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What's New

To see what's new in the latest release of Onshape, check out this list of changes (opens in new tab), and also this Forum post of new features (opens in new tab).
Onshape Help

To get started with Onshape and set up your account and behavior defaults, we highly recommend going through the Primer first. This walks you through pertinent "set and forget" account preferences, how to begin a sketch, make a part, and other basics of Onshape. The estimated time to completion is 50 minutes for all sections, but you are able to select the modules of your choice.

If you are new to Onshape, going through the Primer is a good way to get acquainted with Onshape concepts and some basic functionality.

This master help system contains help for all of the platforms that Onshape works on. In each topic, information for all platforms is explained. In some topics, the information is platform-specific and there are dropdowns for each platform. In other topics, the information is not platform-specific, so the information applies to all platforms.

Feedback

To provide feedback on the Help system itself, click on the blue Feedback button on the right side of your browser.

Use the tool within Onshape to log a ticket for Onshape Support. Expand the Help menu (click the icon) and select Contact Support. Enterprise customers may also reach out to their Customer Success manager.

At the bottom of every topic in the help, you'll find the Was this article helpful? feedback tool (as seen below). Provide your feedback by clicking either the Yes or No button.

Hardware and Graphics Performance Recommendations

Onshape is built on a cloud-based architecture which has two primary and distinct advantages:

- Hardware requirements are significantly less for Onshape than for installed desktop CAD programs.
Onshape technology is lightweight and consists of short, intermittent messages, yielding much greater performance for any given bandwidth.

To ensure optimal GPU performance when using Onshape, browse the recommendations below and compare to your configuration.

You have the ability to let Onshape determine whether your browser is compatible with Onshape here: browser compatibility (opens in new tab).

The browser compatibility checks for and displays the following information:

- Browser and version
- WebGL and extensions
- Renderer
- Performance check
- WebSockets
- Geographical Data
- Onshape server region

This is for informational purposes only, Onshape’s compatibility check does not resolve any issues.

**Graphics Performance: Desktop**

**Browsers**

Onshape currently supports these tested and approved browsers:

- Google Chrome
- Mozilla Firefox
- Safari (Mac OS only)
- Opera
- Microsoft Edge

Microsoft Internet Explorer is currently not supported.
Onshape suggests that you run the 64-bit version of browsers on operating systems that are able to run both 64-bit and 32-bit (i.e., Windows, Linux).

Vivaldi browsers will work with Onshape, however, you need to disable "Allow Gestures" in your browser Mouse settings as this setting conflicts with Onshape's mouse settings.

**WebGL**

Onshape requires WebGL. To ensure that you are taking advantage of the highest performing configuration, first update your graphics drivers to the most recent version from the manufacturer and make sure your preferred browser has WebGL enabled. Most modern browsers enable it by default, but certain hardware or graphics driver configurations will turn it off. If you see an error in Onshape (“It looks like your browser doesn’t have WebGL enabled”, or “Rats! WebGL hit a snag.”) or the browser compatibility (opens in new tab) check page says WebGL is disabled, try the following steps in your browser of choice.

Some graphics cards are blacklisted because of poor WebGL support. A list is available at https://www.khronos.org/webgl/wiki/BlacklistsAndWhitelists (opens in new tab) and steps are listed below to override the blacklist in Chrome and Firefox. Legacy operating systems, such as Windows XP, can lack modern driver and browser support, and hence may not run Onshape, even with these work-arounds.

If you make these changes while running Onshape, simply refresh your browser for the changes to take effect.

**Chrome**

Ensure WebGL is on and hardware accelerated is checked first:

1. Go to chrome://settings.
2. Click the Show advanced settings link.
3. Scroll down to the System section and ensure the Use hardware acceleration when available checkbox is checked.
4. Relaunch Chrome so any changes take effect.
Check Onshape at this point. If it’s still not working, try to force WebGL hardware rendering via the following:

1. Go to chrome://flags.
2. Enable the flag called Override software rendering list.

Firefox

1. Go to about:config.
2. Search for webgl.disabled and ensure its value is false.
3. Go to about:support.
4. Inspect the WebGL Renderer row in the Graphics table:
   a. If the status contains a graphics card manufacturer, model and driver (eg: “NVIDIA Corporation -- NVIDIA GeForce GT 650M OpenGL Engine”), then WebGL is enabled.
   b. If the status is something like “Blocked for your graphics card because of unresolved driver issues” or “Blocked for your graphics driver version”, then your graphics card/driver is blacklisted.
5. If your graphics card/drivers are blacklisted, you may override the blacklist:
   a. Go to about:config.
   b. Search for webgl.force-enabled.
   c. Set to true.
6. Like Chrome, Firefox has a Use hardware acceleration when available checkbox:
   a. Go to Preferences > Advanced > General > Browsing.
   b. However, unlike Chrome, Firefox does not require this checkbox to be checked for WebGL to work.

Safari

1. Click Safari and select Preferences from the menu.
2. Click the Security tab.
4. Select Allow WebGL.
Opera


2. Navigate to Browser and scroll down to System.

3. Ensure that Use hardware acceleration when available is checked.

4. Restart your browser for changes to take effect.

More information for all of these browser settings is available at: https://superuser.com/questions/836832/how-can-i-enable-webgl-in-my-browser (opens in new tab).

Graphics cards

The system check (explained above) also provides insight into the rendering performance of your computer as currently configured. This is expressed as triangles per second (TPS) and represents ‘most’ of the work being done by your computer. The more TPS your computer can render, the faster a model will be rendered to the screen on open and the more smoothly it will rotate.

While it is hard to give numbers for recommended TPS, it is easy to say that for larger assemblies, the Onshape experience is better with a discrete (dedicated) graphics card.

To improve your TPS, you don’t need an expensive graphics card, a low-end gamer card that supports WebGL will work, and generally improve performance significantly. More GPU RAM will enable more and larger datasets to be opened concurrently in Onshape. Without a dedicated graphics card, the rendering task is performed by the CPU and yields significantly lower numbers.

If you see the message shown below, take steps to either install a dedicated graphics card, or take steps to resolve an unrecognized graphics card, explained below.
Many computers have more than one graphics card installed (GPU). Often, on Windows machines with NVIDIA graphics cards, Optimus technology is also involved, meant to optimize the workload between GPUs -- to assign the proper graphics card to applications needing a high performing graphics card and applications needing more battery life and lower performing graphics card. Your out-of-the-box system settings may be using the integrated chip by default to render Onshape in your web browser or when on battery power. To get the best performance, specify that the browser always use the discrete graphics card, or disable automatic graphics switching, depending on your device.

**If you do not have NVIDIA or NVIDIA with Optimus technology, you may skip this section.**

To get the most out of your graphics cards:

- Make sure the graphics card you use is not on the WebGL blacklist, as some video graphics cards do not support WebGL.

  See https://www.khronos.org/webgl/wiki/BlacklistsAndWhitelists (opens in new tab) for more information.

- Onshape performs best when Optimus technology is *not* involved in the management of graphics cards. To ensure that Optimus technology, if present, does not interfere with the best performance of the graphics card when working with Onshape:

  Determine whether or not your computer uses Optimus technology:
* Open the NVIDIA control panel.
  
  Select System Information and then Components. If Optimus technology is present, it will be listed somewhere in the right column.

* If Optimus technology is present, make the NVIDIA graphics card the default card for Onshape (for the browser you use with Onshape), through the NVIDIA control panel.

To make the NVIDIA graphics card the default GPU for Onshape, or rather, the browser you want to use for Onshape:

1. Open the NVIDIA control panel.
2. Navigate to 3D Settings > Manage 3D Settings and then the Program Settings tab.
3. Locate the browser you use for Onshape.
4. Set Select the preferred graphics processor for this program option to High-performance NVIDIA processor.

If this method doesn't work, see below for more options.

**In a nutshell**

You want to use your high performance GPU when an application (like Onshape) demands it. Having a management technology involved (like Optimus) doesn't always result in the performance you are hoping for.

If you don't know what your computer has by way of GPUs, you have the ability to download and use a utility such as Speccy (for Windows) or gfxCardStatus (for Mac) to discover what is installed on your machine.

You want to use the faster, discrete NVIDIA GPU (when available) for Onshape, always. For applications that do not require high performance graphics or require longer battery life, you have the option to use an integrated GPU such as Intel's integrated GPU. To this end, assign the appropriate GPU to a specific browser.

**Alternative**

As a last resort, you could try to go into the machine's BIOS settings and switch off Optimus technology completely and run using the discrete NVIDIA GPU all the time.
This carries serious risk, however, so make sure you know what you’re doing here, or seek help before attempting this solution.

**Displaying memory-intensive models**

Onshape uses a WebGL rendering context in order to display 3D data in the browser. There are occasions when a browser will take away the rendering context. For instance, when changing the display connected to your computer, the browser may need to update the context for the new display. Typically, the context is quickly restored after being taken away.

Occasionally a browser may take away the WebGL context and never return it. One known cause of this situation is when the GPU runs out of memory. This may occur if the GPU has relatively little video RAM to begin with (less than 1 GB), or if the loaded tab has sufficient complexity to exceed the limits of the GPU’s RAM. If context loss issues persist, try loading the document on another device with more video RAM to see if the issue recurs there. Video RAM memory usage may be reduced by lowering tessellation quality for parts contained within the tab.

**Speed of page loading and tessellation quality**

In very large documents, Chrome may run out of memory before the tab (Part Studio or Assembly, for example) is fully loaded. This may occur due to the large number of entities and volume of display data involved.

One way Onshape speeds the loading of the tab is by deferring the loading of ‘less important’ bodies, for example, parts that are out of view or too small to be used. These bodies are loaded when they become ‘important’ enough for viewing as the need arises, for example, when hovering over the part, zooming in, or when hiding some parts so others are visible.

In order to stay under the browser memory threshold, Onshape may unload other unimportant bodies after loading more important ones. If this is necessary, small or invisible, memory intensive parts are unloaded first.

Unloaded parts will appear as semi-transparent boxes that take up the bounds of the part. Parts become ‘important’ as they come into view and take up a significant part of the screen. When this happens, the system automatically starts loading the part. Once it is loaded, the part resolves to its fully-loaded state, and all geometry is visible.
Onshape progressively pulls down finer tessellation for part data in Part Studios and Assemblies, when finer tessellation exists.

The mechanism works by sampling the Part Studio or Assembly to detect poor-quality parts after view manipulation is stopped. If poor quality parts are detected, Onshape requests better representations from the server.

This automatic tessellation refinement might be hindered under the following conditions:

1) The interactive frame rate becomes too slow (the current threshold is 20 FPS)
2) The amount of data exceeds the memory limit

In the case of exceeding the memory limit, Onshape compensates by swapping out older, unused body representations for refined representations of what is currently in view.

**Troubleshooting the performance check**

If the performance check was not executed due to a low refresh rate (less than 60hz), or the check resulted in an unexpectedly low triangle/line count, these troubleshooting steps may lead to better results:

- Ensure that the performance check page window is in focus throughout the test.
- Limit the amount of other activity going on in the test system. If another CPU/GPU-intensive program is running simultaneously, it could steal resources from the test and affect the results.
- In case of dual graphics systems, ensure that you are getting the expected GPU. On some systems the browser is limited to the integrated GPU, which may not perform as well as the discrete GPU. Consult the GPU documentation or the operating system help to ensure the browser is able to access the GPU of your choice.
- The performance test will not be run if the browser is not updating at (at least ) 60 frames per second. Certain combinations of displays, cables, and GPUs will result in a reduced refresh rate. Check your GPU specifications to ensure that the GPU can support a 60 hz refresh rate at the native resolution of your display. Also ensure that your display cable (e.g., HDMI) supports the given resolution and refresh rate. A reduced refresh rate does not affect the overall number of triangles.
Onshape can display, however you may experience some drop in interactive performance. You can use this site to test the refresh rate on your browser: https://www.testufo.com/refreshrate

**Improving rendering performance**

When browser frame rate has dropped below the Onshape threshold for more than three minutes, Onshape displays the message: *Reduced rendering performance detected.*

Before proceeding, it's best to check system compatibility to make sure you are using the best GPU on your system. Click the link in the message or here. Once you have determined that you're using the best GPU, you can also try the following to improve performance:

- Change render mode to “Shaded without edges.” For large assemblies that contain many components, edges can add a significant amount of time to the render loop. If edges are not needed, you can opt for the “Shaded without edges” rendering mode in the View cube dropdown. Remember that you can still select edges even when in “Shaded without edges” mode.

- Disable “Match pixel density.” If your computer has a high-DPI display, significant time can be spent matching the pixel density of the monitor. Improve performance by sacrificing some quality through disabling the “Match pixel density” option. See "Environment profile settings" in the Preferences help topic for more details.

- Disable tessellation overrides. When tessellation is being forced to a higher level of detail, it results in the GPU having to render more triangles and can affect performance. Consider removing the override to avoid this overhead, or force detail to the coarse setting, to lower the total triangle count. See the "Specifying tessellation quality" heading within the "Specifying tessellation quality of parts" on page 309 help topic for more details.

- Make sure your browser is up-to-date. Browser vendors, such as Chrome and Firefox, are continually updating their own code base in order to optimize performance and add functionality. If your browser has not been updated recently, you may be missing out on updates that can have real effects on Onshape performance.
**Internet Connection**

The primary requirement for using Onshape is a reliable Internet connection. An intermittent connection results in a sub-optimal Onshape experience - you will be asked to refresh your browser tab with each dropped connection. However, no work or data is ever lost.

Keep in mind that because using Onshape is not like streaming a movie or using a screen sharing app, it does not require constant use of a high bit-rate (bandwidth).

Instead, as stated earlier, Onshape is 'conversational' meaning that short messages are sent only when an action is being performed. For an (idealized) example, the request to fillet an edge is replied to by a return message describing the facets (triangles) that have changed as a result of the fillet. All rendering, rotation, zooming, sectioning, and selection is handled by the GPU and requires ZERO bandwidth. Because the duration of the messages is small compared to the periods of 'silence', the bandwidth requirement for multiple users scales in a very non-linear way. Are there variations on this? Yes, but this example describes the principle by which multiple users can share even a modest connection. If you must have a metric: if your connection can support a single video stream, you can support a team using Onshape.

No CAD data (other than exported downloads) is transferred to the end user, eliminating the morning 'logger jam' experienced by users of other CAD systems while trying simultaneously to check files out of a vault onto their local machines.

**RAM**

Due to Onshape's full-cloud architecture, increases in RAM provide only marginal gains in performance. However, additional RAM can improve the experience of running multiple browser tabs or other applications simultaneously.

**Mobile Devices**

When using Onshape on your mobile device, you also have access to the same help system as on the browser. Onshape Help offers device-specific help where appropriate. You can access Onshape Help through the browser and through Onshape mobile app (links to the apps below).
This means:

- You get all of the power, precision, functionality, and flexibility of Onshape no matter what platform you’re using
- You can sign in and work from anywhere—without ever having to worry about updates, new versions, installations, memory or storage
- You have access to all of your documents and project files at any time

Supported Android versions

- Lollipop (5.0, 5.1)
- Marshmallow (6.0)
- Nougat (7.0)
- Oreo (8.0)
- Pie (9.0)
- Android 10

Onshape minimally requires Lollipop (5.0).

To check what version of Android your device has installed, go to your settings and check the About or General section.

Graphics Performance

Devices with older GPUs and older implementations of OpenGL ES 2.0 sometimes lack the necessary capabilities to run Onshape’s 3D graphics. This most often happens on devices with a Mali 400 GPU.

When using devices with older GPUs or implementations of OpenGL ES 2.0 an error message will alert you when you open a Part Studio or an Assembly. The error message offers a link to more information as well as a link to continue.

If you elect to proceed after receiving the error message, you will likely notice graphics glitches or a blank, white screen. Eventually, the app may crash.
Why doesn't it work?

The Onshape graphics library uses Vertex Array Objects (VAOs). VAOs are supported natively in OpenGL ES 3.0 and above, but are not supported in the OpenGL ES 2.0 library. Often, implementations of OpenGL ES 2.0 contain extensions that support VAOs, but the support of these extensions is not guaranteed by Android and depend on the service of the manufacturer.

By default, Onshape allows a device with OpenGL ES 2.0 or higher to install the application but it is not until the app is running on a device that it can detect if the device has the necessary capabilities or extensions.

How can I fix this?

Use a device that supports, at the very least, OpenGL ES 2.0 with Vertex Array Object extensions.

A device running the latest version of Android does not necessarily support OpenGL ES 3.0. We recommend that you avoid running Onshape on devices with a Mali 400 or Mali 450 GPU, as these are unable to render 3D graphics properly.

It is recommended that you use a device that runs OpenGL ES 3.

How do I find out what GPU I have?

The most direct way to find out what GPU your device uses is to search for your device and/or serial number on the Internet. For example; you might try browsing or searching for your device on a website such as gsmarena.com or phonearena.com.

The earliest iOS devices supported

- iPad Mini 3
- iPad Air
- iPad Pro

Onshape minimally requires the latest version of iOS 11.

Optimal iOS devices supported

- iPhone 11
- iPhone 7/8/X
- iPad Mini 4
iPad Air 2
iPad Pro

To check what version of iOS your device has installed, go to Settings > General > About > Version.

If you'd like to bookmark Help pages on the iOS platform, you can use your app settings on your iOS device and set "Open Help in browser" to on. When you access the help via Onshape, it will then open in a browser, where you can create bookmarks of specific topics or pages.

Additional information

More resources include:

- [http://alteredqualia.com/texts/optimus/](http://alteredqualia.com/texts/optimus/) -- more information and specific instructions

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Performance Considerations

Part of any modeling process, in any tool, is modeling efficiently and appropriately for manufacturing, workflow, and yes, modeling itself. When you create models using CAD tools effectively, you reduce regeneration times for the entire model. In Onshape, there are specific steps you can take to ensure efficiency of modeling and reduced regeneration times.

Check regeneration times in the Part Studio

Some parts may take a long time to regenerate in your Part Studio; this may result in slower loading times than necessary. Some modeling strategies can result in long regen times, as well as placement of complicated features in the Feature list.

What can you do to reduce regeneration times?

First, understand which features are taking a long time: Use the Regeneration panel in the Feature list to get an idea of times per feature. Click the clock icon in the Feature list to open the Regeneration times panel:
Once you can see which features are taking a longer time to regenerate, you can take action.

Next, understand how you can influence regen times:

- When creating patterns or mirrors, use the part or face pattern or mirror option instead of the feature pattern or mirror option, so that the feature does not need to regenerate for every instance of the pattern. For example:
Suppress features that require longer times to regenerate until you are finished designing. At that time, you can unsuppress those features.

Move longer regenerating features to the end of the Features list.

For more information on using feature patterns and mirror tools, see: "Circular Pattern" on page 986, "Curve Pattern" on page 1010, "Linear Pattern" on page 965, and "Mirror" on page 1037.

For more information on reducing regeneration times, see Working with the Feature list.

Derive parts from versions

Deriving parts can be beneficial to streamlining design both for saving time and for consistency of design. There’s no need to recreate a design in order to iterate off of it or model closely-related parts from it.

However, there are some considerations to take into account when deriving parts. First and foremost, deriving is best done from a version of the document. Since versions are immutable and static, there is no regeneration time involved and no updating to be done when deriving from an entity in a version. You can select a version directly in the Derive dialog. For example:
For more information on versions, see Version Manager. For more information on deriving parts, see Derived tool.

**Don't derive parts across many Part Studios**

As economical and design-savvy it is to derive instead of recreate, there are performance consequences to deriving across many Part Studios, in a daisy-chaining method. Instead, create a version and derive directly from the version. If the design needs to change, create a branch from the version, make the change, and then create another version from which to derive.

How many Part Studios is too many? If you find yourself deriving over three Part Studios, it's time to derive directly from a version.
For more information on deriving parts, see Derived tool.

**Keep tabs per document at a reasonable number**

Despite Onshape documents being a sort of container that can hold many kinds of data and seemingly endless numbers of tabs, best practices are to:

- Keep the number of tabs in a document low, as in less than 100.
- Use 'Move to document' to move specific tabs to another document not only to improve performance within a single document, but also because moving a part to another document enables you to share specific parts of the design with specific third-parties, or for review purposes. See, "Organizing tabs" on page 128 for more information.

**Things to Know**

To use Onshape efficiently (either on a browser, or in the app), here are some helpful things to know about its design, functionality, and user interface.

Onshape runs in a browser, and mobile device apps exist for both iOS and Android. There is no software to download or maintain, ever. On a browser, Onshape updates automatically.

What browsers does Onshape work on? See our Recommendations.

**Mobile Help**

When using Onshape on your mobile device, you also have access to a mobile-specific help system. Onshape Mobile Help offers the same information as the Onshape Help, but for our supported, touch-based mobile operating systems (iOS and Android). Access Onshape Mobile Help through the Onshape mobile app (links to the apps below).

![Download on the App Store](https://example.com/appstore)

![Get it on Google Play](https://example.com/googleplay)

This means:
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How can I fix this?

Use a device that supports, at the very least, OpenGL ES 2.0 with Vertex Array Object extensions.

A device running the latest version of Android does not necessarily support OpenGL ES 3.0. We recommend that you avoid running Onshape on devices with a Mali 400 or Mali 450 GPU, as these are unable to render 3D graphics properly.

It is recommended that you use a device that runs OpenGL ES 3.

How do I find out what GPU I have?

The most direct way to find out what GPU your device uses is to search for your device and/or serial number on the Internet. For example; you might try browsing or searching for your device on a website such as gsmarena.com or phonearena.com.

The earliest iOS devices supported

- iPad Mini 3
- iPad Air
- iPad Pro

Onshape minimally requires the latest version of iOS 11.

Optimal iOS devices supported

- iPhone 11
- iPhone 7/8/X
- iPad Mini 4
- iPad Air 2
- iPad Pro
To check what version of iOS your device has installed, go to Settings > General > About > Version.

If you’d like to bookmark Help pages on the iOS platform, you can use your app settings on your iOS device and set "Open Help in browser" to on. When you access the help via Onshape, it will then open in a browser, where you can create bookmarks of specific topics or pages.

**Files not needed, Onshape documents are project-level**

Onshape does not use files. Instead, it uses Documents and Tabs. A document is a project-level container that consists of as many tabs as you need. Fill your document with different types of tabs such as Part Studio, Assembly, and Drawings tabs. You are even able to use a tab to hold a PDF, a video, or a picture. Tabs are able to hold anything, and documents are able to hold infinite tab. Pay attention to the organization of your documents, however. Complicated models are better organized across many documents, not only for performance sake, but also for the ability to keep track of and reuse parts more easily. See [Document tabs](#) for more details.

For more recommendations on organizing documents, see [Product Structure Organization Tips](#) in the Onshape Learning Center (sign in required).

You have the ability to [import](#) CAD files from other CAD tools either as a document, or as a tab within a document.

Check out our [list of supported file formats](#).

**Automatic and infinite history and restore points**

Because Onshape is full-cloud CAD, every action is automatically saved. Actions are recorded, points in history can be compared and restored at any time. You never need to manually save, and you are always working on the most recent version of your document. See [Version Manager](#) to learn more about Onshape’s unique, built-in data management.

**Start modeling with a sketch**

To start modeling in a new document in Onshape, you must begin with a sketch in a Part Studio. If you import a CAD file or select a document with an existing part, you may begin direct editing that part using feature tools.
Part Studio
Sketch tools
Feature tools

Built-in data management
Onshape has a built-in data management system. You are now able to completely manage versions without leaving Onshape or even your document. This allows for seamless collaboration with real-time updates on changes made to tabs and documents. You also have the ability to create, compare, and merge versions all within Onshape. Read more about Document Management and Collaboration to work efficiently with large or small teams and utilize our built-in data management.

Collaboration
Multiple users are able to work in the same document, and even on the very same part, simultaneously. We call this Simultaneous Editing. When two or more people work together in a document, we refer to them as collaborators. Any feature made or edited by a collaborator is displayed, in real time, to all collaborators. Share, follow, and comment to make simultaneous editing more efficient. Manage versions and history to make the most of any collaboration in Onshape.

See our topic on Sharing and Collaboration to learn more about collaboration.

FeatureScript
FeatureScript is the programming language that all Onshape features are built with. With FeatureScript, you are able to define and create your own custom features in Onshape.

Learn how to create a custom feature or see our FeatureScript Documentation for more information.

On a desktop, you are also able to access our FeatureScript Library and begin using custom features created by others, in your own document right away.
Onshape Primer

The total estimated time of completion is 50 minutes.

Use this primer to become familiar with Onshape's terms, tools, and some of the more common workflows that Onshape has re-engineered to be simpler and more powerful than other systems. This primer steps you through setting up your Onshape profile, understanding the user interface, and the basics of creating and assembling parts. In many lessons, at the left of the screen are some helpful tips that are useful; refer to them at any time. At the top of all the lessons, there is an estimated time of completion right above the description of the lesson.

You can step through one lesson at a time, in sequential order, or use this list to jump to your preferred starting place. We recommend starting from the beginning so you don't miss any important information.

- "Setting Your Preferences" on page 40
- "Creating Parts" on page 49
- "Defining and Constraining Sketches" on page 55
- "Viewing, Selecting, and Shortcuts" on page 64
- "Importing and Editing a Part" on page 71
- Measuring and Mass Properties
- "Working with Versions" on page 78
- "Working with Branches" on page 89
- "Merging Branches" on page 92

Before you start the Primer, there are some things that are worth knowing.

Onshape supports these browsers:

- Google Chrome
- Mozilla Firefox
- Safari (Mac OS only)
- Opera

Internet Explorer is currently not supported.
Supported Android versions

- Lollipop (5.0, 5.1)
- Marshmallow (6.0)
- Nougat (7.0)
- Oreo (8.0)
- Pie (9.0)
- Android 10

Onshape minimally requires Lollipop (5.0).

To check what version of Android your device has installed, go to your settings and check the About or General section.

Graphics Performance

Devices with older GPUs and older implementations of OpenGL ES 2.0 sometimes lack the necessary capabilities to run Onshape's 3D graphics. This most often happens on devices with a Mali 400 GPU.

When using devices with older GPUs or implementations of OpenGL ES 2.0 an error message will alert you when you open a Part Studio or an Assembly. The error message offers a link to more information as well as a link to continue.

If you elect to proceed after receiving the error message, you will likely notice graphics glitches or a blank, white screen. Eventually, the app may crash.

Why doesn't it work?

The Onshape graphics library uses Vertex Array Objects (VAOs). VAOs are supported natively in OpenGL ES 3.0 and above, but are not supported in the OpenGL ES 2.0 library. Often, implementations of OpenGL ES 2.0 contain extensions that support VAOs, but the support of these extensions is not guaranteed by Android and depend on the service of the manufacturer.

By default, Onshape allows a device with OpenGL ES 2.0 or higher to install the application but it is not until the app is running on a device that it can detect if the device has the necessary capabilities or extensions.
How can I fix this?

Use a device that supports, at the very least, OpenGL ES 2.0 with Vertex Array Object extensions.

A device running the latest version of Android does not necessarily support OpenGL ES 3.0. We recommend that you avoid running Onshape on devices with a Mali 400 or Mali 450 GPU, as these are unable to render 3D graphics properly.

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We suggest that you run the 64-bit version of browsers on operating systems that can run both 64-bit and 32-bit (i.e., Windows, Linux).

Context menus are everywhere
Use a right mouse button (RMB) click on an entity to invoke its context menu. Context menus contain commands for that entity in the current context. Context Menus exist for entities in the graphics area, entities in Feature lists, Parts lists, and Drawings, as well as Onshape constructs such as tabs. Right-click throughout the interface to discover Context Menus.

View cube and view tools
Change the view of your workspace inside a Part Studio or Assembly with the View cube. Click the View cube, located in the upper right corner of the graphics area, to chose from a list of different viewing options. Select one to change your view of the graphics area or to change the view settings of your entities.

Use the smaller cube below the View cube to access the View tools which allow you not only shortcuts to standard model views like isometric, Dimetric and Trimetric, but also contains commands to display the model in various visual ways, observe curvature visualization and draft analysis and apply section views.

See View Navigation and Viewing Parts for more information on different views.

Touch gestures for mobile devices
Onshape allows the same functionality on a mobile device as it does on a web browser. However, because of the unique, touch-based interface of smart phones and tablets, there are some basic and important gestures you need to know. (The orange markings below represent touch actions in the interface.)

Select
Tap to select or deselect an entity or tool. Double tap to deselect all selections.
Zoom
Pinch to zoom in or out.
**Scroll**
Swipe to scroll through a Feature list or dialog.

**Open Context menu**
Two-finger tap to open Context menus.
3D rotate

Drag a single finger in the graphics area to 3D rotate.
2D rotate

Two-finger twist in the graphics area to 2D rotate.
Pan

Two-finger drag to pan the graphics area.
Deselect all

Double-tap in the same spot to deselect all selections.
Precision selector
Touch and hold in the graphics area to call the Precision selector. Use crosshairs to pinpoint selection. To select: Release finger when crosshairs are lined up with desired selection entity (when desired entity is highlighted).

See User Interface Basics > Selection for more information.

**Stylus**

Onshape Mobile supports the use of a stylus such as the Apple Pencil.

Use the stylus for any one-finger gesture such as select, deselect, scroll, 3D rotate, and precision select (all mentioned above). The actions to complete the gestures with a stylus are exactly the same as those with a finger except for Precision selector.

Precision selector with stylus

While using the stylus to precision select, press harder to temporarily zoom. Once you release to make a selection, the zoom returns to normal:

![Precision selector with stylus](image)

**Contact Support tool**

Click Contact Support (if you have a professional subscription) or Report a bug (if you have a free subscription), located in the drop down or along the side of our Documentation topics to provide feedback, report bugs, and contact support.
Setting Your Preferences

*The estimated time of completion is 8 minutes.*

This section covers signing in to Onshape, checking your computer graphics compatibility and setting your personal work preferences. Work preferences are set once and then become immediately available to your Onshape account, regardless of what device or browser you access Onshape from.

If you have already created your account, all you need is the following Onshape URL (or the URL of your Onshape enterprise if your company has subscribed to an enterprise plan): [https://cad.onshape.com/signin](https://cad.onshape.com/signin)
If you haven't yet created your account, you need to do that before you can sign in.

When you were added to a company subscription in Onshape, or you signed up for an Onshape plan on our website, you received an email notification. In that notification is a link to Onshape.

1. Open the email and click:

   CREATE YOUR ONSHAPE ACCOUNT

2. In the Onshape form that opens, fill in the fields:

   a. Your email is provided and is the one you were registered with, do not change this.

   b. Create a password with the following requirements:
      i. 8 characters or more in length
      ii. Include at least one of each: numeral, lowercase alphabetic character, and uppercase alphabetic character

   c. Make sure that the password and password confirmation match.

   d. Enter your first and last name and your phone number.

   e. Review Onshape's terms and privacy policy. Check the box to agree.

   f. Check the box to agree to be contacted by Onshape regarding products and services (this consent can be withdrawn at any time).

3. Click: Sign up.

Onshape opens. Now whenever you want to sign in to Onshape, simply go to your Onshape URL and enter your user credentials.
To check your system against Onshape’s requirements and make sure that you have and are using the most beneficial configuration and graphics processor for Onshape:

1. Click 🤔 in the Onshape title bar (near the right end of the bar):

2. Select **System check** from the menu.

3. Wait for the system check to automatically complete. Green check marks appear when a test is completed.

This is an informational step only - for more information on the Performance check for graphics, you can click [Graphics and Performance Recommendations](#) and learn more about how Onshape handles graphics.
Onshape understands that you may have habits and methods of working with your CAD data, and we have settings to make it easier for you to work the way you're accustomed to.

Set these preferences once and that's it. Whenever you sign in, on any device or browser at all, your preferences are already set and available, no resetting is necessary.

1. Open the User menu (click your name in the right top corner of the window).
2. Select My account from the menu.
3. On the page that appears, select Preferences in the left panel.
4. Make your preferential choices about your Onshape environment.

   **Note that each section has its own Save button.**

5. Make sure to click Save for each section.

**Language**

Language - Select your preferred language for the user interface and help system. Upon clicking Save, Onshape refreshes the screen and you are required to sign in again.

Click Save language.

**Units**

Units - Specify the default units for measurements in all documents that you create. To override the default at any time, enter the desired units at the time of creation (while dimensioning a sketch, for example). Otherwise, all numeric values will use these units by default.

Click Save mouse controls.
Mouse controls - Use the drop down menu under "View settings" to select the CAD system with which you are most comfortable. Review the mouse settings for rotate, constrained rotate, pan, and zoom.

- If you are most familiar with SOLIDWORKS, select SOLIDWORKS.
- If you are most familiar with NX 10, select NX 10.
- If you are most familiar with Creo, select Creo.
- If you are most familiar with AutoCAD, select AutoCAD.

Click Save profile settings.

Environment profile settings - To match the pixel density on high resolution displays (perhaps different devices or monitors), you can create profiles here. When using Onshape on a high resolution device or with a high resolution monitor, select the appropriate profile for that device.

Click Save profile settings.

Shortcut toolbars - Configure your own in-app shortcut toolbars accessible through the 's' key while in a Part Studio, Assembly, or Drawing tab or with a sketch open in a Part Studio. This does not alter the window toolbar but rather supplies you with a quick and customized toolbar that appears at your cursor whenever you press the 's' key in empty space.

1. Select the toolbar to create from the dropdown menu.

2. Place a checkmark next to the tools to include in the shortcut menu; remove checkmarks to remove tools from the menu.

Click Save shortcut toolbar settings.

Toolbars - Click "Reset to defaults" to set the customized shortcut toolbars back to the Onshape defaults.

Click Save.
Set background color for imported drawings

**Drawings** - Set the background color (dark or light) of the model space in your imported drawings (.DWG and .DXF files). Note this does not change the color of drawings created in Onshape, just for drawings imported into Onshape.

Click Save drawing settings.

**Material libraries** - Use your own custom material libraries, if you want. Note that the material library must be added by an administrator in order for it to appear in this list so other users can make use of it as well.

For more detailed information on using material libraries, see [Creating new material libraries](#).

**Release management** - For Professional and Enterprise plan users, use the Release management section of the Preferences page to delegate your release approval responsibilities to another user or team.

For more information about release management set up and workflows, see [Release Management](#).

Use custom material libraries with Onshape

Delegate release approval responsibilities

Additional resources

<table>
<thead>
<tr>
<th>Professional and Enterprise plan users</th>
<th>Standard users</th>
</tr>
</thead>
<tbody>
<tr>
<td>My Account Settings</td>
<td>Manage Account</td>
</tr>
</tbody>
</table>

Creating a Sketch

*The estimated time of completion is 4 minutes*

This section covers how to begin creating a sketch in an Onshape document.

The basic container in Onshape differs from the files of other CAD systems. Onshape uses a structure we call a document, and documents can hold all types of CAD data and supporting materials like images, files, PDFs and more.

To begin creating a sketch in Onshape, you first create a Document.

Sign in

Navigate to your enterprise URL or to [https://cad.onshape.com/signin](https://cad.onshape.com/signin) and enter your user credentials.
Create a document

Helpful tip

Onshape documents can contain as many tabs as needed, of any type needed. You can create more, delete, even copy tabs. Right-click on a tab to access its context menu.

Think of Onshape documents as multi-data containers. Not only can you sketch, create parts, assemble parts, and create drawings all in one document, but you are also able to store non-CAD data in any document as well: images, non-translated CAD data, PDFs and more. Just about any type of file you want to keep track of has the ability to be imported into a document.

1. On the Documents page, the page that appears when you sign in to your account, click .
2. Select Document... from the menu.
3. Type a name for your document (eg. "Primer").
4. Click OK.

The document opens and contains two tabs: Part Studio 1 and Assembly 1. Tabs are Onshape's way of allowing you to store more than one type of data together in one place. Look towards the bottom of the window to see the name of the tab. Click a tab to activate it. Each data type is stored in a specific tab type.
Onshape parts typically begin with a sketch. The first tool on the Part Studio toolbar is Sketch (to the right of the Undo/Redo icons).

To create a solid, you need a sketch with closed regions (as opposed to creating a surface, for which only a sketch curve is necessary). Onshape automatically shades all closed regions of a sketch. Below are examples of closed regions and sketch curves:

1. Click Sketch.
2. The dialog opens:

   ![Sketch dialog](image)

   *Blue highlighted fields indicate selection in the graphics area is needed*

3. Select a plane in the graphics area or from the Features list on the left.
4. Press the N key to orient the sketch plane to normal.
5. Click (or press the G key) to select the Rectangle tool.
6. Click in the graphics area to start the rectangle, move the cursor to draw the rectangle.
click the icon in the toolbar.

When selecting entities in the graphics area, Onshape outlines or highlights entities upon hover when they are selectable for the current action.

Helpful tip

- Access the keyboard shortcut map through the Help menu.
- The first dimension you apply scales the entire sketch.

8. Type "4" for the width and press Enter. The height box immediately becomes active. Type "3" for the height and press Enter. (You can edit or delete these dimensions at any time, when the sketch dialog is open.)

Since the default units were defined previously, units need only be entered if they are different from the default.

9. Click to begin another rectangle. Click to set the size of the second rectangle.

10. Type "1.5" for the width and press Enter. Type "3.2" for the height and press Enter.

11. Press Escape key to release (deactivate) the Rectangle tool.

12. Click the check mark in the dialog to accept the actions and close the dialog (to create and save the sketch).

If you accidentally click the red x and cancel the sketch, you have ten seconds to click Restore in the blue message at the
top of the window to restore the sketch.

Notice the sketch is listed in the Features list on the left (as Sketch 1).

Parts are created from sketches and your design goals dictate what Feature tools you use to create a part. In general, you will start with the first group in the Feature toolbar (visible in a Part Studio whenever no sketch dialog is open):

These tools are, from left to right: Extrude, Revolve, Sweep, Loft, and Thicken. The down arrow beside Thicken reveals an additional tool, Enclose.

Click a tool to select it. Click again to deselect it (or press the Escape key).

Creating Parts

*The estimated time of completion is 15 minutes*

This section covers how to begin creating parts in an Onshape document starting with a sketch.

The basic container in Onshape differs from the files of other CAD systems. Onshape uses a structure we call a document, and documents can hold all types of CAD data and supporting materials like images, files, PDFs and more.

To begin creating a part in Onshape, you first create a Document (see more information on creating documents in the [Creating a Sketch](#) topic).
Create a part from the sketch

Helpful tip

All sketches, features, and part in the lists on the left can be renamed. Right-click on the name in the Feature or Parts list to access the context menu for an item.

Parts (solid bodies) are created from sketch regions (closed regions are indicated by shading). Surfaces can be created from any sketch curves, even if they are part of a closed region.

1. Select the Extrude tool in the toolbar.

The dialog opens.

2. Select the enclosed (shaded) regions of both rectangles.

3. Accept the defaults for the remaining fields in the dialog.

4. Click the check mark to accept the actions and close the dialog.

Notice the parts are listed in the Parts list on the left (*Part 1* and *Part 2*).
Creating a part (solid body)

In Onshape, you have the ability to apply features to 2D sketches in a Part Studio to create 3D parts. All Feature tools are found in the Feature toolbar as icons or within a drop down attached to an icon. Two feature tools, Extrude and Revolve, are also available on the Sketch toolbar for easy access and can be used when a sketch is open. Both methods of creating a part are covered here (with a sketch active, and after the sketch is accepted).

Every sketch and feature created are stored parametrically in the Feature list on the left side of the window. You can go back and edit any feature or sketch listed in the Feature list (double-click to open its dialog or right-click it and select Edit).

Creating a part when the Sketch dialog is open

1. With an existing sketch in the graphics area and the sketch dialog active, click Extrude.

   Onshape automatically selects all regions (enclosed areas that are shaded gray). If any regions are nested, the nested regions are not selected. Onshape does its best to select regions that make sense.

   At this point, you can select additional regions, or deselect regions.

2. Make adjustments in the defaults, if desired, including:

   a. **End type** - *Blind* (for a specified distance), *Symmetric* (in two directions equally), *Up to next* (add material up to the next selected entity), *Up to face* (add material up to the next encountered face), *Up to part* (add material up to the next encountered part), *Up to vertex* (add material up to the next vertex), or *Through all* (add material through all encountered material)

   b. Use the arrows to change the direction from the plane that
the material is added, if necessary.

c. Specify a depth for the material.

d. You are also able to create a draft (for any material) or specify a second end position (for a second direction). These options are covered in Extrude feature tool.

3. Click ✔️ to accept the feature.

Creating a part when the sketch dialog is closed

1. After accepting and closing the Sketch dialog, click 📄.

   The only difference between this method and the above method is that Onshape doesn't automatically select any regions for you.

2. Select the regions to create parts from. Remember, for a solid part, select shaded regions.

3. Specify the details, just as in the above instructions.

4. Click ✔️ to accept the feature.

Shortcuts

- Use the 'S' key to display your shortcut feature toolbar (with the sketch dialog closed).

- Use Shift+E to open the Extrude dialog.

- Click 🎨 and select Keyboard shortcuts to access the list of all keyboard shortcuts within Onshape.
Once you have created a part, you are able to use Onshape tools to refine the part, creating fillets, chamfers, drafts, shelling a part, and more. For the entire list of tools available, hover over the icons in the Feature toolbar for explanations and short instructions for each tool.

This example walks you through applying fillets to a part.

1. Once the feature tool dialog is accepted, click Fillet

2. Select the edges of the part you want to round. Selected edges are highlighted in orange.

3. Tweak the feature with the options in the dialog, like:
   a. Tangent propagation - Whether or not you want the tangency of the fillet applied along all tangent faces.
   b. Cross section style - The style of the fillet: circular, conic, or curvature. Check the cross-section of these to see the differences among them.
   c. Radius of the fillet - The size of the round you require.
   d. Variable fillet - Select specific vertices to apply a different fillet size to. Indicate whether you require a smooth transition between the fillets.

4. Click ✔️ to accept the feature.
Shortcuts

- Press the P key to hide planes and show planes in the graphics area.
- Press the N key to orient the active plane to normal.
- Press Escape to turn off/deactivate selected tools.
- Press Enter to accept actions and close a dialog once required information is entered.
- Select the Extrude or Revolve tool with a sketch dialog open and Onshape automatically selects all enclosed regions, including nested regions:
- Drag the manipulator arrow for pulling material when creating parts (example shown below).

- Use the direction arrows in a dialog (when available) to change the direction of the action (example shown below in the Extrude dialog).
Defining and Constraining Sketches

The estimated time of completion is 10 minutes

Onshape provides automatic inferencing to automatically apply constraints to your sketch. Use dimension and manually applied constraints to help fully define a sketch in order to maintain design integrity.

Open the document

You created the document named "Primer" in the previous lesson "Creating Parts" on page 49. Navigate to the Documents page (after signing in to your Onshape account):

1. For enterprise users, click Documents at the top of the page. Everyone else lands on the Documents page upon sign in.

2. Click the My Onshape filter on the left to display your most recently opened documents.

3. Click the title of the document named Primer.

   The document opens to the last active tab.

4. If Part Studio 1 is not open, select it now.

Helpful tip

When you hover over a title in the documents list, an underline appears indicating that you can click to open that document. If you click in the row without the underline appearing, you simply select that row of the table.
To include design integrity immediately in your design, you can use Onshape's automatic inferencing while sketching and apply additional constraints after sketching. Automatic inferencing is the ability to move near another entity to 'wake up' inferencing and apply constraints automatically.

To practice using automatic inferencing while sketching:

1. Click \( \text{Sketch} \) to select the Sketch tool in the toolbar.
2. Select a plane. (Press the 'N' key to orient the plane normal to your view.)
3. Click \( \text{Circle} \) in the toolbar to select the Circle tool.
4. Click in the graphics area to set the center of the circle.
5. Move the cursor away from the first point and click again to set the diameter of the circle.

Now sketch another circle, using the first circle to wake-up inferencing and constrain the circles to each other:

1. With the Circle tool still selected, move the cursor towards the center of the circle just sketched.
2. Notice that the center of the circle is highlighted in orange and a small box appears around it.
3. Move your cursor horizontally away from the center and notice a dashed orange line that follows the cursor along
To suppress automatic inferences when sketching, press the Shift key when mousing. Holding the Shift key when mousing during sketching prevents any automatic constraints from being applied.

Show all constraints by selecting the Show constraints checkbox.

Show one sketch entity's constraints by hovering over that entity. Use the Shift key to keep the constraints visible in order to select one.

Select a constraint or even just hover over with a small white box with a horizontal line in it (the horizontal constraint icon):

4. Click to set the center of the second circle when the horizontal line is visible, and you set a horizontal constraint between the two circles.

5. Click to set the diameter of the second circle.

6. Press Escape to deactivate the Circle tool.

Now test the constraint:

1. Click either circle's perimeter and drag. The circle resizes.

2. Click a circle centerpoint and drag. Notice that the circle will move in a horizontal direction, aligned horizontally with the other circle. Move the circle vertically and the other circle moves with it (barring no other constraints on
one to see what sketch entities are constrained by it (highlighted in orange).

3. To view all of the constraints, mark the **Show constraints** check box in the Sketch dialog.
**Helpful tip**

*Constraint icons are generally either blue or white.* Blue indicates a constraint associated with something outside of the sketch, for instance the origin, or an edge of a previous feature.

White indicates a constraint between entities in that sketch.

Onshape makes it easy to not only to see all the constraints, but also to evaluate them.

Once you check Show constraints, you have options for viewing and evaluating:

- If it is difficult to see all the constraints, click and drag constraint icons to new locations for easier viewing. This does not change them in any way.

- Select a constraint and the associated sketch entities are highlighted as well, so you can see the entities bound by the constraint:

  When you select or hover over a constraint, the sketch entities involved in it are highlighted in yellow, and the entities bound by the constraint are highlighted in orange:

```
Ø3.5  →  Ø2.5
```

*Above, the horizontal constraint is selected and the two circle center-points are indicated as the constrained entities (orange highlighting) and the circles are indicated as the sketch curves involved (yellow highlighting)*
The best workflow for sketching always includes fully-defining the sketch so that altering one part of it does not alter the intent of the design. Defining a sketch includes using dimensions and constraints.

To identify which entities in your sketch are not fully-defined, look at the color of the curves:

- **Blue** indicates not fully-defined; you can drag to move the entity or change its size
- **Black** indicates fully-defined; you cannot drag to move or alter the entity
- **Red** indicates over-defined, an error, and sometimes an unresolvable sketch

You can also check for constraints, as described above. When there are problems with constraints they are displayed in red:

![A fully-defined sketch](image-url)
An over-defined sketch; the red indicates errors or conflicts

If we apply a dimension between the two circles you've sketched, the sketch becomes more defined:

1. First, click and drag one of the circle's center to be coincident with the origin. Just drag the center to the origin and let go of the mouse button when you see a yellow box around the origin.

2. Now, use the 'D' key to activate the Dimension tool (or click it in the toolbar).

3. Click the center of the larger circle, and then click the center of the smaller circle.

4. Move the cursor to a convenient spot to display the dimension and click to set it there.

5. Type "3.5 inches" and press Enter.

6. With the Dimension tool still active, click the edge of the large circle and move the cursor away, click to set the location of the diameter dimension text and type "3 inches". Press Enter.
7. Click the edge of the small circle and move the cursor away, click to set the location of the diameter dimension text and type "2 inches". Press Enter.

8. Press Escape to exit the Dimension tool.

   Notice that both circles are now black and cannot be moved or resized.

9. Click ✔️ to accept the changes.

   To edit a sketch, its dimensions or constraints, double-click the sketch name in the Features list to open the sketch dialog for editing. **Delete a constraint or a dimension** by selecting it and pressing the Delete key.
For more information about Onshape constraints, see the Help section listed below:

**Coincident** - Make two or more entities coincident, including a sketch entity and a plane.

**Concentric** - Make any point coincident with the center of an arc or circle. Also make arcs and circles share a center point.

**Parallel** - Make two or more lines parallel.

**Tangent** - Form a tangent relation between two curves, or between a curve and a plane.

**Horizontal** - Make one or more lines, or sets of points, align horizontally relative to the current sketch plane.

**Vertical** - Make one or more lines, or sets of points, align vertically.

**Perpendicular** - Form a right angle between two lines.

**Equal** - Make two or more sketch curves of the same type equal in size.

**Midpoint** - Constrain a point to the midpoint of a line or arc.

**Normal** - Make a line and curve, or a curve and a plane normal to each other.

**Pierce** - Constrain a sketch entity (point or curve) to be coincident with the intersection point of its sketch plane and an arbitrary curve that is not in its sketch plane. The sketch entity is now constrained to be coincident with the point of intersection.

**Symmetric** - Constrain two geometries (of the same type) to be symmetric relative to a line.

**Fix** - Ground a sketch entity on the sketch plane so that it does not move.

**Curvature** - Create curvature continuous transitions between
sketch splines (and conics) and surrounding geometry.

**Additional resources**

[Sketch Basics](#)  [Introduction to Sketching (Learning Center)](#)

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**Viewing, Selecting, and Shortcuts**

*The estimated time of completion is 6 minutes*

Onshape has distinct methods for viewing models in the graphics area and has a helpful selection strategy and scheme throughout the user interface. Onshape also has keyboard and interface shortcuts you can use for maximum efficiency. This lesson introduces these strategies and schemes to help you use the interface with ease.

**Open the document**

You created the document named "Primer" in the previous lesson [Creating Parts](#). Navigate to the Documents page (after signing in to your Onshape account):

1. For enterprise users, click **Documents** at the top of the page.
2. Click the **My Onshape** filter on the left.
3. Click the title of the document named **Primer**.
4. The document opens to the last active tab. *(If Part Studio 1 is not selected, select it now.)*
This Part Studio contains the sketch you created in *Creating Parts*.

When setting your preferences in Setting Your Preferences you may have selected a familiar CAD system as a basis for mouse movements when viewing 3D parts and assemblies and 2D drawings. In addition to those methods, Onshape also provides a View cube and View tools in the top right of the graphics area. The View cube and View tools allow for easy, and automatic (respectively) rotation of your model space:

1. Click the **arrows** to rotate the view in 45 degree increments.
2. Click one of the **white circles** at the corners of the cube to return to a *trimetric view*.
3. Click one of the **sides** of the cube to view the graphics area normal to that plane (top, bottom, front, back, right, left).
4. The **small cube** (View tools) below the large View cube contains a menu of additional viewing options. Click the small down arrow to open the menu:

---

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- Isometric  - Dimetric  - Trimetric  - Named views
- Zoom to fit  - Zoom to window  - Turn perspective on  - Shaded
- Shaded without edges  - Shaded with hidden edges  - Hidden edges removed  - Hidden edges visible
- Translucent  - Curvature visualization  - Draft analysis  - Turn section view on

When using Onshape on a mobile device, Onshape uses the mobile OS-specific gestures.
Selecting entities

Helpful tips

- Onshape selection is additive. Click to select, click again to deselect.
- Click in white space (or press the space-bar) to deselect multiple selections at once.

Onshape uses color to indicate selection. For example, selected entities in the graphics area like edges, faces, and sketch entities are highlighted in yellow. Names in the Features and Parts lists are highlighted in blue.

Select entities simply by clicking. Onshape selection is additive and a small number appears at the cursor for up to 5 entities selected. When more than 5 entities are selected a small + sign appears next to the number 5.

To deselect, simply click on the entity a second time. To clear an entire selection of multiple entities, click in empty space, press the Space bar, or right-click and select "Clear selection".

The selected sketch is highlighted in yellow in the graphics area and the name of the sketch is highlighted in blue in the Features list; folders (when present) are highlighted in gray.
Undoing, Redoing and Restoring

Every open dialog has two options for being closed or dismissed:

- 🔄 - KEEP the actions taken and close the dialog.
- ✗ - CANCEL all the actions taken and close the dialog.

Recovering from accidentally canceling

If you mistakenly cancel the dialog, you do not necessarily have to recreate your work.

If you click the ✗ of a Sketch dialog after making changes, Onshape displays this message:

![Message](https://example.com/message.png)

To reverse the action of clicking the ✗, click the **Restore** link in the message bubble (available for ten seconds). The sketch is restored to the changes made during the last edit session, before you clicked the ✗.

If you began a new sketch and clicked the ✗, clicking **Restore** restores the sketch as it was before you clicked the ✗.

If you take an action you don’t mean to take, you can use the Undo 🔄 icon (in the toolbar) at any time to undo the last action or series of actions, one by one. Use the Redo 🔄 icon (also in the toolbar) to repeat the last series of actions, one by one.

You can also right-click either icon to see the list of actions and select the action to undo/redo up to.
This list contains only the actions each user has taken in their present session. If you need to undo an action taken by another user or one taken during a previous session, use the Versions and history flyout, explained in Document Management.
When you set your preferences in Setting Your Preferences, you had the chance to set up customized shortcut toolboxes for Sketches, Features, Assemblies, and Drawings. Access your personal shortcut toolbox by pressing the 'S' key.

The 'S' key opens the toolbox appropriate for the tab and toolbar active at the time. For example: when you are in a Part Studio with a sketch open, pressing S opens the Sketch shortcut toolbox. When no sketch is open, pressing S opens the Features shortcut toolbox.

If you didn't customize this toolbox, the toolbox opens with the Onshape defaults.

Selecting a tool from the toolbox closes the box. Clicking anywhere outside of the toolbox also closes the box.

You can also customize the tab toolbar at the top of the window in any Part Studio, Assembly or Feature Studio. Right-click anywhere in the toolbar and select Customize toolbar. For more information, see Toolbars and Document Menu.

Additional resources

- Customizing shortcut toolbars
- Sketch Basics
- Document Toolbar and Document Menu
- Sketch Tools
- Feature and Part Lists
- Shortcut keys map
Importing and Editing a Part

*The estimated time of completion is 8 minutes*

Occasionally, you might have a part or assembly from another system that you want to import into Onshape. Onshape allows the import of files from many systems, but note that when moving between any CAD system, the models lose their parametric history. However, Onshape supplies powerful direct editing tools so you can still modify parts after importing them into the system.

This lesson covers how to import a part from another CAD system and modify it with Onshape’s robust direct editing tools.

**Best import format**

You can import many types of files into Onshape, but if you want to edit the part, the most reliable neutral 3D CAD file format to import into Onshape is Parasolid (.x_t). If the system the file originates in does not support Parasolid, STEP (.step or .stp) is the next best format.

**Choosing where to import in Onshape**

There are two options in the Onshape interface for importing a file, each with a different workflow.

1. Import from the **Documents page** Create > Import action to create a new Onshape document for the imported data.

2. Import from **within an Onshape document**, to import the files into that open Onshape document.
To create a new Onshape document to contain your imported data, import from the Documents page:

1. Click the Create button on the Documents page:
   
   ![Create button](https://via.placeholder.com/150)
   
2. Select **Import files...** from the menu.
3. Select one or many files, each file results in a new Onshape document created.
4. Click **OK** when satisfied with your selection.
5. In the Import dialog that appears, choose one of the following, keeping in mind that your choice applies to all the files you selected:
   - Import to a single document (per file you select)
   - Split into multiple documents (split assemblies and parts into different documents, preserving structure - again, per file you select)
   - Combine to a single Part Studio (combine assemblies to Part Studios only (best for small assemblies - per file selected)
6. Choose appropriate options:
   - **Orient imported models with Y Axis Up** - If the models in the file being imported are in this position (e.g. SOLIDWORKS or Inventor files), and this box is checked, they will remain in this position in Onshape. Onshape's up axis is the Z axis. (This option is not available when splitting one file into multiple documents.)
7. Click **OK** to begin import.

The files are uploaded in sequence and translation of the data happens in the background. You can continue working...
in Onshape during this process. Notifications are posted when processes have finished. Notice that you have one new Onshape document for each file that you imported, unless you selected to split assemblies and parts into multiple documents, then you may have more than one file per imported document.
To import one or more files into an existing Onshape document (with no new document creation):

1. Open the Onshape document into which you want to import files.
2. Click (+) (Insert new element icon) at the bottom of the window.
3. Select Import... from the menu.
4. Select one or more files to import and click Open.
5. Select the preferred import option:
   - **Import to this document** - Import the parts to one Part Studio and the assemblies (in each file) to their own Part Studio (per file) and Assembly tabs/per assembly/file. The parts will not be recreated for all the assemblies, but will live in the Part Studio and be instanced into the relevant assemblies.
   - **Combine to a single Part Studio** - Combine the assemblies and parts in the imported file to a single Part Studio in this open document. This is done per file selected and is best suited for small assemblies only.
   - **Orient imported models with Y Axis Up** - If the models in the file being imported are in this position, and this box is checked, they will remain in this position in Onshape. Onshape's up axis is the Z axis. (This option is not available when splitting one file into multiple documents.)

The files are uploaded in sequence and translation happens in the background. You can continue working in Onshape during this process. Notifications are posted when processes have finished. Notice that the document now has a new tab for each file that you imported.
Where is the imported data?

All uploaded files appear either as individual documents in the Created by me filter on the Documents page or within the document into which they were imported.

Regardless of which import method you used, the Onshape documents containing the imported data have a tab with the original file, and then also Part Studios and any other tabs necessary (Assemblies, for instance) to store the parts and assemblies translated from the original file.
If you are still working on the original file within the original system, and want to update the file as it exists in Onshape, you can. To update the data imported into Onshape:

1. Export the file again from the original system. Note the file name and location.

2. Open the Onshape document containing the originally imported data, right-click the tab with the file name, and select Update.

3. The name of the Onshape tab is not changed (even if the file chosen for the update has a different name). The data is reloaded and re-translated into Onshape.
Editing a part with direct editing tools

Onshape offers a variety of tools for editing parts with parametric histories, but sometimes imported parts do not come into Onshape with their history. In these instances, Onshape’s direct editing tools come in handy:

- Change the radius of a fillet or remove existing fillets
- Delete one or more faces, heal it, or leave it open to create a surface
- Move one or more selected faces linearly or about an axis
- Replace one or more selected faces with another face
- Create a new surface by offsetting an existing face, surface, or sketch region. Set offset distance to 0 to create a copy in place.

Additional resources

Importing files
Supported file formats
List of feature tools

How to Import Data (Learning Center)
Direct Editing (Learning Center)

Measuring and Mass Properties

The estimated time of completion is 3 minutes.

The Onshape Mass and Measure tools are automatic and do not require activation.
Click the respective tool in the lower-right corner to display more Mass properties and Measure information.

<table>
<thead>
<tr>
<th>Mass properties tool</th>
<th>Measure information</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select in the bottom right corner of the model space to open the Mass properties dialog. You can preselect a part or select the part once the dialog is open.</td>
<td>Select in the bottom right corner of the model space to open the Measure dialog. You can preselect two points or lines, edges, faces or parts in the model space or make your selections once the dialog is open.</td>
</tr>
</tbody>
</table>

Additional resources

| "Measure Tool" on page 344 | "Mass Properties Tool" on page 349 |

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**Working with Versions**

*The estimated time of completion is 8 minutes.*

Onshape captures the state of every tab in a workspace every time an edit is completed. Every edit, by every user, across every session is captured so there is always a complete history of every action made in the document. In addition to keeping a history of actions taken in every document, Onshape provides a way for you to capture an immutable version of a document at any point in time as well.

This lesson covers how to preserve a particular state of a document at a particular point in time by creating a version, as well as what you can do with a version once it's created.
Before discussing versions, it's important to understand what an Onshape workspace is. Basically, a workspace is where you do your work. Every Onshape document starts with a workspace called Main. Creating parts, assemblies and drawings is all done in a workspace.

The name of the workspace or version you currently have open is displayed in gray next to the document name in the document title bar. Since you can have many workspaces and versions per document, look there to see which workspace or version you are in at any point in time (below, outlined in blue).

Note that you can work in a workspace, but you can only view a version.
All documents start with a version named *Start* and a workspace named *Main*. The Start version is immutable (as are all versions). It is the starting point of every document.

Click the Manage versions and history icon (outlined in blue, below) in the document toolbar to display a graphical representation of the document history:

**Helpful tips**

- **Selections in the Versions and history flyout** are highlighted in blue.
- **Hovering over an entry in the flyout** highlights it in light blue.
- **Click > Show changes to expand the list of all changes made in that workspace or version**

The icons in the top right of the flyout are:

- ![Create a release candidate - Available only for Professional and Enterprise accounts; discussed in another lesson.](image)
- ![Create a version - Create an immutable snapshot of the](image)
workspace at that particular point in time.

Compare history entries - Compare any two entries in the history graph.
Viewing the history of a document

The history of a document is a list of all actions taken on that document during all sessions and by all users. To see the history:

1. Click "History" in the document toolbar to open the Version and history flyout.

2. Click "Show changes" to see a list of changes for a particular version, workspace or release.

Helpful tips

- Each document can contain many branches, one workspace for each branch, many versions, and many releases.
- Changes are identified by lighter text and a dashed line in the graph with a small solid dot for each change.

Hover over a particular change entry to see by whom and when the change was made.

There are a number of actions you can take on any of the history moments (changes) in the expanded list; right-click over an entry to access the context menu:
Create a version

Helpful tip

To create a version from a workspace, with the workspace and the Version and history flyout open:

Click † and fill out the dialog.

Create a version to indicate to the system that you want to preserve the workspace at a particular moment in time. Versions are immutable, meaning you can never change or edit a version in any way. You can create a new branch (and thereby a workspace) from a version.

1. With the Versions and history flyout open, right-click on the change entry from which you want to create a version.

2. Right-click on the change row and select Create version here...

3. Enter information for the version.

4. Click Create.

Note that a new version node is added to the Versions and history graph:

The V1 (version) was created

The changes along the Main (blue) branch are indicated by blue tick marks along the Main branch once the version changes are expanded.
Select a history entry (click Show changes to see a list of history entries) to view the document at that point in time, or right-click on an entry and select View.

The name of the workspace (or version) you are viewing is always displayed next to the document name in the document toolbar. When viewing a version, directly below that is a link to Restore to the top of that workspace: to restore the state of the workspace back to that point in time. There is also a link to Return to the top of the workspace, which simply shifts the view to that of the workspace.
The change labeled "Myspresso-v4Asmb parts:: Change part appearance" is selected and visible in the document, in the "Main" workspace; to return to the end point of the workspace, select the workspace name in the history or click Return to Main at the top of the dialog.
If you are unsure of how your work in a document differs between workspaces, versions, or between a workspace and a version, you can compare them. Onshape allows you to compare two entities at a time: two versions, two workspaces, or one of each.

Initiate a comparison in the Versions and history flyout, using the icon in the top right corner.

1. With the Versions and history flyout open, click.

2. Select any two entries: a workspace, version, or any specific history entry within a workspace or a version.

3. The first selection is highlighted in blue and when you click the second selection, the compare dialog opens:

   The compare dialog is color-coded, changes made in one version are color-coded blue and changes made in the other version are color-coded red

Select a feature in the Features list on the left side to open another dialog listing the Features created or edited and the specifics of the feature:
The feature selected on the left is also highlighted in the model. Use the slider above the model to change the display emphasis towards one workspace or the other to visualize changes. Note the fields shown for the feature are view-only in the Compare flyout.

Restore is like an Undo action on a larger document and user scale. Since all actions, by all users, across all sessions are listed here, you can select a particular action to restore the workspace to. Right-click on any change point in the Versions and history flyout and select *Restore to <name>*:

Rest assured that even a restore action can be 'undone' by restoring to the action taken just before the Restore action in the History.
Working with Branches

*The estimated time of completion is 5 minutes.*

When you work in Onshape, you do your modeling in workspaces. Regardless of whether you are working with parts, assemblies, drawings and imported data, all the work is done in workspaces. Keep in mind that a single document can contain several workspaces and that workspaces within one document are still completely independent from one another. You can think of workspaces as sandboxes, where engineers can experiment with new design variations without fear of disrupting each other's work.

This lesson covers creating a new workspace by creating a branch.

**What do you start with**

Let's recap and go over that every document in Onshape starts with a version named *Start* and a workspace named *Main*. The workspace *Main* is where you'll begin your work, and the version *Start* remains the unchanged starting point, with no data, and permanently in view-only mode. Versions are moments in time of the workspaces, unable to be changed.

Since you can create many workspaces in a document, you can have many sandboxes where engineers can experiment with a main design without disrupting anyone else's work.

Once happy with a design, you can merge it back into another workspace to fully integrate it with the main design for everyone to work with again.
Remember, a branch is not another file, or a copy, it lives in a document and is simply an independent workspace within that document. You can change designs and data in one workspace without affecting any data in any other workspace.

The process of creating another workspace is called branching. This refers to what happens to the Version and history graph when you create another workspace. You create another branch of the graph.

The most important thing to remember is that in order to create a branch, you must first have a version. Branches are created from versions only.

To create a version, simply click the Create version icon in the Document toolbar, outlined in blue, below:

Enter a name for the version and an optional description, click Create. The Version and history graph now shows a new version, V1, in the graph:
Creating more workspaces: branching

Helpful tip

Then, right-click on the version in the graph and select Branch to create workspace. Type in a name and description for the new workspace and click Create.

The new branch, B1, appears in the graph:

Note that branches are color-coded; follow the color of the branch to see its complete history

Upon creating the new branch and workspace, you are put into that workspace, as you can see by the new workspace name next to the document name in the Document title bar.
Working with multiple branches

Now that you have multiple workspaces in your document, it's important to be aware of what workspace you are in, and how to get back and forth between workspaces. The easiest way to see what workspace you are in is to look at the top of the interface. Next to the document name you will see the workspace name (or the version name if you are in a version). Note that if you are in a version, you will not see any toolbars or Feature lists since versions are view-only.

You can also see the active workspace by clicking the “Manage versions and history” icon in the toolbar.

The active workspace will be highlighted in the graph.

So how do you go back and forth between different workspaces? In the graph, simply select the branch you want to open. You can close the graph at any time; it doesn't affect which workspace you are viewing.

Remember, branches are totally independent workspaces within documents, so you can create a version in a branch the same way you could in the original workspace.

Additional resources

Version Manager Branching and Merging (Learning Center)

Merging Branches

*The estimated time of completion is 5 minutes.*

Onshape allows you to merge design changes from one workspace to another using From > To logic. You merge changes *from* any workspace or version of one branch (the source), *to* another workspace on another branch (the target, always a workspace).

This lesson covers what to do once you have branches with designs you approve and want to merge those designs together again in one branch.
Preparing to merge branches

Helpful tips

- All changes made in the source workspace are merged into the active workspace (the target).
- All merge actions are also recorded in the document history (so it can also be reversed if necessary).

When you have a workspace on one branch (what we call a 'source') you can merge it into another branch (what we call a 'target'). The most important information right now is that you will merge into the active workspace. So make sure to have the workspace you want to merge into open and active in the interface.

1. Open the Versions and history flyout.

2. Onshape recommends first creating a version in the workspace you want to merge (the source workspace). That way you always have that moment in time you can go back to.

   The advantage to creating a version is that it won't change if a user continues to work in the workspace.

   With the source workspace open in the interface, click the Create version icon and fill in the details in the dialog.

3. Once a version is made of the source workspace, go ahead and open the target workspace: the workspace into which you want to merge. (To make a workspace active, select it in the Versions and history graph.)

4. With the Version and history graph open, right-click the source version and select "Merge into current workspace".

   All changes made in the source version are all merged into the target workspace and immediately visible in the interface.
An example
Let’s walk through an example of having a design that requires merging a version into a workspace.

Helpful tip
Changes in the source branch overwrite changes in the target branch, even with entities not involving Part Studios and Assemblies (like drawings, images, pdfs etc).

This base and widget were created in the Main branch of a document. A version was created, V1, and a branch created from that version, B1.

In branch B1, the widget design was then modified as shown below:

In branch Main, the base design was modified by adding holes in the corners, as shown below:
To merge the new widget design in branch B1 into the design in Main, we must:

1. Open the workspace of branch Main so it's visible in the interface, and open the Version and history graph as well:

2. Open the Version and history graph and right-click the workspace of branch B1:

3. Select *Merge into current workspace.*

The current workspace, Main, is updated with the geometry in the workspace of branch B1, including the parametric history of the features from branch B1, resulting in this widget design:
This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape is built from the ground up to enable efficient design processes. This section walks you through getting set up in Onshape for the first time, then presents the basic steps to modeling. There are some settings you can make right away to make your modeling more efficient.

**Set it and forget it account settings**

You can set preferences in your Onshape account to mimic the mouse settings of another CAD system. Why not start with a mouse configuration that is already second nature to you? The preferences you set become immediately available in your Onshape account regardless of what device or browser on which you access Onshape.
1. After signing in to your Onshape account, click the user menu in the top right corner of the window to open the User menu.

2. Click My account.

3. On the next page, select Preferences in the left panel.

4. Select your preferred settings, as follows.

   Note that each setting has its own Save button. If you do not click Save for each change made, the change is not registered.

   a. **Units** - Select your preferred units. Keep in mind that this setting will be the default for all numeric fields in all of the documents you create. You can override these
settings in any numeric field simply by typing the units in the field. (The units will be recalculated and then displayed using your default units.)

b. Click **Save units**.

c. **View manipulation** - Select the CAD system whose mouse settings you are most comfortable with. (If you opt to keep Onshape's default settings, see Setting Preferences to learn about Onshape's mouse settings.)

d. Click **Save view manipulation settings**.

For more information

These are the most relevant settings for your account right now, but for more information on additional settings, see Managing Accounts.

**Create a document to hold all your design data**

Think of Onshape documents as multi-data containers. Not only are you able to sketch, create parts, assemble parts, and create drawings all in one document, but you are also able to store non-CAD data in any document as well: images, non-translated CAD data, PDFs and more. Just about any type of file you want to keep track of can be imported into a document.
1. The Onshape landing page (where you find yourself when you sign in) is called the Documents page because it displays a table listing all of the Onshape documents you have access to.

To access this page from anywhere else in the system, click the Onshape logo in the top left corner of the window (or your company logo if you are part of an enterprise account).

2. Click **Create**.

3. Select **Document** from the menu.

4. Give the document a name (in the dialog that opens).

5. Click **OK** to create and open the document.

Documents, by default, contain two tabs when created: Part Studio 1 and Assembly 1. Tabs are Onshape's way of allowing you to store more than one type of data together.
in one place. Look towards the bottom of the window to see the name of the tab. Click a tab to activate it. Each data type is stored in a specific tab type.

**Begin part design with a sketch**

A Part Studio is used to define parts and has a Feature list (parametric history) that, when regenerated, produces solid bodies called *parts* in Onshape.

Part Studio tabs are where you begin creating parts to be later used in assemblies. All parts begin as sketches, using sketch tools, and then, in the same Part Studio, you use feature tools to create solid bodies (called parts) from the sketches. You can create as many Part Studios in a document as you wish, and create as many parts in a Part Studio as you wish. Keep in mind that it's best to limit parts in a Part Studio to only those that are geometrically relevant to each other. For the best performance, parts not geometrically related to each other should be in their own separate Part Studios. So you'll have multiple parts in a given Part Studio only when those parts are geometrically related to each other.

This is similar to multi-body part modeling in other CAD systems, but is much more powerful. One Onshape Feature list has the ability to drive the shape of multiple, actual parts. Each part is able to be instanced multiple times in assemblies and each instance is able to move independently in the assembly.

In a Part Studio, there are two tool sets: [Sketch Tools](#) and [Feature Tools](#).

Use Sketch tools to create sketches, the foundation of parts. Use Feature tools to create parts from the sketches. Each feature is recorded in the parametric history that is the Feature list.

Parts created in Onshape always begin with a sketch. The first tool on the toolbar when you create or open a document is Sketch (next to the Undo/Redo icons).
To create a solid (versus a surface) you need a sketch with closed regions. (Onshape automatically shades all closed regions of a sketch.)

1. Click Sketch.

A sketch dialog opens.

Blue highlighted fields require selection in the graphics area
2. First, select a plane to sketch on (top, front, right).
   You can select the name of the plane in the Features list or the plane in the graphics area.
   Press the N key to orient the sketch plane to normal (if desired).
3. Click a sketch tool in the toolbar; hover on a tool to see a name and tooltip.
   a. For demonstration purposes, select Rectangle. Click it in the toolbar or press the shortcut key (G) to toggle the tool on, and press it again to toggle the tool off.
   b. Click in the white space (the graphics area) to set one corner of the rectangle, and click again to set the opposite corner of the rectangle.
   c. Onshape automatically displays dimension text. The dimension text with the box around it is the active one: type a dimension and press Enter (you can always change it). The other dimension becomes active. Type another dimension and press Enter. Note that this must be done with the sketch tool still active. When you select another tool or toggle this tool off (by clicking the tool icon again or pressing the Escape key) the tool is deselected and the dimension fields are inactive. Simply double-click a dimension to activate it again.
      Dimensions can also be deleted; click once on the dimension and press the Delete key. Note that dimensions are only visible when the sketch dialog is open.
4. Click the checkmark in the corner of the dialog to accept (save) the sketch and close the dialog.

Notice the sketch is listed in the Features list on the left side of the window; by default the name is 'Sketch 1'.

You can rename everything you create in Onshape: in an open dialog, click the name and a pencil appears to the right. Click the pencil to edit the field. Otherwise, you can right-click on the sketch (or feature) in the Features list and select Rename from the context menu that appears. Specify a new name and and press Enter.
For more detailed information about sketching and tools, see "Sketch Basics" on page 383.

You can also take a self-paced learning course on sketching here: [Starting a Sketch](#).

**Create solid parts**

Parts (solid bodies) are created by selecting a sketch region (closed curves indicated by shading). By contrast, surfaces are created by selecting a curve. In Part Studios, you can create more than one part at a time.
To create a part:

1. Select the Extrude tool 📦 in the toolbar.
2. At the top of the open dialog, select Solid.
3. Select the shaded region of the rectangle.
4. Accept the defaults for the remaining fields in the dialog.
5. Click the checkmark ✅ to accept the actions and close the dialog. (To dismiss the dialog without accepting the actions, click ❌.)

To create more than one part, use an additional Extrude feature, select Solid, and then New in the dialog box:
Notice the part listed in the Parts list at the bottom of the Feature list on the left: Part 1. Onshape uses cross-highlighting to help you locate features and sketches involved in a part. Try this: Click the part in the graphics area and see what becomes highlighted in the Feature and Parts lists:

Selecting a face of the part in the graphics area causes the Extrude feature and the Part name to be highlighted in the Features and Parts lists, respectively.

Now try selecting something in the Feature or Parts list and notice what else is highlighted:
Selecting the sketch name in the Features lists causes the sketch to be highlighted in the graphics area.

Each sketch and feature created are stored parametrically in the Feature list on the left side of the window. You can go back and edit any feature or sketch listed in the feature list.

**Shortcuts**

- Use the 's' key to display your shortcut feature toolbar (with the sketch dialog closed). (Remember you can customize this toolbar through your account settings).
- Use Shift+E to open the Extrude dialog.
- Click 🎬 and select **Keyboard shortcuts** to access the list of all keyboard shortcuts within Onshape.

For more information

For detailed information about using Onshape's Feature tools to create parts, see "Feature Basics" on page 681.

You can also take a self-paced learning course on creating parts here: [Starting a Part](#)

**Start part design by importing a part**

You may have a part or assembly from another system that you want to import into Onshape. Onshape allows the import of files from many systems, but note that when imported, models lose their parametric history. However, Onshape supplies powerful
direct editing tools so you can modify parts post-import. You can also use in-context modeling to use the imported model as a reference for the creation of additional parts.

To learn about all your options for importing files from another CAD system, see "Importing Files" on page 1975.

Once you have imported your design, open the document or Part Studio with the model in it and begin using Onshape's direct editing tools to modify it:

For more detailed information on importing, see "Importing Files" on page 1975. For information about in-context modeling, see "Modeling In-Context" on page 1719.

You can also take a self-paced learning course on importing parts here: Importing a Part.
Onshape provides functionality for assembling parts in Assembly tabs and creating drawings in Drawings tabs.

**Assemble parts**

An Onshape Assembly is a tab type that is used to define the structure and behavior of an assembly. Each Assembly has its own Feature list that contains Instances (of parts and sub-assemblies), Mates, and Mate connectors.

An Assembly contains instances. An instance is a reference to either a part defined in a Part Studio, or to a sub-assembly defined in another Assembly. Instead of creating and inserting commonly used parts, like nuts and bolts, you can instance desired standard content and either replicate it in the assembly, or create patterns using the one instance. This keeps the Assembly from getting cluttered and bogged down by too many unnecessary parts.

Mates are used to position instances and to define how they move.

It's important to understand how Onshape Mates differ from mates in other CAD systems. In older CAD systems, mates are low-level assembly constraints, for example, making two planar faces coincident. As a result, positioning two instances usually requires two or three mates.

In Onshape, mates are high-level entities. There is only one Onshape mate between any two part instances, and the movement (degrees of freedom) between those two instances is embedded in the Mate. For example, a single Mate in Onshape can define a pin slot relationship and may also include movement limits as well.

**Create drawings with a bill of materials**

You create drawings directly from a part or Part Studio, or even Assembly in Onshape. Simply select the entity (part name in a Part Studio or an Assembly or Part Studio tab) right-click and select Create drawing. You have the opportunity to select a drawing template, and then the drawing is created within a new Drawing tab in your document. For more information on drawings, see "Drawing Basics" on page 1742.

To insert a bill of materials, click the Bill of materials icon in the toolbar. For more information on inserting a bill of materials, see "Insert BOM" on page 1925.
Organizing data

It is important to remember that Onshape documents are not files; they are containers that can include parts, assemblies, drawings, imported data and basically anything you need for your project. Although you are able to (and sometimes should) have one part per document, we recommend that you keep all project-related data in one document. Anything you plan on reusing across multiple projects should be in its own document. You are able to link from one document to another in order to cross-use data from one document in another document.

For more information on how best to organize your data within Onshape, check out our Onshape Fundamentals: Data Management (opens in new tab) learning pathway.

Modeling in Onshape: iOS

1. Once signed in to Onshape on your iOS device, tap the button in the lower right.

2. Enter a title for the document.

3. Tap OK.

   When you are part of a company, you have a choice to create the document as its owner, or with the company as the owner.

When you create a document, that document is automatically opened. Onshape opens a new Part Studio tab in the document you just created; Part Studios are where you create Parts. See Part Studio for more information on modeling in Onshape.

   The Documents page is the first page displayed upon subsequent sign ins. While on any other page, tap the back arrow in the upper left to return here.
Part Studio

A Part Studio is used to define parts and has a Feature list (parametric history). Regenerating the Feature list produces Parts.

This is similar to multi-body part modeling in other CAD systems, but is much more powerful. One Onshape Feature list drives the shape of multiple parts. Each part is able to be instanced multiple times in assemblies and each instance is able to move independently in the assembly.

In a Part Studio, there are two tool sets: Sketch Tools and Feature Tools.

Use Feature tools to create parts. Each feature is recorded in the Feature list.

Assembly

An Onshape Assembly is used to define the structure and behavior of an assembly. Each Assembly has its own Feature list that contains Instances (of parts and sub-assemblies), Mates, and Mate connectors.

An Assembly contains instances. A instance is a reference to either a part defined in a Part Studio, or to a subassembly defined in another Assembly.

Mates are used to position instances and to define how they move.

It's important to understand how Onshape mates differ from mates in other CAD systems. In traditional CAD systems, mates are low-level assembly constraints like making two planar faces coincident. As a result, positioning two instances usually requires two or three mates.

In Onshape, mates are high-level entities. There is only one Onshape mate between any two instances, and the movement (degrees of freedom) between those two instances is embedded in the mate. For example, a single mate in Onshape is able to define a pin slot relationship.

Drawing

You have the ability to create drawings directly from a part or Part Studio, or even an Assembly in Onshape browser version. These drawings are viewable on the Onshape mobile platforms.
Importing existing CAD designs

Onshape provides an easy way to import your existing CAD files to load those designs into Onshape. For more information on supported file types, see "Importing Files" on page 1975.

Organizing data

It is important to remember that Onshape documents are not files; they are containers that are able to include parts, assemblies, drawings, imported data and basically anything you need for your project. Although you are able to (and sometimes should) have one part per document, we recommend that you keep all project-related data in one document. Anything you plan on reusing across multiple projects should be in its own document. You have the ability to link from one document to another in order to cross-use data from one document in another document.

Modeling in Onshape: Android

1. Tap the button in the lower right.

2. Enter a title for the document.

3. Tap OK.

When you are part of a company, you have a choice to create the document as its owner, or with the company as the owner.

When you create a document, that document is automatically opened.

Onshape opens a new Part Studio tab in the document you just created; Part Studios are where you create Parts. See Part Studio for more information on modeling in Onshape.

The Documents page is the first page displayed upon subsequent sign ins. While on any other page, tap the back arrow in the upper left to return here.
Simple Modeling Example

The small design example below demonstrates how to create a sketch in Onshape, then create a part, apply fillets, drafts, and shell the part. Onshape provides two main methods of creating parts: from a sketch or from an imported part. When you import a part into Onshape, you can use the powerful direct editing tools to modify the part within Onshape. For more information on direct editing tools, see [Delete face](#), [Move face](#), [Replace face](#) and [Offset surface](#).

In addition to Onshape's Feature tool functionality, you can edit a Feature within the Feature list and see the result without regenerating the entire Feature list by using the [Preview slider](#) and the [Final button](#) in a Feature dialog.

The example below shows the basic steps involved in creating a simple tub.

1. Sketch

2. Extrude
3. Fillet

4. Draft

5. Shell

The features are shown in the Feature list in the order they were created, followed by the Rollback bar.

Drag the Rollback bar to see the model at the point when a feature was created, as in this example with the Rollback bar above the Shell feature:
Editing a feature

Edit a feature by double-clicking on it in the Feature list or through the context menu (right-click on the Feature and select Edit). The system automatically rolls back the model display (as shown by the grayed-out and italic Feature names in the list) to before the feature was added and opens the Feature dialog you selected to edit:

Notice the _Preview slider_ and the _Final_ button at the bottom of the Feature the dialog. By default, the Preview slider is at 70%. The image of the model is a blended image of before the feature is added (30%) and after the feature is added (70%). Sliding to the left increases the 'before' image; sliding to the right increases the 'after' image.

This can be very useful in complex models to confirm that you are editing the correct feature or when the feature change is hard to discern.
The Final button shows the impact of the feature on the final visualization of the model (with the later features fully regenerated).

The Final button is especially useful when the feature is early in the regeneration process and it is hard to predict the impact the edit might have on later features. With the Final button clicked, you can see if the final result is what you intended. You can even see if this edit will make a later feature fail to regenerate.

Onshape Documents

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape has created a new document concept within the CAD industry. Instead of creating and working files, Onshape provides documents in which you can create, import and organize not just your CAD data, but also all supporting data and information that is pertinent to your CAD projects.
To be more specific, Onshape documents can contain parts, assemblies, drawings, imported data, images, and more. These types of data are stored in their own tabs within a document. For example, Part Studio tabs are where you do modeling like creating parts. Assembly tabs hold subassemblies and assemblies, Drawing tabs contain drawings, and so on. Onshape doesn't dictate what can and cannot be in a document. You can put many parts and assemblies in a document or just one part or just one modular subassembly if you wish. You can also use parts or assemblies from one document in another document. This latter strategy is referring to as linking documents. For more information, see "Linking Documents" on page 1446.

Some unique aspects of using Onshape documents are:

- **You can sketch, build, and assemble parts (solid bodies) in the same document** - All of your work is able to be done in a single document with complete parametric history.

- **You can keep all project-related information in one document** - Onshape documents can contain any kind of data: sketches and multiple parts (solid bodies) organized in one or many Part Studios, subassemblies and assemblies organized into one or many Assemblies, drawings, and any other type of information or file you want to import (including CAD data from another system). All of these elements are shown in separate tabs in an Onshape document.

- **You can collaborate with many users in one document** - There's no need to copy documents and send them to coworkers or suppliers: share your document with as many other users as needed and collaborate in the same document at the same time.

When a user shares a document with multiple users, all users have the ability to view (and edit, depending on permissions) the same document (even the same parts) simultaneously.

The document owner has the ability to assign permissions to each user for a specific document, and also revoke those permissions at any time. Ownership is determined at the time of document creation and varies by the type of account:

- In Enterprise accounts, documents are automatically and always owned by the enterprise.
In Professional company accounts, documents are automatically and always owned by the company.

All other accounts are single user and each user owns the documents they create.

Users belonging to more than one professional-level account select the owner company at the time they create the document.

Why is ownership important? Owners of documents have every permission on the document by default, including the ability to transfer ownership of that document to another user. Creators of a document in a Professional company or enterprise account have every permission on the document, except to transfer ownership.

**Keeping project information in one document**

You can, if it makes sense for your project, keep all of your project data in one Onshape document if you wish. By default, documents contain a "Part Studios" on page 279 and an Assembly (you may create as many as you like of both in one document). These are the tabs located at the bottom of the browser window when a document is open. When you open a document, the last tab you accessed before closing the document is opened (made active) by default.

Click a document's name on the Documents page to open it.

Sketch and create parts in Part Studios, and assemble those parts in Assemblies. Note that you are able to create many parts in one Part Studio and Assemblies may contain subassemblies as well. In addition to these types of data, you may also import other files into Onshape which will appear each in their own tab, some examples are:

- PDFs
- CAD files
- Images
- Drawings

You can also:

- Duplicate a tab
- Copy a Part Studio and paste it into another document
• Export an entity (sketch, planar face, part, Part Studio, drawing)
• Create a drawing of a particular part or entire Part Studio or Assembly
• Delete a tab
• Control the order of the tabs (drag and drop)
• Organize tabs into folders (drag and drop)

Managing document versions and history

Onshape's "Document Management" on page 2127 model allows branched editing, merging, and the ability to restore a document to any point in its editing history through the Versions and history flyout feature.

You can, at any milestone, create a read-only version of the document. If you need finer control within a document, for example over a single part or subassembly, you can split the part or subassembly out into separate documents, then reference (link) them back to the main project design document. For more information on linking documents, see "Linking Documents" on page 1446.

You can, at any time, revert a document to a previous point in its history; every action made in a document is saved in the history of the document. You can also review a point in a document's history before restoring to that point. Easily reverse the action since the entire history is always available.

Collaborating with other people

Onshape is designed specifically with collaboration in mind. Depending on the type of account you have, documents you create may be:

• Private - Owned, visible and editable only by you
• Shared - Private documents that you enable other users to view, edit, or edit and share
• Public - Documents you make available for viewing and copying by all Onshape users

Sharing and permissions may be reversed; all documents that are shared may be unshared and all documents you make public may be made private again (except for Free accounts, those documents are always and only public).
You also have the ability to delete documents you own, restore them from Trash, or permanently delete them from Trash.

For more information on sharing and collaborating, see "Sharing and Collaboration" on page 2036.

**Best practices and strategies**

Until you settle on a strategy for organizing your data within Onshape that suits your business needs, we have a few tips:

- Start off using each document as a container for any and all data related to a single, unique design project. Anything that you plan on reusing across multiple projects should live in its own document.

- Fewer parts in a single Part Studio means easier management and better performance. Before creating a second part in a Part Studio, ask "are these parts geometrically related to each other?" If the answer is no, build the second part in another Part Studio. Remember you can add as many Part Studios as you want in the same document (same with all types of tabs).

- Use single Part Studio for multiple parts when the parts have a high degree of geometric interdependency.

- Standard and purchased parts should be linked in to documents as needed and not recreated.

**Creating a document: Desktop**

The home page in Onshape is the Documents page. This page (after you sign in to your account) lists the documents that you have access to, including documents you have created as well as those that have been shared with you, and all documents that have been made public.

Click **Create > Document** (shown below) to create a new document.
Click the Onshape logo in the top left corner of the browser window (anywhere in the user interface) to return to the Documents page.

**Creating a document: iOS**

The home page in Onshape is the Documents page. This landing page (after you sign in to your account) lists the documents that you have access to: documents you have created as well as those that have been shared with you, all documents that have been made public, and Onshape's own Tutorial & Samples documents.
Tap the New Document button in the lower right to create a new document.

**Creating a document: Android**

The home page in Onshape is the Documents page. This landing page (after you sign in to your account) lists the documents that you have access to: documents you have created as well as those that have been shared with you, all documents that have been made public, and Onshape's own Tutorial & Samples documents.
Tap the New Document button in the lower right to create a new document.

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**Document Tabs**

This functionality is available on Onshape's browser, iOS, and Android platforms.

As explained in "Onshape Documents" on page 115, Parts Studios, Assemblies, and non-native files imported into Onshape documents are represented in tabs in the user interface. Click a tab to select it and make it active. Only one tab is active in one browser tab at a time, but you can open any tab in a new browser tab to have multiple active tabs open. In addition to working in a tab, you can perform actions on tabs. Right-click on a tab to access the context menu for that tab.
Creating more tabs in a document

Click the plus sign (Insert new) in the tab bar to create a new Onshape tab:

- **Import** automatically creates a new tab for that file, see "Importing Files" on page 1975
- **Create folder** creates a folder into which you can move tabs in order to organize them, see "Organizing tabs with folders" on page 129
- **Create Drawing**, creates a new empty Drawing tab, see [Create a drawing](#)
- **Create Assembly**, creates a new empty Assembly tab, see [Create an Assembly](#)
- **Create Part Studio**, creates a new empty Part Studio tab, see [Create a Part Studio](#)
Create Feature Studio, creates a new empty Feature Studio, see Create a Feature Studio

Create Material library, creates a new Material library for use in Onshape, see Creating new material libraries

Add application, lets you select an application for use in the document; one that you have already added through the App Store, see "App Store FAQs" on page 2613 for more information

Working with tabs

When working with a lot of data, the number of tabs you start amassing may get a bit unwieldy. Onshape provides a Tab manager to help you organize and find tabs more easily.

To the left of the Insert new icon on the tab bar is the Toggle tab manager icon. The Tab manager opens a flyout on the left side of the Onshape window (moving to the right any Assembly list or Feature list, depending on the type of tab you have open at the time):
The tab manager lists all of the tabs you have in the document.

**Anatomy of the tab manager**

The tab manager has icons at the top to filter the tabs by. The resulting list of tabs can then be ordered according to Tab order in the document, alphabetical by name, or by
type. There is a search box at the top to enter the name of the tab you are looking for, if you know it:

Use the icons at the top of the tab manager to control the search to whatever is selected (remember that selection is additive):
- Search tabs field - Toggle the Search field open and closed.

- Tab order selection - Select the order in which to list the tab names: A-Z or Z-A, or by type of tab.

- Display filter icons - Toggle the tab type icons visible or not visible: Part Studio, Assembly, Drawing, File (only icons for the tab types found in the document are displayed).

- List view - Display tab names only.

- Detail view - Display tab names and details.

At the bottom of the tab manager is a thumbnail view of the currently selected tab.

There are many ways to use the tab manager:

- Use Ctrl+spacebar to view previously opened tabs’ thumbnails. Keep Ctrl depressed and repeatedly press spacebar to proceed through the previously opened tabs. Release both keys to open the tab whose thumbnail is currently active.

- Click and drag tabs in the tab manager list to reorder them in the document.

Keep in mind that tab order is shared among all users in a workspace, and is persistent. For example, if User-1 changes the order of the tabs, User-2 will also see the changes when the workspace is open.

- The active tab and scroll state is not shared, nor persistent. Each user collaborating on a document has their own active tab and their own tab scroll state.

- When a workspace is opened, the active tab is the previously active tab. (When a workspace is opened for the first time, the active tab is the first tab in the series.)

- A newly created tab is placed directly to the right of the currently active tab and is made active immediately.

- When scrolling through tabs, the active tab is always kept in view.

- You can use Ctrl+click to select more than one tab in the tab manager.

Hover over a tab in the tab bar to see a thumbnail of the contents.
Organizing tabs

Moving tabs to other documents

You have the ability to move tabs from one document to another (new or existing) through the Move to document command on the context menu. If moving to a new document, the document is created during this operation.

Whatever is moved creates a link between the moved entity and the original document. Keep in mind that the moved entity is removed from the original document entirely.

When moving an Assembly tab, the Assembly tab and the Part Studios from which the part instances are referenced will all move to the new document.

Move action will be prevented if it would result in a document with no tabs.

Moving a tab to a shared document requires Edit and Link permissions, and moving a tab from a shared folder requires Edit and Copy permissions. (Shared folders are those owned by a company that you are not part of, but are shared into.)

Moving a drawing to a new document without related tabs

If you have a document with a Part Studio and a drawing, and you want to share just the drawing with a vendor or other third-party, you can move the model out of the document, leaving just the drawing:

1. Create a drawing of your model.
2. Create a version of the model.
3. Right-click on the drawing tab and select 'Change to version' so the drawing now references a particular version (instead of the workspace).

Now that the drawing is linked to a version of the model, you can move the model (Part Studio or Assembly) to another document without moving the drawing (if you wish) or move the drawing to another document and it will not include referenced tabs. This leaves you with a document with just the drawing, which you can safely share with a third-party or other user. Keep in mind that the drawing is now versioned and will not indicate if changes are made to the model. You can, of course, right-click on the drawing (in the new document) and select Update linked document and select a newer version of the document to which the drawing is linked.
To derive a drawing into another document on its own, while keeping a linked copy in the master file (so the drawing is in two places):

1. Right-click the Drawing tab and select Duplicate. Now you have two drawings.
2. Move one drawing out of the document to the document you want to share with someone. Both drawings are linked to the original document. However, the duplicate drawing will not be updated if changes are made to the original drawing - you can update the duplicate (and moved) drawing to a newer version of the linked document, if desired (as described above).

**Organizing tabs with folders**

Organize tabs with folders on the Tab bar. Use the + menu to Create a folder:

1. Select Create folder from the menu.
2. A new folder tab appears on the Tab manager, with the name field active.
3. Supply a name for the folder.
4. Drag and drop tabs onto the folder to place them in that folder.

When a folder is active, the Tab bar displays only the tabs in that folder, all other tabs are represented by the All tabs icon 🏡:

![Tab bar with a folder and other tabs]

Select the All tabs icon 🏡 to display all tabs again.

You are able to nest folders: drag and drop a folder onto another folder.

Use the context menu on any folder to act on that folder, including:

- **Rename** - Edit the folder name
- **Move to parent folder** - Move the folder and its contents to the parent folder, if present (tabs and folders within the folder remain within)
- **Unpack folder** - Move any tabs or folders within the folder to the parent folder and delete the folder

These actions are also available through the Tab manager. For more information on folders, see "Documents Page" on page 140.
Acting on tabs

Depending on whether you are in a version or in a workspace, the actions available to you for any particular tab are different.

**Versions:** Using the context menu of a tab, you have the ability to:

- Open in the tab in a new browser tab
- Access the Properties for the tab, including Description
- Display the FeatureScript code for Part Studios only (view only)
- Export the tab

**Workspaces:** Using the context menu of a tab, you have the ability to:

- Open the tab in a new browser tab
- Rename the tab
- Access the Properties for the tab, including Description
- Display the FeatureScript code for that tab (for Part Studios only, in view mode)
- Create a duplicate (copy) of the tab, these tabs are not associative in any way
- Copy the tab to the clipboard, and then paste it into another document using the menu
- Select (this tab) as the document thumbnail
- Move the tab to another document (see above)
- Export the tab
- Delete the tab, even if it is the currently active tab

Right-click on a tab to access a context menu:
Deleting tabs

To delete a tab, simply right-click on the tab you wish to delete, and click Delete:

If deleting the tab might cause conflicts, you will see this warning:
Click Delete to finalize the deletion, or click Cancel to close the dialog without deleting any tabs.

If the tab is able to be deleted without causing any conflicts it will simply be deleted, with no warning, upon clicking Delete.

**Searching and grouping tabs in a document**

Click 🕵️ to open the Tab manager. The toolbar and Feature list (or Parts list in an Assembly) move to the right and the Tab manager opens.

When you are in a version with released objects, the Tab manager has a Released filter 🔄 as well:

You can:
Enter a partial or complete tab name to find an existing tab. (Onshape employs a type-ahead feature for your convenience.)

Use to view the tabs as a list, as shown above, with the tab icon indicating the type of tab and a thumbnail preview.

Use to view the tabs in detail view, shown below, with each line item including a thumbnail preview.

Use the tab type icons to filter by that tab type. Use the Clear button to turn filters off. (Filters appear when their specific entity is present in the document workspace or version (Part Studio, Assemblies, Drawings, and Releases, for example.)

With the tab filters off, folders become visible and actionable through a context menu, specifically, you can:

- On a folder:
  - **Rename** - Edit the folder name
  - **Add selected to a folder** - Create a new folder immediately and add the selection to it
  - **New folder** - Create a new folder inside the selected folder
  - **Unpack folder** - Move the contents of the selected folder to the parent and delete the folder

- On any item in the list:
  - **Add selection to folder** - Create a new folder immediately and add the selection to it
• Click on a tab in the list to open it.

• Use Ctrl-click to select multiple tabs in the list, then use the context menu to:
  • Move those tabs to another document
  • Add selected tabs to a folder (creating a folder if no folder exists)

• Use Ctrl-click to select multiple items in the list, then use the context menu to act on those items, including delete multiple tabs.

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**Document Toolbar and Document Menu**

This functionality is currently available only on Onshape's browser platform.

This functionality is also available on iOS and Android in a limited form.

The Document toolbar is available in all Onshape documents (aligned with the Onshape logo). Below is a map of the functionality available:
1. Onshape or company logo - Click the Onshape logo (or your company logo if you are part of an enterprise account) to return to the Documents page from anywhere else in the system.

2. Document menu - Click this to open a menu of commands available for the document and workspace that is currently active.

3. Manage version and history - Click this to open the "Version Manager" on page 2138 flyout where you can branch, merge, create new workspaces, and view changes and releases.

4. Create version - Click to open the Create new version dialog.

5. Document name (bold) - The name in bold is the document you are currently working in.

You are able to change the document name by clicking on the current name and typing a new one.

6. Workspace name (light gray) - The name in light gray is workspace you are currently working in.

7. Comments - Click to open the Comments dialog for the active document.

8. FeatureScript notices - Click to see any notices sent by the FeatureScript compiler.

9. App Store - Click to access (and sign in if necessary) the Onshape App store for purchasing apps that partner with Onshape.

10. Learning Center - Click to access the Onshape Learning Center (in a new window tab) that offers recorded webinars, technical briefings, and self-paced learning pathways.
11. **Share** - Click to open the Share dialog and make the active document accessible to another user or team and select permissions for those users to have on this document.

12. **Help menu** - Open the Help menu to access: the online Help system, Learning Center (same as the Learning Center button), FeatureScript documentation, a listing of product keyboard shortcuts, a list of What's New for this release, forums, contact Support, check your system for compatibility with Onshape's requirements, and information about this version of Onshape.

13. **User account menu** - Access your account preferences and other settings through this menu. For more information see, "Managing Your Onshape Plan" on page 2398.

**Document menu commands**

The Document menu contains the following commands:

- **Rename a document** - Provide a new name for the document.

- **Document properties** - Enter a description for the document; this description displays on the Documents page, in the Detail panel. If you have access to **Release management** and have Onshape's workflow enabled, this box also includes a Not revision managed checkbox to indicate that this document and its objects not be included in any Release management workflow. You can also assign a category for the document here.

- **Copy workspace** - Create another document with a copy of this workspace in it. The original and the copy are not linked in any way. The copy is opened automatically to the same tab that you were on in the original document.

- **Copy version** - When you are viewing a specific version of a document, this command appears in the menu. Create another document with a copy of the currently open version in it. The original and the copy are not linked in any way and no release metadata is included in the copy (except for Part numbers, if they exist in the source version).

- **Workspace / Version properties** - When you are viewing a specific workspace or version of a document, the appropriate version of this command appears in the menu. View and/or edit the available properties (including a list of tabs, parts...
within the tabs, descriptions, part numbers, revision numbers, states, and more). If Release management is enabled, and automatic part number generation is also enabled, there will be an icon to automatically generate a part number for each part.

- Workspace units - Set the default units for the open workspace. Default units set for a workspace affect all Part Studios and Assemblies in that workspace, all values displayed in sketch dimensions and all other numerical fields (for example, in all feature dialogs). You can set default units for all documents (and workspaces within a document) created through your account in your user profile. Note that the default unit setting has no effect on imported files.

- Print - Print the graphics area of the active tab.

- Close document - Close the document.

---

**Setting Default Units per Workspace**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape defaults to inch/degree for units of measure for all documents and workspaces; this encompasses all measurements in Part Studios and Assemblies, all values displayed in sketch dimensions, drawings, and the default input units for all features as well. (These default units do not affect any external files you import.)

You can specify a different unit of measure in any numeric field and the value will be converted to the default unit automatically. For example, if the default unit is inches, you can still specify a different unit type (for example "10mm") in a numeric field and it will be converted to the proper value in inches.

Default units can be set on an individual workspace basis, through the Document menu, but also for all documents you create, through your account preferences. This topic covers setting default units per workspace.
For information on setting your account preferences (and default units there), see "Managing Your Onshape Plan" on page 2398.

**Setting default units per workspace: Desktop**

1. Open the Document menu and select Workspace units to set the units of measurement and precision used in this Onshape workspace, unless specifically overridden in a dialog (by entering units of choice).

2. Make edits.
Workspace units default to the Units settings on the Accounts > Preferences page, unless overridden in this dialog. These settings encompass all measurements in Part Studios, Assemblies, and Drawings; all values displayed in sketch dimensions as well as the default input units for all features.

New workspaces created from a version inherit the settings of that version.

The decimal place settings:

- Are currently available on browser only
- Are currently applied to the feature dialogs, sketch dimensions, and manipulator dialogs
- Work with the Measure tool and Mass properties tool
  - The Measure tool will display values in scientific notation when the display precision is not sufficient.
  - The Mass properties will display error in measurement, see "Mass Properties Tool" on page 349 for more information.
- Impact the display only; values are rounded internally
- Are not used for computation
- Are used internally to determine the number of decimal places to display, regardless of how many places are entered; if more than the specified number are entered, they will be visible when the field is selected for edit.
- Do not affect any external files imported

3. Click ✓ to save changes, or ✗ to exit without saving.

Setting default units per workspace: iOS

From within a document:

1. Tap the More icon in the upper right corner.
2. Tap the Units icon.

3. Tap to set your desired default length, angle, and mass units of measurement.

Setting default units per workspace: Android

From within a document:

1. Tap the More icon in the upper right corner.

2. Tap the Units icon.

3. Tap to set your desired default length, angle, and mass units of measurement.

4. Tap Save.

Documents Page

This functionality is available on Onshape's browser, iOS, and Android platforms.

The Documents page is the first page displayed when you sign in to your Onshape account. The exception is that Enterprise subscription users will land on the Activity page when you sign in. When on any other page in the system, you can click the Onshape logo (or your company logo) to return to the Documents page.
This page lists all of the Onshape documents you have any permission to, including just view permission. All Onshape documents have permissions that are applied automatically upon creation of the document. There’s a great deal of flexibility in assigning and revoking permissions, so be sure to check out these topics for details: "Share Documents" on page 2045 (for everyone) and "Understanding and Administering Project Roles and Permission Schemes" on page 2276 (for Enterprise users and administrators).

This topic explains the anatomy of this Documents page, explains the tasks you can perform here, how to navigate it and gives a short explanation of using this page to organize and then locate your data using labels, folders, and in some cases projects, and through filters and search functionality.

The page layout is explained below, along with how to navigate and use the specific tools on the page. The platform-specific dropdowns present how to locate the functionality on each platform user interface, as they may differ slightly from each other.

**Document page layout**

This section presents the basic functionality and constructs of the page and the platform-specific topics below present how to use the functionality for each platform type.

The Documents page is customizable to help you view available documents with ease. When you select a filter and layout preference, Onshape displays this page that way until you change them again. For example, if you select the List view and the My Onshape filter, the next time you access the Documents page these settings will still be selected.
Documents page with a document selected and the preview visible in the Details flyout

Document thumbnails are of the last Part Studio opened, but you are able to customize this from within the Part Studio as well. The document name is in bold and the last opened workspace is in light grey beside it. The numerals below explain the matching areas of the Documents page pictured above:

1. Onshape supplies various filters to sort documents into groups. The selected filter also appears above the list of documents (to the right of the Create button). For information on filters, see "Using Onshape Filters" on page 174.

2. Regardless of which filter is selected, the top of the documents list displays those documents most recently opened by the current user (Last opened by me). This section is collapsible.

3. Buttons for:

   - **Share** - Open the Share dialog for the currently selected item (document, folder, or project). Use the Share dialog to assign permissions with which another user can access the selected document.

   - **Trash** - Send the selected document to trash, removing it from the document list. Trash is automatically deleted every 30 days, or whenever an admin wants to empty it; until then you can restore documents from trash back into the document list.
Label - See the labels assigned to the selected document, or create a new label. You can also use the Create button to create labels. For more information on labels, see "Creating and Using Labels" on page 159.

List view - Display the document list in list view (shown above).

Tile view - Display the document list in thumbnail tile view.

4. List of projects (for Enterprise accounts), folders, and documents. For more information on folders, see "Creating and Using Folders" on page 162. For more information on projects, see "Understanding and Administering Project Roles and Permission Schemes" on page 2276.

5. The Create button has multiple options:

Document - Click to create and name a new Onshape document. You are also able to create a new document using cad.onshape.com/new.

Folder - Click to create and name a new Onshape folder.

Import files - Click to import external files into Onshape.

Import from - Click to import files from Google Drive, Dropbox, or Microsoft OneDrive into Onshape.

Project - Click to open the Create new project dialog box, where you are able to specify the name, description, and permission scheme of your project, as well as who has access to it. This option is available for any user who has been granted the Global permission that enables this functionality.

Label - Click to create and name a new Onshape label.

The Create button is available for all users; however, when an enterprise user has read-only access to the folder in which they are trying to create, this button is inactive.

6. The Details flyout displays more details about the document, has a link to click to open the document, and can be toggled off with the Details button. Click this
icon to toggle the flyout open and closed.

Click the Share details icon to display the permissions on the document and the folder or project (if any) it resides within.

Click the versions and history icon to display the versions and history of the document. This panel allows you to perform all actions available through the Versions and history flyout in a document, right from this panel. For more information on how to use the Versions and history graph, see Document Management. Note that you can hover over any entry in the graph and see the description as entered into the properties for that Version. For more information on properties, see "Version Manager" on page 2138.

When searching on Parts, Part Studios, Drawings, or Assemblies, specifically, the Detail pane for a selected document will include a Properties icon:

![Search box, above](image)
Note that any Categories added to the company or enterprise are also available in the Add criteria selection box in the Advanced search dialog.

Click this icon to view Properties information. Click the Edit link at the top of the pane to edit properties.

Also, if the search criteria and results include any revision history information (such as State), the Detail pane for a selected document will include a Revision history icon 🔄:

Click this icon to view Revision history information. The revision of the selected part (in the search results) is highlighted in blue in the Revision history panel. For example, the selected "Part 1" on the left below is highlighted on the right, under Revision history:
**Actions**

You can use the arrow keys to navigate up and down the list of documents.

To multi-select documents:

- Use Ctrl-click to select more than one document at a time.
- Use Shift-click (or Shift-arrow) to select a range of documents.

When a document is highlighted, you have the ability to use the Delete key to move the document to Trash. You are also able to use the context menu to access more commands:

- **Open** - Open the document.
- **Open in new browser tab**
- **Versions and history** - View a list of all the versions of a particular document. For more information, see [Document Management](#).
- **Labels** - Select labels to apply to the selected document(s).
- **Share** - Share a private document with other users, and assign permissions per user. Send an email link to your document to any person. If they are an Onshape user, they can click the link and open the document in Onshape. If they are not yet a user, the document opens in View-only mode. For more information, see "Share Documents" on page 2045.
• **Move to** - Move the selected document or folder to a folder. You can also use drag and drop for this action.

• **Transfer ownership** - Transfer the ownership of the document to an individual or a company.

• **Rename document** - Provide a new name for the document.

• **Copy workspace** - Make a copy of the document default workspace. Copies are placed in the same folder as the document being copied, if you have Edit permission on the folder. If you do not have Edit permission on the folder, the copy is placed at the root level of your Onshape account which is **My Onshape**.

• **Details/Hide details** - Show or hide the Details flyout.

• **Make public** - Available only for private documents; makes the document publicly available to all Onshape users in view-only mode.

• **Send to trash / Send selected to trash** - This option changes depending on selection of documents:
  - **Send to trash** - Move the single currently-selected document to trash.
  - **Send selected to trash** - Move the multiple-selected documents to trash.

You are also able to use drag and drop to delete items.

> Once in Trash, a document can be Restored or Deleted (individually), or you can use the Empty trash button to delete all documents from Trash at once.

• **Remove from recently opened** - Remove the document from your Recently opened filter.

> The actions available to you may change based on permissions.

**Selecting documents and opening documents**

Selecting a document results in the document being highlighted in the list of documents and when the Details flyout is open, the Details of that document are displayed. Click anywhere in the tile or the table row to select a document. The background of the document name turns blue when the document is selected.
By contrast, to open a document, click the actual document name either in the table or the tile. When you have the cursor positioned correctly, the document name is underlined.

**Thumbnail images and Detail pane size**

To change the thumbnail that is displayed, open the document, right-click on the tab you wish to use as the thumbnail and click *Select as document thumbnail*. Keep the following in mind:

- If the tab selected is deleted or moved to another document, the thumbnail reverts to the most recently opened tab.
- Edit permission on the document is required in order to select a new thumbnail.

To adjust the size of the Details flyout and the documents list, place your cursor on the splitter bar between them, then click and drag either left or right. Keep in mind:

- The Details flyout can be resized no smaller than 340px.
- Once the Details flyout is resized, the new position of the splitter bar will be maintained when the flyout is opened again, and when the you sign in again.

While in the *My Onshape filter*, the top of the window displays thumbnails of the *last few documents opened* by you (regardless of ownership). As many thumbnails are shown as will fit in the window at the time.
**Searching**

Use the search bar at the top of the Documents page to search for a document within the active filter, as stated in the search bar:

```
Search in My Onshape
```

Use the down arrow in the Search bar to activate an advanced search menu to search for an Onshape tab type (Document, Part, Part Studio, Assembly, Drawing, or all types) within the active filter and refine the search using additional relevant search criteria.

For more information on what additional search criteria may be used, see the topic [Advanced search](#).

When using the search box to search for a document, the search is conducted within the currently selected filter.
Notifications

The upper-right side of the title bar includes a bell icon indicating notifications (when there are notifications). A number beside the icon indicates unread notifications. Notifications are automatically deleted after 14 days, whether they are read or not.

Notifications may include:

- **A comment on the document** - Open to read and mark as read
- **Status of an upload** - Open to read (mark as read, or delete)
- **Status of an export** - Open to read (mark as read, or delete)
- **Workspace status** - If the workspace falls out of date with the most current Onshape release, it is noted here. An out of date workspace could be the result of:
  - Branching from a version created before the last update
  - Restoring to a version created before the last update

  Information about updating is displayed.

Documents Page: iOS

The Document page on the iOS platform has the same functionality as the web platform. For example, you can click the Details icon at the right end of a document listing to open the Details pane:
To close the Details pane, tap the X in the upper right corner.

Tap the Share details icon to display the permissions on the document and the folder or project (if any) it resides within.
Tap the versions and history icon 🔄 to display the versions and history of the document, and also opens the document. This panel allows you to perform all actions available through the Versions and history flyout in a document. For more information on how to use the Versions and history graph, see Document Management.

Tap the Edit icon (the pencil icon) at the top-right of the pane to rename the document.

Tap the History icon to open the document, and also open the History pane for that document.

Tap the Copy icon to make a copy of the document; you are prompted to name it, and the copy opens automatically.

Tap the Move icon to select a folder to which to move that document.

Tap the Trash can icon to move the document to trash.

Creating a new document

To create a new document:

Tap the button in the lower right.

Enter a title for the document and select either Private or Public.

If you are a member of a company then when you create a new document, you can select a document owner. If you are a member of a company, the default owner of a new document is the company that you are a member of. If you are not a member of a company, the default owner of a new document is you.

Import

To import a file into Onshape:

Tap the button in the lower right corner, to the left of the new document button.
If you are a member of a company then when you import a new document, you can select a document owner. If you are a member of a company, the default owner of a new document is the company that you are a member of. If you are not a member of a company, the default owner of a new document is you.

For more info, see Import.

**Account settings**

To access your Onshape account settings:

Tap the icon in the upper left to access the navigation drawer. Here, you are able to select Account settings and then select from:

- **Profile** - View your name, username, nickname, edit your bio, and view the companies and teams you are a part of.

- **Emails** - View the email addresses you have associated with your Onshape account.

- **Preferences** - View and set the default units for Length, Angle, and Mass.

- **Subscription** - View your Onshape subscription type.

- **Teams** - View the teams you are a part of.

- **Companies** - View the companies you are a part of.

**Sign out**

To sign out of Onshape:

Tap the icon in the upper left to access the navigation drawer. Here, tap **Sign out**.

**Feedback and Help**

To provide feedback, watch videos, and access the help:

Tap the Help icon in the upper right.

From here you are able to tap to access:

- **Help** - the Mobile Help.

- **Videos** - Videos showcasing specific Onshape capabilities.
• **Contact Support** - Ask a question or provide feedback to Onshape in the form of logging a bug or requesting an improvement.

• **About** - See what version of Onshape your device is running and view Terms and Privacy.

**View document info panel**

To view the document information panel and access things like the document history, versions, etc:

Tap the icon to the right of the document.

From here you are able to:

• **Access the document versions** - see [Version Manager](#) for more info.

• **Access the document history** - see [Version Manager](#) for more info.

• **Copy, delete, and share the document** - see [Share](#) for more info on sharing documents.

• **Add or edit the document name and document description** - tap the icon in the upper right of the document info panel to edit the document name:

  ![Edit Document Name](#)

  Tap **Add description** to add or edit the document description.

Note that if you are not the owner of a document you will not be able to delete it. However, if you have a document shared with you, the option to remove yourself from the document is available. Tap **Remove** to remove yourself, or unsubscribe, from the shared document.

**Documents Page: Android**

The Documents page is the first page displayed upon sign in. While within a document, tap the back arrow in the upper left corner to return here.

To create a new document:

Tap the button in the lower right.
Enter a title for the document and toggle on Public, or leave the toggle off for Private.

If you are a member of a company, when you create a new document you have the ability to select a document owner. If you are a member of a company, the default owner of a new document is the company that you are a member of. If you are not a member of a company, the default owner of a new document is you.

**Import**

To import a file into Onshape:

Tap the Plus sign button in the lower right corner of the Documents Page, and then the Import button.

If you are a member of a company, when you import a new document you are able to select a document owner. If you are a member of a company, the default owner of a new document is the company that you are a member of. If you are not a member of a company, the default owner of a new document is you.

For more info, see Import.

**Account settings**

**Share Notifications**

To turn on Share Notifications (a push notification whenever a user shares a document with you):

Tap **Settings** at the bottom right. Select **My Account** and toggle Share Notifications on (box checked) or off (box empty).

**Comment Notifications**

To turn on Comments Notifications (a push notification whenever a user comments on a shared document):
Tap **Settings** at the bottom right. Select **My Account** and toggle Comment Notifications on (box checked) or off (box empty).

**Sign out**

To sign out of Onshape:

Tap the icon in the upper left to access the navigation drawer. Here, tap **Sign out**.

**Feedback and Help**

To provide feedback, watch videos, and access the help:

Tap the Help icon in the upper right.

![Help icon]

From here you are able to tap to access:

- **Help** - the Mobile Help.
- **Videos** - Videos showcasing specific Onshape capabilities.
- **Contact Support** - Ask a question or provide feedback to Onshape in the form of logging a bug or requesting an improvement.
- **About** - See what version of Onshape your device is running and view Terms and Privacy.

**View document info panel**

To view the document information panel and access things like the document history, versions, etc:

Tap the icon to the right of the document.
From here you have the ability to:

- **Access the document versions** - see [Version Manager](#) for more information.
- **Access the document history** - see [Version Manager](#) for more information.
- **Copy the document** - make a copy of the document.
- **Move the document** - see [Folders](#) for more information on moving documents.
- **Delete the document** - delete the document.
- **Remove yourself from document** - see [Share](#) for more information.
- **Share the document** - see [Share](#) for more information.
- **Add or edit the document name and document description** - tap the icon in the upper right of the document info panel to edit the document name, tap the icon to the right of Description to add or edit the document description.

Note that if you are not the owner of a document you will not be able to delete it. However, if you have a document shared with you, the option to remove yourself from it is available.
Creating and Using Labels

This functionality is currently available only on Onshape's browser platform.

Creating and using labels

Onshape provides a way for you to group and label documents to better organize them. You have the ability to label individual documents or groups of documents at a time. You also have the ability to relabel documents, apply more than one label to a document, remove a label from a document, and add and delete labels at will. All labels are user-specific, and no other user sees the labels you apply to documents, regardless of permissions.

You may also create folders to physically group documents. Share permissions are applied to the folder (top-level folders only) and all documents and sub-folders within a folder inherit those Share permissions. For more information about folders, see "Creating and Using Folders" on page 162.

You have two options for creating labels:

- Use the Label command in the Create menu to create a label without immediately applying it to a document. (Shown below.)

- Use the Labels icon located near the Details icon to create a label for selected documents. (Shown below.)
1. Click either Create > Label, (shown above) or the label icon in the top right of the window and then Create new label:

![Create new label dialog box]

2. Enter a name for the label and click Create.

The label appears in the filter list on the left side of the window.

**Applying labels to documents**

You can apply one or more labels to one or more documents at a time:

1. In the Documents list, select one or more documents. Use Ctrl-click to select more than one document.

2. Click the label icon in the upper-right of the window.

3. Select which labels to apply to the document(s) by checking the box next to the desired labels:
4. Click the X in the upper-right of the dialog to save the specifications and close the dialog.

5. Notice the label appear below the document name in the Detail pane (as shown below).

Use the Search box in the dialog to search for the label you want when applying labels to documents. To locate documents with a particular label, select the label filter on the left side of the window. You are also able to drag and drop selected documents onto an existing label to apply that label.

Filtering by labels

To locate documents by label, select the label in the Document filter list:
The breadcrumb at the top of the Documents list displays the current filter selection (or folder path). The documents list is refreshed to list all documents with that label applied.

Deleting labels
Deleting a label has no effect on documents, except to remove the label from them:
1. Select the label in the Documents filter list.
2. Right-click and select Delete label.
3. In the message that appears, click Delete.
You can remove a label from a document by selecting the document and opening the Label dialog (click the Label icon in the toolbar) or right-click the document and select Labels... from the context menu. This opens the dialog with a list of labels: click the checkbox to remove labels from, or apply labels to, that document.

Labels: Mobile devices
On mobile devices, Android and iOS, you currently cannot create labels. You can, however, see labels on the home page that were created via the Onshape browser platform.

Creating and Using Folders
This functionality is available on Onshape's browser, iOS, and Android platforms.
Create folders for organizing documents and also for applying Share permissions on all documents with the folder and within any sub-folders. Note that all Share permissions on a folder are inherited by the documents and folders within that folder.

To learn more about organizing documents using folders, you can follow the self-paced course here: Creating Folders (opens in new tab).

To create a folder, on the Documents page:
1. Select **Create** and then **Folder**.

![Create and Folder options](image)

1. Specify a name for the Folder.

2. Click **Create**.

Notice the blue message at the top of the page:

Folder created successfully in My Onshape

You can use Share on a folder the same way you use Share for documents.

**Viewing folders and their contents**

Click the **My Onshape** filter in the left panel to organize your Documents page into listing Folders and Documents to which you have access. The **Teams** and **Shared with me** filters also display folders to which you have access.

To see the contents (documents and sub-folders) of a specific folder, click the name of the folder (an underline appears when you are in the correct position for clicking).

If there are no documents or folders in that folder, a message appears to that effect.

Breadcrumbs are displayed at the top of the page. Click these breadcrumbs to navigate along the folder tree.

**Deleting folders**

Right-click on the folder entry and select **Send to trash** or drag and drop the folder onto **Trash** in the filter panel. The contents of the folder are also moved to Trash.
Nesting folders

While in a folder (after opening the folder by clicking on the underlined name), click Create, then Folder... to create another folder. Check the breadcrumbs to the right of the Create button to verify that you are inside of a folder. Nested folders inherit the Share permissions applied to the top-level folder.

![Image of folder structure]

In the above illustration, “FolderOne” is the parent folder and “Folder-x510” is the sub-folder.

You have the ability to drag and drop a folder into another folder, provided you have permissions on both folders.

Moving documents among folders

You can move documents into and out of folders.

After a document is created inside a folder, you are able to move it to another folder, or out of a folder altogether:
1. Select the document (or use Ctr-click to select multiple documents).

2. Right-click the folder and select Move to.

3. Select the new folder from the list. (Use the breadcrumbs to navigate to another location, if desired.)

4. Click Move here.

You can use drag and drop to maneuver documents and folders into and out of folders. While in a folder, you can drag a document or folder into another location in the breadcrumbs. In the example below, the test-doc document is being dragged into the aero-project folder. The selection document is highlighted in blue and the target folder is also highlighted in blue:
Your cursor will have a small rectangle appended to it when the move is allowed, and a small circle with a line through it when the move is not allowed.

Keep in mind that when moving a document from one root folder to another, the ownership of that document is transferred to the owner of the folder into which it is moved. The Share permissions also change to be inherited from the new root folder.

**Creating a document in a folder**

Navigate to the folder in which you want to create the document. Once the folder name is visible in the breadcrumbs at the top of the Documents page:

1. Click ![Create](image).
2. Select ![Document](image).
3. Specify a name for the document.
4. Click ![OK](image).

**Sharing a folder**

When a folder is created, only the creator of the folder has any permissions to that folder. (The exception is if the folder is created within a Professional or Enterprise subscription. In that case, the administrator also has access to, and permissions on the folder.)

The creator/owner of the folder is able to share the folder with other users and thereby assign Share permissions to the folder. These permissions are applied on a root folder level. All documents and sub-folders within the root inherit the permissions of the root folder.

You are able to specify additional Share permissions on individual documents within folders, but keep in mind that permissions are additive: these additional permissions are added to the folder permissions to create a larger set of permissions and access.

The Onshape best practice is to keep minimal permissions on the root folder and add additional access and permission on individual documents.
To share a folder:

1. Select the folder on the Documents page.

2. Click the button.

3. Make the appropriate permission selections and enter the email addresses of the people with whom you wish to share the folder (or select a team or company).

4. Click .

When a folder is shared, the permissions on that folder become the shared users’ permissions on any and all documents and folders within that folder.

Only top-level folders can be explicitly shared, and all Share details apply to the contents of the folder, including all documents and any sub-folders, even if they are moved into the folder at a later time.

For more information on Sharing, see "Share Documents" on page 2045.

**Transferring ownership of a folder**

An admin of an enterprise or company may also transfer the ownership of a folder to an individual Onshape user who is not a member of the enterprise or company. Right-click on the document or folder (on the Documents page) and select Transfer ownership.

For more information on transferring ownership, see "Transfer Ownership" on page 2105.

**Copying document workspaces in folders**

When copying a workspace in a folder (thereby creating a new document), you need edit permission on the folder in order for the copy to be placed in the same folder as the original document. If you do not have edit permission on the folder, the copy is placed at the root level of your Onshape account which is My Onshape. A blue notification is displayed at the top of the window upon successful copy that is has been created in My Onshape.

**Folders: iOS**
Create a folder

You can create and use folders to group documents within the user interface. All documents within a folder have the permissions dictate by the root folder. Permissions are assigned when you share a folder with another user. Those permissions are associated with the folder and apply to all documents within the folder. Sub-folders inherit the permissions of their root folder.

To create a folder:
Tap the leftmost button in the lower portion of the screen.
Specify a name for the folder and then tap OK.

Viewing folders and their contents

Tap the My Onshape filter to organize your Documents page into listing Folders and Documents to which you have access. The Teams and Shared with me filters also display folders to which you have access.

To see the contents (documents and sub-folders) of a specific folder, tap the name of the folder.

If there are no documents or folders in that folder, a message appears to that effect.

Breadcrumbs are displayed at the top of the page. You can click these breadcrumbs to navigate along the folder tree.

Nesting folders

While in a folder (after opening the folder by tapping the name), use the icon to create another folder. Check the breadcrumbs at the top of the screen to verify that you are inside of a folder.

Creating a document in a folder

Navigate to the folder in which you want to create the document. Once the folder name is visible in the breadcrumbs at the top of the Documents page:

1. Tap the Create a document icon.
2. Specify a name for the document.
3. Tap OK.

Sharing a folder
When a folder is created, only the creator of the folder has any permissions to that folder. (The exception is if the folder is created within a company or enterprise subscription. In that case, the administrator also has access to, and permissions on the folder.)

The creator/owner of the folder can share the folder with other users and thereby assign permissions to the folder. These permissions are applied on a root folder level. All documents and sub-folders within the root inherit the permissions of the root folder.

You can specify additional Share permissions on individual documents within folders, but keep in mind that permissions are additive: these additional permissions are added to the folder permissions to create a larger set of permissions and access. For this reason, the Onshape best practice recommendation is to keep minimal permissions on the root folder and add additional access and permission through individual documents.

To share a folder:

1. Tap the information icon to the right of the folder name.
2. Tap the Share icon.
3. Make the appropriate permission selections and enter the email addresses of the people with whom you wish to share the folder.

When a folder is shared, the permissions on that folder become the shared users’ permissions on any and all documents and folders within that folder.

Only first-level folders can be explicitly shared, and all Share details apply to the contents of the folder, including all documents and any sub-folders.

Moving a document (or a folder) into or out of a folder

You have the ability to move documents and/or folders into and out of folders. You must have the appropriate permissions on the documents and the folders you wish to move and to move to.

1. Tap the icon next to the document (or folder) in the Documents list to open the Details flyout for that document.
2. Tap the Move icon and then select the folder into which to move the document (or folder).

3. Once the target folder is selected, the contents of that folder are displayed.

4. If this is the correct folder, click Move here (top right corner of the screen). If this is not the correct folder, click the X in the top left corner.

If a transfer of ownership is required to move the document or folder into the new folder, a notification is displayed. Document ownership is transferred to the owner of the folder.
5. Confirm the transfer of ownership (if appropriate) by selecting Transfer ownership, or click Cancel to cancel the move.

6. Once the action is confirmed or dismissed, the Documents page is displayed again.

**Folders: Android**

**Create a folder**

You can create and use folders to group documents within the user interface. All documents within a folder have the permissions dictated by the root folder. Permissions are assigned when you share a folder with another user. Those permissions are associated with the folder and apply to all documents within the folder. Sub-folders inherit the permissions of their root folder.

To create a folder:

Tap the leftmost button in the lower portion of the screen.
Specify a name for the folder and then tap OK.

**Viewing folders and their contents**

Tap the **My Onshape** filter to organize your Documents page into listing Folders and Documents to which you have access. The Teams and Shared with me filters also display folders to which you have access.

To see the contents (documents and sub-folders) of a specific folder, tap the name of the folder.

If there are no documents or folders in that folder, a message appears to that effect.

Breadcrumbs are displayed at the top of the page. Click these breadcrumbs to navigate along the folder tree.

**Nesting folders**

While in a folder (after opening the folder by tapping the name), use the icon to create another folder. Check the breadcrumbs at the top of the screen to verify that you are inside of a folder.

**Creating a document in a folder**

Navigate to the folder in which you want to create the document. Once the folder name is visible in the breadcrumbs at the top of the Documents page:

1. Tap the create a document icon.
2. Specify a name for the document.
3. Tap OK.

**Sharing a folder**

When a folder is created, only the creator of the folder has any permissions to that folder. (The exception is if the folder is created within a company subscription. In that case, the company administrator also has access to, and permissions on, the folder.)

The creator/owner of the folder may share the folder with other users and thereby assign permissions to the folder. These permissions are applied on a root folder level. All documents and sub-folders within the root inherit the permissions of the root folder.
You can specify additional Share permissions on individual documents within folders, but keep in mind that permissions are additive: these additional permissions are added to the folder permissions to create a larger set of permissions and access. For this reason, the Onshape best practice recommendation is to keep minimal permissions on the root folder and add additional access and permission through individual documents.

To share a folder:

1. Tap the overflow menu to the right of the folder name.
2. Tap the Share icon.
3. Make the appropriate permission selections and enter the email addresses of the people with whom you wish to share the folder.

When a folder is shared, the permissions on that folder become the shared users' permissions on any and all documents and folders within that folder.

Only first-level folders may be explicitly shared, and all Share details apply to the contents of the folder, including all documents and any sub-folders.

**Moving a document (or a folder) into or out of a folder**

You can move documents and/or folders into and out of folders. You must have the appropriate permissions on the documents and the folders you wish to move and to move to.

1. Tap the icon next to the document (or folder) in the Documents list to open the Detail panel for that document.
2. Tap the Move icon and then select the folder into which to move the document (or folder).
3. Once the target folder is selected, the contents of that folder are displayed.
4. If this is the correct folder, click Move here (top right corner of the screen). If this is not the correct folder, click the X in the top left corner.

If a transfer of ownership is required to move the document or folder into the new folder, a notification is displayed. Document ownership is transferred to the owner of the folder.
5. Confirm the transfer of ownership (if appropriate) by selecting Transfer ownership, or click Cancel to cancel the move.

6. Once the action is confirmed or dismissed, the Documents page is displayed again.

Using Onshape Filters

This functionality is available on Onshape's browser, iOS, and Android platforms.

You have multiple options for organizing and then locating your data in Onshape. Onshape provides the following methods for organizing your data on the Documents page:

- Use Onshape-supplied filters (as in My Onshape, Recently opened, Create by me, etc)
- Create labels and attach them to documents to act as a filter
- Create folders in which to group documents
- For enterprises, create projects in which to group folders and documents
Create teams of users, not necessarily people in your organization or on your Onshape account (teams can be created by the owner of an account and also by individual members of an account).

All of these constructs act as filters and are listed in the filter list on the left side of the Documents page, except for folders; folders are listed in the documents list and tile views. To filter by any of these, simply select one either in the filter list on the left, or select a folder (or project) in the list or tile view.

**Onshape filters**

On the left of the Documents page is a list of pre-defined filters to aid you in finding folders and documents. Select one to filter the list of documents:

- 🧑‍💻 My Onshape filter - Lists all folders and documents you have created as well as all those shared explicitly with you.

- ⏳ Recently opened filter - Lists documents most recently opened by you or another user with permissions to the document.

  To remove a document from this filter list, select the document and right-click to access the context menu. Select Remove from recently opened.

- 🗄️ Created by me filter - Lists those documents you have created yourself regardless of owner.

- 🏷️ Shared with me filter - Lists all documents shared with you explicitly by another Onshape user.

- 🧑‍➕ Teams filter - If you are a member of a team, those filters are inserted at this point in the list, and include documents and folders shared with the team. Teams are collapsed under the Teams label.

  Note that if the account you belong to has many teams, you see only those teams that you are a member of. Anyone can create a team through Account settings.
Any labels you have created are listed, collapsed under Labels. Labels are user-specific. Create labels from the Create button or from the label icon in the button toolbar at the top of the window.

Public filter- Lists all documents made publicly available to all Onshape users by all other Onshape users. For actions specific to public documents, see "Public documents" on the next page, below.

Google drive or Dropbox filters - If you have previously integrated your Google Drive and/or Dropbox account into Onshape, the filter appears above the Trash filter on your Documents page. To filter your search through your Google Drive account, click the icon. To filter your search through your Dropbox account, click the icon. The email following the icon is the specific account you have integrated. You can integrate more than one account.

Integrate your Dropbox or Google drive account through Integrations in your Account settings.

Trash filter - Lists all documents that you have sent to trash. You see only those documents you have sent to trash. Note that any documents you delete from the Trash, and all those present in Trash when you click Empty Trash, are deleted forever. You can, however, right-click on any item in Trash and Restore it.

When a document is permanently deleted, it cannot be retrieved, viewed, or edited.

In the event you have a document in trash that another document links to, the document in Trash will not be automatically deleted by Onshape. You can delete it from the Trash yourself, in which case Onshape retains any information in that document that has been linked to by live (undeleted) documents so those documents remain valid. If all referencing documents are deleted (sent to Trash and then deleted), the previously retained information will be permanently deleted as well.

Documents in Trash are not upgraded automatically with Onshape automatic releases. If you restore a document (or other item) from Trash after a new release of Onshape, you are prompted to upgrade the document at that time.
Public documents

You have the ability to view all Public documents at once through the Public filter on the Documents page. The following information is displayed both in the list view and the tile view:

- **Name** - The document and workspace names
- **Modified** - The last time and date the document was modified
- **Owned by** - The owner’s user name
- **Likes** - The number of times the document has been ‘Liked’ by users; click the thumbs up icon either in the tile view or on the Details flyout.
- **Links** - The number of times this document (or an entity therein) has been referenced by another document
- **Copies** - The number of times this document has been copied by other users

Public documents can be seen by all Onshape users, in a view-only capacity. However, keep in mind that any user can make a copy of a public document and then use that data. No copyright is inferred upon documents made available publicly. If you
have a Free account, all of the documents you create are public by default. If you wish to create private documents, sign up for a paid subscription of Onshape.

Filters: iOS

To organize the documents list with a specific filter:

Tap the icon in the upper left to open the navigation drawer. Here, tap to select a filter with which to organize the documents list.

Select from:

- **My Onshape** - lists all folders and documents you have created as well as all those shared explicitly with you.

- **Recently opened** - lists documents most recently opened by you or another user with permissions to the document.

- **Created by me** - lists those documents you have created yourself, regardless of owner.

- **Shared with me** - lists all documents shared with you explicitly by another Onshape user or as part of a team or company.

- **Team** - if you are a member of a team, those filters are inserted at this point in the list, and include documents and folders shared with the team. Teams are collapsed under the Teams label. Note that if the account you belong to has many teams, you see only those teams that you are a member of. Anyone can create a team through Account settings.

- **Labels** - any labels you have created are listed, collapsed under the Labels filter. Labels are user-specific.

- **Public** - lists all documents made publicly available to all Onshape users by another Onshape user.

- **Trash** - lists all documents that you have deleted. You see only those documents you have sent to Trash. Note that any documents you delete from the Trash, and all those present in Trash when you click Empty trash, are deleted forever. You can, however, restore items in Trash.
When a document is permanently deleted, it cannot be retrieved, viewed, or edited. In the event you have a document in Trash that another document links to, the document in Trash will not be automatically deleted by Onshape. You can delete it from the Trash yourself, in which case Onshape retains any information in that document that has been linked to by live (undeleted) documents so those documents remain valid. If all referencing documents are deleted (sent to Trash and then deleted), the previously retained information will be permanently deleted as well.

Documents in Trash are not upgraded automatically with Onshape automatic releases. If you restore a document (or other item) from Trash after a new release of Onshape, you are prompted to upgrade the document at that time.

The most recently selected filter is remembered and treated as default upon your next return to the Documents page. Signing out of the app resets the remembered default filter.

Sort documents list

To sort the documents list by name, last modified date, or modifier:

Tap the sort icon in the upper right.

Search documents list

To search for a specific document by name:

Tap the search icon in the upper right.

When using the search button to search for document, the search is conducted under the currently selected filter.

Filters: Android

To organize the documents list with a specific filter:

Tap the icon in the upper left to open the navigation drawer. Here, tap to select a filter with which to organize the documents list.
Select from:

- **My Onshape** - lists all folders and documents you have created as well as all those shared explicitly with you.

- **Recently opened** - lists documents most recently opened by you or another user with permissions to the document.

- **Created by me** - lists those documents you have created yourself.

- **Shared with me** - lists all documents shared with you explicitly by another Onshape user or as part of a team or company.

- **Team** - lists all documents and folders shared with the team.

- **Labels** - any labels you have created are listed, collapsed under the Labels filter.

- **Public** - lists all documents made publicly available to all Onshape users by another Onshape user.

- **Tutorials** - lists all tutorials provided by Onshape; these documents are read-only; feel free to make a copy for yourself in order to obtain edit privileges.

- **Trash** - lists all documents that you have deleted. You see only those documents you have sent to Trash. Note that any documents you delete from the Trash, and all those present in Trash when you click Empty trash, are deleted forever. You can, however, restore items in Trash.

When a document is permanently deleted, it cannot be retrieved, viewed, or edited. In the event you have a document in Trash that another document links to, the document in Trash will not be automatically deleted by Onshape. You can delete it from the Trash yourself, in which case Onshape retains any information in that document that has been linked to by live (undeleted) documents so those documents remain valid. If all referencing documents are deleted (sent to Trash and then deleted), the previously retained information will be permanently deleted as well.

Documents in Trash are not upgraded automatically with Onshape automatic releases. If you restore a document (or other item) from Trash after a new release of Onshape, you are prompted to upgrade the document at that time.
The most recently selected filter is remembered and treated as default upon your next return to the Documents page. Signing out of the app resets the remembered default filter.

**Search documents list**

To search for a specific document by name:

Tap the search icon in the upper right.

When using the search button to search for document, the search is conducted under the currently selected filter.

**Advanced Document Search**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Use the search bar at the top of the Documents page to search for a document inside the active filter.

An advanced search allows you to search for a Document, Part, Part Studio, Assembly, or Drawing within the active filter.

To perform an advanced search, click on the drop-down arrow in the search bar and enter additional search criteria:
Search accepts multiple words as well as non-alphanumeric characters, such as punctuation characters, for example "universal handle" and "universal-handle."

- **Type** - Select a type to search for. You are able to search for a Document, Part, Part Studio, Assembly, or Drawing. Select All to search all types.

- **Name** - Enter the name of the type of entity you are searching for (the name of the Document, Part, Part Studio, Assembly, or Drawing).

- **Description** - Every type is able to have a description assigned to it. Enter some or all of the description of a document, tab, or part you are searching for.

- **Part number** - If you are searching for a Part, Assembly, or Drawing, you can enter a Part number to search for.

- **State** - If you are searching for a Part, Assembly, or Drawing, you can enter a state to search for such as in progress, pending, released, or obsolete.

- **Revision** - If you are searching for a Part, Assembly, or Drawing, you can enter a revision number to search for.

- **Add criteria** - Users of Company and Enterprise accounts have the ability to specify criteria for custom properties when custom properties are active for their company.

- **Results from** - Acquire results from Workspaces, Workspaces and Versions, or Versions; and just the latest versions and workspaces or all of them.

As you enter additional search criteria, the search bar auto-populates with the complete query.

Click Reset to clear the search bar and fields.

Use a **wildcard** (*) in any field but only in a trailing position, as in: 05* to find a string beginning with '05' and followed by any characters. Using a wildcard in a prefix position returns more results than would be meaningful.

**Understanding search results**

The search result list indicates where the found identifier (part name, part number, document name, etc) has been found. For example, with the search criteria as (the blue arrow indicates the wildcard in use):

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The search results appear as:
Each search result indicates which type of entity fulfills the search criteria by the icon preceding the name, followed by the name of the entity. Below that, the workspace or version icon, the document name and workspace name (or version name, when part of the search criteria), the part number, and the release management state.

For example, the first search result in the image above:

- A Part with the name Part 1, in the Untitled document, in Part Studio 1 (as seen in the details pane, on the right side of the Documents page).

The following information is displayed below the item name (shown above, outlined in blue) in this order:
An icon indicating whether or not that item is a workspace, version, or latest revision.

The workspace or revision name, shown in parentheses (note that this only appears when an item is in the Release management workflow).

The part number, highlighted in gray (note that this only appears when an item is in the Release management workflow).

The state of the item (Released, Pending, In progress, Rejected, Obsolete, or Discarded).

Select a search result, then click the Details icon to open the Details pane, where you will find a more detailed description of that item.

Note that labels on documents are only available in the Details pane.

When hovering your cursor over a search result that is not a document (for example, a part) the tab name will be in the tool tip that appears:

Click to open Part 1 in Untitled document > Part Studio 1.

When the search result includes revision information or the searched for entity has a new property defined, the Detail flyout includes a Revision history icon and a Properties icon, respectively:
User Interface Basics: Desktop

This functionality is currently available only on Onshape's browser platform.

This functionality is also available on iOS and Android in a limited form.

Toolbars

Located at the top of the page, these change based on the current work flow. There are 5 main toolbars:
• **The Document toolbar**

Click the question mark to access supporting information like online Help, videos, tutorials, FeatureScript documentation and keyboard shortcuts. You are also able to find links to the Onshape Forum, a What’s New posting, and a mechanism to give feedback.

• **The Feature toolbar**

Select Sketch to open the Sketch toolbar.

• **Sketch toolbar** Open by selecting Sketch on the Feature toolbar.

• **Assembly toolbar**

• **Drawings toolbar**
Part Studio interface

- **Default geometry** - Includes Origin, Top plane, Front plane, Right plane; hover over an entity in the Feature list and then use the 🕒 to toggle hide/view. Resize planes: select to activate drag handles, then drag to desired size:

- **Graphics area** - Displays the active Part Studio, Assembly, or other tab.

- **Feature list and Rollback bar** - In Part Studios: a parametric history of work, containing a Rollback bar to view work at a certain point in the history. In Assemblies, the Feature list contains the Assembly tree structure, Mates, Groups, and Mate con-
nectors. See "Feature and Part Lists" on page 312, "Part Studios" on page 279 and "Assemblies" on page 1410 for more information.

- **Selection** - Works as a toggle, click to select and click again to deselect. No need to use function keys for multiple selections. See "Selection" on page 219 for more information.

- **Dialogs** - Mechanism for creating and editing features. A solid blue field requires selection in the graphics area (click on a sketch, region, part, etc). A field outlined in blue requires keyboard input. See "Dialogs" on page 250 for more information.

- **Undo and Redo** - Undo and redo; undo the last successful action, redo the last undone action; available per user, per tab, per session.

- **Context menus** - Available for all features and tab (right-click on the feature or tab). See "Context Menus" on page 268 for more information.

- **Error indicators** - Color-coded feedback, messaging, constraint icons. See "This functionality is available on Onshape's browser, iOS, and Android platforms." on page 273 for more information.

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**Keyboard Shortcuts and Hotkeys**

This functionality is currently available only on Onshape's browser platform.

Activate the keyboard shortcuts map right in the user interface by pressing Shift+? on your keyboard when in a document. You can even pop it out of the window for continuous display. You can access the shortcuts from anywhere in the product by clicking the Help icon in the upper right of the window and selecting Keyboard shortcuts:
Click the arrow in the upper-right corner to pop this window out of the browser window. Click the x to close the window.
View Navigation and Viewing Parts

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape provides the following default mouse settings:

Windows Mouse

**3D Rotate**: Right-mouse-button-click+drag

Press the Alt key to animate to nearest 'floor down' view (the nearest view without any roll)

Holding Alt+Right Mouse results in horizontal mouse movement around the model, and vertical mouse movement pitches over the model

**Zoom in and out**: Scroll up and scroll down, respectively

**2D pan**: CTRL-right-mouse-button+drag (middle button click+drag)

Keyboard

**3D Rotate**:  
Rotate 15 degrees: arrow key  
Rotate 3 degrees: Shift + arrow key  
Continuously rotate: press and hold arrow key

**Zoom in and out**:  
Zoom in: Shift+z  
Zoom out: z

**2D pan**:  
Ctrl+Shift + arrow key

Touchpad

**3D Rotate**: Right-mouse-button-click+drag

**Zoom in and out**: Pinch out and pinch in, respectively

**2D pan**: CTRL-right-mouse-button+drag
Apple Mac Mouse

**3D Rotate:** Right-mouse-button-click+drag

**Zoom in and out:** Scroll down and scroll up, respectively

**2D pan:** Ctrl+RMB+drag (middle button click+drag)

**Keyboard**

**3D Rotate:**
- Rotate 15 degrees: arrow key
- Rotate 3 degrees: Shift + arrow key
- Continuously rotate: press and hold arrow key

**Zoom in and out:**
- Zoom in: Shift+z
- Zoom out: z

**2D pan:**
- Ctrl+Shift + arrow key

To learn how to customize your mouse settings, see [Managing Your Onshape Account](#).

**Rotate the view in 15 degree increments:** Click arrows around the View Cube.

**Rotate the view in 5 degree increments:** Click Shift + the arrows around the View Cube.

**Return to the Trimetric view:** Click one of the small bubbles at the corners of the View Cube.

**View a particular plane view of the cube:** Click one of the sides of the View Cube (Top, Bottom, Front, Back, Right, Left)
The small cube, View Tools, offers these viewing options:

Shaded

Shaded without edges

Shaded with hidden edges
Hidden edges removed

Hidden edges visible

Translucent

Curvature visualization

Interference detection

**Zooming**

The mouse wheel direction for zoom is configurable in user account preferences.

To zoom:

- **Zoom to fit** (shortcut: f, double-click scroll wheel) - Select this command or use the shortcut key to zoom the entire Part Studio, Assembly or Drawing into view.
- **Zoom to window** (shortcut: w) - Select this command, then click+drag a box around the area you want to zoom to in a Part Studio, Assembly or Drawing. The shortcut key toggles the feature on and off.

- **Zoom to selection** - Select this command to zoom to the selected entities.

**Interference detection**

Detect and view the interference between parts in a Part Studio or Assembly.

1. With more than one part in the Part Studio or Assembly, select Interference detection from the View tools menu:
2. Select two or more parts among which to view any interfering mass.

The interference is shown in red, as above, and the parts involved are listed in the Interferences section of the dialog.

Hover over the part name in the dialog to see cross-highlighting in the graphics area when and vice versa. When the focus is on the Interferences field in the dialog box, the graphics area zooms to fit the selected interference. Hover over the Interferences field in the dialog box and a bounding box appears in the graphics area, surrounding the interference in the model. In the dialog box, the length/width/height of the bounding box also appears on hover:

You can use box select to select entities for interference detection: drag from top left to lower right to include only parts or bodies fully encompassed by the box. Drag from lower right to top left to include any parts or bodies touched by the selection box.

**Curvature visualization**

Represent the reflection of a striped room on the current model. This allows you to see
whether or not the curvature across edges is aligned and continuous:

- When the curvature is aligned across an edge, the edge is smooth and the stripes line up, and then veer off across the edge:

- When the curvature is continuous across an edge, the edge is smooth and there is no change in curvature across the edge. Stripes line up and do not veer off across the edge:

**Draft analysis**

Use Draft analysis to find faces in the model that do not meet a specified minimum amount of draft, discover undercut regions, and see the potential parting line locations for selected geometries.

Select Draft analysis from the View tools menu:
1. In the dialog, indicate the Mold split direction by selecting a plane, face, or edge.

2. Specify the minimum draft angle.

3. Select the part or parts to check.

4. Optionally turn off the indication of red undercut faces (Show undercut regions check box).

Notice the draft analysis flyout in the bottom right corner of the window.

- Faces in blue indicate they meet the specified minimum angle for the draft.
- Faces in yellow indicate they are too steep (i.e. less than the minimum specified draft).
- Faces in red indicate undercut faces.

You can view the exact angle of individual drafts by moving the cursor over the model:
As with other visualization modes, draft analysis remains active until you select something else. While it is active, you are able to edit the part to correct the drafts and see the immediate result of your actions. You are also able to use section views to view places on the model that might otherwise be difficult to see.

To change the details of the draft analysis, click Edit draft analysis in the lower right corner:

Draft analysis works automatically in both directions. Onshape displays acceptable draft in different colors to indicate direction: light blue for side one (positive direction) and dark blue for side two (negative direction). Note that the manipulator arrow points to side one and you can flip it using the directional arrow in the dialog, as shown above next to the Mold split direction field.

The image above shows the draft angle after the directional arrow was clicked to flip the direction of the analysis: the light blue shows the positive direction and the angle is shown as a positive angle. By contrast, the image preceding, the dark blue shows the negative direction and the angle is shown as a negative angle.

**Setting transparency via the context menu**

Set the level of transparency of a part through the Part context menu; right-click on a
part name in the Parts list and select Appearance editor. See "Customizing Parts, Faces, and Features: Appearance" on page 302 for more information.

**View parts and surfaces sectioned with Section view**

Section view is available from the View Tools cube, allowing you to select one or many planes, mate connectors, cylindrical faces, or planar faces to use for sectioning.

You can also use the context menu for a plane to select Section view and turn this view type on.

Once the manipulator is visible, you may move it via the ball (open circle at its center) and snap it to any inference point on the part, surface, or assembly. You are able to view sectioned items in both Part Studios and Assemblies:

1. Select one or many planes, mate connectors, cylindrical faces, or planar faces on the part or surface.
2. Expand the menu on the View Tools cube and select Turn section view on.
You can alternately select the Turn section view on command before making any selections.

3. The part/surface is sectioned at the point you chose in step 1 above (cylindrical face, planar face, plane, or mate connector). A manipulator appears at the last location selected and a dialog opens listing selections:

   Intersecting parts are rendered in red.

4. Click and drag the open circle (ball) of the manipulator to position it. Notice you are able to snap it to any inference point on the part or assembly, including the centroids of cylinders (the white marks below indicate inference points):
5. Use the manipulator to change the depth and/or angle of the section.
   a. Use the arrow to change the depth, dragging in one direction or another. Click the manipulator to flip the direction of the view.
   b. Use the angle indicators to drag at an angle.
   c. Use the numeric field to type the depth or angle of the view.

6. To select a different sectioning plane, click the selection in the dialog box to activate the manipulator.
7. To view the section normal to the section view plane, use shortcut key “n” or right-click and select “View normal to” from the context menu.

8. Select Turn off Section view when you're finished.

Note that you can use Section view and then save the view as a Named View.

**Keyboard shortcuts for view**

- Front view = Shift 1
- Back view = Shift 2
- Left view = Shift 3
- Right view = Shift 4
- Top view = Shift 5
- Bottom view = Shift 6
- Isometric view = Shift 7
- Section view = Shift X
- Named view = Shift V

**Zoom to selection**

Use Zoom to selection to change the view to a close-up of the selected entities.

Make a selection in the graphics area:
Expand the View menu and select Zoom to selection:

**Highlight boundary edges**

Onshape displays holes and disconnects in a model, including laminar edges of surfaces, that may be in need of repair when you select Highlight boundary edges. This
feature is especially helpful when importing parts that require repairing surfaces.

Select Highlight boundary edges in the View tools menu. Edges that may require repair are highlighted in solid red lines when the edges are visible and dashed red lines when the edges are hidden:

Keep in mind that this tool is available only in Part Studios. For more information, see Repairing Imported Models.

3D Rotate Lock: iOS and Android

3D Rotate Lock, when active, locks the user's ability to 3D rotate the graphics area. This is particularly useful when attempting to select and drag an entity.

It is located directly above the View Cube in both Part Studios and Assemblies.

To activate the 3D Rotate Lock, tap the button. To deactivate, tap the button again or commit a feature.

3D Rotate Lock activates by default in certain situations:

- **Part Studio** - If a sketch is open and an entity is selected, the 3D Rotate Lock turns on by default. This allows for the selection to be dragged without the view rotating. Unlock by tapping the button.

- **Assembly** - If an instance, mate connector, or entity is selected, the 3D Rotate Lock turns on by default. This allows for the selection to be dragged without the view rotating. Unlock by tapping the button.

View Cube: iOS and Android
The View cube is located in the upper right corner of the graphics area. When selected, a list of different viewing options appears. Select one to change the view of your graphics area or the view settings of your part(s). This is an easy and quick way to get a well-oriented view of your part(s) without having to 2D/3D rotate or zoom.

**Steps**

1. Tap the View cube. A list of options appears.
2. Tap to select a view option or select Cancel.

3. Scroll to see more options.

**Options explained**

- Top, Bottom, Front, Back, Right, and Left - Select any of these options for a front-facing view of the respective plane.

- Isometric, Dimetric, Trimetric - Select any of these for the respective angled view.

- Zoom to fit - Select to resize the graphics area to fit the screen. This could result in the view zooming in or out.
Perspective View - Toggle perspective view on and off. Perspective view shows the relative distance from the point of view to the model, and creates a vanishing point as the point of view (or imaginary camera) approaches the model. The images below show a front view of the same part without perspective view and with perspective view, respectively.
Shaded - Select to show the part with shaded faces and edges. (Default)
• Shaded without edges - Select to show part shaded, without edges.

• Shaded with hidden edges - Select to show the part shaded and to show hidden edges (edges that aren't in the direct line of sight).

• Hidden edges removed - Select to show the part unshaded, with the hidden edges (edges not in direct line of sight) removed.

• Hidden edges visible - Select to show the part unshaded, with the hidden edges (edges not in direct line of sight) visible.
- Translucent - Select to show the part as translucent.

- Section View - Select to access a manipulator that allows you to adjust a section view of a part, via a plane or planar face.

You must preselect a plane or planar face before selecting Section View. Drag the arrow or either directional manipulator to adjust the section view plane that is created from the preselected plane or planar face. In this case, the Front plane is used to create a section view roughly halfway through the parts.
Named Views

This functionality is available on Onshape's browser and iOS platforms only.

Creating named views: Desktop

You have the ability to create and name views for use later within a workspace. Named views capture the perspective, the zoom scale, and the orientation of the current view.

To create a view and name it in order to retrieve it for later use:

1. Rotate your model into the desired view. For example:

2. Optionally, select Turn perspective on, and/or Zoom to fit. You may also section the view (Turn section view on) and adjust the sectioning.

   Perspective view shows the relative distance from the point of view to the model, and creates a vanishing point as the point of view (or imaginary camera) approaches the model.
3. Access the View Tools menu and select Named views:

![Named views menu]

4. In the dialog, enter a name for the view in the first field:

![Named views dialog]

5. Click the plus sign icon.
6. Notice the view (name) is saved in the next field:

You may create as many named views as you want per workspace.

To delete a named view:

1. Select the view in the second field:
2. Click the \( \times \) icon.

To open the Named views dialog, use the shortcut keys: Shift+v. This opens the dialog next to the View dropdown cube, unless the Named views dialog has been previously repositioned. If it has, it opens in its former location.
If the graphic is in a position that matches a named view at the time the Named view dialog is opened, that view will automatically be selected in the dialog's dropdown menu.

**Creating named views: iOS**

You have the ability to create and name views for use later within a workspace. Named views capture the perspective, the zoom scale, and the orientation of the current view.

To create a view and name it in order to retrieve it for later use:

1. Rotate your model to the desired view, for example:

2. Optionally, tap the View cube (shown above to the left of the blue arrow), select Turn perspective on, and/or Zoom to fit. You may also section the view (Turn section view on) and adjust the sectioning.

   Perspective view shows the relative distance from the point of view to the model, and creates a vanishing point as the point of view (or imaginary camera) approaches the model.
3. Access the View cube menu by tapping the View cube, tap Named views:

4. In the dialog, enter a name for the view in the first field, or tap Select a view to select a previously saved view:

5. Tap the checkmark.

To delete a Named view:
1. Tap Select a view in the second field of the Named views dialog.
2. Swipe your finger from right to left across the desired view to delete it.

---

**Searching for Tools**

This functionality is currently available only on Onshape's browser platform.

Shortcut: alt + C

In Onshape, you have the ability to search for tools and custom features in the toolbar whenever you are in a Part Studio or Assembly.

To begin your search, click on the Search tools button (shown above) located to the right of the toolbar at the top of the page. The Search tools dialog opens:

If you do not have any recently used tools or features, your Search tools dialog will appear as it does above, simply begin typing a word or phrase into the search bar to produce results. As you type, Onshape provides suggestions and results for your search on the left side of the dialog, hover over a tool or feature to see its description on the right side of the dialog:
Click on a tool or feature in the left side of the dialog to begin using it.

In the bottom right corner of the Search tools dialog box, you have the option to check or uncheck the Highlight tool in toolbar box (shown above outlined in blue). With this box checked, the tool or feature you hover over in the Search tools dialog will appear highlighted in orange in the toolbar:

To clear your search, click the x in the top right corner of the dialog.

Click or the Escape key on your keyboard to close the Search tools dialog. The dialog also closes when you select a tool in the toolbar.

To hide the Search tools feature, right-click anywhere in the toolbar to open a drop-down menu:

Click Hide "Search tools," and the Search tools button disappears. Right click in the toolbar again and click Show "Search tools" to replace the button.
Selection

This functionality is available on Onshape's browser, iOS, and Android platforms.

Selection: Desktop Graphics area

Onshape selection works like a toggle. Click to select, click again to deselect. You are also able to click to additively select and deselect (the same behavior you would expect from Ctrl+click).

Specifically:

- To select an entity, click on it. To deselect, click it again.

  The cursor displays a count of selected entities; the displayed cursor count is accurate up to 5 entities (after 5, the cursor maxes out at 5+).

- Clicking additional entities adds them to the selection set.

- Clear selections by clicking in empty space, pressing the Space bar, or by choosing Clear selections from the context menu.

- Select a range of objects (like names in a Feature list) by clicking one end of the range and Shift+click the other end.

- To select a tool in the toolbar, click on it. To deselect, click it again, or use the context menu and select Exit <tool name>, or press the ESC key.

Selection may be made with the cursor on a specific sketch or part entity (sketch curve or part edge, for example) and also by dragging a selection box around or across entities. Selected entities in the graphics area are highlighted. To deselect all selected entities, click in the white space in the graphics area or access the context menu and select Clear selection.

Selecting midpoints for use

When not creating or editing a sketch or feature, you are able to hover over sketch entities and model edges and visualize the midpoints:
Use these midpoints for:

- **Measuring** - Select two points to get measurement information in the Measure tool in the right bottom corner of the interface:

- **Creating planes** - Select midpoints as points to define planes:

- **Use in a sketch** - Select the midpoint (of a sketch entity not in the active sketch), then the Use tool in the Sketch toolbar to use that point in your sketch:
Note that midpoints do not appear for entities not in the active sketch. When creating or editing a sketch, you may select the midpoint of an entity in another (inactive) sketch and use the Use tool to incorporate that point in the active sketch.

Cursor selection examples

- Sketch curve highlighted
• Face highlighted

• Edge highlighted
Box selection examples

Drag **left-to-right** to select the entities that fall entirely within the box (indicated by solid blue outline and blue-shaded selection box):

Notice that despite the selection box having crossed the cylindrical shaft, it was not selected (above).

Drag **right-to-left** to select the entities that the box touches (indicated by dotted yellow outline and yellow-shaded selection box):
Notice that this time when the selection box crossed the cylindrical shaft, it was selected (above).

This functionality works is available in both Part Studios and in Assemblies.

**Selection: iOS and Android**

**Tip**

Graphics area

Onshape selection works like a toggle. Tap to select, tap again to deselect. Double tap (tap twice, quickly, with one finger) to deselect all.

Specifically:

- To select an entity, tap on it. To deselect it, tap on it again.
- Clear selections by choosing **Clear selection** from the [Context Menu](#) or by double tapping anywhere in the graphics area.
- To select a tool, tap on it. To deselect a tool, tap on it again or tap the x in the dialog box.

**Box Select**

Two finger press and hold to bring up the box for box select.

Before picking up your fingers from the screen, use a pinch gesture to adjust the size of the box.
After picking up your fingers from the screen, touch and drag a corner of the box to adjust the size.

Touch and drag the box with one finger to adjust its position, but not its size.

Along the bottom of the screen there are four icons:

- **Cancel** - Tap to cancel box selection
- **Inside box** - Tap to limit the selection to only the entities that are entirely inside the box. In the image below, notice that the cylinder that is not entirely within the box is not selected:

![Diagram showing box selection](image)

- **Crossed by box** - Tap to select any entity that is at least partially inside the box. In the image below, notice that the cylinder at the top of the box that is only partially inside the box is selected along with the other parts:
Confirm - Tap to confirm the box selection.

Precision Selector

Using a finger as a cursor often obstructs one's view and makes it difficult to make precise selections. To help with this, Onshape provides the Precision selector to allow you to carefully select an entity without your finger obstructing your view.

The orange markings below represent a screen touch.
To use the Precision selector, touch and hold with one finger in the graphics area. When the Precision selector appears, drag your finger to move the selector. Use the
crosshairs as a cursor. When the crosshairs line up with the desired selection and the desired entity is highlighted, lift your finger from the screen to make the selection. If you do not want to use the Precision selector, make sure that the crosshairs are not lined up with an entity, then lift your finger and start over.

**Precision selector with stylus**

While using the stylus to precision select, press harder to temporarily zoom. Once you release to make a selection, the zoom returns to normal:

Create Selection

This functionality is available on Onshape's browser and iOS platforms only.

**Create Selection: Desktop**

Onshape provides the Create selection dialog to make selecting related faces, such as faces that define a pocket on a model, easy. This is especially useful in certain commands, such as Delete face and Replace face.

Access this from a Feature tool dialog with this icon +, or select Create selection from the context menu.
Create selection may be used with extrusions, pockets, hole, fillets, tangent connected faces, bounded faces, or edges as the selection criteria. Select one or more faces that the system uses to propagate to select other faces based on the selection criteria. These selections may then be added to a tool dialog value list such as Replace face or Delete face. (See the example below.)

The available selection criteria are:

- **Protrusion** - Selects all faces that are connected to the selected face by a convex edge.
- **Pocket** - Selects all faces that are connected to the selected face by a concave edge.
- **Hole** - Selects all faces that are connected to the selected face as part of the same round hole.
- **Fillets** - Selects all faces on a part which form a constant radius fillet.
- **Tangent connected** - Select all faces that are connected to the selected face by a tangent edge.
- **Bounded faces** - Selects all faces between the selected face and the boundary defined by other edges and faces selected.
- **Select patterns option** - Select all other faces on the part which match the same criteria specified.

Selecting edges enables you to create specific edge selections for detailed filleting or surfacing, for example. The available selection refinement options are:
- **Tangent connected** - Selects all edges tangent to the selected edge. (See example below.)

- **Loop/chain connected** - Select a face (and/or sketch edges) and all edges that form a connected loop on that face (or adjacent to the sketch edge) are selected. Select adjacent faces (or sketch edges) to continue the loop. Unselect a loop if multiple are selected or keep multiple loops if desired.

The face highlighted in orange is the selected face. The edges highlighted in yellow are the automatic selections:

When you select a face, all edges adjacent to that face forming a loop are automatically selected, as shown above.

When you select a face and then an edge, all edges adjacent to the selected edge along the selected face forming a loop are automatically selected, as in:
The orange face is selected as well as the orange edge. The yellow highlighted edges are automatically selected.

- **Equal length/radius** - Select one edge and all the other edges that match that in length or radius are automatically selected:

  ![Equal length/radius example](image)

- **Parallel** - Selects one edge and all other edges parallel to that edge are selected:

  ![Parallel example](image)

- **Select pattern** checkbox - Works with patterns: select one edge and all edges in the pattern are selected automatically. Again, the circle highlighted in orange is the selected edge which is part of a patterned sketch. Onshape automatically selects the rest of the patterned features, highlighted in yellow:

  ![Select pattern example](image)
Examples
Selecting faces

The example below demonstrates using Create selection within the Move face feature, selecting faces:

1. Click the Move face tool.
2. Right-click and select Create selection:

3. Select the type of selection (here, Protrusion), select a face, and Onshape automatically makes the appropriate selections:
4. Click **Add selection** to transfer the selected components to the Move face dialog:

5. Enter the remaining required specifications for the Move face feature.

**Selecting edges**

The example below demonstrates using Create selection within the Fillet feature, selecting edges:
1. Click the **Fillet** tool.

2. Right-click and select **Create selection**.

3. Select the type of selection (here, Tangent connected), select an edge or edges, and Onshape automatically makes the appropriate additional selections (the original selections are highlighted in orange, the yellow highlights indicate the automatic selections):
4. Click **Add selection** to transfer the selected components to the Fillet dialog:

5. Enter the remaining required specifications for the Fillet feature.

**Create Selection: iOS**

Onshape provides the Create selection dialog to make selecting related faces, such as faces that define a pocket on a model, easy. This is especially useful in certain commands, such as **Delete face** and **Replace face**.

**Steps**

1. Access the Create selection dialog either via a Feature tool dialog with the Create selection icon OR via the **Create selection** option in the Context Menu.
The Create selection dialog opens:
2. Chose the Feature type you will be selecting:

- **Protrusion** - Selects all faces that are connected to the selected face by a convex edge.
- **Pocket** - Selects all faces that are connected to the selected face by a concave edge.
- **Hole** - Selects all faces that are connected to the selected face as part of the same round hole.
- **Fillet** - Selects all faces on a part which form a constant radius fillet.
- **Tangent connected** - Selects all faces that are connected to the selected face by a tangent edge.
- **Bounded faces** - Selects all faces between the selected face and the boundary defined by other edges and faces selected.


4. Toggle **Select patterns** on or off.

5. Tap **Add selection** to create the selection per the specifications you set in the dialog.
Example

The example below demonstrates using Create selection within the **Delete face** feature.

1. With the Delete face tool selected and the dialog open, select the Create selection icon:

   ![Create selection dialog](image)

   The *Create selection* dialog opens:
2. Select the type of feature you want to edit, in this case, a **pocket**:

3. Select a face of the pocket you want to edit:

Because you specified that you were creating a selection from a pocket, the other faces that make up the pocket are automatically selected for you:
4. Tap **Add selection** to add the selection of the faces of this pocket as one selection for the Delete face tool to use.

   Note that the Create selection dialog clears to allow you to create another selection.

5. Tap the left facing chevron in the upper left of the dialog to return to the Delete face dialog.

   Note that Delete face now has entities selected:

6. Accept the Delete face dialog.

   Your selected faces are deleted:
Select Other

This functionality is currently available only on Onshape's browser platform.

Shortcut: ` (grave accent key, to the left of the “1” key)

Use Select other to select entities (sketch curves, part faces, etc) that you might not be able to see in the graphics area because they are obscured by other entities.

The keyboard shortcut for this functionality is the grave accent key (directly to the left of the number 1 on a US keyboard):

- ` on the U.S. keyboard
- ’ (on the @ key) on the UK keyboard
- @ on the French keyboard
- ^ on the German keyboard
• ñ on the Spanish keyboard
• Alt+\ on the Italian Mac keyboard and ò on the Italian Windows keyboard

When a part has many faces that you can’t see from one perspective, instead of rotating your model:

1. Select a face (or hover over a face or edge) and then from the context menu, choose Select other or skip the context menu and press the ` key (grave accent key):

   ![Select other dialog]

2. The Select other value list is populated with all faces and edges, working from the one already highlighted and into the part (farther away from your perspective).

3. Select the desired entity from the list with your mouse or cycle through the list using the ` key (grave accent) to proceed down the list and cycle back up the list using Shift+` keys. Press Enter to select the highlighted selection.

4. The Select other dialog closes when a selection is made.

When you open a Feature dialog, the selections you made in the Select other dialog automatically populate the Feature dialog value list.

![Feature dialog]

You also have the ability to open the Feature dialog first, then the Select other dialog.
Triad Manipulator

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Triad Manipulator: Desktop**

Once an instance is inserted into an Assembly, you are able to position it in two ways:

- Use the mouse to click and drag it (referred to as free drag).
- Click on it to activate a triad manipulator (referred to as manipulator drag).

If an instance is fixed, you are unable to drag it. The manipulator does not appear and any attempt to drag the instance results in the following visual cues:

**Repositioning the manipulator itself**

Click a part to visualize the manipulator. Use the center circle (highlighted in orange below) to move the manipulator without moving a part:
As you move the manipulator, you have the option to snap it to any inferred Mate connector or defined Mate connector. Once snapped to a connector, drag the manipulator to move the part in relation to that point:

As you drag the manipulator, (either along a plane or an angle) a numeric field appears:
Enter a numeric value in this field to define the position of the part in relation to the mate connector.

You are able to snap it to other entities in the Assembly to redefine the entity’s position and orientation. You are able to place this center on any mate connection point, then use manipulator drag to move the part in relation to that point.

Use the context menu (right-click with the center of the manipulator selected) for more options like:

- Move to origin (this simply moves the part, placing its reference point at the origin; it does not mate or fix the part).

**Move the instance along an axis**

Use the context menu (right-click with an axis arrow selected) for more options like:

- Align with Z to automatically align the part in the selected direction along the Z axis.
Anti-align with Z to automatically align the part in the selected direction along the -Z axis.

Move the instance within the plane

Rotate the instance around the triad X, Y, or Z axis

Use the context menu (right-click with an angle indicator selected) for more options like:

- Rotate 90 degrees
- Rotate 180 degrees

The part is rotated about the axis that is selected.
An instance not mated and not fixed will move exactly as you specify. A mated instance will try to move as directed within its degrees of freedom. In some cases, the system may not find a solution even though one exists. In these cases, repositioning the manipulator or trying different parts of the manipulator may yield better results.

**Triad Manipulator: iOS and Android**

Once an instance is inserted into an Assembly, you can position it in two ways:

- **Free drag** - touch and drag the part.
- **Manipulator drag** - tap on the part to activate the triad manipulator, then touch and drag the triad manipulator to use it.

**Reposition the manipulator itself**

Touch and drag the circle at the base of the triad manipulator:

This repositions the triad manipulator itself but does not change the instance it belongs to. The triad manipulator that has been activated for an instance remains
related to that instance until it is deselected (tap the selected instance again OR double tap to deselect all selections).

**Move the instance along an axis**

Touch and drag the arrow that represents the axis that you want to move the instance along.

This moves the instance around within the plane of the square you select.

**Move the instance within the plane**

Touch and drag the square that represents the plane you want to move the instance along.

This moves the instance around within the plane of the square you select.

**Rotate the instance around the triad X, Y, or Z axis**

Touch and drag the circle-arc icon that represents the axis you want to rotate the instance about.
This rotates the instance about the axis of the circle-arc icon you select.

Note that an instance not mated and not fixed will move exactly as you specify. A mated instance will try to move as directed within its degrees of freedom. In some cases, the system may not find a solution even though one exists. In these cases, repositioning the manipulator or trying different parts of the manipulator may yield better results.

**Dialogs**

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Dialogs: Desktop**

Dialogs are used wherever user input is required. A typical dialog looks and works something like this:
Selections and other input

There are two types of input accepted into dialogs: selections made in the graphics area or feature list, and keyboard input such as numeric values:

- Fields that are highlighted in blue are populated when you make a selection in the graphics area and in the Feature list.
- Fields that are outlined in blue (and not highlighted) are populated with keyboard input, usually numeric values.

Onshape provides visual representation of the possible states of selected entities. For example:

Healthy selection input

Selection suppressed before opening dialog

Selection suppressed after opening dialog (missing from dialog)
Once the dialog is accepted or rejected, the actions performed when the dialog was open are removed from the Undo|Redo list.

Click in a field to set focus.

Hover in the title box to activate the Edit icon ✏️. Click ✏️ to edit the feature name. (Alternatively, right-click on the feature in the Feature list and select Rename from the context menu.)

Use the Enter key to accept the dialog and close it, use Shift-Enter to accept the dialog, close it, and reinvoke the same function with the dialog empty.

**Example of Active selection field**

**Before a selection is made in graphics area:**

![Fillet 2 dialog](image)

**After a selection is made in graphics area:**

![Face of Shell 1 dialog](image)

**Preview slider and Final button examples**

When creating or editing a feature, the preview (the model in the graphics area) is usually displayed as a blend of the model before and after the feature. The Preview slider is an opacity control that lets you adjust the display opacity of the feature along a scale of 0% (before the feature is applied) and 100% (after the feature is applied).
When you edit a feature, by default Onshape displays the model rolled back to its state when that feature was created, hiding all later features. The Final button displays the final result while you are still editing the feature. If you are editing the last feature, there is no Final button in the dialog, since you are already seeing the final result.

Clicking the Final button shows the part in its Final state with the current editing applied.

**Dialogs: iOS and Android**

Dialogs are used wherever user input is required, both in Part Studios and in Assemblies. Dialogs appear and work similarly on both Android and iOS platforms. A typical dialog looks and works like this:
Extrude 1

- Result body operation type: New
- Faces and sketch regions to extrude: 1 entity selected
- End type: Blind
- Opposite direction
- Depth: 25 mm
- Draft
- Second end position

Loft 1

- Solid: Surface
- New Add Remove Intersect
- Profiles: 2 items
- Start profile condition: None
- End profile condition

Copyright © 2017, Onshape. All rights reserved.
A typical iOS dialog box for an Onshape feature, above left, and a typical dialog Android dialog box, above right

**Accept or reject dialog**

To accept a feature, save changes, and close the dialog, select the checkmark.

To reject a feature and close the dialog without saving, select the x.

```
Extrude 1 ✔️ ❌
```

**Adjust and move the dialog**

By default a dialog will extend to full length. To collapse the dialog completely, tap the top section (or title bar) of the dialog. Tap again to expand it. To adjust the size of the dialog, touch the top and drag to desired length. Once you have specifically adjusted the size, that is where the dialog will expand to when you tap the top section.

**Note:** If the dialog is too small to display all of the information, swipe vertically inside the dialog to scroll up or down.

**Select an option**

Tap a box to select an option. When selected, boxes appear grey.

**Make a selection in the graphics area**

A field highlighted in blue indicates that a selection from the graphics area is required. A grey line of text at the top of the field will explain what type of selection needs to be made. (For example, this particular dialog for Extrude calls for a face and/or sketch region to extrude to be selected.) Once a selection from the graphics area has been made, the field states how many entities have been selected and an arrow appears.

```
Faces and sketch regions to extrude
2 entities selected
```

Tap the arrow to view the list of selected entities. The list appears.
Swipe left on a selection to reveal a delete option. Tap delete to remove that selection.

Make a selection from a list

A box with an arrow and no blue highlighting requires a selection from a list. Tap the arrow to view the list and tap to make a selection from the list.

A checkmark will indicate the selection made.
### End type

- Blind
- Symmetric
- Up to next
- Up to surface
- Up to part
- Through all

### Input a numeric value

Tap a number to open the number pad.

| Depth | 1.0000 in |

Use the number pad to input a value.

See "Numeric Fields" on page 260 for more information.

### Toggle an option

Tap anywhere in a field with a toggle switch to activate/deactivate the option. A grey switch indicates the option is off. A blue switch indicates the option is on.

For example. Opposite direction is a toggle option found in an Extrude dialog.

<table>
<thead>
<tr>
<th>Opposite direction</th>
<th>OFF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Opposite direction</td>
<td>ON</td>
</tr>
</tbody>
</table>

### Using the preview slider and final button
The preview slider is an opacity control that lets you adjust the display opacity of the feature along a scale of 0% (before the feature is applied) and 100% (after the feature is applied).

When you edit a feature, by default Onshape displays the model rolled back to its state when that feature was created, hiding all later features. The Final button displays the final result while you are still editing the feature. If you are editing the last feature, the final button in the dialog is not present since you are already seeing the final result.

![Preview Slider](image)

**Mobile Number Pad**

When a numerical input is required for anything in Onshape, a number pad will appear. Use the number pad to enter a numeric value, change the unit of measurement used, or delete a dimension. Use the green check to accept and save the value entered and close the number pad. Use the red x to close the number pad without saving and return to the graphics area.

External keyboards are compatible with the number pad. You can input a value, use the arrow keys, and backspace. You can even accept the entered value and close the number pad with "Return".

---

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Using the number pad

- Tap the small, circled x in the number display area to clear the current value.
- Tap the button to the right of the number display area to select a unit of measurement for the value.
- Tap the backspace button to backspace one character at a time.
- Tap the keyboard button to open the device’s keyboard to enter values or expressions.

When you tap the keyboard button the number pad minimizes and the device’s keyboard expands. Use the keyboard to enter values or expressions. Entering an unsolvable expression results in a red outline around the numeric field and a deactivated checkmark button. Correct the entered expression or tap the red x to close the keyboard and revert to the initial value. See Numeric fields for more information.

- Tap the red x button to cancel a value and close the number pad without saving.
- Tap the green checkmark button to accept a value, close the number pad, and save.
- Tap Delete to delete the selected dimension from the entity.
Numeric Fields

This functionality is available on Onshape's browser, iOS, and Android platforms.

Numeric Fields: Desktop

When entering values and parameter expressions in numeric fields throughout Onshape, you are able to use the keyboard and also the mouse scroll wheel:

<table>
<thead>
<tr>
<th>Scroll+Key</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>scroll wheel default</td>
<td>increments of 0.1</td>
</tr>
<tr>
<td>Ctrl-scroll wheel</td>
<td>increments of 0.01</td>
</tr>
<tr>
<td>Shift-scroll wheel</td>
<td>increments of 1.0</td>
</tr>
</tbody>
</table>

Numeric value fields throughout Onshape Part Studios and Assemblies accept integers, decimals, parameter expressions and trigonometric functions. Default units dictate the unit when no other unit is entered in the numeric field, but you can always enter any unit. Onshape will convert and display the value in default units. When you click in the field, however, the original units are displayed again.

Accepted unit keywords

<table>
<thead>
<tr>
<th>Keyword type</th>
<th>Keywords accepted</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>mm, millimeter, cm, centimeter, m, meter, in, inch, ft, foot, yd, yard</td>
<td>5mm 10meters 3ft</td>
</tr>
<tr>
<td>Angle units</td>
<td>deg, degree, rad, radian</td>
<td>7deg (or 7 degree) 14rad (or 14 radian)</td>
</tr>
<tr>
<td>Keyword type</td>
<td>Keywords accepted</td>
<td>Examples</td>
</tr>
<tr>
<td>---------------------</td>
<td>----------------------------------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Math functions</td>
<td>+, -, *, /, ^, ceil, floor, round, exp, sqrt, abs, max, min, log, log10</td>
<td>2^3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>abs(-4)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>max(2, 3)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(sqrt(2in * 3mm)) and sqrt(4 in^2)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>exp(2)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ceil(5.667) = 6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>floor(5.667) = 5</td>
</tr>
<tr>
<td>Modulo operator</td>
<td>%</td>
<td>5%2 (returns 1)</td>
</tr>
<tr>
<td>Trigonometric</td>
<td>cos, sin, tan, acos, asin, atan, atan2, cosh, sinh, tanh, asinh, acosh, atanh</td>
<td>These functions are in degrees, not radians. For example: sin (30) = sin(30 deg) = .5</td>
</tr>
<tr>
<td>functions</td>
<td></td>
<td>atan2(4, 5) (Give the polar angle of (5,4) in as an angle)</td>
</tr>
<tr>
<td>Constants</td>
<td>pi, PI, Pi</td>
<td>(3*pi) in</td>
</tr>
</tbody>
</table>

**Using parameter expressions**

Parameter expressions are evaluated as FeatureScript.

- Parameter expressions are available in Part Studios and Assemblies.
- Parameter expressions must either result in a unit-less value, or result in a unit value to the first power.
- After a numeric field has been accepted, the evaluation of the expression is displayed. When the field is active again, the original expression is displayed.
- Use any units (if the field accepts units), but don't mix types (such as degrees and millimeters):

<table>
<thead>
<tr>
<th>Valid</th>
<th>Invalid</th>
</tr>
</thead>
<tbody>
<tr>
<td>3in + 2.5in</td>
<td>3 + 2.5in</td>
</tr>
<tr>
<td>3mm + 2.5in</td>
<td>3mm + 2deg</td>
</tr>
</tbody>
</table>
Plurals of all length and angle units are allowed (for example: feet, radians, etc.).

Most parameters are lengths or angles. Some parameters are unit-less, like Rho and pattern instance counts.

Fractions are supported.

Use parentheses when necessary. For example, \((2*3)(1/3)\).

Global variables and equations are not yet supported. Local variables are supported in Part Studios.

### Order of operations and processing units

For unit-less expressions, all unit-less expressions are accepted and follow the standard order of evaluation. For example: \(3+(2*3)/6\)

For single-unit expressions, all single units are accepted if the expression ends with a unit to the first power. For example: \(3\text{mm} + (2\text{mm}\times3\text{mm})/(6\text{mm})\), and \(3\text{mm} + 2\text{mm}\)

For multiple-unit expressions, all multiple-unit expressions are accepted if the result is a unit to the first power. For example: \(3\text{[unit]} + 3\text{[unit]}\) is accepted, but \(3\text{[unit]} \times 3\text{[unit]}\) is not accepted.

### Trigonometric functions

When creating custom features using FeatureScript, you are able to use trigonometric functions in numeric fields. Keep the following in mind:

Unit-less parameters are accepted. For example: \(\sin(30)\) and \(\sin(\text{asin}(1))\).

Inverse trigonometric functions are accepted. For example: \(\text{atan}(1)\), \(\text{atan}(1)/\text{deg}\). Be aware that \(\text{asin}/\text{acos}/\text{atan}\) return a degree, so you need to divide by degree to get a unit-less value.
For more information on FeatureScript, see Welcome to FeatureScript (opens in new tab) and for more information on creating custom features, see "Add Custom Features" on page 1400.

**Advanced syntax**

Onshape supports syntax/lookup tables such as this:

\[3,5,6,7][2]\] with:

- \[3,5,6,7]\] being the array
- \[2]\] being the position
- 6 being the second value in the array

Remember the array starts at position 0.

Another example is: \[3,5,6,7][3]=7\]

You are also able to use ternary operators (such as '?') which may yield conditional results. For example, say the length of a sketch entity should be 7 inches if the width is greater than 5 inches. It can be written this way:

\#width>5?7:4

Where:

- \#width>5\] is the conditional statement
- ? is the ternary operator
- 7 being if the expression is true (if the width is greater than 5), make the length 7 inches
- 4 being if the expression is false (if the width is 5 or less), make the length 4 inches

**Invalid inputs**

- 3in*3in
- 3 + 3in (Because unit-less + unit does not compute.)
- 3[unit]*3[unit] (This results in [unit]^2, which is not accepted.)
- sin(30)/deg (This results in a 1/deg unit, which is not accepted.)
- Anything resulting in 1/[unit] is not accepted.
- A unit over a unit if there is a separate unit, for example: 3[unit]+1[unit]/2[unit].
Notes

- Inverse trigonometric functions take numeric values and return angles; for example: \( \text{atan}(1) = 45 \) degrees.
- To use a unit-less value (perhaps to enter into a dimension field), divide by the default angle unit; for example: \([\text{atan}(1)/\text{deg}]\).
- Plurals of all length and angle units are allowed (for example: feet, radians, etc.).
- Use parentheses when necessary. For example, \((2*3)\times(1/3)\).
- Fractions are supported.
- Most parameters are lengths or angles. Some parameters are unit-less, like Rho and pattern instance counts.

Global variables and equations are not yet supported. Local variables are supported in Part Studios.

Numeric Fields: iOS and Android

Numeric value fields throughout Onshape Part Studios accept integers, decimals, expressions and trigonometric functions. Default units dictate the unit when no other unit is entered in the numeric field, but you can always enter any unit. Onshape converts and displays the value in default units. When you tap in the field, however, the original units are displayed again.

Expressions are entered into numeric fields using the Mobile Number Pad's keyboard option (see "Numeric Fields" on page 260). Currently, expressions can be used only in Part Studios.

Accepted unit keywords

<table>
<thead>
<tr>
<th>Keyword type</th>
<th>Keywords accepted</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>mm, millimeter, cm, centimeter, m, meter, in, inch, ft, foot, yd, yard</td>
<td>5mm</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10meters</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3ft</td>
</tr>
<tr>
<td>Angle units</td>
<td>deg, degree, rad, radian</td>
<td>7deg (or 7 degree)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>14rad (or 14 radian)</td>
</tr>
<tr>
<td><strong>Keyword type</strong></td>
<td><strong>Keywords accepted</strong></td>
<td><strong>Examples</strong></td>
</tr>
<tr>
<td>---------------------</td>
<td>-----------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| Math functions      | +, -, *, /, ^, ceil, floor, round, exp, sqrt, abs, max, min, log, log10 | $2^3$  
                        |                                      | abs(-4)  
                        |                                      | max(2, 3)  
                        |                                      | (sqrt(2in * 3mm)) and sqrt(4 in^2)  
                        |                                      | exp(2)  
                        |                                      |  |
| Modulo operator     | %                                                         | 5%2 (returns 1)                                                                                                                                 |
| Trigonometric       | cos, sin, tan, acos, asin, atan, atan2, cosh, sinh, tanh, asinh, acosh, atanh | These functions are in degrees, not radians. For example: sin (30) = sin(30 deg) = .5  
                        | functions                         |                                      | atan2(4, 5) (Give the polar angle of (5,4) in as an angle)  
                        |                                      |  |
| Constants           | pi, Pi, Pi                                                | (3*pi) in                                                                                                                                 |

### Using IF statements in expressions

Onshape supports array/lookup tables such as this:

$$[3,5,6,7][2]=6$$  with:

- $$[3,5,6,7]$$ being the array
- $$[2]$$ being the position within the array
- 6 being the value

Remember the array starts at position 0.

Another example is: $$[3,5,6,7][3]=7$$

You can also use ternary operators (such as '?') which can yield conditional results. For example, say the length of a sketch entity should be 7 inches if the width is greater than 5 inches. It can be written this way:

```
#width>5?7:4
```

Where:
• #width>5 is the conditional statement
• ? is the ternary operator
• 7 being if the expression is true (if the width is greater than 5), make the length 7 inches
• 4 being if the expression is false (if the width is 5 or less), make the length 4 inches

How to enter an expression
1. With the number pad open, tap the keyboard button.

The number pad minimizes and a keyboard appears.
2. Enter an expression with the keyboard.
3. Tap the check to accept the expression or the x to cancel.

Using expressions
• Expressions are available in Part Studios and Assemblies.
• Expressions must either result in a unit-less value, or result in a unit value to the 1st power.
• After a numeric field has been accepted, the evaluation of the expression is displayed. When the field is active again, the original expression is displayed.
• Use any units (if the field accepts units), but don't mix types (such as degrees and millimeters):

<table>
<thead>
<tr>
<th>Valid</th>
<th>Invalid</th>
</tr>
</thead>
<tbody>
<tr>
<td>3in + 2.5in</td>
<td>3 + 2.5in</td>
</tr>
<tr>
<td>3mm + 2.5in</td>
<td>3mm + 2deg</td>
</tr>
<tr>
<td>3 + 2</td>
<td>(2*3)(1/3)</td>
</tr>
<tr>
<td>(2<em>3)</em>(1/3)</td>
<td>sqrt(16m)</td>
</tr>
</tbody>
</table>
Plurals of all length and angle units are allowed (for example: feet, radians, etc.).

Most parameters are lengths or angles. Some parameters are unit-less, like Rho and pattern instance counts.

Fractions are supported.

Use parentheses when necessary. For example, $(2*3)/(1/3)$.

Global variables and equations are not yet supported. Local variables are supported in Part Studios.

**Order of operations and processing units**

For unit-less expressions, all unit-less expressions are accepted and follow the standard order of evaluation. For example: $3 + (2*3)/6$

For single-unit expressions, all single units are accepted if the expression ends with a unit to the first degree. For example: $3\text{mm} + (2\text{mm}*3\text{mm})/(6\text{mm})$, and $3\text{mm} + 2\text{mm}$

For multiple-unit expressions, all multiple-unit expressions are accepted if the result is a unit to the first degree. For example: $3[\text{unit}] + 3[\text{unit}]$ is accepted, but $3[\text{unit}] * 3[\text{unit}]$ is not accepted.

**Trigonometric functions**

You can use trigonometric functions in numeric fields. Keep the following in mind:

Unit-less parameters are accepted. For example: $\sin(30)$ and $\sin(\text{asin}(1))$.

Inverse trigonometric functions are accepted. For example: $\text{atan}(1)$, $\text{atan}(1)/\text{deg}$. Be aware that $\text{asin}/\text{acos}/\text{atan}$ return a degree, so you need to divide by degree to get a unit-less value.

**Invalid inputs**

- $3\text{in} \times 3\text{in}$
- $3 + 3\text{in}$ (Because unit-less + unit does not compute.)
- $3[\text{unit}] * 3[\text{unit}]$ (This results in $\text{[unit]}^2$, which is not accepted.)
• \( \sin(30)/\text{deg} \) (This results in a 1/\text{deg} unit, which is not accepted.)

• Anything resulting in 1/\text{unit} is not accepted.

• A unit over a unit if there is a separate unit, for example: 3[\text{unit}]+1[\text{unit}]/2[\text{unit}].

**Notes**

• Inverse trigonometric functions take numeric values and return angles; for example: \( \text{atan}(1) = 45 \) degrees.

• To use a unit-less value (perhaps to enter into a dimension field), divide by the default angle unit; for example: [atan(1)/\text{deg}].

• Plurals of all length and angle units are allowed (for example: feet, radians, etc.).

• Use parentheses when necessary. For example, \( (2*3)*(1/3) \).

• Fractions are supported.

• Most parameters are lengths or angles. Some parameters are unit-less, like Rho and pattern instance counts.

> Global variables and equations are not yet supported. Local variables are supported in Part Studios.

---

**Context Menus**

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Context Menus**

Use a right mouse button (RMB) click on an entity or blank space to invoke a context menu. Context menus contain commands for that entity or workspace in the current context. Context menus exist for entities in the graphics area, entities in Feature lists, Parts lists, Drawings, as well as Onshape constructs such as tabs. Right-click throughout the interface to discover context menus.
To invoke a context menu on a mobile device, two-finger tap in the graphics area at any time.

Some of the situations in which you are able to access context menus are:

- A feature or sketch is open for editing
- An entity is selected in the graphics or drawing area
- A selection is made in the Feature list, Parts list, etc.
- In the graphics area of both Part Studio and Assembly:
  - **Show all** - Show all parts, sketches, and planes
  - **Show all parts** - Show all parts, even those that have been hidden
  - **Create selection** - Open the Create selection dialog for selecting a group of entities to use as a selection in another dialog
  - **Zoom to fit** - Zoom the view of the graphics area to display all entities within view
  - **Isometric** - Adjust the view to Isometric

For example, when a sketch is selected in the Feature list, the context menu is:
- **Rename** - Edit a new name or supply a new name for the sketch
- **Edit** - Open the sketch for editing
- **Copy sketch** - Copy a sketch from one Part Studio to another:
  - Select the sketch in the Feature list, select Copy sketch from context menu.
  - In a different Part Studio, select a plane in the Feature list, select Paste sketch from context menu.
    Note that copying and pasting sketches across Part Studios of differing versions may result in an error condition.
- **Show dimensions** - Show dimensions of sketch; click elsewhere in the graphics area to hide dimensions again.
- **Show/Hide** - Show or hide the selected sketch.
- Show/Hide all sketches - Show or hide sketches except the active sketch.
- Create Drawing of Sketch - Create a Drawing tab with template of your choosing for the selected sketch.
- Export as DXF/DWG - Export the selected sketch as a .DXF or .DWG file (to your local drive).
- Suppress - Visualize the model without the selected feature.
- Add comment - Add a comment directly to that sketch
- Zoom to selection - Zoom so that the currently selected entities fill the screen.
- View normal to - View normal (perpendicular) to the currently selected sketch.
- Clear selection - Deselect all currently selected entities.
- Roll to here - Roll the Feature list back to the selected sketch.
- Roll to end - Present if the rollback bar is not at the end of the Feature list; causes the rollback bar to roll to the end of the Feature list.
- Delete - Remove the selected sketch from the Feature list.

Example

The image below shows the Context menu and available actions with nothing selected.
The image below shows the Context menu and available actions with only the Front plane selected.
Notice some available actions such as **New sketch** (begin sketching on the selected plane), **Offset plane**, **Hide Front plane**, etc.

**Tips**

- The context menu is organized into different sections, vertically. The context menu is context sensitive and available options vary depending on what you have selected when you open it.
- Note that the context menus will not necessarily show all of the same sections, instead they will show only the relevant sections based on your selections in the document.

---

**Error Indicators**

This functionality is available on Onshape's browser, iOS, and Android platforms.
Onshape helps you identify errors and potential issues with error indicators, including:

- **Color in the Feature list** - When there is a problem with a feature, you may see orange text in the Feature list and dialog title.

- When the problem lies with a single field, like an invalid entry in a numeric field, that field is outlined in red. If a selection is the problem, it is red in the selection list and the corresponding part entity or sketch entity is also red:

- **Hover information** - When you see orange text in the Feature list, hover over it for a summary of the issue.

- **Dimension highlighting** - Color is used to indicate the constraint status of dimension: black indicates a driving dimension, and light gray indicates a driven dimension. In the image below, the driving dimension is at the top of the sketch and the driven dimension is at the bottom.
- **Constraint colors** - Constraints normally appear as a gray square with a black icon inside. When there is a problem with a constraint, it will appear as a red square with a white icon.

- **Notifications** - When a general system error occurs, a notification appears in a bubble at the top of the user interface window.
- **Dangling entities** - When an entity is in an error state, and red, selecting it is indicated by the entity turning a darker shade of red:

![Diagram showing selected and not selected entities]

---

**Printing Part Studios and Assemblies**

This functionality is currently available only on Onshape's browser platform.

You have the ability to print (or download an image of) any Part Studio or Assembly in Onshape.
1. Open the tab you wish to print.

2. Expand the Document menu 😵:

3. Select Print.

   A Print dialog opens and a dotted line appears, providing a preview of the printed page border:
4. You are able to click and drag the items (parts, models, drawings) to position it within the dotted page borders using Onshape mouse actions for moving parts.

5. Select the desired paper size.

6. Select Portrait or Landscape orientation.

7. When satisfied with the set up of the page, click to display the page as it will be printed (print preview).

8. Make more specifications as necessary (including destination printer, pages to print, number of copies, layout, color or black and white). You can also specify: paper size, margins, pages per sheet, scale, whether to print on two sides or single side.

9. Click Print.

To download an image, click , choose where you would like to download the image, and click Save.
Part Studios

This functionality is available on Onshape's browser, iOS, and Android platforms.

This topic explains the functionality and mechanics of using Part Studios to create parts as well as how to customize the toolbars.

Part Studios: Desktop

An Onshape Part Studio is a container used to create and edit parts, surfaces, and sketches. The Part Studio lives in a tab within a document and generally consists of the Feature toolbar, the Feature list, and the graphics area containing the model. The Part Studio is not a model, rather it is a design environment meant for designing parts that need to share common references. See below for reference:

- The top margin (Navigation bar) of the Onshape document contains the name of the document (in bold) and the active workspace name to its right.
- The Feature list (on the left of the window) contains the default geometry as well as any features you create. The thick bar is the Rollback bar and has the ability to be
repositioned to generate the Feature list up to its location in the list. At the bottom of this panel is the Parts/Curves/Surfaces lists (referred to commonly as the Parts list). These lists are collapsible for your convenience.

At the top of this panel is the Feature filter and search bar, where you have the ability to filter your search by name, type, folder, or part. Click to open a drop-down menu with various filter options:

![Filter by name or type]

Enter a partial feature name or type to show matching features. Use a special command (prefixed by :) for further control:

- **:name substring**: filter only by feature name
- **:type substring**: filter only by feature type
- **:folder substring**: show features within matching folders
- **:part substring**: show features that directly affect the matching parts
- **:allparts substring**: show features that directly affect any part

For example, **:part part 1** will show all features that directly affect Part 1.

Click the Features and parts icon on the right side of the Feature list to collapse the list entirely. Click the icon again to reopen the list.

- Inside the white space (or graphics area) is where you find the default geometry, origin and planes.
- The Tab manager area at the bottom of the window: all Part Studios, Assemblies, Drawings, imported images and other files. These tabs can be dragged for repositioning, and each has its own context menu (RMB to access).

Use these icons on the Tab bar:
Tab search - Open the Tab search panel to search for a tab by name, or locate using the thumbnail image.

Insert new element - Open the menu from which to create a new tab, including: Part Studio, Assembly, Drawing, Feature Studio, folder. Use this menu to also initiate a file import and add a third-party application.

When the window is smaller, the Sketch tool is resized to 📐.

For more information about sketching, see "Sketch Tools" on page 409.

As a best practice for performance, parts for the entire assembly should not all be modeled in a single Part Studio. Only the parts that must share common references should be in the same Part Studio. When it is time to assemble parts, you can use parts from multiple Part Studios, even from different documents in the same Assembly.

Should you have a need to share common geometric references between a large number of parts, use the Derived part or the In-context feature to maintain relationships between geometries from other Part Studios which enable keeping individual Part Studios smaller.

To learn more about creating multiple parts in one Part Studio, you can follow the self-paced course here: Multi-Part Part Studios (opens in new tab).

Part Studio toolbars

Access the shortcut toolbar with the S key while in a Part Studio, Drawing or Assembly tab.

An example of a sketch tool shortcut toolbar. Customize your shortcut toolbars in your Account preferences.
Customizing toolbars

You can customize the Onshape toolbars in Part Studios, Assemblies and Feature Studios - shown below in a Part Studio:

- Hover anywhere in the toolbar and right-click, then select Customize toolbar...

This activates the ability to edit the toolbar:

- Tools are highlighted in tool sets within the toolbar; you can drag and drop these tool sets to new locations on the toolbar:

- Create a new tool set by dragging a tool icon to the New toolset box that appears when you begin to drag the icon:

- Drag and drop an individual tool out of the toolbar to the Tools box to remove it from the toolbar (you can always drag and drop it back onto the toolbar). Shown below during the drag operation:
After drag is completed:

Click Save to save your changes, Cancel to close without saving, or Reset to default to undo all changes in that type of toolbar (Part Studio, Assembly, or Feature Studio) and restore the toolbar to Onshape original order and content.

When tools are part of a group, you must move the entire group. Once an entire group is removed, you can select individual tools to move back to the toolbar, if desired. To reform the entire group, select Reset to default, which will reset the entire toolbar.

Part Studio context menu

Right-click on a Part Studio tab to access the context menu:

- Open in new browser tab - Open this Part Studio in a new browser tab
- Rename - Rename this Part Studio
- Properties - Access the dialog to provide information about the Part Studio. In the Properties dialog, you are able to provide meta data for the entire Part Studio, or on
a part-by-part basis:

You can use the triangle icon next to the Part Studio name to initiate creating a release package; if you are a member of a company or enterprise, and have permissions to create release packages.

Properties that are grayed out (inactive) are defined and populated through the Company’s properties in Account management. See Manage Companies > Properties for more information.

- **Show code** - Open a panel with the FeatureScript displayed.
- **Duplicate** - Make an immediate copy of this Part Studio. Copies are pasted directly to the original Part Studio. Copies have no association with the original.
- **Copy to clipboard** - Copy a Part Studio to the clipboard in order to paste into another document. Open another Onshape document, click (Insert new element) and select Paste tab to paste the copied tab into a different document.
- **Create drawing of Part Studio x** - Automatically create a drawing of the entire Part Studio (solid bodies/parts only). This creates a new Drawings tab in the document.
• **Move to document** - Move the Part Studio to a new or existing document, creating the new document during this operation. If any part is used in any tab of the original document, a link between the two documents is created and represented in the Feature list with these icons:🔗 a link exists; 🔄 a new version of the document is available. To update to a newer version, click the icon to open the Reference Manager.

• **Export** - Export parts in the Part Studio in a variety of formats with options of where to download or keep in a separate Onshape tab.

• **Delete** - the Part Studio (or any tab), even if it is active. The last remaining tab cannot be deleted.

**Editing and resetting parts properties**

When in a Part Studio, you can access properties for a particular part through the context menu for the part. Right-click the name of the part in the Parts list and select Properties:
The properties can be edited, you can use the icon next to the part name to create a Release (if you have permissions to do so). Use the **Undo/Redo** buttons in the bottom left corner to undo or redo an edit in the dialog.

Use the **Reset all** button to clear all editable part properties of the selected part or parts. (Note the Reset all button is not active when the Part Studio is selected at the top level.) Reset all does more than it appears at first: As the tooltip indicates (on hover), it will also reset some properties you can't see in the dialog (appearance, material, tessellation quality), and, unlike just clearing out the fields manually, it fully clears all manual applications as well. This allows values calculated by custom features, if any, (which are normally overridden by those manual applications) to take effect.

Note that if you have configured properties, **Reset all** will have no effect on those properties.

**Part Studios: iOS**

Newly created documents contain an empty Part Studio and Assembly by default. A Part Studio is used to define parts and has a Feature list (parametric history). One Feature list drives the shape of multiple parts. Use Sketch tools to create 2D geometry, and use Feature tools to create 3D models (or parts) from those sketches, all within a Part Studio.

In a Part Studio, along the top of the screen is the navigation bar. The navigation bar holds a number of tools that varies based on your device's screen width or orientation. Any tools that do not fit due to screen width will be found in the **Document information panel**.

**Return to the Documents Page**

To close the document and return to the documents page, tap the arrow in the upper left corner.
Versions

To create immutable versions and branch versions to create new workspaces, tap the Versions icon in the upper left.

See Version Manager for more info.

History

To view every point of change in the history of the document and restore to any point in that history:

1. Tap the icon in the upper left.
   On a smaller screen, access the Document information panel and then select History from there:

2. Tap the More icon.

3. Select History.

See History for more info.

Comments

To communicate with collaborators, tap the Comments icon in the upper right.
On a smaller screen, access the Document information panel and then select Comments from there:

1. Tap the More icon.

2. Select Comments.

See Comments for more info.

**Collaborators**

To see who you are collaborating with and to access Follow mode, tap the Collaborators icon in the upper right.

See Collaboration and Follow mode for more info.

The Collaborators icon is only available when you are collaborating with someone in real time.

**Help**

To access the Help and other resources such as Videos, Feedback, and About, tap the icon in the upper right.

On a smaller screen, access the Document information panel and then select Help:
1. Tap the More icon.

2. Select Help.

Select one of the following:

- **Help** - to view the documentation for the device you are using.
- **Videos** - open the videos page of the documentation.
- **Feedback** - ask a question, log a bug, or log an improvement.
- **About** - see which version of Onshape you are running.

**Document information panel**

Access the Document information panel which allows you access to any tools (listed above) that do not fit in the navigation bar as well as some other tools listed below.

To access the Document information panel, tap the More icon in the upper right corner.

The Document information panel opens. From here, select from the following:

- **Properties** - to view document properties.
- **Copy** - to make a copy of the workspace you are in.
- **Units** - to view and set default measurement units for the document (such as length, angle, and mass units).
• **Document name** - tap the edit icon in the upper right corner to edit the document name.

• **Document description** - tap Add description to add or edit the document description.

• **Share** - tap the share icon to share the document with individuals, Teams, Companies, or Onshape support. You are also able to set your document to Private or Public.

**Sketch tools**

Tap the New Sketch tool icon to select a plane and start a new sketch.
With a sketch open, tap the New Sketch tool icon to view all of the sketch tools.
See [Sketch tools](#) for more info.

**Feature tools**

Tap the Feature tools icon to view all of the Feature tools.
See [Feature tools](#) for more info.

**Measure and Mass Properties**

Tap the Measure tool to measure any selected entities.
See [Measure tool](#) for more info.

Tap the Mass Properties tool to view the properties of selected parts.
See Mass Properties for more info.

**Feature list**

- Tap the Feature list handle to open the Feature list.
- Touch and drag horizontally or vertically to adjust the size of the Feature list.

See Feature list for more info.

**Rotate lock**

Tap the 3D Rotate lock button to lock the graphics area rotate feature. This is particularly helpful when trying to drag an entity.

See 3D Rotate Lock for more info.

**View Cube**

Tap the View cube to access a list of view to choose from.

See View Cube for more info.
Planes and Origin

- Every Part Studio, by default, has three planes (Top, Front, Right) and an Origin to be used as reference points when sketching or creating a feature.
- You have the ability to create as many planes as you like.
- You have the ability to hide the default planes and Origin from view via the Feature list, but you are not able to delete them.

Tabs

When you open a document, the most recently opened tab is active.
Tap the up-facing chevron (up arrow) to open the Create tab menu.
- Tap a tab to activate it. When you switch tabs, any open feature will be committed.
- Swipe left or right to scroll, horizontally, through the list.
- Filter tabs by Assembly or Part Studio.
- Search for a Part Studio or Assembly by name.
- Create, rename, duplicate, and delete a Part Studio or Assembly. There is no limit to how many Part Studios or Assemblies a document may have.
Duplicate adds a copy of a Part Studio within the document. You are not able to delete a Part Studio or Assembly if it is the only tab in the document. A document must have at least one Part Studio or Assembly (at least one tab).

**Organizing tabs**

Organize tabs with folders on the Tab bar. Use the ellipsis menu to Create a folder:

1. After opening the menu, tap Select.
   a. This selects the current tab to be included in the folder, and puts a check mark on other tabs so you can select them.
2. Select any other tab to be included in the folder.
3. Tap icon.
4. Supply a name for the folder.
5. Tap Create Folder.
6. The folder is created, with the selected tabs inside it, and displayed opened so you can see the contents:
When a folder is active, the Tab bar displays only the tabs in that folder; all other tabs are represented by the All tabs icon:

Select the All tabs icon (second icon from the left, above) to display all tabs again.

Use the context menu on any folder to act on that folder, including:

- Rename - Edit the folder name.
- Select - Select a folder to then create another folder in which to place that folder.
- Unpack - Delete the folder and place the tabs back in their original position.

**Part Studios: Android**

Newly created documents contain an empty Part Studio and Assembly by default. A Part Studio is used to define parts and has a Feature list (parametric history). One Feature list drives the shape of multiple parts. Use Sketch tools to create 2D geometry, and use Feature tools to create 3D models (or parts) from those sketches, all within a Part Studio.

In a Part Studio, along the top of the screen is the navigation bar. The navigation bar holds a number of tools that varies based on your device's screen width or orientation.

Any tools that do not fit due to screen width will be found in the [Document information panel](#).

**Return to the Documents Page**

To close the document and return to the documents page, tap the arrow in the upper left corner.
Versions

To create immutable versions and branch versions to create new workspaces, tap the Versions icon in the upper left.

See Version Manager for more info.

History

To view every point of change in the history of the document and restore to any point in that history:

1. Tap the More icon to access the Document information panel.
2. Tap History.

See History for more info.

Comments
To communicate with collaborators, tap the Comments icon in the upper right.
See [Comments](https://example.com) for more info.

**Collaborators**

![Collaborators icon](https://example.com)

To see who you are collaborating with, tap the Collaborators icon in the upper right.
See [Collaboration](https://example.com) for more info.

The Collaborators icon is only available when you are collaborating with someone in real time.

**Help**

![Help icon](https://example.com)

To access the Help and other resources such as Videos, Feedback, and About, tap the icon in the upper right.
Select one of the following:

- **Help** - to view the documentation for the device you are using.
- **Videos** - open the videos page of the documentation.
- **Feedback** - ask a question, log a bug, or log an improvement.
- **About** - see which version of Onshape you are running.

**Document information panel**

![Document information panel](https://example.com)
Access the Document information panel which allows you access to any tools (listed above) that do not fit in the navigation bar as well as some other tools listed below.

To access the Document information panel, tap the More icon in the upper right corner.

The Document information panel opens. From here, select from the following:

- **Properties** - to view document properties.
- **Copy** - to make a copy of the workspace you are in.
- **Units** - to view and set default measurement units for the document (such as length, angle, and mass units).
- **Document name** - tap the pencil icon in the upper right corner to edit the document name.
- **Document description** - tap the pencil icon to add or edit the document description.
- **Share** - tap the share icon to share the document with individuals, Teams, Companies, or Onshape support. You are also able to set your document to Private or Public.

**Sketch tools**

Tap the New Sketch tool icon to select a plane and start a new sketch.

With a sketch open, tap the New Sketch tool icon to view all of the sketch tools.

**Feature tools**

Tap the Feature tools icon to view all of the Feature tools.

See **Feature tools** for more info.
Measure and Mass Properties

Tap the Measure tool to measure any selected entities.
See Measure tool for more info.

Tap the Mass Properties tool to view the properties of selected parts.
See Mass Properties for more info.

Feature list

- Tap the Feature list handle to open the Feature list.
- Touch and drag horizontally or vertically to adjust the size of the Feature list.
See Feature list for more info.

Rotate lock

Tap the 3D Rotate lock button to lock the graphics area rotate feature. This is particularly helpful when trying to drag an entity.
See 3D Rotate Lock for more info.

**View Cube**

Tap the View cube to access a list of view to choose from.

See View Cube for more info.

**Planes and Origin**

- Every Part Studio, by default, has three planes (Top, Front, Right) and an Origin to be used as a reference points when sketching or creating a feature.
- You have the ability to create as many planes as you like.
You have the ability to hide the default planes and Origin from view via the Feature list, but you are not able to delete them.

**Tabs**

When you open a document, the most recently opened tab is active. Tap the up-facing chevron (up arrow) to open the Create tab menu.

- Tap a tab to activate it. When you switch tabs, any open feature will be committed.
- Swipe left or right to scroll, horizontally, through the list.
- Filter tabs by Assembly or Part Studio.
- Search for a Part Studio or Assembly by name.
- Create, rename, duplicate, and delete a Part Studio or Assembly. There is no limit
to how many Part Studios or Assemblies a document may have.

Duplicate adds a copy of a Part Studio within the document.
You are not able to delete a Part Studio or Assembly if it is the only tab in the document.
A document must have at least one Part Studio or Assembly (at least one tab).

Customizing Parts, Faces, and Features: Appearance

This functionality is available on Onshape's browser, iOS, and Android platforms.

Using the context menu for a specific part (or group of selected parts) you are able to customize not only the color of the part, but also assign materials (and thereby, a density) as well. When you need to control the tessellation (granularity of rendering of parts) for speed or accuracy, use this Appearance dialog also.
In a Part Studio you have the ability to customize the color of a part as a whole and also individual faces and features. You are also able to assign materials (and thereby, a density) to parts as well.

**Default colors**

Onshape has a predetermined color palette and rotation of color assignments as parts are created. (You are also able to assign custom colors to parts, explained below.)

As parts are created, they are rendered in a sequence of eight colors, shown below, from left to right, with the sequence starting over on the 9th part:

![Color sequence](image)

When a part is deleted, the color sequence remains intact with existing parts retaining their color:

![Color sequence](image)

**Customizing appearance: Desktop**

**Customizing part appearance**

The Onshape Appearance editor enables you to manually assign specific colors to specific parts, faces, and features. Once a color is assigned, it is not changed until you change it.

With the Appearance editor, you are also able to indicate that particular parts appear in Part Studios and Assemblies as translucent. This may come in handy when trying to reference parts that are hidden by other parts. Set the transparency in the Part Studio and also see the change in the part in any assemblies it is instanced in.
1. Right-click on a Part name in the Parts list to access the context menu.

Note that you also have the ability to select multiple parts from the graphics area or from the Parts list to assign appearance characteristics to more than one part at a time.
2. Select **Edit appearance**.

![Color Palettes](image)

3. Select a color, or specify the RGB values or the hex value for desired colors.

   Optionally use the Mixer tab to refine the color.

   When you have a color specification you want to save, click the plus sign under Custom colors. To remove a custom color, right-click the color tile and select Delete. You are also able to update a custom color with another color: right-click the color tile and select Update color.

4. Use the Part transparency slider to control transparency (on a scale from 0 - 1; use the slider to specify a value).

5. See below for setting Tessellation quality.

6. Accept ✅.

**Customizing face and feature appearance**

Just as you can customize part appearance, you can also customize specific the color of specific faces and even features. Select the face (in the graphics area) or the feature (in the Features list), right-click, and select Add appearance (or Add appearance to feature) from the context menu.
The context menu will also have a command to Edit appearance <of a part name>. Select that command to change the color of the entire part. Select Add appearance <to selected faces> to change the color of the faces you have pre-selected.

Similarly, the context menu of a feature with an appearance applied will also have a command to Edit feature appearance. Select that command to change the color of the faces created by that feature.

Select Add appearance <to selected faces> to change the color of the faces and features you have pre-selected.

The Face appearance dialog box opens:
In the selection field (shown in blue above), the selected faces are listed. Use the x to remove the selection from the list; select a face in the graphics area to add to the list.

Use the Palette to select an existing color, or the Mixer tab to create a custom color. Click the plus sign under Custom colors to add a color.

Click the check mark when you are satisfied with your choices, to apply the color.

**Appearance panel**

Part Studios that contain a part or surface will show the Appearance panel icon to the right side of the graphics area. Click to open the Appearance panel:

![Appearance panel](image)

Any faces that have had their appearance altered are listed in this panel. This panel enables the following actions:
• Edit existing face, feature faces, and part appearances that are listed in the panel:

• Double-click the row in the panel to edit, and the Appearances dialog opens. Make changes and click the check mark to accept and close the dialog.

• For parts, when the dialog opens you can edit colors for that particular part only; you can not add or remove parts.

• For faces, when the dialog opens you can edit both face selections as well as color selections.

• Click the icon at the top of the panel to open the Face appearance dialog in order to add a new appearance definition; the Face appearance dialog opens:

![Face appearance dialog]

You can preselect faces and the dialog Faces field will be pre-populated with those selections. You can also add and remove selections.

Patterning parts and faces

When parts are patterned, any appearance changes are also patterned. The appearances are all listed in the Appearances panel.
Configuring face colors

You can configure face colors as you would configure any other characteristic of a part or feature. For more information on configuring parts and part properties, see "Configurations" on page 359.

Specifying tessellation quality of parts

In order to maximize performance, Onshape occasionally uses a lesser tessellation quality, which improves the rate of part rendering in the user interface, but may sacrifice some granularity of tessellation as a trade-off. When you need a finer tessellation, you are able to set it in two ways:

- **In a Part Studio** - For a selected part, right-click the part name in the Parts lists and select Edit appearance. In the Appearance dialog, use the drop down menu labeled Tessellation quality and select a quality level:
  - Auto - (Default) The system chooses the quality; balancing performance and quality
  - Coarse
  - Medium
  - Fine
  - Very fine - Never automatically chosen by the system; this setting requires user selection

  Note that when choosing a finer tessellation quality, Onshape performance may slow down until the tessellation is complete.

- **In an Assembly** - Right-click on the part for which to improve tessellation and select **Use best available tessellation**. Similarly, to set tessellation quality back to automatic: right-click on the part and select **Use automatic tessellation setting**.

A part or assembly that has tessellation turned on is indicated by this icon in the Instances list. For example:
Customizing appearance: iOS

The Onshape Appearance editor enables you to manually assign specific colors to specific parts. Once a color is assigned, it is not changed until you change it.

With the Appearance editor, you are also able to indicate that particular parts appear in Part Studios and Assemblies as translucent. This may come in handy when trying to reference parts that are hidden by other parts. Set the transparency in the Part Studio and also see the change in the part in any assemblies it is instanced in.

To use the Appearance editor:

1. Select the ☰ to the right of a part name in the Parts list to access the overflow menu.
2. Select **Appearance** from the menu to open the Appearance Editor.

- Free drag the selector to select a color. The RGB values will adjust according to your selection.
- Slide the arrows left or right to adjust the color's intensity.
- Use the slider to adjust the opacity OR tap the value to the right of $\alpha$ to use a number pad for input.
- Use the sliders to adjust the RGB values OR tap the value itself to open a number pad and input a value.

**Customizing appearance: Android**

The Onshape Appearance editor enables you to manually assign specific colors to specific parts. Once a color is assigned, it is not changed until you change it.

With the Appearance editor, you can also indicate that particular parts appear in Part Studios and Assemblies as translucent. This may come in handy when trying to reference parts that are hidden by other parts. Set the transparency in the Part Studio and also see the change in the part in any assemblies it is instanced in.
To use the Appearance editor:

1. Select the to the right of a part name in the Parts list to access the overflow menu.

2. Select Appearance from the menu to open the Appearance Editor.

Use the slider to adjust the opacity OR tap the value to the right of \( \alpha \) to use a number pad for input.

Use the sliders to adjust the RGB values OR tap the value itself to open a number pad and input a value.

Feature and Part Lists

This functionality is available on Onshape's browser, iOS, and Android platforms.

The Part Studio Feature list consists of a list of features and a list of parts (shown in the desktop screenshot below). The pane on the left of the window includes, starting at the top:

- If you have created configurations, they appear at the top of this list, with a drop-down to allow selection of the configuration
• The number of features in the Part Studio
• A search box (to search for features, parts, surfaces, or curves by name or type)
• The list itself includes the default geometry of origin and planes
• Each feature in the Part Studio
• The rollback bar
• A bar that separates the parts/curves/surfaces list from the features
• The number of parts, surfaces, meshes, composite parts, and curves in the Part Studio
• Each part in the Part Studio (created in the Part Studio or derived)
• When viewing a release version of the document, this indicator ▶ appears next to any parts that have been released in that particular version
• Each surface and also curve (created in the Part Studio or derived)

You are able to drag the separator bar to show more or less of the Feature list area and more or less of the Parts list area. Each area (Feature and Part) also scrolls independently of the other.

Features are accompanied by the tool icon which created them. This enables you to rename using descriptive names and still be able to tell what kind of Feature is represented. The model displayed in the Graphics area is visualized up to the position of the Rollback bar in the Feature list.

**Feature and Parts Lists: Desktop**

**Working with the Feature List**

The Feature list contains a list of all sketches and features created in the Part Studio. It also contains a list of parts (including surfaces and curves), as seen towards the bottom of the list. There are many ways to work with the Feature list:

• Use **Show regeneration times** (with this icon at the top of the list ◉) to open a display of all features and the time each takes to regenerate. You are able to use this to estimate which time-costly features to suppress to maximize modeling time without regenerating features you don’t need at the moment.
Hover over a feature to see the corresponding highlighting on the model (indicated by the blue arrows above). Use the Content menu in the Feature list to suppress a specific feature.

- **Search for features** - Use the search box and filter at the top of the Feature list to filter your list; the feature filter gives you the ability to filter your search by name, type, folder, or part. Click to open a dropdown menu with various filter options (shown below in the image on the right): the ellipsis that appears indicates there are more entries, hover over the ellipsis to view those filtered out features:
Enter a partial feature name or type to show matching features. (The ellipsis that appears indicates there are more entries, hover over the ellipsis to view those filtered out features.) Use a special command (prefixed by :) for further control:

- **:namesubstring:** filter only by feature name
- **:typesubstring:** filter only by feature type
- **:foldersubstring:** show features within matching folders
- **:partssubstring:** show features that directly affect the matching parts
- **:allpartssubstring:** show features that directly affect any part

**Drag the rollback bar** - Visualize a model at the point of the rollback bar; all features listed beneath the rollback bar become temporarily suppressed. You are also
able to right-click on a feature in the list and select “Roll to here” for immediate and precise rollback. Right-click on the rollback bar and select "Roll to end" to return the rollback bar to the end of the Feature list.

If you are a collaborator with view-only permission and you change the position of the rollback bar, you no longer will see real-time updates of any changes made to the rollback bar by another collaborator. To fix this, reload your browser.

- **Rename a feature** - Right-click the name in the Feature list and select Rename to edit the Feature name directly in the Feature list without opening the dialog.

- **Make selections** - Click a feature/sketch name in the list to supply a selection for a dialog (or make the selection in the graphics area)

- **Reorder features** - Drag an item in the Feature list (or multiple items - just click to select additional items, they do not have to be contiguous, in the Feature list) to parametrically reorder them

- **Suppress a feature** - Use the Suppress command from the context menu of a feature to visualize the model without that feature

- **Hide or show features** - Use the Hide/Show command from the context menu of a feature to hide that entity from the graphics area view (or show it); you are also able to hover next to the name and click the 🔄 icon.

- **Hide or show parts, surfaces, curves, meshes, and composite parts** - Hover over the Parts, Composite parts, Curves, Meshes, or Surfaces title in the Parts list to see the 🔄 icon. Click the icon to hide all entities in that list, click again to show all entities.

- **Group sketches, features, and planes into folders** - Select the items you wish to group together in the Features list, then right-click and select Add selection to folder... A Folder name dialog appears in which to enter a name for the folder. The name of the folder appears in the Features list, with an arrow to expand the folder to see the contents. You can right-click on just the folder (with no other selections made) and select Unpack folder to remove the folder and return the items to being listed singly in the Features list. You can also make a selection and then click the folder icon at the top of the Features list.

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All rights reserved.
Make the selection of items to group; right-click and select "Add selections to folder..."; specify a name for the folder; click the checkmark.

The folder appears in the Features list; expand to see the contents.

- **Collapse the Feature list** - Click the Features and parts icon on the right side of the Feature list to collapse the list entirely. Click the icon again to reopen the Feature list.

- Use the **Part context menu** - In the Parts list, use the context menu (RMB) to access commands for the parts, surfaces, and curves, including but not limited to:
  - **Rename** - Enter a new name for the item.
  - **Properties** - Edit metadata for the item.
  - **Assign material** - Use the Onshape material library or a custom library (if available) to assign a material (and therefore density) to the item.
  - **Edit appearance** - Change the color of the item.
  - **Copy** - Make a copy of the item in order to paste it somewhere else.
  - **Create drawing** - Create an Onshape drawing of the selected item.
  - **Export** - Export the item.
  - **Release** - Create a Release candidate and begin the release process for the selected item.
  - **Revision history** - Display the revision history of the selected item. If there is no revision history, a message is displayed.
• **Hide** - Hide the item from view.
• **Hide other** - Hide items other than the one selected.
• **Hide all** - Hide all items, including the one selected.
• **Isolate** - Make other items muted in color and unavailable for selection with the ability to expand opacity by either distance to the selected item or by connectivity to the selected item.
• **Make transparent** - Make the selected item transparent.
• **Add comment** - Add a comment and tag the selected item in the comment.
• **Zoom to selection** - Zoom the window so that the entire item is visible and centered in the graphics area, such that the largest possible view of the item is created.
• **Delete** - Discard the selected item.

**Feature and Part Lists: iOS**

The Part Studio Feature list consists of a list of features and a list of parts. The pane on the left of the window includes, starting at the top:

• The number of features in the Part Studio
• The list itself includes the default geometry of origin and planes
• Each feature in the Part Studio
• The rollback bar
• A bar that separates the parts/curves/surfaces list from the features
• Each part in the Part Studio (created in the Part Studio or derived)
• Each surface and also curve (created in the Part Studio or derived)
- Tap the Feature list handle to open the Feature list to default width.
- Touch and drag the handle vertically or horizontally to set the size (height or width) of the Feature list.
- Tap the handle to close the Feature list, tap again to open it to its previously set width.

The workspace is still active when the Feature list is open.

Features are accompanied by the tool icon which created them. This enables you to rename with descriptive names and still be able to tell what kind of Feature is represented. The model displayed in the Graphics area is visualized up to the position of the Rollback bar in the Feature list.

**Working with the Feature list**

The Feature list contains a list of all sketches and features created in the Part Studio. It also contains a list of parts, as seen towards the bottom of the list. There are many ways to work with the Feature list:

- **Drag the rollback bar** - Visualize a model at the point of the rollback bar; all features listed beneath the rollback bar become temporarily suppressed. You are also able to tap the overflow menu associated with a feature in the list, and select "Roll to here" for immediate and precise rollback.
To drag the rollback bar to a specific point in the Feature list: tap **Reorder** in the upper right then drag the rollback bar. The graphics area updates. Tap **Done** in the upper right to place the rollback bar where it is and continue modeling. You must have copy or edit permission on the document in order to use the rollback bar.

- **Make selections** - Tap a feature, sketch, or part name in the list to supply a selection for a dialog (or make the selection in the graphics area).

- **Reorder features** - Tap the Reorder features icon in the upper right corner of the dialog then touch and drag a feature or sketch name in the list to parametrically reorder them.

- **Suppress a feature** - Tap the overflow menu of the feature you wish to suppress, then select "Suppress" to visualize the model without that feature. (You are also able to unsuppress a feature.)

- **Hide or show features** - Tap the Hide/Show icon to the right of the name of the feature, sketch, or part that you wish to hide or show.

**Feature and Part Lists: Android**

The Part Studio Feature list consists of a list of features and a list of parts. The pane on the left of the window includes, starting at the top:

- A drop down menu listing available configurations for the Part Studio (Configurations are created on the browser system and are able to be seen and selected on Android mobile devices)

- The number of features in the Part Studio

- The list itself includes the default geometry of origin and planes

- Each feature in the Part Studio

- The rollback bar

- A bar that separates the parts/curves/surfaces list from the features

- Each part in the Part Studio (created in the Part Studio or derived)

- Each surface and also curve (created in the Part Studio or derived)
Tap the Feature list handle to open the Feature list to default width.

Touch and drag the lower right corner of the Feature list border to set the size (height or width) of the Feature list.

Tap the handle to close the Feature list, tap again to open it to its previously set width.

Tap the down arrow next to the Configuration (if configurations are present in the Part Studio) to select a particular configuration.

The workspace is still active when the Feature list is open.

Features are accompanied by the tool icon which created them. This enables you to rename with descriptive names and still be able to tell what kind of Feature is represented. The model displayed in the Graphics area is visualized up to the position of the Rollback bar in the Feature list.
Working with the Feature list

The Feature list contains a list of all sketches and features created in the Part Studio. It also contains a list of parts, as seen towards the bottom of the list. There are many ways to work with the Feature list:

- **Drag the rollback bar** - Visualize a model at the point of the rollback bar; all features listed beneath the rollback bar become temporarily suppressed. You are also able to tap the overflow menu associated with a feature in the list, and select "Roll to here" for immediate and precise rollback.

To drag the rollback bar to a specific point in the Feature list: tap the Reorder features icon in the upper right corner of the dialog then touch and drag the rollback bar. The graphics area updates. Tap the icon in the upper right again, to place the rollback bar where it is and continue modeling.

- **Make selections** - Tap a feature, sketch, or part name in the list to supply a selection for a dialog (or make the selection in the graphics area).

- **Reorder features** - Tap the Reorder features icon in the upper right corner of the dialog then touch and drag a feature or sketch name in the list to parametrically reorder them.

- **Suppress a feature** - Tap the overflow menu of the feature you wish to suppress, then select "Suppress" to visualize the model without that feature. (You can also unsuppress a feature.)

- **Hide or show features** - Tap the Hide/Show icon to the right of the name of the feature, sketch, or part that you wish to hide or show.

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Customizing Parts: Materials

This functionality is available on Onshape's browser, iOS, and Android platforms.

Using the context menu for a specific part (or group of selected parts) you are able to customize not only the color of the part, but also assign a material (and thereby, a
density), as well as specify a custom density without assigning a material. Onshape provides a material library for your use, and you are also able to add (or remove) your own custom material libraries.

**Customizing Materials: Desktop**

**Assigning materials to parts**

You have the ability to assign a material to a part (or group of selected parts) through the context menu. When material is assigned to a part, the "Mass Properties Tool" on page 349 then also displays density-related information.

To assign material to a part:

1. Select a part (or group of parts), right-click to open the context menu and select **Assign material**.
2. Select the Library tab.
3. Select a material library from the first drop down menu (the Onshape Material Library is displayed by default, if present):

   ![Material Library](image)

   4. Select a material from the second drop down menu. Note that each material has a density value listed with it.
5. Click ![Checkmark](image)

When assigning materials, note that:

- Parts with no material assigned have zero mass.
- Units are shown in the current document units.
You are able to search for a material by entering the name or category in the search box:

Assigning a custom density to a part

You have the ability to assign a custom material name and corresponding density to a part (or group of selected parts) through the Custom tab in the Material dialog.

To assign a custom density and a material name to a part:

1. Select a part (or group of parts), right-click to open the context menu and select Assign material.
2. Select the Custom tab.
3. Enter a name for material and a value for the density:
4. Click.

The value specified is used to calculate Mass properties. Note that the material name and value are not stored for future use with other parts, but you can enter the same information for other parts if you wish.

Creating new material libraries

You are able to add material libraries to your Onshape account and share it with other
users as well. For company accounts, the company admin is able to add the material library, and it automatically becomes available to all company members once the admin checks the box on the Company > Preferences page in My account.

The main workflow for adding a custom material library is:

1. Export the Onshape material library from Onshape (use the format of this library and simply edit the contents).
2. Import the library into a spreadsheet editing tool and edit it to contain the information you want. Export the file.
3. Import the library into a new Onshape document.
4. Share the document with others so they may also use it.

More detailed instructions follow.

Export the Onshape material library

1. While in your Onshape account, search the Public filter for the std.mat document, or use this link:

   Onshape standard material library (opens in new tab) (std.mat)

2. Open the document.

3. Right-click on the std.mat tab and select Export.

The tab is exported as a CSV file (in UTF-8 format).

Import the library into a spreadsheet, edit, and export

1. Open the CSV file of the exported Onshape material library in a spreadsheet editor. Note that the file must be in UTF-8 format.

2. Leave the first row as the necessary column names: Category, Name, Density.

   Note that densities are always recorded in the spreadsheet as kg/m³.

3. Edit the rows as desired to record your custom materials. Feel free to use any categories that serve your purpose.

   Note that you are able to create multiple custom material libraries to aid in organizing materials for your users.
4. Save the spreadsheet with a new name.

5. Export the spreadsheet as a CSV file (in UTF-8 format) from the editor.

**Import the edited library to Onshape**

1. Create a new document in your Onshape account. The name is irrelevant because the name of the document is not used for the name of the library; the name of the tab is used for the name of the library.

2. In the new document, click to create another tab, shown below (to hold the material library).

![Create Material Library](image)

3. Right-click on the new **Material Library tab** and select **Update**.

4. In the dialog, select the custom material library CSV file (in UTF-8 format), and click **Open**.

5. The file contents is displayed, if this is the correct file, click **OK**.
If it’s not the right file, click [Cancel] and restart at step 2, above.

6. Click [Create] to create a version of this document. (Give the version a name and click Create.)

7. Delete unnecessary tabs from the document, leaving just the Material Library tab.

8. Rename the tab to be what you want the library name to be and appear as in the Material dialog.
Share the library with others, if desired

To share access to the new library with other users:

1. Click Share.

2. Select the individuals, Team, or company to share the document (and thereby the library) with.

3. The minimum permission must be View.

4. Refer the other users to Adding a material library in "Managing custom material libraries" on the next page, above, for instructions.

Using new custom material libraries

Even if the new material library has not been added to your Preferences, you are still able to use it through the Material dialog:

1. While in a Part Studio with geometry present, right-click the part or part name and select Assign material.

2. Click the plus sign icon next to the library name.
3. Use the filters to search through the material libraries available to you:

Click a column header to sort by that column, if desired.

4. Click Add.

5. Notice the new library name in the Material dialog.

**Managing custom material libraries**

Onshape provides a default material library for your use, and you are also able to create, add, and remove material libraries.

Adding a material library

> Note that to add a material library, you must have access to a library added through the steps in "Creating new material libraries" on page 324

If a material library document has been shared with you (you need View permission on that document), you can simply add the material library to your Preferences list in your account settings.

1. Click your name in the top right corner of the Onshape window.

2. Select **My account**.

3. Select the **Preferences** tab on the left.
4. Scroll to the bottom of the displayed page to see Material libraries:

![Image of My account Preferences]

5. Click the **Add material library** button.

6. In the Add material library dialog, use the filters to help find the material library you wish to add.

7. Select the material library and click **Add**.

The new material library is added to the list on the Preferences page. This new library will be available through the Material library dialog in all other documents to which you have access.

**Removing a library**

Be aware that the Remove action is immediate. No confirmation is required.

1. On the **Preferences** tab of your account settings, scroll to the bottom of the page to **Material libraries**.

2. Click the **Remove** button next to the library you wish to remove.
You can remove the Onshape material library if you wish. To add it back, use this document: Onshape standard material library (std.mat)

Updating a material library

If you need to add or remove materials from a custom material library that has already been added to your Onshape account:

1. Make the necessary changes in the material library CSV file (in UTF-8 format) and save it.
2. In Onshape, open the document containing the custom material library (as a tab).
3. Right-click on the custom material library tab and select Update.
4. Select the updated CSV file and click OK.
5. Make a version to make the changes available to all users the document has been shared with.

Users are not notified that the material library has changed, but have immediate access to the updated version. Any parts to which a material have been assigned are not updated with any changes made to the library. If a particular material specification has been changed, the material has to be reapplied to the part in order for changes to take effect.

Customizing Materials: iOS

In a Part Studio, you are able to assign a material (and thereby, a density) to a Part. You are also able to assign an appearance (color and/or transparency) to a part.

Assigning materials to parts

You have the ability to assign a material to a part or group of parts. When material is assigned to a part, the Mass Properties Tool then displays density-related information.

To assign material to a part:
1. Select a part in the Feature list and select the corresponding overflow menu.

2. Select Material, a Material dialog opens.

   ![Material dialog](image)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>Undefined</td>
<td></td>
</tr>
</tbody>
</table>

3. Select the drop down within the dialog, and scroll and tap to select a property.

   After you select a property, the property values are shown in the dialog.

   ![Material dialog](image)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>2.52e-1</td>
<td>lb/in^3</td>
</tr>
</tbody>
</table>

**Customizing Materials: Android**

In a Part Studio, you have the ability to assign a material (and thereby, a density) to a Part. You also have the ability to assign an **appearance** (color and/or transparency) to a part.

**Assigning materials to parts**

You have the ability to assign a material to a part or group of parts. When material is assigned to a part, the [Mass Properties Tool](#) then displays density-related information.

To assign material to a part:
1. Select a part in the Feature list and select the corresponding overflow menu.

![Feature list with overflow menu](image)

2. Select Material, a Material dialog opens.

![Material dialog](image)

3. Select the drop down within the dialog, and scroll and tap to select a property.

After you select a property, the property values are shown in the dialog.
Visualizing Curvature

This functionality is currently available only on Onshape's browser platform.

Visualize the curvature combs on a sketch or part in a Part Studio.

Steps

Select one or more sketch curves or Part edges:

1. Open context menu (RMB-click) and select Show curvature.

2. Use the slider at the bottom of the dialog to adjust the magnitude of the combs,
   use the check boxes to choose whether or not to show curvature combs, inflection points, or minimum radius.
Example of what a sketch curve looks like with Show curvature combs, Show sketch inflection points, and Show minimum radius all selected in the dialog box.
Image example of what a sketch curve looks like with only Show curvature combs selected in the dialog box.

3. Click and drag a curve to adjust the curvature, if necessary. The combs update dynamically while you drag.

4. When finished, close the Curvature dialog; click ✗.

With the slider towards left of center:
With the slider right of center:
**Showing curvature during feature creation**

You are also able to show the curvature combs for a feature in-process, for example, during an extrude:

1. With the feature dialog open, right-click in the graphics area and select Show curvature to open the Show curvature dialog:
2. Select a curve for the feature:

3. You can show the curvature for the selected edge used to create the new feature (uncheck Show for previewed edges), or select Show for previewed edges to see the curvature combs for the edge of the new feature being created.

You are also able to show curvature combs, inflection points, and the minimum radius by selecting the boxes to the left of those options in the Show curvature dialog box (shown below).
Example of a feature with Show for previewed edges, Show curvature combs, Show inflection points, and Show minimum radius all selected in the Show curvature dialog box.

**Isolating Parts**

This functionality is available on Onshape's browser, iOS, and Android platforms.

When modeling part that are geometrically reliant on one another, and when editing in-context, Onshape provides some convenient tools you can use for better accessibility to parts that may be hidden by other parts:
• Hide (selection)
• Hide other parts
• Hide all parts
• Isolate (selection)
• Hide part on hover shortcut key “y”

Use these commands to access the parts required for your tasks, instead of painstakingly finding and moving parts out of the way to access the relevant entities.

Use the shortcut key “y” to hide a part under the cursor (part will show highlighting on hover). To show the part again, use the context menu “show all” or “show all parts” commands.

Access these commands from the context menu for selected parts in a Part Studio.

**Hiding parts**

Hide all parts, selected parts, or ‘all other’ parts excluding those selected to aid in visualizing necessary entities. Select parts in the graphics area or from the Parts list; box select also works for selecting. This command is modal: hide/show.

**Hide example**

Select the parts to hide and click Hide in the context menu. The selected parts before Hide:
The model after Hide:

 Hide other parts example

Select the parts you want to visualize and click Hide other parts in the context menu:

All other parts are hidden except those selected.

**Isolating parts**

Shortcut: Shift-i

Isolate works similarly to Hide, with the difference that unselected parts remain visually present for reference, but muted in color and unavailable for selection until you exit Isolate mode. Any Mate connectors of non-selected parts are also muted and
unavailable for selection. As with the Hide commands, you are able to select the parts in the Feature list, graphics area, and with the box select functions.

Use Isolate with individual parts, multiple parts, and groups.

Notice the dialog that appears. Use the Expand distance slider to slowly select parts in order of proximity to the isolated parts:

To exit the Isolate command, you can:
• Click the X on the dialog to close the dialog and exit Isolate.
• Open the context menu and select, Exit Isolate.
• Use the Esc (escape) key.

---

**Measure Tool**

![Measure Tool Image](image)

This functionality is available on Onshape's browser, iOS, and Android platforms.

The Onshape Measure tool is available in Part Studios, for sketches and parts, and in Assemblies for parts and assemblies. The Measure tool appears in the bottom right corner of the interface in Part Studios and Assemblies, (shown below, in a screenshot from Onshape's desktop application).
In addition, selecting any entity from the graphics area also automatically shows the pertinent measurements without having to explicitly open the Measure dialog.

**Measure Tool: Desktop**

The Measure tool displays measurements dynamically whenever you select entities.

1. Select any entity. The measure tool shows measurements for that entity.
2. Select another entity. The measure tool shows measurements between the entities.

When a sketch entity is selected, a sketch entity icon indicates the type.

**Using values**

You are able to use the information displayed to enter values elsewhere in the system, for example, as a dimension.

1. With the Measurement dialog expanded, click to highlight the value you want to copy. One click captures the maximum precision value, clicking a second time captures the lower precision.
2. Use keyboard shortcuts to copy the value.

**Interpreting the measure information**

When you hover over Measurement information in the flyout, the measurement is visualized in the graphics area, depicting the exact measurement referred to. For example:

Sketch entity icons indicate the type of entity selected
Minimum distances between entities are shown as bold dotted lines:

- Changes in X are shown in red
- Changes in Y are shown in green
- Changes in Z are shown in blue
- Center distances are shown in black

Note that when measuring to the center of a circle, you can select a planar face, edge, and edge of a cylinder

Angles appear as thin dotted lines:

**Measure Tool: iOS**

**Steps**

1. Tap the Measure tool to active it.
With the Measure tool active:

2. Tap to select desired entities to measure.

   The relevant measurement information appears in the dialog at the bottom of the screen. Tap the top of the dialog to collapse or expand it. Swipe inside the dialog to scroll up or down.

3. Tap the X in the upper right corner of the Measure dialog to close it and exit the measurement tool.

   ![Measurement Info](image)

   Note that the measurement tool displays information using the units of measurement that you have set for your document. To change the units of measurement, select **Units** from the document info panel and select desired unit.

**Measure Tool: Android**

The Onshape Measure tool is available in Part Studios, for sketches and parts, and in Assemblies for parts and assemblies; it appears in the far right of the main toolbar at the top of the screen. It displays measurement dynamically whenever you select an entity. The tool shows all possible measurements for the selected entity/entities.
including, but not limited to, minimum distance between entities, total surface area, angles, length, parallel distance, etc.

**Steps**

1. Tap the Measure tool to active it.

   ![Measure tool icon]

   With the Measure tool active:

2. Tap to select desired entities to measure.

   The relevant measurement information appears in the dialog at the bottom of the screen. Tap the top of the dialog to collapse or expand it. Swipe inside the dialog to scroll up or down.

3. Tap the X in the upper right corner of the Measure dialog to close it and exit the measurement tool.

   ![Measurement info]

   Measurement Info
   Min dist: 0.385 in
   Min dist: ΔX:0.119 ΔY:0.107 ΔZ:0.350 in
   Center dist: 0.350 in
   Center dist: ΔX:0.000 ΔY:0.000 ΔZ:0.350 in
Mass Properties Tool

This functionality is available on Onshape's browser, iOS, and Android platforms.

The Onshape Mass properties tool is available in Part Studio and Assemblies for parts and assemblies. Find the Mass properties tool in the bottom right corner of the interface, the scales icon. (Shown below in a screenshot from Onshape's browser application.)

You can also use this tool to override properties and supply your own values for mass, center of mass, and inertia.

Properties are additive: for each additional part you select, its properties are added to the calculations in the dialog. When you apply materials to parts, the density of the material is used in the calculations in the Mass properties dialog. If a part has no material assigned, no figure for that part is used in the calculation (and a note is displayed in the dialog to that effect).
Results of mass property calculations are approximate. The calculation of the properties can vary in accuracy, depending on the complexity of the geometry.

Enabling Show calculation variance displays the value and the difference between the lower and upper bound of the calculated value. If Show calculation variance is not enabled, the computed value without the bounds is displayed. The computations of the values are not affected by the state of the Show calculation variance checkbox.

**Materials** may be applied to parts through the context menu on a part in the Parts list (or the graphics area).

**Mass Properties Tool: Desktop**

**Steps**

1. To access the Mass Properties dialog, select a part in the Parts list. (You can also open the dialog first and select after.)

2. Click the small scale icon that appears in the bottom right corner of the interface.
3. If you wish to override calculations and enter your own values for Mass, Center of mass, and Mass moments of inertia, place a check in the Override box for the property you wish to override and enter the desired value. Appropriate recalculation are made once a new value is entered.

For any intersecting parts, the properties are calculated for each individual whole part and added together.

**Using values**

You have the ability to use the information displayed to enter values elsewhere in the system, for example, as a dimension.

1. With the Mass properties dialog expanded, click the value to view and highlight the max precision, click again to toggle the view to value with default decimal place setting; use shortcut keys to copy to clipboard.

2. The Mass Properties dialog provides the following information, presented from top down as shown in the tool:

- A list of selected parts - Hover over a part in the list and a small red x appears beside it. Use this x to remove the part and its properties from the dialog and calculations. Alternately, you can click the selected part in the Parts lists to deselect it.

- Select a mate connector (optional) to calculate the Moments of Inertia more accurately (instead of to the common centroid of the selected parts (as described below):
- Mass of all parts that have a material applied
- Volume of all selected parts
- Surface area of all selected parts
- Center of mass of all parts that have a material applied
- Moments of inertia - With respect to the common centroid of the selected parts (not the Part Studio origin) and reported using the densities of the materials assigned to the selected parts. Any selected parts without materials assigned are omitted from the calculation. If no materials are assigned to any selected parts, no calculation is made.

**Mass Properties Tool: iOS**

*Mass Properties tool shown in Part Studio toolbar on iOS application.*

*Mass Properties tool shown in Assemblies toolbar on iOS application.*
In a Part Studio

- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.

<table>
<thead>
<tr>
<th>Mass Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 entities selected</td>
</tr>
</tbody>
</table>

- Mass: 1.2 lb
- Volume: 14.069 in³
- Surface area: 99.893 in²
- Center of mass:
  - X: -2.521 in
  - Y: -1.73 in
  - Z: 1.448 in
- Moments of inertia: lb in²
  - Lxx: 2.477e0
  - Lxy: 0
  - Lxz: 0
  - Lyx: 0
  - Lyy: 3.8e0
  - Lyz: 0
  - Lzx: 0
  - Lzy: 0
  - Lzz: 4.27e0

The Mass properties dialog provides the following information, presented from top down as shown in the dialog.

Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document).

To change the default units for your document, tap the Document menu icon in the upper left, select **Units**, and select desired length, angle, and mass units.

- Selected entities field - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.
- Mass of all selected entities.
- Volume of all selected entities.
- Surface area of all selected entities.
- Center of mass.
- Moments of inertia - with respect to the common centroid of the selected parts (not the Part Studio origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in^3).
- If you select more than one part at a time, the combined properties of the parts are calculated.
- Any intersecting parts have the properties of each individual whole part calculated and added together.

**In an Assembly**
- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.

<table>
<thead>
<tr>
<th>Mass Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 entities selected</td>
</tr>
<tr>
<td><strong>Mass</strong>: 1.2 lb</td>
</tr>
<tr>
<td><strong>Volume</strong>: 14.069 in^3</td>
</tr>
<tr>
<td><strong>Surface area</strong>: 99.893 in^2</td>
</tr>
<tr>
<td><strong>Center of mass</strong>:</td>
</tr>
<tr>
<td>X: -2.521 in</td>
</tr>
<tr>
<td>Y: -1.73 in</td>
</tr>
<tr>
<td>Z: 1.448 in</td>
</tr>
<tr>
<td><strong>Moments of inertia</strong>: lb in^2</td>
</tr>
<tr>
<td>Lxx: 2.477e0</td>
</tr>
<tr>
<td>Lxy: 0</td>
</tr>
<tr>
<td>Lxz: 0</td>
</tr>
<tr>
<td>Lyy: 3.8e0</td>
</tr>
<tr>
<td>Lyz: 0</td>
</tr>
<tr>
<td>Lzz: 4.27e0</td>
</tr>
</tbody>
</table>

Copyright © 2017, Onshape. All rights reserved.
The Mass Properties dialog provides the following information, presented from top down as shown in the dialog.

Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document).

To change the default units for your document, tap the Document menu icon the upper left, select **Units**, and select desired length, angle, and mass units.

- Selected entities field - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.

- Mass of all selected entities.

- Volume of all selected entities.

- Surface area of all selected entities.

- Center of mass.

- Moments of inertia - with respect to the common centroid of the selected parts (not the Assembly origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in³).

- If you select more than one part at a time, the combined properties of the parts are calculated.

- Any intersecting parts have the properties of each individual whole part calculated and added together.

**Mass Properties Tool: Android**

*Mass Properties tool shown in Part Studios toolbar on Android application.*

*Mass Properties tool shown in Assemblies toolbar on Android application.*
In a Part Studio

- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.

![Mass properties dialog](image)

2 entities selected

- **Mass**: 0.134 lb
- **Volume**: 3.503 in³
- **Surface area**: 32.134 in²
- **Center of mass**:
  - X: 0.274 in
  - Y: 0 in
  - Z: 1.077 in
- **Moments of inertia**: lb in²
  - Lxx: 8.4221E⁻²
  - Lyy: 1.9177E⁻¹
  - Lzz: 2.3073E⁻¹

The Mass properties dialog provides the following information, presented from top down as shown in the dialog.

Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document).

To change the default units for your document, tap the Document menu icon in the upper left, select **Units**, and select desired length, angle, and mass units.

- **Selected entities field** - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.
• Mass of all selected entities.
• Volume of all selected entities.
• Surface area of all selected entities.
• Center of mass.
• Moments of inertia - with respect to the common centroid of the selected parts (not the Part Studio origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in^3).
• If you select more than one part at a time, the combined properties of the parts are calculated.
• Any intersecting parts have the properties of each individual whole part calculated and added together.

**In an Assembly**
• To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
• You are able to select multiple parts at a time.
• Parts may be selected in both the graphics area and the Feature list.
The Mass Properties dialog provides the following information, presented from top down as shown in the dialog.

- **Mass**: 0.134 lb
- **Volume**: 3.503 in³
- **Surface area**: 32.134 in²
- **Center of mass**: X: 0.274 in, Y: 0 in, Z: 1.077 in
- **Moments of inertia**: lb in²
  - Lxx: 8.4221E-2
  - Lyy: 1.9177E-1
  - Lzz: 2.3073E-1
  - Lxy: 0
  - Lyz: 0
  - Lzx: 0

Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document).

To change the default units for your document, tap the Document menu icon the upper left, select **Units**, and select desired length, angle, and mass units.

- **Selected entities field** - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option.
  - Deleting an entity from the list will remove it from the dialog and calculations.

- **Mass of all selected entities.**
- **Volume of all selected entities.**
- **Surface area of all selected entities.**
- **Center of mass.**
Moments of inertia - with respect to the common centroid of the selected parts (not the Assembly origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in³).

If you select more than one part at a time, the combined properties of the parts are calculated.

Any intersecting parts have the properties of each individual whole part calculated and added together.

---

**Configurations**

This functionality is currently available only on Onshape’s browser platform.

This functionality is also available on iOS and Android in a limited form.

Configurations are created only on a browser (in a Part Studio or an Assembly) but can be accessed and changed on mobile platforms, both iOS and Android. See "Using configurations" on page 377 for more details. For information creating configurations in an Assembly, see "Assembly Configurations" on page 1499.

Create part families by creating variations of an entire Part Studio or specific part. You are able to configure any feature or parameter value and even part properties, custom part properties, face and part appearance, and sketch text. For example, you are able to configure the depth of an extrude feature, the application of a fillet feature, the faces selected for a fillet, the FeatureScript of a custom feature, and part numbers, colors, and materials.

All of the features and parameters you configure in one Part Studio are referred to as a Configuration. Each Part Studio can have one Configuration. You are able to, however, create multiple Configuration inputs within one Configuration. This is especially
helpful when the feature or parameter values you want to configure are not necessarily related to each other. For example, when the length and diameter of a part are not related to whether a fillet is applied, you can use two Configuration inputs. This allows more flexibility and can aid in keeping each Configuration input from becoming unnecessarily complicated.

The Configuration inputs you define in a Part Studio become options in the Insert dialog when you are inserting parts into an Assembly or Drawing. For example, you create a Configuration input to place a flange at either the top or the left side of a sheet metal part. When inserting the sheet metal part into an Assembly, you select not only the part, but the configuration of the flange:
When more than one person is working in the same document, each sees their own selected configuration, except when working in Follow Mode; at that point the follower sees the configuration selected by the leader.

Below is an explanation of the basic steps for creating a Configuration with a single Configuration input in Onshape, and then an explanation of creating additional Configuration inputs in the same Part Studio. Lastly, there’s an explanation of configuring part properties within any Configuration input.

**Basic steps: Creating one Configuration input table**

With a model or sketch in the workspace, open the Configuration panel:

1. Click to the right-side of the graphics area (below the View cube):
2. The Configurations panel opens:
3. Click (as shown above) to open a table:

By default, the caret to the left of 'Configuration' is expanded (shown above to the right of the blue arrow), click the caret when you are done with a section of the panel to collapse that section.

4. Click in the first row to activate it and enter the names of the input in the Name column. For example, to apply a flange to different sides of a sheet metal part, you might name the rows Top, Left, Right. Use Tab to move from one row to the next.
The active row is indicated by a blue bar to the left of the row.

5. To configure a parameter value for the indicated row, click + Configure features.

6. Open the feature that contains the parameter (click it in the Feature list) and select the parameter. The parameter is then outlined with a broken yellow line and a new column is created for that parameter in the table.

For example, to configure the side of the sheet metal part to attach the flange, open the Flange feature and select the Edges or side faces to flange selection. Notice the new column in the table:
The column name defaults to the Feature name (as a top-level heading) plus the field name (as the subordinate-level heading), in this case *Extrude 1* is the Feature name and *Faces and sketch regions to extrude* is the field name.

Hover over the fields in the feature dialog to see which parameters can be configured. Parameters available for configuration are highlighted in yellow when you hover over them.

7. To edit a configured instance of the parameter:
   a. If the parameter is an entered value, click on the row in the table and enter a new value.
   b. If the parameter is a selection in a dialog, double-click the row in the table to open the feature dialog. For example: click **1 entity** in the first row.

   The appropriate field in the feature dialog is highlighted in blue. Make your selection on the model (or sketch) for this parameter.

8. When finished defining the configurations, click the check mark on the Feature dialog to close it.

9. Repeat step 6 through 8 for each row.

10. Repeat steps 5 through 8 to add another feature parameter to the table.

11. To test the inputs with the model, in the Feature list, under **Configurations**, use the down arrow to select from the menu:

The model should update accordingly. If it doesn’t, check the model for design intent and the configurations definition for accurate selection.
Cross referencing configured features

Hover over a feature in the configured table to see the feature in the model space (as shown below):

**Creating additional configuration inputs**

A Part Studio configuration can contain one or more configuration inputs. The steps above explain how to create a list type configuration input which results in a list of configuration choices when inserting a part into an Assembly or Drawing. You can create more than one of these configuration inputs (to keep one input from becoming too complicated or duplicating parameters) and also create different types of inputs. Other types of inputs you can configure are Configuration variable and Checkbox.

Once you have a configuration input defined (using the steps above), you can either add to that using the button at the top of the Configuration panel, or create additional configuration inputs using the button at the bottom of the Configuration panel:
When creating configuration inputs, you have choices on the type:

- **List** - Creates a table of feature parameters in the Part Studio and presents as a list of selections when inserting the part (or parts) into an Assembly or Drawing. (This type is explained above.)

- **Configuration variable** - Creates a variable that can be used in any feature and in FeatureScript. Types of variables include: Length, Angle, Integer, Real, and Text. Enter the value of the variable at insertion time.

- **Checkbox** - Creates a check box to turn features on or off, like Fillets and Chamfers, and can also be used to suppress or unsuppress features. This type presents a check box to check/uncheck during insertion time. Once created, use the +Configure features button to select the associated feature/s.

Step-by-step instructions follow.

**Creating a list input**
When created this way, a List input dialog is displayed. The name you give the configuration input becomes a variable in the system. This is different from the name when created using the basic steps above; that name is not a variable in the system.

1. Click the + Add configuration input button.

2. In the List input, enter a name for the configuration input.
   'Default' is supplied as the first option name; you can click it to change it.

3. Enter additional option names for the first column of the list table. Use the Tab key to add option names.

4. To configure a parameter value for each option (the selected option is indicated by a vertical blue bar to the left of the name), click + Configure features.

5. Open the feature that contains the parameter (click it in the Feature list) and select the parameter. The parameter is then outlined with a broken yellow line and a new column is created for that parameter in the table.

Hover over the fields in the feature dialog to see which parameters can be configured. Parameters available for configuration are highlighted in yellow when you hover over them.

6. To configure each instance of the parameter, double-click on the row in the table. For example: double-click 1 selection in the first row.
The appropriate field in the feature dialog is highlighted in blue. Make your selection on the model (or sketch) for this parameter.

7. When finished defining the configurations, click the check mark on the dialog to close it.

8. Repeat step 6 through 8 for each row.

9. Repeat steps 5 through 8 to add another feature parameter to the table.

10. To test the inputs with the model, in the Feature list, under **Configurations**, use the down arrow to select from the menu:

![Feature list with configurations](image)

### Creating a configuration variable input

1. Click the arrow to the right side of the **Add configuration input** button.

2. Select **Configuration variable**.

3. Enter a name for the variable input (this becomes an actual variable in the system, referenced by using #<variable-name>).

4. Select a type for the variable: Length, Angle, Integer, Real, Text. Text can be any type of text that can be used in custom FeatureScript.

5. Enter values for the type of variable you selected.

6. Click the check mark to save your definition.

7. Apply the variable to a feature:

   a. Double-click a feature in the Feature list to open it.

   b. For a sketch, you can right-click a dimension, select Configure dimensions and then either **Configuration** or **Set to #<variable-name>**
a. Close the feature dialog.

b. Test the value by selecting it in the Configurations list above the Feature list on the left side of the page:

```
<table>
<thead>
<tr>
<th>Configurations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configuration</td>
</tr>
<tr>
<td>Value</td>
</tr>
</tbody>
</table>
```

**Creating a checkbox input**

1. Click the arrow to the right side of the button.

2. Select **Checkbox**:

```
<table>
<thead>
<tr>
<th>Checkbox Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
</tr>
<tr>
<td>Default</td>
</tr>
</tbody>
</table>
```

1. Enter a name for the input.

   The configuration input has one column with an empty checkbox row and a checked checkbox row:
2. To configure a parameter value, click + Configure features.

3. Open the feature (click it in the Feature list) that contains the parameter and select the parameter. The parameter is then outlined with a broken yellow line and a new column is created for that parameter in the table. (Parameters that are configured in another input are outlined with a broken yellow line and are unavailable for configuration.)

   In this example, Unsuppressed is selected as a configuration parameter.

4. Click × to close the feature dialog.

5. In the Configuration input table, the Feature parameter column has two rows, both with checked check boxes. Uncheck the check box next to the unchecked box in the first column so the table resembles this:

   In the Configurations list, above the Feature list, this configuration input presents a checkbox to turn the Extrude on (unsuppressed) or off (suppressed).

**Editing configurations and tables**

Once a configuration is created, you can use the menu to act on it in the Part Studio:
- **Copy table** - Copy the entire Input table, you can then paste the table into a spreadsheet for record-keeping or editing. You can likewise paste from a spreadsheet back into a configuration Input table.

- **Rename** - Select this action to rename the configuration input.

- **Edit FeatureScript IDs** - This dialog will change the internal FeatureScript identifiers of the input Configuration. Click inside the dialog boxes to change IDs. When finished, select Break reference and change IDs in the lower right corner.

This dialog is typically only useful if you need to build a Part Studio inside a custom FeatureScript feature.

- **Delete** - Select this action to immediately delete the configuration input; no warning is given.
For all tables, use the context menu (right-click) to operate on rows or columns:

- **Switch to** - When right-clicking a row that is not currently selected Input, you have the menu item prefaced with "Switch to" a different Input.

- **Set as default** - When right-clicking a row that is not currently the default Input, select this to set it as the new default.

- **Duplicate row** - Create a duplicate row; this is especially convenient when preparing to paste a new table into this one. See "Copying and pasting into and out of input tables" on page 380 below, for more information.

- **Move up** - Move the selected row up one level in the table.
- **Move down** - Move the selected row down one level in the table.
- **Rename** - Rename the Input.
- **Delete row** - Delete the selected row.

You can click and drag individual column edges in the table to resize them; in the case of stacked column labels, click and drag the bottom label, indicated in the illustration above by the blue arrow.

**Configuring part properties**

Onshape has a mechanism for also configuring part properties for each of the configuration inputs and options you have previously defined, directly from the Configuration panel. The properties available to be configured include: Part name, material, appearance, description, part number, vendor, project, product line, title 1, title 2 and title 3. If you have a professional or enterprise account, you can also configure custom properties.

To configure a part property:

1. With an existing configuration input in the Configuration panel, click at the top of the panel (shown below to the left of the blue arrow):
2. Click

3. Select the part property you wish to configure (custom part properties are included in the list). (This example uses Appearance.)

A table is created with the previously selected configuration input in the first column and the part property in the second column:

4. In the Configuration column, use the down arrow to select from the list of configuration options.

5. In the Appearance column (part property), double-click to open a dialog from which to select the value (in this case, the Appearance editor).
6. Select or specify the value and the table is populated with your choice:

7. Click \( \checkmark \) to close the property dialog and accept the value.

8. To add more part properties for another configuration option, click \( \text{+} \).

9. Select a new configuration option from the first column.

10. In the Appearance column (part property), double-click to open the dialog from which to select the value.

11. Select or specify the value and the table is populated with your choice.

12. Click to close the property dialog and accept the value.
13. Repeat as necessary to configure the part properties for the necessary configuration options.

Note that when configuring part numbers and you have automatic part number generation turned on, you can right-click in the column and select Generate next part number:

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Part number</th>
</tr>
</thead>
<tbody>
<tr>
<td>short</td>
<td></td>
</tr>
<tr>
<td>tall</td>
<td></td>
</tr>
</tbody>
</table>

**Using configurations**

You are able to use the configuration inputs you create in the Part Studio to test the results and use that information to tweak design intent. However, the main point of creating configuration inputs is to provide options for the parts you use during production workflows like creating Assemblies and Drawings.

**To test configurations in Part Studios**, use the Configurations area at the top of the Feature list to select configuration input parameters to see how they affect the parts in the Part Studio:
When inserting parts into Assemblies or Drawings, select the desired inputs directly in the Insert dialog during the insertion process, on all platforms (browser, iOS and Android):

*Browser insert dialog, above*
Android insert dialog, above
In an Assembly, configured parts are indicated by the icon in the Instances list:

**iOS insert dialog, above**

**Copying and pasting into and out of input tables**
You can copy and paste into and out of a configuration input table, to aid in entering or editing input values.

To copy a configuration input table:

1. Open the menu in the upper right corner, next to +Configure features.

2. Select Copy table:

3. Once you have copied the table, you can paste it into a spreadsheet:

Note that the column names also come in with the table, as shown above. Now you can edit the table and copy/paste it back into Onshape:

1. Select just the rows and columns with data (not the column names or headings), as shown in blue below:
You can also pad your table with extra empty rows, if you wish. Just include the extra rows in the spreadsheet when selecting for the copy command.

2. Issue a Copy command.

3. In the Onshape Configuration table, click the top, left cell of the table.

4. Issue a keyboard Paste command:

Onshape automatically replaces whatever data was in the rows and columns of the configuration input table with the data that was copied. Onshape also includes the default units for each input parameter, automatically.

Note that if there are more rows copied from the spreadsheet than are in the Onshape configuration input table, those rows are included in the paste. Onshape creates the rows on the fly.

However, if there are more columns copied from the spreadsheet than are in the Onshape configuration input table, those columns are not included in the paste. Onshape does not yet create columns on the fly. You can, however, create additional columns (configured features) in the configuration table before pasting.
Sketch Basics

This functionality is available on Onshape's browser, iOS, and Android platforms.

Sketch Basics: Desktop

When creating sketches in Onshape, you use this Sketch tools toolbar:

Access the Sketch shortcut toolbar with the S key while in an active sketch (with a Sketch dialog open):

Customize the shortcut toolbar through your Onshape account Preferences page.

To customize the toolbar of Part Studios, Assemblies, or Feature Studios, see "Document Toolbar and Document Menu" on page 134.

In Onshape, sketches are created in Part Studios and consist of sketch curves (line segments, polygons, rectangles, splines, etc). Sketches are the basis for models and are stored parametrically, visible in the Feature list as its own entity.

You are able to rigidly transform geometry in an active sketch simultaneously through the context menu once sketch entities are selected.

You are able to copy sketches within a Part Studio, copy a sketch to another Part Studio, and derive a sketch for use in another Part Studio.
To access the Sketch toolbar and begin sketching, click the Create sketch icon in the Feature toolbar:

When the window is smaller, the Sketch tool may be resized to

To learn more about creating sketches in Onshape, you can follow the self-paced course here: [Sketching (opens in new tab)].

To learn about creating solid bodies and parts from sketches, using Feature tools, you can follow the self-paced course here: [Part Design Using Part Studios. (opens in new tab)]

Basic workflow

You have the ability to create as many sketches as necessary in a Part Studio and Extrude into as many parts as you want.

1. In a Part Studio, click Sketch and notice the Sketch dialog opens:

2. Select the plane to sketch on (you are only able to sketch on one plane at a time).

   You are also able to select a Mate connector (inferred or existing) as the sketch plane if you have an existing sketch or part.

   Click the icon in the sketch dialog to see inferred Mate connectors when you mouse over the sketch or part. The sketch will respect the coordinate system defined by the mate connector.
Above, the circles and squares and the centroid are inferred Mate connectors visible upon mouse-over.

Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:

3. Select a Sketch tool from the Sketch toolbar.

4. Click in the graphics area to create the sketch curve. Different tools require different numbers of clicks (as specified in those topics).

   Some tools allow you to specify dimensions while you sketch, for example:
Sketch dimension appears as the sketch curve is drawn.

When the sketch curve is drawn the suggested dimension value appears in a box. Type a value (or expression) to dimension the sketch curve.

Or, continue sketching and the curve remains un-dimensioned. Dimensions specified for two of the three line segments.

- Toggle between multiple dimension boxes using the Alt+arrow key (for example, in rectangles).
- You can also use a variable for a dimension, simply enter ";=#" and the variable name, as in: =#d.
- Dimension a sketch at a later time using the Dimension tool.

5. Use automatic inferencing to apply constraints while sketching including coplanar vertices and edges of an existing (but separate) sketch or feature. As you move the cursor over existing sketches or features, coplanar vertices and edges will 'wake up' (be highlighted) indicating you can select them to create an inferred constraint.

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For example, in the image below the gear was created with a sketch on the Top plane. In a later sketch, to create a circle on the Top plane, hover over a vertex on the gear to constrain the circle to the edge of the gear:

6. Add manual constraints as appropriate.

7. Accept the sketch and close the dialog with ✓

Canceling the sketch with x closes the dialog and does not record the sketch actions taken when the dialog was open. To reverse the action of clicking the x, click the Restore link in the message bubble that appears:

Inferencing and constraints

As you sketch and pass over points or lines, you may awaken inferences. To see all constraints, check the Show constraints checkbox in the Sketch dialog. To see only the over-defined constraints (constraints that result in the sketch being over-constrained), check the Show overdefined checkbox (checked by default) with the Show constraints box unchecked.
Line styles

As you sketch and then create models, notice the line styles of your sketches and edges of your models change or differ from each other. Read on to understand line styles in Onshape.

Sketch lines

<table>
<thead>
<tr>
<th>Active sketch</th>
<th>Selected line, active sketch</th>
<th>Inactive sketch</th>
<th>Selected line, inactive sketch</th>
</tr>
</thead>
</table>

![Sketch line styles](image)

Construction lines

<table>
<thead>
<tr>
<th>Active</th>
<th>Active, selected</th>
<th>Inactive</th>
<th>Inactive, selected</th>
</tr>
</thead>
</table>

![Construction line styles](image)

Sketch lines obstructed by model geometry

The single line in the middle of the part, below, is a construction line.
Hidden lines

The part edges are dark and solid, the sketch lines are lighter and solid, and the construction line is light and dashed (going through the middle of the part).

Used edges (projected edges)

Use (project) an edge of a part into another sketch. Below, the circular edge (highlighted) is used and results in a straight line in the new sketch:
Transforming sketches

Use the context menu > Transform sketch entities command (available when at least one sketch entity is selected) to move sketch entities simultaneously.

The manipulation triad appears, drag to manipulate selected sketch entities:
The center of the triad is used for free drag, allowing for repositioning of the triad without changing the transform operation. Free drag snaps to sketch inferences, and normal drag does not.

Drag the highlighted (above) angle indicator to rotate the sketch.

Result, below:

Pre- and post-selection is supported; entities can be added and removed during the operation.

Click off the sketch or press Enter to commit the transform and exit the operation. Press Esc to cancel the operation.

In the case of no rotation or 180 degree rotation, internal constraints are unchanged.
In the case of 90 degree or 270 degree rotation, horizontal and vertical constraints swap.

In some cases, construction geometry may be added to maintain degrees of freedom.

Directed dimensions are deleted, and may be replaced with construction geometry and minimum dimensions.

Transform is supported for images, text, DWG, and DXF:

![Diagram](image)

For more information, see "Transform Sketch" on page 573.

**Copying sketches**

Sketches must be selected in the Feature list in order to be copied and then pasted into either an open sketch, or via the Paste into sketch command from the context menu:

1. Select the sketch in the Feature list, right-click to access the context menu, and select Copy sketch.
2. Either open a sketch, right-click and select Paste into sketch on the context menu.
3. Or right-click the sketch to paste into, in the Feature list and select Paste into sketch.

**Copying sketches to another Part Studio**

1. Select a sketch in Part Studio A Feature list, right-click and select Copy sketch.
3. Either select an existing sketch, right-click and select Paste sketch entities.
4. Or create a new sketch, select a sketch plane, right-click and select Paste sketch entities.
Deriving a sketch

You must have a sketch in a Part Studio in order to derive it in another Part Studio. You need not have an existing sketch nor create a new sketch before inserting a derived sketch.

1. In the second Part Studio, click Derived

2. In the dialog, select the sketches to derive; you can select more than one.

3. Close the dialog with ✓.

Sketches are placed on the plane upon which they were created. When the original sketch is edited, the changes are reflected in the derived sketch.

Commenting on a sketch

Place comments on a particular sketch for later reference or for other collaborators. You are also able to indicate that you want to receive email notifications of other users’ comments on the sketch.

1. Right-click on the sketch in the Feature list and select Add comment.

2. Type a comment, optionally indicate that you wish to receive email notifications of others’ comments.

3. Close Comments panel.

If another user has been shared on the document and has selected Receive comment email notifications, an email is sent to that email address with the text of your comment in it.

Sketch Basics: iOS

When creating sketches in Onshape, you use the New Sketch tool and the Sketch tools.
In Onshape, sketches are created in Part Studios and consist of sketch curves (line segments, polygons, rectangles, splines, etc). Sketches are the basis for models and are stored parametrically, visible in the Feature list as its own entity.

You are able to rigidly transform geometry in an active sketch simultaneously through the context menu once sketch entities are selected.

You are able to copy sketches within a Part Studio, copy a sketch to another Part Studio, and derive a sketch for use in another Part Studio.
To access the Sketch tools and begin sketching, tap the New Sketch tool in the toolbar.

_basic workflow_

You have the ability to create as many sketches as necessary in a Part Studio and Extrude into as many parts as you want.

1. In a Part Studio, tap the New Sketch tool. The Sketch dialog opens.

2. Tap to select the plane to sketch on (you are only able to select one plane at a time).

   You are also able to select a Mate connector (inferred or existing) as the sketch plane. The sketch will respect the coordinate system defined by the mate connector.
3. Select a Sketch tool from the Sketch toolbar.

4. Tap in the graphics area to create the sketch curve. Different tools require different numbers of taps (as specified in those topics).

5. Dimension a sketch using the Dimension tool.

6. Use automatic inferencing to apply constraints while sketching.

7. Add manual constraints as appropriate.

8. Tap the checkmark to accept the sketch and close the dialog.

Canceling the sketch by tapping the x closes the dialog and does not record the sketch actions taken when the dialog was open.

Inferencing and constraints

As you sketch and pass over points or lines, you may awaken inferences. You are also able to manually add constraints using the constraint tools.

See Automatic Inferencing and Constraints for more info.

Line styles

As you sketch and then create models, notice the line styles of your sketches and edges of your models change or differ from each other. Read on to understand line styles in Onshape.
Sketch lines

Active:

Active, selected:

Inactive:
Inactive, selected:

Construction lines

Active:

Active, selected:
Inactive:

Inactive, selected:
Sketch lines obstructed by model geometry

The single line in the middle of the part below is a construction line.

Hidden lines

The part edges are dark and solid, the sketch lines are lighter and solid, and the construction line is light and dashed (going through the middle of the part).

Used edges (projected edges)

Use (project) an edge of a part into another sketch. Below, the circular edge (highlighted) is used and results in a straight line in the new sketch.

Copying sketches

You have the ability to copy and paste a sketch within, or across, Part Studios.
To copy a sketch

- Select the sketch in the Feature list, open the Context Menu, and select "Copy sketch"
- Select the overflow menu of the sketch in the Feature list and select "Copy sketch"
- While editing or creating a sketch, open the Context Menu and select "Copy sketch"

Once you have copied a sketch, you have the ability to move into a different Part Studio, and paste the sketch in a sketch or Feature in the other Part Studio.

To paste a sketch

- Select a sketch in the Feature list, open the Context Menu, and select "Paste into sketch"
- Select the overflow menu of a sketch in the Feature list and select "Paste into sketch"
- While editing or creating a sketch, open the Context Menu and select "Paste into sketch"
- While editing a feature, select a plane or planar face, open the Context Menu, and select "Paste into sketch"

Once you have pasted a sketch, touch and drag, or tap, to position the pasted sketch.

**Deriving a sketch**

You must have a sketch in a Part Studio in order to derive it in another Part Studio. You do not need to have an existing sketch, nor do you need to create a new sketch, before inserting a derived sketch.

1. While in a Part Studio, select the Derived tool.

A list of Part Studios and their features appears. If the list is lengthy, use the search box to search for a Part Studio or feature by name.
2. Tap to select a Part Studio.

3. Tap to select a sketch to derive into the current Part Studio.

See Derived for more info.

**Sketch Basics: Android**

When creating sketches in Onshape, use the New Sketch tool and the Sketch tools.

In Onshape, sketches are created in Part Studios and consist of sketch curves (line segments, polygons, rectangles, splines, etc). Sketches are the basis for models and are stored parametrically, visible in the Feature list as its own entity.

You are able to rigidly transform geometry in an active sketch simultaneously through the context menu once sketch entities are selected.

You are able to copy sketches within a Part Studio, copy a sketch to another Part Studio, and derive a sketch for use in another Part Studio.

To access the Sketch tools and begin sketching, tap the New Sketch tool in the toolbar.
Basic workflow

You have the ability to create as many sketches as necessary in a Part Studio and Extrude into as many parts as you want.

1. In a Part Studio, tap the New Sketch tool. The Sketch dialog opens.

   ![Sketch dialog](image)

   - Select a sketch plane.
   - Sketch plane
   - No entities selected
   - SHOW CONSTRAINTS

2. Tap to select the plane to sketch on (you are only able to select one plane at a time).

   You are also able to select a Mate connector (inferred or existing) as the sketch plane. The sketch will respect the coordinate system defined by the mate connector.

   ![Sketch and Mate connector](image)

3. Select a Sketch tool from the Sketch toolbar.
4. Tap in the graphics area to create the sketch curve. Different tools require different numbers of taps (as specified in those topics).

5. Dimension a sketch using the Dimension tool.

6. Use automatic inferencing to apply constraints while sketching.

7. Add manual constraints as appropriate.

8. Tap the checkmark to accept the sketch and close the dialog.

**Inferencing and constraints**

As you sketch and pass over points or lines, you may awaken inferences. You are also able to manually add constraints using the constraint tools. To see all constraints, tap the Show constraints button in the Sketch dialog.

See Automatic Inferencing and Constraints for more info.

**Line styles**

As you sketch and then create models, notice the line styles of your sketches and edges of your models change or differ from each other. Read on to understand line styles in Onshape.

**Sketch lines**

**Active:**

![Sketch lines](image)

**Active, selected:**
Inactive:

Inactive, selected:
Construction lines

Active:

Active, selected:

Inactive:
Sketch lines obstructed by model geometry

The single line in the middle of the part below is a construction line.
Hidden lines

The part edges are dark and solid, the sketch lines are lighter and solid, and the construction line is light and dashed (going through the middle of the part).

Used edges (projected edges)

Use (project) an edge of a part into another sketch. Below, the circular edge (highlighted) is used and results in a straight line in the new sketch:

Copying sketches

You are able to copy and paste a sketch within, or across, Part Studios.

To copy a sketch

- Select the sketch in the Feature list, open the Context Menu, and select "Copy sketch"
- Select the overflow menu of the sketch in the Feature list and select "Copy sketch"
- While editing or creating a sketch, open the Context Menu and select "Copy sketch"

Once you have copied a sketch, you have the ability to move into a different Part Studio, and paste the sketch in a sketch or Feature in the other Part Studio.

To paste a sketch

- Select a sketch in the Feature list, open the Context Menu, and select "Paste into sketch"
• Select the overflow menu of a sketch in the Feature list and select "Paste into sketch"
• While editing or creating a sketch, open the Context Menu and select "Paste into sketch"
• While editing a feature, select a plane or planar face, open the Context Menu, and select "Paste into sketch"

Once you have pasted a sketch, you have the option to touch and drag, or tap, to position the pasted sketch.

Deriving a sketch
You must have a sketch in a Part Studio in order to derive it in another Part Studio. You do not need to have an existing sketch, nor do you need to create a new sketch, before inserting a derived sketch.

1. While in a Part Studio, select the Derived tool.

A list of Part Studios and their features appears. If the list is lengthy, use the search box to search for a Part Studio or feature by name.

2. Tap to select a Part Studio.

3. Tap to select a sketch to derive into the current Part Studio.

See Derived for more info.

Sketch Tools
This functionality is available on Onshape's browser, iOS, and Android platforms.

Sketch Tools: Desktop
The Sketch toolbar appears when you:

- Create a new sketch by clicking the Create new sketch tool \( \text{Sketch} \) in the Feature toolbar
- Open an existing sketch for editing

It contains all the tools necessary to create a 2D sketch from which you create a 3D feature or part.

A small arrow beside a tool icon indicates a drop-down menu:

The icon beside the arrow representing the group is the last tool of that group previously chosen.

When you access Extrude or Revolve from the Sketch toolbar, your open sketch is accepted and the dialog is closed. At that point, the feature dialog (Extrude or Revolve) is opened with all regions automatically selected.

Access the Sketch shortcut toolbar with the S key while in an active sketch (with a Sketch dialog open):

Customize the Sketch shortcut toolbar through your Onshape account Preferences page.

To learn more about customizing the toolbars of Part Studios, Assemblies, or Feature Studios, see Toolbars and Document Menu.

Search tools

Shortcut: alt + C
In Onshape, you have the ability to search for tools in the Sketch toolbar as well as the Assembly toolbar and Feature toolbar: see Toolbars and Document Menu, Assemblies, and Feature tools for more information on those specific topics.

**Tips**

- The Escape key exits a tool selection.
- You are able to apply constraints (including dimensions) between sketch curves and planes.
- You are able to use expressions and trigonometric functions in numeric fields in Part Studios.
- The sketch context menu is a quick way to access many commands available for sketches.

  For more information, see "Context Menus" on page 268.

**Sketch Tools: iOS**

Sketch tools are used to create the 2D sketches that are required to make 3D features and parts.

At the top of your opened document is the main toolbar:
The main toolbar has Undo/Redo buttons to the left, Sketch and Feature tool icons to the right of the Undo/Redo buttons, and the Measure tool and Mass Properties tool to the right.

See Measure tool or Mass Properties for more info.

Tap the New sketch tool to begin a sketch:

A sketch dialog opens:
When you begin a new sketch, the New sketch tool now houses all of the sketch tools. Tap to see the list of sketch tools:
Scroll vertically to see the entire list of Sketch tools.

Note that when a sketch is open, the Construction tool moves to the main toolbar for your convenience. See [Construction](https://www.onshape.com) for more information on the Construction tool.

**Get started**

1. Select the New sketch tool.

   A sketch dialog opens.
2. Select a plane or planar face of a part.
3. Select a sketch tool and start sketching.

**Copy & Paste**

You have the ability to copy and paste a sketch within, or across, Part Studios.

To copy a sketch:
- Select the sketch in the Feature list, open the Context Menu, and select "Copy sketch"
- Select the overflow menu of the sketch in the Feature list and select "Copy sketch"
- While editing or creating a sketch, open the Context Menu and select "Copy sketch"

Once you have copied a sketch, you are able to move into a different Part Studio, and paste the sketch in a sketch or Feature in the other Part Studio.

To paste a sketch:
- Select a sketch in the Feature list, open the Context Menu, and select "Paste into sketch"
- Select the overflow menu of a sketch in the Feature list and select "Paste into sketch"
- While editing or creating a sketch, open the Context Menu and select "Paste into sketch"
- While editing a feature, select a plane or planar face, open the Context Menu, and select "Paste into sketch"

Once you have pasted a sketch, touch and drag, or tap, to position the pasted sketch.

**Tips**

- With no tools active, touch and drag points or sketch entities to adjust them.
- You are able to apply constraints (including dimensions) between sketch curves and planes.

**Sketch Tools: Android**
Sketch tools are used to create the 2D sketches that are required to make 3D features and parts.

At the top of your opened document is the main toolbar:

The main toolbar has Undo/Redo buttons to the left, Sketch and Feature tool icons to the right of the Undo/Redo buttons, and the Measure tool and Mass Properties tool to the right.

See Measure tool or Mass Properties for more info.

Tap the New sketch tool to begin a sketch:

A sketch dialog opens:
When you begin a new sketch, the New sketch tool now houses all of the sketch tools. Tap to see the list of sketch tools:

<table>
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<tr>
<th>Tool</th>
<th>Tool</th>
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<tr>
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<td>Fill arc</td>
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<td>Tangent</td>
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<td>Midpoint</td>
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</table>

Scroll vertically to see the entire list of Sketch tools.

Note that when a sketch is open, the Construction tool moves to the main toolbar for your convenience. See Construction for more information on the Construction tool.

Get started
1. Select the New sketch tool.
   A sketch dialog opens.
2. Select a plane or planar face of a part.
3. Select a sketch tool and start sketching.

Copy & Paste
You are able to copy and paste a sketch within, or across, Part Studios.

To copy a sketch:
- Select the sketch in the Feature list, open the Context Menu, and select "Copy sketch"
Select the overflow menu of the sketch in the Feature list and select "Copy sketch"

While editing or creating a sketch, open the Context Menu and select "Copy sketch"

Once you have copied a sketch, you are able to move into a different Part Studio, and paste the sketch in a sketch or Feature in the other Part Studio.

To paste a sketch:

- Select a sketch in the Feature list, open the Context Menu, and select "Paste into sketch"
- Select the overflow menu of a sketch in the Feature list and select "Paste into sketch"
- While editing or creating a sketch, open the Context Menu and select "Paste into sketch"
- While editing a feature, select a plane or planar face, open the Context Menu, and select "Paste into sketch"

Once you have pasted a sketch, you can touch and drag, or tap, to position the pasted sketch.

Tips

- With no tools active, touch and drag points or sketch entities to adjust them.
- You have the ability to apply constraints (including dimensions) between sketch curves and planes.

Line

Sketch a line segment or series of line segment.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Line: Desktop**

Shortcut: I

In the Sketch toolbar:

Sketch a line segment or series of line segment.

**Steps**

1. Click to begin and end line segments, continuing to create attached segments.
2. Or, click and drag to start one segment and release to end.

**Line: iOS**
Sketch a line segment or series of line segment.

Steps

1. Tap to select the tool.

2. Tap to set the start point of the line.

3. To set the second point of the line, either tap again or touch and drag then release.

   Repeat step two to add as many points to your line as you like.

**To begin sketching a new line you can either:**

- Double tap to end the first line, then tap to set the first point of a new line.

Or

- Tap the line tool to deselect it, then tap the line tool to select it again and begin a new line.

---

**Sketch 1**

---

**Line: Android**

Sketch a line segment or series of line segment.
Steps

1. Tap to set the start point of the line.

2. To set the second point of the line, either tap again or touch and drag then release.
   Repeat step two to add as many points to your line as you like.

To begin sketching a new line you can either:

- Double tap to end the first line, then tap to set the first point of a new line.

Or

- Tap the line tool to deselect it, then tap the line tool to select it again and begin a new line.

---

**Sketch 1**

---

**Corner Rectangle**

Sketch a rectangle starting with a corner point.

---

Copyright © 2017, Onshape. All rights reserved.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Corner Rectangle: Desktop**

Shortcut: g

In the Sketch toolbar:

Sketch a rectangle starting with a corner point.

Steps

1. Click to start a corner, click to end at diagonal corner.
2. Or click and drag from corner to diagonal corner and release.

Hold the ALT key while sketching to constrain two rectangle sides to be equal (resulting in a square).
Corner Rectangle: iOS

Sketch a rectangle starting with a corner point.

Steps

1. Tap to set the start point (a corner) of the rectangle.

2. To set the end point (opposite corner) of the rectangle, either tap again or touch and drag then release.
Corner Rectangle: Android

Sketch a rectangle starting with a corner point.

Steps

1. Tap to set the start point (a corner) of the rectangle.

2. To set the end point (opposite corner) of the rectangle, either tap again or touch and drag then release.
Center Point Rectangle

Sketch a rectangle starting with its center point.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Center Point Rectangle: Desktop

Shortcut: r

In the Sketch toolbar:
Sketch a rectangle starting with its center point.

Steps

1. Click 🔄 to set the center point, click again to set a corner.

2. Or click and drag from center point to corner and release.

Hold the ALT key while sketching to constrain two rectangle sides to be equal (resulting in a square).

---

**Center Point Rectangle: iOS**

Sketch a rectangle starting with its center point.

Steps

1. Tap 🔄 to set the start point (the center) of the rectangle.

2. To set the end point (a corner) of the rectangle, either tap 🔄 again or touch and drag then release.
Center Point Rectangle: Android

Sketch a rectangle starting with its center point.

Steps

1. Tap 📐 to set the start point (the center) of the rectangle.

2. To set the end point (a corner) of the rectangle, either tap 📐 again or touch and drag then release.
Center Point Circle

Sketch a circle starting with its center point.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Center Point Circle: Desktop

Shortcut: c

In the Sketch toolbar:
Sketch a circle starting with its center point.

Steps

1. Click 👉 to set the center point and click again to set the radius.
2. Or click and drag to set the center point, release to set the radius.

**Center Point Circle: iOS**

Sketch a circle starting with its center point.

Steps

1. Tap 👉 to set the start point (the center) of the circle
2. To set the end point (any outside point) of the circle, either tap 👉 again or touch and drag then release.
**Center Point Circle: Android**

Sketch a circle starting with its center point.

Steps

1. Tap 🔄 to set the start point (the center) of the circle

2. To set the end point (any outside point) of the circle, either tap 🔄 again or touch and drag then release.
### 3 Point Circle

Sketch a circle by defining three points along its circumference.

This functionality is available on Onshape's browser, iOS, and Android platforms.

### 3 Point Circle: Desktop

In the Sketch toolbar:

Sketch a circle by defining three points along its circumference.
Steps

1. Click 🔄 to set start point, click to set second point and click to set diameter.

2. Or click and drag to set start point, release to set diameter and click to set third point.

3 Point Circle: iOS

Sketch a circle by defining three points along its circumference.

Steps

1. Tap 🔄 to set the start point (an outside point) of the circle.

2. To set the second point (the point opposite the first), tap 🔄 again or touch and drag then release.

3. To set the diameter with the third and final point, tap 🔄 again or touch and drag then release.
3 Point Circle: Android

Sketch a circle by defining three points along its circumference.

Steps

1. Tap to set the start point (an outside point) of the circle.

2. To set the second point (the point opposite the first), tap again or touch and drag then release.

3. To set the diameter with the third and final point, tap again or touch and drag then release.
Sketch an ellipse using a center point, major axis, and minor axis.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Ellipse: Desktop**

In the Sketch toolbar:
Sketch an ellipse using a center point, major axis, and minor axis.

Steps

1. Click 🔄 to initiate ellipse.
2. Drag and click to set desired major axis.
3. Drag and click to set minor axis.

Ellipse: iOS

Sketch an ellipse using a center point, major axis, and minor axis.

Steps

1. Tap 🔄 to set the start point (center point) of the ellipse.
2. To set the major axis of the ellipse, tap 🔄 again or touch and drag then release.
3. To set the minor axis of the ellipse, tap 🔄 again or touch and drag then release.
**Ellipse: Android**

Sketch an ellipse using a center point, major axis, and minor axis.

**Steps**

1. Tap 🖐️ to set the start point (center point) of the ellipse.

2. To set the major axis of the ellipse, tap 🖐️ again or touch and drag then release.

3. To set the minor axis of the ellipse, tap 🖐️ again or touch and drag then release.
3 Point Arc

Sketch an arc by defining the two end points and then the radius point.

This functionality is available on Onshape's browser, iOS, and Android platforms.

3 Point Arc: Desktop

Shortcut: a

In the Sketch toolbar:
Sketch an arc by defining the two end points and then the radius point.

Steps

1. Click 👉 to set start point.

2. Click 👉 to set second point.

3. Click 👉 to set radius (or click and drag to set start point, release to set second point and click to set radius).

3 Point Arc: iOS

Sketch an arc by defining the two end points and then the radius point.

Steps

1. Tap 📡 to set the first end point of the arc.

2. To set the second end point of the arc, tap 📡 again or touch and drag then
release.

3. To set the radius of the arc, tap \(\rightarrow\) again or touch and drag then release.

**3 Point Arc: Android**

Sketch an arc by defining the two end points and then the radius point.

**Steps**

1. Tap \(\rightarrow\) to set the first end point of the arc.

2. To set the second end point of the arc, tap \(\rightarrow\) again or touch and drag then release.

3. To set the radius of the arc, tap \(\rightarrow\) again or touch and drag then release.
Tangent Arc

Sketch an arc at the end of a line.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Tangent Arc: Desktop

In the Sketch toolbar:
Sketch an arc at the end of a line.

Steps
1. Click .
2. Click to start.
3. Click to end.

Or click and drag to start, release to end.

Tangent Arc: iOS

Sketch an arc at the end of a line.

Steps
1. Tap at the end point of a line to set the start point of the arc.
2. To set the end point of the arc, tap or touch and drag then release.
**Tangent Arc: Android**

Sketch an arc at the end of a line.

**Steps**

1. Tap at the end point of a line to set the start point of the arc.

2. To set the end point of the arc, tap or touch and drag then release.
**Center Point Arc**

Sketch an arc by defining center, start, and end points.

This functionality is available on Onshape’s browser, iOS, and Android platforms.

**Center Point Arc: Desktop**

In the Sketch toolbar:

Sketch an arc by defining center, start, and end points.
Steps

1. Click 🖕️.
2. Click a center point on a sketch entity.
3. Click start point.
4. Click end point.

Center Point Arc: iOS

Sketch an arc by defining center, start, and end points.

Steps

1. Tap 🖕️ to set the start point (center point) of the arc.
2. To set one end point of the arc, tap 🖕️ again or touch and drag then release.
3. To set the other end point of the arc, tap 🖕️ again or touch and drag then release.
Center Point Arc: Android

Sketch an arc by defining center, start, and end points.
Steps

1. Tap \( \bullet \) to set the start point (center point) of the arc.

2. To set one end point of the arc, tap \( \bullet \) again or touch and drag then release.

3. To set the other end point of the arc, tap \( \bullet \) again or touch and drag then release.
Conic

Sketch an ellipse, parabola, or hyperbola by defining start, end, and control points, and a rho value.

This functionality is currently available only on Onshape's browser platform.

In the Sketch toolbar:

Steps

1. Click 🔄.
2. Click to indicate a start point.
3. Click to indicate an end point.
4. Click to indicate a control point.

5. Enter a rho value.

For example, various rho values yield different shapes:
- Greater than 0.5 yields a hyperbola shape
- 0.5 yields a parabola shape
- Less than 0.5 yields an elliptical shape
Inscribed Polygon

Sketch an inscribed polygon (polygon on the outside of the drawn circle).

This functionality is available on Onshape's browser, iOS, and Android platforms.

Inscribed Polygon: Desktop

In the Sketch toolbar:

Sketch an inscribed polygon (polygon on the outside of the drawn circle).
Steps

1. Click 🏠.
2. Click to start.
3. Drag to set the circumference.

Notice that the circle drawn for the polygon uses the construction flag.

4. At this point you have two options:
   a. Use the keyboard to enter the number of sides. Then click to lock the circumference and a value field appears for the number of sides: you can use the keyboard again or drag to set the number of sides. Click again or press Enter.
   b. Click to lock the circumference and a value field appears for the sides: you can use the keyboard again or drag to set the number of sides. Click again or press Enter.

   Drag towards the polygon to reduce the number of sides, click to set. Minimum sides = 3.

   Drag away from the polygon to increase the number of sides, click to set. Maximum sides = 50.

Inscribed Polygon: iOS
Sketch an inscribed polygon (polygon on the outside of the drawn circle).

Steps

1. Tap to set the start point (center point) of the polygon.

2. To set the second point (radius) of the polygon, tap again or touch and drag then release.

   The number of sides appears in the middle of the polygon.

   - Touch and drag counterclockwise around the center point to decrease the number of sides the polygon has. Minimum sides = 3.

   - Touch and drag clockwise around the center point to increase the number of sides the polygon has. Maximum sides = 50.

When you release, the number of sides is set.
Inscribed Polygon: Android

Note that after you sketch a polygon, the next polygon you sketch defaults to the number of sides of the previously sketched polygon.
Sketch an inscribed polygon (polygon on the outside of the drawn circle).

Steps

1. Tap 📊 to set the start point (center point) of the polygon.

2. To set the second point (radius) of the polygon, tap 📊 again or touch and drag then release.
   
   The number of sides appears in the middle of the polygon.

   ⊗ Touch and drag counterclockwise around the center point to decrease the number of sides the polygon has. Minimum sides = 3.

   ⊗ Touch and drag clockwise around the center point to increase the number of sides the polygon has. Maximum sides = 50.

   When you release, the number of sides is set.
Note that after you sketch a polygon, the next polygon you sketch defaults to the number of sides of the previously sketched polygon.
**Circumscribed Polygon**

Sketch a circumscribed polygon (polygon on the inside of the drawn circle).

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Circumscribed Polygon: Desktop**

In the Sketch toolbar:

Sketch a circumscribed polygon (polygon on the inside of the drawn circle).

Steps

1. Click 🟢.

2. Click to start.

3. Drag to set the circumference.

   Notice that the circle drawn for the polygon uses the construction flag.

4. At this point you have two options:

   a. Use the keyboard to enter the number of sides. Then click to lock the circumference and a value field appears for the number of sides: you can use the keyboard again or drag to set the number of sides. Click again or press Enter.

   b. Click to lock the circumference and a value field appears for the sides: you can use the keyboard again or drag to set the number of sides. Click again or press Enter.

   Drag towards the polygon to reduce the number of sides, click to set. Minimum sides = 3.
Drag away from the polygon to increase the number of sides, click to set. Maximum sides = 50.

**Circumscribed Polygon: iOS**

Sketch a circumscribed polygon (polygon on the inside of the drawn circle).

Steps

1. Tap to set the start point (center point) of the polygon.

2. To set the second point (radius) of the polygon, tap again or touch and drag then release.
   
   The number of sides appears in the middle of the polygon.

- Touch and drag counterclockwise around the center point to decrease the number of sides the polygon has. Minimum sides = 3.

- Touch and drag clockwise around the center point to increase the number of sides the polygon has. Maximum sides = 50.

   When you release, the number of sides is set.
Note that after you sketch a polygon, the next polygon you sketch defaults to the number of sides of the previously sketched polygon.

Circumscribed Polygon: Android

Sketch a circumscribed polygon (polygon on the inside of the drawn circle).

Steps

1. Tap to set the start point (center point) of the polygon.

2. To set the second point (radius) of the polygon, tap again or touch and drag then release.

   The number of sides appears in the middle of the polygon.

   • Touch and drag counterclockwise around the center point to decrease the number of sides the polygon has. Minimum sides = 3.

   • Touch and drag clockwise around the center point to increase the number of sides the polygon has. Maximum sides = 50.

When you release, the number of sides is set.
Note that after you sketch a polygon, the next polygon you sketch defaults to the number of sides of the previously sketched polygon.

**Spline**

Sketch a multiple point curve with points along its length.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Spline: Desktop**

In the Sketch toolbar:
Sketch a multiple point curve with points along its length.

Steps

1. Click .
2. Click to start, click to establish points, double-click to end.
3. Click and drag any point along the spline to make adjustments.
4. Use the tangent handles (the white points) to modify the tangency of the spline curve. Tangent handles cannot be deleted.

You can use the spline handles (or any spline points) with dimensions:
You can use spline handles (or any spline points) with constraints:

Tips

- Splines may be closed by connecting the trailing point to the first point only.
- Spline tangent handles (the white points) may be moved along the spline or pulled “away” from the spline to adjust the tangency of the curve.
- You can add more spline points with the Spline point tool.
- Visualize the curvature in more detail with the Show curvature context menu com-
Spline: iOS

Sketch a multiple point curve with points along its length.

Steps

1. Tap to set the first point of the spline.

2. To set the next point of the spline, either tap again or touch and drag to release.

To begin sketching a new spline you can either:

• Double tap to end the first spline, then tap to set the first point of a new spline.

OR

• Tap the spline tool to deselect it, then tap the spline tool to select it again and begin a new line.
Once you finish sketching a spline, with the sketch open, you can touch and drag any point along the spline to reposition it.

You can use the spline handles (or any spline points) with dimensions:
You can use spline handles (or any spline points) with constraints:

**Spline: Android**

Sketch a multiple point curve with points along its length.

Steps

1. Tap to set the first point of the spline.

2. To set the next point of the spline, either tap again or touch and drag to release.

To begin sketching a new spline you can either:

- Double tap to end the first spline, then tap to set the first point of a new spline.
OR

- Tap the spline tool to deselect it, then tap the spline tool to select it again and begin a new line.

Once you finish sketching a spline, with the sketch open, you can touch and drag any point along the spline to reposition it.

You can use the spline handles (or any spline points) with dimensions:
You can use spline handles (or any spline points) with constraints:

**Spline Point**

Add points along a spline.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Spline Point: Desktop**
In the Sketch toolbar:

Add points along a spline.

Steps

1. Click .

2. Click anywhere along a spline to add points to it.

3. Drag the points to modify the spline.

For more information about splines, tangency handles (the white points), and creating splines, see "Spline" on page 460.

**Spline Point: iOS**

Add points along a spline.

Steps

1. Tap anywhere along a spline to add a point to it.

2. Touch and drag the points to modify the spline.
### Spline Point: Android

Add points along a spline.

**Steps**

1. Tap anywhere along a spline to add a point to it.
2. Touch and drag the points to modify the spline.
Point

Create points.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Point: Desktop**

In the Sketch toolbar:

Create points.
Steps

1. Click ◊.
2. Click to create a point.

You are able to apply many of the same sketch constraints that you can to other sketch entities.

---

**Point: iOS**

Create points.

Steps

1. Tap to set a point.
2. Double tap to stop setting points.
Point: Android

Create points.

Steps

1. Tap to set a point.
2. Double tap to stop setting points.
Add up to 250 characters of text to a sketch (you are able to copy and paste into the text dialog). Treat sketch text as most other sketch entities: extrude, dimension, and apply constraints.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Text: Desktop

In the Sketch toolbar:
Add up to 250 characters of text to a sketch (you are able to copy and paste into the text dialog). Treat sketch text as most other sketch entities: extrude, dimension, and apply constraints.

Steps

1. Click A.

2. Click and drag to establish the position and size of the text box. (The lower left corner and the height define the text position and size of the first line of text.)

   Note that a horizontal constraint is applied to the lower edge of the box by default.

In the dialog that appears:

1. Enter the text as you wish it to appear. (You are able to see a preview of your text in the preview box at the top of the dialog.)

2. Select a font from the drop-down menu.

   Be sure to select a font that supports your language.

4. Optionally, select one of the icons to flip the text:

- Flip the text about the horizontal center of the text frame
- Flip the text about the vertical center of the text frame

For example, when the above text is flipped about the vertical center of the text frame, the result is this:

![Flipped Text]

**Tips**

- To edit existing sketch text, select Edit text from the context menu.
- There is a limit of 250 characters per text box.
- When entering more than one line of text, the first line of text appears in the box and the subsequent lines appear 'outside' of the box. This is because the box represents the line length and height (text baseline to text ascender).
You are able to dimension the text box.

- Dragging the box without fixing or constraining it moves the box; note that the box may not be visible during the move.
- To rotate the box, remove the horizontal constraint on the lower edge, apply a Fix constraint to one corner, then drag an opposite corner.
- To resize the box, dragging a corner (make sure one other corner is fixed). If the box has been dimensioned, you are not able to resize it by dragging.
- Note that when you drag the text box, you see only the text box until the box is stationary again, then you see the text.

Text: iOS

Add up to 250 characters of text to a sketch (you are able to copy and paste into the text dialog). Treat sketch text as most other sketch entities: extrude, dimension, and apply constraints.

Steps

1. With a Sketch open, tap 📑
2. Tap to set starting point (corner) of the text box.
3. Tap to set the end point (opposite corner) of the text box.

Note that a horizontal constraint is applied to the lower edge of the box by default.

The Text dialog opens.

4. Type to enter text. You are able to enter up to 250 characters.
5. Tap in the font field to view a list of fonts to chose from.
6. Tap **B** or *I* to toggle bold or italic text, respectively.

7. Tap checkmark.

<table>
<thead>
<tr>
<th>Text</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>OpenSans</td>
<td>≥</td>
<td></td>
</tr>
<tr>
<td><strong>B</strong></td>
<td></td>
<td><em>I</em></td>
</tr>
<tr>
<td>Default text</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Tips

• To edit existing sketch text, tap on an entity of the text (a part of a character or a text box line), then select Edit Text from the context menu.

Note: two-finger tap to bring up the context menu. See "Context Menus" on page 268 for more info.

• There is a limit of 250 characters per text box.
When you hit return to enter more than one line of text, the first line of text appears in the box and subsequent lines appear 'outside' of the box. This is because the box represents the line length and height (text baseline to text ascender).

You are able to dimension the text box.

Dragging the box without fixing or constraining a point moves the box; note that the box may not be visible during the move.

To rotate the box, remove the horizontal constraint on the lower edge, apply a Fix constraint to one corner, then drag an opposite corner.

To resize the box, drag a corner (make sure the other corner is fixed). If the box has been dimensioned, you are not able to resize it by dragging.

Note that when you drag the text box, you see only the text box until you lift your finger, then you see the text.

Text: Android

Add up to 250 characters of text to a sketch (you are able to copy and paste into the text dialog). Treat sketch text as most other sketch entities: extrude, dimension, and apply constraints.

Steps

1. With a Sketch open, tap \[A\].

2. Tap to set starting point (corner) of the text box.

3. Tap to set the end point (opposite corner) of the text box.

   Note that a horizontal constraint is applied to the lower edge of the box by default.

The Text dialog opens.

4. Type to enter text. You are able to enter up to 250 characters.

5. Tap in the font field to view a list of fonts to chose from.

6. Tap \[B\] or \[I\] to toggle bold or italic text, respectively.

7. Tap checkmark.
Tips

- To edit existing sketch text, tap on an entity of the text (a part of a character or a text box line), then select Edit Text from the context menu.

Note: two-finger tap to bring up the context menu. See "Context Menus" on page 268 for more info.
• There is a limit of 250 characters per text box.
• When you hit return to enter more than one line of text, the first line of text appears in the box and subsequent lines appear 'outside' of the box. This is because the box represents the line length and height (text baseline to text ascender).
• You are able to dimension the text box.
• Dragging the box without fixing or constraining a point moves the box; note that the box may not be visible during the move.
• To rotate the box, remove the horizontal constraint on the lower edge, apply a Fix constraint to one corner, then drag an opposite corner.
• To resize the box, drag a corner (make sure the other corner is fixed). If the box has been dimensioned, you are not able to resize it by dragging.
• Note that when you drag the text box, you see only the text box until you lift your finger, then you see the text.

Use

Project (or convert) an edge, edges, and silhouettes of a part or sketch onto the active sketch plane.

This functionality is available on Onshape’s browser, iOS, and Android platforms.

Use: Desktop

Shortcut: u

In the Sketch toolbar:
Project (or convert) an edge, edges, and silhouettes of a part or sketch onto the active sketch plane.

Steps

1. Create a sketch or part.
2. Start another sketch.
3. Click \[\square\], then an edge, edges, or silhouettes from the first sketch or part.

In this example, the highlighted edge of the part was selected to use (project) onto the sketch plane, resulting in the highlighted horizontal line.

When hovering over entities, a preview of the projected lines will appear.

Using silhouettes

When viewing a model normal to a sketch any visible boundary that isn't an edge is a silhouette; where the surface transitions from facing you to facing away from you.

How does using a silhouette work?

1. Click \[\square\].

   Note that there is NO pre-select behavior for silhouettes.
2. Hover over the face for which you want a silhouette.

You should see highlights. If the actual silhouette is out of the sketch plane you will see two. One that is the 'real' silhouette, one which is the projection in the sketch plane. Both are selectable.

When hovering over entities, a preview of the projected lines will appear.

If no highlights are visible, you may be running into a limitation, see Tips below.

3. Hover on and then click on a highlight to project that silhouette. (The highlight being hovered over in order to be selected, above, is shown in yellow highlight.)
Note that when multiple silhouettes are available, you are able to click a face to select all silhouettes, or hover over an individual silhouette and click to select just the ones you wish to use:

This is what the highlight looks like during hover:

After selection and projection onto another plane:
Tips

- All used edges update when the underlying geometry changes. However, this doesn't react well to changes of geometry type (circle to line, etc.) caused by model changes.

- Some things about Onshape Use may be different from other systems, including:
  - Onshape does not constrain the ends of the silhouette. You are able to choose how to fix the ends.
  - Onshape does not distinguish between "bits" of silhouettes, like in this example of a cylinder with a hole through it:
Onshape does not use a face, like the cylinder above with a hole through it, and automatically extract either edges or silhouettes and sew them all together.

Onshape only uses silhouettes that are trackable. This is enables a level of certainty the silhouette may still be updated later.

Supported silhouettes include: cylinders, cones, tori, spheres, extruded surfaces, and any surface with one silhouette.

Silhouettes that are self-intersecting after projection are not usable.

**Use: iOS**

Project (or convert) an edge, edges, and silhouettes of a part or sketch onto the active sketch plane.

Steps

1. Create a sketch or part.
2. Start another sketch.
3. Tap ✶, then select an edge, edges, or silhouette from the first sketch or part.

Here, the highlighted face was selected to use (project) onto the Right sketch plane, resulting in the black sketch lines that form a rectangle on the Right sketch plane.
Using silhouettes

When viewing a model normal to a sketch any visible boundary that isn't an edge is a silhouette; where the surface transitions from facing you to facing away from you.

How does using a silhouette work?

1. Tap .
   
   Note that there is NO pre-select behavior for silhouettes.

2. Use the Precision selector to hover over the face for which you want a silhouette.
   
   See "Selection" on page 219 for more info.
The projection of the silhouette appears in pink on the active sketch plane.

3. Hover over (line the cross-hairs up with) a specific pink silhouette section and it turns yellow to indicate that it is selected. Release to select it and that silhouette is now projected onto your current sketch plane.
4. If you do not specify a selection by hovering over a pink silhouette to turn it yellow, then releasing to select will project all of the silhouette options.

This may be used as a more efficient way of projecting an entire silhouette at once.

Tips

- All used edges update when the underlying geometry changes. However, this doesn't react well to changes of geometry type (circle to line, etc.) caused by model
Some things about Onshape Use may be different from other systems, including:

- Onshape does not constrain the ends of the silhouette. You are able to choose how to fix the ends.
- Onshape does not distinguish between "bits" of silhouettes, like in this example of a cylinder with a hole through it:

![Image of a cylinder with a hole through it.]

- Onshape does not use a face, like the cylinder above with a hole through it, and automatically extract either edges or silhouettes and sew them all together.
- Onshape only uses silhouettes that are trackable. This enables that the silhouette can still be updated later.
- Supported silhouettes include cylinders, cones, tori, spheres, extruded surfaces, and any surface with one silhouette.
- Silhouettes that are self-intersecting after projection are not usable.

**Use: Android**

Project (or convert) an edge, edges, and silhouettes of a part or sketch onto the active sketch plane.
Steps

1. Create a sketch or part.
2. Start another sketch.
3. Tap  
   , then select an edge, edges, or silhouette from the first sketch or part.

Here, the highlighted face was selected to use (project) onto the Right sketch plane, resulting in the black sketch lines that form a rectangle on the Right sketch plane.

Using silhouettes

When viewing a model normal to a sketch any visible boundary that isn't an edge is a silhouette; where the surface transitions from facing you to facing away from you.

How does using a silhouette work?

1. Tap  
   .

   Note that there is NO pre-select behavior for silhouettes.

2. Use the Precision selector to hover over the face for which you want a silhouette.
   See "Selection" on page 219 for more info.
The projection of the silhouette appears in pink on the active sketch plane.

3. Hover over (line the cross-hairs up with) a specific pink silhouette section and it turns yellow to indicate that it is selected. Release to select it and that silhouette is now projected onto your current sketch plane.
4. If you do not specify a selection by hovering over a pink silhouette to turn it yellow, then releasing to select will project all of the silhouette options.

Use this as a more efficient way of projecting an entire silhouette at once.

Tips
- All used edges update when the underlying geometry changes. However, this doesn't react well to changes of geometry type (circle to line, etc.) caused by model changes.
Some things about Onshape Use may be different from other systems, including:

- Onshape does not constrain the ends of the silhouette. You are able to choose how to fix the ends.
- Onshape does not distinguish between "bits" of silhouettes, like in this example of a cylinder with a hole through it:

![Sketch 3]

- Onshape does not use a face, like the cylinder above with a hole through it, and automatically extract either edges or silhouettes and sew them all together.
- Onshape only uses silhouettes that are trackable. This enables that the silhouette may still be updated later.
- Supported silhouettes include cylinders, cones, tori, spheres, extruded surfaces, and any surface with one silhouette.
- Silhouettes that are self-intersecting after projection are not usable.

Intersection

Project (or convert) the intersection of a surface or face and the active sketch plane onto the sketch plane.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Intersection: Desktop**

In the Sketch toolbar:

Project (or convert) the intersection of a surface or face and the active sketch plane onto the sketch plane.

**Steps**

1. Create a sketch using an intersecting plane as the sketch plane.
2. Select 📋 from the Use icon dropdown menu.
3. Select the surface or face with which to create the new sketch.

The resulting sketch seen with the part, below.

![Intersection: Desktop](image)

**Tips**

- The sketch updates when the underlying geometry changes.
- The sketch is constrained with the Intersection and Pierce constraints where the sketch intersects the plane or surface/face of the original model.

**Intersection: iOS**
Project (or convert) the intersection of a surface or face and the active sketch plane onto the sketch plane.

Steps

1. Create a sketch using an intersecting plane as the sketch plane.

2. Select ![image](image-url).

3. Select the surface or face with which to create the new sketch.

The resulting sketch seen with the part, below.

Tips

- The sketch updates when the underlying geometry changes.
- The sketch is constrained with the Intersection constraint, and the Pierce constraint where the sketch intersects the plane or surface/face of the original model.

**Intersection: Android**

Project (or convert) the intersection of a surface or face and the active sketch plane onto the sketch plane.

Steps

1. Create a sketch using an intersecting plane as the sketch plane.

2. Select ![image](image-url).

3. Select the surface or face with which to create the new sketch.

The resulting sketch seen with the part, below.
Tips

- The sketch updates when the underlying geometry changes.
- The sketch is constrained with the Intersection constraint, and the Pierce constraint where the sketch intersects the plane or surface/face of the original model.

Construction

Sketch new construction geometry or convert existing geometry into construction geometry. Construction geometry are sketch entities used in creating other geometry but not used in creating features.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Construction: Desktop

Shortcut: q (to toggle Construction state on and off)

In the Sketch toolbar:
Sketch new construction geometry or convert existing geometry into construction geometry. Construction geometry are sketch entities used in creating other geometry but not used in creating features.

Steps

There are two approaches one may take when drawing construction geometry:

- Draw the sketch entities first, select the sketch entities to toggle, then select \textbf{\textdollar}: 
Select $\cdot$, then a sketch tool and draw the sketch entities in Construction mode:

Tips

- Select $\cdot$ and then a sketch tool to create construction geometry.
- Select sketch entities and then $\cdot$ to toggle construction mode on and off.

Construction: iOS

In the Sketch toolbar:

If you are in a sketch, the Construction tool is located to the right of the Feature tool icon.
1. Sketch new construction geometry or convert existing geometry into construction geometry.

2. Select the sketch entity that you want to put into construction mode.

3. Then, with desired sketch entity selected, select the Construction tool to set the entity as a construction.

Any entity in construction mode is also able to be toggled out of construction mode in the same way.

Alternatively, select the Construction tool, then begin sketching in construction mode.

The Construction tool works for all sketch entities. Below are just a few examples.

Construction: Android

In the Sketch toolbar:
If you are in a sketch, the Construction tool is located to the right of the Feature tool icon.

1. Sketch new construction geometry or convert existing geometry into construction geometry.
2. Select the sketch entity that you want to put into construction mode.
3. Then, with desired sketch entity selected, select the Construction tool to set the entity as a construction.

Any entity in construction mode is also able to be toggled out of construction mode in the same way.

Alternatively, select the Construction tool, then begin sketching in construction mode.

The Construction tool works for all sketch entities. Below are just a few examples.
Create fillets or rounds with a specified radius along one or more lines, arc, and splines. This functionality is available on Onshape's browser, iOS, and Android platforms.

**Fillet (Sketch): Desktop**
Shortcut: Shift-f

In the Sketch toolbar:

Create fillets or rounds with a specified radius along one or more lines, arc, and splines.

Steps

1. Click or press Shift-f.

2. Select a point or two sketch curves.

3. The radius dialog opens, click in the dialog and enter the radius.

Tips

To apply more than one fillet of the same size, make the first selection, enter the radius, then click the other points to fillet. The first value entered becomes the default and that size fillet will be applied to all selected points. You are able to change the first value to change all values.
If the default fillet size is too large for a selected point, the fillet line is displayed in red.

You are able to drag the manipulator to resize the fillet.

You are also able to click to activate the value and enter a new value for the fillet size:
Examples

**Line, make selections**

1. Click first line; no highlighting occurs.
2. Click and drag second line to estimate size of fillet.
3. Enter value for fillet radius and press Enter.

**Vertex, make selections**

1. Click \( \bigcirc \).
2. Click a vertex.
3. Enter the radius value and press Enter.
4. Each subsequent click with the Fillet tool selected results in equal-sized fillets.
Spline, make selections

1. Click the .
2. Click the left spline near the top (not the point).
3. Click the right spline near the top (not the point).
4. Enter the fillet radius value and press Enter.

You may notice a small, open circle after the fillet is applied, where the lines used to meet. This is a virtual sharp that is added to the sketch as reference geometry. This virtual sharp will retain the coincident constraints on the two lines, as well as a dimension (radius of the fillet). You may want to use this as a reference point for adding constraints, for example. You may also choose to simply ignore it. (See the "Vertex, make selections" example above to see the virtual sharp.)

Fillet (Sketch): iOS

Sketch new construction geometry or convert existing geometry into construction geometry. Construction geometry are sketch entities used in creating other geometry but not used in creating features.

Steps

1. Tap to select the point of a corner to fillet. You are also able to select two sketch curves.
2. Use the Number Pad to enter a value for the radius of the fillet.
You may notice a small, open circle after the fillet is applied, where the lines used to meet. This is a virtual sharp that is added to the sketch as reference geometry. This virtual sharp will retain the coincident constrains on the two lines as well as a
dimension (radius of the fillet). You may want to use this as a reference point for adding constraints, for example.

**Fillet (Sketch): Android**

Sketch new construction geometry or convert existing geometry into construction geometry. Construction geometry are sketch entities used in creating other geometry but not used in creating features.

Steps

1. Tap to select the point of a corner to fillet. You are also able to select two sketch curves.

2. Use the Number Pad to enter a value for the radius of the fillet.
You may notice a small, open circle after the fillet is applied, where the lines used to meet. This is a virtual sharp that is added to the sketch as reference geometry. This virtual sharp will retain the coincident constrains on the two lines as well as a dimension (radius of the fillet). You may want to use this as a reference point for adding constraints, for example.

**Trim**

Trim a curve to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the entire curve is deleted.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Trim: Desktop**
Shortcut: m

In the Sketch toolbar:

Trim a curve to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the entire curve is deleted.

Steps

1. Click ⊳.

2. Click entities to trim away.

Trim: iOS

Trim a curve to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the entire curve is deleted.
Steps

1. Tap to select a curve segment to trim.

2. Continue tapping to select more curve segments to trim.

If you select a curve segment that has intersecting geometry, the trim only trims up to the intersecting geometry. If you select a curve segment that has no intersecting geometry, the trim will delete the segment entirely.
Sketch 1

Trim: Android
Trim a curve to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the entire curve is deleted.

Steps
1. Tap to select a curve segment to trim.
2. Continue tapping to select more curve segments to trim.

If you select a curve segment that has intersecting geometry, the trim only trims up to the intersecting geometry. If you select a curve segment that has no intersecting geometry, the trim will delete the segment entirely.
Extend

Extend a line to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the line ends at the release point.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Extend: Desktop
Shortcut: x

In the Sketch toolbar:

Extend a line to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the line ends at the release point.

Steps

No intersecting or boundary geometry:

1. Click |.
2. Click the point to extend.
3. Click new location for the point.

Intersecting geometry:
Extend: iOS

Extend a line to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the line ends at the release point.

Steps

1. Tap \( \rightarrow \) to select the line or arc to extend.

2. Tap \( \rightarrow \) to set the new location for the end point of the line or arc.

If there is intersecting geometry found, the extension snaps to that geometry.
Extend a line with no intersecting geometry
Extend a line with intersecting geometry
**Extend: Android**

Extend a line to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the line ends at the release point.

**Steps**

1. Tap to select the line or arc to extend.

2. Tap to set the new location for the end point of the line or arc.

If there is intersecting geometry found, the extension snaps to that geometry.
Extend a line with no intersecting geometry
Extend a line with intersecting geometry
**Sketch Split**

Split open or closed sketch curves into multiple segments. Open curves require one or more points to split with; closed curves require two or more points.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sketch Split: Desktop**

In the Sketch toolbar:
Split open or closed sketch curves into multiple segments. Open curves require one or more points to split with; closed curves require two or more points.

Steps

1. Click 🕒.

2. Click the sketch curve to split; click one or more locations along the curve.

Tips

- Click on one or more points to split an open curve.
- Click two or more points to split a closed curve.

**Split Entity: iOS**

Split open or closed sketch curves into multiple segments. Open curves require one or more points to split with; closed curves require two or more points.

Steps

1. Tap 🕒 to place a point to split with.

2. If required (if you are splitting a closed curve) tap to set another point to split with.
Split an open curve

Sketch 1
Split a closed curve

Sketch 1

**Split Entity: Android**

Split open or closed sketch curves into multiple segments. Open curves require one or more points to split with; closed curves require two or more points.

**Steps**

1. Tap ![point icon] to place a point to split with.

2. If required (if you are splitting a closed curve) tap to set another point to split with.
Split an open curve

Sketch 1
Split a closed curve

Offset

Offset the selected curve or loop at a specified distance and direction from the original.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Offset: Desktop
Shortcut: o

In the Sketch toolbar:

Offset the selected curve or loop at a specified distance and direction from the original.

Steps

1. Click or press the o key.
2. Select one or more curves to offset.
3. Indicate the direction (click the direction arrow) and enter distance value of the offset.
   You are also able to use the arrow to drag the offset distance to a value or closer to the correct value and then type in the final value.
   Note that to change the direction of the offset, you can also use a negative distance value.
4. Press Enter.
5. If needed, click on additional curves to set offsets of equal distance.
Select a single entity

Chain select a loop

**Offset: iOS**

Offset the selected curve or loop at a specified distance and direction from the original.
Steps

1. Tap 🔄 to select an entity to offset.

2. Tap another entity to continue selecting entities to offset.

3. To set the offset distance and direction either:
   - Touch and drag the manipulator arrow.
   OR
   - Tap the arrow to flip the direction and tap anywhere to open the number pad and enter a specific value for the distance.
**Offset: Android**

Offset the selected curve or loop at a specified distance and direction from the original.

**Steps**

1. Tap 🔄 to select an entity to offset.
2. Tap another entity to continue selecting entities to offset.
3. To set the offset distance and direction either:
   - Touch and drag the manipulator arrow.
   - OR
   - Tap the arrow to flip the direction and tap anywhere to open the number pad and enter a specific value for the distance.
Slot

Create a slot around selected sketch curves (including splines, lines, arcs but no closed profiles).

This functionality is available on Onshape's browser, iOS, and Android platforms.

Slot: Desktop

In the Sketch toolbar:

Create a slot around selected sketch curves (including splines, lines, arcs but no closed profiles).

Steps

Pre-selecting sketch curves and then applying Slot creates slots of equal size across all curves, linked together with one dimension:

1. Select sketch curves (either individually or with box select):
The user has the option to either select the Slot tool itself first and then select the centerlines to create slots from, or select the centerlines first, followed by the Slot tool.

2. Click \( - \).

3. Double-click the dimension to edit it.

   Diameter dimension controls the width of the slot and can be edited before or after the slot is finished being placed.

Chain selection

Applying the Slot command to individual sketch curves in sequence, links the dimensions of the slots:

1. Select one sketch curve:
2. Click 🐸.

3. With the Slot command still selected, click more sketch curves to apply the same (unlinked) dimension:
4. When you edit the dimension, all slots are changed:

**Slot: iOS**

Use the Slot tool in either of two ways:

- Select the Slot tool and then the individual sketch curve.
- Pre-select sketch curves and then apply Slot to create slots of equal size across all curves, linked together with one dimension.

**Steps**

1. Tap 💡.
2. Tap to select a sketch curve.
   
   A slot is created around that sketch curve.
3. Tap 💡 to begin creating a slot around a different sketch curve.
4. Tap to select a sketch curve.
Pre-select steps

1. Tap to preselect sketch curves.

2. Tap to apply slots of equal sizes across all curves, linked together with one dimension.
**Slot: Android**

Use the Slot tool in either of two ways:

- Select and then the individual sketch curve.
- Pre-select sketch curves and then apply Slot to create slots of equal size across all curves, linked together with one dimension.

**Steps**

1. Tap.
2. Tap to select a sketch curve.
   
   A slot is created around that sketch curve.
3. Tap to begin creating a slot around a different sketch curve.
4. Tap to select a sketch curve.
Sketch 1

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Pre-select steps

1. Tap \( \text{\ding{257}} \) to preselect sketch curves.

2. Tap \( \text{\ding{257}} \) to apply slots of equal sizes across all curves, linked together with one dimension.
Mirror (Sketch)

Create the reflection of one or more selected sketch entities about a specified line.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Mirror (Sketch): Desktop

In the Sketch toolbar:
Create the reflection of one or more selected sketch entities about a specified line.

Use the Mirror tool in either of two ways: first pre-select the entities to mirror and then the tool, or select the tool and then the entities.

Steps: Pre-selected entities

1. Select one or more sketch entities.

2. Click ☒. (You are prompted to select the mirror line.)

3. Select the line across which to mirror: a sketch line, a plane, or a part face or edge.
4. As soon as you click the mirror line, the sketch resolves:

Steps: No pre-selected entities

1. Click 📝. (You are prompted to select a mirror line.)

2. Select the line. (You are prompted to select entities.)

3. As you select entities, they are mirrored. When you are finished, press ESC to exit the Mirror tool.

Mirror (Sketch): iOS
Mirror one or more selected sketch entities about a specified line.

You have the ability to use the Mirror tool in either of two ways:

- Select the tool and then the mirror line and the entities.
- Pre-select the mirror line, then select the tool, then select the entities to mirror.

Steps

1. Tap \( \text{[Line]} \).
2. Tap to select a line across which to mirror.
3. Tap to select the entities to mirror.
Pre-select steps

1. Tap to select a line across which to mirror.

2. Tap \[ \text{[ ]} \].

3. Tap to select the entities to mirror.
Mirror (Sketch): Android
Mirror one or more selected sketch entities about a specified line.

You have the ability to use the Mirror tool in either of two ways:

- Select the tool and then the mirror line and the entities.
- Pre-select the mirror line, then select the tool, then select the entities to mirror.

Steps

1. Tap to select the Mirror tool.
2. Tap to select a line across which to mirror.
3. Tap to select the entities to mirror.
Pre-select steps

1. Tap to select a line across which to mirror.

2. Tap to select the Mirror tool.

3. Tap to select the entities to mirror.
Linear Sketch Pattern

Create multiple instances of sketch entities uniformly in one or two directions.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Linear Sketch Pattern: Desktop

In the Sketch toolbar:

Create multiple instances of sketch entities uniformly in one or two directions.

Steps

1. Select the sketch entity or entities to pattern and then click 📚:

The initial pattern created is:
Double-click to enter the number of instances in the pattern. Press Enter twice.

Click and drag the arrow head to change the distance between entities; drag the arrow base to move the pattern at an angle.

Double-click to specify the distance between entities.

2. Move the mouse to white space and notice the icon is a mouse with a green button. Click to accept and set the sketch pattern.

Tips

- You are able to delete any sketch entity in the pattern without affecting the integrity of the pattern. Changing the number of sketch entity occurrences does not reinstate a deleted entity; the space for that entity is left empty.

- In linear sketch patterns, you are able to drag the arrow manipulator's base to position the pattern at an angle (the base is shown highlighted below):
When you create more than ten pattern entities, the system shows only up to the first 10 to prevent dips in performance from generating the preview.
For more information, see "Circular Sketch Pattern" on page 565.

**Linear Sketch Pattern: iOS**

Create multiple instances of sketch entities uniformly in one or two directions.

**Steps**

1. Tap the Linear sketch pattern tool.

2. Tap to select the sketch or sketches to pattern.

   An initial pattern is created. The distance between patterned sketch entities is displayed as a number. The number of sketch entity instances is displayed as "numberX".

3. Double tap on a distance or instance value to edit it.

4. Touch and drag the square manipulators to adjust the axis of that row of instances.

5. Touch and drag the arrow manipulators to adjust the distance between entities.

6. Double tap to finish editing the linear sketch pattern.
Tips

- You are able to delete any sketch entity in the pattern without affecting the integrity of the pattern. Changing the number of sketch entity occurrences does not reinstate a deleted entity; the space for that entity is left empty.

- When you generate more than 10 pattern entities, the system shows only up to the first 10 to prevent dips in performance from generating the preview.

For information on circular sketch patterns, see "Circular Sketch Pattern" on page 565.

Linear Sketch Pattern: Android

Create multiple instances of sketch entities uniformly in one or two directions.

Steps

1. Tap the Linear sketch pattern tool.

2. Tap to select the sketch or sketches to pattern.

   An initial pattern is created. The distance between patterned sketch entities is displayed as a number. The number of sketch entity instances is displayed as "num-berX".
3. Double tap on a distance or instance value to edit it.
4. Touch and drag the square manipulators to adjust the axis of that row of instances.
5. Touch and drag the arrow manipulators to adjust the distance between entities.
6. Double tap to finish editing the linear sketch pattern.
Tips

- You are able to delete any sketch entity in the pattern without affecting the integrity of the pattern. Changing the number of sketch entity occurrences does not reinstate a deleted entity; the space for that entity is left empty.
When you generate more than 10 pattern entities, the system shows only up to the first 10 to prevent dips in performance from generating the preview.

For information on circular sketch patterns, see "Circular Sketch Pattern" below.

Circular Sketch Pattern

Create multiple instances of sketch entities uniformly about an axis or mate connector. Circular patterns may be open or closed, as described below.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Circular Sketch Pattern: Desktop

In the Sketch toolbar:

Create multiple instances of sketch entities uniformly about an axis or mate connector. Circular patterns may be open or closed, as described below.

Steps: Radial pattern

1. Select the sketch entity or entities to pattern and then click  

The initial pattern created is:
Steps: Entity to pattern radially (open pattern):

1. Select the sketch to pattern and then click:

2. Move the mouse to white space and notice the icon is a mouse with a green button. Click to accept and set the sketch pattern.
2. After initial pattern created, click and drag the arrow head to reduce the angle dimension and open the pattern:

![Diagram showing pattern dynamic reduction](image)

3. Once the pattern is accepted, when you change the instance count, it keeps the pattern open:

![Diagram showing pattern acceptability](image)

**Tips**

- Circular patterns default to closed patterns (360°). However, you are able to click and drag the manipulator (arrow head) to change the angle value and create an open pattern.
Circular patterns are initially created about the origin (but not constrained to it). Click and drag the center icon to reposition and resize the pattern.

You are able to delete any sketch entity in the pattern without affecting the integrity of the pattern. Changing the number of sketch entity occurrences does not reinstate a deleted entity; the space for that entity is left empty.

For information on linear sketch patterns, see "Linear Sketch Pattern" on page 557.

**Circular Sketch Pattern: iOS**

Create multiple instances of sketch entities uniformly about an axis or mate connector. Circular patterns may be open or closed, as described below.

Steps

1. Tap 🔄.

2. Tap to select the sketch or sketches to pattern.
   
   An initial closed pattern is created. The number of sketch entity instances is displayed as "numberX".

3. Double tap on the instance value to edit it.

4. Touch and drag the arrow manipulator to change the circular pattern from a closed pattern to an open pattern.

5. With an open pattern, touch and drag the arrow or double tap on the degrees value to change the axis of the pattern.

6. Touch and drag the square manipulator at the center of the pattern to adjust the placement of the origin of the pattern.

7. Double tap to finish editing the circular sketch pattern.
Tips

- Circular patterns default to closed patterns (360 degrees). However, you are able to touch and drag the manipulator arrow to change the angle value and create an open pattern.

- Circular patterns are initially created about the origin (but not constrained to it). Touch and drag the center icon to reposition and resize the pattern.

- You are able to delete any instance in the pattern without affecting the integrity of the pattern. Changing to number of sketch entity occurrences does not reinstate a deleted entity; the space for that entity is left empty.

For information on linear sketch patterns, see "Linear Sketch Pattern" on page 557.

Circular Sketch Pattern: Android

Create multiple instances of sketch entities uniformly about an axis or mate connector. Circular patterns may be open or closed, as described below.
Steps

1. Tap 💡.

2. Tap to select the sketch or sketches to pattern.
   
   An initial closed pattern is created. The number of sketch entity instances is displayed as "numberX".

3. Double tap on the instance value to edit it.

4. Touch and drag the arrow manipulator to change the circular pattern from a closed pattern to an open pattern.

5. With an open pattern, touch and drag the arrow or double tap on the degrees value to change the axis of the pattern.

6. Touch and drag the square manipulator at the center of the pattern to adjust the placement of the origin of the pattern.

7. Double tap to finish editing the circular sketch pattern.

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Tips

- Circular patterns default to closed patterns (360 degrees). However, you are able to touch and drag the manipulator arrow to change the angle value and create an open pattern.

- Circular patterns are initially created about the origin (but not constrained to it). Touch and drag the center icon to reposition and resize the pattern.

- You are able to delete any instance in the pattern without affecting the integrity of the pattern. Changing to number of sketch entity occurrences does not reinstate a deleted entity; the space for that entity is left empty.

For information on linear sketch patterns, see "Linear Sketch Pattern" on page 557.

_Tips_ are text-based explanations and tips for using Onshape, a cloud-based product design platform. They are designed to help users understand how to use Onshape's features and tools effectively.

Transform Sketch

Transform adjusts a sketch's location and orientation.

Transform Sketch: Desktop

In the Sketch toolbar:

Transform adjusts a sketch's location and orientation.

Steps

1. Click 🏡.

2. Select the sketch entities you want to move.
3. Use the manipulator to drag and orient the sketch.

4. Click in space when the sketch is placed and oriented as desired.

When rotating via the manipulator, an angle field activates. Enter an angle, press Enter, and click in space to set the angle:

Place the manipulator’s ball point to orient a snap point for the sketch:
1. Drag the ball point to a point on the sketch you wish to use as a snap point:

2. Use the plane to drag the sketch ball point to the snap point on another sketch (in this case the lower-left point of the square):

3. Click off the sketch to set the transform:

For more information, see "Transforming sketches" on page 390
Tips

• Pre- and post-selection is supported; entities can be added and removed during the operation.
• Click off the sketch or press Enter to commit the transform and exit the operation. Press Esc to cancel the operation.
• In the case of no rotation or 180 degree rotation, internal constraints are unchanged.
• In the case of 90 degree or 270 degree rotation, horizontal and vertical constraints swap.
• In some cases, construction geometry may be added to maintain degrees of freedom.
• Directed dimensions are deleted, and may be replaced with construction geometry and minimum dimensions.
• Transform is also supported for images, text, DWG, and DXF.

Transform Sketch: iOS
Transform adjusts a sketch's location and orientation.

Steps

1. With the sketch opened for editing, select the sketch entities you want to move.

2. In the Feature tool area, tap 🏡.

3. Use the manipulator to drag and orient the sketch.

4. Click in space when the sketch is placed and oriented as desired.
When rotating via the manipulator, an angle field activates. Tap in the numeric field, enter an angle, tap the checkmark, and tap in space to set the angle:

Place the manipulator’s ball point to orient a snap point for the sketch:
1. Drag the ball point to a point on the sketch you wish to use as a snap point:

2. Use the **plane** to drag the sketch ball point to the snap point on another sketch (in this case the lower-left point of the square):

3. Click off the sketch to set the transform:
Tips

- Pre- and post-selection is supported; entities may be added and removed during the operation.
- Tap off the sketch or tap the checkmark to commit the transform and exit the operation.
- In the case of no rotation or 180 degree rotation, internal constraints are unchanged.
- In the case of 90 degree or 270 degree rotation, horizontal and vertical constraints swap.
- In some cases, construction geometry may be added to maintain degrees of freedom.
- Directed dimensions are deleted, and may be replaced with construction geometry and minimum dimensions.
- Transform is also supported for images, text, DWG, and DXF.

**Transform Sketch: Android**

Transform adjusts a sketch's location and orientation.

Steps

1. With the sketch opened for editing, select the sketch entities you want to move.

2. In the Feature tool area, tap 🏡.

3. Use the manipulator to drag and orient the sketch.

4. Click in space when the sketch is placed and oriented as desired.
When rotating via the manipulator, an angle field activates. Tap in the numeric field, enter an angle, tap the checkmark, and tap in space to set the angle:

Place the manipulator’s ball point to orient a snap point for the sketch:
1. Drag the ball point to a point on the sketch you wish to use as a snap point:

2. Use the **plane** to drag the sketch ball point to the snap point on another sketch (in this case the lower-left point of the square):

3. Click off the sketch to set the transform:
Tips

• Pre- and post-selection is supported; entities may be added and removed during the operation.

• Tap off the sketch or tap the checkmark to commit the transform and exit the operation.

• In the case of no rotation or 180 degree rotation, internal constraints are unchanged.

• In the case of 90 degree or 270 degree rotation, horizontal and vertical constraints swap.

• In some cases, construction geometry may be added to maintain degrees of freedom.

• Directed dimensions are deleted, and may be replaced with construction geometry and minimum dimensions.

• Transform is also supported for images, text, DWG, and DXF.

Insert DXF or DWG

Insert DXF or DWG files into a sketch as sketch entities. The DXF or DWG must have already been imported into the currently open document (or another document you own or that has been shared with you, creating a link to that document). It is recommended that you insert DXF or DWG files into an empty sketch, though it is possible to insert into a sketch with existing sketch entities.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Insert DXF and DWG as Sketch Entities: Desktop

In the Sketch toolbar:
Insert DXF or DWG files into a sketch as sketch entities. The DXF or DWG must have already been imported into the currently open document (or another document you own or that has been shared with you, creating a link to that document). It is recommended that you insert DXF or DWG files into an empty sketch, though it is possible to insert into a sketch with existing sketch entities.

Supported formats

Currently, the supported export format is Release. The following formats are supported for import:

- Release 9
- Release 10
- Release 11
- Release 13
- Release 14
- 2000
- 2004
- 2007
- 2010
- 2013

All Onshape supported formats can be found here.

Steps

1. Click to create a new sketch.
2. Select a plane.
3. Click .
4. In the dialog that appears, select the Units (at the bottom of the dialog) for the sketch entity:
5. Optionally, check Use file origin position to position the geometry from the file relative to the current Part Studio origin in the same way the geometry is positioned relative to the DXF/DWG file origin. (Otherwise, the geometry is positioned so that the center of the geometry extents -as calculated in the form of a 2D box containing all entities- is at the Part Studio origin.)

6. Then select a DXF or DWG file (that has been previously imported in the current document), use Other documents to locate a file in another document that you have created or that has been shared with you, or use Import (at the bottom of the dialog) to import a new file to be used immediately.

   When importing a file from within this dialog, once the import is complete, the file is listed in the dialog. Select it to insert it into the sketch.

7. Selecting the file to insert or import automatically closes the dialog.
Tips

- You are able to insert DXF/DWG files that have already been imported into your document or another document that you have created or has been shared with you. These show up as tabs and also in the Insert DXF dialog.

- Make sure to select the units in the dialog first; selecting the file automatically closes the dialog.

- The Insert action is recorded in the Undo/Redo stack for the document.

- When dimensioning the inserted sketch, the first dimension applied automatically scales the entire sketch.

- If some geometry in the inserted sketch isn't supported, Onshape inserts the supported geometry and displays a message about unsupported geometry not being shown.

Insert DXF and DWG as Sketch Entities: iOS

Insert DXF or DWG files into a sketch as sketch entities. The DXF or DWG must have already been imported into the currently open document (or another document you own or that has been shared with you, creating a link to that document). It is recommended that you insert DXF or DWG files into an empty sketch, though it is possible to insert into a sketch with existing sketch entities.

Steps

1. Tap the New sketch tool to create a new sketch.

2. Select a plane.

3. Tap the Insert tool.

4. In the dialog that appears, select a drawing from either your current workspace, or browser other documents. You are also able to search for a drawing by name.

5. Tap to select the drawing you want to insert.

6. Toggle Use file origin position on, to position the geometry from the file relative to the current Part Studio origin in the same way the geometry is positioned relative to the DXF/DWG file origin. (Otherwise, the geometry is positioned so that the cen-
ter of the geometry extents - as calculated in the form of a 2D box containing all entities - is at the Part Studio origin.)

7. Tap on the blue unit to select units.

8. Tap **Insert Drawing**.

Supported formats

Currently, the supported export format is Release. The following formats are supported for import:

- Release 9
- Release 10
- Release 11
- Release 13
- Release 14
- 2000
- 2004
- 2007
- 2010
- 2013

All Onshape supported formats can be found here.
Tips

- You are able to insert DXF/DWG files that have already been imported into your document or another document that you have created or has been shared with you. These show up as tabs and also in the Insert DXF dialog.

- Make sure to select the units in the dialog first; selecting the file automatically closes the dialog.

- The Insert action is recorded in the Undo/Redo stack for the document.

- When dimensioning the inserted sketch, the first dimension applied automatically scales the entire sketch.

- If some geometry in the inserted sketch isn’t supported, Onshape inserts the supported geometry and displays a message about unsupported geometry not being shown.

Insert DXF and DWG as Sketch Entities: Android

Insert DXF or DWG files into a sketch as sketch entities. The DXF or DWG must have already been imported into the currently open document (or another document you own or that has been shared with you, creating a link to that document). It is recommended that you insert DXF or DWG files into an empty sketch, though it is possible to insert into a sketch with existing sketch entities.

Steps

1. Tap the New sketch tool to create a new sketch.

2. Select a plane.

3. Tap the Insert tool.

4. In the dialog that appears, select a drawing from either your current workspace, or browser other documents. You are also able to search for a drawing by name.

5. Tap to select the drawing you want to insert.

6. Toggle Use file origin position on, to position the geometry from the file relative to the current Part Studio origin in the same way the geometry is positioned relative to the DXF/DWG file origin. (Otherwise, the geometry is positioned so that the cen-
ter of the geometry extents - as calculated in the form of a 2D box containing all entities - is at the Part Studio origin.)

7. Tap on the blue unit to select units.

8. Tap **Insert Drawing**.

Supported formats

Currently, the supported export format is Release. The following formats are supported for import:

- Release 9
- Release 10
- Release 11
- Release 13
- Release 14
- 2000
- 2004
- 2007
- 2010
- 2013

All Onshape supported formats can be found here.
Tips

- You are able to insert DXF/DWG files that have already been imported into your document or another document that you have created or has been shared with you. These show up as tabs and also in the Insert DXF dialog.
- Make sure to select the units in the dialog first; selecting the file automatically closes the dialog.
- The Insert action is recorded in the Undo/Redo stack for the document.
- When dimensioning the inserted sketch, the first dimension applied automatically scales the entire sketch.
- If some geometry in the inserted sketch isn't supported, Onshape inserts the supported geometry and displays a message about unsupported geometry not being shown.

Insert Image

Use an imported image as a basis for a sketch. Upload an image to your document, then open that image in a sketch. Create sketch geometry using the image as a guide. Supported image types include: PNG, JPEG, and BMP.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Insert Image: Desktop

In the Sketch toolbar:

Use an imported image as a basis for a sketch. Upload an image to your document, then open that image in a sketch. Create sketch geometry using the image as a guide. Supported image types include: PNG, JPEG, and BMP.
Steps

1. Click 📷.

2. In the dialog that appears (by default it appears on top of the Feature list), enter a search phrase to locate an image file, or select one from the list. If there are no image files listed, use the Import link at the bottom of the dialog.

   You are also able to click Browse documents in the dialog to browse for a document that has an image file already uploaded. Inserting an image from another Onshape document (that you own or has been shared with you) links the documents. You are only able to link documents that have at least one version created.

3. Click and drag to position the image in the graphics area. (The aspect ratio of the image is maintained and indicated by a dashed line as you drag.)

   Note that a horizontal constraint is applied to the lower edge of the box by default.
4. To reposition the image, delete the Horizontal constraint (click and press Delete), then click and drag the image:

Note that the image becomes semi-transparent as you move it, for better visibility during placement.

5. To rotate the image, fix one corner and drag another.

You can sketch on top of the image. Dimensioning the sketch geometry the first time scales the image as well.

Tips

- To rotate an image, remove the horizontal constraint, fix one corner and drag another corner.
- To move an image, remove all constraints and click and drag the image to the desired location.
• You can sketch on top of the image.

• Dimensioning sketch geometry scales the image as well, but only the first dimension applied scales the image. To rescale the image, remove additional dimensions and adjust the remaining one dimension.

• When you Show/Hide the sketch, the image is also shown or hidden.

• This feature respects the alpha channel, so if it is transparent, it will remain that way in Onshape.

• You can copy/paste an image (as a sketch entity) within a Part Studio and from one Part Studio to another.

• You can use the context menu and Edit image command to select another image file or upload a new one.

• If the source of the image changes, it may be updated in its tab using the Update option on the tab context menu. This also updates the image wherever it is used in the document.

Insert Image: iOS and Android

Use an imported image or a photo as a basis for a sketch. Upload an image to your document from your mobile device or select a photo from your mobile device, then open that image in a sketch. Create sketch geometry using the image as a guide. Supported image types include: PNG, JPEG, and BMP.

Steps

1. Tap the New sketch tool to create a new sketch.

2. Select a plane.

3. Tap 📸.

   In the dialog that appears, select search criteria to locate an image file in the current document or another document, or select a photo from the device.

   To use an image in another document, that document must first be versioned. You are also able to search for a drawing by name.

4. Tap to select the drawing you want to insert.
5. Tap the check mark to close the dialog.

6. Tap to set the first corner of the image, and tap again to set the opposite corner.
   (The aspect ratio of the image is maintained and indicated by a dashed line as you drag.)

7. To rotate the image, fix one corner and drag another.

8. You can sketch on top of the image. Dimensioning the sketch geometry the first time scales the image as well:

   Inserted Images linked to from another document are indicated in the Feature list by a link icon.

   ![Link Icon]

   When a newer version of the document from which you inserted the image is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab.

   ![Link Icon]

   Tips

   • To rotate an image, remove the horizontal constraint, fix one corner and drag another corner.

   • To move an image, remove all constraints and click and drag the image to the desired location.

   • You can sketch on top of the image.

   • Dimensioning sketch geometry scales the image as well, but only the first dimension applied scales the image. To rescale the image, remove additional dimensions and adjust the remaining one dimension.

   • When you Show/Hide the sketch, the image is also shown or hidden.

   • This feature respects the alpha channel, so if it is transparent, it will remain that way in Onshape.

   • You can copy/paste an image (as a sketch entity) within a Part Studio and from one Part Studio to another.
If the source of the image changes, it may be updated in the Update option on the tab context menu. This also updates the image wherever it is used in the document.

**Dimension**

Add horizontal, vertical, shortest distance, angular, diametrical, arc length, or radial dimensions to sketch geometry and between sketch geometry and planes. You are able to specify dimensions as driven (reference) or driving.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Dimension: Desktop**

Shortcut: d

In the Sketch toolbar:

Add horizontal, vertical, shortest distance, angular, diametrical, arc length, or radial dimensions to sketch geometry and between sketch geometry and planes. You can specify dimensions as driven (reference) or driving.

You have the ability to use the Measure tool to measure anything in the graphics area.

Some tools allow you to dimension as you sketch.

You can also use the Show dimensions command in the context menu (RMB) of a sketch to view existing dimensions.

**Steps**
1. Click or press the D key.

2. Select the entity (or entities between which) to dimension and the location of the dimension.
   The dialog opens on the placement of the dimension.

3. Enter a value and press Enter to accept the value. You can also use a variable for a dimension, simply enter "=#" and the variable name, as in: =#d.

Note that you can enter negative values for dimensions (length, linear distance, and angles), thereby flipping the direction of the entity.

The image below illustrates a positive dimension, notice the position of the rectangle in relation to the horizontal line:

![Positive Dimension Diagram]

The following image illustrates a negative dimension value in the dialog:

![Negative Dimension Diagram]

When the dialog is accepted, the rectangle flips direction based on the negative dimension value:
You are able to use **expressions and trigonometric functions** in numeric fields in Part Studios.

Delete a dimension by selecting it and pressing the Delete key, or select it and select Delete from the context menu.

To edit a placed dimension, double-click the value to activate the field, then enter the new value. Press Enter to accept the change.

**Linear distance**

1. Click .
2. Click a line to dimension one line, or two lines to dimension the distance between them.
   
   When dimensioning between two lines, note that this will imply parallelism without a visual parallel constraint.
3. Drag to visualize the dimension.
4. Click again to access the numeric value field.
5. Type value and press Enter.

**Diagonal distance**

1. Click .

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2. Click corner points diagonal to each other.
3. Drag to visualize the dimension.
4. Click again to access the numeric value field.
5. Type value and press Enter.

Length or height

1. Click .
2. Click the edge of the circle.
3. Drag cursor into or away from the circle.
4. Click to activate numeric value field.
5. Type value and press Enter.

Diameter
Angle

1. Click .
2. Click each line.
3. Move cursor into angle.
4. Click to activate numeric value field.
5. Type value and press Enter.

You are also able to drag the label to the quadrant for which you want to define the angle:
Direct distance

1. Click .
2. Click each endpoint of the lines.
3. Move cursor away at an angle to get shortest distance between the points.

Linear distance

1. Click .
2. Click each endpoint of the lines.
3. Move cursor straight up for linear distance.
Radius

1. Click \( \text{\textbullet} \) .
2. Click the edge of the arc.
3. Move cursor into or away from arc.
4. Click to enter numeric value field.
5. Type value and press Enter.

Arc length / curve length

1. Click \( \text{\textbullet} \) .
2. Click each arc endpoint.
3. Move cursor to arc line.
4. When arc line is highlighted, click to activate numeric value field.
5. Type value and press Enter.
Between sketch geometry and plane

Centerline dimensions

Create a centerline dimension between a circle, point, or non-construction line to a construction line; for instance, to dimension a part for a revolve operation. Start a distance dimension between one of these sketch entities and then move the mouse to the opposite side of the construction line. Moving the mouse across the construction line toggles the state between distance and centerline dimensions:

1. Start the dimension between the entity and the construction line, resulting in a distance dimension:
2. Move the mouse to the opposite side of the construction line to toggle the state to a centerline dimension.

3. Enter the value and press Enter.

Driven dimensions

Driven dimensions are useful for maintaining design intent, such as keeping a clearance or wall thickness above a certain value.

- Dimensions are driving by default. Right-click on a dimension value to select "Driving/Driven" from the context menu.
- Driving dimensions appear black and can be edited.
- Driven dimensions appear light gray and cannot be edited (Toggle it to 'driving' and then edit, if necessary.)
• When a dimension added to a sketch over-defines the sketch, the dimension is automatically made 'driven'.

• You are able to add driven dimensions anywhere a driving dimension can be added.

• Driven dimensions reflect the value of the implied dimension; it does not change geometry.

• When a dimension is switched from driven to driving, it changes the geometry; if changing a driven dimension to driving causes the sketch to be over-constrained, red indicators appear as usual.

**Dimension: iOS**
Add horizontal, vertical, shortest distance, angular, diametrical, arc length, or radial dimensions to sketch geometry and between sketch geometry and planes. You are able to specify dimensions as driven (reference) or driving.

**Steps**

1. Tap Dimension tool.

2. Select the entity (or entities) between which to dimension, and the location of the dimension.

   The dialog opens on the placement of the dimension.

3. Enter a value and select the checkmark to accept the value.

**Diagonal distance**

1. Tap to select a corner point.

2. Tap to select the opposite corner point.

3. The current value appears, tap on it to open the number pad.

4. Use number pad to enter a dimension.

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Length or height

1. Tap to select a vertical or horizontal line.
2. The current value of the length or height appears, tap on it to open the number pad.
3. Use the number pad to enter a dimension.

Diameter

1. Tap to select the edge of circle.
2. The current value appears, tap on it to open the number pad.
3. Use the number pad to enter a dimension.
Angle

1. Tap to select a line.
2. Tap to select another line.
3. The current value of the angle between the two lines appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Linear distance / direct distance

1. Tap to select a point.
2. Tap to select another point.
3. The current value appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Radius

1. Tap the edge of an arc to select it.
2. The current value of the radius appears, tap on it to open the number pad.
3. Use the number pad to enter a dimension.
Arc length / curve length

Arc length

1. Tap to select both points of an arc.
2. Tap on the curve of the arc to select it.
3. The arc length value appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Spline curve length

1. Tap a spline to select it.
2. The curve length value appears, tap on it to open the number pad.
3. Use the number pad to enter a dimension.
Between sketch geometry and plane

1. Tap to select a point.
2. Tap to select a plane.
3. The current value appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Centerline dimensions

Create a centerline dimension between a circle, point, or non-construction line to a construction line; for instance, to dimension a part for a revolve operation.

1. Tap to select the point to dimension.
2. Tap to select the construction line to create as the centerline.
   
   A dimension is set between the point and the construction line.
3. Touch and drag the dimension across the construction line.
   
   Once the dimension crosses the construction line, the construction line becomes the centerline and the dimension adjusts accordingly.
Driven dimensions

- Driven dimensions are useful for maintaining design intent, such as keeping a clearance or wall thickness above a certain value.

- Dimensions are driving by default. Tap to select a dimension, then two-finger tap to bring up the context menu. In the context menu you can tap to Toggle Driven/Driving dimensions.

- Driving dimensions appear black and can be edited.

- Driven dimensions appear light gray and are not able to be edited (toggle a driven dimension to driving and then edit, if necessary).
• When a dimension added to a sketch over-defines the sketch, the dimension is automatically made 'driven'.

• You are able to add driven dimensions anywhere a driving dimension can be added.

• Driven dimensions reflect the value of the implied dimensions; it does not change geometry.

• When a dimension is switched from driven to driving, it changes the geometry; if changing a driven dimension to driving causes the sketch to be over-constrained, red indicators appear as usual.

**Negative dimensions**

Onshape allows users to enter negative values for some dimensions but not all. See below for which dimensions accept negative values.

iOS allows negative values for:

• Distance dimension
• Angle dimension
• Line length
iOS does not allow negative values for:

- Arc dimension
- Spline dimension
- Radius or Diameter dimension

**Dimension: Android**

Add horizontal, vertical, shortest distance, angular, diametrical, arc length, or radial dimensions to sketch geometry and between sketch geometry and planes. You are able to specify dimensions as driven (reference) or driving.

**Steps**

1. Tap Dimension tool.
2. Select the entity (or entities) between which to dimension, and the location of the dimension.
   
   The dialog opens on the placement of the dimension.
3. Enter a value and select the checkmark to accept the value.

**Diagonal distance**

1. Tap to select a corner point.
2. Tap to select the opposite corner point.
3. The current value appears, tap on it to open the number pad.
4. Use number pad to enter a dimension.
Length or height

1. Tap to select a vertical or horizontal line.

2. The current value of the length or height appears, tap on it to open the number pad.

3. Use the number pad to enter a dimension.
Diameter

1. Tap to select the edge of circle.
2. The current value appears, tap on it to open the number pad.
3. Use the number pad to enter a dimension.
Angle

1. Tap to select a line.
2. Tap to select another line.
3. The current value of the angle between the two lines appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Linear distance / direct distance

1. Tap to select a point.
2. Tap to select another point.
3. The current value appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Radius

1. Tap the edge of an arc to select it.

2. The current value of the radius appears, tap on it to open the number pad.

3. Use the number pad to enter a dimension.
Arc length / curve length

Arc length

1. Tap to select both points of an arc.
2. Tap on the curve of the arc to select it.
3. The arc length value appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Spline curve length

1. Tap a spline to select it.

2. The curve length value appears, tap on it to open the number pad.

3. Use the number pad to enter a dimension.
Between sketch geometry and plane

1. Tap to select a point.
2. Tap to select a plane.
3. The current value appears, tap on it to open the number pad.
4. Use the number pad to enter a dimension.
Centerline dimensions

Create a centerline dimension between a circle, point, or non-construction line to a construction line; for instance, to dimension a part for a revolve operation.

1. Tap to select the point to dimension.

2. Tap to select the construction line to create as the centerline.

   A dimension is set between the point and the construction line.

3. Touch and drag the dimension across the construction line.

   Once the dimension crosses the construction line, the construction line becomes the centerline and the dimension adjusts accordingly.
Driven dimensions

- Driven dimensions are useful for maintaining design intent, such as keeping a clearance or wall thickness above a certain value.

- Dimensions are driving by default. Double tap a dimension to open the Number pad, then tap the Driven/Driving button to switch the dimension type. The button says what type of dimension you have selected. Tap the button to switch to the opposite type of dimension.

- Driving dimensions appear black and can be edited.

- Driven dimensions appear light gray and are able to be edited (toggle a driven dimension to driving and then edit, if necessary).
When a dimension added to a sketch over-defines the sketch, the dimension is automatically made 'driven'.

- You are able to add driven dimensions anywhere a driving dimension can be added.

- Driven dimensions reflect the value of the implied dimensions; it does not change geometry.

- When a dimension is switched from driven to driving, it changes the geometry; if changing a driven dimension to driving causes the sketch to be over-constrained, red indicators appear as usual.

**Negative dimensions**

Onshape allows users to enter negative values for some dimensions but not all. See below for which dimensions accept negative values.

Android allows negative values for:

- Distance dimension

- Angle dimension
● Line length
● Spline dimension

Android does not allow negative values for:
● Arc dimension
● Radius or Diameter dimension

---

**Coincident**

Make two or more entities coincident, including a sketch entity and a plane.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Coincident: Desktop**

Shortcut: i

In the Sketch toolbar:

Makes two or more entities coincident, including a sketch entity and a plane.

Constraints can be toggled on while you make selections. Toggle Coincident on and each pair of entities you select are constrained to each other. Click Coincident again to toggle off, or select another tool to toggle off automatically.
Steps

1. Select the entities.
2. Click 🖹️.
The infinite, underlying geometry of the two entities is made coincident.

**Coincident: iOS and Android**

Makes two or more entities coincident.
1. Select the entities to apply the coincident constraint to. (You are able to select two or more lines, curves, points, or a sketch entity and a plane).

2. Select 🔧.

The infinite, underlying geometry of the two entities is made coincident.
Concentric

Make any point coincident with the center of an arc or circle. Also make arcs and circles share a center point.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Concentric: Desktop

In the Sketch toolbar:
Steps

1. Select the circle.
2. Select the arc.
3. Select ☰.

Constraints can be toggled on while you make selections. Toggle Concentric on and each pair of entities you select are constrained to each other. Click Concentric again to toggle off, or select another tool to toggle off automatically.

**Concentric: iOS and Android**

Make circles and arcs share a common center point.

1. Select the arcs or circles to share the same center.
2. Select the Concentric constraint tool.
Parallel

Make two or more lines parallel.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Parallel: Desktop**

Shortcut: b

In the Sketch toolbar:

Make two or more lines parallel.

Steps

1. Select each line.

2. Click .
Tips

Constraints can be toggled on while you make selections. Toggle Parallel on and each pair of entities you select are constrained to each other. Click Parallel again to toggle off, or select another tool to toggle off automatically.

Parallel: iOS and Android

Make two or more lines parallel.

1. Select a line with which to make the others parallel.
2. Select the other line(s).
3. Select ✂️.
Tangent

Form a tangent relation between two curves, or between a curve and a plane.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Tangent: Desktop**

Shortcut: t

In the Sketch toolbar:

Form a tangent relation between two curves, or between a curve and a plane.

Constraints can be toggled on while you make selections. Toggle Tangent on and each pair of entities you select are constrained to each other. Click Tangent again to toggle off, or select another tool to toggle off automatically.

Steps

1. In the graphics area, select two or more curves.

2. Click or press the T key.

Examples

**Select two or more curves:**

**Click the Tangent constraint tool:**
Tangent: iOS and Android

Form a tangent relation between two or more curves.

1. Select two or more curves OR select curve(s) and a plane.

2. Tap ⬇️.
Horizontal

Make one or more lines, or sets of points, align horizontally.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Horizontal: Desktop**

Shortcut: h

In the Sketch toolbar:

Make one or more lines, or sets of points, align horizontally.

**Steps**

Constraints can be toggled on while you make selections. Toggle Horizontal on and each pair of entities you select are constrained to each other. Click Horizontal again to toggle off, or select another tool to toggle off automatically.

1. Select one or more lines or points.

2. Click ——.
Horizontal: iOS and Android

Make one or more lines, or sets of points, horizontal.

1. Select one or more lines, OR two or more points.

2. Select —.
Vertical

Make one or more lines, or sets of points, align vertically.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Vertical: Desktop**

Shortcut: v

In the Sketch toolbar:

Make one or more lines, or sets of points, align vertically.

Constraints can be toggled on while you make selections. Toggle Vertical on and each pair of entities you select are constrained to each other. Click Vertical again to toggle off, or select another tool to toggle off automatically.

Steps

1. Select one or more lines or points.

2. Click .
Vertical: iOS and Android

Make one or more lines, or sets of points vertical.

1. Select one or more lines, or sets of points.

2. Tap +.
Perpendicular

Form a right angle between two lines.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Perpendicular: Desktop

In the Sketch toolbar:

Steps

1. Select two lines.

2. Click \( \square \).

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Constraints can be toggled on while you make selections. Toggle Perpendicular on and each pair of entities you select are constrained to each other. Click Perpendicular again to toggle off, or select another tool to toggle off automatically.

**Perpendicular: iOS and Android**

Form a right angle between two lines.

1. Select two or more lines.
2. Tap \( \perp \).

**Sketch 1**
Make two or more sketch curves of the same type equal in size.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Equal: Desktop**

Shortcut: e

In the Sketch toolbar:
Constraints can be toggled on while you make selections. Toggle Equal on and each pair of entities you select are constrained to each other. Click Equal again to toggle off, or select another tool to toggle off automatically.

Steps

1. Select two or more sketch curves.

2. Click $\mathbf{=}$.
If one sketch curve is dimensioned, that size is used.

**Equal: iOS and Android**

Make two or more sketch curves of the same type equal in size.

1. Select two or more curves.
2. Select $\equiv$.

If one of the selected curves is dimensioned, that size is used.
Midpoint

Constrain a point to the midpoint of a line or arc.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Midpoint: Desktop**

In the Sketch toolbar:

Constraints can be toggled on while you make selections. Toggle Midpoint on and each pair of entities you select are constrained to each other. Click Midpoint again to toggle off, or select another tool to toggle off automatically.

Steps

1. Select a line and a point.

2. Click 🔄️.
Or select an arc and a point, then click .

The point will be constrained to the midpoint of the line or arc.

**Midpoint: iOS and Android**
Constrain a point to the midpoint of a line or arc.

1. Select a point, and a line or an arc.

2. Select \( \bullet \).

The point will be constrained to the midpoint of the line or arc.

**Sketch 1**

![Sketch Image]
Sketch 1
Make a line and curve, or a curve and a plane normal to each other.

This functionality is available on Onshape’s browser, iOS, and Android platforms.

**Normal: Desktop**

In the Sketch toolbar:
Steps

1. Select a line and a curve, or a curve and a plane.

2. Click 🎨.

Constraints can be toggled on while you make selections. Toggle Normal on and each pair of entities you select are constrained to each other. Click Normal again to toggle off, or select another tool to toggle off automatically.

**Normal: iOS and Android**

Make a line and a curve, or a curve and a plane normal to each other.

1. Select a line or a plane, and a curve.

2. Tap 🎨.

The selected line or plane and the selected curve are now normal to each other.
Pierce

Constrain a sketch entity (point or curve) to be coincident with the intersection point of its sketch plane and an arbitrary curve that is not in its sketch plane. The sketch entity is now constrained to be coincident with the point of intersection.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Pierce: Desktop**

In the Sketch toolbar:

Steps

1. Select a sketch point or curve and an edge outside of the sketch (intersecting with the sketch plane).

2. Click .

Constraints can be toggled on while you make selections. Toggle Pierce on and each pair of entities you select are constrained to each other. Click Pierce again to toggle off, or select another tool to toggle off automatically.

The key is that the edge has to pass through the plane.
Pierce: iOS and Android

Constrain a sketch entity (point or curve) to be coincident with the intersection point of its sketch plane and an arbitrary curve that is not in its sketch plane.

In this example, a sketch point of the rectangular sketch on the Top plane is constrained with the intersection of the curve of the circle on the Right sketch plane.

1. Select a sketch point or curve and an edge outside of the sketch (intersecting with the sketch plane).
2. Select ⌘.

The key is that the edge has to pass through the plane.

The sketch entity is now constrained to be coincident with the point of intersection.
**Symmetric**

Constrain two geometries (of the same type) to be symmetric relative to a line.

This functionality is available on Onshape's browser, iOS, and Android platforms.
**Symmetric: Desktop**

In the Sketch toolbar:

Steps

1. Pre-select a line, or linear edge.
2. Select two other geometries (of similar type to each other).
3. Click Symmetric.

Tips

Constraints can be toggled on while you make selections. Toggle Symmetric on and each pair of entities you select are constrained to each other. Click Symmetric again to toggle off, or select another tool to toggle off automatically.

Symmetric only constrains the underlying curve to be symmetric. For example, when applying the Symmetric constraint to two arcs, the underlying circles are made symmetric but **not** the end points (as shown above). You would need to add those separately and/or drag them closer to what is needed:
Symmetric: iOS and Android

Constrain two geometries (of the same type) to be symmetric relative to a line:

1. Pre-select a line, or linear edge.
2. Select two other geometries (of similar type to each other).
3. Tap $\Sigma$.

Sketch:
After Symmetric constraint is applied:

[Diagram of Sketch 1 with symmetric constraint applied]
Symmetric only constrains the underlying curve to be symmetric. For example, when applying Symmetric constraint to two arcs, the underlying circles are made symmetric but not the end points (as shown above). You would need to add those separately and/or drag them closer to what is needed.

---

**Fix**

Ground a sketch entity on the sketch plane so that it does not move.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Fix: Desktop**

In the Sketch toolbar:
Constraints can be toggled on while you make selections. Toggle Fix on and each entity that is selected is constrained to the sketch plane so that it will not move. Click Fix again to toggle off, or select another tool to toggle Fix off automatically.

Steps
1. Select a sketch entity.
2. Click ▶️.

**Fix: iOS and Android**

Ground a sketch entity on the sketch plane so that it does not move.
1. Select a sketch entity.
2. Tap ▶️.

The selected entity is now fixed to the sketch plane.

---

**Curvature**

Create curvature continuous transitions between sketch splines (and conics) and surrounding geometry.

This functionality is available on Onshape's browser, iOS, and Android platforms.

In the Sketch toolbar:

Steps
To ensure that the splines or conics are curvature continuous:
1. Select the geometry involved.

2. Click.

Automatic Inferencing

This functionality is available on Onshape's browser, iOS, and Android platforms.

The Onshape sketch editor has the ability to assign constraints to certain entities automatically. For example, create a line and hover one of the endpoints above the origin and a dotted line appears indicating a vertical inference between that endpoint and the origin.
When sketching, Onshape displays inferences for Horizontal and Vertical alignment between an entity and the origin and/or another entity. In some cases, inference only occurs when the cursor is moved near another entity to 'wake up' the inferencing between the two entities. Some commonly used wake up inferences are: horizontal, vertical, midpoint, parallel, and coincident.

Steps

1. Create two lines with a perpendicular constraint between them.
2. Move cursor near line until inferencing ‘wakes up’.
3. Draw a line.

When sketching, Onshape indicates relationships with other sketch entities. In the illustration below, the bottom (blue) line is the one being drawn. When it is parallel to the other line it turns to a dotted line and the other is highlighted in orange to show that there is a relation present. (The parallel constraint icon is also visible in this example.)

To suppress automatic inferences, press the Shift key when mousing.

Working with Constraints

Constraints are available and viewable when a sketch is being created or otherwise open for editing.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Working with Constraints: Desktop**

Constraints applied between entities in two sketches (for instance, when you Use an entity from one sketch in another sketch) are differentiated by a blue background. Constraints may be applied manually and some are created when geometry is created as you sketch. Upon hover, the referenced constraint’s background is a darker blue:

![Constraint Example](image)

The Use constraint shown above (with the blue background) constrains a vertex in the rectangle’s sketch with the center point of the circle in the circle’s sketch.

With a sketch open, hover over a sketch entity, like a line or arc, to see the constraints for that entity. As you move the mouse to hover over entities, constraints will appear only for the highlighted entity. To keep all constraints visible, use the Shift key as you move the mouse.

Entities are highlighted in orange upon hover, with the exception of referenced constraints which have a blue background and a darker blue background upon hover. Related entities are highlighted with yellow, as when you select a constraint and the coordinating entities is also highlighted.
Constraints created automatically

These constraints are not available in the Constraint section of the toolbar, but are created automatically during specific actions as described below:

- **Quadrant** - Constrains a point to be coincident to an ellipse and either the major or minor axis of that ellipse. Can be made by inference, dragging something to, or placing something on one of the points on an ellipse.

- **Use** - Constrains a sketch entity in one sketch to an entity in another sketch; made by selecting the Use tool and then an entity (sketch entity, face, or edge) in a different sketch or feature.

- **Intersection** - Constrains the end points of an open curve (resulting from using the Intersection tool) with Pierce constraints so that they lie on the edges of the intersected face; for a closed curve, constrains the sketch entities with Intersection constraints.

**Tips**

You have the ability to interact with constraint icons:

- Click and drag the icon or group of icons to a different location.

- Hover over a single constraint icon to see which entities are highlighted, indicating the constraint applies to them.

- Delete a constraint: click a single constraint icon and press Delete or select Delete from the context menu.
• In the Sketch dialog, check Show constraints to display all constraints defined for the sketch.

• Conflicting constraints are shown as white symbols on a red background.

When sketching, constraint indicators appear next to the mouse cursor as the curves snap to inferences.

Constraints: iOS and Android

Add, apply, and edit constraints to help define a part in Onshape. Constraints are listed in the sketch dialog when a sketch is being created or otherwise open for editing.

Constraints may be applied between entities in a sketch or between entities in two or more sketches (for instance, when you Use an entity from one sketch in another sketch). Constrains are able to be applied manually and some are created when geometry is created as you sketch. See Automatic Inferencing for more information on automatically created constraints.

In the sketch dialog, tap on a constraint in the list to highlight the relevant sketch geometry. For example, if two lines are constrained horizontally to each other and you tap the horizontal constraint, both lines are highlighted.
Add, apply, and edit constraints to help define a part in Onshape. Constraints are only available and viewable when a sketch is being created or otherwise open for editing. For information on using specific constraints while on a mobile device, see the specific constraint topic.

Troubleshooting Sketch Geometry

This functionality is also available on Onshape's iOS and Android platforms.

The color of sketch entities indicate its constrained status:

- Blue means under-constrained
- Black means fully constrained
- Red means a constraint problem (over-constrained)

The color of a constraint icon indicates its constrained status: black on gray is well-defined, white on red indicates a problem.

Adding more dimensions or constraints will further constrain the sketch. Dragging entities may help you understand what constraints or dimensions you may want to add.
Feature Basics

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape has somewhat unique feature tools you use to create parts. Many of our tools are 'combination' tools and serve many purposes like creating new material, adding additional material, removing material, and also creating intersecting material. For example, the Extrude tool is used for all of the above.

This topic walks you through the basics of feature tool dialogs: how to use them, what the color paradigm means, and the other unique characteristics that make Onshape features easy to use.

Feature Basics: Desktop

When creating Features in Onshape, you use this Feature tools toolbar:
In Onshape, features are applied to 2-D sketches in a Part Studio to create 3-D parts. All Feature tools are in the Feature tools toolbar as icons or within drop down menus. Two Feature tools, Extrude and Revolve, are available on the Sketch tools toolbar for efficiency and may be used when a sketch is open. Each Feature created is stored parametrically, that is, visible in the Feature list as its own entity.

Access the Feature shortcut toolbar with the S key while in the graphics area (with no open dialog):

Customize this shortcut toolbar through your Onshape account Preferences page.

The Feature tools toolbar is always accessible at the top of the workspace in a Part Studio. If you are editing a sketch, you will see the Sketch toolbar, close the sketch to see the Feature toolbar. You can also customize the main Feature toolbar at the top of the page, for more information see "Part Studios" on page 279.

**Basic workflow**

You can create as many features (and therefore parts) as you want, in a Part Studio.

1. In a Part Studio, with an existing sketch or part, click to select the Feature tool you want. That tool's dialog opens, for example:
When a Feature dialog opens, two fields are active: the first (blue) field requiring selection in the graphics area with a mouse click, and the first (white) text field requiring keyboard input. Use the Tab key to move focus from one field to the next, sequentially, starting with the first active text field. You can also use the arrow keys to expand a dropdown menu, and the Enter key to accept a selection in the menu. Use the Enter key again to accept the input and close the dialog.

2. Using the dialog, and by selecting entities in the workspace (sketches, part faces, or surfaces) fill out the required parameters.

3. When you are done filling out the parameters, visualize changes using the Preview slider.

4. Accept the feature by clicking ✓.

You can cancel a feature at any time by clicking ✗ or by pressing the Escape key.

Each Feature tool requires different information to complete the feature. For more specific information on what is required for each tool, see Feature tools.
In addition to using a mouse to navigate through the dialog, you can:

- Use the Tab key to move from one field to another.
- Use arrow keys to navigate a dropdown menu (Blind, above) and the Enter key to select from the menu.
- Use the spacebar to check/uncheck options (Draft and Second end position, above).
- Click the ✓ to commit the feature (or press Enter).
- Click ✗ to cancel the feature and close the dialog.

**Title** - The title is red if you have not completely filled out the dialog, or if the information entered has resulted in an error. This prevents you from committing a broken feature.

If you have specified all of the information, correctly, needed to complete the feature, then the title is black. This indicates that you may commit the feature successfully.
• **Blue text and underline** - Blue text with a blue underline indicates a selected item. Click to select an item from a horizontal list.

```
<table>
<thead>
<tr>
<th>Solid</th>
<th>Surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Add</td>
</tr>
<tr>
<td></td>
<td>Remove</td>
</tr>
<tr>
<td></td>
<td>Intersect</td>
</tr>
</tbody>
</table>
```

• **Blue highlight** - A highlighted field indicates that a selection from the graphics area is required. Click in the field, then click in the graphics area to make one or more selections. Click on the x in the right of the field, to quickly remove a selection.

```
Face of Sketch 1
Blind
```

• **Drop down menus** - Click to open a drop down menu, then click to select an option.

```
Face of SKETCH 1
Symmetric
Blind
Symmetric
Up to next
Up to face
Up to part
Up to vertex
Through all
```

• **Opposite direction** - When applicable, click the icon to toggle the direction of the feature.

```
Blind
```

• **Input field** - Click to input a value. You have the ability to specify a unit of measurement by adding it to the value, or you are able to set default units for all of your documents.

```
Blind
Depth 5 ft
```

• **Check boxes** - Check boxes indicate an optional specification that can be applied to the feature. Click to check or uncheck a box to use optional specifications.
In the image above, a Draft is being applied to the extrude feature but a second end position is not.

- **Preview slider bar** - The Preview slider is an opacity control that lets you adjust the display opacity of the feature along a scale of 0% (before the feature is applied) to 100% (after the feature is applied).

When you edit a feature, by default Onshape displays the model rolled back to its state when that feature was created, hiding all later features. The Final button displays the final result while you are still editing the feature. If you are editing the last feature, there is no Final button in the dialog, since you are already seeing the final result.

If you have permission to copy a document, but not edit the document, you are still able to open the document, view it, and also open Feature dialogs within the document. Without Edit permission, however, you will not be able to make any changes to the document, unless you first make a copy, and all dialog fields are grayed out and inactive:
Contextual Help - Click on the 🔎 in the lower right of any dialog to open the Help documentation to the related topic.

Reordering parameters

Some feature dialogs accept the selection of multiple entities (called array parameters) whose selection order is critical in order to create certain geometry. For example, when selecting profiles to use in a Loft feature, the order in which you select the sketches defines the direction of the loft. Onshape allows the reordering of these entities directly in the dialog.

For example, if you have selected three profiles for a loft and then want to add a fourth, but the fourth profile invalidates the geometry. The first time you create the loft, the dialog has three profiles and the model is valid:
When you want to add the fourth profile, and you just select it, it is added to the bottom of the list of profiles. If it doesn't occur in an order that makes sense, an error occurs. However, you can click the opposing arrows (circled in blue above) at the top of the Profiles field to activate the reorder mechanism (circled in blue below):

Click and drag the reorder icon to reorder the selected profile to the correct position in the dialog:
Drop to place the profile in the desired position:

Commenting on a Feature

Place comments on a particular feature for later reference or for other collaborators. You are also able to indicate that you want to receive email notifications of other users' comments on the feature.

1. Right-click on the feature in the Feature list and select Add comment.
2. Type a comment, optionally indicate that you wish to receive email notifications of others' comments.
3. Click Add to add the comment or Cancel to close the Comments flyout without adding a comment.

If another user has share permissions on the document and has selected Receive comment email notifications, an email is sent to that email address with the text of your comment in it.

Click on the comments icon at the top to open the comments flyout:

When the comments flyout is open, any Features that have been commented on, will have an icon next to them in the Features list.

**Feature Basics: iOS**

When creating Features in Onshape, you use the Feature tools in the Feature toolbar.
Feature tools create, modify, or manipulate 2D and 3D geometry to create new parts, modify existing ones, or generate construction tools for later use. Features are stored parametrically, visible in the Feature list as its own entity.

The Feature toolbar is always accessible at the top of the graphics area in a Part Studio. If you are editing a sketch, there are two Feature tools available in the Feature toolbar, for efficiency; Extrude and Revolve. Close the sketch to see and use the other Feature tools.

**Basic workflow**

You have the ability to create as many features (and therefore parts) as you want, in a Part Studio.

1. In a Part Studio, with an existing sketch or part, tap to select the Feature tool you want. That tool's dialog opens:
2. Using the dialog, and by selecting entities in the workspace (sketches, part faces, or surfaces) fill out the required parameters.

3. When you are done filling out the parameters, visualize changes using the Preview slider.

4. Accept the feature by tapping the checkmark.

You have the ability to cancel a feature at any time by tapping the x.

Each Feature tool requires different information to complete the feature. For more specific information on what is required for each tool, see Feature tools.

For more information on how to fill out the required parameters in a dialog, see Dialogs.
Feature Basics: Android

When creating Features in Onshape, you use the Feature tools in the Feature toolbar.

Feature tools create, modify, or manipulate 2D and 3D geometry to create new parts, modify existing ones, or generate construction tools for later use. Features are stored parametrically, visible in the Feature list as its own entity.

The Feature toolbar is always accessible at the top of the graphics area in a Part Studio. If you are editing a sketch, there are two Feature tools available in the Feature toolbar, for efficiency; Extrude and Revolve. Close the sketch to see and use the other Feature tools.

You have the ability to create as many features (and therefore parts) as you want, in a Part Studio.

1. In a Part Studio, with an existing sketch or part, tap to select the Feature tool you want. That tool's dialog opens:
2. Using the dialog, and by selecting entities in the workspace (sketches, part faces, or surfaces) fill out the required parameters.

3. When you are done filling out the parameters, visualize changes using the Preview slider.

4. Accept the feature by tapping the checkmark.

You have the ability to cancel a feature at any time by tapping the x.

Each Feature tool requires different information to complete the feature. For more specific information on what is required for each tool, see Feature tools.

For more information on how to fill out the required parameters in a dialog, see Dialogs.
Feature Tools

Feature tools create, modify, or manipulate 3-dimensional geometry to create new parts, modify existing ones, or generate construction tools for later use.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Feature Tools: Desktop

The Feature toolbar

Access the Feature shortcut toolbar with the S key while in the graphics area (with no open dialog):

Customize the Feature shortcut toolbar through your Onshape account Preferences page.

To learn more about customizing the toolbars of Part Studios, Assemblies, or Feature Studios, see "Part Studios" on page 279.

Steps

1. Generate the requisite base geometry for your intended Feature tool (see information on individual Feature tools for relevant requirements).
2. Select your Feature tool of choice.
3. Select geometry as required.
4. Input parameters.
5. Select direction and any additional options.
6. Visualize changes using the Preview slider.

7. Click ✔️ to generate the feature or ✗ to cancel.

**Search tools**

Shortcut: alt + C

In Onshape, you have the ability to search for tools in the Feature toolbar as well as the Assembly toolbar and Sketch toolbar: see Toolbars and Document Menu, Assemblies, and Sketch tools for more information on those specific topics.

**Tips**

- The Escape key exits a tool selection.
- Use the Preview slider to check the potential result to make sure it's what you intend. Slide right to see more, slide it to the left to see less.
- Use the Final button to view your model from the perspective of the bottom of the Feature List, after all calculations are made. This may help you see the final result of editing you may be doing towards the top of the Feature List and how it affects the final outcome.
- Use the Undo/Redo buttons while you are editing to revert an action or reinstate an action made while the sketch or feature is open.
- Use the Undo/Redo buttons after closing a sketch or edit dialog to revert an editing session, or subsequently reinstate the changes made during that editing session.

**Feature Tools: iOS**

Feature tools create, modify, or manipulate 2D and 3D geometry to create new parts, modify existing ones, or generate construction tools for later use.

At the top of your opened document is the main toolbar:
The main toolbar has Undo/Redo buttons to the left, Sketch and Feature tool icons to the right of the Undo/Redo buttons, and the measure tool to the right.

Tap the Feature tools icon to view all of the Feature tools:
Tap to select a Feature tool.

**Get started**

1. Generate the requisite base geometry for your intended Feature tool (see information on individual Feature tools for what each tool requires).

2. Select a Feature tool.

3. Select required geometry.

4. Input parameters.

5. Select direction and any additional options.

6. Visualize your changes using the preview slider.

7. Select the check to generate the feature or select the x to cancel.
Tips

- Use the **Preview slider** to check the potential result of the feature to make sure it's what you intend. Slide right to see more, slide left to see less.

- Use the **Final button** to view your model from the perspective of the bottom of the Feature list, after all calculations are made. This may help you to see the final result of editing you may be doing towards the top of the Feature list and how it affects the final outcome.

- Use the **Undo|Redo** buttons **while you are editing** to revert an action or reinstate an action made while the sketch or feature is open.

- Use the **Undo|Redo** buttons **after closing a sketch or edit dialog** to revert an editing session, or subsequently reinstate the changes made during that editing session.

**Example**

1. With a part in your Part Studio, tap the Shell tool.

2. Shell requires that you select one or more part faces to remove in order to hollow out the part.

   You have the ability to either select the tool and then the face to shell, or the face to shell and then the tool.
If you select the tool first then as soon as you select the face to remove, it is removed. If you select the face to remove first then as soon as you select the tool, it is removed.

3. Specify the wall thickness (the system supplies a default). You are able to select the unit of your choice as well.

4. Use the *Opposite direction* toggle to specify whether the wall thickness should be applied to the inside of the part face or the outside.

5. Move the Preview slider to the left to visualize the part before the feature is
applied.
6. Move the Preview slider to the right to visualize the part after the feature is applied.

7. Tap the check to accept or tap the x to close without committing any changes.

**Feature Tools: Android**

Feature tools create, modify, or manipulate 2D and 3D geometry to create new parts, modify existing ones, or generate construction tools for later use.

At the top of your opened document is the main toolbar:
The main toolbar has Undo/Redo buttons to the left, Sketch and Feature tool icons to the right of the Undo/Redo buttons, and the measure tool to the right.

Tap the Feature tools icon to view all of the Feature tools:

Tap to select a Feature tool.
Get started

1. Generate the requisite base geometry for your intended Feature tool (see information on individual Feature tools for what each tool requires).

2. Select a Feature tool.

3. Select required geometry.

4. Input parameters.

5. Select direction and any additional options.

6. Visualize your changes using the preview slider.

7. Select the check to generate the feature or select the x to cancel.

Tips

- Use the **Preview slider** to check the potential result of the feature to make sure it's what you intend. Slide right to see more, slide left to see less.

- Use the **Final button** to view your model from the perspective of the bottom of the Feature list, after all calculations are made. This may help you to see the final result of editing you may be doing towards the top of the Feature list and how it affects the final outcome.

- Use the **Undo|Redo** buttons while you are editing to revert an action or reinstate an action made while the sketch or feature is open.

- Use the **Undo|Redo** buttons after closing a sketch or edit dialog to revert an editing session, or subsequently reinstate the changes made during that editing session.

Example

1. With a part in your Part Studio, tap the Shell tool.
2. Shell requires that you select one or more part faces to remove in order to hollow out the part.

You have the ability to either select the tool and then the face to shell, or the face to shell and then the tool.

If you select the tool first then as soon as you select the face to remove, it is removed. If you select the face to remove first then as soon as you select the tool, it is removed.

3. Specify the wall thickness (the system supplies a default). You are able to select the unit of your choice as well.

4. Use the *Opposite direction* toggle to specify whether the wall thickness should be applied to the inside of the part face or the outside.

5. Move the Preview slider to the left to visualize the part before the feature is applied.
6. Move the Preview slider to the right to visualize the part after the feature is applied.
7. Tap the check to accept or tap the x to close without committing any changes.

**Extrude**

Add depth to a selected region or planar face along a straight path. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Use Extrude to create parts or surfaces.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Extrude: Desktop**

Shortcut: Shift-e

In the Feature toolbar:

In the Sketch toolbar:

Add depth to a selected region or planar face along a straight path. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Use Extrude to create parts or surfaces.

**Steps for creating solids**

From the Sketch or Feature toolbar:
1. Click

![Extrude 4 window]

2. **Select Solid** Creation type.

   When Extrude Solid is selected at the time a sketch is open, Onshape automatically selects all the closed regions in the sketch, and if present, nested entities:

   ![Extruded solids]

3. **Select a Result operation type:**

   - **New** - Create new material that results in a new part.
   - **Add** - Create material added to the existing material.
- **Remove** - Take material away from a part.
- **Intersect** - Leave material only where intersections exist.

4. Select Faces and sketch regions to extrude.

5. Select End type:
   - **Blind** - To a specified distance (as entered in the Depth field).
   - **Symmetric** - To a specified total distance, half the distance in both directions about the sketch plane.
   - **Up to next** - Up to the next face or faces encountered in the specified direction. If it doesn’t completely terminate, then the Extrude fails.
   - **Up to face** - Up to the infinite face underlying the selected face or plane.
   - **Up to part** - Up to the next part encountered in the specified direction.
   - **Up to vertex** - Up to a selected point (vertex) or mate connector (inferred or existing). Click the in order to select inferred Mate connectors.

Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:
Up to next, Up to Face, Up to part, and Up to vertex all support extruding with an offset in one or two directions, indicated by the single and double arrows in the example below:

With Up to face, Up to part, and Up to vertex, checking **Offset** to specify a distance for the extrude to fall short of the part, face, vertex, or next entity by the specified distance.

- **Through all** - Through all selected parts.

6. Specify whether to switch to the opposite direction, optional.

7. Check **Draft** to create an automatic draft during the Extrude operation with the sketch plane as the neutral plane, and specify the number of degrees for the draft.

8. Optionally, check to extrude in a **second end position** about the sketch plane.

   Extruding in a second end position offers all the same end conditions and a separate depth field, as well as the option to create a draft (with the sketch plane as the neutral plane).

9. Enter a depth (for each end position, if necessary).

10. If necessary, select a **Merge scope** (or Merge with all) to select parts or surfaces with which to merge the new (additional) part or surface.

11. Click ✓.

Remember, you are able to use the Preview slider to visualize the result before accepting the feature (with the check).

**Steps for creating surfaces**

From the Sketch or Feature toolbar:
1. Click [ ].

![Extrude 4 dialog box]

2. Select **Surface** Creation type:

3. Select a Result operation type:
   - **New** - Create new material that results in a new part or surface.
   - **Add** - Create material added to the existing material.

4. Select sketch curves to extrude.

5. Select End type:
   - **Blind** - To a specified distance (as entered in the Depth field).
   - **Symmetric** - To a specified total distance, half the distance in both directions about the sketch plane.
   - **Up to next** - Up to the next face or faces encountered in the specified direction. If it doesn’t completely terminate, then the Extrude fails.
   - **Up to face** - Up to the infinite face underlying the selected face or plane.
   - **Up to part** - Up to the next part encountered in the specified direction.
   - **Up to vertex** - Up to a selected point (vertex) or Mate connector (inferred or existing). Click the [ ] in order to select inferred Mate connectors.
Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:

Up to next, Up to Face, Up to part, and Up to vertex all support extruding with an offset in one or two directions as shown in the example below where Up to Next is accompanied with an Offset of 1 inch in both directions:

Checking **Offset** and specifying a distance results in the extrude falling short of the part, face, vertex, or next entity by the specified distance.

- **Through all** - Through all selected parts.

6. Specify whether to switch to the opposite direction, optional.
7. Optionally, check to extrude in a second end position about the sketch plane.

   Extruding in a second end position offers all the same end conditions and a separate depth field, as well as the option to create a draft (with the sketch plane as the neutral plane).

8. Enter a depth (for each end position, if necessary).

9. Click ✓.

   Remember, you are able to use the Preview slider to visualize the result before accepting the feature (with the check).

**Steps for extruding on a sheet metal flat pattern**

When extruding on a flat pattern sketch, an abbreviated Extrude dialog box appears with options to add or remove material from the sheet metal. Click the checkmark to accept any changes.

![Extrude dialog box](image)

   Remember, you are able to use the Preview slider to visualize the result before accepting the feature (with the check).

**Extrude: New/Add (new material)**

Create new material or material that results in a new part or surface

**New** - Create new material that results in a new part
**Add** - Create material and add to the existing material

When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you can check Merge with all to add all touching or intersecting parts to the merge scope.

Note that if the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Extrude: Remove (cut material)**

Take material away from a part; not available for surfaces.

**Remove** - Take material away
**Extrude: Intersect**

Leave material only where intersections exist; not available for surfaces.

**Intersect** - Leave material only where intersections exist.

**Extrude: Second Direction**

Extrude in two directions differently about the sketch plane.

Create an offset from the sketch plane by flipping the extrude second direction:
End type examples

Up to next, Up to Face, Up to part, and Up to vertex all support extruding with an offset in one or two directions. Checking **Offset distance** and specifying a distance results in the extrude falling short of the part, face, vertex, or next entity by the specified distance.

Blind

To a specified distance in one direction

Symmetric

To a specified distance equally in both directions around the sketch plane
Up to next

Up to the next geometry encountered in the given direction; if there is no existing geometry encountered, the extrude cut fails; note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed.

Up to face

Up to the infinite face underlying the selected face or plane (select a plane or face of a part); note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed.
Up to part

Up to the next part encountered in the given direction; if there is no part encountered, the extrude will fail. Note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed.

Up to vertex

Up to the selected point (vertex) or mate connector (inferred or existing).
Through all

Through all selected parts
**Merge scope**

Merge scope is available with parts and surfaces and allows you to select the specific part or surface with which to merge the newly created part or surface. By default, Merge all is selected. You can uncheck that box to access the Merge scope field, then select the part or surface with which to merge. Surfaces must be merged with surfaces and parts with parts.

Merge scope: with all

Merge extrusion with all parts it intersects
Merge scope: particular part

Select a specific part with which to merge

**Extrude: iOS**

Add depth to a selected region or planar face along a straight path. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Use Extrude to create parts or surfaces.

**Steps**

1. Tap Extrude tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts.
   - **Surface** - Create a surface along a sketch curve.

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.
4. Select Faces and sketch regions to extrude.

5. Select End type:
   - **Blind** - To a specified distance (entered in the Depth field).
   - **Symmetric** - To an equal distance is both directions symmetrically along the sketch plane.
   - **Up to next** - Up to the next geometry encountered in the given direction; if there is no existing geometry encountered, the extrude cut fails.
   - **Up to face** - Up to the infinite face underlying the selected face or plane.
   - **Up to part** - Up to the next part encountered in the given direction; if there is no part encountered, the extrude fails.
   - **Up to vertex** - Up to the selected point (vertex) or mate connector.
   - **Through all** - Through all selected parts.

6. Specify whether to switch to the opposite direction, optional.

7. Toggle on to create an automatic Draft during the Extrude operation with the sketch plane as the neutral plane, and specify the number of degrees for the draft, optional.

8. Toggle on to extrude in a second end position about the sketch plane, optional.

9. Enter a depth (for each end position, if necessary).

10. Tap checkmark.

**Extrude nested sketches**

1. With a sketch open, select the Extrude tool.

2. Onshape automatically selects all of the regions in the sketch, except when nested sketch entities exist. In this case, Onshape selects only the region between the nested sketch entities.
Extrude: New/Add (new material)

**New** - Create new material that results in a new part

**Add** - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you can toggle **Merge with all** to add all touching or intersecting parts to the merge scope.

Note that if the Boolean is set to Add, Remove, or Intersect, and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Extrude: Remove (cut material)**

**Remove** - take material away

**Extrude: Intersect**

**Intersect** - Leave material only where intersections exist
**Extrude: Surface**

Create a surface along a sketch curve.

**Extrude: Second Direction**

Extrude in two directions differently about the sketch plane.
End type examples

These examples all show the extrude of a simple part or surface, creating new material.

Blind

To a specified distance in one direction
Symmetric

To a specified distance equally in both directions around the sketch plane
Up to the next geometry encountered in the given direction; if there is no existing geometry encountered, the extrude cut fails. Note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed.
Up to face

Up to the infinite face underlying the face or plane (select a plane or face of a part). Note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed.
Up to part

Up to the next part encountered in the given direction; if there is no part encountered, the extrude fails. Note that the sketch region or entity being extruded but fall entirely within the target entity for the extrude to succeed.
Up to vertex

Up to the selected point (vertex) or mate connector
Through all

Through all selected parts:
**Merge scope**

Merge scope: with all

Merge extrusion with all parts it intersects

Merge scope: particular part

Select a specific part with which to merge
**Extrude: Android**

Add depth to a selected region or planar face along a straight path. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Use Extrude to create parts or surfaces.

**Steps**

1. Tap Extrude tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts.
   - **Surface** - Create a surface along a sketch curve.

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select Faces and sketch regions to extrude.

5. Select End type:
   - **Blind** - To a specified distance (entered in the Depth field).
   - **Symmetric** - To an equal distance is both directions symmetrically along the sketch plane.
   - **Up to next** - Up to the next geometry encountered in the given direction; if there is no existing geometry encountered, the extrude cut fails.
   - **Up to face** - Up to the infinite face underlying the selected face or plane.
   - **Up to part** - Up to the next part encountered in the given direction; if there is no part encountered, the extrude fails.
   - **Up to vertex** - Up to the selected point (vertex) or mate connector.
   - **Through all** - Through all selected parts.

6. Specify whether to switch to the opposite direction, optional.

7. Toggle on to create an automatic Draft during the Extrude operation with the sketch plane as the neutral plane, and specify the number of degrees for the draft, optional.

8. Toggle on to extrude in a second end position about the sketch plane, optional.

9. Enter a depth (for each end position, if necessary).

10. Tap checkmark.
Extrude nested sketches

1. With a sketch open, select the Extrude tool.

2. Onshape automatically selects all of the regions in the sketch, except when nested sketch entities exist. In this case, Onshape selects only the region between the nested sketch entities.

Extrude: New/Add (new material)

New - Create new material that results in a new part
Add - Create material and add to the existing material

When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you can toggle Merge with all to add all touching or intersecting parts to the merge scope.

Note that if the Boolean is set to Add, Remove, or Intersect, and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.
**Extrude: Remove (cut material)**

Remove - take material away

**Extrude: Intersect**

Intersect - Leave material only where intersections exist

**Extrude: Surface**

Create a surface along a sketch curve.
Extrude: Second Direction

Extrude in two directions differently about the sketch plane.

End type examples

These examples all show the extrude of a simple part or surface, creating new material.

Blind

To a specified distance in one direction
Symmetric

To a specified distance equally in both directions around the sketch plane
Up to the next geometry encountered in the given direction; if there is no existing geometry encountered, the extrude cut fails. Note that the sketch region or entity being
extruded must fall entirely within the target entity for the extrude to succeed
Up to face

Up to the infinite face underlying the face or plane (select a plane or face of a part). Note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed.
Up to part

Up to the next part encountered in the given direction; if there is no part encountered, the extrude fails. Note that the sketch region or entity being extruded but fall entirely within the target entity for the extrude to succeed.
Up to vertex

Up to the selected point (vertex) or mate connector
Through all

Through all selected parts:
Merge scope

Merge scope: with all

Merge extrusion with all parts it intersects

Merge scope: particular part

Select a specific part with which to merge
Revolve

Create, add to, subtract from, or intersect parts by revolving sketch regions or planar faces about a central axis, or surfaces by revolving lines and curves about a central axis.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Revolve: Desktop**

In the Feature toolbar:

**Steps for creating solids**

From the Sketch or Feature toolbar:

1. Click 🌊.

2. Select **Solid** Creation type:

   When Revolve Solid is selected at the time a sketch is open, Onshape automatically selects all the closed regions in the sketch.
3. Select a line, cylindrical edge, or arc to revolve around an axis, or Mate connector (inferred or existing).

4. Activate the Revolve axis field, then click the axis or click the ⌁ in order to select inferred Mate connectors about which to revolve.

Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:

5. Choose whether you want to:
   - **New** - Create a new solid
   - **Add** - Add to an existing solid
   - **Remove** - Subtract from an existing solid
   - **Intersect** - Keep the intersection of two (or more) solids

6. Select a Revolve type:
   - **Full** - Revolve about the axis 360 degrees
   - **One direction** - Revolve in one direction for a specified number of degrees
   - **Symmetric** - Revolve in both directions for a specified number of degrees

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Two directions - Revolve in both directions for the same or different numbers of degrees

7. If necessary, select a Merge scope (or Merge with all) to select parts or surfaces with which to merge the new (additional) part or surface.

8. Click ✓.

Steps for creating surfaces

From the Sketch or Feature toolbar:

1. Click ✓.

2. Select Surface Creation type:

3. Choose whether you want to:
   - New - Create a new surface
   - Add - Add to an existing surface

4. Select edges or a sketch to revolve.

5. Activate the Revolve axis field, then click the axis or Mate connector (inferred or existing) about which to revolve.

   Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:
6. Select a Revolve type:
   - **Full** - Revolve about the axis 360 degrees
   - **One direction** - Revolve in one direction for a specified number of degrees
   - **Symmetric** - Revolve in both directions for a specified number of degrees
   - **Two directions** - Revolve in both directions for the same or different numbers of degrees

7. If necessary, select a Merge scope (or Merge with all) to select surfaces with which to merge the new (additional) surface.

8. Click ⬗️.
**New** - Create new material that results in a new part.

**Add** - Create material and add to the existing material.

When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you have the ability to check Merge with all to add all touching or intersecting parts to the merge scope.
If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Revolve: Remove**

Take material away:

![Diagram of Revolve: Remove]

**Revolve: Intersect**

Leave material only where intersections exist:

![Diagram of Revolve: Intersect]
End type examples

Full

Revolve about the axis 360 degrees:

One direction

Revolve in one direction for a specified angle:
Two Directions

Revolve in two directions for specific angles:

Symmetric

Revolve in both directions for the same angle:
Project a selected region or planar face about an axis or Mate connector (inferred or existing). Create a new part or modify an existing one by adding or removing material, or intersecting parts in its path. You also have the ability to create parts or surfaces.

**Merge scope**

Merge scope is available with parts and surfaces and allows you to select the specific part or surface with which to merge the newly created part or surface. By default, Merge all is selected. You are able to uncheck that box to access the Merge scope field, then select the part or surface with which to merge. Surfaces must be merged with surfaces and parts with parts.
Merge scope: with all

Merge with all parts that touch or intersect the geometry being created:

Merge scope: with specific part

Merge with selected part:

Revolve: iOS
Create, add to, subtract from, or intersect parts by revolving sketch regions or planar faces about a central axis, or surfaces by revolving lines and curves about a central axis.

**Steps**

1. Tap Revolve tool.

![Image of Revolve 1 tool]

2. Select Creation type:
   - **Solid** - Create parts or modify existing parts by revolving a sketch region or face about an axis.
   - **Surface** - Create a surface by revolving a sketch curve or edge about an axis.

3. Select a Result body operation type:
   - **New** - Create new material.
   - **Add** - Create material and add to the existing material.
• **Remove** - Take material away.
• **Intersect** - Leave material only where intersections exist.

4. Select Faces and sketch regions to revolve.

5. Select an axis about which to revolve.

6. Select a Revolve type:
   • **Full** - Revolve about the axis 360 degrees.
   • **One direction** - Revolve in one direction for a specified angle.
   • **Symmetric** - Revolve in both directions for the same angle.
   • **Two directions** - Revolve in two directions at the same angle OR two different angles.

7. Tap checkmark.

**Revolve: New/Add (new material)**

**New** - Create new material that results in a new part

**Add** - Create material and add to the existing material

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When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts is you can check **Merge with all** to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Revolve: Remove (cut material)**

Remove - take material away
**Revolve: Intersect**

*Intersect* - Leave material only where intersections exist

**Revolve: Surface**

Create a surface along a sketch curve.
End type examples

These examples all show the revolve of a simple part or surface, creating new material.

Full

Revolve about the axis 360 degrees.
One direction

Revolve in one direction for a specified angle.
Symmetric

Revolve in both directions for the same angle.
Two directions

Revolve in two directions at the same angle OR two different angles.
Revolve: Android

Create, add to, subtract from, or intersect parts by revolving sketch regions or planar faces about a central axis, or surfaces by revolving lines and curves about a central axis.

Steps

1. Tap Revolve tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts by revolving a sketch region or face about an axis.
   - **Surface** - Create a surface by revolving a sketch curve or edge about an axis.

3. Select a Result body operation type:
   - **New** - Create new material.
   - **Add** - Create material and add to the existing material.
• **Remove** - Take material away.

• **Intersect** - Leave material only where intersections exist.

4. Select Faces and sketch regions to revolve.

5. Select an axis about which to revolve.

6. Select a Revolve type:

• **Full** - Revolve about the axis 360 degrees.

• **One direction** - Revolve in one direction for a specified angle.

• **Symmetric** - Revolve in both directions for the same angle.

• **Two directions** - Revolve in two directions at the same angle OR two different angles.

7. Tap checkmark.

**Revolve: New/Add (new material)**

**New** - Create new material that results in a new part

**Add** - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts is you can check **Merge with all** to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Revolve: Remove (cut material)**

**Remove** - take material away:
**Revolve: Intersect**

*Intersect* - Leave material only where intersections exist:

**Revolve: Surface**

Create a surface along a sketch curve:
End type examples

These examples all show the revolve of a simple part or surface, creating new material.

Full

Revolve about the axis 360 degrees:
One direction

Revolve in one direction for a specified angle:
Symmetric

Revolve in both directions for the same angle:
Two directions

Revolve in two directions at the same angle OR two different angles:
Sweep

Define a shape using a selected region, curves, or planar face moving along a path (either solid or surface). Create a new part or modify an existing one by adding or removing material, or intersecting parts in its path.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sweep: Desktop**

In the Feature toolbar:

Define a shape using a selected region, curves, or planar face moving along a path (either solid or surface). Create a new part or modify an existing one by adding or removing material, or intersecting parts in its path.

**Steps for creating solids**
1. Click 🎨.

![Sweep 1 dialog box](image)

2. Select Solid Creation type.

3. Select a Result operation type:
   - **New** - Create a new solid
   - **Add** - Add to an existing solid
   - **Remove** - Subtract from an existing solid
   - **Intersect** - Keep the intersection of two (or more) solids

4. Select the face or edge to sweep.

5. Click to make the **Sweep path** field active. Select a line segment or curve on the sketch or an edge on the part.

6. Select **Keep profile orientation** to maintain the profile relationship along the sweep path. Uncheck this field to maintain the profile relationship with the global plane.

   With Keep profile orientation:
Without Keep profile orientation:

7. If necessary, select a Merge scope (or Merge with all) to select parts with which to merge the new (additional) part.

8. Click  

If the sweep feature references a face or sketch region that is along the sweep path (not at an end) the path is broken where the profile plane touches it and the sweep is created in both directions, for the length of the sweep path.

For best results, the profile and the path should touch.

Steps for creating surfaces

1. Click  

2. Select Surface Creation type.
3. Select a Result operation type:
   - **New** - Create a new surface
   - **Add** - Add to an existing surface

4. Select the face or edge to sweep.

5. Click to make the **Sweep path** field active. Select a line segment or curve on the sketch or an edge.

6. Select **Keep profile orientation** to maintain the profile relationship along the sweep path. Uncheck this field to maintain the profile relationship with the global plane.

   With Keep profile orientation:

   ![With Keep profile orientation](image1)

   Without Keep profile orientation:

   ![Without Keep profile orientation](image2)

7. If necessary, select a Merge scope (or Merge with all) to select surfaces with which to merge the new (additional) surface.

8. Click ✅.
If the sweep feature references a face or sketch region that is along the sweep path (not at an end) the path is broken where the profile plane touches it and the sweep is created in both directions, for the length of the sweep path.

For best results, the profile and the path should touch.

**Sweep: New/Add (new material)**

**New** - Create new material that results in a new part:

**Add** - Add material to the existing material:
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you can check Merge with all to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Sweep: Remove (cut material)**

Remove material from existing material; not available for surfaces.
**Sweep: Intersect**

Leave material only where geometry intersects; not available for surfaces.

**Merge scope**

Merge scope is available with parts and surfaces and allows you to select the specific part or surface with which to merge the newly created part or surface. By default, Merge all is selected. You are able to uncheck that box to access the Merge scope field, then select the part or surface with which to merge. Surfaces must be merged with surfaces and parts with parts.
Merge scope: with all

Merge with all parts the new part intersects:

Merge scope: particular part

Select a specific part with which to merge:
**Sweep: iOS**

Define a shape using a selected region, curves, or planar face moving along a path (either solid or surface). Create a new part or modify an existing one by adding or removing material, or intersecting parts in its path.

**Steps**

1. Tap Sweep tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts by sweeping a sketch region along a path.
   - **Surface** - Create a surface by sweeping a sketch curve along a path.

3. If you select to create a solid, also select a result body operation type:
   - **New** - Create a new solid.
   - **Add** - Add to an existing solid.
   - **Remove** - Subtract from an existing solid.
   - **Intersect** - Keep the intersection of two (or more) solids.

4. Select the face or edge to sweep.

5. Specify the sweep path (select a line segment or curve on the sketch or an edge on the part).

6. Specify whether to keep the profile orientation.
7. Tap the checkmark.

**Sweep: New/Add (new material)**

**New** - Create new material that results in a new part

When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.

- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).

- A shortcut to selecting multiple touching or intersecting parts, check **Merge with all**
to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Sweep: Remove (cut material)**

*Remove* - Take material away

**Sweep: Intersect**

*Intersect* - Leave material only where intersections exist

**Sweep: Surface**

Create a surface by sweeping a sketch curve along a path.
**Sweep: Android**

Define a shape using a selected region, curves, or planar face moving along a path (either solid or surface). Create a new part or modify an existing one by adding or removing material, or intersecting parts in its path.

**Steps**

1. Tap Sweep tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts by sweeping a sketch region along a path.
   - **Surface** - Create a surface by sweeping a sketch curve along a path.

3. If you select to create a solid, also select a result body operation type:
   - **New** - Create a new solid.
- Add - Add to an existing solid.
- Remove - Subtract from an existing solid.
- Intersect - Keep the intersection of two (or more) solids.

4. Select the face or edge to sweep.
5. Specify the sweep path (select a line segment or curve on the sketch or an edge on the part).
6. Specify whether to keep the profile orientation.
7. Tap the checkmark.

**Sweep: New/Add (new material)**

**New** - Create new material that results in a new part

**Add** - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you can check **Merge with all** to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Sweep: Remove (cut material)**

**Remove** - take material away
**Sweep: Intersect**

*Intersect* - Leave material only where intersections exist

**Sweep: Surface**

Create a surface by sweeping a sketch curve along a path.
Loft

Use profiles (sketch regions or sketch curves) and optional guide curves to define shapes that smoothly transition between them. Create parts or surfaces or modify existing parts or surfaces.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Loft: Desktop**

In the Feature toolbar:

Use profiles (sketch regions or sketch curves) and optional guide curves to define shapes that smoothly transition between them. Create parts or surfaces or modify existing parts or surfaces.

**Steps for creating solids**

1. Click 🔄.
2. Select Solid Creation type.

3. Specify a Result operation type:
   - **New** - Create a new solid.
   - **Add** - Add to an existing solid.
   - **Remove** - Subtract from an existing solid.
   - **Intersect** - Keep only the intersection of two (or more) solids.

4. Select profiles (a region, face, edge, or point) and then optional cross-sections (in order of the loft direction) and finally the end (region, face, edge, or point).

   To select a set of tangentially connected curves as a single chain, click the arrow next to the desired selection in the dialog to expand the selection field. (A blue field is an active field.) Select more curves to create a composite selection.

   For example: To select both circles in the end loft position, select the first circle, then click in the field where the first selection appears and then click the second selection:
5. To refine the shape further, select a **Start or End Profile condition** to define the derivative constraints on the start and end profiles:

a. Normal to profile - Causes the loft to touch the profile normal to the profile plane.

b. Tangent to profile - Causes the loft to touch the profile tangent to the profile plane.

c. Match tangent - Causes the loft to match the tangents of loft faces to the tangents of model faces adjacent to the profile face (if available).

d. Match curvature - Same as Match tangent, but applies to a curvature constraint.

6. Optionally, use a **guide** curve or curves for the loft to follow; it is not necessary for the guide curves to be touching the outsides of the profiles, only for them to intersect.

a. Select the box next to **Guides and continuity**.

b. Select the curve (or curves) to act as guides.

To select tangentially connected curves as a single guide, click the down arrow next to the selected guide to open the field for more selections. Make additional selections:
1 and 2: Each of these is a single guide selection (“Edge of Loft 1” and “Edge of Sketch 3”).

3: Click the arrow next to the guide name to expand the field.

4: The blue highlighting indicates that field is active. At this point, you can select more adjacent curves to create a composite guide selection.

For further definition, use the **Continuity** condition on the guide. Continuity can be:

- **Match tangent** - Causes the loft to match the tangents of loft faces to the tangents of the guides adjacent to the profile.

- **Match curvature** - Same as Match tangent, but applies to a curvature constraint.

Make sure that your sketch is consistent with what you are selecting, if the sketch is inconsistent with Match tangent and it is selected, the loft will fail. The same is true for Match curvature.

7. To create a centerline equivalent, select a **Path** for the loft to follow (and create intermediate sections along the path for the loft to reference).

   a. Click the box next to Path.

   b. Select edges, curves, and sketches to act as the **path** (centerline guide) of the loft.

   c. Specify the section count (number of intermediate sections) to be used along the path. The more sections used, the more closely the path is followed.

   For example: Straight line selected as the path, section count = 3
8. Optionally, select **vertices** to have more control on the twist of the resulting surface. If there are guides, those are used for alignment, if not Onshape estimates the proximity within the existing vertices. It is best to have at least two vertices on each profile and use the matching vertices to control twist:

a. Click Match vertices.

b. Select one set of vertices (one vertex on each region/face/edge/point).

9. If necessary or desired, select a Merge scope (or Merge with all) to select parts with which to merge the new (additional) part. See more on Merge scope below.

10. Click ☑️

**Steps for creating surfaces**
1. Click 🕒.

2. Select **Surface** Creation type.

3. Specify a Result operation type:
   - **New** - Create a new surface.
   - **Add** - Add to an existing surface.

4. Select **profiles** (a region, face, edge, or point) and then optional cross-sections (in order of the loft direction) and finally the end (region, face, edge, or point).

   To select a set of tangentially connected curves as a single chain, click the arrow next to the desired selection in the dialog to expand the selection field. (A blue field is an active field.) Select more curves to create a composite selection.

5. To refine the shape further, select a **Profile condition** to define the derivative constraints on the start and end profiles:
a. **Normal to profile** - Causes the loft to touch the profile with tangents on the profile plane.

b. **Tangent to profile** - Causes the loft to touch the profile with tangents on the profile plane.

c. **Match tangent** - Causes the loft to match the tangents of loft faces to the tangents of model faces adjacent to the profile face (if available).

d. **Match curvature** - Same as Match tangent, but applies to a curvature constraint.

6. Optionally, use a guide curve or curves for the loft to follow; guide curves must be touching the outsides of the profiles, not the centers.

a. Select the box next to Guides and continuity.

b. Select the curve (or curves) to act as guides.

To select tangentially connected curves as a single guide, click the down arrow next to the selected guide to open the field for more selections. Make additional selections:

1 and 2: Each of these is a single guide selection (“Edge of Loft 1” and “Edge of Loft 2”).

3: Click the arrow next to the guide name to expand the field.

4: The blue highlighting indicates that field is active. At this point, you can select more adjacent curves to create a composite guide selection.

For further definition, use the **Continuity** condition on the guide. Continuity can be:
• **Match tangent** - Causes the loft to match the tangents of loft faces to the tangents of the guides adjacent to the profile.

• **Match curvature** - Same as Match tangent, but applies to a curvature constraint.

Make sure that your sketch is consistent with what you are selecting, if the sketch is inconsistent with Match tangent and it is selected, the loft will fail. The same is true for Match curvature.

7. **Trim guides** and **Trim profiles** become available when *Guides and continuity* is selected. These options allow you to control how the guides influence the loft operation:

**Trim guides** - Trim the loft operation to the boundaries of the guides:

The image below is without any Trim selected, the loft extends the length of the profiles and also the length of the guides (which are highlighted below):

![Guides and continuity](image)

The image below is with Trim guides selected, the loft is trimmed to the intersection of the profiles and the guides, along the guides:
Trim profiles - Trim the loft operation to the intersection of the profiles and the guides, along the profiles. The image below shows the profiles trimmed:

The image below shows both Trim guides and Trim profiles selected, which trims the loft along both the guides and the profiles:

8. To create a centerline equivalent, select a Path for the loft to follow (and create intermediate sections along the path for the loft to reference).
a. Click the box next to Path.

b. Select edges, curves, and sketches to act as the path (centerline guide) of the loft.

c. Specify the section count (number of intermediate sections) to be used along the path. The more sections used, the more closely the path is followed.

For example: Straight line selected as the path, section count = 3

Spline selected as the path, section count = 10

9. Optionally, select **vertices** to have more control on the twist of the resulting surface. If there are guides, those are used for alignment, if not Onshape estimates the proximity within the existing vertices. It is best to have at least two vertices on each profile and use the matching vertices to control twist:
a. Click Match vertices.

b. Select one set of vertices (one vertex on each region/face/edge/point).

10. If necessary or desired, select a Merge scope (or Merge with all) to select surfaces with which to merge the new (additional) surface. See more on Merge scope below.

11. Click ✓.

**Loft with guides and continuity**

**Surface / Add / Guides / Match curvature** - Create material and add it to the existing material.

---

**Loft: New/Add (new material)**

New - Create new material that results in a new part or surface.
**Add** - Create material and add to the existing material. (This example is merge with all existing material; you could also select one part as the merge scope.)

- When adding material, you have the option to merge that material with other parts that touch or intersect its geometry.
- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).

A shortcut to selecting multiple touching or intersecting parts, you can check Merge with all to add all touching or intersecting parts to the merge scope.

Note that if the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Loft: Remove (cut material)**

Take material away from existing material by selecting sketches along the loft profile; not available for surfaces.

**Loft: Intersection**

Leave material only where selected geometry overlaps; if necessary, select Merge with all to complete the process; not available for surfaces.
Loft with path (centerline guide)

Select a path to use as a centerline equivalent (guide) for the loft and as a way to control the global shape of the loft. It is not necessary for this guide to be at the center. Specify the number of intermediate sections along the path to fine-tune the shape of the loft along the path.

Loft with no path:
Loft with path and 2 intermediate sections:
Loft with path and 20 intermediate sections:
Merge scope

Merge scope is available with parts and surfaces and allows you to select the specific part or surface with which to merge the newly created part or surface. By default, Merge all is selected. You are able to uncheck that box to access the Merge scope field, then select the part or surface with which to merge. Surfaces must be merged with surfaces and parts with parts.
Merge scope: with all

Merge extrusion with all parts it intersects

Merge scope: particular part

Select a specific part with which to merge
End conditions

‘None’ end condition

‘Normal to profile’ end condition

Causes the loft to touch the profile with tangents parallel to the profile’s normal. In this example, both profiles are ‘normal to profile.’
You are able to use Magnitude to adjust the shape according to one profile or another by increasing one Magnitude. In this example, the bottom profile’s Magnitude is increased from 1 to 3, thereby extending further the bottom profile shape towards the top profile:

‘Tangent to profile’ end condition

Causes the loft to touch the profile with tangents on the profile plane. This example has Tangent to profile applied to the top profile only:
You are able to use Magnitude to adjust the shape according to one profile or another by increasing one Magnitude. In this example, the top profile’s Magnitude is increased from 1 to 3, thereby extending further the top profile shape towards the bottom profile:

‘Match curvature’ end condition

Causes the loft to touch the profile with tangents on the profile plane, with a curvature constraint.
'Match tangent’ end condition

Before Match tangent is selected:

After Match tangent is selected:

You are able to use Magnitude to adjust the shape according to one profile or another by increasing one Magnitude. In this example, the bottom profile’s Magnitude is increased from 1 to 3, thereby extending further the bottom profile shape towards the top profile:
**Match vertices**

Select one set of vertices (one vertex on each profile).

**Tips**

- For best results, all profiles should have the same number of curve segments.
- Vertex selection must be one vertex from each profile.
- Profiles (regions) and guides to be used in a loft operation each must be a single entry in the entry field.
- When working with multi-edge guide curves make sure one sketch defines the guide; select it from the Feature list.
- Make sure to select profiles (regions, faces, edges, or points) in the correct order from the start of the loft to the end.
- Guide curves need to be smooth (multi-edge curves must be tangent), and they must touch the profile (use Coincident or Pierce constraints).
- After creating the loft, use the Final button during editing to visualize the result and fine tune the operation.
- Nested loops in profiles are currently not supported.
- To select tangentially connected curves as a single guide, select them from the Feature list as a complete sketch, or from the Parts list as Curves.

Loft: iOS

Use profiles (sketch regions or sketch curves) and optional guide curves to define shapes that smoothly transition between them. Create parts or surfaces or modify existing parts or surfaces.

Steps

1. Tap Loft tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts.
   - **Surface** - Create a surface along a sketch curve.

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select Profiles to loft:
   First select the start profile (a region, face, edge, or point), and then optional cross-sections (in order of the loft direction) to help bound the loft, and finally the end (a region, face, edge, or point).

5. Select a path (centerline guide) for the loft to follow (and create intermediate sections along the path for the loft to reference).
   a. Tap to toggle **Path**.
   b. Select edges, curves, and sketches to act as the path (centerline guide) of the loft.
   c. Specify the section count (number of intermediate sections) to be used along the path. The more sections used, the more closely the path is followed.

6. Select a control type (to help define the loft) or end conditions:
   - **None**
   - **Guides** - Select the guide lines (guide lines must be touching the outsides of the profiles, not the centers).
     To select a set of connected curves as a single chain, select them from the Feature list as a complete sketch.
   - **End conditions** - Select start profile condition and End profile condition (derivative constraints on the start and end profiles):
     - **Normal to profile** - Causes the loft to touch the profile with tangents parallel to the profile’s normal
Tangent to profile - Causes the loft to touch the profile with tangents on the profile plane

Match tangent - Causes the loft to match the tangents of loft faces to the tangents of model faces adjacent to the profile face (if available)

Match curvature - Same as Match tangent, but applies to a curvature constraint.

For each end condition (start profile and end profile), you can specify a magnitude (use the number pad to change these values).

7. Select optional vertices to match (to define corresponding locations on each profile):
   a. Tap to toggle Match vertices.
   b. Select one set of vertices (one vertex on each region/face/edge/point).

Loft: New/Add (new material)

New - Create new material that results in a new part
Add - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.

- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).

- A shortcut to selecting multiple touching or intersecting parts, you can check Merge with all to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Loft: Remove (cut material)**
**Remove** - Take material away

**Loft: Intersect**

**Intersect** - Leave material only where intersections exist
Loft: Surface
**Loft with path (centerline guide)**

Select a path to use as a centerline guide for the loft. Specify the number of intermediate sections along the path to fine-tune the shape of the loft along the path.

Loft with no path:

Loft with path and 1 section:
Loft with path and 20 intermediate sections:
Control type examples

None

Perform a loft without a control type.
Guides

Select guide lines (guide lines must be touching the outsides of the profiles, not the centers).

To select tangentially connected curves as a single guide, select them from the Features list as a complete sketch.
End conditions

Select start profile condition and end profile condition (derivative constraints on the start and end profiles). For each end condition (start profile and end profile), you are able to specify a magnitude.

**Normal to profile** - Causes the loft to touch the profile with tangents parallel to the profile's normal
**Tangent to profile** - Causes the loft to touch the profile with tangents on the profile plane
**Match tangent** - Causes the loft to match the tangents of loft faces to the tangents of model faces adjacent to the profile face (if available)
Match curvature - Same as Match tangent, but applies to a curvature constraint.
**Match vertices**

Select one set of vertices (one vertex on each region/face/edge/point).
Tips

- For best results, all profiles should have the same number of curve segments.
- Vertex selection must be one vertex from each profile.
- Profiles (regions) and guides to be used in a loft operation each must be a single entry in the entry field.
- When working with multi-edge guide curves make sure one sketch defines the guide; select it from the Feature list.
- Make sure to select profiles (regions, faces, edges, or points) in the correct order from the start of the loft to the end.
- Guide curves need to be smooth (multi-edge curves must be tangent), and they must touch the profile (use Coincident or Pierce Constraints).
- After creating the loft, use the Final button during editing to fine tune the operation.
- Nested loops in profiles are currently not supported.

Loft: Android

Use profiles (sketch regions or sketch curves) and optional guide curves to define shapes that smoothly transition between them. Create parts or surfaces or modify existing parts or surfaces.
Steps

1. Tap Loft tool.
2. Select Creation type:
   - **Solid** - Create parts or modify existing parts.
   - **Surface** - Create a surface along a sketch curve.

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select Profiles to loft:
   First select the start profile (a region, face, edge, or point), and then optional cross-sections (in order of the loft direction) to help bound the loft, and finally the end (a region, face, edge, or point).

5. Select a path (centerline guide) for the loft to follow (and create intermediate sections along the path for the loft to reference).
   a. Tap to toggle **Path**.
   b. Select edges, curves, and sketches to act as the path (centerline guide) of the loft.
   c. Specify the section count (number of intermediate sections) to be used along the path. The more sections used, the more closely the path is followed.

6. Select a control type (to help define the loft) or end conditions:
   - **None**
   - **Guides** - Select the guide lines (guide lines must be touching the outsides of the profiles, not the centers).
     To select a set of connected curves as a single chain, select them from the Feature list as a complete sketch.
   - **End conditions** - Select start profile condition and End profile condition (derivative constraints on the start and end profiles):
     - **Normal to profile** - Causes the loft to touch the profile with tangents parallel to the profile’s normal
- **Tangent to profile** - Causes the loft to touch the profile with tangents on the profile plane

- **Match tangent** - Causes the loft to match the tangents of loft faces to the tangents of model faces adjacent to the profile face (if available)

- **Match curvature** - Same as Match tangent, but applies to a curvature constraint.

  For each end condition (start profile and end profile), you are able to specify a magnitude (use the number pad to change these values).

  7. Select optional vertices to match (to define corresponding locations on each profile):

  a. Tap to toggle **Match vertices**.

  b. Select one set of vertices (one vertex on each region/face/edge/point).

**Loft: New/Add (new material)**

**New** - Create new material that results in a new part
**Add** - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.

- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).

- A shortcut to selecting multiple touching or intersecting parts, check **Merge with all** to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Loft: Remove (cut material)**
**Remove** - Take material away

**Loft: Intersect**

**Intersect** - Leave material only where intersections exist
Loft: Surface
Loft with path (centerline guide)

Select a path to use as a centerline guide for the loft. Specify the number of intermediate sections along the path to fine-tune the shape of the loft along the path.

Loft with no path:

Loft with path and 1 section:
Loft with path and 20 intermediate sections:
Control type examples

None

Perform a loft without a control type.
Guides

Select guide lines (guide lines must be touching the outsides of the profiles, not the centers).

To select tangentially connected curves as a single guide, select them from the Features list as a complete sketch.
End conditions

Select start profile condition and end profile condition (derivative constraints on the start and end profiles). For each end condition (start profile and end profile), you are able to specify a magnitude.

**Normal to profile** - Causes the loft to touch the profile with tangents parallel to the profile's normal
**Tangent to profile** - Causes the loft to touch the profile with tangents on the profile plane.
**Match tangent** - Causes the loft to match the tangents of loft faces to the tangents of model faces adjacent to the profile face (if available)
**Match curvature** - Same as Match tangent, but applies to a curvature constraint.
Match vertices

Select one set of vertices (one vertex on each region/face/edge/point).
**Tips**

- For best results, all profiles should have the same number of curve segments.
- Vertex selection must be one vertex from each profile.
- Profiles (regions) and guides to be used in a loft operation each must be a single entry in the entry field.
- When working with multi-edge guide curves make sure one sketch defines the guide; select it from the Feature list.
- Make sure to select profiles (regions, faces, edges, or points) in the correct order from the start of the loft to the end.
- Guide curves need to be smooth (multi-edge curves must be tangent), and they must touch the profile (use Coincident or Pierce Constraints).
- After creating the loft, use the Final button during editing to fine tune the operation.
- Nested loops in profiles are currently not supported.
Add depth to a surface. Create a new part or modify an existing one by giving thickness to a surface and convert it to a solid, adding or removing material from an existing part or surface, or intersecting parts in its path.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Thicken: Desktop**

In the Feature toolbar:

Add depth to a surface. Create a new part or modify an existing one by giving thickness to a surface and convert it to a solid, adding or removing material from an existing part or surface, or intersecting parts in its path.

**Steps**

1. Click 📦.

2. Select whether to:
   - **New** - Create a new solid
   - **Add** - Add to an existing solid
   - **Remove** - Subtract material from an existing part
   - **Intersect** - Keep only intersecting materials

3. Select the part face (or surface) in the graphics area.
4. Specify the thickness of material to be added or removed.

5. Optionally, select a direction using the arrows.

6. Specify a value for Direction 2 to thicken the part or surface in the opposite direction as well.

7. Optionally, specify a Merge Scope to indicate whether to incorporate the new material with all parts or a specific part, where appropriate.

8. Click ✓.

**Thicken: New/Add (new material)**

**New** - Create new material that results in a new part:

![New Material](image1)

**Add** - Create material and add to the existing material:

![Add Material](image2)

**Thicken: Remove (cut material)**

Take material away:
**Thicken: Intersect**

Leave material only where geometry overlaps:

**Tips**

When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.

- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).

- A shortcut to selecting multiple touching or intersecting parts, you are able to check Merge with all to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Thicken: iOS**
Add depth to a surface. Create a new part or modify an existing one by giving thickness to a surface and convert it to a solid, adding or removing material from an existing part or surface, or intersecting parts in its path.

**Steps**

1. Tap Thicken tool.

![Thicken 1](image)

**Select surfaces or faces to thicken.**

Result body operation type

- **New** - Create a new solid.
- **Add** - Add to an existing solid.
- **Remove** - Subtract material from an existing part.
- **Intersect** - Keep only intersecting materials.

2. Select a result body operation type:

3. Select faces and surfaces to thicken.

4. Specify the thickness of material to be added or removed for the first direction.

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5. Optionally, select a direction using the **Opposite direction** toggle.

6. Specify a value for **Direction 2** to thicken the part or surface in the opposite direction as well.

7. Optionally, specify a Merge scope to indicate whether to incorporate the new material with all parts or a specific part, where appropriate.

8. Tap the checkmark.

**Thicken: New/Add (new material)**

**New** - Create new material that results in a new part

**Add** - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, check **Merge with all** to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Thicken: Remove (cut material)**

**Remove** - Take material away
**Thicken: Intersect**

*Intersect* - Leave material only where intersections exist

*Thicken: Android*
Add depth to a surface. Create a new part or modify an existing one by giving thickness to a surface and convert it to a solid, adding or removing material from an existing part or surface, or intersecting parts in its path.

**Steps**

1. Tap Thicken tool.

---

**Thicken 1**

Select surfaces or faces to thicken.

**New** Add Remove Intersect

Faces and surfaces to thicken

No entities selected

**Direction 1**

0.25 in

**Opposite direction**

0.0 in
2. Select a result body operation type:
   - **New** - Create a new solid.
   - **Add** - Add to an existing solid.
   - **Remove** - Subtract material from an existing part.
   - **Intersect** - Keep only intersecting materials.

3. Select faces and surfaces to thicken.

4. Specify the thickness of material to be added or removed for the first direction.

5. Optionally, select a direction using the **Opposite direction** toggle.

6. Specify a value for **Direction 2** to thicken the part or surface in the opposite direction as well.

7. Optionally, specify a Merge scope to indicate whether to incorporate the new material with all parts or a specific part, where appropriate.

8. Tap the checkmark.

**Thicken: New/Add (new material)**

**New** - Create new material that results in a new part
Add - Create material and add to the existing material
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, check **Merge with all** to add all touching or intersecting parts to the merge scope.

If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

**Thicken: Remove (cut material)**

**Remove** - take material away
Thicken: Intersect

Intersect - Leave material only where intersections exist
Enclose

Create a part by selecting all boundaries surrounding an empty space to form a solid. Use any set of surfaces and solids (including planes and faces) that intersect each other or connect at a boundary to create a volume.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Enclose: Desktop

In the Feature toolbar:

Create a part by selecting all boundaries surrounding an empty space to form a solid. Use any set of surfaces and solids (including planes and faces) that intersect each other or connect at a boundary to create a volume.

Steps

1. Click 📚.

2. Select the entities that surround the volume to be enclosed.

   Optionally select Keep tools to retain the selected entities at the creation of the new part. If Keep tools is not selected, those owning parts of any selection (not from a sketch or a plane) will be deleted.

3. Click ✔️.
Examples

In the first image, the surfaces and the plane are selected as boundaries. In the second image, the surfaces are deleted (no Keep tools) and the volume bounded by the plane and surfaces is now a solid part.

Keep tools

When Keep tools is selected, the surfaces remain and the volume is a solid part.

Tip

If the selection of boundaries results in multiple solids, Onshape automatically combines the solids to form one part.

Enclose: iOS

Create a part by selecting all boundaries surrounding an empty space to form a solid. Use any set of surfaces and solids (including planes and faces) that intersect each other or connect at a boundary to create a volume.
Steps

1. Tap 

2. Select the entities that surround the volume to be enclosed.

   Optionally select Keep tools to retain the selected entities at the creation of the new part. If Keep tools is not selected, those owning parts of any selection (not from a sketch or a plane) will be deleted.

3. Tap 

Examples

In the first image, the surfaces and the plane are selected as boundaries. In the second image, the surfaces are deleted (no Keep tools) and the volume bounded by the plane and surfaces is now a solid part.
Keep tools

When Keep tools is selected, the surfaces remain and the volume is a solid part:

Tip

If the selection of boundaries results in multiple solids, Onshape automatically combines the solids to form one part.

Enclose: Android

Create a part by selecting all boundaries surrounding an empty space to form a solid. Use any set of surfaces and solids (including planes and faces) that intersect each other or connect at a boundary to create a volume.

Steps

1. Tap 📋.
2. Select the entities that surround the volume to be enclosed.

   Optionally select Keep tools to retain the selected entities at the creation of the new part. If Keep tools is not selected, those owning parts of any selection (not from a sketch or a plane) will be deleted.

3. Tap ✔.

Examples

In the first image, the surfaces and the plane are selected as boundaries. In the second image, the surfaces are deleted (no Keep tools) and the volume bounded by the plane and surfaces is now a solid part.

Keep tools

When Keep tools is selected, the surfaces remain and the volume is a solid part.

Tip

If the selection of boundaries results in multiple solids, Onshape automatically combines the solids to form one part.
Fillet

Round sharp interior and exterior edges and define as a standard constant radius, more stylized conic or variable.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Fillet: Desktop

Shortcut: Shift-f

In the Feature toolbar:

Round sharp interior and exterior edges and define as a standard constant radius, more stylized conic or variable.

Steps

1. Click 📐.
2. Select any edges or faces of the part you want to round or fillet. Onshape automatically applies the correct feature to the edge.

When filleting sheet metal, you might have to select the exact corner (not an adjacent edge). You are able to select any combination of edges on parts, and corners on sheet metal. For more information on using Fillet and Chamfer tools with sheet metal, see "Sheet Metal Model" on page 1317.

3. By default, Tangent propagation is set to extend the fillet to tangent edges. Uncheck if you don’t want to extend the fillet to all tangent edges.

4. Select a cross section type:

- Circular - Fillet has a circular edge with the radius value you enter
- Conic - Fillet has a conical edge with the radius value you enter and optionally a Rho value to define the style of the fillet:
  
  ![Conic Fillet Examples]

  Rho 0.25 - Elliptical curve  Rho 0.5 - Parabolic curve  Rho 0.999 - Hyperbolic curve

- Curvature - Fillet matches the curvature of the surrounding edges with a radius value you enter and optionally a Magnitude value between 0 and .999 to tweak the tangency. Turn Curvature visualization on in the small View cube to see the
tangency more clearly:

When entering a radius value you also have the ability to use the drag manipulator, as indicated in the image below by an arrow, to visualize the fillet and approach an estimated value:

5. By default, Allow edge overflow is set to allow an edge to be 'deleted' if necessary. When Allow edge overflow is enabled, the fillet can modify edges on the face created by the fillet to make a smooth, continuous surface. When the option is
disabled, the edge on the filleted face is preserved:

Notice how the original edge remains in place when Allow edge overflow is not selected:

6. Check Variable fillet to vary the shape and size of the fillet by selecting vertices to which to apply specific values. (This is available for all Cross section types.)

   a. Select a vertex (available vertices are indicated by black dots in the model).

   b. Adjust definition as described above for each cross section type (Circular, Conic, Curvature).

   In this example, the Circular fillet has a radius of 8mm (seen in the upper portion of the view), and the two orange (selected) vertices have radii of 2mm:
7. If you have applied a variable fillet, you have the option to check Smooth transition to smooth out the lines between the fillet vertices.

Turn Curvature visualization on to aid in identifying this benefit; Curvature visualization is available in the small View cube menu.

8. Click ✓

**Tangent propagation**

Select one face to fillet:

Then select Tangent propagation to extend the fillet to all tangent faces:
**Variable fillet**

Apply the desired fillet (here, 0.2 radius):

Check Variable fillet and select as many vertices (2 orange vertices, below) as needed and supply new radius (here, 0.8) for each vertex selected:

**Smooth transition**
This option is available only when a variable fillet has been applied. Turn on Curvature visualization to see the effects. Notice now the stripes transition smoothly from one face to another:

**Circular cross-section**

With variable fillet:

**Conic cross-section**

Rho value less than 0.5 (0.1):
Rho value 0.5:

Rho value greater than 0.5 (0.999):

**Curvature cross-section**

With magnitude 0.5:

With magnitude 0.999:

**Fillet: iOS**

Round sharp interior and exterior edges and define as a standard constant radius, more stylized conic or variable.

**Steps**
1. Tap Fillet tool.

2. Select any edges or faces of the part you want to round or fillet. Onshape automatically applies the correct feature to the edge.

   When filleting sheet metal, you might have to select the exact corner (not an adjacent edge). You are able to select any combination of edges on parts, and corners on sheet metal. For more information on using Fillet and Chamfer tools with sheet metal, see "Sheet Metal Model" on page 1317.

3. By default, Tangent propagation is set to extend the fillet to tangent edges. Turn off if you don’t want to extend the fillet to all tangent edges.
4. Select a cross section type:
   - Circular - Fillet has a circular edge with the radius value you enter
   - Conic - Fillet has a conical edge with the radius value you enter and optionally a Rho value to define the style of the fillet:
     - Rho 0.25 - Elliptical curve
     - Rho 0.5 - Parabolic curve
     - Rho 0.999 - Hyperbolic curve

   - Curvature - Fillet matches the curvature of the surrounding edges with a radius value you enter and optionally a Magnitude value between 0 and .999 to tweak the tangency.

When entering a radius value you can also use the drag manipulator, as indicated in the image below by an arrow, to visualize the fillet and approach an estimated value:

5. By default, Allow edge overflow is set to allow an edge to be 'deleted' if necessary. When Allow edge overflow is enabled, the fillet can modify edges on the face created by the fillet to make a smooth, continuous surface. When the option is disabled, the edge on the filleted face is preserved:
Notice how the original edge remains in place when Allow edge overflow is not selected:

6. Turn Variable fillet on to vary the shape and size of the fillet by selecting vertices to which to apply specific values. (This is available for all cross section types.)

   a. Select a vertex (available vertices are indicated by black dots in the model).
   
   b. Adjust definition as described above for each cross section type (Circular, Conic, Curvature).

   In this example, the Circular fillet has a radius of 8mm (seen in the upper portion of the view), and the two orange (selected) vertices have radii of 2mm:
7. If you have applied a variable fillet, you can optionally check Smooth transition to smooth out the lines between the fillet vertices.

8. Tap the checkmark.

**Tangent propagation**

Select one face to fillet:

Then select Tangent propagation to extend the fillet to all tangent faces:
Variable fillet

Apply the desired fillet (here, 0.2 radius):

Check Variable fillet and select as many vertices (2 orange vertices, below) as needed and supply new radius (here, 0.8) for each vertex selected:

Smooth transition

This option is available only when a variable fillet has been applied. Turn on Curvature visualization to see the effects. Notice now the stripes transition smoothly from one face to another:
Circular cross-section

With variable fillet

Conic cross-section

Rho value less than 0.5 (0.1):

Rho value 0.5:
Rho value greater than 0.5 (0.999):

Curvature cross-section
With magnitude 0.5:

With magnitude 0.999:

Fillet: Android
Round sharp interior and exterior edges and define as a standard constant radius, more stylized conic or variable.

Steps
1. Tap Fillet tool.
2. Select any edges or faces of the part you want to round or fillet. Onshape automatically applies the correct feature to the edge.
When filleting sheet metal, you might have to select the exact corner (not an adjacent edge). You are able to select any combination of edges on parts, and corners on sheet metal. For more information on using Fillet and Chamfer tools with sheet metal, see Sheet Metal Model.

3. By default, Tangent propagation is set to extend the fillet to tangent edges. Turn off if you don’t want to extend the fillet to all tangent edges.

4. Select a cross section type:

- Circular - Fillet has a circular edge with the radius value you enter
- Conic - Fillet has a conical edge with the radius value you enter and optionally a Rho value to define the style of the fillet:
  
  Rho 0.25 - Elliptical curve  Rho 0.5 - Parabolic curve  Rho 0.999 - Hyperbolic curve

- Curvature - Fillet matches the curvature of the surrounding edges with a radius value you enter and optionally a Magnitude value between 0 and .999 to tweak the tangency.

When entering a radius value you can also use the drag manipulator, as indicated in the image below by an arrow, to visualize the fillet and approach an estimated value:
5. By default, Allow edge overflow is set to allow an edge to be 'deleted' if necessary. When Allow edge overflow is enabled, the fillet can modify edges on the face created by the fillet to make a smooth, continuous surface. When the option is disabled, the edge on the filleted face is preserved:

Notice how the original edge remains in place when Allow edge overflow is not selected:
6. Turn Variable fillet on to vary the shape and size of the fillet by selecting vertices to which to apply specific values. (This is available for all cross section types.)
   a. Select a vertex (available vertices are indicated by black dots in the model).
   b. Adjust definition as described above for each cross section type (Circular, Conic, Curvature).

   In this example, the Circular fillet has a radius of 8mm (seen in the upper portion of the view), and the two orange (selected) vertices have radii of 2mm:

7. If you have applied a variable fillet, you have the option to check Smooth transition to smooth out the lines between the fillet vertices.

8. Tap the checkmark.

Tangent propagation
Select one face to fillet:

Then select Tangent propagation to extend the fillet to all tangent faces:

**Variable fillet**

Apply the desired fillet (here, 0.2 radius):

Check Variable fillet and select as many vertices (2 orange vertices, below) as needed and supply new radius (here, 0.8) for each vertex selected:
Smooth transition

This option is available only when a variable fillet has been applied. Turn on Curvature visualization to see the effects. Notice now the stripes transition smoothly from one face to another:

Circular cross-section

With variable fillet:
**Conic cross-section**
Rho value less than 0.5 (0.1):

Rho value 0.5:

Rho value greater than 0.5 (0.999):

**Curvature cross-section**
With magnitude 0.5:
Chamfer

Break sharp edges with a bevel. Define by the distance to break from the edge and by the angle made with the surface.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Chamfer: Desktop**

In the Feature toolbar:

Break sharp edges with a bevel. Define by the distance to break from the edge and by the angle made with the surface.

**Steps**

1. Click 🗄.
2. Select any edges or faces of the part to which to apply the chamfer.

When applying a chamfer to sheet metal, you might have to select the exact corner (not an adjacent edge). You can select any combination of edges on parts, and corners on sheet metal. For more information on using Fillet and Chamfer tools with sheet metal, see "Sheet Metal Model" on page 1317.

3. Enter a width for the chamfer; Onshape applies the 45 degree angle by default.

4. Optionally, check Tangent propagation to extend the selection along surrounding edges.

5. Click ✓.

**Equal-distance Chamfer**

Before:

![Before Image]

With:

![With Image]
Two-distance Chamfer

Before:

After:
Distance-and-angle Chamfer

Before:

After:

In this example, the distance is measured along the bottom face from the selected edge, and the angle is measured starting from the bottom face. If you use the Directional arrows and flip the distance measurement, you get the distance and angle measured with respect to the vertical face:
Cylindrical Chamfer

This example shows a chamfer on a cylindrical edge:

After:
Chamfer: iOS

Break sharp edges with a bevel. Define by the distance to break from the edge and by the angle made with the surface.

Steps
1. Tap Chamfer tool.
2. Select entities to chamfer (any edges or faces of a part).

3. Specify chamfer type:
   - **Equal distance** - to a specified distance in equal directions.
   - **Two distances** - to two individually specified distances.
   - **Distance and angle** - to a specified distance and angle.

4. Specify Distance (the width of the chamfer).

5. Optionally, toggle to use **Tangent propagation** to automatically extend the fillet to tangent edges.

6. Tap the checkmark.

**Equal-distance Chamfer**

This shows the chamfer of a single edge, using an equal distance of 0.4 inches:
Two-distance Chamfer
This shows the chamfer of a single edge, using one distance of 0.4 inches and another distance of 0.8 inches:
Distance-and-angle Chamfer

Before:

After:

In this example, the distance is measured along the bottom face from the selected edge, and the angle is measured starting from the bottom face. If you use the Directional arrows and flip the distance measurement, you get the distance and angle measured with respect to the vertical face:
Cylindrical Chamfer

This example shows a chamfer on a cylindrical edge:

After:
Chamfer: Android

Break sharp edges with a bevel. Define by the distance to break from the edge and by the angle made with the surface.

Steps

1. Tap Chamfer tool.
2. Select entities to chamfer (any edges or faces of a part).

3. Specify chamfer type:
   - **Equal distance** - to a specified distance in equal directions.
   - **Two distances** - to two individually specified distances.
   - **Distance and angle** - to a specified distance and angle.

4. Specify Distance (the width of the chamfer).

5. Optionally, toggle to use **Tangent propagation** to automatically extend the fillet to tangent edges.

6. Tap the checkmark.

### Equal-distance Chamfer

This shows the chamfer of a single edge, using an equal distance of 0.4 inches:
Two-distance Chamfer

This shows the chamfer of a single edge, using one distance of 0.4 inches and another distance of 0.8 inches:
Distance-and-angle Chamfer

Before:

After:

Chamfer 1

Entities to chamfer
1 entity selected

Chamfer type
Distance and angle

Distance
35.0 mm

Opposite direction

Angle
25.0 deg

Tangent propagation

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In this example, the distance is measured along the bottom face from the selected edge, and the angle is measured starting from the bottom face. If you use the Directional arrows and flip the distance measurement, you get the distance and angle measured with respect to the vertical face:

**Cylindrical Chamfer**

This example shows a chamfer on a cylindrical edge:
Draft

Apply a taper to one or more selected faces, or a parting line, in order to facilitate pulling a part from a mold.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Draft: Desktop**

In the Feature toolbar:

Apply a taper to one or more selected faces, or a parting line, in order to facilitate pulling a part from a mold. For information on analyzing a draft, see "Draft analysis" on page 198.

**Steps for neutral plane**

1. Click and select Neutral plane.

2. With focus on the *Neutral plane* field in the dialog, click on the entity to act as the neutral plane. (You can select the face of the part or a mate connector to act as a neutral plane.)

3. Click in the *Entities to draft* field, then select (all of) the faces to which to apply the draft.

4. Specify the degree of draft in the numeric field.
5. Indicate whether to apply the draft along tangent propagation; this applies the draft to all tangent faces.

Note that: Tangent propagation selects only faces that are steeper than the draft angle. In all cases, fillets that are not steep will be reapplied.

6. Optionally, indicate to Reapply fillets: steep fillet faces are treated as draft faces, not fillet faces. This converts those fillet faces to cones. Frequently, with large draft angles, this produces undesirable geometry. Leaving Reapply fillets unchecked, steep fillet faces are treated as fillets and reblended. This results in cylindrical faces instead of cones, and more often produces a desirable result.

7. Optionally, use the slider to visualize the difference between before the draft is applied and after.

8. Click ✅.

**Steps for parting line**

Drafting along a parting line requires an existing parting line along a face (or faces) before beginning the draft operation. Create a parting line using the Split tool and split a face (or faces), not a part.
1. Click and select Parting line.

2. With focus on the Pull direction field in the dialog, click on the face or Mate connector (inferred or existing) of the part to act as a neutral plane, along the primary axis.

3. Click in the Parting edges field, then select the line you created as the parting line.
   Check Parting line propagation to extend the draft along the entire parting line to tangential faces only. Multi-select is also available for selecting multiple line segments, tangential or not.

4. Select the sides of the parting to which to apply the draft: One sided, Symmetric, or Two sided. (Use the Switch face arrow to flip from one face to the opposing face.)

5. Specify the degree of draft in the numeric field.

6. Optionally, indicate to Reapply fillets: steep fillet faces are treated as draft faces, not fillet faces. This generates cones and preserves the parting line edges. Frequently, with large draft angles, this produces undesirable geometry. Leaving Reapply fillets unchecked, steep fillet faces are treated as fillets and reblended. This results in cylindrical faces instead of cones, modifies the parting line edges and more often produces a desirable result.
7. Optionally, use the slider to visualize the difference between before the draft is applied and after.

8. Click ✔️.

Draft: iOS

Apply a taper to one or more selected faces, or a parting line, in order to facilitate pulling a part from a mold.

Steps

1. Tap Draft tool.
2. Select a Neutral plane using a face or a mate connector (along the primary axis).
3. Select Entities to draft.
4. Specify Draft angle.
5. Optionally, toggle to switch draft to the opposite direction.
6. Optionally, toggle to use tangent propagation.
7. Optionally, toggle to reapply the fillet.

Note that:

- Tangent propagation selects only faces that are steeper than the draft angle.
- With Reapply fillets toggled off: steep fillet faces are treated as draft faces, not fillet faces. This generates cones and preserves the parting line edges. Frequently, with large draft angles, this produces undesirable geometry.
- In all cases, fillets that are not steep are reapplied.
8. Tap the checkmark.

**Draft: Android**

Apply a taper to one or more selected faces, or a parting line, in order to facilitate pulling a part from a mold.
Steps

1. Tap Draft tool.

2. Select a Neutral plane using a face or a mate connector (along the primary axis).
3. Select Entities to draft.
4. Specify Draft angle.
5. Optionally, toggle to switch draft to the opposite direction.
6. Optionally, toggle to use tangent propagation.
7. Optionally, toggle to reapply the fillet.
   
   Note that:
   
   - Tangent propagation selects only faces that are steeper than the draft angle.
   - With Reapply fillets toggled off: steep fillet faces are treated as draft faces, not fillet faces. This generates cones and preserves the parting line edges. Frequently, with large draft angles, this produces undesirable geometry.
   - In all cases, fillets that are not steep are reapplied.

8. Tap the checkmark.
Rib

Create ribs in parts at multiple locations based on a sketch.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Rib: Desktop**

In the Feature toolbar:

Create ribs in parts at multiple locations based on a sketch.

**Steps**

1. Click 📋.
2. Select the sketch curves from which to create ribs.

3. Select the parts to incorporate the ribs.

4. Specify the desired thickness of the rib.
5. Indicate how to extend the rib: normal (perpendicular) to the rib sketch plane, or parallel to the sketch plane. Use the directional arrows to flip the direction, if necessary.

The rib must be entirely within the bounds of the part or the operation will fail.

6. In the case of sketch curves that do not intersect with the part, select Extend profiles to part to extend the sketch curve to the part. Lines are extended, arcs are extended by straight lines from the ends of the arc:
7. Use Merge ribs to add the ribs to the existing part. Uncheck this box to create individual new parts of the ribs.

8. Click ✅.

**Normal to sketch plane**

Extend the rib normal to the sketch plane:
Parallel to sketch plane

Extend the rib parallel to the sketch plane:

Extend profiles to part

Extends the sketch profile to the part edge:
Merge ribs

Merges the ribs with the part and with other intersecting:
Rib: iOS

Create ribs in parts at multiple locations based on a sketch.

Steps

1. Tap Rib tool
2. Select the sketch curves from which to create ribs.
3. Select the parts to incorporate ribs.
4. Specify the desired thickness of the rib.
5. Indicate how to extend the rib:
   - **Normal to sketch plane** - perpendicular to the sketch plane
   - **Parallel to sketch plane** - parallel to the sketch plane
6. Optionally toggle **Opposite direction** to change the direction of the rib.
7. Optionally toggle **Extend profiles to part** - in the case of sketch curves that do not intersect with the part, this will extend the sketch curves to the part. Lines are extended, arcs are extended by straight lines from the ends of the arc.
8. Optionally toggle **Merge ribs** to add the ribs to the existing part. Leave Merge ribs off to create individual new parts of the ribs.

**Example**

1. Tap Rib tool.

2. Select the sketch curves from which to create ribs.
3. Select the parts to incorporate ribs.

4. Specify the desired thickness of the rib.

   The rib must be entirely within the bounds of the part or the operation will fail.

5. Indicate how to extend the rib: normal (perpendicular) to the rib sketch plane, or parallel to the sketch plane. Use the **Opposite direction** toggle, if necessary.

6. In the case of sketch curves that do not intersect with the part, select **Extend profiles to part** to extend the sketch curve to the part. Lines are extended, arcs are extended by straight lines from the ends of the arc:
7. Toggle on **Merge ribs** to add the ribs to the existing part.

8. Tap checkmark.

**Rib: Android**

Create ribs in parts at multiple locations based on a sketch.
Steps

1. Tap Rib tool.
2. Select the sketch curves from which to create ribs.

3. Select the parts to incorporate ribs.

<table>
<thead>
<tr>
<th>Rib 1</th>
<th>✓</th>
<th>✗</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select sketch profiles to create the ribs.</td>
<td>sketch profiles</td>
<td>No entities selected</td>
</tr>
<tr>
<td>Sketch profiles</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Parts</td>
<td>No entities selected</td>
<td></td>
</tr>
<tr>
<td>Thickness</td>
<td>0.1 in</td>
<td></td>
</tr>
<tr>
<td>Rib extrusion direction</td>
<td>Parallel to sketch plane</td>
<td></td>
</tr>
<tr>
<td>Opposite direction</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Extend profiles to part</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Merge ribs</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
4. Specify the desired thickness of the rib.

5. Indicate how to extend the rib:
   - **Normal to sketch plane** - perpendicular to the sketch plane
   - **Parallel to sketch plane** - parallel to the sketch plane

6. Optionally toggle **Opposite direction** to change the direction of the rib.

7. Optionally toggle **Extend profiles to part** - in the case of sketch curves that do not intersect with the part, this will extend the sketch curves to the part. Lines are extended, arcs are extended by straight lines from the ends of the arc.

8. Optionally toggle **Merge ribs** to add the ribs to the existing part. Leave Merge ribs off to create individual new parts of the ribs.

**Example**

1. Tap Rib tool.

2. Select the sketch curves from which to create ribs.
3. Select the parts to incorporate ribs.

4. Specify the desired thickness of the rib.

   The rib must be entirely within the bounds of the part or the operation will fail.

5. Indicate how to extend the rib: normal (perpendicular) to the rib sketch plane, or parallel to the sketch plane. Use the **Opposite direction** toggle, if necessary.

6. In the case of sketch curves that do not intersect with the part, select **Extend profiles to part** to extend the sketch curve to the part. Lines are extended, arcs are extended by straight lines from the ends of the arc:
7. Toggle on **Merge ribs** to add the ribs to the existing part.

8. Tap checkmark.
Remove material from a part to produce a cavity of constant wall thickness with the option to remove zero faces (hollow) to many faces of the part (shell).

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Shell: Desktop**

In the Feature toolbar:

Remove material from a part to produce a cavity of constant wall thickness with the option to remove zero faces (hollow) to many faces of the part (shell).

**Steps**

1. Click 📝.

2. With focus on the Faces to shell field of the dialog, click on the part face or faces to remove. (The rest of the part will be hollowed out, forming a shell.)
   
   Optionally, check the Hollow box to shell (hollow) the part without removing any faces:

3. In the numeric value field, enter a value for the thickness of the part wall.
   
   A hollowed-out part will show edges when displayed with Hidden edges visible:
4. Click ✓.

**Tips**

The direction arrows next to the numeric field in the dialog allow you to select whether to create the shell wall by using the part face as the inside of the shell or the outside of the shell.

**Shell: iOS**

Remove material from a part to produce a cavity of constant wall thickness with the option to remove zero faces (hollow) to many faces of the part (shell).

**Steps**

1. Tap Shell tool.
2. Optionally, toggle **Hollow** to shell (hollow) the part without removing any faces.

3. Select Faces to remove. The part face or face you select is removed and the rest of the part is hollowed out, forming a shell.

4. Specify Shell thickness. The thickness of the part wall.

5. Optionally, toggle to switch to the Opposite direction.

6. Tap the checkmark.
Example

![Example Image](image)

**Shell: Android**

Remove material from a part to produce a cavity of constant wall thickness with the option to remove zero faces (hollow) to many faces of the part (shell).
Steps

1. Tap Shell tool.

2. Optionally, toggle **Hollow** to shell (hollow) the part without removing any faces.

3. Select Faces to remove. The part face or face you select is removed and the rest of the part is hollowed out, forming a shell.

4. Specify Shell thickness. The thickness of the part wall.

5. Optionally, toggle to switch to the Opposite direction.

6. Tap the checkmark.
Example
Hole

Create simple, countersink, and counterbore holes at sketch points or circle centers, using ANSI or ISO standards or custom specifications.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Hole: Desktop

In the Feature toolbar:

Create simple, countersink, and counterbore holes at sketch points or circle centers, using ANSI or ISO standards or custom specifications.

Steps

1. Create a sketch with sketch points.

2. Select the points where you want to create holes, then click . You can click the in order to select inferred Mate connectors as points instead of sketch points.
Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:

Note that the selections for the last hole created are presented as defaults when you open the Hole dialog.
You can also click on a sketch and it will automatically select all the points on the sketch, not including those used as *construction geometry*.

3. Select a hole style:
   a. Simple (a uniform-diameter drilled hole)
   b. Counterbore
   c. Countersink

4. Select a termination condition:
   - **Through** - Completely through the selected part
   - **Blind** - To a specified depth in the selected part
   - **Blind in last** - To a specified depth in the last/bottom of multiple selected parts; this places the tapped portion of the hole in the last part and clearance in all other parts

5. Select a standard, or choose Custom for non-standard specifications.
   
   If you select a standard and then edit the default, the standard specification automatically changes to Custom.

6. Currently offered standards include:
   - Custom
   - ANSI
   - ISO
• DIN
• PEM®

After choosing a standard, select the specifications that suit your needs:

• **Hole type** - Clearance, Tapped, Drilled
• **Size** - Select from the list of sizes
• **Fit** - Free, Close, Normal, or Loose
• **Drill size** - Select a drill size for Drilled type holes
• **Pilot drill diameter** - For holes that require pilot holes
• **% diametric engagement** - Select a value for Tapped type holes to define thread engagement by varying the pilot hole size. A higher percentage decreases the pilot hole size and increases thread engagement. A lower percentage increases the pilot hole size and decreases thread engagement.
• **Threads/inch and percent thread engagement** - The number of threads per inch and the percent of thread engagement
• **Tapped depth** - The full thread depth of a tapped hole, in document units (or specify other units)
• **Tap clearance** - The number of threads between the bottom of the tapped hole and the bottom thread

Percent thread engagement refers to how much of the thread is available due to the change in the diameter of the tap hole.

When choosing **Custom**, enter:

• **Diameter** - The diameter of the hole itself
• **Counterbore diameter and Counterbore depth** - The diameter of the counterbore and the depth of the counterbore
• **Countersink diameter, Countersink angle** - The diameter of the countersink and the angle of the countersink
• **Depth** - The depth of the hole itself, inclusive of the counterbore or countersink depth
7. Indicate **Start from sketch plane** to start the hole from the sketch plane (this allows you to have the depth end at the same location on the part when the holes start at different heights):

For example, all the sketch points for the holes are on one plane:

![Sketch points on one plane](image)

For a counterbore hole with a depth of 1 inch, when the holes do not start from the sketch plane, the shorter hole contains only the counterbore while the deepest hole contains the counterbore and bolt shaft:

![Counterbore hole](image)

But when the holes are started from the sketch plane the shaft is placed first and the counterbore is shortened:

![Shaft placed first](image)

The same scenario holds true for countersink holes. Simple holes are just adjusted for depth.

8. When creating tapped holes, you can check 'Tap through all' to create thread for the full length of the hole.

9. With focus in the Sketch points to place holes field in the dialog, select points in the sketch (any points including, corners of geometry, line ends, spline points,
circle centers, etc) where the hole centers are to be placed. (Box selection of multiple points is also an option.)

10. With focus in the Merge scope field, select the part(s) to contain the holes.

11. Click ✓.

**Simple**

A uniform-diameter drilled hole:
Counterbore

Countersink

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Through sheet metal

Holes can be created on active sheet metal features. If the hole has a counterbore or a countersink, the counterbore or countersink diameter of the holes will be put into the sheet metal normal to the surface (note the normal cut for the countersink, below).

Termination condition examples

Through

Completely through the selected part or parts.

A simple hole with Through set as the termination condition and all four plates selected:
Blind

To a specified depth in the selected part.

A simple hole with Blind selected as the termination condition and a depth set to 10 inches, passing through just the top two plates:
Blind in last

To a specified depth in the last/bottom of multiple selected parts; placing the tapped portion of the hole in the last part and clearance in all other parts.

A simple hole with Blind in last selected as the termination condition and all four plates selected, passing through all four parts with the tapped portion of the hole in the last (or bottom) part:
**Tips**

- All material is cleared between the point on the sketch plane and the hole:
- If you change the sketch, the hole feature recomputes.
- This feature includes logic to determine a good starting depth for the hole, useful for curved or irregular surfaces. This is the default. If you want the hole start to be located at the sketch plane, check this box (effectively turning off the starting depth logic). Checking this box also allows you to create overlapping holes.
- In the case of collision or overlap of hole features or the hole does not lie completely on the target part, the hole will be drilled at 0 depth relative to the sketch plane.
- If a standard does not provide a counterbore, countersink or tap diameter, these values will be reset if they are out of range.

**Hole: iOS**

Create simple, countersink, and counterbore holes at sketch points or circle centers, using ANSI or ISO standards or custom specifications.
Steps

1. Tap the Hole tool.

2. Select a hole style:
   - **Simple** - A uniform-diameter drilled hole
   - **Counterbore**
   - **Countersink**
3. Select an end (termination) condition:
   - **Through** - Completely through the selected part
   - **Blind** - To a specified depth in the selected part
   - **Blind in last** - To a specified depth in the last/bottom of multiple selected parts; this places the tapped portion of the hole in the last part and clearance in all other parts

4. Select a standard, or choose **Custom** for non-standard specifications.

   If you select a standard and then edit the default, the standard specification automatically changes to Custom.

   When choosing a standard, select the appropriate specifications:
   - **Hole type** - Clearance, Tapped, Drilled
   - **Size** - From the list of standard sizes
   - **Fit** - Close, Free/Standard
   - **Drill size** - where appropriate
   - **Pilot drill diameter** - where appropriate
   - **% diametric engagement** - where appropriate
   - **Threads/inch and percent thread engagement** - where appropriate (Percent thread engagement refers to how much of the thread is available due to the change in the diameter of the tap hole.)
   - **Tapped depth** - the full thread depth of a tapped hole, in document units (or specify other units)
   - **Tap clearance** - The number of threads between the bottom of the tapped hole and the bottom thread

   When choosing Custom, enter:
   - Hole diameter
   - Counterbore/Countersink diameter - where appropriate
• Counterbore depth, Countersink angle - where appropriate
• Hole depth

5. Optionally, toggle on Start from sketch plane to start the hole from the sketch plane (this may change where the counterbore/countersink, clearance, and tapped portions of the screw are located).

6. In the Sketch points to place holes field, select points in the sketch at which to place holes (box select is available).

7. In the Merge scope field, select the part(s) to contain the holes.

8. Tap checkmark.

**Simple**

A uniform-diameter drilled hole

---

**Counterbore**
Countersink
Termination condition examples

Through

Completely through the selected part or parts.

This shows a simple hole with Through set as the termination condition and all four plates selected. Thus, the hole is through all four plates.
Blind

To a specified depth in the selected part.

This shows a simple hole with **Blind** selected as the termination condition and a depth set to 10 inches. Thus, the hole is created 10 inches through the plates, and only passes through the top two plates.
Blind in last

To a specified depth in the last/bottom of multiple selected parts; this places the tapped portion of the hole in the last part and clearance in all other parts.

This shows a simple hole with **Blind in last** selected as the termination condition and all four plates selected. Thus, the hole passes through all four parts, but the tapped portion of the hole is in the last (or bottom) part.
Tips

- All material is cleared between the point on the sketch plane and the hole.
- If you change the sketch, the hole feature recomputes.
- This feature includes logic to determine a good starting depth for the hole, useful for curved or irregular surfaces. This is the default. If you want the hole start to be located at the sketch plane, toggle on Start from sketch plane (effectively turning off the starting depth logic). Toggling this option on also allows you to create overlapping holes.
- In the case of collision or overlap of hole features or the hole does not lie completely on the target part, the hole is drilled at 0 depth relative to the sketch plane.
- If a standard does not provide a counterbore, countersink, or tap diameter, these values are reset if they are out of range.
• With **Blind in last** selected, the depth entered is the depth of the hole through the last selected part. For example, if you specify a depth of 1 inch and select three parts, the hole will go through the first two parts completely and 1 inch into the third part.

**Hole: Android**

Create simple, countersink, and counterbore holes at sketch points or circle centers, using ANSI or ISO standards or custom specifications.

**Steps**

1. Tap the Hole tool.
Hole 1

No hole points selected.

**Simple**  Counterbore  Countersink

**Through**  Blind  Blind in last

Opposite direction

Standard
Custom

Diameter
0.25 in

Sketch points to place holes
No entities selected

Merge scope
No entities selected
2. Select a hole style:
   - **Simple** - A uniform-diameter drilled hole
   - **Counterbore**
   - **Countersink**

3. Select an end (termination) condition:
   - **Through** - Completely through the selected part
   - **Blind** - To a specified depth in the selected part
   - **Blind in last** - To a specified depth in the last/bottom of multiple selected parts; this places the tapped portion of the hole in the last part and clearance in all other parts

4. Select a standard, or choose **Custom** for non-standard specifications.
   
   If you select a standard and then edit the default, the standard specification automatically changes to Custom.

   When choosing a standard, select the appropriate specifications:
   - **Hole type** - Clearance, Tapped, Drilled
   - **Size** - From the list of standard sizes
   - **Fit** - Close, Free/Standard
   - **Drill size** - where appropriate
   - **Pilot drill diameter** - where appropriate
   - **% diametric engagement** - where appropriate
   - **Threads/inch and percent thread engagement** - where appropriate (Percent thread engagement refers to how much of the thread is available due to the change in the diameter of the tap hole.)
   - **Tapped depth** - the full thread depth of a tapped hole, in document units (or specify other units)
   - **Tap clearance** - The number of threads between the bottom of the tapped hole and the bottom thread

   When choosing Custom, enter:
- Hole diameter
- Counterbore/Countersink diameter - where appropriate
- Counterbore depth, Countersink angle - where appropriate
- Hole depth

5. Optionally, toggle on *Start from sketch plane* to start the hole from the sketch plane (this may change where the counterbore/countersink, clearance, and tapped portions of the screw are located).

6. In the *Sketch points to place holes* field, select points in the sketch at which to place holes (box select is available).

7. In the *Merge scope* field, select the part(s) to contain the holes.

8. Tap checkmark.

**Simple**

A uniform-diameter drilled hole
Counterbore

Countersink
Termination condition examples

Through

Completely through the selected part or parts.

This shows a simple hole with **Through** set as the termination condition and all four plates selected. Thus, the hole is through all four plates.
Blind

To a specified depth in the selected part.

This shows a simple hole with **Blind** selected as the termination condition and a depth set to 10 inches. Thus, the hole is created 10 inches through the plates, and only passes through the top two plates.
Blind in last

To a specified depth in the last/bottom of multiple selected parts; this places the tapped portion of the hole in the last part and clearance in all other parts.

This shows a simple hole with **Blind in last** selected as the termination condition and all four plates selected. Thus, the hole passes through all four parts, but the tapped portion of the hole is in the last (or bottom) part.
Tips

- All material is cleared between the point on the sketch plane and the hole.
- If you change the sketch, the hole feature recomputes.
- This feature includes logic to determine a good starting depth for the hole, useful for curved or irregular surfaces. This is the default. If you want the hole start to be located at the sketch plane, toggle on *Start from sketch plane* (effectively turning off the starting depth logic). Toggling this option on also allows you to create overlapping holes.
- In the case of collision or overlap of hole features or the hole does not lie completely on the target part, the hole is drilled at 0 depth relative to the sketch plane.
- If a standard does not provide a counterbore, countersink, or tap diameter, these values are reset if they are out of range.
With **Blind in last** selected, the depth entered is the depth of the hole through the last selected part. For example, if you specify a depth of 1 inch and select three parts, the hole will go through the first two parts completely and 1 inch into the third part.

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**Linear Pattern**

Replicate selected parts, faces, or features and arrange them in a row or grid pattern. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating circular patterns, see "Circular Pattern" on page 986. Linear pattern may also be used during an active sheet metal operation.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Linear Pattern: Desktop**

In the Feature toolbar:

Replicate selected parts, faces, or features and arrange them in a row or grid pattern. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating circular patterns, see "Circular Pattern" on page 986. Linear pattern may also be used during an active sheet metal operation.

**Steps**

1. Click 📁.
2. Select a Result operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from existing material.
   - **Intersect** - Leave material only where geometry overlaps.

3. Select the pattern type:
   - **Part** - To pattern an individual part
   - **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc).
   - **Face** - To pattern a specific face on a specific part

4. With focus on the Entities to pattern field, select entities to replicate into a pattern.
   
   When selecting Faces to pattern, the "Create Selection" on page 229 can be useful to select related faces.

5. Set focus in the Direction field, and then select an edge or face of the part or a Mate connector (inferred or existing) along which to place the replicated pattern entities.
6. Enter the distance between each pattern entity, and then the number of repetitions, or Instance count (the minimum number of instances you are able to use is 1). Select a direction for the pattern in the workspace (shown below as the highlighted edge).

7. Use Centered to make the seed instance/face/feature as the center of the pattern. In this case, the instance count is N instances from the seed (inclusive of the seed) in one direction and N instances from the seed (inclusive of the seed) in the other direction. With N being the number you enter in the Instance count field in the dialog:

8. Select second direction to extend the pattern in a second direction; select the second direction.

9. Select Apply per instance to regenerate the feature for each instance (specified in Instance count), in lieu of simply recreating the initial pattern instance.

10. Click ✓

**Linear part pattern**

Pattern an individual part:
Pattern a sheet metal part; in this case the blue outer wall was selected:

**Linear feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc) without Apply per instance selected (a faster, more lightweight feature pattern). This is not available for sheet metal.

In the following example, the initial extrude was Up to next (bringing the extrude up to the face of the surface). *Without* Apply per instance selected, the features patterned (extrude and fillet) are patterned from the initial features and not regenerated (so the Up to next does not update for each instance of the pattern):
Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc) *with* Apply per instance selected. This is available for sheet metal.

In the following example, the initial extrude was Up to next (bringing the extrude up to the face of the surface). With Apply per instance selected, the features patterned (extrude and fillet) are regenerated for each instance of the pattern so Up to next is applied to each instance of the pattern.

**Linear face pattern**

Pattern a specific face on a specific part:
Linear pattern: **New/Add (new material)**

**New** - Create new material that results in a new part or sheet metal:

**Add** - Create material and add to the existing material:
**Linear pattern: Remove (cut material)**

Take material away; select the part to pattern, and then Remove:

**Linear pattern: Intersect**

Leave material only where geometry overlaps; select the part to pattern, and then Intersect:
Tips

- When selecting a face, edge, or Mate connector (inferred or existing) to set the Direction, use the Directional arrows to flip the result if necessary.

- When you select a face, edge, or Mate connector (inferred or existing) for the Direction, you are using the direction that is 'normal to' the face, edge, or mate connector.

- When patterning a feature, you are able to select anything in the feature list, in any order. Regardless of the order selected, the features are applied in the order listed in the Feature list.

- If you select a pattern in the Feature list, you will pattern that pattern, but not the seed. In order to get the seed included, select it as well.

- When patterning a boolean feature (Boolean, Split, etc), you must also select the features the boolean was applied to.

- When creating Feature patterns, all aspects of a feature are applied; for example, the end conditions in an extrude feature. (By contrast, Face patterns do not recognize these types of modifiers.)

Linear Pattern: iOS

Replicate selected parts, faces, or features and arrange them in a row or grid pattern. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating circular patterns, see "Circular Pattern" on page 986. Linear pattern may also be used during an active sheet metal operation.

Steps

1. Tap Linear pattern tool.
2. Select the Pattern type:

- **Part** - To pattern an individual part
- **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)
- **Face** - To pattern a specific face on a specific part
3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select entities (parts, features, or faces) to pattern.

5. Specify direction of the pattern using the number pad or drag manipulator.

6. Specify the instance count.

7. Optionally, toggle to switch to the opposite direction.

8. Optionally, toggle **Centered** to make the seed instance/face/feature as the center of the pattern.

9. Optionally, toggle to add a second direction. If so, be sure to complete the steps for the second direction.

10. Tap the checkmark.

**Linear part pattern**

Pattern an individual part.

One cylindrical part is patterned 5 times using Linear part pattern to create new material.
Linear feature pattern

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)

An extrude remove feature is patterned 10 times using Linear feature pattern to remove material from the existing part.
Linear face pattern
Pattern a specific face on a specific part.

The cylindrical face of the part is patterned 5 times using Linear face pattern to add material to the existing part.

**Linear Pattern: New/Add (new material)**

**New** - Create new material that results in a new part.
Add - Create material and add to the existing material.

Linear Pattern: Remove (cut material)
Take material away.
Linear Pattern: Intersect
Leave material only where intersections exist.

Linear Pattern: Android
Replicate selected parts, faces, or features and arrange them in a row or grid pattern. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating circular patterns, see "Circular Pattern" on page 986. Linear pattern may also be used during an active sheet metal operation.

Steps
1. Tap Linear pattern tool.

![Linear pattern tool](image)

**Linear pattern 1**

- **Select parts to pattern.**
- **Pattern type**
  - Part pattern

**New** Add Remove Intersect

**Entities to pattern**
- **No entities selected**

**Direction**
- **No entities selected**

**Distance**
- 1.0 in

**Instance count**
- 2

- **Opposite direction**
- **Centered**
- **Second direction**
2. Select the Pattern type:
   - **Part** - To pattern an individual part
   - **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)
   - **Face** - To pattern a specific face on a specific part

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select entities (parts, features, or faces) to pattern.

5. Specify direction of the pattern using the number pad or drag manipulator.

6. Specify the instance count.

7. Optionally, toggle to switch to the opposite direction.

8. Optionally, toggle **Centered** to make the seed instance/facefeature as the center of the pattern.

9. Optionally, toggle to add a second direction. If so, be sure to complete the steps for the second direction.

10. Tap the checkmark.

**Linear part pattern**

Pattern an individual part.

One cylindrical part is patterned 5 times using Linear part pattern to create new material.
Linear feature pattern

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)

An extrude remove feature is patterned 10 times using Linear feature pattern to remove material from the existing part.
Linear face pattern
Pattern a specific face on a specific part.

The cylindrical face of the part is patterned 5 times using Linear face pattern to add material to the existing part.

**Linear Pattern: New/Add (new material)**

**New** - Create new material that results in a new part.
Add - Create material and add to the existing material.

Linear Pattern: Remove (cut material)
Take material away.
Linear Pattern: Intersect

Leave material only where intersections exist.

Circular Pattern

Replicate selected parts, faces, or features about an axis or Mate connector (inferred or existing). Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating linear patterns,
Circular pattern may also be used during an active sheet metal operation.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Circular Pattern: Desktop**

In the Feature toolbar:

Replicate selected parts, faces, or features about an axis or Mate connector (inferred or existing). Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating linear patterns, see "Linear Pattern" on page 965. Circular pattern may also be used during an active sheet metal operation.

**Steps**

1. Click 🍃:

   ![Circular pattern 1](image)
2. Select a Result operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from existing material.
   - **Intersect** - Leave material only where geometry overlaps.

3. Select the pattern type: Part, Feature, or Face:
   - **Part** - To pattern an individual part
   - **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc).
   - **Face** - To pattern a specific face on a specific part

4. With focus on the Entities to pattern field, select entities to replicate into a pattern.
   When selecting Faces to pattern, the "Create Selection" on page 229 can be useful to select related faces.

5. Set focus in the Axis of pattern field, and then select an edge, face, or conic or cylindrical face of the part, linear sketch entity or Mate connector (inferred or existing) about which to place the replicated pattern parts. (Click the in order to select inferred Mate connectors.)

   Once a Mate connector is selected, click the Mate connector icon in the dialog field (outlined in blue below) to open a dialog with which to edit the Mate connector:
6. Enter the distance between each pattern part, and then the number of repetitions, or Instance count (the minimum number of instances you are able to use is 1).

7. Use Centered to make the seed instance/face/feature as the center of the pattern.

8. The Equal spacing box allows you to place the pattern parts within the specified degrees.

9. Select Apply per instance to regenerate the feature for each instance (specified in Instance count), in lieu of simply recreating the initial pattern instance.

**Circular part pattern**

Pattern an individual part

One section was created and then patterned as Add to create this part:
For sheet metal, this blade was selected as the part and an edge was selected as the pattern axis.

**Circular feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc) without Apply per instance selected (a faster, more lightweight feature pattern). This is not available for sheet metal.

In the following example, the initial extrude was Up to next (bringing the extrude up to the face of the surface). *Without* Apply per instance selected, the features patterned (extrude and fillet) are patterned from the initial features and not regenerated (so the Up to next does not update for each instance of the pattern):

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc) *with* Apply per instance selected. This is available for sheet metal.

In the following example, the initial extrude was Up to next (bringing the extrude up to the face of the surface). *With* Apply per instance selected, the features patterned
(extrude and fillet) are regenerated for each instance of the pattern so Up to next is applied to each instance of the pattern.

Circular face pattern

Pattern a specific face on specific part

The selected faces are the cylindrical face and its top face (for two cylinders), then patterned at 90 degree angles, 4 instances.

The selected face on this sheet metal piece is the flange. The axis or Mate connector (inferred or existing) is the highlighted circle.
Circular pattern: New/Add (new material)

New - Create new material that results in a new part:

Add - Create material and add to the existing material (in this instance, Merge with all was selected):

Circular pattern: Remove (cut material)

Take material away; select the part to pattern and then Remove:
Circular pattern: Intersect

Select the part to pattern, and then Intersect:

Tips

- When selecting a face or edge to set the Direction, you can use the Directional arrows 🔄 to flip the result if necessary.

- When you select a face for the Direction, you are using the direction that is 'normal to' the face.

- When patterning a feature, you can select anything in the feature list, in any order. Regardless of the order selected, the features are applied in the order listed in the Feature list.

- If you select a pattern in the Feature list, you will pattern that pattern, but not the seed. In order to get the seed included, select it as well.

- When patterning a boolean feature (Boolean, Split, etc), you must also select the features the boolean was applied to.
When creating Feature patterns, all aspects of a feature are applied; for example, the end conditions in an extrude feature. (By contrast, Face patterns do not recognize these types of modifiers.)

**Circular Pattern: iOS**

Replicate selected parts, faces, or features about an axis or Mate connector (inferred or existing). Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating linear patterns, see "Linear Pattern" on page 965. Circular pattern may also be used during an active sheet metal operation.

**Steps**

1. Tap Circular pattern tool.
2. Specify Pattern type:

- **Part** - To pattern an individual part
- **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)
- **Face** - To pattern a specific face on a specific part
3. Select a Result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create new material and add to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select entities (parts, features, or faces) to pattern.

5. Select the axis of the pattern.

6. Specify the angle. This angle will be the distance between each part, unless you chose to use Equal spacing (see step 9 for more info).

7. Specify the instance count.

8. Optionally, toggle to switch to the opposite direction.

9. Optionally, toggle to use equal spacing. Equal spacing will place the patterned entities within the specified angle/degrees.

10. Optionally, toggle **Centered** to make the seed instance/face/feature as the center of the pattern.

11. Tap checkmark.

**Circular part pattern**

Pattern an individual part.

A part is patterned 4 times using Circular part pattern around the axis of a sketch circle to add material to the existing part.
**Circular feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)

An extrude feature is patterned 4 times using Circular feature pattern around the axis of a sketch circle to add material to the existing part.
Circular face pattern

Pattern a specific face on a specific part.

The highlighted cylindrical faces of the part are patterned 4 times using Circular face pattern to add material to the existing part.
Circular Pattern: New/Add (new material)

New - Create new material that results in a new part.
Add - Create material and add to the existing material.
Circular Pattern: Remove (cut material)

Take material away.
Circular Pattern: Intersect

Leave material only where intersections exist.

Circular Pattern: Android

Replicate selected parts, faces, or features about an axis or Mate connector (inferred or existing). Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. For information on creating linear patterns, see "Linear Pattern" on page 965. Circular pattern may also be used during an active sheet metal operation.

Steps

1. Tap Circular pattern tool.
2. Specify Pattern type:

- **Part** - To pattern an individual part
- **Feature** - To pattern a specific feature (or features) listed in the Feature list (an
extrude, fillet, sweep, sketch, etc.)

- **Face** - To pattern a specific face on a specific part

3. Select a Result body operation type:

- **New** - Create new material that results in a new part.
- **Add** - Create new material and add to the existing material.
- **Remove** - Take material away from a part.
- **Intersect** - Leave material only where intersections exist.

4. Select entities (parts, features, or faces) to pattern.

5. Select the axis of the pattern.

6. Specify the angle. This angle will be the distance between each part, unless you chose to use Equal spacing (see step 9 for more info).

7. Specify the instance count.

8. Optionally, toggle to switch to the opposite direction.

9. Optionally, toggle to use equal spacing. Equal spacing will place the patterned entities within the specified angle/degrees.

10. Optionally, toggle **Centered** to make the seed instance-face-feature as the center of the pattern.

11. Tap checkmark.

**Circular part pattern**

Pattern an individual part.

A part is patterned 4 times using Circular part pattern around the axis of a sketch circle to add material to the existing part.
**Circular feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)

An extrude feature is patterned 4 times using Circular feature pattern around the axis of a sketch circle to add material to the existing part.
Circular face pattern

Pattern a specific face on a specific part.

The highlighted cylindrical faces of the part are patterned 4 times using Circular face pattern to add material to the existing part.
Circular Pattern: New/Add (new material)

**New** - Create new material that results in a new part.
Add - Create material and add to the existing material.
Circular Pattern: Remove (cut material)

Take material away.
**Circular Pattern: Intersect**
Leave material only where intersections exist.

**Curve Pattern**
Replicate selected parts, faces, or features along a sketch curve (or series of adjacent curves, edges on solid parts, and edges on wire parts) in the order of selection. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. Curve pattern may also be used during an active sheet metal operation.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Curve Pattern: Desktop**
In the Feature toolbar:
Replicate selected parts, faces, or features along a sketch curve (or series of adjacent curves, edges on solid parts, and edges on wire parts) in the order of selection. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. Curve pattern may also be used during an active sheet metal operation.

Steps

1. Click 🔄.

2. Select the pattern type:
   - **Part** - To pattern an individual part
   - **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.). Note that Feature does not work for sheet metal; see the Face pattern type for sheet metal modifications.
   - **Face** - To pattern a specific face on a specific part

3. Select a Result operation type:
   - **New** - Create new material that results in a new part
   - **Add** - Create new material and add to the existing material
   - **Remove** - Take material away from existing material
   - **Intersect** - Leave material only where geometry overlaps
4. With focus on the *Entities to pattern* field, select entities to replicate into a pattern.

When selecting Faces to pattern, the "Create Selection" on page 229 is useful to select related faces.

5. Set focus in the *Path to pattern along* field, and then select a sketch curve (or series of adjacent curves, edges on solid parts, and edges on wire parts) along which to place the replicated pattern entities.

6. Enter the number of instances you want the pattern to have (the minimum number of instances you are able to use is 1).

7. Use *Keep orientation* to preserve the original orientation of the part/face/feature being patterned.

8. Select Apply per instance to regenerate the feature for each instance (specified in Instance count), in lieu of simply recreating the initial pattern instance.

9. Click ✓.

You are unable to set a distance between entities in the curve pattern, but you are able individually delete any entities after the pattern is complete.

**Curve part pattern**

Pattern an individual part.

A part was patterned along two adjacent sketch curves 10 times, creating new material:
**Curve feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc) without Apply per instance selected (a faster, more lightweight feature pattern). This is not available for sheet metal.

In the following example, the initial extrude was Up to next (bringing the extrude up to the face of the surface). *Without* Apply per instance selected, the features patterned (extrude and fillet) are patterned from the initial features and not regenerated (so the Up to next does not update for each instance of the pattern):

![Diagram of curve feature pattern example](image)

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc) *with* Apply per instance selected. This is available for sheet metal.

In the following example, the initial extrude was Up to next (bringing the extrude up to the face of the surface). With Apply per instance selected, the features patterned (extrude and fillet) are regenerated for each instance of the pattern so Up to next is applied to each instance of the pattern:
Curve face pattern

Pattern a specific face on a specific part.

A face was patterned along one sketch curve 5 times to remove material from the existing material:

Curve pattern: New/Add (new material)

New - Create new material that results in a new part:
**Add** - Create material and add to the existing material (in this instance, Merge with all was selected):

**Curve pattern: Remove (cut material)**

Take material away; select the part to pattern and then Remove:
Curve pattern: Intersect

Leave material only where geometry overlaps.

Before:
After:
**Tips**

- When patterning a feature, you can select anything in the feature list, in any order. Regardless of the order selected, the features are applied in the order listed in the Feature list.

- If you select a pattern in the Feature list, you will pattern that pattern, but not the seed. In order to get the seed included, select it as well.

- When patterning a boolean feature (Boolean, Split, etc), you must also select the features the boolean was applied to.

- When creating Feature patterns, all aspects of a feature are applied; for example, the end conditions in an extrude feature. (By contrast, Face patterns do not recognize these types of modifiers.)
You are unable to set a distance between entities in the curve pattern, but you are able to individually delete any entities after the pattern is complete.

If you use more than one sketch curve (or edge on solid parts, or edge on wire parts) to direct your pattern and it does not result as expected, try selecting the sketch curves in a different order.

**Curve Pattern: iOS**

Replicate selected parts, faces, or features along a sketch curve (or series of adjacent curves, edges on solid parts, and edges on wire parts) in the order of selection. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. Curve pattern may also be used during an active sheet metal operation.

**Steps**

1. Tap Curve Pattern tool.
2. Select the pattern type:
   - **Part** - To pattern an individual part
   - **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)
   - **Face** - To pattern a specific face on a specific part

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part
   - **Add** - Create new material and add to the existing material
   - **Remove** - Take material away from existing material
   - **Intersect** - Leave material only where geometry overlaps
4. With focus on the *Entities to pattern* field, select entities to replicate into a pattern.

5. Set focus on the *Path to pattern along* field, and then select a sketch curve (or series of adjacent curves, edges on solid bodies, and edges on wire bodies) along which to place the replicated pattern entities.

6. Enter the number of instances you want the pattern to have.

7. Toggle on *Keep orientation* to preserve the original orientation of the part/-face/feature being patterned.

8. Tap the checkmark.

You are unable to set a distance between entities in the curve pattern, but you are able to individually delete any entities after the pattern is complete.

**Curve part pattern**

Pattern an individual part.

A part was patterned along two adjacent sketch curves 10 times, creating new material.

**Curve feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.).

An extrude feature was patterned along two adjacent sketch curves 15 times, adding to the existing material.
Curve face pattern

Pattern a specific face on a specific part.

A face was patterned along one sketch curve 5 times to remove material from the existing material.
Curve pattern: New/Add (new material)

**New** - Create new material that results in a new part.
Add - Create material and add to the existing material (in this instance, *Merge with all* was selected).

Curve pattern: Remove (cut material)

Take material away; select the part to pattern and then Remove.
Curve pattern: Intersect

Leave material only where geometry overlaps.

Before:
After:
Tips

- When patterning a feature, you are able to select anything in the feature list, in any order. Regardless of the order selected, the features are applied in the order listed in the Feature list.

- If you select a pattern in the Feature list, you will pattern that pattern, but not the seed. In order to get the seed included, select it as well.

- When patterning a boolean feature (Boolean, Split, etc.), you must also select the features the boolean was applied to.

- When creating Feature patterns, all aspects of a feature are applied; for example, the end conditions in an extrude feature. (By contrast, Face patterns do not recognize these types of modifiers.)
You are unable to set a distance between entities in the curve pattern, but you are able to individually delete any entities after the pattern is complete.

If you use more than one sketch curve (or edge on solid bodies, or edge on wire bodies) to direct your pattern and it does not result as expected, try selecting the sketch curves in a different order.

**Curve Pattern: Android**

Replicate selected parts, faces, or features along a sketch curve (or series of adjacent curves, edges on solid parts, and edges on wire parts) in the order of selection. Create new parts or modify existing parts by adding or removing material, or intersecting parts in its path. Curve pattern may also be used during an active sheet metal operation.

**Steps**

1. Tap Curve Pattern tool.
2. Select the pattern type:
   - **Part** - To pattern an individual part
   - **Feature** - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)
   - **Face** - To pattern a specific face on a specific part

3. Select a Result body operation type:
   - **New** - Create new material that results in a new part
   - **Add** - Create new material and add to the existing material
   - **Remove** - Take material away from existing material
   - **Intersect** - Leave material only where geometry overlaps
4. With focus on the *Entities to pattern* field, select entities to replicate into a pattern.

5. Set focus on the *Path to pattern along* field, and then select a sketch curve (or series of adjacent curves, edges on solid bodies, and edges on wire bodies) along which to place the replicated pattern entities.

6. Enter the number of instances you want the pattern to have.

7. Toggle on *Keep orientation* to preserve the original orientation of the part/-face/feature being patterned.

8. Tap the checkmark.

You are unable to set a distance between entities in the curve pattern, but you are able to individually delete any entities after the pattern is complete.

**Curve part pattern**

Pattern an individual part.

A part was patterned along two adjacent sketch curves 10 times, creating new material.

**Curve feature pattern**

Pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.).

An extrude feature was patterned along two adjacent sketch curves 15 times, adding to the existing material.
**Curve face pattern**

Pattern a specific face on a specific part.

A face was patterned along one sketch curve 5 times to remove material from the existing material.
**Curve pattern: New/Add (new material)**

**New** - Create new material that results in a new part.
**Add** - Create material and add to the existing material (in this instance, *Merge with all* was selected).

**Curve pattern: Remove (cut material)**

Take material away; select the part to pattern and then Remove.
Curve pattern: Intersect

Leave material only where geometry overlaps.

Before:
After:
Tips

- When patterning a feature, you are able to select anything in the feature list, in any order. Regardless of the order selected, the features are applied in the order listed in the Feature list.

- If you select a pattern in the Feature list, you will pattern that pattern, but not the seed. In order to get the seed included, select it as well.

- When patterning a boolean feature (Boolean, Split, etc.), you must also select the features the boolean was applied to.

- When creating Feature patterns, all aspects of a feature are applied; for example, the end conditions in an extrude feature. (By contrast, Face patterns do not recognize these types of modifiers.)
You are unable to set a distance between entities in the curve pattern, but you are able to individually delete any entities after the pattern is complete.

If you use more than one sketch curve (or edge on solid bodies, or edge on wire bodies) to direct your pattern and it does not result as expected, try selecting the sketch curves in a different order.

---

**Mirror**

Replicate one or more selected parts or surfaces about a specified plane or planar face. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Mirror may also be used during an active sheet metal operation.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Mirror: Desktop**

In the Feature toolbar:

Replicate one or more selected parts or surfaces about a specified plane, planar face, or mate connector. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Mirror may also be used during an active sheet metal operation.

**Steps**

1. Click 🔄.
2. Select the Result operation type:
   - **Part** - Mirror an individual part or surface
   - **Feature** - Mirror a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, etc). Note that Feature does not work for sheet metal; see the Face pattern type for sheet metal modifications.
   - **Face** - Pattern a specific face on a specific part or surface

3. Select Result operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create material added to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. With the focus on the Entities to mirror field, select entities to mirror.

   When selecting Faces to pattern, the "Create Selection" on page 229 can be useful to select related faces.

5. Click in the Mirror plane field to give it focus, then select the plane, planar face or mate connector (inferred or existing) about which to mirror. Click the in order to select inferred Mate connectors.

   Once a Mate connector is selected, click the Mate connector icon in the dialog field (outline in blue below) to open a dialog with which to edit the Mate connector:
Notice that with the slider towards the right, you get an instant preview of the result.

6. Select whether to merge the new entity with other entities that touch or intersect its geometry:

- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching or intersecting parts, you can check Merge with all to add all touching or intersecting parts to the merge scope.

Note that if the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will result in an error. For New, no merge scope is available since New does not boolean the result.

7. Click ✓

**Mirror: Part**

Mirror an individual part

This part is mirrored across a plane to add material to the existing material:
**Mirror: Feature**

Mirror a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc)

This Extrude Remove feature is mirrored across the plane to remove material from the part:

**Mirror: Face**

Mirror a specific face (or faces) on a specific part.

Several faces are mirrored across the plane to add material to the existing part:
**Mirror: New/Add (new material)**

**New** - Create material that results in a new part:

![New material](image1)

**Add** - Create material added to the existing material:

![Add material](image2)

Add material to a sheet metal part:
Mirror: Remove (cut material)

Take material away from existing material:

Mirror: Intersect

Leave material only where geometry overlaps

Before:
Mirror: iOS

Replicate one or more selected parts or surfaces about a specified plane or planar face. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Mirror may also be used during an active sheet metal operation.

Steps

1. Tap Mirror tool.
2. Select the mirror type:
   - **Part mirror** - To mirror an individual part
   - **Feature mirror** - To mirror a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)
   - **Face mirror** - To pattern a specific face on a specific part

3. Select result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create material added to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select entities to mirror.

5. Select mirror plane.

6. Tap the checkmark.
**Mirror: Part**

Mirror an individual part.

A part is mirrored across the Right plane to add material to the existing material.

**Mirror: Feature**

Mirror a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.).

An extrude remove feature is mirrored across the Right plane to remove material from the existing part.
**Mirror: Face**

Mirror a specific face on a specific part.

Several faces are mirrored across the Right plane to add material to the existing part.
**Mirror: New/Add (new material)**

**New** - Create new material that results in a new part.
**Add** - Create material added to the existing material.
**Mirror: Remove (cut material)**

Take material away from a part.

**Mirror: Intersect**

Leave material only where intersections exist.

Before Mirror intersect:
After Mirror intersect:
Mirror: Android

Replicate one or more selected parts or surfaces about a specified plane or planar face. Create a new part or surface or modify an existing one by adding or removing material, or intersecting parts in its path. Mirror may also be used during an active sheet metal operation.

Steps

1. Tap Mirror tool.
2. Select the mirror type:

- **Part mirror** - To mirror an individual part

- **Feature mirror** - To mirror a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.)

- **Face mirror** - To pattern a specific face on a specific part
3. Select result body operation type:
   - **New** - Create new material that results in a new part.
   - **Add** - Create material added to the existing material.
   - **Remove** - Take material away from a part.
   - **Intersect** - Leave material only where intersections exist.

4. Select entities to mirror.

5. Select mirror plane.

6. Tap the checkmark.

**Mirror: Part**

Mirror an individual part.

A part is mirrored across the Right plane to add material to the existing material.
**Mirror: Feature**

Mirror a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc.).

An extrude remove feature is mirrored across the Right plane to remove material from the existing part.

**Mirror: Face**

Mirror a specific face on a specific part.

Several faces are mirrored across the Right plane to add material to the existing part.
**Mirror: New/Add (new material)**

*New* - Create new material that results in a new part.
Add - Create material added to the existing material.
**Mirror: Remove (cut material)**

Take material away from a part.

**Mirror: Intersect**

Leave material only where intersections exist.

Before Mirror intersect:
After Mirror intersect:
Boolean

Modify parts by merging parts or surfaces together (Union), removing a part/surface from a target (Subtract), or calculating the intersection between two or more parts or surfaces (Intersect).
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Boolean: Desktop**

In the Feature toolbar:

Modify parts by merging parts or surfaces together (Union), removing a part/surface from a target (Subtract), or calculating the intersection between two or more parts or surfaces (Intersect).

**Steps**

1. Click 🔄.

2. Select an operation type:
   - **Union** - Merge parts or surfaces.
   - **Subtract** - Remove parts, allowing for an offset between parts or surfaces, if desired.
   - **Intersect** - Merge parts or surfaces, keeping material only where intersections exist.

3. Select tools.
   - If the operation is Subtract, also select:
     - **Targets** - The part or surface being acted upon
     - **Offset** (optional) - Create a gap between the remaining parts or surfaces
**Keep tools** (optional) - Check to keep the entities, or uncheck to remove the entities from the Part Studio

If you select Offset, also select Faces to offset, Offset distance, Direction of offset, and optionally toggle Reapply fillet.

- If the operation is Intersect, optionally select Keep tools.

4. Click 🔄.

**Boolean: Union**

Boolean to merge.

**Before:**

![Before image](image)

**After:**

![After image](image)
**Boolean: Subtract**

Boolean to remove parts.

Before:

After:
Boolean to remove parts and apply an offset between parts (or surfaces).

Before:

After:
**Boolean: Intersect**

Boolean to merge parts or surfaces, keeping material only where overlapping geometry exists.

Before:

![Before Image]

After:

![After Image]
**Tips**

- With the Subtract option, you have the choice to use the Keep tools checkbox to keep the parts or surfaces used to cut the main entity. This is useful when creating fitted parts or surfaces within the Part Studio.

- Use the Intersect option to keep only the material that intersects the selected parts or surfaces.

- When parts or surfaces are merged and as a result some entities no longer exist, the attributes of the earlier selected entity (such as part name) are retained.

**Boolean: iOS**

Modify parts by merging parts or surfaces together (Union), removing a part or surface from a target (Subtract), or calculating the intersection between two or more parts or surfaces (Intersect).

**Steps**

1. Tap Boolean tool.
2. Select an operation type:

- **Union** - Merge parts or surfaces
- **Subtract** - Remove parts or surfaces
- **Intersect** - Merge parts or surfaces, keeping material only where intersections exist.

3. Select tools.

   If the operation is **Subtract**, also select:

   - Targets
   - Offset (optional)
   - Keep tools (optional)

   If you select Offset, also select faces to offset, offset distance, direction of offset, and optionally toggle reapply fillet.

   If the operation is **Intersect**, also select:

   - Keep tools (optional)

4. Tap checkmark.

**Boolean: Union**

Boolean to merge parts or surfaces.

Before:
After:
**Boolean: Subtract**

Boolean to remove parts or surfaces.

Before:
After:

Boolean: Intersect
Boolean to merge parts or surfaces, keeping material only where intersections exist.

Before:

During:
After:
Tips

- With the Subtract option, you have the choice to use the Keep tools checkbox to keep the parts or surfaces used to cut the main entity. This is useful when creating fitted parts or surfaces within the Part Studio.

- Use the Intersect option to keep only the material that intersects the selected parts.

- When parts or surfaces are merged and as a result some entities no longer exist, the attributes of the earlier selected entity (such as part name) are retained.

**Boolean: Android**

Modify parts by merging parts or surfaces together (Union), removing a part or surface from a target (Subtract), or calculating the intersection between two or more parts or surfaces (Intersect).

**Steps**

1. Tap Boolean tool.

![Boolean Tool Interface]

- **Boolean 1**
  - **Union**
  - **Subtract**
  - **Intersect**

- **Tools**
  - **Cannot resolve entities.**
  - **No entities selected**

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2. Select an operation type:
   - **Union** - Merge parts or surfaces
   - **Subtract** - Remove parts or surfaces
   - **Intersect** - Merge parts or surfaces, keeping material only where intersections exist.

3. Select tools.
   If the operation is **Subtract**, also select:
   - Targets
   - Offset (optional)
   - Keep tools (optional)
     If you select Offset, also select faces to offset, offset distance, direction of offset, and optionally toggle reapply fillet.
   If the operation is **Intersect**, also select:
   - Keep tools (optional)

4. Tap checkmark.

**Boolean: Union**

Boolean to merge parts or surfaces.

Before:
After:
**Boolean: Subtract**

Boolean to remove parts or surfaces.

Before:
After:

Boolean: Intersect
Boolean to merge parts or surfaces, keeping material only where intersections exist.

Before:

During:
After:
Tips

- With the Subtract option, you have the choice to use the Keep tools checkbox to keep the parts or surfaces used to cut the main entity. This is useful when creating fitted parts or surfaces within the Part Studio.
- Use the Intersect option to keep only the material that intersects the selected parts.
- When parts or surfaces are merged and as a result some entities no longer exist, the attributes of the earlier selected entity (such as part name) are retained.

Split

Separate an existing part or face into multiple new parts or faces using a plane, mate connector, surface or face of a part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Split: Desktop

In the Feature toolbar:

Separate an existing part or face into multiple new parts or faces using a plane, mate connector or implicit mate connector, surface or face of a part.

Split Part

1. Click 📁.
2. Select Part as the type of entity to split.

3. Select the parts or surfaces to split.

4. Select the entity to split with: a plane, mate connector or implicit mate connector, surface, or face. Click the mate connector icon to visualize (on hover) implicit mate connectors from which to choose. You can also select explicitly created mate connectors in the Feature list.

5. Optionally toggle to Keep tools.

   Toggle on Keep tools to keep the entity that the split was made with. Unselect to remove the entity the split was made with when the feature is generated. The exception is when a face is selected, Keep tools defaults to selected face and the face is kept.

6. Optionally select Trim to face boundaries to limit the split to the bounding edges of the face selected to perform the split. Otherwise, Onshape extends the edges of the selected face. See the example below:

   With Trim to face boundaries selected, the arc selected as the entity to split with is limited to bounding edges of the surface selected and splits the part along one corner:
With **Trim to face boundaries** unselected, Onshape uses the extended edges of the selected arc and splits the part along all intersections of that extended edge:

**Trim to face boundaries** is valid only when a face is selected. When a full surface is selected, this option is ignored.

**Keep tools** is valid only when a full surface is selected (when using surfaces). When a face is selected, this option defaults to on.

If a multi-face surface is selected as a tool with Trim to face boundaries selected, you must select a single face of the surface as the tool in order to also trim.

7. Click ✅.

**Split Face**

1. Click 📛.
2. Select Face as the type of entity to split.

3. Select the faces to split.

4. Select the entity to split with: a plane, mate connector or implicit mate connector, surface, or face. Click the mate connector icon to visualize (on hover) implicit mate connectors from which to choose. You can also select explicitly created mate connectors in the Feature list.

5. Click ✓.

Examples

Split Part

Split a part using a plane, creating another part:
Split Surface

Split a surface with the Right plane, creating three separate surfaces:

Split Face

Split a face with the Right plane, creating two separate faces on the one part:
Split Face with sketch entities

Split: iOS
Separate an existing part or face into multiple new parts or faces using a plane, surface or face of a part.

Split Part
1. Tap Split tool.
2. Select Part as the type of entity to split.

3. Select the parts or faces to split.

4. Select the entity to split with: a plane, surface, or face.

5. Optionally, toggle to keep tools.

   Toggle on Keep tools to keep the entity that the split was made with. Leave Keep tools toggled off and the entity that the split was made with will be automatically removed when the feature is generated. The exception is when a face is selected as the tool, Keep tools then default to the selected face and the face is kept.

   Keep tools is valid only when a full surface is selected (when using surfaces to split with). When a face is selected as the tool, this option defaults to On.

6. Optionally select Trim to face boundaries to limit the split to the bounding edges of the face selected to perform the split. Otherwise, Onshape extends the edges of the selected face.
If a multi-face surface is selected as a tool with Trim to face boundaries selected, you must select a single face of the surface as the tool in order to also trim.

7. Tap checkmark.

**Split Face**

1. Tap Split tool.

2. Select Face as the type of entity to split.

3. Select the faces to split.

4. Select the entity to split with: a surface, plane, or sketch.

   When splitting a face with a surface that intersects it, you can elect to keep the surface (check *Keep tool surfaces*).

5. Tap checkmark.

**Examples**
Split Part

A part is split with the Right plane, creating another part.

Split Surface

A surface is split by the Right plane, creating three separate surfaces.
Split Face

A face of a part is split by the Right plane, creating two separate faces on the one part.
Split: Andriod

Separate an existing part or face into multiple new parts or faces using a plane, surface or face of a part.

Split Part

1. Tap Split tool.
2. Select Part as the type of entity to split.

3. Select the parts or faces to split.

4. Select the entity to split with: a plane, surface, or face.

5. Optionally, toggle to keep tools.

   Toggle on **Keep tools** to keep the entity that the split was made with. Leave **Keep tools** toggled off and the entity that the split was made with will be automatically removed when the feature is generated. The exception is when a face is selected as the tool, Keep tools then default to the selected face and the face is kept.

   Keep tools is valid only when a full surface is selected (when using surfaces to split with). When a face is selected as the tool, this option defaults to On.

6. Optionally select **Trim to face boundaries** to limit the split to the bounding edges of the face selected to perform the split. Otherwise, Onshape extends the edges of the selected face.

   If a multi-face surface is selected as a tool with Trim to face boundaries selected, you must select a single face of the surface as the tool in order to also trim.
7. Tap checkmark.

**Split Face**

1. Tap Split tool.

2. Select Face as the type of entity to split.

3. Select the faces to split.

4. Select the entity to split with: a surface, plane, or sketch.
   
   When splitting a face with a surface that intersects it, you can elect to keep the surface (check **Keep tool surfaces**).

5. Tap checkmark.

**Examples**
Split Part

A part is split with the Right plane, creating another part.

Split Surface

A surface is split by the Right plane, creating three separate surfaces.
Split Face

A face of a part is split by the Right plane, creating two separate faces on the one part.
**Transform**

Adjust a part's (and its Mate connector or axis, if desired) location and orientation in 3D space with the option to copy the part in place. When transforming a part, you also have the ability to select its Mate connector (inferred or existing), or axis, so the Mate connector stays with the part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Transform: Desktop**
In the Feature toolbar:

Adjust a part's (and its Mate connector or axis, if desired) location and orientation in 3D space with the option to copy the part in place. When transforming a part, you also have the ability to select its Mate connector (inferred or existing) or axis, so the Mate connector stays with the part.

**Steps**

1. Click 🔄.

2. Select a part (and its Mate connector or axis, if desired) to move.

3. Select the method of moving the part (transform type):
   - **Translate by line** - Move the part along a selected line: such as a part edge.
   - **Translate by distance** - Move the part by a specified value and in a direction indicated by the selection of an entity or mate connector.
   - **Translate by XYZ** - Move the part along an axis by a specified value. Optionally, use the drag manipulator that appears to position the part along axis.
   - **Transform by Mate connectors** - Specify two Mate connectors (inferred or existing), or axis, by which to reorient the placement of the part. Use the directional arrows to flip the orientation:
- **Rotate** - Move the part about an axis or Mate connector (inferred or existing) specified by selecting an entity.

- **Copy in place** - Make a copy of the part at the same location; this creates a separate and independent part enabling you to:
  - Make changes to one part and use both to create different parts during a later operation.
  - Make a copy of a part prior to a series of operations enabling you to reference the original state for ancillary operations.
  - Create multiple copies of a part in order to create multiple variants.

  If you need to create multiple copies of a part at once, use the Pattern feature with a distance of 0 (zero).

- **Scale** - Scale a part by a specific factor and select a point to scale about; selecting Scale presents a Scale Uniformly checkbox. Uncheck this box to specify your own scale for X, Y, and Z axes, and also select a point or Mate connector (inferred or existing) to scale about. Using a Mate connector (inferred or existing), or axis, to scale about changes the coordinate system to be relative to the Mate connector or axis.

  Optionally, select **Copy part** to duplicate the part.

4. Click ✔️.
**Translate by line**

Before translate by line:

After translating along the selected line specified by a part edge:

**Translate by distance**

Before translate by distance:

After translating by specifying a value and a direction:
Translate by XYZ

After translating along an axis by a specified distance with the copy part option checked:

Transform by Mate connectors or axis

Before transform by Mate connectors or axis:
After transform from one Mate connector to another:

**Rotate**

Before rotate:

After rotate:

**Copy in place**

The copy is placed exactly where the part is originally. Note that the Parts list reflects the new part and part count.
Scale uniformly

Before scale uniformly:

After scale uniformly:
**Non-uniform scale**

Before scale:

After scale:

**Transform: iOS**

Adjust a part's (and its Mate connector or axis, if desired) location and orientation in 3D space with the option to copy the part in place. When transforming a part, you also have the ability to select its Mate connector (inferred or existing), or axis, so the Mate connector stays with the part.

**Steps**

1. Tap Transform.
2. Select entities to transform or copy.

3. Select the Transform type (method of moving the entities):

   - **Translate by line** - Select an entity (such as a part edge)
   - **Translate by distance** - Select an entity to indicate direction and specify a value, OR use the drag manipulator that appears.
   - **Translate by XYZ** - Specify axis values to move along, OR use the drag manipulator that appears to position the part along the axis
   - **Transform by mate connectors** - Specify two mate connectors by which to reorient the placement of the part.
• Flip the primary axis, Z orientation of the instances.

• Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) by 90 degrees at a tap.

• **Rotate** - Move the part about an axis specified by selecting an entity and specify the angle, OR use the drag manipulator that appears.

• **Copy in place** - Make a copy of the part at the same location; this creates a separate and independent part enabling you to:
  - Make changes to one part and use both to create different bodies during a later operation.
  - Make a copy of a part prior to a series of operations enabling you to reference the original state for ancillary operations.
  - Create multiple copies of a part in order to create multiple variants.

  **Tip:** If you need to create multiple copies of a part at once, use the Pattern feature with a distance of 0 (zero).

• **Scale uniformly** - Scale a part by a specific factor and select a point to scale about; optionally copy the part.

4. Optionally, toggle to switch to the opposite direction.

5. Select lines or points.

6. Optionally, toggle to copy part.

7. Tap checkmark.

**Translate by line**

The part is translated by the sketch line above it, to a new location across the Right plane:
Translate by distance

The part is translated by distance. Specifically, it is translated 3.1215 inches from its original position. With Translate by distance, you can only translate in one direction at a time:

Translate by XYZ

The part is translated from its original position in several directions at once:
Transform by Mate connectors

Two parts are transformed to share the same Mate connector location:
**Rotate**

The part is rotated 45 degrees from its original position:

**Copy in place**

Use copy in place to copy one or multiple parts in their exact location.

**Scale uniformly**

The part is scaled at 1.5 times its original size:
Non-uniform scale

Before scale:

After scale:

Transform: Android

Adjust a part's (and its Mate connector or axis, if desired) location and orientation in 3D space with the option to copy the part in place. When transforming a part, you also have the ability to select its Mate connector (inferred or existing), or axis, so the Mate connector stays with the part.

Steps

1. Tap Transform.
2. Select entities to transform or copy.

3. Select the Transform type (method of moving the entities):

- **Translate by line** - Select an entity (such as a part edge)
- **Translate by distance** - Select an entity to indicate direction and specify a value, OR use the drag manipulator that appears.
- **Translate by XYZ** - Specify axis values to move along, OR use the drag manipulator that appears to position the part along the axis
• **Transform by mate connectors** - Specify two mate connectors by which to reorient the placement of the part.

  - 🔄 Flip the primary axis, Z orientation of the instances.

  - ⏰ Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) by 90 degrees at a tap.

• **Rotate** - Move the part about an axis specified by selecting an entity and specify the angle, OR use the drag manipulator that appears.

• **Copy in place** - Make a copy of the part at the same location; this creates a separate and independent part enabling you to:
  - Make changes to one part and use both to create different bodies during a later operation.
  - Make a copy of a part prior to a series of operations enabling you to reference the original state for ancillary operations.
  - Create multiple copies of a part in order to create multiple variants.

  **Tip:** If you need to create multiple copies of a part at once, use the Pattern feature with a distance of 0 (zero).

• **Scale uniformly** - Scale a part by a specific factor and select a point to scale about; optionally copy the part.

4. Optionally, toggle to switch to the opposite direction.

5. Select lines or points.

6. Optionally, toggle to copy part.

7. Tap checkmark.

**Translate by line**

The part is translated by the sketch line above it, to a new location across the Right plane:
Translate by distance

The part is translated by distance. Specifically, it is translated 3.1215 inches from its original position. With Translate by distance, you are only able to translate in one direction at a time:

Translate by XYZ

The part is translated from its original position in several directions at once:
Transform by mate connectors

Two parts are transformed to share the same mate connector location:
**Rotate**

The part is rotated 45 degrees from its original position:

**Copy in place**

Use copy in place to copy one or multiple parts in their exact location.

**Scale uniformly**

The part is scaled at 1.5 times its original size:
Non-uniform scale

Before scale:

After scale:

Wrap

Wrap a surface or sketch entity around a cylinder.

This feature currently only works on cylindrical parts and surfaces.

This functionality is available on Onshape's browser, iOS, and Android platforms.
Wrap: Desktop

In the Feature toolbar:

Wrap a surface or sketch entity around a cylinder.

This feature currently only works on cylindrical parts and surfaces.

Steps

1. Select .

2. Select the Tools, which are faces or sketch regions that you are wrapping around the cylinder.

3. Select the Target, the cylinder you are wrapping faces or sketch regions around.

4. To change the position of the tools:
   - **Angle** - Change the degrees of tilt.
   - **U Shift** - Move the result horizontally.
   - **V Shift** - Move the result vertically.
5. Select the check mark when finished.

**Steps for embossing or engraving**

1. Click Solid at the top of the dialog.

2. Adjust the settings in the Wrap dialog box to your preference.

a. Choose whether you want to:

- **New** - Create a new solid
- **Add** - Add to an existing solid
- **Remove** - Subtract from an existing solid
- **Intersect** - Keep the intersection of two (or more) solids

b. Choose the thickness of the solid.

c. Select the Trim to Target option to make the tools not visible when it goes beyond the borders of the target.

d. Select the Specify Anchor Points option to match one point of the tool to the target, assigning location.

e. If necessary or desired, select Merge scope (or Merge with all) to select parts with which to merge the new (additional) part.

**Steps for producing surfaces**

1. Select Surface at the top of the dialog if not there already.

2. Adjust the settings in the Wrap dialog box to your preference.

a. Select the Trim to Target option to make the tools not visible when it goes beyond the borders of the target.
b. Select the Specify Anchor Points option to match one point of the tool to the target, assigning location.

Steps for splitting the target face

1. Select Split at the top of the dialog.
2. Adjust the settings in the Wrap dialog box to your preference.

a. Select the Specify Anchor Point option to match one point of the tool to the target, assigning location.

Wrap: iOS
Basic steps
1. Tap 🔄.

Wrap 1

Select planar faces or sketch regions to wrap.

**Tools**

No entities selected

**Target**

No entities selected

Flip alignment

Trim to target

Specify anchor points

Angle

0 deg

Angle opposite direction

U shift

0 in

U shift opposite direction

V shift

0 in

V shift opposite direction

Final
2. Select Solid, Surface, or Split.

3. Select the Tools, which are the faces or sketch regions that you are wrapping around the cylinder.

4. Select the Target, the cylinder you are wrapping faces or sketch regions around.

5. To change the position of the tools, select one of the following:
   - **Angle** - Change the degrees of tilt.
   - **U Shift** - Move the result horizontally.
   - **V Shift** - Move the result vertically.

6. Tap the check mark when finished.

**Steps for Embossing or Engraving**

1. Select **Solid** at the top of the dialog.

2. Adjust the settings in the Wrap dialog box to your preference.
   a. Choose whether you want to:
      - **New** - Create a new solid
      - **Add** - Add to an existing solid
      - **Remove** - Subtract from an existing solid
      - **Intersect** - Keep the intersection of two (or more) solids
   b. Enter a figure for the thickness of the solid.
   c. Toggle flip alignment, if necessary.
   d. Select the Trim to Target option to make the tools not visible when it goes beyond the borders of the target.
   e. Select the Specify anchor points option to match one point of the tool to the target, assigning location.

Copyright © 2017, Onshape. All rights reserved.
f. Specify an angle measurement to angle the wrap

g. Toggle Angle opposite direction, if desired.

h. Toggle U shift opposite direction, if desired.

i. Enter a measurement for a U shift opposite direction, if desired.

j. Toggle V shift opposite direction, if desired.

k. Enter a measurement for a V shift, if desired.

l. Enter a thickness measurement, if desired.

m. Toggle thickness opposite direction, if desired.

n. If necessary or desired (for operations that add or remove material to an existing part) select Merge scope (or Merge with all) to select parts with which to merge the new part.

3. Tap the checkmark to accept and close the dialog.

**Steps for Producing Surfaces**

1. Select **Surface** at the top of the dialog if not there already.

2. Adjust the settings in the Wrap dialog box to your preference.

a. Select the Trim to Target option to make the tools not visible when it goes beyond the borders of the target.

b. Select the Specify Anchor Points option to match one point of the tool to the target,
assigning location.

3. Tap the checkmark to accept and close the dialog.

**Steps for Splitting the Target Face**

1. Select Split at the top of the dialog.

2. Adjust the settings in the Wrap dialog box to your preference.

a. Select the Specify Anchor Point option to match one point of the tool to the target, assigning location.

3. Tap the checkmark to accept and close the dialog.

**Wrap: Android**
Basic Steps
1. Tap 🌐.
### Wrap 2

Select planar faces or sketch regions to wrap.

#### Solid Surface Split

<table>
<thead>
<tr>
<th>Tools</th>
<th>No entities selected</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Target</th>
<th>No entities selected</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Flip alignment</th>
<th>off</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trim to target</td>
<td>off</td>
</tr>
<tr>
<td>Specify anchor points</td>
<td>off</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Angle</th>
<th>0.0 deg</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle opposite direction</td>
<td>off</td>
</tr>
<tr>
<td>U shift</td>
<td>0.0 mm</td>
</tr>
<tr>
<td>U shift opposite direction</td>
<td>off</td>
</tr>
<tr>
<td>V shift</td>
<td>0.0 mm</td>
</tr>
<tr>
<td>V shift opposite direction</td>
<td>off</td>
</tr>
</tbody>
</table>
2. Select Solid, Surface, or Split.

3. Select the Tools, which are the faces or sketch regions that you are wrapping around the cylinder.

4. Select the Target, the cylinder you are wrapping faces or sketch regions around.

5. To change the position of the tools, select one of the following:
   - **Angle** - Change the degrees of tilt.
   - **U Shift** - Move the result horizontally.
   - **V Shift** - Move the result vertically.

6. Tap the check mark when finished.

**Steps for embossing or engraving**

1. Select **Solid** at the top of the dialog.

2. Adjust the settings in the Wrap dialog box to your preference.

   a. Choose whether you want to:
      - **New** - Create a new solid
      - **Add** - Add to an existing solid
      - **Remove** - Subtract from an existing solid
      - **Intersect** - Keep the intersection of two (or more) solids

   b. Enter a figure for the thickness of the solid.

   c. Toggle flip alignment, if necessary.

   d. Select the Trim to Target option to make the tools not visible when it goes beyond the borders of the target.

   e. Select the Specify anchor points option to match one point of the tool to the target, assigning location.
f. Specify an angle measurement to angle the wrap

g. Toggle Angle opposite direction, if desired.

h. Toggle U shift opposite direction, if desired.

i. Enter a measurement for a U shift opposite direction, if desired.

j. Toggle V shift opposite direction, if desired.

k. Enter a measurement for a V shift, if desired.

l. Enter a thickness measurement, if desired.

m. Toggle thickness opposite direction, if desired.

n. If necessary or desired (for operations that add or remove material to an existing part, select Merge scope (or Merge with all) to select parts with which to merge the new (additional) part.

3. Tap the checkmark to accept and close the dialog.

**Steps for producing surfaces**

1. Select **Surface** at the top of the dialog if not there already.

2. Adjust the settings in the Wrap dialog box to your preference.

a. Select the Trim to Target option to make the tools not visible when it goes beyond the borders of the target.

b. Select the Specify Anchor Points option to match one point of the tool to the target,
assigning location.

3. Tap the checkmark to accept and close the dialog.

**Steps for splitting the target face**

1. Select Split at the top of the dialog.

2. Adjust the settings in the Wrap dialog box to your preference.

   a. Select the Specify Anchor Point option to match one point of the tool to the target, assigning location.

3. Tap the checkmark to accept and close the dialog.

---

**Delete Part**

![Delete icon] Delete one or more parts or surfaces; this is a parametric operation that creates a delete-part feature and is able to be undone.

This functionality is available on Onshape’s browser, iOS, and Android platforms.

**Delete Part: Desktop**
In the Feature toolbar:

Delete one or more parts or surfaces; this is a parametric operation that creates a delete-part feature and is able to be undone.

Steps

1. Click 

When deleting a non-composite part, the selection in the dropdown is irrelevant. Just select the part to be deleted. To use Delete part with a composite part, see "Converting closed composite parts to individual parts" on page 1314 for more information.

2. Select the part or surface to delete.

3. Click 

Notice that the deleted part or surface is no longer listed in the Feature list, and a new feature appears, Delete Part.

Tips

- Delete Part is useful when you want to use a part as a tool part in multiple Boolean features and later discard it.

- Note that selecting a face or edge in the graphics area will highlight the feature that created it in the feature list, and pressing the Delete button will delete the feature, not the part. This action also creates a parametric operation, and is able to be undone.

- You are able to click to select more than one part at a time, with either method of deleting (through the Part list or the Delete part tool).
Part colors are re-sequenced when a part is deleted (unless the colors are custom-assigned), according to the Onshape automatic color sequence. See "Part Studios" on page 279 for more information.

Delete Part: iOS and Android

Delete one or more parts or surfaces; this is a parametric operation that creates a delete-part feature and is able to be undone.

Steps

1. Tap Delete part tool.

2. Select the part or surface to delete (you are able to select in the graphics area or in the Feature list).

3. Tap the checkmark.

4. Notice that the deleted part or surface is no longer listed in the Feature list, and a new Feature appears, Delete Part.

Tips

● Delete part is useful when you want to use a part as a tool body in a Boolean multiple times and Boolean and then later discard it.
• Note that selecting a face or edge in the graphics area will highlight the feature that created it in the feature list, and pressing the Delete button will delete the feature, not the part. This action also creates a parametric operation, and is able to be undone.

• You are able to tap to select more than one part at a time, with either method of deleting (through the Part list or the Delete part too).

Part colors are re-sequenced when a part is deleted (unless the colors are custom-assigned), according to the Onshape automatic color sequence.

---

**Modify Fillet**

Alter or remove existing fillets or rounds; this Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Modify Fillet: Desktop**

In the Feature toolbar:

Alter or remove existing fillets or rounds; this Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.
Steps

1. Click 🔄.

![Modify fillet 1](image)

2. Select the fillet faces to change or remove.

3. Make the select to either Change the radius of the fillet, or Remove the fillet.

4. When changing the radius, enter a new value.

5. Click ✅.

Tips

- Keep Reapply fillet checked to ensure that the modified fillet flows nicely into any derivative fillets. Unchecking this parameter may result in undesired feature characteristics.

- In the case of many fillets that run into each other, it may be difficult to select all necessary faces. You can make it easier by using the "Create Selection" on page 229 option on the context menu.

**Modify Fillet: iOS and Android**

Alter or remove existing fillets or rounds; this Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

Steps

1. Tap Modify fillet tool:
2. Select the face or faces (with fillets) to modify.

3. Enter a new value for the radius.

4. Select to reapply the fillet.

5. Tap the checkmark.

---

**Delete Face**

Remove geometry from a part. Select whether to heal the surrounding faces (by extending until they intersect), cap the void, or leave the void open. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Delete Face: Desktop**
In the Feature toolbar:

Remove geometry from a part. Select whether to heal the surrounding faces (by extending until they intersect), cap the void, or leave the void open. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

Steps

1. Click 

2. On the model, select the part face or faces to delete.

3. Select how to treat the remaining void:

   - **Heal** - Extend the surrounding faces until they intersect.
   - **Cap** - Place a face over the remaining void.
   - **Leave open** - Do nothing, leaving the face removed and void visible; this creates a surface out of the existing solid.

4. Check Delete fillet faces to indicate whether or not to delete the adjacent filleted faces as well.

5. Click 

Tips

The "Create Selection" on page 229 arrow (next to the Faces field) is useful when selecting related faces for Delete face.

**Delete Face: iOS and Android**
Remove geometry from a part. Select whether to heal the surrounding faces (by extending until they intersect), cap the void, or leave the void open. This Direct Editing tool is especially convenient if you don’t have the parametric history of the part, as is often the case with an imported part.

Steps

1. Tap the Delete face tool.

2. Select faces to delete.

3. Select how to treat the remaining void:
   - Heal - Extend the surrounding faces until they intersect.
   - Cap - Place a face over the remaining void.
   - Leave open - Do nothing, leaving the face removed and the void visible.

4. Optionally, toggle to delete fillet faces. This deletes adjacent filleted faces as well.

5. Tap the checkmark.

Before:
During:

After:
Move Face

Translate, rotate, or offset one or more selected faces. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Move Face: Desktop

In the Feature toolbar:

Translate, rotate, or offset one or more selected faces. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

Steps
1. Click 🎨.

2. Select faces to move.

3. Select the type of move:
   
   **Offset** - Typically used with non-planar faces to increase or decrease a radius.
   
   - Select any face or combination of faces (Faces field).
   - Blind End type - Enter a distance, flip Directional arrow if necessary.
   - Up to entity End type - Select a part or surface.
   - Specify the value of the offset (Numeric field).

   Use the direction arrows to change the direction of the offset, if necessary.

   **Translate** - Move one or more faces:

   - Select any face or combination of faces (Faces field) to move.
   - In the Direction field, select a Mate connector (inferred or existing) or axis (to define the direction vector parallel to the selected Mate connector), an edge (to define the direction vector parallel to the selected edge), or a face (to define the direction vector normal to the selected face).
   - Blind End type - Enter a distance, flip Directional arrow if necessary.
   - Up to entity End type - Select a vertex or parallel face.

   Optionally use Offset distance to create an offset between faces.
**Rotate** - Rotate one or more faces a specified number of degrees.

- Select any face or combination of faces (Faces field).
- Select the axis or Mate connector (inferred or existing) to rotate from (Axis field).
- Specify the number of degrees to rotate.

Use the direction arrows to change the direction of the rotation, if necessary.

4. Click ✅.

**Tips**

The "Create Selection" on page 229 (next to the Faces field) can be useful to select related faces for Move face.

**Move Face: iOS**

Translate, rotate, or offset one or more selected faces. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

**Steps**

1. Tap Move Face tool.
2. Select move.

3. Select the type of move:

   **Translate** - Move one or more faces in a specified direction for a specified distance
   a. Select any face or combination of faces (Faces field) to move.
   b. Select an edge to define a vector (Direction field).
   c. Blind End type - Enter a distance, toggle Opposite direction if necessary.
   d. Up to entity End type - Select a vertex or parallel face.

   **Rotate** - Rotate one or more faces a specified number of degrees
   a. Select any face or combination of faces (Faces field).
   b. Select the axis to rotate from (Axis field).
   c. Specify the number of degrees to rotate.
Toggle Opposite direction to change the direction of the rotation, if necessary.

**Offset** - Offset one or more faces a specified distance (typically used with non-planar faces to increase or decrease a radius).

a. Select any face or combination of faces (Faces field).
b. Blind End type - Enter a distance, toggle Opposite direction, if necessary.
c. Up to entity End type - Select a vertex, edge, or face.
d. Specify the value of the offset (Numeric field).

   Toggle Opposite direction to change the direction of the offset, if necessary.

4. Optionally, toggle to reapply fillet.
5. Tap the checkmark.

**Translate**

Move one or more faces in a specified direction for a specified distance.

This shows one face, moved 0.75 inches.
**Rotate**

Rotate one or more faces a specified number of degrees

This shows one face rotated 20 degrees.
Offset

Offset one or more faces a specified distance.
This shows one face, offset by 0.2 inches.
Move Face: Android

Translate, rotate, or offset one or more selected faces. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

Steps
1. Tap Move Face tool.
2. Select move or create:
   - **Move** - Select faces to move
   - **Create** - Select face and surfaces to reference in order to create a new face

3. Select the type of move or create:
   - **Translate** - Move one or more faces in a specified direction for a specified distance
     
a. Select any face or combination of faces (Faces field) to move.
b. Select an edge to define a vector (Direction field).
c. Blind End type - Enter a distance, toggle Opposite direction if necessary.
d. Up to entity End type - Select a vertex or parallel face.
   - **Rotate** - Rotate one or more faces a specified number of degrees
     
a. Select any face or combination of faces (Faces field).
b. Select the axis to rotate from (Axis field).
c. Specify the number of degrees to rotate.
   
   Toggle Opposite direction to change the direction of the rotation, if necessary.
   - **Offset** - Offset one or more faces a specified distance (typically used with non-planar faces to increase or decrease a radius).
     
a. Select any face or combination of faces (Faces field).
b. Blind End type - Enter a distance, toggle Opposite direction, if necessary.
c. Up to entity End type - Select a vertex, edge, or face.
d. Specify the value of the offset (Numeric field).
   
   Toggle Opposite direction to change the direction of the offset, if necessary.

4. Optionally, toggle to reapply fillet.

5. Tap the checkmark.

**Translate**

Move one or more faces in a specified direction for a specified distance.

This shows one face, moved 0.75 inches.
**Rotate**

Rotate one or more faces a specified number of degrees

This shows one face rotated 20 degrees.
**Offset**

Offset one or more faces a specified distance.
This shows one face, offset by 0.2 inches.
Replace Face

Trim a face or extend a face to a new surface. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Replace Face: Desktop

In the Feature toolbar:

Steps

1. Click 🗑.

2. On the model, select the face you want to trim or extend.
3. Select the surface to use for the replacement.

Notice that a surface has been created in this example, to use to extend the face to. You have the ability to hide and unhide this part in the Parts list of the Feature list box:

4. Optionally provide an offset distance, or flip the alignment.

- Offset distance - Specify a distance by which to separate the face.
- Flip alignment - Select to change the face being replaced by to an opposing face. The default replacement face is the one closest to the face being replaced:
5. Click ✓.

**Tips**

The "Create Selection" on page 229 arrow ▶️ (next to the Faces field) is useful when selecting related faces for Replace face.

**Replace Face: iOS**

Trim a face or extend a face to a new surface. This Direct Editing tool is especially convenient if you don’t have the parametric history of the part, as is often the case with an imported part.

**Steps**

1. Tap Replace face tool.

2. Select faces to replace.

![Replace face 1](image-url)

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3. Select surface to replace with.
4. Optionally, toggle to flip alignment.
5. Specify offset distance.
6. Optionally, toggle to switch to opposite direction.
7. Tap the checkmark.

Before:

![Image of the before state](image1)

During:
After:
Replace Face: Android

Trim a face or extend a face to a new surface. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

Steps

1. Tap Replace face tool.
2. Select faces to replace.
3. Select surface to replace with.
4. Optionally, toggle to flip alignment.
5. Specify offset distance.
6. Optionally, toggle to switch to opposite direction.
7. Tap the checkmark.

Before:
During:

After:
Offset Surface

Create a new surface by offsetting an existing face, surface, or sketch region. Set offset distance to 0 to create a copy in place.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Offset Surface: Desktop

In the Feature toolbar:
Create a new surface by offsetting an existing face, surface, or sketch region. Set offset distance to 0 to create a copy in place.

**Steps**

1. Click 🔄.

![Offset surface 1](image)

Offset surface 1

- Faces, surfaces, and sketch regions
- Offset: 0.25 in

2. Select faces, surfaces, and sketch regions to offset.

3. Specify distance of offset and use the direction arrows icon to change the direction of the offset, if necessary.

4. Click ✅.

**Examples**

Select multiple faces

Part before selection and offset:

![Part](image)

Resulting surface:
Faces selected are the revolved face and the planar face and results in one surface.

Transform resulting surface

Select the surface and then the Transform tool to move the surface away from the original part, if desired:

Offset Surface: iOS

Create a new surface by offsetting an existing face, surface, or sketch region. Set offset distance to 0 to create a copy in place.

Steps

1. Tap 📊.
2. Select faces, surfaces, and sketch regions to offset.

3. Specify distance of offset and use the direction arrows icon to change the direction of the offset, if necessary.

4. Tap Checkmark.

**Examples**

Select multiple faces

Part before selection and offset:

[Diagram of part before selection and offset]

Resulting surface:
Faces selected are the revolved face and the planar face and results in one surface.

Transform resulting surface

Select the surface and then the Transform tool to move the surface away from the original part, if desired.

Offset Surface: Android

Create a new surface by offsetting an existing face, surface, or sketch region. Set offset distance to 0 to create a copy in place.

Steps

1. Tap 📝.
2. Select faces, surfaces, and sketch regions to offset.

3. Specify distance of offset and use the direction arrows icon to change the direction of the offset, if necessary.

4. Tap checkmark.

**Examples**

Select multiple faces

Part before selection and offset:

Resulting surface:
Faces selected are the revolved face and the planar face and results in one surface.

Transform resulting surface

Select the surface and then the Transform tool to move the surface away from the original part, if desired.

Fill

Create a surface (or a part from surfaces) by defining boundaries and refine the surface with boundary conditions (instead of requiring the use of reference surfaces).
This functionality is available on Onshape's browser and iOS platforms only.

**Fill: Desktop**

In the Feature toolbar:

Create a surface (or a part from surfaces) by defining boundaries and refine the surface with boundary conditions (instead of requiring the use of reference surfaces).

**Steps**

1. Click 🍳.

2. Select the Edges, which act as the boundaries of the fill.
3. Optionally, define **Continuity** for each selected edge or curve (you can select Curvature visualization from the small View cube menu to see the effects):

   a. **Position** - Make edges meet with no tangent or curvature relationship to each other

   b. **Tangency** - Create an implicit tangency (normal to the plane of the selected surface) between the boundaries and the new surface (as if you had a reference surface). This works for sketch selections only, not for other planar curves.

   c. **Curvature** - Match the actual curve of the adjacent surface

   For example, for the left most selected edge, below, each of the Continuity options were selected. Note the differences in the visualization stripes:

   Position: Tangency: Curvature:

4. Select **Guides** (points, vertices, points on a sketch, or curves) with which to influence the shape. The resulting surface intersects these points which will lie in the interior of the boundary. When curves are selected, the surface pushes through the curves and you can select from two types of calculations:

   a. **Sampled** - Uses the Sample Size to determine the number of vertices along the curves are used to calculate the surface. A long Sample size may result in the surface following the entire curve:

   Sample size of 3:
Depending on the Sample size, some rippling of the surface may occur.

b. **Precise** - Uses the exact curve to form the surface. Note that this option requires very carefully designed and selected curves. See the examples below for more information.

5. Check **Show iso curves** to evaluate how the underlying surface is defined.

   The untrimmed underlying surface is shown as a mesh to display the iso parametric curves, enabling you to evaluate the quality of the underlying surface.

6. Click ✔.

**Examples**

The edges of the surfaces and the two bridging curves are selected as boundaries. A new surface is created:
The Guide vertices are selected to further define the shape of the surface:

When using the Precise option with guide curves especially, if the curves meet each other, the intersection must be at a point such that the curve and the point are tangent to each other. Also, when using the Precise option, the curves must touch the boundary but not cross the boundary.
Guides that do not intersect

Guides that meet tangentially

Guide that touches one boundary

Guide that does not touch a boundary

This scenario can work when Sampled is selected:
But will not succeed when Precise is selected.

Guides that intersect at tangent plane

Guides that meet tangent criteria

Show iso curves

Show iso curves is selected to display the iso parametric curves, enabling you to evaluate the quality of the underlying surface:
Tips

- When selecting edges, you might see red dots; these indicate missing or open curves in the boundary.

- If the operation results in a closed surface (creating a volume), Onshape automatically creates a solid part (if Add is selected). If the creation of a part is an undesired result, use New to keep all surfaces and not create a part.

Fill: iOS

Create a surface (or a part from surfaces) by defining boundaries and refine the surface with boundary conditions (instead of requiring the use of reference surfaces).

Steps

1. Tap Fill tool.
2. Select the Edges, which are boundaries of the fill.

   Optionally, define Continuity for each selected edge or curve:

   a. **Tangent** - Create an implicit tangency (normal to the plane of the surface) between the boundaries and the new surface (as if you had a reference surface)

   b. **Curvature** - Match the actual curve of the other surface
c. **Position** - Make edges meet with no tangent or curvature relationship to each other

3. Select Guide (points, vertices, points on a sketch, or curves) with which to influence the shape. The resulting surface intersects these points. When curves are selected, the surface pushes through the curves and you can select from two types of calculations:

a. Sampled - Uses the Sample size to determine the number of vertices along the curves used to calculate the surface. A large sample size may result in the surface following the entire curve. A smaller Sample size may result in some rippling of the surface.

b. Precise - uses the exact curve to form the surface. Note that this option requires very carefully designed and selected curves.

4. Check Show iso curves to evaluate how the underlying surface is defined.

The untrimmed underlying surface is shown as a mesh to display the iso parametric curves, enabling you to evaluate the quality of the underlying surface.

5. Tap the checkmark.

**Examples**

The edges of the surfaces and the two bridging curves are selected as boundaries. A new surface is created.
Guide vertices
The Guide vertices are selected to further define the shape of the surface.

Show iso curves
Show iso curves is selected to display the iso parametric curves, enabling you to evaluate the quality of the underlying surface.

Tips
When selecting edges, you might see red dots; these indicate missing or open curves.
If the operation results in a closed surface (creating a volume), Onshape automatically creates a solid part (if Add is selected). If the creation of a part is an undesired result, use New to keep all surfaces and not create a part.

**Move Boundary**

Move boundary edges of a surface in order to extend or trim it.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Move Boundary: Desktop**

In the Feature toolbar:

1. Click 📦.

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2. Select a surface or boundary edges to move:

3. Adjust the settings in the Move boundary dialog box to your preference.
   a. Choose an end condition with the dropdown arrow:

   **Blind** - Enter a distance manually to extend or trim to.
**Up to face** - Select a target entity to extend or trim to.

**Up to part/surface** - Select a part or surface to extend or trim to.

**Up to vertex** - Select a vertex to extend or trim to.

b. To trim a surface or boundary edge, you have the option to enter a negative distance in the Distance field of the dialog box:

![Move boundary 1 dialog box](image)

Click the opposite direction icon, or manually drag the manipulator arrow back (shown below):

![Manipulator arrow](image)

c. Check the box next to Maintain curvature to make all the extended surface curvature continuous with the original surface boundary.
d. Check the box next to Tangent propagation to extend the surface for all edges tangent to the first edge selection.

4. Click the green check mark in the upper right corner of the dialog box to apply your changes. Click the to cancel without applying your changes.

**Move Boundary: iOS**

**Steps**

1. Tap .
2. Select a surface or boundary edge to move.
3. Adjust the settings in the Move Boundary dialog box to your preference.
a. Choose an end condition with the side arrow:

**Blind** - Enter a distance manually to extend or trim to.

**Up to face** - Select a target entity to extend or trim to.

**Up to part/surface** - Select a part or surface to extend or trim to.

**Up to vertex** - Select a vertex to extend or trim to.

b. Trim a surface or boundary edge by tapping the opposite direction switch or manually drag the manipulator arrow back.

c. Tap the Maintain curvature switch to make all the extended surface curvature continuous with the original surface boundary.

d. Tap the Tangent propagation switch to extend the surface for all the edges tangent to the first edge selection.

4. Tap Checkmark.

**Move Boundary: Android**

**Steps**

1. Tap ✅.

2. Select a surface or boundary edge to move.

3. Adjust the settings in the Move Boundary dialog box to your preference.
a. Choose an end condition with the side arrow:
   
   **Blind** - Enter a distance manually to extend or trim to.
   
   **Up to face** - Select a target entity to extend or trim to.
   
   **Up to part/surface** - Select a part or surface to extend or trim to.
   
   **Up to vertex** - Select a vertex to extend or trim to.

b. Trim a surface or boundary edge by tapping the opposite direction switch or manually drag the manipulator arrow back.

c. Tap the Maintain curvature switch to make all the extended surface curvature continuous with the original surface boundary.

d. Tap the Tangent propagation switch to extend the surface for all the edges tangent to the first edge selection.

4. Tap Checkmark.

---

**Plane**

Create a new construction plane.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Plane: Desktop**

In the Feature toolbar:

Create a new construction plane. Note that you can use the context menu for a plane to turn Section view on, more information below.

**Steps**

1. Click 📊.
2. Select an entity on which to base the new plane.

3. Make further specifications where necessary (see below).

4. Click \(\checkmark\).

You are able to create a plane based on the relative position to another entity, including:

- **Plane** - Select another plane, planar face, or Mate connector (inferred or existing).
- **Point** - Select a vertex, sketch point, Mate connector (inferred or existing), or the origin
- **Line** - Select a linear edge, sketch line, or cylindrical face to get its axis

Note that pre-selecting a planar face (solid or plane) and creating a plane defaults to Point normal plane.

Once you select the geometry with which to create the plane, you can adjust the normal of the plane to be one direction or the opposite: either check the Flip normal box, or leave it unchecked. You can see which direction is normal by looking at the name of the plane in the graphics area.
The image above shows the plane normal to the normal of the selected face.

The image above shows the plane with the normal flipped (the box is checked), and you can see by the name on the plane that the normal is reversed.

**Offset**

Create a plane a specified distance from another plane (or the XY plane of a Mate connector, inferred or existing) using a plane and a distance value.

Offset from a planar face:
Offset from a Mate connector:

Use the manipulator to drag the new plane to the desired distance; the numeric distance field in the dialog automatically updates. Click the manipulator arrow to flip the direction.

Offset from another plane:
**Plane Point**

Create a plane that passes through a point, parallel to a plane, using a plane and a point:

**Line Angle**

Create a plane that passes through a line at an angle, using a line, reference geometry (such as a plane, point, or axis) and an angle value:
Use the manipulator to drag the angle specification; see that the Angle field in the dialog automatically updates. Click the manipulator arrow to flip the direction.

**Point Normal**

Create a plane that is normal to the line and passes through the point, using: a straight axis (a straight line segment or anything that defines an axis (circle, arc, cylindrical face, revolved face, etc) and a point (vertex). The point is always the origin of the plane and the axis or line is always the normal of the plane:
Note that pre-selecting a planar face (solid or plane) and creating a plane defaults to Point normal plane.

**Three Point**

Create a plane that passes through three points, using three points.

The starting sketches, on two planes:

![Starting sketches](image)

The resulting third plane:

![Resulting third plane](image)
Mid Plane

Create a plane at the intersection of two other planes:
Flip alignment:
**Curve Point**

Create a curve point plane that passes through the point, perpendicular to the curve. Use one curve (or edge) defining the normal of the plane and one point (Mate connector or vertex) defining the origin of the plane. The plane normal is always tangent to the curve:
Tips

- Use the keyboard shortcut, p, to hide/unhide all planes.

- Once the plane is created, select it then use the context menu (RMB) and select Section view:
Click the X in the dialog to exit the Section view.

**Plane: iOS**

Create a new construction plane.

**Steps**

1. Tap Plane tool.
2. Select entities on which to base the new plane.

3. Select a plane type:
   - **Offset** - Create a plane a specified distance from another plane using a plane, planar face, or Mate connector (inferred or existing) and a distance value.
   - **Plane Point** - Create a plane that passes through a point, parallel to a plane, using a plane and a point or Mate connector (inferred or existing).
   - **Line Angle** - Create a plane that passes through a line at an angle, using a line, a vertex and an angle value.
   - **Point Normal** - Create a plane that is normal to the line and passes through the point.
   - **Three Point** - Create a plane that passes through three points, using three points.
   - **Mid Plane** - Create a plane at the intersection of two other planes.
   - **Curve Point** - Create a curve-tangent point plane that passes through the point, perpendicular to the curve.

4. Specify the distance or angle (depending on which plane type you have selected).
5. Optionally, toggle to switch to the opposite direction.

6. Tap the checkmark.

**Offset**

Create a plane a specified distance from another plane using a plane and a distance value. You are able to offset from a planar face or from another plane or Mate connector (inferred or existing).

Select entity on which to base the new plane and specify the distance.

---

**Plane Point**

Create a plane that passes through a point, parallel to a plane, using a plane and a point (or Mate connector, inferred or existing).

Select a plane for the new plane to remain parallel with and a point for the new plane to pass through.
Line Angle

Create a plane that passes through a line at an angle, using a line and an angle value.

Select a line for the plane to pass through and specify the angle of the plane. Select an additional point, plane or axis to specify where the angle is to be measured from.
**Point Normal**

Create a plane that is normal to the line and passed through the point, using: a straight axis (a straight line segment or anything that defines an axis (circle, arc, cylindrical face, revolved face, etc.) and a point. The point is always the origin of the plane and the axis or line is always the normal of the plane.

Select a line for the plane to pass through and a point for the plane to be normal to.
As shown in the image above, the plane is normal to the point on the corner of the sketch on the Right plane and it is passing through the selected line.

**Three Point**
Create a plane that passes through three points, using three points.

Select the three points for the plane to pass through.

As shown in the image below, the plane passes through all three points:

**Mid Plane**
Create a plane at the intersection of two other planes.
Select two planes for the new plane to intersect or a planar face and vertex or Mate connector (inferred or existing).

Curve Point
Create a curve-tangent point plane that passes through the point, perpendicular to the curve. Use one curve (or edge) defining the normal of the plane and one point defining the origin of the plane. The plane normal is always tangent to the curve. Select a curve for the plane to be perpendicular to and select a point to define the origin of the plane.

Plane: Android
Create a new construction plane.
Steps
1. Tap Plane tool.

2. Select entities on which to base the new plane.

3. Select a plane type:
   - **Offset** - Create a plane a specified distance from another plane using a plane (or Mate connector, inferred or existing) and a distance value.
   - **Plane Point** - Create a plane that passes through a point, parallel to a plane, using a plane and a point (or Mate connector, inferred or existing).
- **Line Angle** - Create a plane that passes through a line at an angle, using a line and an angle value.

- **Point Normal** - Create a plane that is normal to the line and passes through the point.

- **Three Point** - Create a plane that passes through three points, using three points.

- **Mid Plane** - Create a plane at the intersection of two other planes.

- **Curve Point** - Create a curve-tangent point plane that passes through the point, perpendicular to the curve.

4. Specify the distance or angle (depending on which plane type you have selected).

5. Optionally, toggle to switch to the opposite direction.

6. Tap the checkmark.

**Offset**

Create a plane a specified distance from another plane using a plane (or Mate connector, inferred or existing) and a distance value. You are able to offset from a planar face or from another plane.

Select entity on which to base the new plane and specify the distance.

**Plane Point**

Create a plane that passes through a point, parallel to a plane, using a plane and a
point (or Mate connector, inferred or existing).

Select a plane for the new plane to remain parallel with and a point for the new plane to pass through.

**Line Angle**

Create a plane that passes through a line at an angle, using a line and an angle value.

Select a line for the plane to pass through and specify the angle of the plane.
Point Normal

Create a plane that is normal to the line and passed through the point, using: a straight axis (a straight line segment or anything that defines an axis (circle, arc, cylindrical face, revolved face, etc.) and a point (or vertex). The point is always the origin of the plane and the axis or line is always the normal of the plane.

Select a line for the plane to pass through and a point for the plane to be normal to.
As shown in the image above, the plane is normal to the point on the corner of the sketch on the Right plane and it is passing through the selected line.

**Three Point**
Create a plane that passes through three points, using three points.
Select the three points for the plane to pass through.

As shown in the image below, the plane passes through all three points:

**Mid Plane**
Create a plane at the intersection of two other planes.
Select two planes for the new plane to intersect.

**Curve Point**

Create a curve-tangent point plane that passes through the point, perpendicular to the curve. Use one curve (or edge) defining the normal of the plane and one point (or vertex) defining the origin of the plane. The plane normal is always tangent to the curve.

Select a curve for the plane to be perpendicular to and select a point (or vertex) to define the origin of the plane.
**Helix**

Create a helix using a conical or cylindrical face, or circular edge.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Helix: Desktop**

In the Feature toolbar:

Create a helix using a conical or cylindrical face, or circular edge.

A helix may be used for sweeps (to create a simple spring). A helix does not consume the part used to create it.

**Steps**

1. With a cone, cylinder, or circular sketch in the graphics area, click 🔄.

   ![Helix Tool](image)

   If you don't see the Helix icon, expand the Plane/Mate connector icon group: 🔄 or 🖥.

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2. When selecting a conical or cylindrical face:
   a. Select either Turns or Pitch on which to base the size of the helix.
   b. Specify the direction of the turns, Clockwise or Counterclockwise.
   c. Specify the number of revolutions for Turns or the dimension of the Helical pitch (the distance traveled axially in each revolution).
   d. Indicate the Start angle (the measurement from a reference point on the cylinder or cone; the start of the revolve or the x-axis of an extruded circle), in preferred units.

3. When using a circular edge:
   a. Select either: Height and Turns, Height and Pitch, or Turns and Pitch.
   b. Specify the appropriate values as determined by your choice above:
      - **Clockwise / Counterclockwise** - Specify the direction of the turns
      - **Height** - The overall height of the helix (you can also use the drag manipulator)
      - **Turns** - The number of turns to the helix
      - **Helical pitch** - The distance traveled axially in each revolution
      - **Revolutions** - The number of revolutions to the helix
      - **Start angle** - The measurement from the circular edge

4. Click ✓

   **Examples**
Creating a spring

1. Create a helix as described above:

   ![Helix Diagram]

   If it helps, you can hide the cone or cylinder (use the eye symbol in the Parts list).

2. Create a curve point plane using the helix and the vertex of the helix:

   ![Curve Point Plane Diagram]

   The plane in the image above is a new plane, intersecting the helix vertex and normal to the edge of the helix.
3. Sketch a circle on the plane, using ( ) the helix vertex for the center of the circle:

4. Sweep the circle along the helix (path):

5. Click ✔️.
Creating a plane point plane

1. Create the helix as described above:

2. Create a plane point plane using the helix vertex and a plane.

3. Click ✓.

**Helix: iOS**

Create a helix using a conical or cylindrical face, or circular edge.

**Steps**

1. Tap the Helix tool.
2. Select a Helix type:
   - **Turns** - The number of turns of the helix
   - **Pitch** - The distance traveled axially in each revolution
   - **Height and Turns** - The overall height of the helix and the number of turns of the helix
   - **Height and Pitch** - The overall height of the helix and the distance traveled axially in each revolution
   - **Turns and Pitch** - The number of turns to the helix and the distance traveled axially in each revolution

If you want to create a helix from a **conical** or **cylindrical face**, you can use any of the above Helix types. If you want to create a helix from a **circular edge** or a **circular sketch**, you must select Height and Turns, Height and Pitch, or Turns and Pitch as the Helix type.
3. When using a **circular edge**:
   
a. Select one of: Height and Turns, Height and Pitch, or Turns and Pitch.
   
b. Select Clockwise or Counterclockwise to set the direction of the turns.
   
c. Specify the appropriate values as determined by your selection for step 3a.

   i. Height - The overall height of the helix
   
   ii. Helical pitch - The distance traveled axially in each revolution
   
   iii. Revolutions - The number of revolutions from the circular edge
   
   iv. Start angle - The measurement from the circular edge

4. When selecting a **conical or cylindrical face**:
   
a. Select a Helix type.
   
b. Select Clockwise or Counterclockwise to set the direction of the turns.
   
c. Specify the appropriate values as determined by your selection in step 4a.

   i. Height - The overall height of the helix.
   
   ii. Helical pitch - The distance traveled axially in each revolution
   
   iii. Revolutions - The number of revolutions from the circular edge.
   
   iv. Start angle - The measurement from the circular edge.

5. Tap the checkmark.

**Create a Helix from a conical or cylindrical face**
Create a Helix from a circular sketch

Create a spring from a helix

1. Create a helix from either a conical or cylindrical face or a circular edge, as described above.
If you used a cone or cylinder to make the helix, it helps to hide it. Do so with the hide icon in the feature list.

2. With the Plane tool, create a plane using the plane type **Plane Point**. Select a plane and select a point from either end of the helix.
3. On the new plane, sketch a circle around the point of the helix.

4. Sweep the circle as a solid, using the helix as the sweep path.
5. Tap the checkmark.

Helix: Android

Create a helix using a conical or cylindrical face, or circular edge.

Steps

1. Tap the Helix tool.
2. Select a Helix type:

- **Turns** - The number of turns of the helix
- **Pitch** - The distance traveled axially in each revolution
- **Height and Turns** - The overall height of the helix and the number of turns of the helix
- **Height and Pitch** - The overall height of the helix and the distance traveled axially in each revolution
• **Turns and Pitch** - The number of turns to the helix and the distance traveled axially in each revolution

If you want to create a helix from a **conical** or **cylindrical face**, you can use any of the above Helix types. If you want to create a helix from a **circular edge** or a **circular sketch**, you must select Height and Turns, Height and Pitch, or Turns and Pitch as the Helix type.

3. When using a **circular edge**:
   a. Select one of: Height and Turns, Height and Pitch, or Turns and Pitch.
   b. Select Clockwise or Counterclockwise to set the direction of the turns.
   c. Specify the appropriate values as determined by your selection for step 3a.
      i. **Height** - The overall height of the helix
      ii. **Helical pitch** - The distance traveled axially in each revolution
      iii. **Revolutions** - The number of revolutions from the circular edge
      iv. **Start angle** - The measurement from the circular edge

4. When selecting a **conical or cylindrical face**:
   a. Select a Helix type.
   b. Select Clockwise or Counterclockwise to set the direction of the turns.
   c. Specify the appropriate values as determined by your selection in step 4a.
      i. **Height** - The overall height of the helix.
      ii. **Helical pitch** - The distance traveled axially in each revolution
      iii. **Revolutions** - The number of revolutions from the circular edge.
      iv. **Start angle** - The measurement from the circular edge.

5. Tap the checkmark.

**Create a Helix from a conical or cylindrical face**
Create a Helix from a circular sketch

Create a spring from a helix

1. Create a helix from either a conical or cylindrical face or a circular edge, as described above.

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If you used a cone or cylinder to make the helix, it helps to hide it. Do so with the hide icon in the feature list.

2. With the Plane tool, create a plane using the plane type **Plane Point**. Select a plane and select a point from either end of the helix.
3. On the new plane, sketch a circle around the point of the helix.

4. Sweep the circle as a solid, using the helix as the sweep path.
5. Tap the checkmark.

**Projected Curve**

Create a Curve as the intersection of two projected sketches. Note that the projections from one sketch must intersect the other in one contiguous curve for this operation to succeed. The resulting Curve feature is listed in the Feature list and the curve itself is listed in the Parts list (when selecting a curve, use the Parts list).

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Projected Curve: Desktop**

In the Feature toolbar:
Create a Curve as the intersection of two projected sketches. Note that the projections from one sketch must intersect the other in one contiguous curve for this operation to succeed. The resulting Curve feature is listed in the Feature list and the curve itself is listed in the Parts list (when selecting a curve, use the Parts list).

**Steps**

With two non-parallel sketch curves in the graphic area:

1. Click 🌡️.

2. With focus in the First sketch field, select the First sketch (on the model or in the Feature list).

3. With focus in the Second sketch field, select the Second sketch (on the model or in the Feature list).

4. Click ✅ to accept.

You are not able to show/hide the feature associated with a Curve; use the show/hide functionality in the Parts list instead.

**Example**

The first sketch is selected:
The second sketch is selected and the resulting curve appears:

The curve is listed in the Parts list under Curves:

**Projected Curve: iOS**
Create a Curve as the intersection of two projected sketches. Note that the projections from one sketch must intersect the other in one contiguous curve for this operation to succeed. The resulting Curve feature is listed in the Feature list and the curve itself is listed in the Parts list (when selecting a curve, use the Parts list).

Steps

With two non-parallel sketch curves in the graphic area:

1. Tap the Projected curve tool.

2. Select the First sketch.

3. Select the Second sketch.

4. Tap the checkmark.

You are unable to show/hide the feature associated with a Curve; use the show/hide functionality in the Parts list instead.
Projected Curve: Android

Create a Curve as the intersection of two projected sketches. Note that the projections from one sketch must intersect the other in one contiguous curve for this operation to succeed. The resulting Curve feature is listed in the Feature list and the curve itself is listed in the Parts list (when selecting a curve, use the Parts list).

Steps

With two non-parallel sketch curves in the graphic area:
1. Tap the Projected curve tool.

2. Select the First sketch.

3. Select the Second sketch.

4. Tap the checkmark.

You are unable to show/hide the feature associated with a Curve; use the show/hide functionality in the Parts list instead.
3D Fit Spline

Create a 3D fit spline through a series of vertices. Creates a curve which is listed in the Parts list under Curves.

This functionality is available on Onshape's browser, iOS, and Android platforms.

3D Fit Spline: Desktop

In the Feature toolbar:

Create a 3D fit spline through a series of vertices or edges that are tangential continuous. Creates a curve which is listed in the Parts list under Curves.

Steps
1. Click  

2. Click to select whether to use vertices, or edges at the top of the dialog.

   a. For splines using vertices:
      i. Click to select vertices along which to create the 3D fit spline.
      ii. Optionally, check the box to create a closed spline.
      iii. If the spline is not closed, you have the option to click a line or a Mate connector (inferred or existing) to select a **Start direction**.
      iv. Enter a value or click and drag the directional arrow in the graphics area to adjust the **Start magnitude**.
      v. Optionally, click to select **Match curvature at start** to match the curvature of the edge or face selected as the start direction.
      vi. Optionally, click to select **Match curvature at end** to match the curvature of the edge or face selected as the end direction.
      vii. Optionally, click a line or Mate connector (inferred or existing) to select an **end direction**.
      viii. Enter a value or click and drag the directional arrow in the graphics area to adjust the **End magnitude**.
      ix. Optionally, click to select **Match curvature at end** to match the curvature of the edge or face selected as the end direction.
b. For splines using edges, click to select tangentially continuous edges with which to create the 3D fit spline.

3. Click ✅.

A curve is created, listed under Curves in the Parts list. You are not able to show/hide the 3D Fit Spline feature; use the show/hide functionality in the Curves (Parts) list instead.

Example of a spline using edges

1. Click ✗.

2. Select edges (must be tangentially continuous) to create a 3D fit spline.

3. Check the box to create the spline.

4. Click ✅.
Example of a closed spline

1. Click □.

2. Select vertices to create a 3D fit spline.

3. Check the box to create a closed spline.

4. Click ✔.
Matching curvature

1. Click ⬇️.

2. Select vertices to create a 3D fit spline:

3. Select Start and End directions (highlighted edges):
4. Select *Match curvature at start* and *Match curvature at end*:

![Diagram of 3D fit spline](image)

5. Click ✅.

**3D Fit Spline: iOS**

Create a 3D fit spline through a series of vertices, or edges that are tangential continuous. Creates a curve which is listed in the Parts list under Curves.

**Steps**

1. Tap the 3D fit spline tool.
2. Tap to select whether to use vertices, or edges at the top of the dialog.

   a. For splines using vertices:
      
      i. Tap to select vertices along which to create the 3D fit spline.

      ii. Optionally, check the box to create a closed spline.

      iii. If the spline is not closed, you have the option to click a line or a Mate connector (inferred or existing) to select a **Start direction**.

      iv. Enter a value or drag the directional arrow in the graphics area to adjust the **Start magnitude**.
v. Optionally, select **Match curvature at start** to match the curvature of the edge or face selected as the start direction.

vi. Optionally, select **Match curvature at end** to match the curvature of the edge or face selected as the end direction.

vii. Optionally, select a line or Mate connector (inferred or existing) to select an **end direction**.

viii. Enter a value or drag the directional arrow in the graphics area to adjust the **End magnitude**.

ix. Optionally, select **Match curvature at end** to match the curvature of the edge or face selected as the end direction.

   b. For splines using edges, select tangentially continuous edges with which to create the 3D fit spline.

3. Tap the checkmark.

A curve is created, listed under Curves in the Parts list. You are unable to show/hide the 3D Fit Spline feature; use the show/hide functionality in the Curves (Parts) list instead.
Example of a closed spline

1. Tap the 3D fit spline tool.

2. Select vertices to create a 3d fit spline.

3. Check the box to create a closed spline.

4. Tap the checkmark.

3D Fit Spline: Android

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Create a 3D fit spline through a series of vertices. Creates a curve which is listed in the Parts list under Curves.

**Steps**

1. Tap the 3D fit spline tool.
Select at least two vertices to make a spline.

**Vertices**  **Edges**

**Vertices**
No entities selected

- **Closed spline**
- **Start direction**
  No entities selected
  - **Start magnitude**
    - 1.0
- **Opposite direction**
- **Match curvature at start**
- **End direction**
  No entities selected
  - **End magnitude**
    - 1.0
- **Opposite direction**
- **Match curvature at end**
2. Tap to select whether to use vertices, or edges at the top of the dialog.
   a. For splines using vertices:
      i. Tap to select vertices along which to create the 3D fit spline.
      ii. Optionally, check the box to create a closed spline.
      iii. If the spline is not closed, you have the option to click a line or a Mate connector (inferred or existing) to select a **Start direction**.
      iv. Enter a value or drag the directional arrow in the graphics area to adjust the **Start magnitude**.
      v. Optionally, select **Match curvature at start** to match the curvature of the edge or face selected as the start direction.
      vi. Optionally, select **Match curvature at end** to match the curvature of the edge or face selected as the end direction.
      vii. Optionally, select a line or Mate connector (inferred or existing) to select an **end direction**.
      viii. Enter a value or drag the directional arrow in the graphics area to adjust the **End magnitude**.
      ix. Optionally, select **Match curvature at end** to match the curvature of the edge or face selected as the end direction.
b. For splines using edges, select tangentially continuous edges with which to create the 3D fit spline.

3. Tap the checkmark.

A curve is created, listed under Curves in the Parts list. You are unable to show/hide the 3D Fit Spline feature; use the show/hide functionality in the Curves (Parts) list instead.

**Example of a closed spline**

1. Tap the 3D fit spline tool.

2. Select vertices to create a 3d fit spline.

3. Check the box to create a closed spline.

4. Tap the checkmark.
**Bridging Curve**

Create a Curve connecting any two points or vertices. The resulting Curve is listed in the Feature list and the Parts list.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Bridging Curve: Desktop**

In the Feature toolbar:

Create a Curve connecting any two points or vertices. The resulting Curve is listed in the Feature list and the Parts list.

**Steps**

With at least two sketch points or vertices in the graphic area:
1. Click 🔄.

2. Select a point or vertex to act as the First side.

3. Select a point or vertex to act as the Second side.

4. Choose for each point or vertex, select a continuity definition:
   a. **Match tangent** - The curve will end at the vertex and be tangent to the edge
   b. **Match position** - The curve will end at the vertex
   c. **Match curvature** - The curve will match the curvature of the guide curve in addition to the tangency.

5. When Match tangent is selected for at least one vertex or point, you have the option to specify a **Magnitude**, any positive number (defaults to 1), to further define the shape of the curve. Magnitude is a scaling factor applied to Onshape’s default calculation. The closer you get to zero the straighter the line. The larger the Magnitude the faster the curve 'shoots out of' the selected point or vertex.

6. When Match tangent is selected for two or more vertices or points, you have the option to specify a **Bias** value between 1.000e-4 and 0.999. This weights the tangency toward one side of the curve or the other. A value of 0.5 weights the tangency equally. This determines whether the curve 'shoots out more' from one side (closer to 0) or the other side (closer to 1).

7. Click ✅ to accept.

**Example**
In addition to entering a numeric value for **Magnitude** you are able to drag the arrow to adjust the Magnitude of the curve as shown below on the left corner of the new curve:

In addition to entering a numeric value for **Bias** you are able to drag the circle to adjust the Bias of the curve as shown below, affecting the left side of the curve:
**Tips**

- You are unable to show/hide the feature associated with a Curve; use the show/hide functionality in the Parts list instead.

- If you have chosen Match position then you do not need to select an edge.

- If you select a vertex for one of the lists and there is only one edge coming out of the vertex then you do not need to select the edge.

- If you select a vertex and more than one edge comes out of it then you need to select an edge.

- If you select a vertex and an edge then the vertex must be at one end of the edge (but doesn't have to be a vertex of the edge, just in the same place).
You may also select an edge but no vertex. If you do that, Onshape will try to work out which vertex you mean based on the selection(s) for the other side. Override Onshape by selecting an edge.

**Bridging Curve: iOS**

Create a Curve connecting any two points or vertices. The resulting Curve is listed in the Feature list and the Parts list.

**Steps**

With at least two sketch points or vertices in the graphic area:

1. Tap the Bridging curve tool.

2. Select a point or vertex to act as the First side.
3. Select a point or vertex to act as the Second side.

4. Choose either:

a. Match tangent - The curve will end at the vertex and be tangent to the edge

b. Match position - The curve will end at the vertex

5. Specify a Magnitude, any positive number (Defaults to 1.) - A scaling factor applied to Onshape's default calculation. The closer you get to zero the straighter the line. The larger the magnitude the faster the curve 'shoots out of' the selected edge.

6. Specify a Bias between 1.000e-4 and 0.999 - Determines whether the curve 'shoots out more' from one side (closer to 0) or the other side (closer to 1).

7. Tap the checkmark.

You are unable to show/hide the feature associated with a Curve; use the show/hide functionality in the Parts list instead.
Tips

- If you have chosen Match position then you do not need to select an edge.
- If you select a vertex for one of the lists and there is only one edge coming out of the vertex then you do not need to select the edge.
- If you select a vertex and more than one edge comes out of it then you need to select an edge.
- If you select a vertex and an edge then the vertex must be at one end of the edge (but doesn't have to be a vertex of the edge, just in the same place).
- You may also select an edge but no vertex. If you do that, Onshape will try to work out which vertex you mean based on the selection(s) for the other side. Override Onshape by selecting an edge.

Bridging Curve: Android

Create a Curve connecting any two points or vertices. The resulting Curve is listed in the Feature list and the Parts list.

Steps

With at least two sketch points or vertices in the graphic area:

1. Tap the Bridging curve tool.
Bridging curve 1

Select a single vertex for both sides of the bridge.

First side
No entities selected

Match position  Match tangent

Second side
No entities selected

Match position  Match tangent

Magnitude  1.0

Bridging curve 1

Select a single vertex for both sides of the bridge.

Second side
No entities selected

Match position  Match tangent

Magnitude  1.0

Bias  0.5
2. Select a point or vertex to act as the First side.

3. Select a point or vertex to act as the Second side.

4. Choose either:
   a. Match tangent - The curve will end at the vertex and be tangent to the edge
   b. Match position - The curve will end at the vertex

5. Specify a Magnitude, any positive number (Defaults to 1.) - A scaling factor applied to Onshape's default calculation. The closer you get to zero the straighter the line. The larger the magnitude the faster the curve 'shoots out of' the selected edge.

6. Specify a Bias between 1.000e-4 and 0.999 - Determines whether the curve 'shoots out more' from one side (closer to 0) or the other side (closer to 1).

7. Tap the checkmark.

You are unable to show/hide the feature associated with a Curve; use the show/hide functionality in the Parts list instead.
**Tips**

- If you have chosen Match position then you do not need to select an edge.
- If you select a vertex for one of the lists and there is only one edge coming out of the vertex then you do not need to select the edge.
- If you select a vertex and more than one edge comes out of it then you need to select an edge.
- If you select a vertex and an edge then the vertex must be at one end of the edge (but doesn't have to be a vertex of the edge, just in the same place).
- You may also select an edge but no vertex. If you do that, Onshape will try to work out which vertex you mean based on the selection(s) for the other side. Override Onshape by selecting an edge.
Represent multiple edges as one Curve. Select multiple adjacent edges, sketch entities and other curves. Selecting non-contiguous edges may result in multiple Curves created. Selections for each Curve must meet at their vertices. (Curves are listed in the Parts > Curves list.)

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Composite Curve: Desktop**

This functionality is also available on Onshape’s iOS and Android platforms.

In the Feature toolbar:

Represent multiple edges as one Curve. Select multiple adjacent edges, sketch entities and other curves. Selecting non-contiguous edges may result in multiple Curves created. Selections for each Curve must meet at their vertices. (Curves are listed in the Parts > Curves list.)

**Steps**

With two or more contiguous sketch curves in the graphics area:

1. Click 🔄.
2. Select each sketch curve. (Curves must not intersect or overlap.)

3. Click ✅ to accept.

Selecting non-contiguous edges can result in more than one Curve being created:

You are not able to show/hide the Composite Curve feature; use the show/hide functionality in the Parts (Curves) list instead.

**Using Composite curves**

To use a Composite Curve in a feature tool like Loft, select the Curve in the Curves (Parts) list as the profile:
Composite Curve: iOS

Represent multiple edges as one Curve. Select multiple adjacent edges, sketch entities and other curves. Selecting non-contiguous edges may result in multiple Curves created. Selections for each Curve must meet at their vertices. (Curves are listed in the Parts > Curves list.)

Steps

With two or more contiguous sketch curves in the graphics area:

1. Tap the Composite curve tool.

2. Select each sketch curve. (Curves must not intersect or overlap.)

3. Tap the checkmark.
Selecting non-contiguous edges can result in more than one Curve being created:

You are unable to show/hide the Composite Curve feature; use the show/hide functionality in the Parts (Curves) list instead.

**Using Composite curves**

To use a Composite Curve in a feature tool like Loft, select the Curve in the Curves (Parts) list as the profile:
Composite Curve: Android

Represent multiple edges as one Curve. Select multiple adjacent edges, sketch entities and other curves. Selecting non-contiguous edges may result in multiple Curves created. Selections for each Curve must meet at their vertices. (Curves are listed in the Parts > Curves list.)

Steps

With two or more contiguous sketch curves in the graphics area:

1. Tap the Composite curve tool.
2. Select each sketch curve. (Curves must not intersect or overlap.)

3. Tap the checkmark.

Selecting non-contiguous edges can result in more than one Curve being created:

You are unable to show/hide the Composite Curve feature; use the show/hide functionality in the Parts (Curves) list instead.

**Using Composite curves**

To use a Composite Curve in a feature tool like Loft, select the Curve in the Curves (Parts) list as the profile:
Mate Connector

Mate connectors are local coordinate system entities located on or between parts or surfaces and used within a mate to locate and orient part or surface instances with respect to each other. You can also use mate connectors to create planes.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Mate Connector: Desktop

Shortcut: Ctrl-m

Shortcut: k (to show/hide Mate connectors)

In the Feature toolbar:
In the Assembly toolbar:

Mate connectors are local coordinate system entities located on or between parts or surfaces and used within a Mate to locate and orient part or surfaces instances with respect to each other.

Two part or surface instances are positioned in an assembly by creating a Mate. The two instances are positioned by aligning a Mate connector defined on one instance with a Mate connector defined on the other instance.

Mate connectors are able to be used to define an axis for Features requiring the selection of an axis. The Z axis of the Mate connector is used.

Mate connectors can be defined explicitly or implicitly:

- **Explicit mate connectors:**
  - Define using the Mate connector tool in the toolbar in an Assembly or in a Part Studio
  - Listed at the highest level in the Feature list
  - Can be selected from the Feature list during the creation of a mate

- **Implicit mate connectors:**
  - Define while creating a Mate, in an Assembly and some features in a Part Studio
  - Listed at a sub-level in the Feature list in an Assembly, beneath the Mate in which it is used
  - Can not be selected from the Feature list during the creation of a Mate

When a Mate connector is used in more than one Mate, it is listed only once in the Feature list, with the first Mate that uses it (if implicit) or as its own original top-level feature in the list.

To learn more about Mates, see "Mates" on page 1544. To learn about Mates and Mate Connectors watch the video below.
Use the shortcut key k to hide/show Mate connectors.

**Steps**

1. Click 🕒.

![Mate connector 1](image.png)

2. Choose between creating a mate connector on a part (entity) or between parts:
   - **On entity** - Create a Mate connector on a part:

![Part with Mate connector](image.png)

   - **Between entities** - Create a Mate connector halfway between two entities on
3. Select a point on the part for the Mate connector:
   - Roll over any face to activate the potential Mate connectors and select a point.
   - Or click anywhere on a face to automatically place the Mate connector at the centroid point.

4. Specify options, if desired (as shown in options examples below).

5. Click □.

**Visualize Mate connector points**

With the Mate connector dialog open, moving the cursor over a part 'wakes up' default inference points and the inference point closest to the cursor highlights as a Mate connector. As you continue to mouse over the part, different default inference points appear.

To lock Mate inferences when you see the one you want to select, depress the Shift key when mousing.

Each face and edge of a part has default inference points:
   - At the centroid
   - At the midpoints
   - At the corners

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At the centroids of any region contained in a planar face (for example, holes and slots)

Before the default Mate connector is highlighted at the centroid (seen above), you might see the centroid point icon (seen below):

For cylindrical faces, inference points appear on the axis of the cylindrical and partial cylindrical face:
Select a planar face that has a partial cylindrical edge and the Mate connector inference points include the centroid of the axis:

Hover over the edge of the partial cylindrical face and the default Mate connector appears at the centroid of the axis:
To zero in on a specific inferenced point or default Mate connector without waking up others as you move the cursor, use the SHIFT key to prevent other Mate connectors from appearing.

**Realign Mate connectors**

Check Realign to change the orientation of the Mate connector along a primary and (optionally) a secondary axis.

- **Realign**

**Move Mate connectors**

- **Move** - Move the Mate connector a specified distance in a specified direction. The fields are presented in this order:
  - X translation
  - Y translation
  - Z translation
  - You are also able to use the Rotate field to specify a rotation of a specified number of degrees.

Flip primary axis of Mate connectors

Flip the primary axis 180 degrees.
Reorient secondary axis of Mate connectors

Move the primary axis one quadrant at a time through the X/Y coordinates.

**Inference points and defaults**

The inference points for potential Mate connectors available when you select an edge or face are:

- **Planar face** - Parallel to the face at every vertex, arc center, edge midpoint, and the face centroid
- **Cylindrical face** - Perpendicular to the face axis at the middle and ends
- **Linear edge or sketch line** - Perpendicular to the line at the middle and ends
- **Circular edge or sketch circle** - Perpendicular to the line at the middle and ends

**Hide and show Mate connectors**

Once created, you are able to hide or show Mate connectors in both Part Studios and Assemblies:

- Use the context menu in the Feature list (Hide, Hide other mate connectors/Show, Show all mate connectors) - Hide other mate connectors hides all mate connectors but the one you have selected.
- Use the icon in the Feature list to hide a specific mate connector.
- Hiding/showing mate connectors in a Part Studio or Assembly is exclusive to the Part Studio or Assembly. Mate connectors hidden in a Part Studio are visible when inserted into the Assembly. You have the ability to view Mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

**Tips**
• If the behavior is not what you expected, try flipping the primary and/or secondary axis on the Mate connector.

• Use the Shift key to keep the Mate connectors you want visible as you move the pointer to select one. This can be useful when the inferred point for potential Mate connector you want is on or near an edge.

• All Mate connectors are listed in the Feature list; you are able to hide/show them, edit and adjust, change, and use different orientations of the connectors.

A Mate connector may be created in both the Assembly and the Part Studio. Creating a Mate connector in the Part Studio has two advantages:

• You are able to reference sketch entities in the Part Studio. This gives you the ability to define the Mate connector in more positions than are possible in an Assembly.

• A Mate connector defined in a Part Studio is available for reuse on every instance of that part in every assembly in which it is instanced.

When creating a Mate connector in the Part Studio, there is an additional option in the Mate connector dialog called Owner Part.

Mate connector dialog in Assembly

Mate connector dialog in Part Studio

In a Part Studio with more than one part, it is sometimes unclear which part owns the Mate connector. Use Owner Part to specify which part owns the Mate connector.

Mate Connector: iOS
Mate connectors are local coordinate system entities located on or between parts or surfaces and used within a mate to locate and orient part instances with respect to each other.

**Steps**

1. Tap Mate connector tool.

2. Select origin type:
   - **On entity** - Create a Mate connector on a part.
   - **Between entities** - Create a Mate connector halfway between two entities on the part or surface.

3. Select origin entity.
4. Optionally, toggle to realign.
   If you do realign, select primary and secondary axis entities.

5. Optionally, toggle to move.
   If you do move, specify X,Y, and Z values.
   Also select rotation axis and specify rotation angle.


7. Tap the checkmark.

**Visualize Mate connector points**

Each *face and edge* of parts and surfaces have default inference points:

- At the centroid
- At the midpoints
- At the corners
For **cylindrical faces**, inference points appear on the axis of the cylindrical and partial cylindrical face:
Select or hover over a planar face that has a **partial cylindrical edge** and the Mate connector inference points include the centroid of the axis:

Select or hover over the edge of the partial cylindrical face and default Mate connector inference points appear:
On entity

Select the origin entity then select the Mate connector inference point on which to place the Mate connector.
Between entities

Select an origin entity and the Mate connector inference point in line with where you want to place the Mate connector. Select another entity and the Mate connector is placed in line with the Mate connector inference point, between the two entities you selected.
Realign Mate connectors

Toggle on Realign to change the orientation of the Mate connector along a primary and (optionally) a secondary axis.

Select a primary and secondary axis along which to realign the Mate connector.
Move Mate connectors

Move the Mate connector to a specified distance in a specified direction. The fields are presented in this order:

- X translation
- Y translation
- Z translation

You are also able to use a rotation axis and specify a number of degrees to rotate the Mate connector.
Hide and show Mate connectors

Once created, you are able to hide or show Mate connectors in both Part Studios and Assemblies:

- Use the context menu in the Feature list (Hide, Hide other Mate connectors/Show, Show all Mate connectors) - Hide other Mate connectors hides all Mate connectors but the one you have selected.
- Use the Hide/Show icon (the eye icon) in the Feature list to hide a specific Mate connector.
- Hiding/showing Mate connectors in a Part Studio or Assembly is exclusive to the Part Studio or Assembly. Mate connectors hidden in a Part Studio are visible when inserted into the Assembly. You are able to view Mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

Mate Connector: Android

Mate connectors are local coordinate system entities located on or between parts and surfaces and used within a mate to locate and orient part and surface instances with respect to each other.

Steps

1. Tap Mate connector tool.
2. Select origin type:
   - **On entity** - Create a Mate connector on a part or surface
   - **Between entities** - Create a Mate connector halfway between two entities on the part or surface.

3. Select origin entity.

4. Optionally, toggle to realign.
   - If you do realign, select primary and secondary axis entities.
5. Optionally, toggle to move.
   If you do move, specify X, Y, and Z values.
   Also select rotation axis and specify rotation angle.


7. Tap the checkmark.

**Visualize Mate connector points**

Each **face and edge** of a part has default inference points:

- At the centroid
- At the midpoints
- At the corners
For **cylindrical faces**, inference points appear on the axis of the cylindrical and partial cylindrical face:
Select or hover over a planar face that has a **partial cylindrical edge** and the Mate connector inference points include the centroid of the axis:

Select or hover over the edge of the partial cylindrical face and default mate connector inference points appear:
Select the origin entity then select the Mate connector inference point on which to place the mate connector.
Between entities

Select an origin entity and the Mate connector inference point in line with where you want to place the Mate connector. Select another entity and the Mate connector is placed in line with the Mate connector inference point, between the two entities you selected.
Realign Mate connectors

Toggle on Realign to change the orientation of the Mate connector along a primary and (optionally) a secondary axis.

Select a primary and secondary axis along which to realign the Mate connector.
**Move Mate connectors**

Move the Mate connector to a specified distance in a specified direction. The fields are presented in this order:

- X translation
- Y translation
- Z translation

You are also able to use a rotation axis and specify a number of degrees to rotate the Mate connector.
Hide and show Mate connectors

Once created, you are able to hide or show Mate connectors in both Part Studios and Assemblies:

- Use the context menu in the Feature list (Hide, Hide other Mate connectors/Show, Show all Mate connectors) - Hide other Mate connectors hides all Mate connectors but the one you have selected.
- Use the Hide/Show icon (the eye icon) in the Feature list to hide a specific Mate connector.
- Hiding/showing Mate connectors in a Part Studio or Assembly is exclusive to the Part Studio or Assembly. Mate connectors hidden in a Part Studio are visible when inserted into the Assembly. You are able to view Mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

Derived

Insert parts, sketches, surfaces, helices, planes, or Mate connectors from one Part Studio into another in the same or a different document (thereby linking the documents), with an associative link. You also have the ability to insert these entities into a Part Studio from a different version of the same document.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Derived: Desktop

In the Feature toolbar:

Insert parts, sketches, surfaces, helices, planes, or Mate connectors from one Part Studio into another in the same or a different document (thereby linking the documents),
with an associative link. You also have the ability to insert these entities into a Part Studio from a different version of the same document.

Steps

1. While in a Part Studio, click 📷.

   A list of Part Studios in this document and their features appears.

   If the list is lengthy, use the Search box to search for a Part Studio or feature by name.

2. You are able to select Current document and derive from a different version, or you may select Other documents and select from their features and parts (shown below).
You can select from other documents only if that document has one or more versions. A notice is displayed regarding the state of the document: if no version exists or if a newer version exists, with an option to create a version immediately if necessary. See Linking Documents for more information.
3. Use the filters and search bar to find and select a document.
   i. You have the ability to search by part/assembly name and other properties, including Custom properties.
   ii. Search accepts multiple words as well as non-alphanumeric characters, such as punctuation.
   iii. The filter to the left of the Search bar shows common properties (or all properties) to search by. You are able to use more than one filter at a time.
   iv. Each search result indicates which type of entity fulfills the search criteria by the icon preceding the name, followed by the name of the entity. Below that, the workspace or version icon, the document name and workspace name (or version name, when part of the search criteria), the part number, and the release management state (shown below).

![Part Icon](image)

4. Select one or many features of that document.

5. Click ✅.

If you have derived a part (or sketch) from another document, a linked icon appears next to the Derived feature in the Feature list to indicate that it is linked to another document.

🔗

When a newer version of the document from which you derived the part is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab.

🔗

See [Linking Documents](#) for more information on linked documents and how to update them.

**Inserting released derived parts**

When inserting a part derived from another Part Studio, you have the option to search
for parts that have been released. The icons in the second row of the dialog (below the Other documents filter) can also help you find the parts you are looking for:

![Derived 1](image)

*From left to right, filter for released parts, create a new version, view the version graph*

Use the ▲ icon to find parts that have been released (within the selected document):

![Derived 1](image)

When you have inserted a released part for a derived part, the feature icon represents that it is a derived feature and the release icon indicates it is a released part:

![Features (5)](image)

**Tips**

- You are not able to select a derived feature for insertion more than once in the same operation. You are, however, able to reopen the Derived dialog and insert
the same derived feature an additional time. For example, if you want two of the same part in the target Part Studio, you must select the part once, close the Derived dialog, then reopen the dialog and select the part a second time.

- You can swap out one derived feature for another one, even if the second derived feature is in a different Part Studio:
  - In the target Part Studio, simply double-click the Derived feature to open the Derived dialog. (Or right-click on the feature in the Feature list and select Edit.)
  - Select a new document, if necessary, or Part Studio and then the feature to insert.
  - Click the checkmark to accept your selections and close the dialog.
  - Note that subsequent features may fail - check your Part Studio and make any necessary adjustments.

- After selecting a sketch you can use that sketch to perform an extrude in the target Part Studio. In the parent Part Studio, when you make a change to the sketch, such as a dimension, the change is reflected in the target Part Studio. You are able to use the sketch in many Part Studios as a derived feature. Then in each Part Studio, continue with varied designs.

- Derived features have a one-way correspondence: from the parent Part Studio to the target Part Studio. When you change the feature in the parent Part Studio, the change is reflected in the target Part Studio, but not vice versa.

- This feature does not accept circular references. For example, when inserting a feature from Part Studio A to Part Studio B, you cannot insert any feature from Part Studio B to Part Studio A. The operation will fail due to the circular reference attempted from Part Studio A to B to A again.
The visibility setting of a part is irrelevant when inserting a derived part - derived parts are always visible when they are inserted, regardless of what their setting is in the original Part Studio (workspace or version).

**Derived: iOS**

Insert parts, sketches, surfaces, helices, planes, or Mate connectors from one Part Studio into another in the same or a different document (thereby linking the documents), with an associative link. You also have the ability to insert these entities into a Part Studio from a different version of the same document.

**Steps**

1. While in a Part Studio, select the Derived tool.

   ![Derived tool interface](image)

   A list of Part Studios and their features appears. If the list is lengthy, use the search box to search for a Part Studio or feature by name.

2. Tap to select from the list.
3. Optionally, select **Browse documents** to view Part Studios of other documents. Use the filters to find and select a document, and then select one or many features of that document.

You have the ability to select from other documents only if that document has one or more versions. A notice is displayed regarding the state of the document if no version exists or if a newer version exists. If no version exists, tap to create a version in that document.

4. Tap the checkmark.

If you derived a part, it is listed in the Parts list (under the Features list) where you can toggle the eye icon to hide/show that part.

If you derived a sketch, you can toggle the eye icon next to the derived feature (in the Features list) to hide/show that sketch.

If you have derived a part (or sketch) from another document, a linked icon appears next to the Derived feature in the Feature list to indicate that it is linked to another document.

When a newer version of the document from which you derived the part is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab.

See [Linking Documents](#) for more information on linked documents and how to update them.

**Inserting released derived parts**

When inserting a part derived from another Part Studio, you have the option to search for an insert parts that have been released. The icons in the second row of the dialog (below the Other documents filter) can also help you find the parts you are looking for:
Release icon filter for released parts

Use the ▲ icon to find parts that have been released (within the selected document):

When you have inserted a released part for a derived part, the feature icon represents that it is a derived feature and the release icon indicates it is a released part:

Tips
You have the ability to insert a derived feature from only one parent Part Studio at a time. Open the Derived dialog again to select from an additional Part Studio.

You are unable to select a derived feature for insertion more than once in the same operation. You are able to reopen the Derived dialog and insert the same derived feature an additional time. For example, if you want two of the same part in the target Part Studio, you must select the part once, close the Derived dialog, then reopen the dialog and select the part a second time.

Derived features have a one-way correspondence: from the parent Part Studio to the target Part Studio. When you change the feature in the parent Part Studio, the change is reflected in the target Part Studio, but not vice versa.

This feature does not accept circular references. For example, you are unable to insert a feature from Part Studio A to Part Studio B and then to Part Studio A again, the operation will fail.

The visibility setting of a part is irrelevant when inserting a derived part - derived parts are always visible when they are inserted, regardless of what their setting is in the original Part Studio (workspace or version).

**Derived: Android**

Insert parts, sketches, surfaces, helices, planes, or Mate connectors from one Part Studio into another in the same or a different document (thereby [linking the documents](#)), with an associative link. You also have the ability to insert these entities into a Part Studio from a different version of the same document.

**Steps**

1. While in a Part Studio, select the Derived tool.
A list of Part Studios and their features appears. If the list is lengthy, use the search box to search for a Part Studio or feature by name.

2. Tap to select from the list.

3. Optionally, select **Browse documents** to view Part Studios of other documents. Use the filters to find and select a document, and then select one or many features of that document.

You have the ability to select from other documents only if that document has one or more versions. A notice is displayed regarding the state of the document.
If no version exists or if a newer version exists, if no version exists, tap to create version in that document.

4. Tap the checkmark.

If you derived a part, it is listed in the Parts list (under the Features list) where you are able to toggle the eye icon to hide/show that part.

If you derived a sketch, you have the ability to toggle the eye icon next to the derived feature (in the Features list) to hide/show that sketch.

If you have derived a part (or sketch) from another document, a linked icon appears next to the Derived feature in the Feature list to indicate that it is linked to another document.

When a newer version of the document from which you derived the part is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab.

See Linking Documents for more information on linked documents and how to update them.

**Inserting released derived parts**

When inserting a part derived from another Part Studio, you have the option to search for an insert parts that have been released. The icons in the second row of the dialog (below the Other documents filter) can also help you find the parts you are looking for:

![Derived 2 filter](image)

*Release icon filter for released parts*
Use the ▲ icon to find parts that have been released (within the selected document):

When you have inserted a released part for a derived part, the feature icon represents that it is a derived feature and the release icon indicates it is a released part:

Tips

● You have the ability to insert a derived feature from only one parent Part Studio at a time. Open the Derived dialog again to select from an additional Part Studio.
You are unable to select a derived feature for insertion more than once in the same operation. You are able to reopen the Derived dialog and insert the same derived feature an additional time. For example, if you want two of the same part in the target Part Studio, you must select the part once, close the Derived dialog, then reopen the dialog and select the part a second time.

Derived features have a one-way correspondence: from the parent Part Studio to the target Part Studio. When you change the feature in the parent Part Studio, the change is reflected in the target Part Studio, but not vice versa.

This feature does not accept circular references. For example, you are not able to insert a feature from Part Studio A to Part Studio B and then to Part Studio A again, the operation will fail.

The visibility setting of a part is irrelevant when inserting a derived part - derived parts are always visible when they are inserted, regardless of what their setting is in the original Part Studio (workspace or version).

Variable

(x)

Create a variable for use in expressions in a Part Studio, and assign a value. Use the variable in dimensions and expressions. Variables are features in Onshape, so the placement in the Feature list is important. Create a variable before the feature in which you will use it or create it on the fly during an operation that needs the value.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Variable: Desktop

In the Feature toolbar:
Create a variable for use in expressions in a Part Studio, and assign a value. Use the variable in dimensions and expressions. Variables are features in Onshape, so the placement in the Feature list is important. Create a variable before the feature in which you will use it.

**Steps**

1. While in a Part Studio, click \( x \).

2. In the dialog:
   a. Enter a name for the variable (and by which to reference it).

   Use only English letters and numbers in the name (at least one English letter followed by letters and/or numbers). Variable names are case-sensitive.

   b. Select a type:
      - Length - A numeric value representing a length (decimal, integer, fraction)
      - Angle - A numeric value representing an angle (decimal, integer, fraction)
      - Number - A numeric value (decimal, integer, fraction)
      - Any - Any of the above, a numeric value with different units, or a FeatureScript value such as boolean, map, array, string, or a function. See https://cad.onshape.com/FsDoc/variables.htm#standard-types and examples below.

   c. Enter a value (and optionally, units for Length, Angle, and Any).

3. Click \( \checkmark \).

**Variables in dimensions**

Create a dimension, in the dimension field enter \# and the variable name (and optionally, as part of an expression, as shown below):
Save the dimension; the variable is replaced with the value and the expression (if applicable) is solved:

When you double-click the dimension for editing, the variable (and expression) is displayed:

Variable in a solid part feature

Use variables anywhere you use expressions in a Part Studio. For example, in an extrude or revolve operation.

Start the operation as usual (in this case, Revolve); in the numeric value field, enter # and the variable name (or optionally, as part of an expression):
Accept the feature.

When you edit the feature, the solution is displayed in the numeric value field:

Click in the field and the variable (and expression, if applicable) is displayed.

**Variable autofill feature**

When you have defined variables, entering a hashtag (or pound sign, underscore, or any letter) in a numeric dialog field opens the variable autofill feature as shown below:
The dialog on the left is an Extrude dialog with the Depth field active; the box on the right is a dimension field in a sketch.

You can keep typing to filter the list to reasonable options, use the arrow keys to move up and down in the list, and use the Enter key to accept a variable from the list when it is highlighted. You can also use the mouse scroll wheel to scroll the variable list and a mouse click to select a particular variable. Note that FeatureScript functions are also included in the list when the pound sign is not used:

Also note the following:
- Using the Escape key closes the dropdown without selecting anything. Removing focus from the dropdown also closes it without selecting anything.
- Compute values are shown in light gray, and will not be inserted (only the variable or function name is inserted). Likewise with function inputs, because the input placeholder names are not inserted.
- Only variables that exist in the current context are shown; variables further down in the Feature list are not shown.

**Using Arrays in Variables**

Variable values may contain expressions. You are able to specify an array with an index, and the index can be a variable. This allows you to change the value of the variable by changing the value of the index variable.

To use arrays in variables, you must first set up a zero-based index.

1. Create a variable and set the name to ‘config’. Set the value of #config to ‘2’.
2. Create a variable and set the name to ‘diameter’. Set the value of #diameter to 
   \[0.25, 0.5, 1][#config]\].
   
   It is the second set of brackets [#config] that serves as the index pointer in the array.

3. Create a variable and set the name to ‘length’. Set the value of #length to \[2, 4, 10][#config]\].
4. Create a circle.
5. Create a line.
6. Set the diameter of the circle to ‘#diameter’.
   
   Since #config = 2, the diameter of the circle is 1.
7. Set the length of the line to ‘#length’.
   
   Since #config = 2, the length of the line is 10.

Change the value of the index variable to change the indices of all array variables.

**Creating Variables on the fly**

You can create variables as you need them, when entering a numeric value into a
field for example, and the variable is saved preceding the current operation in the Feature list.

For example, when creating a sketch, when the dimension field pops up, instead of entering a value for the dimension, you can type the pound sign # and the list of existing variables is displayed, along with an option for a New variable:

Select New variable to open the Variable dialog, to create a new variable and use it in the current field. (Otherwise, select one of the existing variables, if available.)

The Create variable dialog gives the option of creating a variable for a Feature (default) or a Configuration:
Enter a name for the new variable, as well as a value. Click the check mark to save the new variable and to use it in the open feature.

**FeatureScript functions in Variables**

You are able to use FeatureScript functions in a variable, following the FeatureScript syntax.

For example, you might create a variable of type Any, named “Adjust”, to store a function that doubles a given length and adds 2.5mm, as follows:

```plaintext
function(len) { return len * 2 + 2.5 mm; }
```

and then reference that variable in an expression, such as:

```plaintext
#Adjust(20mm)
```
Tips

- When you change the value of a variable (edit it as you would any feature), all operations that use the variable are automatically updated.
- When you change the name of a variable, you have the choice to propagate the change everywhere the variable is used:

Place a check next to Update all references.

**Variable: iOS**

Create a variable for use in expressions in a Part Studio, and assign a value. Use the variable in dimensions and expressions. Variables are features in Onshape, so the placement in the Feature list is important. Create a variable before the feature in which you will use it.

**Steps**

1. While in a Part Studio, select the Variable tool.
2. In the dialog:

a. Enter a name for the variable (and by which to reference it).

   Use only English letters and numbers in the name (at least one English letter followed by letters and/or numbers). Variable names are case-sensitive.

b. Select a type:
   - Length - A numeric value representing a length (decimal, integer, fraction)
   - Angle - A numeric value representing an angle (decimal, integer, fraction)
   - Number - A numeric value (decimal, integer, fraction)
   - Any - Any of the above, a numeric value with different units, or a FeatureScript value such as boolean, map, array, string, or a function. See https://cad.onshape.com/FsDoc/variables.htm#standard-types and examples below.

c. Enter a value (and optionally, units for Length, Angle, and Any).

3. Tap the checkmark.

The variable you created is listed in the Feature List and displays the name and value of the variable.

**Variables in dimensions**

Create a dimension, in the dimension field enter # and the variable name (and
optionally, as part of an **expression**, as shown below):

Save the dimension; the variable is replaced with the value and the expression (if applicable) is solved:

When you double-click the dimension for editing, the variable (and expression) is displayed:

**Variable in a solid body feature**

Use variables anywhere you use expressions in a Part Studio. For example, in an extrude or revolve operation:

Start the operation as usual (in this case, Revolve); in the numeric value field, enter `#` and the variable name (or optionally, as part of an expression). For example: `(#{x})` deg.

Accept the feature.
When you edit the feature, the solution is displayed in the numeric value field. For example: 4 deg.

Click in the field and the variable (and expression, if applicable) is displayed.

**Using Arrays in Variables**

Variable values are able to contain expressions but must evaluate to a scalar value. You have the ability to specify an array with an index, and the index can be a variable. This allows you to change the value of the variable by changing the value of the index variable.

To use arrays in variables you must first set up a zero-based index.

1. Tap the Variable tool and set the name to "config." Set the value of #config to 2.

2. Tap the Variable tool and set the name to "diameter." Set the value of #diameter to [0.25, 0.5, 1] [#config].

   It is important that you include the second set of brackets that contain "#config", because this is what allows the value of the variable #config to point to the correct place in the array.

3. Tap the Variable tool and set the name to "length." Set the value of #length to [2, 4, 10] [#config].

   It is important that you include the second set of brackets that contain "#config", because this is what allows the value of the variable #config to point to the correct place in the array.

4. Tap the sketch tool and sketch a circle.

5. Tap the sketch tool and sketch a line.

6. Set the diameter of the circle to "#diameter".

7. Set the length of the line to "#length".

   The diameter of the circle is set to 1, and the length of the line is set to 10.

8. Edit the #config variable, set the value to 0.

The sketch updates and the circle diameter is now set to 0.25 and the line length is set to 2.
You are able to do this for as many arrays as you like, and your arrays can hold infinite places (they are not limited to only three values as this example shows).

**FeatureScript functions in Variables**

You are able to use FeatureScript functions in a variable, following the FeatureScript syntax.

For example, you might create a variable of type Any, named “Adjust”, to store a function that doubles a given length and adds 2.5mm, as follows:

```feature
function(len) { return len * 2 + 2.5 mm; }
```

and then reference that variable in an expression, such as:

`#Adjust(20mm)`

---

**Tips**

- When you change the value of a variable (edit it as you would any feature), all operations that use the variable are automatically updated.
- Variable names are case-sensitive.
- You have the ability to input a variable as a stand-alone value or as part of an equation.
- You have the ability to use a variable in a dimension, expression, or feature.
- In cases where you specify a variable without a unit, expressions will also be
unitless or assume the unit of the workspace. But if you explicitly add units to a variable value, then any expression should also match that unit when being written in order to be valid.

**Variable: Android**

Create a variable for use in expressions in a Part Studio, and assign a value. Use the variable in dimensions and expressions. Variables are features in Onshape, so the placement in the Feature list is important. Create a variable before the feature in which you will use it.

**Steps**

1. While in a Part Studio, select the Variable tool.
2. In the dialog:
   a. Enter a name for the variable (and by which to reference it).
      
      Use only English letters and numbers in the name (at least one English letter followed by letters and/or numbers). Variable names are case-sensitive.
      
   b. Select a type:
      
      - Length - A numeric value representing a length (decimal, integer, fraction)
      - Angle - A numeric value representing an angle (decimal, integer, fraction)
      - Number - A numeric value (decimal, integer, fraction)
      - Any - Any of the above, a numeric value with different units, or a FeatureScript value such as boolean, map, array, string, or a function. See https://cad.onshape.com/FsDoc/variables.htm#standard-types and examples below.
      
   c. Enter a value (and optionally, units for Length, Angle, and Any).

3. Tap the checkmark.

The variable you created is listed in the Feature List and displays the name and value of the variable.

**Variables in dimensions**

Create a dimension, in the dimension field enter # and the variable name (and optionally, as part of an expression, as shown below):

![Example dimensions](image)

Save the dimension; the variable is replaced with the value and the expression (if applicable) is solved:
When you double-click the dimension for editing, the variable (and expression) is displayed:

Variable in a solid body feature

Use variables anywhere you use expressions in a Part Studio. For example, in an extrude or revolve operation:

Start the operation as usual (in this case, Revolve); in the numeric value field, enter # and the variable name (or optionally, as part of an expression). For example: (#x) deg.

Accept the feature.

When you edit the feature, the solution is displayed in the numeric value field. For example: 4 deg.

Click in the field and the variable (and expression, if applicable) is displayed.

Using Arrays in Variables

Variable values are able to contain expressions but must evaluate to a scalar value. You have the ability to specify an array with an index, and the index may be a variable. This allows you to change the value of the variable by changing the value of the index variable.

To use arrays in variables you must first set up a zero-based index.
1. Tap the Variable tool and set the name to "config." Set the value of #config to 2.

2. Tap the Variable tool and set the name to "diameter." Set the value of #diameter to [0.25, 0.5, 1] [#config].

   It is important that you include the second set of brackets that contain "#config", because this is what allows the value of the variable #config to point to the correct place in the array.

3. Tap the Variable tool and set the name to "length." Set the value of #length to [2, 4, 10] [#config].

   It is important that you include the second set of brackets that contain "#config", because this is what allows the value of the variable #config to point to the correct place in the array.

4. Tap the sketch tool and sketch a circle.

5. Tap the sketch tool and sketch a line.

6. Set the diameter of the circle to "#diameter".

7. Set the length of the line to "#length".

   The diameter of the circle is set to 1, and the length of the line is set to 10.

8. Edit the #config variable, set the value to 0.

   The sketch updates and the circle diameter is now set to 0.25 and the line length is set to 2.

   You are able to do this for as many arrays as you like, and your arrays are able to hold infinite places (they are not limited to only three values as this example shows).

**FeatureScript functions in Variables**

You are able to use FeatureScript functions in a variable, following the FeatureScript syntax.

For example, you might create a variable of type Any, named “Adjust”, to store a function that doubles a given length and adds 2.5mm, as follows:

```
function(len) { return len * 2 + 2.5 mm; }
```

and then reference that variable in an expression, such as:
# Adjust(20mm)

**Tips**

- When you change the value of a variable (edit it as you would any feature), all operations that use the variable are automatically updated.

- Variable names are case-sensitive.

- You have the ability to input a variable as a stand-alone value or as part of an equation.

- You have the ability to use a variable in a dimension, expression, or feature.

---

**Composite Part**

A part comprised of selected bodies (parts, surfaces, curves, and/or points) grouped together to act as one part. You can create a closed composite part (one which consumes the bodies involved) or an open composite part (one that does not consume the bodies involved).
A composite part can be used like any other part: you can apply features to it, insert it into an assembly, release it, create a drawing of it, include it as a single line item in a BOM, assign metadata, and even revision it as a single part.

This functionality is available on Onshape's browser, iOS, and Android platforms.

In the Feature toolbar:

Steps

1. Click

2. Select the parts, curves, surfaces, and/or points (also referred to as 'bodies') to be involved in the composite part:
Notice that a Composite part is listed below the parts in the list; the selected bodies (entities) are listed in the Composite part 1 dialog box (you can rename this feature).

3. Check the Closed checkbox to indicate that the bodies involved should no longer exist as separate entities.

Leave the box unchecked to indicate that the bodies should remain as separate entities in addition to being part of the composite part.
4. When satisfied with your selections, click the checkmark \(\checkmark\) to accept and close the dialog.

Converting closed composite parts to individual parts

When a closed composite part is created either through an import or manually in the Part Studio, you can change your mind and convert it to an open composite part or remove the composite part altogether and return to separate bodies. Use the Delete part tool, as shown below:

1. Select the Delete part \(\square\) in the Feature toolbar in the Part Studio.

2. Select *Dissolve composite parts* in the drop down in the dialog:
Dissolve composite parts reduces the composite part to its included bodies and removes the composite:

Before dissolving: one composite part and no surfaces in the list

After dissolving: no composite parts but many surfaces in the list

3. Click the checkmark to accept and close the dialog.

You can use Dissolve composite part on both a closed composite part and an open composite part.

You can select to delete the composite part or individual bodies, if the part is an open composite.

*Ignore composite parts* lets you select a specific body to remove; resulting in the ability to select a body and not the entire composite part:
When Ignore composite part is selected, the individual bodies can be selected; here the body outlined in yellow was selected for removal.

Tips

- Use the context menu (RMB) on the composite part in the Composite parts list or on the part itself in the graphics area to Hide other composite parts or Hide all composite parts:

<table>
<thead>
<tr>
<th>Hide</th>
<th>Hide other composite parts</th>
<th>Hide all composite parts</th>
<th>Add comment...</th>
<th>Delete...</th>
</tr>
</thead>
</table>

- Composite parts are data-managed as a single part: you can release a composite part, assign metadata to it, revise a composite part, and each composite part is listed as a single line item in any bill of materials.

- A single body can be made a composite, if you wish.

- Creating a composite doesn’t change the bodies selected in any way. If a body owns a mate connector, the mate connector stays with that body after it’s made into a composite part as well.
Reminder: A closed composite part consumes the bodies involved and they no longer appear in the Parts list as separate bodies. The Parts list in the Part Studio will list only the composite - graphically you still see and can act on the individual bodies. An open composite does not consume the bodies involved and they are still listed in the Parts list in the Part Studio along with the composite part. You can still act on the individual bodies if you wish.

You can also create composite parts using FeatureScript. See the FeatureScript documentation for more information.

You can use other feature tools on composite parts, for example Delete and Transform. Be aware that open and closed composites may work differently with the same feature.

This is especially useful for restoring a body or a surface after an import process that fractures the body. Parts that are fractured during import will automatically be made composite as part of the same process. For more information on importing, see "Importing Files" on page 1975.

Composite parts never own Mate connectors; mate connectors are always owned by individual parts.

When creating a closed composite, and you select another composite (any type), the selected composite is consumed. When you use a composite in an open composite, the composite is not consumed.

Two open composites can overlap parts.

When creating a mate connector on a closed composite, you get the name of the Part+feature name -- in the dialog, not the part name (because the part no longer exists on its own).

Sheet Metal Model

Create sheet metal parts by converting existing parts, extruding sketch curves (including arcs and splines to create rolled sheet metal), or thickening faces or sketches.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sheet Metal Model: Desktop**

In the Feature toolbar:

Create sheet metal parts by converting existing parts, extruding sketch curves (including arcs and splines to create rolled sheet metal), or thickening faces or sketches. All operations on active sheet metal models are automatically represented as a flat pattern, and joints and bends are listed in a sheet metal table. The folded, flat, and table views are available and updated simultaneously and in real time. Sheet metal models may consist of multiple parts, and multiple sheet metal models may be active simultaneously.

The Sheet metal model tool activates a sheet metal feature. Consequently, features affecting that sheet metal model affect it as piece of sheet metal. The [Finish sheet metal](#) tool deactivates the sheet metal feature, allowing features to affect the model as a 3D model and not as sheet metal.

For example, when the sheet metal model is active, any features piercing the sheet metal are perpendicular to the walls whereas when deactivated, any feature piercing the walls are subject to the angle between the feature and the wall.

You are also able to use the additional [sheet metal tools](#) to create flanges, define and modify joints, convert bends to rips and vice versa, define corner parameters, as well as view the sheet metal as a 3D part and in flattened view simultaneously. Onshape also provides a sheet metal table listing bends and rips, where you are able to edit corner radii and joint types as well as create a drawing of the sheet metal part.

**Sheet Metal Model: Convert**

Create a sheet metal model by enclosing existing parts:
1. While in a Part Studio, click 📑.

2. Select the type of sheet metal operation: Convert

3. Select the parts to enclose.

4. Select the faces to exclude from the operation.

5. Select the edges to define bends; edges not selected are made into rips.

   Arrows or splines not selected in the "Edges or cylinders to bend" field become tangent joints, not bends. See the example below.

6. Specify applicable options:
   a. Clearance from input - The relative offset between the sheet metal and the part selected to be enclosed by the sheet metal
   b. Clearance includes bends - Check this to include the bend within the clearance value
   c. Keep input parts - Keep the selected parts (enclosed by the sheet metal) or not
d. **Thickness** - The thickness of the sheet metal

e. **Bend radius** - The inside radius of the bends created

f. **Bend K Factor** - The fraction of material thickness on which the neutral axis lies on a bend. (Default is 0.45.)

g. **Rolled K Factor** - The fraction of material thickness on which the neutral axis lies on a section of rolled wall. (Default is 0.5.)

h. **Minimal gap** - The smallest gap between the sheet metal edges defining a rip

i. **Corner relief type** -

   - **Square - Sized**
     
     ![Square - Sized](image)

     Flat view: [Flat view image]  
     3D view: [3D view image]

   - **Rectangle - Scaled**
     
     ![Rectangle - Scaled](image)

     Flat view: [Flat view image]  
     3D view: [3D view image]

   - **Round - Sized**
     
     ![Round - Sized](image)

     Flat view: [Flat view image]  
     3D view: [3D view image]

   - **Round - Scaled**
     
     ![Round - Scaled](image)

     Flat view: [Flat view image]  
     3D view: [3D view image]

   - **Closed**
     
     ![Closed](image)

     Flat view: [Flat view image]  
     3D view: [3D view image]

   - **Simple**
     
     ![Simple](image)

     Flat view: [Flat view image]  
     3D view: [3D view image]
j. **Corner relief scale** - The scale of the corner opening (for Scaled openings), a value between 1.00 and 2.00.

k. **Corner relief width** - The measurement of the width of the corner opening (for Sized openings), in default units or specified units.

l. **Bend relief type** - The shape of the bend relief:

   - Rectangle - Scaled

   ![Rectangle Scaled](image)

   - Obround - Scaled

   ![Obround Scaled](image)

   - Tear

   ![Tear](image)

   **Bend relief depth scale** - A value between 1.00 and 5.00. Once you enter a value it becomes the default across all documents.

   A value of 1 results in an obround bend relief perfectly touching the bend and a rectangular bend relief matches the depth of the obround:

   ![Bend relief depth scale](image)
Any value past 1 adds depth via the formula: 
\[ ((\text{depth scale} - 1) \times \text{bendRadius}) \]
**Bend relief width scale** - A value between 0.0625 and 2.00. The width of the bend relief is calculated via the formula: thickness \* width scale. Once you enter a value it becomes the default across all documents.

7. Click to accept the feature; the Sheet metal model is listed in the Feature list.

8. Apply any other specific sheet metal features now. See note below.

9. Click Finish sheet metal model if you want to continue to add non-sheet metal features to your part.

a. Click to close the Sheet metal feature; the Finish sheet metal model feature is listed in the Feature list.

---

**Sheet Metal Model: Extrude**

Create a sheet metal model by extruding sketch curves:
1. While in a Part Studio, click

![Image]

2. Select the sheet metal operation Extrude.

3. Select the sketch curves to extrude.

4. Select the End type: Blind, Symmetric, Up to next, Up to face, Up to part, Up to vertex.

5. Drag or set the depth.

6. Set the thickness of the sheet metal.

7. Set the inside radius of the bends.

8. Specify the **K Factor** - The fraction of material thickness on which the neutral axis lies.

9. Set **Minimal gap** - The smallest gap between sheet metal edges.

10. **Corner relief type:**

    - **Square - Sized**

    ![Flat view] ![3D view]
11. **Corner relief scale** - The scale of the corner opening (for Scaled corners); a value between 1.00 and 2.00.

12. **Bend relief type** - The shape of the bend relief:
   - **Rectangle - Scaled:**
   - **Obround - Scaled:**
   - **Tear:**
13. **Bend relief depth scale** - Between 1.00 and 2.00

14. **Bend relief width scale** - A value between 0.0625 and 2.00

15. Click ✅ to accept the feature; the Sheet metal model is listed in the Feature list.

16. Apply any other specific sheet metal features now. See note below.

17. Click **Finish sheet metal model** if you want to continue to add non-sheet metal features to your part.

18. Click ✅ to close the Sheet metal feature; the Finish sheet metal model feature is listed in the Feature list, has a flat pattern and a sheet metal table listing the bends and joints:

   ![Sheet metal table and flat pattern](image)

When using an arc or spline to define a rolled wall, the sheet metal table and the flat pattern is a bit different. No bend is listed for the rolled wall, instead a tangent joint is listed in the table:
Similarly, no bend lines are shown in the flat pattern, since no bends are made.

**Extruding an arc as a bend**

You may decide to extrude an arc as a bend, however, by selecting the arc while the "Arcs to extrude as bends" field is active, as shown below:

This results in the selected arc (or arcs) being made into bends (instead of tangent joints) and listed in the bend table with their radius grayed out. Since the arc has a radius in the sketch, you are unable to edit it in the bend table.
Sheet Metal Model: Thicken

Create a sheet metal model by thickening faces, surfaces or sketch regions:

1. While in a Part Studio, click 📋.

2. Select the type of sheet metal operation: Thicken

3. Select sketch regions, planar surfaces or faces of a part.

4. Select the edges to create bends; edges not selected are made into rips or, in the case of arcs and splines, are made into tangent joints.

5. Specify applicable options:
   a. **Clearance from input** - The relative offset between the sheet metal and the part selected to be enclosed by the sheet metal
   b. **Clearance includes bends** - Check to include the clearance for bends
   c. **Thickness** - The thickness of the sheet metal
   d. **Bend radius** - The inside radius of the bends created
e. **Bend K Factor** - The fraction of material thickness on which the neutral axis lies.
   (Default is 0.45.)

f. **Rolled K Factor** - The fraction of material thickness on which the neutral axis lies on a section of rolled wall. (Default is 0.5.)

g. **Minimal gap** - The smallest gap between sheet metal edges

h. **Corner relief type** - The shape of the corner relief:
   - **Square - Sized**
     
     Flat view: 3D view:
   
   - **Rectangle - Scaled**
     
     Flat view: 3D view:
   
   - **Round - Sized**
     
     Flat view: 3D view:
   
   - **Round - Scaled**
     
     Flat view: 3D view:
   
   - **Closed**
     
     Flat view: 3D view:
   
   - **Simple**
     
     Flat view: 3D view:

i. **Corner relief scale** - The scale of the corner opening (for Scaled corners); a value between 1.00 and 2.00.
j. Bend relief type:
   - Rectangle:
   - Obround:
   - Tear:

k. Bend relief depth scale - Between 1.00 and 2.00.

l. Bend relief width scale - Between 0.0625 and 2.00.

6. Click ✔️ to accept the feature; the Sheet metal model is listed in the Feature list.

7. Apply any other specific sheet metal features now. See note below.

8. Click Finish sheet metal model ✔️ if you want to continue to add non-sheet metal features to your part.

9. Click ✔️ to close the Sheet metal model feature; the Finish sheet metal model feature is listed in the Feature list.

When using an arc or spline to define a rolled wall, the sheet metal table and the flat pattern is a bit different. No bend is listed for the rolled wall, instead a tangent joint is listed in the table:
Similarly, no bend lines are shown in the flat pattern, since no bends are made.

**Thickening an arc as a bend**

You are able to decide to thicken an arc as a bend, however, by selecting the arc while the "Arcs to extrude as bends" field is active, as shown below:

This results in the selected arc (or arcs) being made into bends (instead of tangent joints) and listed in the bend table with their radius grayed out. Since the arc has a radius in the sketch, you are unable to edit it in the bend table.
Only a subset of available features can be added to active sheet metal models (Extrude > Remove, Move face, Boolean, and Hole for example). All features are represented in the feature list, the model in the graphics area, the sheet metal table, and the flat pattern. For example, you are able to Extrude > Remove a sketch to create an opening in an active sheet metal model. The resulting opening’s sides are always perpendicular to the wall it pierces.

Other Extrude options that modify the model such as Add or Intersect are not allowed until the model is deactivated using the Finish sheet metal model tool.

Keep in mind that subsequent features modifying a deactivated sheet metal model act on the part as they would any non-sheet metal part, and do not affect the flat pattern or table values.

**Treating sheet metal edges**

Use Fillet and Chamfer to soften sheet metal edges, if necessary. Be aware that selecting a sheet metal edge must be done at the very corner and not an adjacent edge. Note that advanced Fillet and Chamfer features (conic fillet for example) are not available on sheet metal at this time.

Onshape retains the original construction edge despite a fillet or chamfer and uses that construction edge for future features. For example, creating a flange on a filleted edge ignores the fillet and creates the flange along the original edge. You may use the Move face tool to move the edge of the flange back to the edge of the fillet if desired:

Fillet a corner:

Create a flange:
Use Move face to move the edge of the flange:

Re-fillet the corner:

**Patterning faces and flanges on sheet metal**

You have the ability to pattern sheet metal faces and flanges using the Face parameter of the patterning tools (Linear pattern, Circular pattern, and Curve pattern) as well as mirroring sheet metal faces and flanges. The tools function normally, just make sure to select "Face" as the Pattern type, or to select a face when mirroring:
**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- 🎫 **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

- 🎫 **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- 🎫 **Tab** - Add a tab to a sheet metal flange.
- Make joint - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- Corner - Modify a corner type and relief scale.

- Bend relief - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.

- Modify joint - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.

- Sheet metal table and flat view - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

**Sheet Metal Model: iOS**

Create sheet metal parts by converting existing parts, extruding sketch curves (including arcs and splines to create rolled sheet metal), or thickening faces or sketches.

**Sheet Metal Model: Convert**

Create a sheet metal model by enclosing existing parts:

1. While in a Part Studio, tap 

2. Select the type of sheet metal operation: Convert

3. Select the parts to enclose.

4. Select the faces to exclude from the operation.

5. Select the edges to define bends; edges not selected are made into rips.

   Arcs or splines not selected in the "Edges or cylinders to bend" field become tangent joints, not bends.
6. Specify applicable options:

a. **Clearance from input** - The relative offset between the sheet metal and the part selected to be enclosed by the sheet metal

b. **Clearance includes bends** - Check this to include the bend within the clearance value

c. **Keep input parts** - Keep the selected parts (enclosed by the sheet metal) or not

d. **Thickness** - The thickness of the sheet metal

e. **Bend radius** - The inside radius of the bends created

f. **Bend K Factor** - The fraction of material thickness on which the neutral axis lies on a bend. (Default is 0.45.)

g. **Rolled K Factor** - The fraction of material thickness on which the neutral axis lies on a section of rolled wall. (Default is 0.5.)

h. **Minimal gap** - The smallest gap between the sheet metal edges defining a rip

i. **Corner relief type** -

   - **Square - Sized**
     
     Flat view: ![Flat view]
     3D view: ![3D view]

   - **Rectangle - Scaled**
     
     Flat view: ![Flat view]
     3D view: ![3D view]

   - **Round - Sized**
     
     Flat view: ![Flat view]
     3D view: ![3D view]

   - **Round - Scaled**
     
     Flat view: ![Flat view]
     3D view: ![3D view]
• **Closed**

Flat view: 3D view:

• **Simple**

Flat view: 3D view:

j. **Corner relief scale** - The scale of the corner opening (for Scaled openings), a value between 1.00 and 2.00.

k. **Corner relief width** - The measurement of the width of the corner opening (for Sized openings), in default units or specified units.

l. **Bend relief type** - The shape of the bend relief:

• Rectangle - Scaled

• Obround - Scaled

• Tear

• **Bend relief depth scale** - A value between 1.00 and 5.00. Once you enter a value it becomes the default across all documents.

A value of 1 results in an obround bend relief perfectly touching the bend and a rectangular bend relief matches the depth of the obround.

Any value past 1 adds depth via the formula: 
\[(\text{depth scale} - 1) \times \text{bendRadius}\].
**Bend relief width scale** - A value between 0.0625 and 2.00. The width of the bend relief is calculated via the formula: thickness * width scale. Once you enter a value it becomes the default across all documents.

7. Tap ✔️ to accept the feature; the Sheet metal model is listed in the Feature list.

8. Apply any other specific sheet metal features now. See note below.

9. Tap Finish sheet metal model if you want to continue to add non-sheet metal features to your part.

   a. Tap ✔️ to close the Sheet metal feature; the Finish sheet metal model feature is listed in the Feature list.

---

**Sheet Metal Model: Extrude**

Create a sheet metal model by extruding sketch curves:

1. While in a Part Studio, tap 📖.

2. Select the sheet metal operation Extrude.

3. Select the sketch curves to extrude.

4. Select the End type: Blind, Symmetric, Up to next, Up to face, Up to part, Up to vertex.

5. Drag or set the depth.

6. Set the thickness of the sheet metal.

7. Set the inside radius of the bends.

8. Specify the **K Factor** - The fraction of material thickness on which the neutral axis
lies.

9. Set **Minimal gap** - The smallest gap between sheet metal edges.

10. **Corner relief type:**
   - **Square - Sized**
   - Flat view: ![Flat view](image)
   - 3D view: ![3D view](image)
   - **Rectangle - Scaled**
   - Flat view: ![Flat view](image)
   - 3D view: ![3D view](image)
   - **Round - Sized**
   - Flat view: ![Flat view](image)
   - 3D view: ![3D view](image)
   - **Round - Scaled**
   - Flat view: ![Flat view](image)
   - 3D view: ![3D view](image)
   - **Closed**
   - Flat view: ![Flat view](image)
   - 3D view: ![3D view](image)
   - **Simple**
   - Flat view: ![Flat view](image)
   - 3D view: ![3D view](image)

11. **Corner relief scale** - The scale of the corner opening (for Scaled corners); a value between 1.00 and 2.00.

12. **Bend relief type** - The shape of the bend relief:
   - **Rectangle - Scaled**:
13. **Bend relief depth scale** - Between 1.00 and 2.00

14. **Bend relief width scale** - A value between 0.0625 and 2.00

15. Tap ☑️ to accept the feature; the Sheet metal model is listed in the Feature list.

16. Apply any other specific sheet metal features now. See note below.

17. Tap **Finish sheet metal model** if you want to continue to add non-sheet metal features to your part.

### Extruding an arc as a bend

You may decide to extrude an arc as a bend, however, by selecting the arc while the "Arcs to extrude as bends" field is active.

This results in the selected arc (or arcs) being made into bends (instead of tangent joints).

### Sheet Metal Model: Thicken

Create a sheet metal model by thickening faces, surfaces or sketch regions:

1. While in a Part Studio, tap 📚.

2. Select the type of sheet metal operation: Thicken

3. Select sketch regions, planar surfaces or faces of a part.

4. Select the edges to create bends; edges not selected are made into rips or, in the case of arcs and splines, are made into tangent joints.
5. Specify applicable options:

a. **Clearance from input** - The relative offset between the sheet metal and the part selected to be enclosed by the sheet metal

b. **Clearance includes bends** - Check to include the clearance for bends

c. **Thickness** - The thickness of the sheet metal

d. **Bend radius** - The inside radius of the bends created

e. **Bend K Factor** - The fraction of material thickness on which the neutral axis lies.  
   (Default is 0.45.)

f. **Rolled K Factor** - The fraction of material thickness on which the neutral axis lies on a section of rolled wall. (Default is 0.5.)

g. **Minimal gap** - The smallest gap between sheet metal edges

h. **Corner relief type** - The shape of the corner relief:

   - **Square - Sized**
     - Flat view: 
     - 3D view:

   - **Rectangle - Scaled**
     - Flat view: 
     - 3D view:

   - **Round - Sized**
     - Flat view: 
     - 3D view:

   - **Round - Scaled**
     - Flat view: 
     - 3D view:

   - **Closed**
     - Flat view: 
     - 3D view:
• Simple

Flat view: 3D view:

i. **Corner relief scale** - The scale of the corner opening (for Scaled corners); a value between 1.00 and 2.00.

j. **Bend relief type**:
   - **Rectangle:**
   - **Obround:**
   - **Tear:**

k. **Bend relief depth scale** - Between 1.00 and 2.00.

l. **Bend relief width scale** - Between 0.0625 and 2.00.

6. Tap ✔️ to accept the feature; the Sheet metal model is listed in the Feature list.

7. Apply any other specific sheet metal features now. See note below.

8. Tap [Finish sheet metal model](#) if you want to continue to add non-sheet metal features to your part.

9. Tap ✔️ to close the Sheet metal model feature; the Finish sheet metal model feature is listed in the Feature list.
**Thickening an arc as a bend**

You may decide to thicken an arc as a bend, however, by selecting the arc while the "Arcs to extrude as bends" field is active.

This results in the selected arc (or arcs) being made into bends (instead of tangent joints).

Only a subset of available features can be added to active sheet metal models (Extrude > Remove, Move face, Boolean, and Hole for example). All features are represented in the feature list, the model in the graphics area, the sheet metal table, and the flat pattern. For example, you can Extrude > Remove a sketch to create an opening in an active sheet metal model. The resulting opening’s sides are always perpendicular to the wall it pierces.

Other Extrude options that modify the model such as Add or Intersect are not allowed until the model is deactivated using the Finish sheet metal model tool.

Keep in mind that subsequent features modifying a deactivated sheet metal model act on the part as they would any non-sheet metal part, and do not affect the flat pattern or table values.

**Treating sheet metal edges**

Use Fillet and Chamfer to soften sheet metal edges, if necessary. Be aware that selecting a sheet metal edge must be done at the very corner and not an adjacent edge. Note that advanced Fillet and Chamfer features (conic fillet for example) are not available on sheet metal at this time.

Onshape retains the original construction edge despite a fillet or chamfer and uses that construction edge for future features. For example, creating a flange on a filleted edge ignores the fillet and creates the flange along the original edge. You may use
the Move face tool to move the edge of the flange back to the edge of the fillet if desired:

Fillet a corner:

Create a flange:

Use Move face to move the edge of the flange:

Re-fillet the corner:
Patterning faces and flanges on sheet metal

You have the ability to pattern sheet metal faces and flanges using the Face parameter of the patterning tools (Linear pattern, Circular pattern, and Curve pattern) as well as mirroring sheet metal faces and flanges. The tools function normally, just make sure to select "Face" as the Pattern type, or to select a face when mirroring.

Additional sheet metal tools

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- **Tab** - Add a tab to a sheet metal flange.

- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- **Corner** - Modify a corner type and relief scale.

- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.

- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

---

**Sheet Metal Flange**

Create a wall for each edge selected for a sheet metal model, connected by a bend.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sheet Metal Flange: Desktop**

In the Feature toolbar:

Create a wall for each edge selected for a sheet metal model, connected by a bend.

**Steps**

To create a flange:
1. While in a Part Studio, click 

2. Select the edges or side faces along which to create flanges.

3. Specify the details of the flange:

   a. Flange alignment:

      - **Inner** - Align the inner wall of the flange wall with the inner edge of the side lip.

      ![Inner Flange Alignment Example]

      - **Outer** - Align the outer wall of the flange wall with the outer edge of the side lip.

      ![Outer Flange Alignment Example]
Middle - Align the mid-plane of the flange wall with a theoretical line halfway between the inner and outer edge of the lip.

b. **End type of the wall** - Blind (specify a distance, or length, of the flange), Up to entity (select the entity or Mate connector -inferred or existing- to extend the flange to), Up to entity with offset (select the entity or Mate connector -inferred or existing- to extend towards and specify the offset value)

c. Select an angle control type to specify how to orient the angle of the flange bend:
   - **Bend angle** - Enter a specific angle value, measured from the edge from which to extend the flange.
   - **Align to geometry** - Select an edge or Mate connector (inferred or existing) to align the flange parallel to.
   - **Angle from direction** - Select an edge or Mate connector (inferred or existing) from which to measure an angle for the flange.

d. Choose a direction using the Flip arrow. (Toggle the direction with this arrow.)

e. **Automatic miter** - Check to automatically trim or extend the intersecting flanges for a miter. Leave unchecked to specify a custom miter angle

f. **Use model bend radius** - Check to use the inside bend radius specified for the sheet metal model, or leave unchecked to enter a custom value (Bend radius)

4. Click to accept the feature; the Flange in the Feature list.

To modify a flange, use a Direct Edit tool such as Move face.
**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- **Tab** - Add a tab to a sheet metal flange.

- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- **Corner** - Modify a corner type and relief scale.

- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.

- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.
Sheet metal table and flat view - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

Sheet Metal Flange: iOS
Create a wall for each edge selected for a sheet metal model, connected by a bend.

Steps
To create a flange:

1. While in a Part Studio, tap 🔄.
2. Select the edges or side faces along which to create flanges.
3. Specify the details of the flange:
   a. Flange alignment:
      - **Inner** - Align the inner wall of the flange with the selected edge.
      - **Outer** - Align the outer wall of the flange with the selected edge.
      - **Middle** - Align the mid-plane of the flange wall with the selected edge.
   b. **End type of the wall** - Blind (specify a distance, or length, of the flange), Up to entity (select the entity or Mate connector - inferred or existing- to extend the flange to), Up to entity with offset (select the entity or Mate connector - inferred or existing- to extend towards and specify the offset value)
   c. Select an angle control type to specify how to orient the angle of the flange bend:
      - **Bend angle** - Enter a specific angle value, measured from the edge from which to extend the flange.
      - **Align to geometry** - Select an edge or Mate connector (inferred or existing) to align the flange parallel to.
• **Angle from direction** - Select an edge or Mate connector (inferred or existing) from which to measure an angle for the flange.

d. Choose a direction using the Flip arrow. (Toggle the direction with this arrow.)

e. **Automatic miter** - Check to automatically trim or extend the intersecting flanges for a miter. Leave unchecked to specify a custom miter angle.

f. **Use model bend radius** - Check to use the inside bend radius specified for the sheet metal model, or leave unchecked to enter a custom value (Bend radius)

4. Tap ✔️ to accept the feature; the Flange in the Feature list.

To modify a flange, use a Direct Edit tool such as Move face.

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**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- 🌩️ Flange - Create a wall for each edge selected, connected to the selected edge with a bend.

- 🗂️ Tab - Add a tab to a sheet metal flange.

- 📖 Make joint - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- Corner - Modify a corner type and relief scale.
- Bend relief - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
- Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

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### Sheet Metal Hem

Create one or more hems on existing sheet metal parts.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sheet metal hem: Desktop**

Create one or more hems on existing sheet metal parts.

In the Feature toolbar:

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**Steps**

To create one or more hems on existing sheet metal parts, first have a sheet metal part in the Part Studio:
1. While in a Part Studio, click 🍃.

![Hem 1](image)

2. Select the edges or side faces along which to create hems.

3. Choose a direction using the Flip arrow. (Toggle the direction with this arrow.)

4. Specify the Hem type:

   **Tear drop**

   Make the hem a tear drop shape; the length extends from the very back of the curved edge to the front edge of the hem.

![Hem 1](image)

Specifications:

**Inner radius** - The measurement of the inner radius of the hem bend.

**Minimal gap** - When checked, this field uses the value specified as the Minimal gap value specified in the Sheet metal model feature. (The value representing the distance between the end of the rolled hem and the sheet being hemmed.)

**Gap** - When Minimal gap is unchecked, specify the specific gap you require.

**Total length** - The length of the hem from the outer most edge of the hem to the end of the hem.
**Hem alignment** - The alignment of the hem in relationship to the outer edge of the hem bend: **Outer** aligns the outside edge of the hem with the edge of the sheet metal. **In place** adds the outside edge beginning at the edge of the sheet metal:

![Hem alignment diagrams](image)

*An outer tear drop hem with outer specified, first image, and in place specified, second image*

**Corner type** - Where hems come together on sheet metal, specify which type of corner to form: **Simple** results in a straight line in the flat view (linear in the flat). **Closed** closes up the corner as much as possible so the corner can be welded and results in more complicated geometry in the flat:

![Corner type diagrams](image)

*An simple corner in the model and in the flat view, above*

*An closed corner in the model and in the flat view, above*

**Rolled**

Roll the hem over the edge, using a specific radius.
Specifications:

**Inner radius** - The measurement of the inner radius of the hem.

**Angle** - The angle of the roll, determining how far the roll extends towards the sheet metal wall.

*The first image shows an angle of 270 degrees and the second image has an angle of 200 degrees*

**Hem alignment** - The alignment of the hem in relationship to the outer edge of the hem bend: **Outer** aligns the outside edge of the hem with the edge of the sheet metal. **In place** adds the outside edge beginning at the edge of the sheet metal:

*A rolled hem with Outer specified, first image, and In place specified, second image*

**Corner type** - Where hems come together on sheet metal, specify which type of corner to form: **Simple** results in a straight line in the flat view (linear in the flat). **Closed** closes up the corner as much as possible so the corner can be welded and results in more complicated geometry in the flat:
A Simple corner in the model and in the flat view, above

A Closed corner in the model and in the flat view, above

**Straight**

Align the hem along the sheet metal material.

Specifications:

**Flattened** - When checked, places the hem flush against the sheet metal material, using the Sheet metal model's Minimal gap for the space between hem and sheet metal.

**Inner radius** - When Flattened is unchecked, you can specify measurement of the inner radius of the hem.

**Total length** - The length of the hem from the outer most edge of the hem to the end of the hem.

**Hem alignment** - The alignment of the hem in relationship to the outer edge of the hem bend: **Outer** aligns the outside edge of the hem with the edge of the sheet metal. **In place** adds the outside edge beginning at the edge of the sheet metal:
A rolled hem with Outer specified, first image, and In place specified, second image

Corner type - - Where hems come together on sheet metal, specify which type of corner to form: Simple results in a straight line in the flat view (linear in the flat). Closed closes up the corner as much as possible so the corner can be welded and results in more complicated geometry in the flat:

5. Click to accept the feature; the Hem is listed in the Feature list.

All dialog selections will default to previously specified values for subsequent hem operations, across sessions and documents.

When you have the flat view open, notice in the bend table that hems are listed there, along with their radius and angle. You can move them up or down in the table, but currently you cannot edit them in the table.

Sheet metal hem: iOS

Create one or more hems on existing sheet metal parts.
To create one or more hems on existing sheet metal parts, first have a sheet metal part in the Part Studio:

1. While in a Part Studio, click 🔍.

2. Select the edges or side faces along which to create hems.

3. Choose a direction using the Flip arrow. (Toggle the direction with this arrow.)

4. Specify the Hem type:

   **Tear drop**
   
   Make the hem a tear drop shape; the length extends from the very back of the
curved edge to the front edge of the hem.

Specifications:

**Inner radius** - The measurement of the inner radius of the hem bend.

**Minimal gap** - When checked, the Minimal gap uses the value specified as the Minimal gap in the Sheet metal model feature.

**Gap** - When Minimal gap is unchecked, specify the specific gap you require.

**Total length** - The length of the hem from the outer most edge of the hem to the end of the hem.

**Hem alignment** - The alignment of the hem in relationship to the outer edge of the hem bend: **Outer** aligns the outside edge of the hem with the edge of the sheet metal. **In place** adds the outside edge beginning at the edge of the sheet metal:

A tear drop hem with **Outer** specified, first image, and **In place** specified, second image

**Corner type** - Where hems come together on sheet metal, specify which type of corner to form: **Simple** results in a straight line in the flat view (linear in the flat). **Closed** closes up the corner as much as possible so the corner can be welded and results in more complicated geometry in the flat:
A Simple corner in the model and in the flat view, above

A Closed corner in the model and in the flat view, above

Rolled

Roll the hem over the edge, using a specific radius.

Specifications:

**Inner radius** - The measurement of the inner radius of the hem.

**Angle** - The angle of the roll, determining how far the roll extends towards the sheet metal wall.

The first image shows an angle of 270 degrees and the second image has an angle of 200 degrees

**Hem alignment** - The alignment of the hem in relationship to the outer edge of the hem bend: **Outer** aligns the outside edge of the hem with the edge of the sheet metal. **In place** adds the outside edge beginning at the edge of the sheet metal.
A rolled hem with Outer specified, first image, and In place specified, second image

**Corner type** - Where hems come together on sheet metal, specify which type of corner to form: **Simple** results in a straight line in the flat view (linear in the flat). **Closed** closes up the corner as much as possible so the corner can be welded and results in more complicated geometry in the flat:

*Simple corner in the model and in the flat view, above*

*Closed corner in the model and in the flat view, above*

**Straight**

Align the hem along the sheet metal material.

Specifications:
**Flattened** - When checked, places the hem flush against the sheet metal material, using the Sheet metal model's Minimal gap for the space between hem and sheet metal.

**Inner radius** - When Flattened is unchecked, you can specify measurement of the inner radius of the hem.

**Total length** - The length of the hem from the outer most edge of the hem to the end of the hem.

**Hem alignment** - The alignment of the hem in relationship to the outer edge of the hem bend: **Outer** aligns the outside edge of the hem with the edge of the sheet metal. **In place** adds the outside edge beginning at the edge of the sheet metal:

![A rolled hem with Outer specified, first image, and In place specified, second image](image)

**Corner type** - Where hems come together on sheet metal, specify which type of corner to form: **Simple** results in a straight line in the flat view (linear in the flat). **Closed** closes up the corner as much as possible so the corner can be welded and results in more complicated geometry in the flat:

![A Simple corner in the model and in the flat view, above](image)
5. Click ✅ to accept the feature; the Hem is listed in the Feature list.

All dialog selections will default to previously specified values for subsequent hem operations, across sessions and documents.

When you have the flat view open, notice in the bend table that hems are listed there, along with their radius and angle. You can move them up or down in the table, but currently you cannot edit them in the table.

**Additional sheet metal tools**

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- **Tab** - Add a tab to a sheet metal flange.

- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- **Corner** - Modify a corner type and relief scale.

- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.

- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.
- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

---

**Sheet Metal Tab**

Add a tab to a sheet metal flange, remove interfering material if appropriate, or bridge two flanges from the same sheet metal model. Sheet metal tabs are reflected in the model and in the flat pattern.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sheet Metal Tab: Desktop**

In the Feature toolbar:

Add a tab to a sheet metal flange, remove interfering material if appropriate, or bridge two flanges from the same sheet metal model. Sheet metal tabs are reflected in the model and in the flat pattern.

**Steps**

To create a tab:
1. While in a Part Studio, with a sheet metal model active, click 

![Tab 1](image)

2. Select a sketch representing the tab parallel to the flange on which you wish to place the tab. Note that you can use multiple sketches.

3. In the **Flange to merge** field, select the flange on which you wish to place the tab. Note that you can select any number of flanges that are parallel to the sketch.

4. Click 

   to accept the feature; the Flange in the Feature list.

**Selecting two sketches on one flange (or wall):**

![Sketch Example](image)

Result:

![Result Example](image)

**Selecting one sketch parallel to two walls (flanges):**
Result:

Creating a tab with a Subtraction scope:
Use the Subtraction scope when removing interfering material. The sketch that spans two flanges:

The resulting tabs with the Subtraction scope selected (highlighted in orange):
The result (the Subtraction offset in this case is 0.05 - the minimal default):

The flat pattern:

Note that the Subtraction scope is not limited to the selection of sheet metal parts; normal parts can also be selected for the Subtraction scope.

**Bridging two flanges or walls**

The sketch that spans two flanges:
The resulting tabs with both flanges selected as the Flange to merge:

The result:

**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- 🛠 **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.
- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- **Tab** - Add a tab to a sheet metal flange.

- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- **Corner** - Modify a corner type and relief scale.

- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.

- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.

- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

**Sheet Metal Tab: iOS**

Add a tab to a sheet metal flange, remove interfering material if appropriate, or bridge two flanges from the same sheet metal model. Sheet metal tabs are reflected in the model and in the flat pattern.

**Steps**

To create a tab:

1. While in a Part Studio, with a sheet metal model active, tap **Tab**.
2. Select a sketch representing the tab parallel to the flange on which you wish to place the tab. Note that you are able to use multiple sketches.
3. In the **Flange to merge field**, select the flange on which you wish to place the tab. Note that you are able to select any number of flanges that are parallel to the sketch.

4. Tap ✅ to accept the feature; the Flange in the Feature list.

**Selecting two sketches on one flange (or wall)**

![Diagram of selecting two sketches on one flange](image)

Result:

![Result diagram](image)

**Selecting one sketch parallel to two walls (flanges)**

![Diagram of selecting one sketch parallel to two walls](image)

Result:
Creating a tab with a Subtraction scope

Use the Subtraction scope when removing interfering material. The sketch that spans two flanges:

The resulting tabs with the Subtraction scope selected (highlighted in orange):

The result (the Subtraction offset in this case is 0.05 - the minimal default):
The flat pattern:

Note that the Subtraction scope is not limited to the selection of sheet metal parts; normal parts are also able to be selected for the Subtraction scope.

**Bridging two flanges or walls**

The sketch that spans two flanges:

The resulting tabs with both flanges selected as the Flange to merge:
Additional sheet metal tools

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- Flange - Create a wall for each edge selected, connected to the selected edge with a bend.
- Tab - Add a tab to a sheet metal flange.
- Make joint - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- Corner - Modify a corner type and relief scale.
- Bend relief - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

Sheet Metal Make Joint

Join or extend existing sheet metal walls into a bend or rip (that is displayed in the Bend/Rip table), creating associativity between the two edges. You have the ability to configure and further define the style of the bend or rip once it is created. To modify an existing joint, use Modify joint.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Sheet Metal Make Joint: Desktop

In the Feature toolbar:

Join or extend existing sheet metal walls into a bend or rip (that is displayed in the Bend/Rip table), creating associativity between the two edges. You have the ability to configure and further define the style of the bend or rip once it is created. To modify an existing joint, use Modify joint.

Steps

To start a Sheet Metal Make joint operation:
1. While in a Part Studio, click 

![Make joint 1](image)

2. Select the side faces or edges of intersection walls to comprise the rip feature.

3. Select a rip style:
   a. Edge joint
   b. Butt joint - Direction 1
   c. Butt joint - Direction 2
      
      Note that only 90° joints can be styled as Butt joints; non-90° joints must be Edge joints.

4. To convert the rip to a bend, open the Sheet metal table and flat view

![Sheet metal table and flat view](image)
**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- **Tab** - Add a tab to a sheet metal flange.

- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- **Corner** - Modify a corner type and relief scale.

- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.

- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.
- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

**Sheet Metal Make Joint: iOS**

Join or extend existing sheet metal walls into a bend or rip (that is displayed in the Bend/Rip table), creating associativity between the two edges. You have the ability to configure and further define the style of the bend or rip once it is created. To modify an existing joint, use **Modify joint**.

**Steps**

To start a Sheet Metal Make joint operation:

1. While in a Part Studio, tap 🌋.
2. Select the side faces or edges of intersection walls to comprise the rip feature.
3. Select a rip style:
   a. Edge joint
   b. Butt joint - Direction 1
   c. Butt joint - Direction 2
      
      Note that only 90° joints can be styled as Butt joints; non-90° joints must be Edge joints.

4. To convert the rip to a bend, open the **Sheet metal table and flat view** 📖.
Additional sheet metal tools

- Flange - Create a wall for each edge selected, connected to the selected edge with a bend.
- Tab - Add a tab to a sheet metal flange.
- Make joint - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- Corner - Modify a corner type and relief scale.
- Bend relief - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

Sheet Metal Corner

Modify a corner on a sheet metal model by selecting an edge, vertex, or face of the existing corner.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Sheet Metal Corner: Desktop

In the Feature toolbar:

Modify a corner on a sheet metal model by selecting an edge, vertex, or face of the existing corner.

Steps

To modify a corner on a sheet metal model:

1. While in a Part Studio with an existing sheet metal model, click .
2. Select an edge, vertex, or face of a corner on the sheet metal model in the graphics area.

3. Select the type of corner relief:
   - **Square - Sized**
     [Flat view] [3D view]
   - **Rectangle - Scaled**
     [Flat view] [3D view]
   - **Round - Sized**
     [Flat view] [3D view]
   - **Round - Scaled**
     [Flat view] [3D view]
   - **Closed**
     [Flat view] [3D view]
   - **Simple**
     [Flat view] [3D view]

4. Specify:
   - Corner relief scale (for scaled corners) to scale the space: a value between 1.00 and 2.00.
   - Corner relief width (for sized corners) to size the space.

5. Click to accept the feature.

**Additional sheet metal tools**
When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.
- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.
- **Tab** - Add a tab to a sheet metal flange.
- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- **Corner** - Modify a corner type and relief scale.
- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.
- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.
- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

**Sheet Metal Corner: iOS**

Modify a corner on a sheet metal model by selecting an edge, vertex, or face of the existing corner.

**Steps**

To modify a corner on a sheet metal model:
1. While in a Part Studio with an existing sheet metal model, tap 

2. Select an edge, vertex, or face of a corner on the sheet metal model in the graphics area.

3. Select the type of corner relief:
   - **Square - Sized**
     - Flat view: 
     - 3D view: 
   - **Rectangle - Scaled**
     - Flat view: 
     - 3D view: 
   - **Round - Sized**
     - Flat view: 
     - 3D view: 
   - **Round - Scaled**
     - Flat view: 
     - 3D view: 
   - **Closed**
     - Flat view: 
     - 3D view: 
   - **Simple**
     - Flat view: 
     - 3D view: 

4. Specify:
   - Corner relief scale (for scaled corners) to scale the space: a value between 1.00 and 2.00.
   - Corner relief width (for sized corners) to size the space.
5. Tap ✔️ to accept the feature.

**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- Flange - Create a wall for each edge selected, connected to the selected edge with a bend.
- Tab - Add a tab to a sheet metal flange.
- Make joint - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- Corner - Modify a corner type and relief scale.
- Bend relief - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
- Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

---

**Sheet Metal Bend Relief**

Modify a bend relief (the small cut made where the bend end meets a free edge), and by specifying a shape, and either a depth and relief width scale, or specific bend relief depth. You are also able to select to extend the bend relief in the opposite direction in the case of a flange, for example (illustrations below).

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Sheet Metal Bend Relief: Desktop**
In the Feature toolbar:

Modify a bend relief (the small cut made where the bend end meets a free edge), and by specifying a shape, and either a depth and relief width scale, or specific bend relief depth. You are also able to select to extend the bend relief in the opposite direction in the case of a flange, for example (illustrations below).

**Steps**

To modify a bend relief on an existing sheet metal model:

1. While in a Part Studio with an existing sheet metal model, click 

2. Select any face, vertex, or edge of the bend end.

3. Specify a **Bend relief type**: 

   - Square - Sized
   - Rectangle - Scaled
   - Obround - Scaled
4. For **scaled bend reliefs** - Specify the Bend relief depth scale in the range 1.00 - 2.00 and the width scale in the range 0.0625 - 2.00

5. For **sized bend reliefs** - Specify the bend relief depth.

6. For bend reliefs with a flange and collision in the sheet metal flat pattern, use **Extend bend relief** to flip the direction of the relief cut and extend it to the end of the sheet metal:

   Select an edge or point of the bend relief, before extend:

   ![Extend bend relief](image)

   After extend:

   ![Extend bend relief after](image)

7. Click to accept the feature.

**Additional sheet metal tools**

When a sheet metal model is active (in the process of being created or edited), additional tools are available:
- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.

- **Tab** - Add a tab to a sheet metal flange.

- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).

- **Corner** - Modify a corner type and relief scale.

- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.

- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.

- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

**Sheet Metal Bend Relief: iOS**

Modify a bend relief (the small cut made where the bend end meets a free edge), and by specifying a shape, and either a depth and relief width scale, or specific bend relief depth. You are also able to select to extend the bend relief in the opposite direction in the case of a flange, for example (illustrations below).

**Steps**

To modify a bend relief on an existing sheet metal model:
1. While in a Part Studio with an existing sheet metal model, tap 

2. Select any face, vertex, or edge of the bend end.

3. Specify a **Bend relief type**:
   - Square - Sized
   - Rectangle - Scaled
   - Obround - Scaled
   - Obround - Sized:
   - Tear

4. For **scaled bend reliefs** - Specify the Bend relief depth scale in the range 1.00 - 2.00 and the width scale in the range 0.0625 - 2.00.

5. For **sized bend reliefs** - Specify the bend relief depth.

6. For bend reliefs with a flange and collision in the sheet metal flat pattern, use **Extend bend relief** to flip the direction of the relief cut and extend it to the end of the sheet metal:

   Select an edge or point of the bend relief, before extend:

   After extend:
7. Tap ✅ to accept the feature.

Additional sheet metal tools

When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- 📋 Flange - Create a wall for each edge selected, connected to the selected edge with a bend.
- 📋 Tab - Add a tab to a sheet metal flange.
- 📋 Make joint - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- 📋 Corner - Modify a corner type and relief scale.
- 📋 Bend relief - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
- 📋 Finish sheet metal model - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.
Sheet Metal Modify Joint

This functionality is currently available only on Onshape's browser platform.

Editing an existing sheet metal model's bends, rips, and joints using the Sheet metal table creates a Modify joint feature in the Feature list. Edit this feature as you would any other; right-click the feature entry to access the context menu.

In the Feature list

After converting a bend to a rip (using the table commands, explained above), a Modify joint feature is listed in the Feature list. You can right-click and Edit this new joint:

1. Right-click the Modify joint feature in the Feature list.
2. Select Edit.
3. In the dialog:
   a. Confirm the joint edge you want to modify (the Entity) or select a different one.
   b. Confirm the Joint type: Bend or Rip.
   c. For bends, keep the model properties default for the radius, or uncheck that box and specify a new Bend radius.
4. Click ✔️ to accept changes.

The Modify joint feature appears in the Feature list, indicated by this icon 🔄.

Additional sheet metal tools
When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.
- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.
- **Tab** - Add a tab to a sheet metal flange.
- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- **Corner** - Modify a corner type and relief scale.
- **Bend relief** - Modify a bend relief (the small cut made where the bend end meets the free edge), depth and relief width.
- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.
- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.
- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

### Sheet Metal Table and Flat View

This functionality is currently available only on Onshape’s browser platform.
Edit an existing sheet metal model’s bends, rips, and joints. View the sheet metal flat pattern.

**Editing sheet metal in the table**

1. With an existing sheet metal model in a Part Studio, click (midway along the right side of the window)

For example:
You can use the caret (next to the table title) to expand or collapse either table to make things easier to view.

2. Select the sheet metal model to edit from the dropdown at the top of the panel.
3. Click to select rows of the table; click again to deselect.
   - Notice the cross-highlighting: what you select in the table is highlighted in the flat panel and in the model and vice versa.
   - You are able to multi-select rows (click to select, click again to deselect).

   Selecting a rip or bend in the graphics area also selects (and scrolls to) the corresponding row in the table.

4. Right-click to access the context menu and commands: Move up/down in the table, Convert to rip/bend.

5. To change a rip joint type, use the dropdown menu on the far right of the table row.
   - Note that only 90° joints are able to be styled as Butt joints; non-90° joints must be Edge joints.

6. Double-click a radii value to enter a new one.

As you make edits, the flat pattern updates as well as the sheet metal model in the graphics area. Features are created in the feature list, for example, a Modify joint feature is created when you convert a bend to a rip.

**Editing sheet metal in the Feature list**

After converting a bend to a rip (using the table commands, explained above), a Modify joint feature is listed in the Feature list. You are able to right-click and Edit this new joint:

1. Right-click the Modify joint feature in the Feature list.
2. Select Edit.
3. In the dialog:

![Modify joint dialog]

a. Confirm the joint edge you want to modify (the Entity) or select a different one.
b. Confirm the Joint type: Bend or Rip.
c. For bends, keep the model properties default for the radius, or uncheck that box and specify a new Bend radius.
d. For rips, select the type of joint: Edge joint, Butt joint - Direction 1, or Butt joint - Direction 2.
   Note that only 90° joints can be styled as Butt joints; non-90° joints must be Edge joints.
4. Click ✅ to accept changes.

The Modify joint feature appears in the Feature list, indicated by this icon ⦿.

**Sketching on a flat pattern**

You have the ability to sketch directly on a flat pattern for the purpose of making marks used in sheet metal manufacturing. Right-click on the sheet metal flat pattern:

1. Select New sketch.
   The sketch dialog opens with the preselected face of the sheet metal flat pattern as the sketch plane.
2. Select a sketch tool and create a sketch as you normally would.
3. When finished sketching, click the checkmark.
When exporting a sheet metal flat pattern, you have the option of including sketches with the export.

**Extruding a flat pattern sketch**

When extruding on a flat pattern sketch, an abbreviated Extrude dialog box appears with options to add or remove material from the sheet metal. Click the checkmark to accept any changes.

![Extrude dialog box](image)

**Exporting DXF/DWG of flat pattern**

Right-click on the flattened view of the sheet metal flat pattern to select the option to export as a DWG or DXF file:
1. Right-click on the flat pattern, select Export DXF/DWG of flat pattern.

2. In the dialog:

   ![Export as DXF/DWG dialog]

   - Specify a file name.
   - Select a format, DWG or DXF.
   - Select a version.
   - Select a download option.
   - Select the parameters for your export:
     a. Export splines as polylines.
     b. Set z-height to zero and normals to positive.
     c. Include or hide bend centerlines.
     d. Include or hide bend tangent lines.
     e. Include or hide visible sketches - Select this to include any sketches created on the flat pattern.
     f. All model flat patterns to Zip of separate files.

3. Additional sheet metal tools

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All rights reserved.
When a sheet metal model is active (in the process of being created or edited), additional tools are available:

- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.
- **Hem** - Create a hem for each edge/face selected, on an existing sheet metal part.
- **Tab** - Add a tab to a sheet metal flange.
- **Make joint** - Convert the intersection of two walls into a joint feature, either a bend (walls joined by cylindrical geometry) or a rip (small gap between two walls).
- **Corner** - Modify a corner type and relief scale.
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- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

---

**Finish Sheet Metal Model**

This tool closes (deactivates) the sheet metal model and causes the sheet metal part to be treated like any non-sheet metal solid part. This allows you to perform...
post-fabrication operations on it, such as drilling at an angle through a face, welding corners, and adding form features and custom FeatureScript features.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Finish Sheet Metal Model: Desktop**

In the Feature toolbar:

This tool closes (deactivates) the sheet metal model and causes the sheet metal part to be treated like any non-sheet metal solid part. This allows you to perform post-fabrication operations on it, such as drilling at an angle through a face, welding corners, and adding form features and custom FeatureScript features.

Leaving the sheet metal model unfinished (active) causes the sheet metal parts to be treated as sheet metal. If you do not intend to perform post-fabrication operations on the sheet metal model, the Finish sheet metal model feature is unnecessary.

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- **Flange** - Create a wall for each edge selected, connected to the selected edge with a bend.

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**Finish Sheet Metal Model: iOS**

This tool closes (deactivates) the sheet metal model and causes the sheet metal part to be treated like any non-sheet metal solid part. This allows you to perform post-fabrication operations on it, such as drilling at an angle through a face, welding corners, and adding form features and custom FeatureScript features.

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- **Modify joint** - Make changes to an existing joint, such as converting a bend to a rip. Currently available through the flat view table.

- **Sheet metal table and flat view** - Open and close the Rip/Bend tables and the visualization of the sheet metal model flat pattern. Use this table to convert rips to bends and vice versa.

- **Finish sheet metal model** - Closes (deactivates) the Sheet metal model; creates a feature in the Feature list.

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**Add Custom Features**

Custom features (and custom tables) are written in a programming language called FeatureScript and are created in an Onshape tab called a **Feature Studio**. These custom features are able to be added to your Feature toolbar for use in documents to which you have write access. Custom tables are available immediately in the document in which they are created, and you can add them to any other document (from a version). See **Custom Tables** for more information on adding custom tables to Part Studios, and **FeatureScript Custom Tables** for information on creating custom tables in FeatureScript.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Add Custom Features: Desktop**

In the Feature toolbar:

Custom features are written in a programming language called FeatureScript and are created in an Onshape tab called a Feature Studio. These custom features are able to be added to your Feature toolbar for use in documents to which you have write access.

You have the ability to add custom features to your toolbar that were defined in other documents and add custom features to your toolbar that were defined in the same workspace.

**Steps: Custom features defined in other documents**

Add custom features to your Feature toolbar from any document to which you have a minimum permission set of “View, copy & export.” You have the ability to add custom features when you are in a Part Studio that you have write access to, or when you are viewing a version that contains the Feature Studio that defines the features.

1. While in a Part Studio, click [+] on the Feature toolbar.

2. In the dialog, locate the document in which the desired custom feature is defined:

   a. **FeatureScript samples** - This filter lists all Onshape-supplied documents containing FeatureScript samples for you to try.

   b. **Onshape standard filters** - Use these filters as you do on the Documents page to locate a specific document.

   c. **Search box** - Enter the name of a document or paste the URL of a document containing FeatureScript (usually acquired through a Share action).
Custom features are linked from specific versions of other documents; the latest version is selected by default. If there is no version, you are able to request that the document owner (or someone with write access) create one.

3. Clicking on any document name displays the custom features defined in it.

4. Select the top level icon to insert all custom features inside it (each represented by its own icon on your toolbar), or select one feature.

   The custom feature icon appears on the Feature toolbar.

   To remove the custom feature before closing the dialog, select the custom feature in the dialog again. (This toggles the custom feature in and out of the toolbar.)

5. If there is more than one version of the document, the latest version is displayed by default. Click  to access the version graph and select a different version.

6. Click the X in the upper-right corner of the dialog to close it.

   The custom feature is now available for use on your Feature toolbar.

Use a custom feature

Using a custom feature creates a feature in the Part Studio Feature list, just like any other Onshape feature. Custom features linked to from another document are indi-

ated in the Feature list by this icon .

Custom features behave like other Onshape features; they can be edited, suppressed, hidden, and deleted.

Update a custom feature

When a newer version of the document from which you inserted the custom feature is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab:

 This is a notification only and no action is required.

To update the version of the custom feature being used:
1. Click the update icon (or right-click the feature and select Update linked document) to access the Reference manager, in which you are able to choose update options.

2. To update to the latest version, click Update all.

3. To update to a specific version:
   a. Click Selective update:
   b. Select the document (if there is more than one) and click for that document:
   c. Select the version to update to.
   d. Repeat for any other documents in the list, if necessary.
   e. Click Update selected.

Remove a custom feature

To remove a custom feature from your toolbar, right-click the icon in the toolbar and select Remove.

The custom feature will no longer appear in the toolbar when editing Part Studios, unless that feature had been used and exists in the Feature list. If the Feature list contains a custom feature, and your toolbar does not have a corresponding custom feature icon (either because it was removed or the document was shared with you and your toolbar never contained the icon), the Linked custom features in this Part Studio icon appears in the toolbar and the custom feature is available from the dropdown:

This allows users with access to that Part Studio to continue to use that custom feature.

Open the document of a linked custom feature

To re-familiarize yourself with a custom feature in your document and how it works, right-click the custom feature icon in the toolbar and select Open linked document. The document containing the custom feature’s Feature Studio opens in another tab.

**Steps: Add custom features from the same workspace**
If you have written your own custom feature in this workspace, the custom features defined in it are automatically available for use from the Custom features drop down in the Feature toolbar:

1. Click the icon to list all custom features defined in the workspace:

![Feature toolbar]

2. Select the custom feature to use.

Custom features from the current workspace automatically update when the FeatureScript that defines the feature is edited and committed. This is useful for quickly testing when developing custom features.

Share a custom feature

To make your custom features available to others:

1. Create a version of the document containing the custom feature FeatureScript.
2. Either:
   a. Share the document with specific individuals or teams, specifying at least “Can view, copy & export” permissions.
   b. Make the document public.

**Using a custom icon for a custom feature**

A custom feature icon is specified by uploading the icon as an SVG blog tab in the document, and then referencing it. For more information, see the instructions in [Custom Icons](#) in the FeatureScript documentation.

**Tips**

- Onshape automatically lists custom features alphabetically.
- If you choose to update to a version that is not the latest, the ‘out of date’ icon remains.
- Updating the referenced version of a custom feature does not change the version of the custom feature pointed to by your toolbar. To update the version pointed to by
the toolbar, remove the existing icon and add a new custom feature that points to
the newer version.

- After updating a custom feature, you may have to edit the feature for it to regenerate
  without errors (for example, with the addition of fields that require input).
- Once you have used a linked custom feature in your document, you have access to
  it even if the source document is deleted or unshared.
- Adding custom features to your toolbar is an account setting and not a document
  setting. The icon (and associated custom feature) is available in all of your doc-
  uments.
- Adding a custom feature to your toolbar (or opening a document containing a cus-
  tom feature) automatically turns on the FeatureScript notices, indicated by [v] in the
  Navigation bar. These notices provide feedback that may be useful to the
  developer of the custom feature.

**Important**

FeatureScript has been designed with security in mind. To protect you, FeatureScript
runs in a tight sandbox and limits the impact of the feature to the Part Studio in which it
is used. This ensures that using custom features written even by untrusted users is rel-
atively safe.

A custom feature cannot:

- Modify anything other than the Part Studio in which it is used.
- “Infect” your Onshape account in any way.
- Communicate anything back to its author or anyone else.
- Affect Part Studio regeneration after it is removed from the Feature list.
- Modify other features in the Feature list.

A malicious (or poorly written) custom feature may:

- Take a long time to regenerate, or otherwise consume excessive resources, inter-
  fering with your ability to work with the Part Studio until you remove the custom fea-
  ture.
- Modify variable values or geometry in the Part Studio in an attempt to cause harm.
Publishing malicious FeatureScript is against the Onshape Terms of Use and will not be tolerated.

Please report malicious custom features using the Feedback button in the Help menu.

**Add Custom Features: iOS**

Custom features are written in a programming language called FeatureScript and are created in an Onshape tab called a [Feature Studio](#). These custom features are able to be added to your Feature toolbar for use in documents to which you have write access.

**Steps: Adding a custom feature**

Add custom features to your Feature toolbar from any document to which you have a minimum permission set of "Link document." You have the ability to add custom features when you are in a Part Studio to which you have edit access or when you are viewing a version that contains the Feature Studio that defines the features.

1. Tap on the Add custom features icon.

![Add custom features icon](#)

   The Add custom features flyout opens.

2. Tap to select between current workspace and browse documents.

3. Tap to select a document from which to add a custom feature.

4. Tap to select the version of the document.

5. Tap to select the custom feature you wish to add.

   Tap a drop down to select all of the custom features within it or tap individual custom features.

   The added custom feature is highlighted in blue in the flyout and it is added to your feature toolbar.

6. Tap the checkmark to close the flyout.

   Custom features linked to from another document are indicated in the Feature list by a link icon.
When a newer version of the document from which you inserted the custom feature is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab.

See Linked documents for more information on linked documents and how to update them.

To remove a custom feature from your feature toolbar:

1. Tap Manage in the lower right of your feature toolbar.
2. Tap the x next to the custom feature you wish to remove.
3. Tap Done when you are done managing the custom features.

Important

FeatureScript has been designed with security in mind. To protect you, FeatureScript runs in a tight sandbox and limits the impact of the feature to the Part Studio in which it is used. This ensures that using custom features written even by untrusted users is relatively safe.

A custom feature cannot:

- Modify anything other than the Part Studio in which it is used.
- "Infect" your Onshape account in any way.
- Communicate anything back to its author or anyone else.
- Affect Part Studio regeneration after it is removed from the Feature list.
- Modify other features in the Feature list.

A malicious (or poorly written) custom feature may:

- Take a long time to regenerate, or otherwise consume excessive resources, interfering with your ability to work with the Part Studio until you remove the custom feature.
- Modify variable values or geometry in the Part Studio in an attempt to cause harm.
Publishing malicious FeatureScript is against the Onshape Terms of Use and will not be tolerated.

Please report malicious custom features using the Contact Support button in the Help menu.

**Add Custom Features: Android**

Custom features are written in a programming language called FeatureScript and are created in an Onshape tab called a Feature Studio. These custom features are able to be added to your Feature toolbar for use in documents to which you have write access.

**Steps: Using an added custom feature**

You must add a custom feature to the Feature toolbar using Onshape on a browser before you are able to access it on a mobile device. See the Onshape Browser Help topic Custom Feature for more info.

On a mobile device in a Part Studio:

1. Tap the Features icon to access the Feature toolbar.

   The custom feature added using Onshape on a browser, is available in the Feature toolbar on your mobile device.

2. Tap to use the custom feature.

3. Complete the dialog and tap the checkmark.

4. The custom feature that has been used is listed in the Feature list, just like any other Onshape feature.

Custom features behave like other Onshape features; they are able to be edited, suppressed, hidden, and deleted.

Custom features linked to from another document are indicated in the Feature list by a link icon.

When a newer version of the document from which you inserted the custom feature is created, the link icon in the Feature list highlights in blue, and an identical icon
appears on the Part Studio tab.

See Linking Documents for more information on linked documents and how to update them.

**Important**

FeatureScript has been designed with security in mind. To protect you, FeatureScript runs in a tight sandbox and limits the impact of the feature to the Part Studio in which it is used. This ensures that using custom features written even by untrusted users is relatively safe.

A custom feature **does not have the ability to**:

- Modify anything other than the Part Studio in which it is used.
- "Infect" your Onshape account in any way.
- Communicate anything back to its author or anyone else.
- Affect Part Studio regeneration after it is removed from the Feature list.
- Modify other features in the Feature list.

A malicious (or poorly written) custom feature **may**:

- Take a long time to regenerate, or otherwise consume excessive resources, interfering with your ability to work with the Part Studio until you remove the custom feature.
- Modify variable values or geometry in the Part Studio in an attempt to cause harm.

Publishing malicious FeatureScript is against the Onshape Terms of Use and will not be tolerated.

Please report malicious custom features using the Contact Support button in the Help menu.
Assemblies

This functionality is available on Onshape's browser, iOS, and Android platforms.

An Onshape Assembly tab is where you define a hierarchical structure of part and sub-assembly instances of an Assembly. It is also where you define degrees of freedom and relations. You are able to have more than one Assembly tab in a document. One Assembly can instance another Assembly as a subassembly, and/or instance a part directly. You are able to instance parts from the same document or other documents to which you have permissions (and that are versioned).

**Assemblies: Desktop**

To create additional Assemblies, use the plus sign menu at the bottom of the window (shown below, under the blue arrow) and select Create Assembly.

To learn more about creating Assemblies in Onshape, you can follow the self-paced course here: [Onshape Assemblies](#) (opens in new tab).

**Assembly toolbar**
The Assembly toolbar is active when an Assembly tab is active.

The types of mates shown on the toolbar (Fastened mate, Slider mate, etc) are collectively referred to as Mates.

Access the Assembly shortcut toolbar with the S key while in an Assembly:

Customize the toolbar through your Onshape account [Preferences page](#).

You can **customize the Onshape toolbars** in Part Studios, Assemblies and Feature Studios - shown below in a Part Studio:

- Hover anywhere in the toolbar and right-click, then select Customize toolbar...

This activates the ability to edit the toolbar:

- Tools are highlighted in tool sets within the toolbar; you can drag and drop these tool sets to new locations on the toolbar:
Create a new tool set by dragging a tool icon to the New toolset box that appears when you begin to drag the icon:

Drag and drop an individual tool out of the toolbar to the Tools box to remove it from the toolbar (you can always drag and drop it back onto the toolbar). Shown below during the drag operation:

After drag is completed:

Click Save to save your changes, Cancel to close without saving, or Reset to default to undo all changes in that type of toolbar (Part Studio, Assembly, or Feature Studio) and restore the toolbar to Onshape original order and content.

When tools are part of a group, you must move the entire group. Once an entire group is removed, you can select individual tools to move back to the toolbar, if
To reform the entire group, select Reset to default, which will reset the entire toolbar.

Search tools

Shortcut: alt + C

In Onshape, you have the ability to search for tools in the Assembly toolbar as well as the Feature toolbar and Sketch toolbar: see Document Toolbar and Document Menu, Feature tools and Sketch tools for more information on those specific topics.

Basic steps to assembling parts

1. "Insert Parts and Assemblies" on page 1430.
2. Create "Mate Connector: Desktop" on page 1255.
3. Create "Mates" on page 1544.
4. Create "Relations: Desktop" on page 1664 if desired.

The tools and functionality available for Assemblies include:

- "Insert Parts and Assemblies: Desktop" on page 1431 dialog - For selecting parts and subassemblies to include in an assembly
- "Triad Manipulator: Desktop" on page 244 - For moving parts and assemblies around the graphics areas and for movement between parts
- "Mates" on page 1544 - For defining movement between parts
- "Mate Connector: Desktop" on page 1255 - For defining where parts connect to each other
- "Snap Mode: Desktop" on page 1622 - Drag and drop parts to create Mate connectors and Mates on the fly.
- "Group" on page 1618 - For defining spatial relationships between parts
- "Assembly List: Desktop" on page 1459 - A list of part instances and mate features in an Assembly

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- Context Menus in Assemblies - Select from a list of actions
- "Assembly Measure tool: Desktop" on page 1694 - Acquire measurement information about part edges and faces

**Assembly context menu**

Right-click on an Assembly tab to access the context menu:

- **Delete** - the tab, even if it is active. The last remaining tab cannot be deleted.
- **Open in new browser tab** - Open this Assembly in a new browser tab.
- **Rename...** - Rename this Assembly.
- **Properties...** - Access the dialog to provide information about the Assembly. In the Properties dialog, you can provide meta data for the entire Assembly. Properties that are grayed out (inactive) are defined and populated through the Company’s properties in Account management. See [Manage Companies > Properties](#) for more information.
- **Duplicate** - Copy this Assembly tab and insert the copy into this same document. All references to the original Assembly or other Assemblies are maintained.
- **Copy to clipboard** - Make a copy of this Assembly tab on the clipboard. You can then use the menu in another document and the Paste tab command to add the Assembly tab into that document. When an Assembly tab is copy/pasted into another document, the Part Studio and Assemblies from which it was created are also pasted into the other document. No references to the original document are maintained.
- **Create Drawing of x...** - Automatically create a drawing of the entire Assembly (solid bodies/parts only). This creates a new Drawings tab in the document.
- **Select as document thumbnail** - Use this assembly image as the thumbnail for the document on the Documents page.
- **Move to document...** - Move the Assembly to a new document, creating the document during this operation. If any part or assembly is used in any tab of the original document, a link between the two documents is created. Note that, the Assembly tab and the Part Studios from which the part instances are referenced
will all move to the new document. This action will be prevented if it would result in a document with no tabs.

- **Export** - Export parts in the Assembly in a variety of formats with options of where to download or keep in a separate Onshape tab.

- **Release** - For user who have the ability to create release packages, this command opens the Release candidate dialog.

**Reconfiguring instances and subassemblies**

Access context menus on individual instances and subassemblies in the Assemblies list to rearrange them into new subassemblies and/or move them to new subassemblies. Note that when copying and moving subassemblies, all mates are maintained.

If the action (of moving or copying) would result in the modification of a linked (external) document, the action will fail. Linked (external) documents are immutable versions and are not able to be modified.
If you want to replace a part in the assembly with a different part not yet in the assembly, use the **Replace part** tool

**Move to new subassembly**

1. Right-click on the part to move and select Move to new subassembly.

2. A new Assembly appears in the list and a new Assembly tab appears in the document. The part is placed within the new Assembly in the list, and the Assembly is automatically named Assembly $X <x>$ where $X$ is the next consecutive Assembly number in the document and $<x>$ is the instance of that assembly in the current Assembly tab.
A new Assembly tab is created. Use the context menu on the tab to rename the assembly, if desired.

A new Mate Features entity is also created and placed directly within the new Assembly in the list. If the part had no mates, this new Mate Feature is empty.

Create new subassembly

1. Right-click in the Assembly list and select Create new subassembly.

2. A new Assembly appears in the list, without any parts or mate features. The new assembly is automatically named Assembly X <x> where X is the next consecutive Assembly number in the document and <x> is the instance of that assembly in the current Assembly tab.

You are also able to drag and drop entities to reconfigure subassemblies (into and out of other subassemblies) and to rearrange the order of entities in the list.

Assemblies: iOS
Newly created documents contain an empty Part Studio and Assembly by default. An Onshape Assembly is where you define a hierarchical structure of part and sub-assembly instances of an assembly. It is also where you define how they are able to move. You are able to have more than one Assembly tab in a document. One assembly is able to instance another assembly as a sub-assembly, and/or instance a part directly.

Return to Documents Page

To close the document and return to the documents page, tap the arrow in the upper left corner.

Versions

To create immutable versions and branch versions to create new workspaces, tap the Versions icon in the upper left.

See Version Manager for more info.

History

To view every point of change in the history of the document and restore to any point in that history:

1. Tap the icon in the upper left.
   
   On a smaller screen, access the Document information panel and then select History from there:

2. Tap the More icon.

3. Select History.
See History for more info.

Comments

To communicate with collaborators, tap the Comments icon in the upper right.

On a smaller screen, access the Document information panel and then select Comments from there:

1. Tap the More icon.

2. Select Comments.

See Comments for more info.

Collaborators

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To see who you are collaborating with and to access Follow mode, tap the Collaborators icon in the upper right.

See [Collaboration](#) and [Follow mode](#) for more info.

The Collaborators icon is only available when you are collaborating with someone in real time.

**Help**

To access the Help and other resources such as Videos, Feedback, and About, tap the icon in the upper right.

On a smaller screen, access the Document information panel and then select Help:

1. Tap the More icon.

2. Select **Help**.

Select one of the following:

- **Help** - to view the documentation for the device you are using.
- **Videos** - open the videos page of the documentation.
- **Contact Support** - ask a question, log a bug, or log an improvement.
- **About** - see which version of Onshape you are running.

**Document information panel**
Access the Document information panel which allows you access to any tools (listed above) that do not fit in the navigation bar as well as some other tools listed below.

To access the Document information panel, tap the More icon in the upper right corner.

The Document information panel opens. From here, select from the following:

- **Properties** - to view document properties.
- **Copy** - to make a copy of the workspace you are in.
- **Units** - to view and set default measurement units for the document (such as length, angle, and mass units).
- **Document name** - tap the edit icon in the upper right corner to edit the document name.
- **Document description** - tap Add description to add or edit the document description.
- **Share** - tap the share icon to share the document with individuals, Teams, Companies, or Onshape support. You are also able to set your document to Private or Public.

**Insert parts and assemblies**

Tap the Insert tool to insert a part or assembly into the active Assembly.

See [Insert Parts and Assemblies](#) for more info.

**Mates and Relations**

Tap the Mates tool to view the list of mates, relations, and other Assembly tools.

See [Mates](#) for more info.

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See [Relations](#) for more info.

**Measure and Mass Properties**

Tap the Measure tool to measure any selected entities.

See [Measure tool](#) for more info.

Tap the Mass Properties tool to view the properties of selected parts.

See [Mass Properties](#) for more info.

**Assembly list**

- Tap the Assembly list handle to open the Assembly list.
- Touch and drag horizontally or vertically to adjust the size of the Assembly list.

See [Assembly list](#) for more info.

**Rotate Lock**

Tap the 3D Rotate lock button to lock the graphics area rotate feature. This is particularly helpful when trying to drag an entity.

**View Cube**
Tap the View cube to access a list of views to choose from.

See View Cube for more info.

**Tabs**

When you open a document, the most recently opened tab is active.

Tap the up-facing chevron (up arrow) to open the Create tab menu.

- Tap a tab to activate it. When you switch tabs, any open feature will be committed.
- Swipe left or right to scroll, horizontally, through the list.
- Filter tabs by Assembly or Part Studio.
- Search for a Part Studio or Assembly by name.
- Create, rename, duplicate, and delete a Part Studio or Assembly. There is no limit to how many Part Studios or Assemblies a document can have.
Duplicate adds a copy of a Part Studio within the document. You are unable to delete a Part Studio or Assembly if it is the only tab in the document. A document must have at least one Part Studio or Assembly (at least one tab).

**Assemblies: Android**

Newly created documents contain an empty Part Studio and Assembly by default. An Onshape Assembly is where you define a hierarchical structure of part and sub-assembly instances of an assembly. It is also where you define how they can move. You can have more than one Assembly tab in a document. One assembly can instance another assembly as a sub-assembly, and/or instance a part directly.

**Return to Documents Page**

To close the document and return to the documents page, tap the arrow in the upper left corner.

**Versions**
To create immutable versions and branch versions to create new workspaces, tap the Versions icon in the upper left.

See Version Manager for more info.

History

To view every point of change in the history of the document and restore to any point in that history:

1. Tap the More icon to access the Document information panel.
2. Tap History.

See History for more info.

Comments

To communicate with collaborators, tap the Comments icon in the upper right.

See Comments for more info.
Collaborators

To see who you are collaborating with, tap the Collaborators icon in the upper right. See Collaboration for more info.

The Collaborators icon is only available when you are collaborating with someone in real time.

Help

To access the Help and other resources such as Videos, Feedback, and About, tap the icon in the upper right.

Select one of the following:

- **Help** - to view the documentation for the device you are using.
- **Videos** - open the videos page of the documentation.
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Document information panel

Access the Document information panel which allows you access to any tools (listed above) that do not fit in the navigation bar as well as some other tools listed below.

To access the Document information panel, tap the More icon in the upper right corner.
The Document information panel opens. From here, select from the following:

- **Properties** - to view document properties.
- **Copy** - to make a copy of the workspace you are in.
- **Units** - to view and set default measurement units for the document (such as length, angle, and mass units).
- **Document name** - tap the pencil icon in the upper right corner to edit the document name.
- **Document description** - tap the pencil icon to add or edit the document description.
- **Share** - tap the share icon to share the document with individuals, Teams, Companies, or Onshape support. You are also able to set your document to Private or Public.

**Insert parts and assemblies**

Tap the Insert tool to insert a part or assembly into the active Assembly.

See [Insert Parts and Assemblies](#) for more info.

**Mates and Relations**

Tap the Mates tool to view the list of mates, relations, and other Assembly tools.

See [Mates](#) for more info.

See [Relations](#) for more info.

**Measure and Mass Properties**

Tap the Measure tool to measure any selected entities.
See Measure tool for more info.

Tap the Mass Properties tool to view the properties of selected parts.
See Mass Properties for more info.

**Assembly list**

- Tap the Assembly list handle to open the Assembly list.
- Touch and drag horizontally or vertically to adjust the size of the Assembly list.

**Rotate Lock**

Tap the 3D Rotate lock button to lock the graphics area rotate feature. This is particularly helpful when trying to drag an entity.

**View Cube**
Tap the View cube to access a list of views to choose from.

See View Cube for more info.

**Tabs**

When you open a document, the most recently opened tab is active.

Tap the up-facing chevron (up arrow) to open the Create tab menu.
Tap a tab to activate it. When you switch tabs, any open feature will be committed.

- Swipe left or right to scroll, horizontally, through the list.
- Filter tabs by Assembly or Part Studio.
- Search for a Part Studio or Assembly by name.
- Create, rename, duplicate, and delete a Part Studio or Assembly. There is no limit to how many Part Studios or Assemblies a document can have.

Duplicate adds a copy of a Part Studio within the document.
You are unable to delete a Part Studio or Assembly if it is the only tab in the document.
A document must have at least one Part Studio or Assembly (at least one tab).

Insert Parts and Assemblies

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This functionality is available on Onshape's browser, iOS, and Android platforms.

Insert Parts and Assemblies inserts instances of parts, assemblies, sketches and surfaces into the active Assembly. You can instance specific parts (released or unreleased), sketches and surfaces defined in a Part Studio (or the entire Part Studio), and assemblies defined in a different Assembly tab, as well as all of these things from other documents (linking documents). Filters for each object type are available in the dialog when the corresponding entities exist in the current document or other documents (being browsed).

**Insert Parts and Assemblies: Desktop**

Shortcut: i

The Assembly toolbar is active when an Assembly tab is active.

Insert Parts and Assemblies inserts instances of parts, assemblies, sketches and surfaces into the active Assembly. You are able to instance specific parts, sketches and surfaces defined in a Part Studio (or the entire Part Studio), and assemblies defined in a different Assembly tab, as well as all of these things from other documents (linking documents). Filters for each object type are available in the dialog when the corresponding entities exist in the current document or other documents (being browsed).

During insertion, the default positioning is the alignment of the Part Studio origin (of the part or subassembly being inserted) with the Assembly origin (of the assembly being inserted into) for the first instance, with subsequent instances of the same part being inserted at a slight offset.

Fixing a part is different from applying a mate. Fix (found in the context menu for a part instance) is specific to the assembly in which it is applied; it does not carry over to any other assembly that part is inserted into.
Note that when deleting an instance or feature from an assembly, all related features (Mate connectors, Mates, relations) are also deleted. The only exception is Mate groups, these are not deleted.

**Inserting from current document**

1. Click the Insert parts and assemblies tool.

![Insert parts and assemblies tool](image)

2. Select **Current document** to insert from this document or **Other documents** to find another document from which to select.

You can also select to insert Onshape-supplied standard parts from **Standard content**.
3. Once a document is selected, select **Part Studios** or **Assemblies** to filter further.

a. Within **Part Studios**, you can insert the entire Part Studio or select specific parts, sketches and surfaces. Use the icons to filter on sketches , surfaces , parts , and composite parts , or use the search field to locate a specific part.

b. Within **Assemblies**, use the search field to locate an assembly by name.

i. You can search by part/assembly name and other properties (including Custom properties).

ii. Search accepts multiple words as well as non-alphanumeric characters, such as punctuation.

iii. The filter to the left of the Search bar shows common properties (or all properties) to search by. You are able to use more than one filter at a time.

iv. Each search result indicates which type of entity fulfills the search criteria by the icon preceding the name, followed by the name of the entity. Below that, the workspace or version icon, the document name and workspace name (or version name, when part of the search criteria), the part number, and the release management state (shown below).

![Part Studio Icon](image)

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- **View released items** to insert a particular revision of an object into the assembly.

The Instances list in the Assembly will reflect the parts that have been previously released with a solid triangle:
4. Any parts that have configurations are shown with their options to select from in order to apply a specific configuration at the time of insertion:

If you click the parts to insert and then accept to close the dialog, the parts are inserted into the Assembly with the Assembly and Part Studio origins aligned.

When inserting more than one instance of a part, the subsequent instances are inserted at a slight offset from the first.

However, if you move the cursor into the graphics area, then your selection appears at your cursor.

When viewing Parts in this dialog, you see the default detail view of the Part Studio, under that, the individual parts in that Part Studio. Click on the Part Studio name to insert the entire contents of the Part Studio, or click individual parts. This works the same for assemblies.
5. Drag to reposition the parts, assemblies or sketches.

When inserted into an Assembly, sketches can have Mate connectors and be assembled with parts, other sketches, and assemblies.

**Changing configurations**

After a part with configurations has been inserted into an Assembly, you have the ability to change the configuration of it:

1. Right-click on the part (or the part name in the Instances list) and select **Change configuration**.

   A Change configuration dialog opens:

   ![Change configuration dialog](image)

2. Select a new configuration option.

3. Click ✔️ when you are satisfied with your selection. (Use ✗ to cancel the operation.)

   To see which configuration is currently active in the Assembly, hover over a part in the Instances list and a tooltip appears with the configuration information:
Note that you are unable to change the configuration in the tooltip; follow the instructions directly above.

**Assembling immediately**

When a component you are inserting has a Mate connector already defined (from within the Part Studio) and you have Snap Mode turned on in the Assembly, then:

- When dragging the component into the Assembly from the Insert dialog, you can snap the source Mate connector on the component to other Mate connectors in the assembly. (These appear upon hover.)
- If the component being inserted has more than one explicitly-defined Mate connector, you are able to use the Control key to cycle through the Mate connectors and stop on the appropriate one to use as the new Snap Mode source Mate connector.
- If you snap to a target Mate connector and accept the insert (closing the dialog), a Fastened Mate is applied between the source and target Mate connectors.
- You may pan and rotate freely as you insert a part instance (or subassembly), even in Snap mode.

If no explicitly-defined Mate connectors are on the component being inserted, then a normal non-snap free drag is available, even if Snap mode is turned on.

**Using linked or revisioned parts or assemblies**

Once inserted into an assembly, a revisioned part can be updated to reflect an updated revision. Click on the icon in the Instances list to open the Reference manager. There you are able to **Update to the latest**, or make a **Selective update**. Parts that have a newer version available have this icon in the Instances list: 🔄. Click the icon to drill down to the revision you wish to select:
- **Version graph** to insert from a particular version of the document

- **Create version** to create a new version of the document on the fly, in order to select a versioned entity

Double-click a part in the Instances list to open a create or edit in-context session for that part.

**Tips**

When inserting parts and assemblies, you are able to search within other documents:
• Notice that the same filters are available here as are on the Documents page, including teams and labels.

• Drill down by selecting a filter, and then a document, or use the Search box if you know the name of the entity you are looking for.

• Once a part, sketch, or surface is inserted into an Assembly, you can change the version of a particular entity: right-click on the entity name in the Instances list and select Change to version... in the context menu.

**Insert Parts and Assemblies: iOS**

**Steps**

1. Tap the Insert tool.
2. Select **Current Workspace** to insert from this document workspace or select **Browse documents** to find another document from which to insert.

a. If you select Browse documents, select a filter or search to find the document from which to insert, then tap to select the document.

You are able to select from other documents only if that document has one or more versions. A notice is displayed regarding the state of the document if no version exists or if a newer version exists. If no version exists, tap to create a version in that document.
b. You have the ability also choose to select a previously-released part or assembly to insert. Released objects are indicated by a solid triangle icon to the right:

3. Select **Parts** or **Assemblies** to view the document's Part Studios or Assemblies.
   
a. Within Part Studios, tap to insert an entire Part Studio or tap to select specific parts.
   
b. Within Assemblies, tap to insert an Assembly.

When viewing Parts, you see the default detail view of the Part Studio, under that, the individual parts in that Part Studio. Tap on the Part Studio name to insert the entire contents of the Part Studio, or tap individual parts. This works the same for assemblies. The insert dialog displays an "Inserted" count that updates with each inserted entity. You can insert multiple entities into an Assembly at a time.

4. Any parts that have configurations are shown with their options to select from in order to select the desired configuration at the time of insertion into the Assembly.
Configurations are created in the Onshape on a browser, and can be seen and worked with on an iOS device.

5. Optionally, tap the left facing arrow (chevron) in the upper right of the Insert dialog to collapse the dialog and view the Assembly.

6. Tap the checkmark.

The default positioning of the inserted entities is the alignment of the Part Studio origin (of the part or subassembly being inserted) with the Assembly origin (of the assembly being inserted into).

If you have inserted a part or assembly from another document, a linked icon appears next to the Derived feature in the Feature list to indicate that it is linked to another document.
When a newer version of the document from which you inserted part studio or assembly is created, the link icon in the instance list highlights in blue, and an identical icon appears on the Assembly tab.

See Linking Documents for more info on linked documents and how to update them.

Tips

- The default positioning of inserted entities is the alignment of the Part Studio origin (of the part or subassembly being inserted) with the Assembly origin (of the assembly being inserted into).

- If a part has been given Mate connectors in a Part Studio, it will have them when it is inserted into the Assembly. A part with Mate connectors explicitly defined in a Part Studio will have a drop down in the Assembly Instance list that contains the Mate connectors which belong to the part.

- Fixing a part is different from applying a mate. Fix (found in the context menu for a part) is specific to the assembly in which it is applied; it does not carry over to any other assembly that part is inserted into.

- When deleting an instance or feature from an assembly, all related features (Mate connectors, Mates, relations) are also deleted. The only exception is Mate groups, these are not deleted.

Insert Parts and Assemblies: Android

Steps

1. Tap the Insert tool.

2. Select Current Workspace to insert from this document workspace or select Browse documents to find another document from which to insert.
a. If you select Browse documents, select a filter or search to find the document from which to insert, then tap to select the document.

You have the ability to select from other documents only if that document has one or more versions. A notice is displayed regarding the state of the document if no version exists or if a newer version exists. If no version exists, tap to create a version in that document.

a. You are also able to choose to select a previously-released part. Use the solid triangle icon to filter on documents with released Parts or Assemblies:
3. Select **Parts** or **Assemblies** to view the document's Part Studios or Assemblies.

   a. Within Part Studios, tap to insert an entire Part Studio or tap to select specific parts.

   b. Within Assemblies, tap to insert an Assembly.

   When viewing Parts, you see the default detail view of the Part Studio, under that, the individual parts in that Part Studio. Tap on the Part Studio name to insert the entire contents of the Part Studio, or tap individual parts. This works the same for assemblies.

   The insert dialog displays an "Inserted" count that updates with each inserted entity. You are able to insert multiple entities into an Assembly at a time.

4. Any parts that have configurations are shown with their options to select from in order to select the desired configuration at the time of insertion into the Assembly:
Configurations are created in the Onshape on a browser, and can be seen and worked with on an Android device.

5. Optionally, tap the left facing arrow (chevron) in the upper right of the Insert dialog to collapse the dialog and view the Assembly.

6. Tap the checkmark.

The default positioning of the inserted entities is the alignment of the Part Studio origin (of the part or subassembly being inserted) with the Assembly origin (of the assembly being inserted into).

If you have inserted a part or assembly from another document, a linked icon appears next to the Derived feature in the Feature list to indicate that it is linked to another document.

When a newer version of the document from which you inserted part studio or assembly is created, the link icon in the instance list highlights in blue, and an identical icon appears on the Assembly tab.
See Linking Documents for more info on linked documents and how to update them.

**Tips**

- The default positioning of inserted entities is the alignment of the Part Studio origin (of the part or subassembly being inserted) with the Assembly origin (of the assembly being inserted into).

- If a part has been given Mate connectors in a Part Studio, it will have them when it is inserted into the Assembly. A part with Mate connectors explicitly defined in a Part Studio will have a drop down in the Assembly Instance list that contains the Mate connectors which belong to the part.

- Fixing a part is different from applying a mate. Fix (found in the context menu for a part) is specific to the assembly in which it is applied; it does not carry over to any other assembly that part is inserted into.

- When deleting an instance or feature from an assembly, all related features (Mate connectors, Mates, relations) are also deleted. The only exception is Mate groups, these are not deleted.

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**Linking Documents**

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Linking Documents: Desktop**

You can insert a part or assembly from a version of one Onshape document into an assembly in another Onshape Document, thereby linking the documents. Moving a part from one document to another creates a link to that part in any Assembly in which the part is previously inserted. You can also insert a part or assembly from a different
version of the same document. (Using the Derived features also creates a link between documents or between versions of a single document.)

Linking documents allows you to create references from one document to data in a version of another document. For example, an assembly in Document A can instance a part defined in version V1 of Document B.

There are no changes to the behavior of parts and assemblies that are all defined within one Onshape Document. Changes to parts instantly propagate to assemblies within the same document. However, you control exactly whether and when to update references to newer versions of the part or assembly in the document in which you have inserted the part or assembly.

Linking documents in this manner is especially valuable when designs mature and you want to apply different permissions and version control to the parts and assemblies defined in other documents. It is also useful for any reuse of standard parts and assemblies.

To learn more about using linked documents in Onshape, follow the self-paced course here: External References (opens in new tab)

**How it works**

Consider a case where one Onshape document (UsingDoc) contains an assembly that instances a part that resides in another Onshape document (RefDoc). In Onshape, versions are always immutable, so anything defined in a version of RefDoc is stable and recoverable. Since linked document references are to versions, this means that every change in the history of UsingDoc is also stable and recoverable. And, as a result, versions created in UsingDoc are also stable and recoverable. This is a fundamental architectural advantage Onshape has relative to traditional file-based CAD.

Instead of changes to RefDoc propagating into UsingDoc with no recourse, you are informed when new versions are available and you choose whether to use them or not. If it becomes obvious that the new version causes a problem, you can use the document history to restore back to a prior working state.

A key aspect of linking documents is that all of Onshape’s document permissions work seamlessly. You decide when a document should change from Editable to View
only to Reference only on a per user basis - and you are always able to change permissions whenever you want.

If a newer version of the part or assembly becomes available in the source document, you are notified via the icon in the document into which you inserted the part or assembly. A blue background appears around the linked icon to indicate that a newer version is available.

Steps

In an Assembly:

1. Click Insert to open the Insert dialog.

2. The default is to insert from the current workspace. To insert a part or assembly from another document, select Other documents.

You can insert from another document only when that document has versions. If the selected document doesn’t have any versions, or you do not have edit permission (to create a version) Onshape displays a notification and allows you to version the document immediately:

Click the link to open the Create version dialog for that document:
Enter the appropriate and necessary information, click Create and return to the Insert dialog.

3. Select a part or assembly from the document.

4. Click the check to close the dialog.

If a newer version of the part or assembly becomes available in the source document, you are notified via the icon in the document into which you inserted the part or assembly. A blue background appears around the linked icon to indicate that a newer version is available.

**Opening a linked document**

Once a part is inserted from another document, creating a linked document, you are able to open the document from which the part is linked, if needed, for editing the part or obtaining a closer inspection of the part:

1. Right-click on the part in the Instances or Features list and select Open linked document.

2. A new browser tab opens, with the linked document open to the version that was linked to and the Part Studio of the part you selected active.

3. Make whatever inspection you need, or edit the part, if desired.

4. If you edit the part, in the tab you'll see the blue icon that indicates another version of the part is available. Click that icon to open the Reference manager.

**Errors you may encounter**

If the linked document you are trying to open is in the Trash, you will get an error:  
*Failed to load document for workspace. Cannot open a document in the trash.*
Restore the document from Trash.

If the document is permanently deleted, you will see the message: Failed to load document for version. Resource does not exist, or you do not have permission to access it.

If you do not have permission to edit the linked document, you will see this message: You cannot modify this feature because you cannot access the referenced document.

Tips
- While the case described here is for Linked Documents, you can reference parts and assemblies defined in different versions on the same document as well.
- Select a linked item in the Feature list, right-click and select Open linked document to open the linked document in a new tab in your browser window.
- To allow another user to link to your document, share the document with at least Read/Copy/Export permissions (or higher).
- If you then unshare the document (remove a user from the list in the Share dialog), the removed user is blocked only from updating to a newer version and creating links to that. Any links already used will still work.

Linking Documents: iOS

How it works
Consider a case where one Onshape document (UsingDoc) contains an assembly that instances a part in another Onshape document (RefDoc). In Onshape, versions are always immutable, so anything defined in a version of RefDoc is stable and recoverable. Since Linked document references are to versions, this means that every change in the history of UsingDoc is also stable and recoverable. And, as a result, versions created in UsingDoc are also stable and recoverable. This is a fundamental architectural advantage Onshape has relative to traditional file-based CAD.

Instead of changes to RefDoc propagating into UsingDoc with no recourse, the user is informed when new versions are available and chooses whether to use them or not. If it becomes obvious that the new version causes a problem, you can use the document history to restore a working prior state.
A key aspect of linking documents is that all of Onshape’s document permissions work seamlessly. You decide when a document should change from Editable to View only to Reference only on a per user basis - and you always have the ability to change permissions whenever you want.

**How to link documents**

There are three ways in which you can link documents.

- **Insert Parts and Assemblies**
  
  In an Assembly, tap the Insert Part and Assemblies icon:

  ![Insert Parts and Assemblies icon](image)

  Follow the steps in the [Insert Parts and Assemblies](#) topic to select a part or assembly to insert.

- **Derive**
  
  In a Part Studio, tap the Derived tool:

  ![Derived tool](image)

  Follow the steps in the [Derived](#) topic to select a part to insert.

- **Add Custom Feature**
  
  In a Part Studio, tap the Add Custom feature tool:

  ![Add Custom Feature](image)

  Follow the steps in the [Custom Feature](#) topic to select a custom feature to add to the feature toolbar.

Inserted parts or assemblies, derived parts, and custom features linked to from another document are indicated in the Instance list or the Feature list by a link icon. Tabs will also have a link icon to indicate that something in that tab is linked to another document.

![Link icon](image)

When a newer version of the document from which you inserted a part or
assembly, derived a part, or added a custom feature is created, the link icon highlights in blue. This indicates that you are able to update your document to the newest version of the document to which it is linked.

**Updating linked documents**

When a reference document is updated, Onshape adds an updated link icon next to the part in the Instance list, feature in the Feature list, and also in the respective tab.

Tap the overflow menu next to an updated link icon (in the Feature list, Instance list, or on a tab) and select **Update** to access the Reference manager.

Reference manager

You have the ability to either update all of your references to their latest versions at once or select specific references to update.

**Update to latest** - to update all of your references to their latest versions:

1. Tap Update to latest.
   
   Available newer versions are listed.

2. Tap Update All.

**Selective update** - to choose specific references to update to their latest versions:

1. Tap Selective update.
   
   Available newer versions are listed, with checkboxes.

2. Tap to select one or more items from the list.

3. Tap Update Selected.

 Updating linked documents

When a referenced document is updated, Onshape adds a linked icon next to the part in your document:
The gray link icon indicates a part referenced (linked) from another document. The white link icon with the blue background indicates that a part referenced from another document now has another version. This notification is also visible on the Assembly tab.

To update a linked document:

Where you see a white link icon with a blue background (either in the instance list or in the Assembly tab) tap the overflow menu or tap the tab.

Where you see a white link icon with a blue background (either in the instance list or in the assembly tab) tap the overflow menu.

Tap to select **Update** or **Update link** from the overflow menu.
Tips

- While the case described here is for Linked Documents, you are also able to reference parts and assemblies defined in different versions on the same document as well.

- Select a linked item in the Feature list, tap the overflow menu, and select Open linked document to open the linked document in a new tab in your document.

- To allow another user to link to your document, share the document with at least Read/Copy/Export permissions (or higher).

- If you then unshare the document (remove a user from the list in the Share dialog), the removed user is blocked only from updating to a newer version and creating links to that. Any links already used will still work.

Linking Documents: Android

How it works

Consider a case where one Onshape document (UsingDoc) contains an assembly that instances a part in another Onshape document (RefDoc). In Onshape, versions are always immutable, so anything defined in a version of RefDoc is stable and recoverable. Since Linked document references are to versions, this means that every change in the history of UsingDoc is also stable and recoverable. And, as a result, versions created in UsingDoc are also stable and recoverable. This is a fundamental architectural advantage Onshape has relative to traditional file-based CAD.

Instead of changes to RefDoc propagating into UsingDoc with no recourse, the user is informed when new versions are available and chooses whether to use them or not. If it becomes obvious that the new version causes a problem, you are able to use the document history to restore a working prior state.
A key aspect of linking documents is that all of Onshape’s document permissions work seamlessly. You decide when a document should change from Editable to View only to Reference only on a per user basis - and you are always able to change permissions whenever you want.

**How to link documents**

There are three ways in which you are able to link documents.

- **Insert Parts and Assemblies**
  
  In an Assembly, tap the Insert Part and Assemblies icon:
  
  ![Insert Part and Assemblies icon]

  Follow the steps in the [Insert Parts and Assemblies](#) topic to select a part or assembly to insert.

- **Derive**
  
  In a Part Studio, tap the Derived tool:
  
  ![Derived tool]

  Follow the steps in the [Derived](#) topic to select a part to insert.

- **Add Custom Feature**
  
  In a Part Studio, tap the Add Custom feature tool:
  
  ![Add Custom feature tool]

  Follow the steps in the [Custom Feature](#) topic to select a custom feature to add to the feature toolbar.

Inserted parts or assemblies, derived parts, and custom features linked to from another document are indicated in the Instance list or the Feature list by a link icon. Tabs will also have a link icon to indicate that something in that tab is linked to another document.

When a newer version of the document from which you inserted a part or
assembly, derived a part, or added a custom feature is created, the link icon highlights in blue. This indicates that you are able to update your document to the newest version of the document to which it is linked.

**Updating linked documents**

When a reference document is updated, Onshape adds an updated link icon next to the part in the Instance list, feature in the Feature list, and also in the respective tab.

Tap the overflow menu next to an updated link icon (in the Feature list, Instance list, or on a tab) and select **Update** to access the Reference manager.

**Reference manager**

You have the ability to either update all of your references to their latest versions at once or select specific references to update.

**Update to latest** - to update all of your references to their latest versions:

1. Tap Update to latest.
   
   Available newer versions are listed.

2. Tap Update All.

**Selective update** - to choose specific references to update to their latest versions:

1. Tap Selective update.
   
   Available newer versions are listed, with checkboxes.

2. Tap to select one or more items from the list.

3. Tap Update Selected.

**Updating linked documents**

When a referenced document is updated, Onshape adds a linked icon next to the part in your document:
The gray link icon indicates a part referenced (linked) from another document. The white link icon with the blue background indicates that a part referenced from another document now has another version. This notification is also visible on the Assembly tab.

To update a linked document:

Where you see a white link icon with a blue background (either in the instance list or in the Assembly tab) tap the overflow menu or tap the tab.

Where you see a white link icon with a blue background (either in the instance list or in the assembly tab) tap the overflow menu.

Tap to select **Update** or **Update link** from the overflow menu.
Tips

While the case described here is for Linked Documents, you are also able to reference parts and assemblies defined in different versions on the same document as well.
Select a linked item in the Feature list, tap the overflow menu, and select Open linked document to open the linked document in a new tab in your document.

To allow another user to link to your document, share the document with at least Read/Copy/Export permissions (or higher).

If you then unshare the document (remove a user from the list in the Share dialog), the removed user is blocked only from updating to a newer version and creating links to that. Any links already used will still work.

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**Assembly List**

The list in an Assembly tab contains a list of all instances, Groups, Mate connectors and Mates defined for the Assembly. Use the context menu to act any of these entities.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Assembly List: Desktop**

The list in an Assembly tab contains a list of all instances, Groups, Mate connectors and Mates defined for the Assembly. Use the context menu to act any of these entities.
To find an object in the Assembly list, click on it in the graphics area. The list automatically scrolls to the selected object and highlights it (indicated in the image below with blue arrows). If the selected object is not expanded in the list, right-click it in the graphics area and select **Go to item in list**. If the item is already visible and expanded in the list, this menu command will not be present.
The Assembly list consists of a list of Parts instances and a list of Mate features:

- The Instances section lists all part instances inserted into the Assembly. They are listed by name and with the instance number in brackets <2>. For example, a part (Housing) that has been inserted into an Assembly twice would be listed as Housing <1> and Housing <2>. If you change the name of a part in a Part Studio, that change is reflected in the Assembly as well.

- Mate features include:
  - **Mate connectors** - Specified points on a part used to position parts in an Assembly.
  - **Mates** - Specify the degrees of freedom between two Mate connectors.
  - **Groups** - Instances rigidly grouped together.

- You can act on the Instances and Mate features in the Feature list:
  - **Hide/Show** - To more easily view parts and their Mate connectors, you have the ability to hide parts that may obscure other parts. Right-click on the instance in the Feature list and click Hide, or hover over the instance name and click the eye icon.
  - **Fix** - Fix a part in place, right-click on the instance name and click Fix. (To remove the fix, right-click again and click Unfix.) When an instance is fixed, this icon appears beside it in the list:
  - **Drag** - Move a part instance name or subassembly name in the Feature list to a new location in the list.

- Go directly to the Part Studio the part was built in; right-click on the instance name and click Switch to <Part Studio name>.

- **Suppress** - Suppress a mate, part instance, or subassembly through the Feature list context menu or the context menu available on the feature in the graphics area. When a part or assembly is suppressed, all mates associated with that part or assembly become inactive.

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- **Properties** - Access the Properties dialog (for viewing and editing properties) for a particular part or assembly in the Instances list. Right-click the part (or assembly) name and select Properties.

- You can select the entire Assembly by clicking the Assembly name (shown below to the right of the blue arrow) at the top of the Instances list.

- You can also right-click on the Assembly name in the Instances list to open a context menu:

  The context menu allows you to perform such actions as:
  
  - Properties - Edit properties for the selected item.
  - Rename - Change the name of the selected item.
  - Check interference - Check to see if any parts in the assembly are interfering with other parts.
- Create Drawing - Create an Onshape Drawing of the selected item.
- Add exploded view - Create an exploded view of the assembly.
- Export - Export the selected items.
- Release - Create a release candidate for the selected items.
- Revision history - View the revision history of the selected items. If no revision history exists, a message to that effect is displayed.
- Add comment - Add a comment on the selected items, tagging them in the comment.

Should you have any errors in your assembly, they will be displayed in the Instances list with a red icon (as shown below):

Instances (3)
- Assembly 1
  - Origin
  - Part 1 <1>
- Circular pattern 1
  - Part 1 <2>
- Items (0)
- Mate Features (2)

Click the dropdown arrow to the left of the error icon to see where the error is located:
To get more information about the error, hover your cursor over the error:

Instances (3)
- Assembly 1
  - Origin
  - Part 1 <1>
- Circular pattern 1
  - Part 1 <2>
- Items (0)
- Mate Features (2)
  - Fastened 1
  - Fastened 2
Any time you hover over a mate or Mate connector, you see read-only information about the mate or Mate connector. To edit the mate or Mate connector, double-click it in the list.

Click the Instances and mates icon on the right side of the Instances list (shown below, to the left of the blue arrow) to collapse the list entirely. Click on the icon again to reopen the list.

Tips
Drag and drop any assembly instance into (or out of) another, or drag it to the top level.

You can right-click on an assembly instance for more actions, including restructuring commands:

- **Move to new subassembly** to create a new Assembly tab and insert this assembly into it automatically
- **Create new subassembly** to reinsert this assembly into this same Assembly again

**Improve tessellation quality, or decrease tessellation quality to improve performance** - You are able to use the context menu to *Use best available tessellation* to improve the rendering quality of the part, and conversely, *Use automatic tessellation setting* to allow the system to select a tessellation quality for balanced performance purposes. This will also be specified in Part Studios through the "Customizing Parts, Faces, and Features: Appearance" on page 302.

Keep in mind that the tessellation quality is determined in the Part Studio and is then used by the Assembly. The Assembly may use a lower quality tessellation in order to increase performance if necessary, but will never use a higher quality than what is set in the Part Studio.

A part or assembly that has tessellation turned on is indicated by this icon 🥇 in the Instances list. For example:
Assembly List: iOS

The list in an Assembly tab contains a list of all Parts instances, Groups, Mate connectors, Mates, and Relations defined for the Assembly. It is called the Instance list.
• Tap the Instance list handle to open the Instance list to default width.
• Touch and drag the handle vertically or horizontally to set the size (height or width) of the Instance list.
• Tap the handle to close the Instance list, tap again to open it to its previously set width.

The workspace is still active when the Instance list is open.

**Working with the Instance list**

The Instance list contains a list of all Parts instances, Groups, Mate connectors, Mates, and Relations defined for the Assembly. There are many ways to work with the Instance list:

• The **Instances** lists all part instances inserted into the Assembly. They are listed by name and with the instance number in brackets <2>. For example, a part (Housing) that has been inserted into an Assembly twice would be listed as Housing <1> and Housing <2>. If you change the name of a part in a Part Studio, that change is reflected in the Assembly as well.

• Mate features include:
  
  • **Mate connectors** - Specified points on a part used to position parts in an Assembly.
  
  • **Mates** - Specify the degrees of freedom between two Mate connectors.
• **Groups** - Instances rigidly grouped together.
• **Relations** - Constrain degrees of freedom between mates.

You are able to act on the Instances and Mate features in the Instance list:

• **Hide/Show** - To more easily view parts and their Mate connectors, you have the ability to hide parts that may obscure other parts. Tap on the overflow menu icon to the right of the instance in the Instance list and tap **Hide**, or tap the 🕵️.

• **Fix** - Fix a part in place, tap on the overflow menu icon to the right of the instance name and tap **Fix**. (To remove the fix, tap the overflow menu again and tap **Unfix**.)

• **Suppress** - Suppress a mate, part instance, or subassembly through the Instance list overflow menu or the context menu available on the feature in the graphics area.

• **Drag** - Move a part instance name or subassembly name in the Instance list to a new location in the list: tap **Edit** in the upper right of the Instance list, then touch and drag the icon to the right of the instance or subassembly you want to move.

**Tips**

• Drag and drop any assembly instance into (or out of) another, or drag it to the top level.

• You are also able to tap to select an assembly instance, then two finger tap or tap the context menu icon (three vertical dots) to bring up the context menu for more actions, including restructuring commands:

  • **Move to new subassembly** to create a new Assembly tab and insert this assembly into it automatically

  • **Create new subassembly** to reinsert this assembly into this same Assembly again

**Assembly List: Android**

The list in an Assembly tab contains a list of all Parts instances, Groups, Mate connectors, Mates, and Relations defined for the Assembly. It is called the Instance list.
• Tap the Instance list handle to open the Instance list to default width.
• Touch and drag the lower right corner of the Instance list border to set the size (height or width) of the Instance list.
• Tap the handle to close the Instance list, tap again to open it to its previously set width.

The workspace is still active when the Instance list is open.

Working with the Instance list

The Instance list contains a list of all Parts instances, Groups, Mate connectors, Mates, and Relations defined for the Assembly. There are many ways to work with the Instance list:

• The **Instances** lists all part instances inserted into the Assembly. They are listed by name and with the instance number in brackets <2>. For example, a part (Housing)
that has been inserted into an Assembly twice would be listed as Housing <1> and Housing <2>. If you change the name of a part in a Part Studio, that change is reflected in the Assembly as well.

- **Mate features include:**
  - **Mate connectors** - Specified points on a part used to position parts in an Assembly.
  - **Mates** - Specify the degrees of freedom between two Mate connectors.
  - **Groups** - Instances rigidly grouped together.
  - **Relations** - Constrain degrees of freedom between mates.

- **You are able to act on the Instances and Mate features in the Instance list:**
  - **Hide/Show** - To more easily view parts and their Mate connectors, you have the ability to hide parts that may obscure other parts. Tap on the overflow menu icon to the right of the instance in the Instance list and tap **Hide**, or tap the eye icon.
  - **Fix** - Fix a part in place, tap on the overflow menu icon to the right of the instance name and tap **Fix**. (To remove the fix, tap the overflow menu again and tap **Unfix**.)
  - **Suppress** - Suppress a mate, part instance, or subassembly through the Instance list overflow menu or the context menu available on the feature in the graphics area.
  - **Drag** - Move a part instance name or subassembly name in the Instance list to a new location in the list: tap the icon in the upper right of the Instance list, then touch and drag the icon to the right of the instance or subassembly you want to move.

**Tips**

- Drag and drop any assembly instance into (or out of) another, or drag it to the top level.
- You are also able to tap to select an assembly instance, then two finger tap or tap the context menu icon (three vertical dots) to bring up the context menu for more actions, including restructuring commands:
• **Move to new subassembly** to create a new Assembly tab and insert this assembly into it automatically
• **Create new subassembly** to reinsert this assembly into this same Assembly again

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### Managing Assemblies

This functionality is currently available only on Onshape's browser platform.

To aid in the process of assembling parts, Onshape provides some convenient tools:

• **Hide (selection)**
• Hide other parts
• Hide all parts
• Isolate (selection)
• Hide part on hover shortcut key “y”
• Make transparent

Use these commands to access the parts and mates required for your tasks, instead of painstakingly finding and moving parts and subassemblies out of the way to access the relevant entities.

Use the shortcut key “y” to hide a part under the cursor (part will show highlighting on hover). To show the part again, use the context menu “show all” or “show all parts” commands.

Access these commands from the context menu for selected parts in an assembly.

The examples use this model:
**Hiding parts**

Hide all parts, selected parts, or ‘all other’ parts excluding those selected to aid in visualizing necessary entities for assembling or evaluating movement of an assembly. Select parts in the graphics area or from the Parts list. You are also able to use box select for selecting multiple parts. This command is modal: hide/show.

**Hide example**

Select the parts to hide and click Hide in the context menu. The selected parts before Hide:
The model after Hide:

Hide other parts example

Select the parts you want to visualize and click Hide other parts in the context menu:
**Isolating parts**

Isolate works similarly to Hide, with the difference that unselected parts remain visually present for reference, but muted in color and unavailable for selection until you exit Isolate mode. Any Mate connectors and Mates of non-selected parts are also muted and unavailable for selection. As with the Hide commands, you are able to select the parts in the Feature list, graphics area, and with the box select functions. All Mates and Mate connectors of selected parts are also available for selection in the Mate process.

Use Isolate with individual parts, multiple parts, and groups.
This command is modal: Isolate/Exit Isolate.

As with the Hide commands, you can use box select.

**Steps**

1. Select a part (or parts) to isolate, either in the Feature list or on the model.

2. Right-click in empty space and select Isolate from the context menu:

3. The selected part is visible and the remainder of the model is faded out. The Isolate dialog opens:
4. Use the dropdown and sliding scale to include more top-level parts or sub-assemblies in the Isolate command:

a. **Expand: connectivity** - Expands the Isolate command to bring into view the top-level parts and subassemblies that are mated to the selected part. The first drag brings the entire subassembly into view. The second drag brings another top-level part or entire subassembly into view.
b. **Expand: distance** - Expands the Isolate command to bring into view other parts based on distance in all directions, incrementally, from the center of the selected part. The parts are not necessarily mated to each other. The slider may bring into view more than one part at a time.

**Making parts transparent**

When modeling, you may need to see a part that is occluded visually by another part.
or parts, in order to perform some task. To this end, Onshape provides the "Make transparent" command in the context menu for parts and assemblies.

Use these commands to access the parts required for your tasks, instead of painstakingly finding and hiding to access the relevant entities.

Use the shortcut key SHIFT+“t” to make a part under the cursor transparent (the part under the cursor is highlighted). This command is cumulative; you can use it on part after part, keep all transparent until you select "Exit make transparent" from the context menu, or close the dialog that opens. You can also multi-select parts and then select the command.

A dialog box opens when you initiate the command:

In a Part Studio, you can Expand by distance to use the slider to make each successive part transparent as you drill through to the part you need.

In an Assembly, you can Expand by distance (as in a Part Studio) or by connectivity to use the slider to make each successively connected part transparent as the slider moves.

This command is also available when editing part in-context. For more information on in-context editing, see "Modeling In-Context" on page 1719.
Standard Content

Onshape standard content is created by Onshape, kept in documents within Onshape and maintained by Onshape. There is never any risk of this content disappearing or not working from one release to another. We have integrated time-saving process in this workflow to simplify the insertion of standard content.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Supported standard content includes:

- **ANSI** - Bolts and screws, nuts, and washers
- **DIN** - Bolts and screws, nuts, washers, and shaft keys
- **ISO** - Bolts and screws, nuts, and washers
- **PEM®** - from PennEngineering® including their Unified category of self-clinching nuts

Standard content is identified in the Assembly list with this icon:

![Standard Content Icon]

**Standard Content: Desktop**

Shift+Insert, Alt+Insert

Select standard content directly from the filter provided in the Insert parts and assemblies dialog in an Onshape Assembly.

**Steps**
1. Click the **Standard content** filter on the Insert parts and assemblies dialog.

2. Select the specifics about the content you want to insert: **Standard**, **Category**, **Class**, and **Component**.

3. Select 📝 (the auto-size icon) and then a hole on the model to automatically select the proper size in the dialog.
   
   Or, select the size manually from the dropdown.

   If there is no standard part that matches the size of the hole selected, a message is displayed:

   ```none
   Unable to find matching size for selection. Selection may be too small or large for specific component.
   ```

4. The part number and description, if present in the system, are automatically supplied. If you wish, click **Edit** to activate the Part number and Description fields and type metadata regarding the standard content part. This information will then be
available for that part for all users of the Onshape account.

This field is company-wide if you are part of a company, or limited to your user account if you are not part of a company.

A preview of the selected part is shown.

5. Click **Insert** to automatically insert the part if a planar or cylindrical face or an edge is already selected. If no selection is made, the part appears at your mouse location: drag it to the preferred location and click to snap to a Mate connector (inferred or existing).

Use the **Insert closest to selection** and **Insert furthest from selection** icons when inserting subsequent standard content to automatically place the content in the stack of content mated to the selected part. **To flip the orientation of the part** being inserted, press the 'a' key on your keyboard.

In the example below, Insert closest to selection will insert at the shorter (leftmost) arrow and Insert furthest from selection will insert at the longer (rightmost) arrow:

6. Click 🔄 on the dialog to save your actions and create instances of the standard content in the assembly and list it in the Instances lists (along with the Mate connector, inferred or existing).

**How it works**

When inserting, Onshape has the ability to detect a mated bolt or screw, and if a washer is selected for insertion into the same hole, Onshape places and fastens the washer at the bottom of the bolt or screw head.
Bolts have only one reference point available (called a mate). Washers and nuts have two mates available, so they are able to be stacked and you have the ability to insert at the **closest or furthest from selection**. When inserting a second washer, for example, if you select Closest to selection (or use Alt+Insert) the washer is fastened to the top of the selected washer, against the bolt head. If you select Furthest from selection (or use Shift+Insert) the washer is fastened to the bottom of the selected washer, against the part.

**Pre-selection**, or the lack thereof, impacts the insert process for standard content. In the absence of selection (on the model), clicking Insert initiates drag mode. The standard content appears at your mouse cursor and you drag it to the desired location, mate references will appear, and click to place the part.

To facilitate the insertion process, you have the ability to pre-select individual holes (their circular edge or cylindrical face) to place standard content specifically. Make your selection prior to clicking the Insert button on the dialog.

You also have the ability to pre-select a planar face. In this case, upon clicking the Insert button, the standard content is fastened to all holes on the selected faces, whether the size of the selected standard content is appropriate or not.

If the holes you select have been created with the Onshape Hole feature, the inserted standard content with find the 'fine' or 'coarse' thread definition of the feature and apply it. If the hole was not created through the Hole feature, Onshape defaults to 'coarse'.

**Editing the standard content instance**

There may be times when you want to edit a particular instance of standard content in an assembly:

1. Right-mouse click on the part in the assembly or the instance in the Instance list.
2. Select Edit standard content instance from the menu.
3. The Insert dialog opens and you are able to change whichever characteristics you wish; you may want to change the material or the length, for example.
4. Click ✅ to save your changes (or click Cancel to close the dialog without saving changes).

**Standard Content: iOS**
Steps

1. Tap the **Standard content** tab on the Insert parts and assemblies dialog.

2. Select the specifics about the content you want to insert: **Standard**, **Category**, **Class**, and **Component**.

3. Select the size.

   If there is no standard part that matches the size of the hole selected, a message is displayed:

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4. A preview of the selected part is shown.

5. Tap **Insert** to automatically insert the part if a planar or cylindrical face or an edge is already selected. If no selection is made, the part appears at the origin. Drag it to the preferred location and click to snap to a Mate connector (inferred or existing).

6. Tap the check on the dialog to save your actions and create instances of the standard content in the assembly and list it in the Instances lists (along with the Mate connector, inferred or existing).

Once an instance of standard content is inserted into an Assembly, it is listed in the Assembly list. To edit the instance, tap the three-button menu and select Edit to open the Insert parts and assemblies dialog again. Make your new selections and tap Update to change the part instance in the Assembly.

**How it works**

When inserting, Onshape has the ability to detect a mated bolt or screw, and if a washer is selected for insertion into the same hole, Onshape places and fastens the washer at the bottom of the bolt or screw head.

Bolts have only one reference point available (called a mate). Washers and nuts have two mates available, so they are able to be stacked and you have the ability to insert at the **closest or furthest from selection**. When inserting a second washer, for example, if you select Closest to selection (or use Alt+Insert) the washer is fastened to the top of the selected washer, against the bolt head. If you select Furthest from selection (or use Shift+Insert) the washer is fastened to the bottom of the selected washer, against the part.

**Pre-selection**, or the lack thereof, impacts the insert process for standard content. In the absence of selection (on the model), clicking Insert initiates drag mode. The standard content appears at your mouse cursor and you drag it to the desired location, mate references will appear, and click to place the part.

To facilitate the insertion process, you have the ability to pre-select individual holes (their circular edge or cylindrical face) to place standard content specifically. Make your selection prior to clicking the Insert button on the dialog.
You also have the ability to pre-select a planar face. In this case, upon clicking the Insert button, the standard content is fastened to all holes on the selected faces, whether the size of the selected standard content is appropriate or not.

If the holes you select have been created with the Onshape Hole feature, the inserted standard content will find the 'fine' or 'coarse' thread definition of the feature and apply it. If the hole was not created through the Hole feature, Onshape defaults to 'coarse'.

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There may be times when you want to edit a particular instance of standard content in an assembly:

1. Right-mouse click on the part in the assembly or the instance in the Instance list.
2. Select Edit standard content instance from the menu.
3. The Insert dialog opens and you have the ability to change whichever characteristics you wish; you may want to change the material or the length, for example.
4. Click to save your changes (or click Cancel to close the dialog without saving changes).

**Standard Content: Android**

**Steps**
1. Tap the **Standard content** tab on the Insert parts and assemblies dialog.

2. Select the specifics about the content you want to insert: **Standard**, **Category**, **Class**, and **Component**.

3. Select the size.

   If there is no standard part that matches the size of the hole selected, a message is displayed:

   ```
   Unable to find matching size for selection. Selection may be too small or large for specific component.
   ```

4. A preview of the selected part is shown.
5. Tap **Insert** to automatically insert the part if a planar or cylindrical face or an edge is already selected. If no selection is made, the part appears at the origin. Drag it to the preferred location and click to snap to a Mate connector (inferred or existing).

6. Tap the check on the dialog to save your actions and create instances of the standard content in the assembly and list it in the Instances lists (along with the Mate connector, inferred or existing).

Once an instance of standard content is inserted into an Assembly, it is listed in the Assembly list. To edit the instance, tap the three-button menu and select **Edit** to open the Insert parts and assemblies dialog again. Make your new selections and tap Update to change the part instance in the Assembly.

**How it works**

When inserting, Onshape has the ability to detect a mated bolt or screw, and if a washer is selected for insertion into the same hole, Onshape places and fastens the washer at the bottom of the bolt or screw head.

Bolts have only one reference point available (called a mate). Washers and nuts have two mates available, so they are able to be stacked and you have the ability to insert at the **closest or furthest from selection**. When inserting a second washer, for example, if you select Closest to selection (or use Alt+Insert) the washer is fastened to the top of the selected washer, against the bolt head. If you select Furthest from selection (or use Shift+Insert) the washer is fastened to the bottom of the selected washer, against the part.

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To facilitate the insertion process, you have the ability to pre-select individual holes (their circular edge or cylindrical face) to place standard content specifically. Make your selection prior to clicking the **Insert** button on the dialog.

You also have the ability to pre-select a planar face. In this case, upon clicking the **Insert** button, the standard content is fastened to all holes on the selected faces, whether the size of the selected standard content is appropriate or not.
If the holes you select have been created with the Onshape Hole feature, the inserted standard content with find the 'fine' or 'coarse' thread definition of the feature and apply it. If the hole was not created through the Hole feature, Onshape defaults to 'coarse'.

**Editing the standard content instance**

There may be times when you want to edit a particular instance of standard content in an assembly:

1. Right-mouse click on the part in the assembly or the instance in the Instance list.
2. Select Edit standard content instance from the menu.
3. The Insert dialog opens and you are able to change whichever characteristics you wish; you may want to change the material or the length, for example.
4. Click ✓ to save your changes (or click Cancel to close the dialog without saving changes).

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**Standard Content Appendix**

This functionality is available on Onshape's browser, iOS, and Android platforms.

This lists all of the standard content Onshape provides for your convenience. If the standard content you require is listed here, you don't have to model it - you can simply use the Insert parts and assemblies dialog to select the standard content you need. Check back here often to see what is available, since Onshape updates this list often.

**ANSI**

**Bolts & Screws**

<table>
<thead>
<tr>
<th>Countersunk bolts</th>
<th>Countersunk bolt, Countersunk square neck bolt</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set screws (hex socket)</td>
<td>Oval point hex socket set screw, Half dog point hex socket set screw, Flat point hex socket set screw, Cup point hex socket set screw, Cone point hex socket set screw</td>
</tr>
<tr>
<td>Category</td>
<td>Examples</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>--------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Set screws (square head)</td>
<td>Oval point square head set screw, Half dog point square head set screw,</td>
</tr>
<tr>
<td></td>
<td>Flat point square head set screw, Dog point square head set screw, Cup</td>
</tr>
<tr>
<td></td>
<td>point square head set screw, Cone point square head set screw</td>
</tr>
<tr>
<td>Set screws (slotted)</td>
<td>Oval point slotted headless set screw, Half dog point slotted headless</td>
</tr>
<tr>
<td></td>
<td>set screw, Flat point slotted headless set screw, Dog point slotted</td>
</tr>
<tr>
<td></td>
<td>headless set screw, Cup point slotted headless set screw, Cone point</td>
</tr>
<tr>
<td></td>
<td>slotted headless set screw</td>
</tr>
<tr>
<td>Socket head screws</td>
<td>Socket countersunk head cap screw, Socket button head cap screw, Socket</td>
</tr>
<tr>
<td></td>
<td>head cap screw, Socket head shoulder screw</td>
</tr>
<tr>
<td>Lag screws</td>
<td>Hex lag screw, Square lag screw</td>
</tr>
<tr>
<td>T-head bolts</td>
<td>T-head bolt</td>
</tr>
<tr>
<td>Square head bolts</td>
<td>Askew head bolt, Countersunk elevator bolt, Square head bolt</td>
</tr>
<tr>
<td>Round head bolts</td>
<td>Round head ribbed neck bolt, Round head step bolt, Round head fin neck</td>
</tr>
<tr>
<td></td>
<td>bolt, Round head short square neck bolt, Round head square neck bolt,</td>
</tr>
<tr>
<td></td>
<td>Round head bolt</td>
</tr>
<tr>
<td>Tapping screws</td>
<td>Oval countersunk trim head tapping screw, Hex washer head tapping screw,</td>
</tr>
<tr>
<td></td>
<td>Pan head tapping screw, Hex head tapping screw, Fillister head tapping</td>
</tr>
<tr>
<td></td>
<td>screw, Flat countersunk trim head tapping screw, Oval countersunk head</td>
</tr>
<tr>
<td></td>
<td>tapping screw, Oval undercut countersunk head tapping screw, Flat</td>
</tr>
<tr>
<td></td>
<td>undercut countersunk head tapping screw, Flat countersunk head tapping</td>
</tr>
<tr>
<td></td>
<td>screw</td>
</tr>
<tr>
<td>Machine screws</td>
<td>Round head machine screw, Truss head machine screw, Flat countersunk</td>
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<tr>
<td></td>
<td>head (100) machine screw, Binding head machine screw, Fillister head</td>
</tr>
<tr>
<td></td>
<td>machine screw, Flat countersunk head (82) machine screw, Hex head machine</td>
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<td></td>
<td>screw, Hex washer machine screw, Oval countersunk head machine screw,</td>
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<td></td>
<td>Pan head machine screw</td>
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<tr>
<td>Category</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Hex bolts</td>
<td>Hex cap screw, Hex flange bolt, Hex bolt, Heavy hex structural bolt, Heavy hex finished bolt, Heavy hex bolt</td>
</tr>
<tr>
<td>Miniature screws</td>
<td>Pan head miniature screw, Countersunk flat head miniature screw, Fillister head oval crown miniature screw, Fillister head chamfered miniature screw, Binding head miniature screw</td>
</tr>
<tr>
<td>Nuts</td>
<td></td>
</tr>
<tr>
<td>Hex slotted nuts</td>
<td>Hex slotted nut, Hex thick slotted nut, Heavy hex slotted nut</td>
</tr>
<tr>
<td>Cap nuts</td>
<td>Acorn high, Acorn low crown nut, Acorn high crown nut</td>
</tr>
<tr>
<td>Hex nuts</td>
<td>Hex jam nut, Hex thick nut, Hex nut, Hex flange nut, Heavy hex nut, Heavy hex jam nut</td>
</tr>
<tr>
<td>T-slot nuts</td>
<td>T-slot nut</td>
</tr>
<tr>
<td>Square nuts</td>
<td>Square nut, Heavy square nut</td>
</tr>
<tr>
<td>Machine screw nuts</td>
<td>Square machine screw nut, Hex machine screw nut</td>
</tr>
<tr>
<td>Hex flat nuts</td>
<td>Hex flat nut, Hex flat jam nut, Heavy hex flat nut, Heavy hex flat jam nut</td>
</tr>
<tr>
<td>Washers</td>
<td></td>
</tr>
<tr>
<td>Spring lock washers</td>
<td>Extra-duty spring lock washer, Regular spring lock washer, High collar spring lock washer, Heavy-duty spring lock washer</td>
</tr>
<tr>
<td>Toothed lock washers</td>
<td>Type B internal/external tooth lock washer, Type B heavy internal tooth lock washer, Type B external tooth lock washer, Type A internal/external tooth lock washer, Type A heavy internal tooth lock washer, Type A external tooth lock washer, Type B internal tooth lock washer, Type A internal tooth lock washer</td>
</tr>
<tr>
<td>Type A plain washers</td>
<td>Type A flat washer</td>
</tr>
<tr>
<td>Type B plain washers</td>
<td>Type B wide flat washer, Type B regular flat washer, Type B narrow flat washer</td>
</tr>
</tbody>
</table>
Pins & Studs

**Dowel & straight pins**
- Taper pin, Hardened ground machine dowel pin,
- Hardened ground production dowel pin, Ground dowel pin, Straight pin

**Spring pins**
- Slotted spring pin

**Cotter pins**
- Grooved T-head cotter pin

**Studs**
- Round-head grooved drive stud

Retaining Rings

**Internal retaining rings**
- Internal retaining ring

**External retaining rings**
- External retaining ring

Shaft Keys

**Parallel key**
- Standard parallel key ANSI B17.1

**Tapered key**
- Tapered key ANSI B17.1

**DIN**

**Bolts & Screws**

**Cross recess head screws**
- PH Pan head cross recess screw with collar DIN 967, PZ
- Raised countersunk head screw DIN EN ISO 7047, PH
- Raised countersunk head screw DIN EN ISO 7047, PZ
- Countersunk flat head screw DIN EN ISO 7046-1, PH
- Countersunk flat head screw DIN EN ISO 7046-1, PZ Pan head screw DIN EN ISO 7045, PH Pan head screw DIN EN ISO 7045, PZ Pan head cross recess screw with collar DIN 967

**Self-tapping screws**
- Cross recessed oval head raised countersunk tapping screw DIN 7051, Cross recessed flat head countersunk tapping screw DIN 7050, Cross recessed pan head tapping screw DIN EN ISO 7049, Cross recessed pan head tapping screw with collar DIN 967

**Miscellaneous bolts**
- Wing screw DIN 316
<table>
<thead>
<tr>
<th>Classification</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slotted head screws</td>
<td>Slotted pan head screw with shoulder DIN 923, Slotted raised countersunk head screw DIN EN ISO 2010, Slotted countersunk flat head screw DIN EN ISO 2009, Slotted pan head screw DIN EN ISO 1580, Slotted cheese head screw DIN EN ISO 1207</td>
</tr>
<tr>
<td>Square head bolts</td>
<td>Square head bolt collar and dog point DIN 480, Square head bolt with short dog point DIN 479, Square head bolt with collar DIN 478</td>
</tr>
<tr>
<td>Socket head screws</td>
<td>Hex socket head cap screw DIN 912, Hex socket head cap screw DIN 7984, Hex socket head countersunk screw grade C DIN 7991</td>
</tr>
<tr>
<td>Eye bolts</td>
<td>Eye bolt DIN 444, Lifting eye bolt DIN 580</td>
</tr>
<tr>
<td>Set screws (hex head)</td>
<td>Hex set screw fine pitch with full dog point DIN 561, Hex set screw with full dog point DIN 561, Hex set screw fine pitch with half dog point and flat cone point DIN 564, Hex set screw with half dog point and flat cone point DIN 564</td>
</tr>
<tr>
<td>Set screws (hex socket)</td>
<td>Hex socket set screw with cone point DIN 914, Hex socket set screw with dog point DIN 915, Hex socket set screw with flat point DIN 913, Hex socket set screw with cup point DIN 916</td>
</tr>
<tr>
<td>Set screws (slotted)</td>
<td>Slotted set screw with cone point DIN EN 27434, Slotted set screw with dog point DIN EN 27435, Slotted set screw with flat point DIN EN 24766, Slotted set screw with cup point DIN EN 27436</td>
</tr>
<tr>
<td>Hex head bolts &amp; screws</td>
<td>Hex bolt structural short grade C DIN 6914, Hex head structural bolt DIN 7990, Hex head bolt grade C DIN EN 24016, Hex head bolt grade C DIN EN 24018, Hex head bolt grade C DIN 931-2, Hex head bolt grade B DIN EN 24015, Hex head bolt grade A &amp; B fine pitch DIN EN 28765, Hex head screw grade A &amp; B fine pitch DIN EN 28676, Hex head bolt grade A &amp; B DIN EN 24014, Hex head screw grade A &amp; B DIN EN 24017</td>
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<tr>
<td>Category</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Pins</td>
<td>Parallel pins: Parallel pin DIN EN ISO 8734, Parallel pin ISO 8734</td>
</tr>
<tr>
<td></td>
<td>Nuts: Hex nuts: Hex nut style 2 grade A &amp; B DIN EN 24033, Hex nut style 2 grade A &amp; B fine pitch DIN EN 28674, Hex thin nut fine pitch grade A &amp; B DIN EN 28675, Hex thin nut grade A &amp; B DIN EN 24035, Hex nut style 1 grade C DIN EN 24034, Hex nut style 1 grade A &amp; B fine pitch DIN EN 28673, Hex nut style 1 grade A &amp; B DIN EN 24032</td>
</tr>
<tr>
<td></td>
<td>Square nuts: Square nut grade C DIN 557, Square weld nut DIN 928, Square thin nut grade B DIN 562</td>
</tr>
<tr>
<td>Washers</td>
<td>Lock washers: Spring lock washer DIN 127 Type A, Spring lock washer DIN 127 Type B, Conical spring lock washer DIN 6908, Conical spring lock washer DIN 6796</td>
</tr>
<tr>
<td>Retaining Rings</td>
<td>Internal retaining rings: Internal retaining ring DIN 472, Internal heavy retaining ring DIN 472</td>
</tr>
<tr>
<td></td>
<td>External retaining rings: External retaining ring DIN 471, External heavy retaining ring DIN 471</td>
</tr>
<tr>
<td>Shaft Keys</td>
<td>Parallel key: Parallel key 6885-1</td>
</tr>
<tr>
<td></td>
<td>Tapered key: Tapered key 6883</td>
</tr>
<tr>
<td></td>
<td>Stressed tapered key: Stressed tapered key 6886</td>
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### ISO

**Bolts & Screws**

<table>
<thead>
<tr>
<th>Description</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hex head bolts &amp; screws</td>
<td>Hex bolt structural short grade C ISO 7412, Hex head bolt grade A &amp; B fine pitch ISO 8765, Hex head screw grade A &amp; B fine pitch ISO 8676, Hex structural bolt grade C ISO 7411, Hex flange bolt small ISO 4162, Hex head screw grade C ISO 4018, Hex head screw grade A &amp; B ISO 4017, Hex head bolt grade C ISO 4016, Hex head bolt grade B ISO 4015, Hex head bolt grade A &amp; B ISO 4014</td>
</tr>
<tr>
<td>Socket head screws</td>
<td>Socket button head screw ISO 7380, Socket head shoulder screw ISO 7379, Hex socket head cap screw ISO 4762, Hexalobular socket pan head screw ISO 14583, Hex socket countersunk head screw ISO 10642, Hexalobular socket cheese head screw ISO 14580</td>
</tr>
<tr>
<td>Set screws (hex socket)</td>
<td>Cone point hex socket set screw ISO 4027, Cup point hex socket set screw ISO 4029, Dog point hex socket set screw ISO 4028, Flat point hex socket set screw ISO 4026</td>
</tr>
<tr>
<td>Set screws (slotted)</td>
<td>Cup point slotted set screw ISO 7436, Dog point slotted set screw ISO 7435, Cone point slotted set screw ISO 7434, Flat point slotted set screw ISO 4766</td>
</tr>
<tr>
<td>Cross recess head screws</td>
<td>PZ Raised countersunk head screw ISO 7047, PH Raised countersunk head screw ISO 7047, PZ Countersunk flat head screw ISO 7046-1, PH Countersunk flat head screw ISO 7046-1, PZ Pan head screw ISO 7045, PH Pan head screw ISO 7045</td>
</tr>
<tr>
<td>Slotted head screws</td>
<td>Slotted raised countersunk head screw ISO 2010, Slotted countersunk flat head screw ISO 2009, Slotted pan head screw ISO 1580, Slotted cheese head screw ISO 1207</td>
</tr>
<tr>
<td>Self-tapping screws</td>
<td>Hexalobular socket countersunk self-tapping screw ISO 14586, Hexalobular socket pan head self-tapping screw ISO 14585</td>
</tr>
</tbody>
</table>
### Nuts

<table>
<thead>
<tr>
<th>Type</th>
<th>Specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prevailing torque nuts fine pitch</td>
<td>Prevailing torque nut fine pitch ISO 10513, Prevailing torque nut fine pitch ISO 10512, Prevailing torque flange nut fine pitch ISO 12126, Prevailing torque flange nut fine pitch ISO 12125</td>
</tr>
<tr>
<td>Hex nuts fine pitch</td>
<td>Hex thin nut grade A &amp; B fine pitch ISO 8675, Hex nut grade A &amp; B fine pitch ISO 8674, Hex nut grade A &amp; B fine pitch ISO 8673</td>
</tr>
<tr>
<td>Prevailing torque hex nuts</td>
<td>Prevailing torque nut ISO 7720, Prevailing torque regular nut ISO 7719, Prevailing torque flange nut ISO 7044, Prevailing torque flange nut ISO 7043, Prevailing torque high nut ISO 7042, Prevailing torque nut ISO 7041, Prevailing torque nut ISO 7040, Prevailing torque nut thin ISO 10511</td>
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</tbody>
</table>

### Washers

<table>
<thead>
<tr>
<th>Type</th>
<th>Specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plain washers</td>
<td>Chamfered plain washer hardened ISO 7416, Plain washer hardened ISO 7415</td>
</tr>
<tr>
<td>Plain washers</td>
<td>Plain washer extra large grade C ISO 7094, Plain washer small grade A ISO 7092, Plain washer normal grade C ISO 7091, Chamfered plain washer normal ISO 7090, Plain washer small grade A ISO 10673, Plain washer normal grade A ISO 10673, Plain washer large grade A ISO 10673, Clevis pin plain washer ISO 8738, Plain washer large grade A ISO 7093-1, Plain washer normal grade A ISO 7089</td>
</tr>
</tbody>
</table>

### O-rings

<table>
<thead>
<tr>
<th>Type</th>
<th>Specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>O-rings</td>
<td>ISO 3601-1 O-rings for industrial applications, ISO 3601-1:2012 O-rings for aerospace applications</td>
</tr>
</tbody>
</table>
NAS
Sheet Metal Fasteners

Clinch nuts  Self-locking clinch nut with self-locking threads NASM 45938, Self-locking clinch nut without self-locking threads NASM 45938, Hex head self-locking clinch nut NASM 45938/2

Flush nuts  Self-clinching flush nut NASM 45938/4, Plain clinch flush nut NASM 3214

Standoffs  Self-clinching blind standoff NAS 1129, Self-clinching thru standoff NAS 1129

PEM®
Unified

Self-clinching nuts  CLA™ Self-clinching nuts, HNL™ Prevailing torque locknuts, H™ NutsS-RT™ Free-running locknuts, SH™ Hard panel nuts, SL™ TRI-DENT® Prevailing torque locknuts, SMPS™/SMPP™ Nuts (For thin sheets), S™/SS™/CLS™/CLSS™/SP™ Self-clinching nuts

Self-clinching studs  FH™/FHS™/FHA™ Flush-head studs, FH4™/FHP™ Flush-head studs for stainless steel sheets, FHL™/FHLS™ Flush, Low-displacement head studs, HFH™/HFHS™/HFHB™ Heavy-duty studs, HFE™/THFE™ Heavy-duty studs for thin sheets, HFG8™ Heavy-duty, high tensile strength studs, HFLH™ Hard panel studs, SGPC™ Swaging collar studs, TFH™/TFHS™ Non-flush studs

Self-clinching pins  FH™/FHS™/FHA™ Flush-head pins, TPS™/TP4™ Flush-head pilot pins
<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Self-clinching standoffs</td>
<td>BSO™/BSOS™/BSOA™/BSO4™ Blind threaded standoffs, DSOS™/DSO™ Threaded standoffs (for close-to-edge applications), SOSG™/SOAG™ Grounding standoffs, SO™/SOS™/SOA™/SO4™ Through-hole threaded standoffs, SO™/SOS™/SOA™/SO4™ Through-hole unthreaded standoffs, TSO™/TSOS™/TSOA™/TSO4™ Threaded standoffs for thin sheets</td>
</tr>
<tr>
<td>Metric</td>
<td></td>
</tr>
<tr>
<td>Self-clinching nuts</td>
<td>CLA™ Self-clinching nuts, HNL™ Prevailing torque locknuts, H™ Nuts, S-RT™ Free-running locknuts, SH™ Hard panel nuts, SL™ TRI-DENT® Prevailing torque locknuts, SMPS™/SMPP™ Nuts (For thin sheets), S™/SS™/CLS™/CLSS™/SP™ Self-clinching nuts</td>
</tr>
<tr>
<td>Self-clinching studs</td>
<td>FH™/FHS™/FHA™ Flush-head studs, FH4™/FHP™ Flush-head studs for stainless steel sheets, FHL™/FHLS™ Flush, Low-displacement head studs, TFH™/TFHS™ Non-flush studs, HFE™/THFE™ Heavy-duty studs for thin sheets, HFH™/HFHS™/HFHB™ Heavy-duty studs, HF109™ Heavy-duty, high tensile strength studs, HFLH™ Hard panel studs, SGPC™ Swaging collar studs, FHX™ Flush-head studs with X-Press™ thread profile</td>
</tr>
<tr>
<td>Self-clinching pins</td>
<td>FH™/FHS™/FHA™ Flush-head pins, TPS™/TP4™ Flush-head pilot pins, TPXS™ Self-clinching pilot pins</td>
</tr>
<tr>
<td>Self-clinching standoffs</td>
<td>BSO™/BSOS™/BSOA™/BSO4™ Blind threaded standoffs, DSOS™/DSO™ Threaded standoffs (For close-to-edge applications), SOSG™/SOAG™ Grounding standoffs, SO™/SOS™/SOA™/SO4™ Through-hole threaded standoffs, TSO™/TSOS™/TSOA™/TSO4™ Threaded standoffs for thin sheets, SO™/SOS™/SOA™/SO4™ Through-hole unthreaded standoffs</td>
</tr>
</tbody>
</table>
SAE O-rings

All O-rings AS568 O-rings

How to request standard content

If you’d like to request that Onshape add particular standard content to our list, use one of these methods:

- Submit a Support or Feedback request from within Onshape
- Log your request in our Forums, here: https://forum.onshape.com/

Assembly Configurations

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

In Onshape, you have the ability to create your own configurations inside an Assembly regardless of whether or not you have Part Studio configurations. They are altogether separate and one is not important to the other.

Assembly configurations work mechanically the same way as Part Studio configurations. The difference is that in an Assembly, you are only able to configure Mates (not to be confused with Mate connectors), Instances, and patterns.

When more than one person is working in the same document, each sees their own selected configuration, except when working in Follow Mode; at that point the follower sees the configuration selected by the leader.

Below is an example of an Assembly with the configuration panel icon on the right side of the window, shown below to the right of the blue arrow:

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Basic steps: Creating one Configuration input table

With a part instance or assembly in the workspace, open the Configuration panel:

1. Click ![image](image.png) to the right-side of the graphics area (as shown above)

2. The Configurations panel opens:
Add configurations to drive different design variants
Configurations enable you to create multiple variants of an Assembly, for example: instance configurations, patterns, mates, mate connectors and suppression states.

Learn more about configurations
3. Click (as shown above outlined in blue) to open a table:

By default, the caret to the left of 'Configuration' is expanded (shown above to the right of the blue arrow), click the caret when you are done with a section of the panel to collapse that section.

4. Click in the first row to activate it and enter the names of the input in the Name column. For example, to configure a pattern on a part, you might name the rows 2x2 and 4x4. Use Tab to move from one row to the next.
The active row is indicated by a blue bar to the left of the row.

5. To configure a parameter value for the indicated row, click

6. Select the Mate, instance or pattern that contains the parameter (click it in the Assembly list) and select the parameter in the dialog that opens (shown below outlined in blue). The parameter is then outlined with a broken yellow line and a new column is created for that parameter in the table (shown below, in the Configuration panel, to the left of the blue arrow).

The column name defaults to the name of the entity selected (as a top-level heading) plus the field name (as the subordinate-level heading), in this case *Box <1>* is the entity selected, and the field names have been changed to *2 x 2* and *Long* by the user.
7. To edit a configured instance of the parameter:

a. If the parameter is an entered value, click on the row in the table and enter a new value.

b. If the parameter is a selection in a dialog, double-click the row in the table to open the feature dialog.

   The appropriate field in the feature dialog is highlighted in blue. Make your selection on the model or in the Instances lists for this parameter.

8. When finished defining the configurations, click the Done button in the orange message at the top of the window, or close the dialog box.

9. Repeat step 6 through 8 for each row.

10. Repeat steps 5 through 8 to add another feature parameter to the table.

11. To test the inputs with the model, in the Feature list, under Configurations, use the down arrow to select from the menu:

![Configuration Panel]

   The model should update accordingly. If it doesn’t, check the model for design intent and the configurations definition for accurate selection.

**Configuring assembly properties**

Onshape has a mechanism for also configuring assembly properties for each of the configuration inputs and options you have previously defined, directly from the Configuration panel. The properties available to be configured include: description, part number, revision, vendor, project, product line, title 1, title 2, title 3, not revision managed, and exclude from BOM.

To configure an Assembly property:

1. With an existing configuration input in the Configuration panel, click

   ![Configured assembly properties]

   at the top of the panel (shown below outlined in blue):
2. Click [Add property].

3. Select the assembly property you wish to configure (custom properties are included in the list). (This example uses Project.)
A table is created with the previously selected configuration input in the first column and the property in the second column:

<table>
<thead>
<tr>
<th>Configurations</th>
<th>Configured assembly properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Properties for Assembly 1</td>
<td></td>
</tr>
<tr>
<td>Configuration</td>
<td>Project</td>
</tr>
<tr>
<td>2x2</td>
<td></td>
</tr>
</tbody>
</table>

4. In the Configuration column, use the down arrow to select from the list of configuration options.

5. In the Project column (project property), enter the name.

6. To add more part properties for another configuration option, click +.

7. Select a new configuration option from the first column.

8. In the Project column (project property), enter the name.

9. Repeat as necessary to configure the properties for the necessary configuration options.

When configuring part number properties, you can right-click and select Generate new part number, when automatic part number generation is turned on (through Release management preferences):

<table>
<thead>
<tr>
<th>Configurations</th>
<th>Configured assembly properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Properties for Assembly 1</td>
<td></td>
</tr>
<tr>
<td>Configuration</td>
<td>List input 1</td>
</tr>
<tr>
<td>base offset</td>
<td>base</td>
</tr>
<tr>
<td>base not offset</td>
<td>base</td>
</tr>
</tbody>
</table>

Changing configurations
After a part with configurations has been inserted into an Assembly, you can change the configuration of it, on any device:

1. Right-click on the part (or the part name in the Instances list) and select **Change configuration**.

A Change configuration dialog opens:

2. Select a new configuration option.

3. Click ✓ when you are satisfied with your selection. (Use ✗ to cancel the operation.)

For iOS and Android devices, tap the three dot menu and select 'Change configuration' to access the configuration dropdown and select a different configuration. Tap the Generate button to generate the new configuration of the part.
Copying and pasting into and out of tables

You can copy and paste into and out of a configuration table, to aid in entering or editing input values.

To copy a configuration table:
1. Open the menu in the upper right corner, next to +Configure features.

2. Select Copy table.

3. Once you have copied the table, you can paste it into a spreadsheet.

Note that the column names also come in with the table. Now you can edit the table and copy/paste it back into Onshape:

1. Select just the rows and columns with data (not the column names or headings).

You can also pad your table with extra empty rows, if you wish. Just include the extra rows in the spreadsheet when selecting for the copy command.

2. Issue a Copy command.

3. In the Onshape Configuration table, click the top, left cell of the table.

4. Issue a keyboard Paste command.

Onshape automatically replaces whatever data was in the rows and columns of the configuration input table with the data that was copied. Onshape also includes the default units for each input parameter, automatically.

Note that if there are more rows copied from the spreadsheet than are in the Onshape configuration input table, those rows are included in the paste. Onshape creates the rows on the fly.

However, if there are more columns copied from the spreadsheet than are in the Onshape configuration input table, those columns are not included in the paste. Onshape does not yet create columns on the fly. You can, however, create additional columns (configured features) in the configuration table before pasting.

---

**Bill of Materials**

This functionality is currently available only on Onshape's browser platform.

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Use the Onshape Bill of Materials (BOM) functionality to automatically create a BOM from any workspace Assembly. You have the ability to insert parts and assemblies into an Assembly post-release, from an Onshape version, or assemble the parts and sub-assemblies and then release the Assembly all at once. For more information on Release management, see "Release Management" on page 2169.

Onshape BOMs include a default set of properties as columns, and you are able to add or remove columns at will. Define "Custom Properties " on page 2585 and include those in the BOM as well, if you wish. You are also able to supply Display names for all Onshape-supplied properties through the Custom properties page of Company settings.

**Loading and viewing the table**

All Onshape Assemblies have a BOM table icon on the far right of the graphics area, below the View tools icon.

1. In an Assembly, click the BOM table icon on the right edge of the graphics area.

2. When the table opens, Onshape retrieves the data for the Assembly and populates the table (if there are parts or assemblies present, see the top example below). If no parts or assemblies are present, Onshape opens with default column names (properties) displayed (see the bottom example below):

![BOM Table Example](image-url)
Bill of material with parts listed, above

Bill of material with no parts, above

3. To populate an empty BOM, simply create your assembly in the Assembly tab, following the instructions in "Insert Parts and Assemblies" on page 1430.

4. In the BOM panel, select how to view the information (through the BOM type menu):

   - **Flattened** - This view provides a simple list of parts by item number, with no indication of subassemblies.
   
   - **Structured** - This view provides a list of parts including indication of expandable subassemblies. Subassemblies are indicated with small right-facing caret; for example:

     ![Subassembly Example](image)

     When viewing in Structured format, double-click the cell with the caret (the subassembly) to expand the list below the cell and see the parts included in the subassembly, labeled with the subassembly item number followed by a dot and then the part item number (9, 9.1, 9.2, and 9.3 in the example below):
The item numbers assigned to the subassemblies and parts reflect the order of the instances in the Instances list. If you reorder the instances in the list, the BOM table updates to reflect that new order.

Before, "002c-Oil Tank Nozzle" is seventh in the Instances list and item number 7 in the BOM table:

After the part is moved to first place in the Instances list, it is updated to item number 1 in the BOM table:

Locating items in the model and Instances list

To locate items from your Instances list in your model, click the item name in the Instances list and the selected item will become highlighted in the model and the
BOM table. You are also able to click the item name in the BOM table, and the selected item will become highlighted in the model and Instances list.

**Generating part numbers**

If you have automatic part numbering turned on in the Release management settings, in a BOM you can:

- Generate part numbers for a particular part - Right-click the Part number cell of the part and select **Generate next part number**.

- Generate part numbers for any parts in the table that are missing part numbers - Right-click the Part number column header and select **Generate missing part numbers**.

**Additional table data**

Item number, Quantity, Part number and Description are the default properties displayed for parts and subassemblies as columns in the BOM table. You are able to add more columns, according to the properties defined for your account.

1. Click the **Add column** drop down and select a property to insert from the list of properties. Additional columns are added at the far right side of the table.

2. For properties marked as editable (in your account Properties), you are able to click in the cell and add data. This data is saved for the specific part, in the specific property and is available throughout your document and company for that part.

For example, you are able to enter information in the Vendor cell for each part. That information is saved in that part's Properties, Vendor field:

The Vendor cell with a value in a BOM table:
The Properties dialog for the part:

Pay attention to where you allow a property to be edited, however. When creating a property (through the Properties tab in your account settings), you are able to mark it "Edit value in workspace", "Edit value in version" or both. If you mark it "Edit value in workspace" only, you will not be able to edit that value in a BOM table for any released parts since released parts become part of a version. You would have to branch a workspace from that version, edit the values, then either create a version, or re-release the parts.

3. To choose to exclude a specific part or subassembly and its corresponding data from the BOM table without deleting it from record: right-click in the row and select
Exclude from BOM. The row of data is removed completely from the table but the part still exists in the Assembly. The remaining items are renumbered accordingly.

4. To leave a record in the table of the information that's being excluded, you can click on the overflow menu at the top right of the table and select **Show excluded**. This displays the row in the table, however, the item number is replaced with a dash, indicating that the item is not included. This construct works the same for subassemblies; when a subassembly is excluded, all the of the parts are also excluded.

When items are excluded from the BOM, the remaining items are renumbered accordingly.
To include an item in the BOM again, select Show excluded, then right-click the row to re-include and select **Include in BOM**.

5. Another way to exclude/include items from a BOM is to use the Exclude from BOM checkbox in the parts Properties dialog.

Adding Items
You have the ability to insert non-geometric items (items not modeled in CAD, like glue, tape, paint, thread locker, etc) into your bill of material, through the BOM panel in an Assembly tab. Onshape refers to these types of entities simply as *items*.

There is some setup required in order to have the information available for insertion into the bill of material. Refer to "Managing Your Onshape Professional Subscription" on page 2492 if you have a Professional subscription or to "Managing Your Onshape Enterprise Subscription" on page 2551 if you have an Enterprise subscription. Once you have items added to your subscription (in your company settings Properties), you can follow these instructions to insert those items into a bill of material within an Assembly tab.
1. In the Bill of Material panel, select the *Insert items* icon at the top of the panel:

![Bill of Materials panel](image)

2. In the dialog:

![Insert items dialog](image)

3. Select the item to include in the BOM, and the quantity. You can search the items and also filter by category.
4. Click Insert to add the selected items to the BOM.

a. When you click Insert, the item is added to the Items list in the Instances list first. You can click Undo to remove items, if you change your mind about an item.

b. When you click the checkmark to accept your actions and close the dialog, the items appear in the BOM panel.

You can edit the items through the Instance list.

Right-click an item to:
• Edit - Change the quantity
• Suppress - Temporarily take the item out of the BOM (and assembly)
• Add comment - Add a comment to the assembly that directly references the item
• Delete - Delete the item from the assembly and the BOM

Export to CSV
Use the overflow menu's Export to CSV command to export the entire BOM table to a comma-separated file. If subassemblies are not expanded, the entire BOM will not be exported.

Formatting the table
Resize the table in the following ways:
• Click and drag the left-most edge of the panel to resize the panel, larger or smaller. There are limits to how large and small you can make the panel.
• Resize rows by clicking and dragging the column header border to the left or right. Note that the first column is stationary for reference; it cannot be moved or resized.
• You have the ability to move any column to the right or left, or remove it from the table through the context-menu on the column name:

<table>
<thead>
<tr>
<th>Part number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PT-08</td>
<td>Move left</td>
</tr>
<tr>
<td></td>
<td>Move right</td>
</tr>
<tr>
<td>PT-03</td>
<td>Remove column</td>
</tr>
</tbody>
</table>

• To add a removed column back to the table, use the Add column drop down at the top right of the panel.

Considerations around released or releasing parts
Keep in mind that when using imported or released parts in an Assembly, the properties will display in the BOM table as read-only. To create a BOM with editable fields, you can branch from the version to create a new workspace. The fields in the BOM will be editable per the settings for each property (through the account settings).
When you insert released (revisioned) parts into the Assembly and then create the BOM table, the state property will be "Released."

**Using BOMs in drawings**

You are able to insert Onshape BOM tables into your drawings. See the "Insert BOM" on page 1925 topic.

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**Mate Connector**

Mate connectors are local coordinate system entities located on or between entities and used within a Mate to locate and orient instances with respect to each other.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Mate Connector: Desktop**

Shortcut: Ctrl-m

In the Feature toolbar:

In the Assembly toolbar:

Mate connectors are local coordinate system entities located on or between entities and used within a Mate to locate and orient instances with respect to each other.
Two instances are positioned in an assembly by creating a Mate. The two instances are positioned by aligning a Mate connector defined on one instance with a Mate connector defined on the other instance.

Mate connectors can be defined explicitly or implicitly:

- **Explicit mate connectors:**
  - Define using the Mate connector tool in the toolbar in an Assembly or in a Part Studio
  - Listed at the highest level in the Feature list
  - Can be selected from the Feature list during the creation of a mate

- **Implicit mate connectors:**
  - Define while creating a Mate, in an Assembly and some features in a Part Studio
  - Listed at a sub-level in the Feature list in an Assembly, beneath the Mate in which it is used
  - Can not be selected from the Feature list during the creation of a Mate

When a Mate connector is used in more than one Mate, it is listed only once in the Feature list, with the first Mate that uses it (if implicit) or as its own original top-level feature in the list.

To learn more about Mates, see "Mates" on page 1544. To learn about Mates and Mate Connectors watch the video below.

Use the shortcut key k to hide/show Mate connectors.

**Steps**

1. Click 🕒.

2. Choose between creating a Mate connector on an entity or between entities:

3. Select a point on the entity for the Mate connector:
   - Roll over any face to activate the potential Mate connectors and select a point.
   - Or click anywhere on a face, sketch, or surface to automatically place the Mate connector at the centroid point.
You are also able to select the Origin as an entity; select the Origin in the graphics area, or in the Instances list.

4. Specify options, if desired (as shown in options examples below).

5. Click ✓.

**Visualize Mate connector points**

With the Mate connector dialog open, moving the cursor over an entity ‘wakes up’ default inference points and the inference point closest to the cursor highlights as a Mate connector. As you continue to mouse over the entity, different default inference points appear.

To lock Mate inferences when you see the one you want to select, press the Shift key when mousing.

Each face and edge of an entity has default inference points:

- At the centroid
- At the midpoints
- At the corners
- At the virtual sharps of conical faces:

*Inference points at the top, middle, bottom, and at the virtual sharp (where the sides of the conical face would meet if extended) of the conical face*
At the centroids of any region contained in a planar face (for example, holes and slots):

Before the default Mate connector is highlighted at the centroid (seen above), you might see the centroid point icon (seen below):

For cylindrical faces, inference points appear on the axis of the cylindrical and partial cylindrical face:
Select a planar face that has a partial cylindrical edge and the Mate connector inference points include the centroid of the axis:

Hover over the edge of the partial cylindrical face and the default Mate connector appears at the centroid of the axis:
To zero in on a specific inferenced point or default Mate connector without waking up others as you move the cursor, use the SHIFT key to prevent other Mate connectors from appearing.

**Realign Mate connectors**

Change the orientation of the Mate connector along a primary and (optionally) a secondary axis:

**Move Mate connectors**

- **Move** - Move the Mate connector a specified distance in a specified direction. The fields are presented in this order:
  - X translation
  - Y translation
  - Z translation
  - You also have the ability to use the Rotate field to specify a rotation of a specified number of degrees.
You are able to use *expressions and trigonometric functions* in numeric fields in Assemblies.

**Inference points and defaults**

The inference points for potential Mate connectors available when you select an edge or face are:

**Planar face** - Parallel to the face at every vertex, arc center, edge midpoint, and the face centroid

**Cylindrical face** - Perpendicular to the face axis at the middle and ends

**Linear edge or sketch line** - Perpendicular to the line at the middle and ends

**Circular edge or sketch circle** - Perpendicular to the line at the middle and ends

**Hide and show Mate connectors**

Once created, you are able to hide or show Mate connectors in both Part Studios and Assemblies:

- Use the context menu in the Feature list (Hide, Hide other Mate connectors/Show, Show all Mate connectors) - Hide other Mate connectors hides all Mate connectors but the one you have selected.

- Use the [ ] icon in the Feature list to hide a specific Mate connector.

- Hiding/showing Mate connectors in a Part Studio or Assembly is exclusive to the Part Studio or Assembly. Mate connectors hidden in a Part Studio are visible when
inserted into the Assembly. You are able to view Mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

**Tips**

- If the behavior is not what you expected, try flipping the primary and/or secondary axis on the Mate connector.
- Use the SHIFT key to keep the Mate connectors you want visible as you move the pointer to select one. This can be useful when the inferred point for potential Mate connector you want is on or near an edge.
- All Mate connectors are listed in the Feature list; you can hide/show them, edit and adjust, change, and use different orientations of the connectors.
- A Mate connector is able to be created in both the Assembly and the Part Studio. Creating a Mate connector in the Part Studio has two advantages:
  - A Mate connector defined in a Part Studio is available for reuse on every instance of that entity in every assembly in which it is instanced.
  - When creating a Mate connector in the Part Studio, there is an additional option in the Mate connector dialog called Owner Part.

In a Part Studio with more than one part, it may be unclear which part owns the Mate connector. Use Owner Part to specify which part owns the Mate connector.
Mate Connector: iOS

Steps

1. Tap Mate connector tool.

2. Select origin type:
   - **On entity** - Create a Mate connector on a part.
   - **Between entities** - Create a Mate connector halfway between two entities on the part.

3. Select origin entity.

4. Optionally, toggle to realign.
   If you do realign, select primary and secondary axis entities.

5. Optionally, toggle to move.
   If you do move, specify X, Y, and Z values.
Also select rotation axis and specify rotation angle.

6. Tap the checkmark.

**Visualize Mate connector points**

Each **face and edge** of a part has default inference points:

- At the centroid
- At the midpoints
- At the corners
For **cylindrical faces**, inference points appear on the axis of the cylindrical and partial cylindrical face:
Select or hover over a planar face that has a **partial cylindrical edge** and the Mate connector inference points include the centroid of the axis:

Select or hover over the edge of the partial cylindrical face and default Mate connector inference points appear:
On entity

Select the origin entity then select the Mate connector inference point on which to place the Mate connector.
Between entities

Select an origin entity and the Mate connector inference point in line with where you want to place the Mate connector. Select another entity and the Mate connector is placed in line with the Mate connector inference point, between the two entities you selected.
Realign Mate connectors

Toggle on Realign to change the orientation of the Mate connector along a primary and (optionally) a secondary axis.

Select a primary and secondary axis along which to realign the Mate connector.
Move Mate connectors

Move the Mate connector to a specified distance in a specified direction. The fields are presented in this order:

- X translation
- Y translation
- Z translation

You are also able to use a rotation axis and specify a number of degrees to rotate the Mate connector.
Hide and show Mate connectors

Once created, you are able to hide or show Mate connectors in both Part Studios and Assemblies:

- Use the context menu in the Feature list (Hide, Hide other Mate connectors/Show, Show all Mate connectors) - Hide other Mate connectors hides all Mate connectors but the one you have selected.

- Use the Hide/Show icon (the eye icon) in the Feature list to hide a specific Mate connector.

- Hiding/showing Mate connectors in a Part Studio or Assembly is exclusive to the Part Studio or Assembly. Mate connectors hidden in a Part Studio are visible when inserted into the Assembly. You are able to view Mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

**Mate Connector: Android**

**Steps**

1. Tap Mate connector tool.
2. Select origin type:
   - **On entity** - Create a Mate connector on a part.
   - **Between entities** - Create a Mate connector halfway between two entities on the part.

3. Select origin entity.

4. Optionally, toggle to realign.
   - If you do realign, select primary and secondary axis entities.

5. Optionally, toggle to move.
   - If you do move, specify X, Y, and Z values.
Also select rotation axis and specify rotation angle.

6. Tap the checkmark.

**Visualizing Mate connector points**

Each *face and edge* of a part has default inference points:

- At the centroid
- At the midpoints
- At the corners
For **cylindrical faces**, inference points appear on the axis of the cylindrical and partial cylindrical face:
Select or hover over a planar face that has a **partial cylindrical edge** and the Mate connector inference points include the centroid of the axis:

Select or hover over the edge of the partial cylindrical face and default Mate connector inference points appear:
Select the origin entity then select the Mate connector inference point on which to place the Mate connector.
Between entities

Select an origin entity and the Mate connector inference point in line with where you want to place the Mate connector. Select another entity and the Mate connector is placed in line with the Mate connector inference point, between the two entities you selected.
Realign Mate connectors

Toggle on Realign to change the orientation of the Mate connector along a primary and (optionally) a secondary axis.

Select a primary and secondary axis along which to realign the Mate connector.
Move Mate connectors

Move the Mate connector to a specified distance in a specified direction. The fields are presented in this order:

- X translation
- Y translation
- Z translation

You are also able to use a rotation axis and specify a number of degrees to rotate the Mate connector.
Hide and show Mate connectors

Once created, you are able to hide or show Mate connectors in both Part Studios and Assemblies:

- use the context menu in the Feature list (Hide, Hide other Mate connectors/Show, Show all Mate connectors) - Hide other Mate connectors hides all Mate connectors but the one you have selected.
- Use the Hide/Show icon (the eye icon) in the Feature list to hide a specific Mate connector.
- Hiding/showing Mate connectors in a Part Studio or Assembly is exclusive to the Part Studio or Assembly. Mate connectors hidden in a Part Studio are visible when inserted into the Assembly. You are able to view Mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

Mates

This functionality is available on Onshape's browser, iOS, and Android platforms.

Mates in Onshape are different than mates in old CAD systems. There is only one Onshape Mate between any two instances, and the movement (degrees of freedom) between those two instances is embedded in the Mate. Mates contain their own coordinate systems, so you ever only need to use one Mate to define the degrees of freedom between two entities. At the time of placing a mate between two entities, Onshape will offer points on each entity to which to align with the mate's coordinate system. The suggested locations are based on the underlying geometry of the part and changing the geometry will change the location of the mate. This can be undesirable in certain situations, but you can also:

- Add an explicit mate connector to an entity exactly where you want it if the geometry does not already allow an implicit mate connector while creating a mate. For more information, see "Mate Connector" on page 1519.
• Insert a layout or reference sketch in an Assembly to use for aligning mate connectors.

**Mates: Desktop**

Shortcut: m

Use the shortcut key j to hide/show mates in an assembly.

Note that you are able to mate entities to the Origin in an Assembly. You are also able to Fix an entity in order to test the movement of assigned Mates using the context menu or drag.

Entities include: parts, assemblies, subassemblies, sketches, and surfaces.

**Mate dialog**

Mates are defined through the Mate dialog:

You select the type of mate to create, then select the **mate connectors** (one for each part). You are also able to check the box to apply limits of movement. Other options/action include:

- Flip the primary axis, Z orientation, of the instances.
- Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a click.
- Preview the animation of unlimited movement of the mate, ignoring all other...
mates in the assembly.

- **Solve** - Solve all assembly mates including this one.

Many mates offer the ability to set an Offset distance for defining a fixed space between the parts being mated, as well as distance [Limits](#) for movement. Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.

Offset distances are visualized in the graphics area as dashed lines between the Mates, displaying the value and the axis. Enter the distance in the dialog.

When you click on a Mate, graphics are displayed to indicate the direction of the X, Y, and Z as defined by the Mate along with the offset and the range of limits dimensions (if any).

Applying an offset should be viewed as moving the entire coordinate system. The offset is relative to the Mate connector selected first.
When the Offset box is checked, many mates also offer an option to specify rotation about a specific axis: Slider, Revolute, Pin Slot, Planar, and Fastened mates include the option, as see below:
Select the axis about which to rotate, above, then enter the degrees of rotation.

When you open a Mate dialog and select two Mate connectors, a head-up display appears at your cursor:

Click the checkmark in the head-up display to commit the current mate and start a new Mate. The Mate dialog box stays open, and you are free to continue selecting Mates.

Offset distances are visualized in the graphics area as dashed lines between the Mates, with editable values. Drag the part to a desired position, double-click the distance value and enter a new value. These values are not persisted; you are able to use them to estimate values to enter in the Offset field in the dialog, or place parts in precise positions in order to take measurements. For example:
Mate values for axes and rotational movement (above).

Mate value in edit (above).

**Mate context menu**

Use the Mate context menu to access the following commands:

- **Rename** - Specify a different name for the Mate
- **Edit...** - Change the Mate definition
Reset - After an assembly is dragged to test movement of Mates, use Reset to return the assembly to its starting/home position (assuming constraints don't restrict that)

Animate - Drive the assembly from a single Mate (or single DOF within a mate)

Hide - Remove from view (Show displays the Mate again)

Show all mates - Show all Mate connectors

Isolate - Dim and inactivate all other parts except those selected (or associated with a selected Mate). When in Isolate mode, Exit isolate appears at the top of this menu. For more information, see "Managing Assemblies" on page 1472.

Make transparent - Dim parts nearest to the mate selected. Use the slider on the Make transparent dialog to extend the transparency out to other parts either by distance from, or by connectivity to, the selected mate.

Suppress - Visualize the assembly without the Mate (and without deleting the Mate)

Clear selection - Clear all selections

Delete - Remove the Mate from the assembly

Reordering mate features

Once a Mate is created and listed in the Mate Features list, select a Mate (or Ctrl-click to select more than one) and drag/drop to reorder them in the list. This will help put the most important mate features higher and more visible in the list. Onshape solves mates simultaneously so order won't affect a Mate.

Setting mate values for movement

You have the ability to specify Mate values of all mates except Ball, Fastened, and Tangent. Onshape provides visual cues for distances, and provides distance values, in default units, from the second Mate connector selected to the first. Specify limits in positive and negative values.
In this example, the Mate connector on the box was the first one selected in the dialog; the Mate connector on the cylinder was the second selected. Notice that the Y value is negative and the X value is positive.

Now, switch the order of Mate connector selection and notice the distance values:

Notice that in this scenario, the Y value is positive and the X value is positive. This is due to the order of measurement from one Mate connector to the other. It's important to remember that the measurement is made from the second selected Mate connector to the first, along the coordinate system.

Use these distance visualizations to estimate the value to enter in the Limits box:

1. When the Limits check box is present for a Mate, click to enable degrees of freedom fields to enter values for the minimum and maximum distances, as measured from the second Mate connector selected, to the first selected.
2. Using the distance visualization as a guide (drag the part to activate), enter a minimum and maximum value.

3. Use the Play button to animate the movement, including limits.

You are able to use expressions and trigonometric functions in numeric fields in Assemblies.

Animating movement within an assembly

Use the Animate command (found in the context menu for mates and mate indicators) to drive the assembly from a single Mate (or single DOF within a Mate). Other Mates and relations in the assembly are also enforced and honored.

If you have defined limits for the Mate, those values are used as the start and stop points during the animation.

1. Right-click on a mate or mate indicator and select Animate.

2. Animate works with only one degree of freedom at a time, so if the Mate has more than one, you are prompted to select one.

3. Enter Start and End values. If Limits are specified in the Mate definition, those values are automatically populated in the Start value and End value fields. If no Limits are specified in the Mate dialog, enter values now.

a. **Start value** - The minimum distance measured along the degree of freedom’s axis. (By default, the value as specified in the Mate Minimum Limit.)

b. **End value** - The maximum distance measured along the degree of freedom’s axis. (By default, the value as specified in the Mate Maximum Limit.)
Note that you are able to enter up to 36000 degrees here (100 revolutions), specifically helpful for visualizing degrees of freedom in high-ratio gears and rack and pinion relations.

4. Specify Steps, a linear map from the start to end value, inclusive, interpolated at each step. The minimum number of steps is 2. By default, playback is around 60 steps/second.

5. Select a playback type, Single or Reciprocate to play the animation of the degrees of freedom once or continuously until you manually stop it, respectively.

Current value is a read-only field and is populated during animation as the Mate moves through the degrees of freedom, in your specified units. When the motion stops (either automatically or manually), Current value displays the point at which the motion was stopped.

Animate supports all Mate types but it’s not recommended to use Fastened, Tangent, or Ball as the driving mate.

Tips

- The Animate command works with various graphics modes, like Isolate, Mate indicators and Mate connectors.

- Animate helps you explore the relationships between Mates, their constraint systems, and gives you a way to show off your design (especially with the playback loop feature, specifically for rotational degrees of freedom).

**Offset entities during assembly**

Offsetting entities from one another during assembly is available for the following Mate types:

- **Planar offset** - Along the Z axis
- **Slider offset** - Along the X and Y axes
- **Revolute offset** - Along the Z axis
- **Pin slot offset** - Along the Z axis
- **Fastened offset** - Along the X, Y, and Z axes
You are also able to drag the entities and observe the distance values in the graphics area. These will help determine the specific values to enter in the dialog.

You are able to use expressions and trigonometric functions in numeric fields in Assemblies.

Copying/Pasting assembled entities

You are able to copy and paste entities that have been mated in an Assembly:

1. Select the entities.
2. From the context menu, select Copy items:
3. From the context-menu, paste the items:

The entities are pasted directly where the mouse click occurs.

Notice that the entities, mate connectors, and mates are also duplicated in the Assembly list.

Mate indicators

In addition to being visible in the Assembly list, mates have indicators in the graphics area as well. You have the ability to hide the entities and mate connectors in the Assembly list in order to see these mate indicators more clearly. These indicators give hints at the type of motion they define as well as the current state: blue/white indicates good Mates, gray indicates suppressed or inactive, and red indicates a problem:

- **Fastened**
- **Revolute**
- **Slider**
- **Planar**
- **Cylindrical**
More tips for visualizing Mates:

- Select a part, right-click for the context menu and select Show mates.
- Hover over a Mate, right-click for the context menu where you are able to take action on the Mate.
- Select a Mate, Mate connector, or Mate relation in the graphics area and its associated instances and Mate feature are highlighted in the list.

**Concepts**

- There is exactly one Mate between any two instances.
- Fixing an entity is different from applying a Mate. Fix (found in the context menu) is specific to the assembly in which it is applied; it does not carry over to any other assembly that entity is inserted into.
- The Mate positions two instances in relationship to each other, aligning a Mate connector on each instance.
The initial position is often a best guess. There are two tools to correct the position:

- The Flip primary axis tool flips the major (Z) orientation.

- The Reorient secondary axis tool adjusts the orientation in 90 degree increments.

- The play button ▶️ animates the allowed movement between the Mate being created.

- The Solve button regenerates the mate in process and the movement of all Mates, so you have the ability to see how your changes affect the entire assembly.

The Mate type then specifies the degrees of freedom behavior.

**Example**

1. Select a Mate (for example 👩‍🔬) to open the dialog:
2. Select one automatic Mate connector on each entity (you can also Mate to the Origin):

3. If necessary, adjust the orientation using Flip Primary Axis or Rotate Secondary Axes.

4. Accept the Mate.

In the example above, only automatic Mate connectors were used. In most mating cases, automatic Mate connectors will work fine. In some less common cases, it may be useful to create Mate connectors ahead of time. You have the ability to create Mate connectors in either the Assembly or in the Part Studio.

**Mates: iOS**

<table>
<thead>
<tr>
<th>Mate connector</th>
<th>Fastened mate</th>
<th>Revolute mate</th>
<th>Slider mate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Planar mate</td>
<td>Cylindrical mate</td>
<td>Pin slot mate</td>
<td>Ball mate</td>
</tr>
<tr>
<td>Parallel mate</td>
<td>Tangent mate</td>
<td>Group</td>
<td>Snap mode</td>
</tr>
</tbody>
</table>

Mates in Onshape are different than mates in traditional CAD systems. There is only one Onshape Mate between any two instances, and the movement (degrees of freedom) between those two instances is embedded in the Mate.
Note that you are able to mate an entity to the Origin in an Assembly. You are also able to Fix an entity, in order to test the movement of assigned Mates, using the context menu. Entities include: parts, assemblies, subassemblies, sketches, and surfaces.

**Mate dialog**

Mates are defined through the Mate dialog:

![Mate dialog](image)

Select the type of Mate to create, then select the Mate connectors (one for each part).

**Mate type** - The *Mate type* field displays the type of Mate you are using. Tap to open a list of Mate types and tap to select one.

**Mate connectors** - The next section, *Mate connectors*, is highlighted in blue. This indicates that a selection from the graphics area is required. Two Mate connectors (one from each instance being mated) must be selected.

**Offset** - Tap to set an offset distance for defining a fixed space between the parts being mated.

**Limits** - Tap to set distance limits for movement.

- Flip the primary axis, Z orientation of the instances.

- Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.
**Solve** - Tap to solve all assembly Mates including the current one.

**Limiting movement**

You have the ability to specify movement limits of all Mates except Ball, Fastened, and Tangent.

Inside the dialog of a Mate that allows limits (Revolute, Slider, Planar, Cylindrical, and Pin slot) toggle the Limits button on and limit fields appear.

Enter desired limit specifications and tap Solve to visualize the changes.

**Offset entities during assembly**

Offsetting entities from one another during assembly is available for the following Mate types:

- Planar offset - Along the Z axis
- Slider offset - Along the X and Y axes
- Revolute offset - Along the Z axis
- Pin slot offset - Along the Z axis
- Fastened offset - Along the X,Y, and Z axes

You are also able to drag the entities and observe the distance values in the graphics area. These will help determine the specific values to enter in the dialog.

You are able to use expressions and trigonometric functions in numeric fields in Assemblies.

**Mate indicators**

In addition to being visible in the Assembly list, Mates have indicators in the graphics area as well. You are able to hide the entities and Mate connectors in the Assembly list in order to see these Mate indicators more clearly. These indicators give hints at the type of motion they define as well as the current state: blue/white indicates good Mates, gray indicates suppressed or inactive, and red indicates a problem.

![Fastened](image-url)
More tips for visualizing Mates:

- Two-finger tap for the context menu, tap **Show all** to show everything listed in the Instance list, including Mate connectors and Mate indicators.
- Tap on a Mate in the Instance list.

**Concepts**

- There is exactly one Mate between any two instances.
- Fixing an entity is different from applying a Mate. Fix (found in the context menu) is specific to the assembly in which it is applied; it does not carry over to any other assembly that entity is inserted into.
The Mate positions two part instances in relationship to each other, aligning a Mate connector on each instance.

**Before Mate**

![Before Mate Image]

**After Mate**

![After Mate Image]

The initial position is often a best guess. There are two tools to correct the position:

フォーク The **Flip primary axis** tool flips the major (Z) orientation.
The Reorient secondary axis tool adjusts the orientation in 90 degree increments.
The **Solve** button regenerates the mate in process and the movement of all Mates, so you have the ability to see how your changes affect the entire assembly.

The **Mate type** specifies the movement behavior.

**Mates: Android**
Mates in Onshape are different than mates in traditional CAD systems. There is only one Onshape Mate between any two instances, and the movement (degrees of freedom) between those two instances is embedded in the Mate.

Note that you are able to mate an entity to the Origin in an Assembly. You are also able to Fix an entity, in order to test the movement of assigned Mates, using the context menu. Entities include: parts, assemblies, subassemblies, sketches, and surfaces.

**Mate dialog**

Mates are defined through the Mate dialog:

Select the type of Mate to create, then select the Mate connectors, inferred or existing (one for each part).
**Mate type** - The *Mate type* field displays the type of Mate you are using. Tap to open a list of Mate types and tap to select one.

**Mate connectors** - The next section, *Mate connectors*, is highlighted in blue. This indicates that a selection from the graphics area is required. Two Mate connectors (one from each instance being mated) must be selected.

**Offset** - Tap to set an offset distance for defining a fixed space between the parts being mated.

**Limits** - Tap to set distance limits for movement.

- Flip the primary axis, Z orientation of the instances.

- Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

**Solve** - Tap to solve all assembly Mates including the current one.

**Limiting movement**

You are able to specify movement limits of all Mates except Ball, Fastened, and Tangent.

Inside the dialog of a Mate that allows limits (Revolute, Slider, Planar, Cylindrical, and Pin slot) toggle the Limits button on and limit fields appear.

Enter desired limit specifications and tap Solve to visualize the changes.

**Offset entities during assembly**

Offsetting entities from one another during assembly is available for the following Mate types:

- Planar offset - Along the Z axis
- Slider offset - Along the X and Y axes
- Revolute offset - Along the Z axis
- Pin slot offset - Along the Z axis
- Fastened offset - Along the X, Y, and Z axes

You are also able to drag the entities and observe the distance values in the graphics area. These will help determine the specific values to enter in the dialog.
You are able to use expressions and trigonometric functions in numeric fields in Assemblies.

**Mate indicators**

In addition to being visible in the Assembly list, Mates have indicators in the graphics area as well. You have the ability to hide the entities and Mate connectors in the Assembly list in order to see these Mate indicators more clearly. These indicators give hints at the type of motion they define as well as the current state: blue/white indicates good Mates, gray indicates suppressed or inactive, and red indicates a problem.

- **Fastened**
- **Revolute**
- **Slider**
- **Planar**
- **Cylindrical**
- **Pin slot, with an arrow in the direction of the slot**
- **Ball**
- **Tangent**
- **Parallel**
More tips for visualizing Mates:

- Two-finger tap for the context menu, tap **Show all** to show everything listed in the Instance list, including Mate connectors and Mate indicators.
- Tap on a Mate in the Instance list.

**Concepts**

- There is exactly one Mate between any two instances.
- Fixing an entity is different from applying a Mate. Fix (found in the context menu) is specific to the assembly in which it is applied; it does not carry over to any other assembly that entity is inserted into.
- The Mate positions two part instances in relationship to each other, aligning a Mate connector on each instance.

**Before Mate**

![Before Mate Image]

**After Mate**

![After Mate Image]
The initial position is often a best guess. There are two tools to correct the position:

- The **Flip primary axis** tool flips the major (Z) orientation.

- The **Reorient secondary axis** tool adjusts the orientation in 90 degree increments.
The Solve button regenerates the Mate in process and the movement of all Mates, so you have the ability to see how your changes affect the entire assembly.

The Mate type specifies the movement behavior.

Fastened Mate

Mate two entities and remove all degrees of freedom between them.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Fastened Mate: Desktop

Mate two entities and remove all degrees of freedom between them.

Begin by creating Mate connectors on each entity, or use the implicit Mate connectors visible upon hover.
Steps

1. Click 

   ![Image of Fastened 4](image)

2. Select two Mate connectors (inferred or existing) to use.
   
   If you want to supply an offset distance, check Offset and supply a distance.
   
   Fastened Mates are able to offset the entities along any combination of the three axes.

3. Click the entity to access the manipulator.

4. Check Offset to supply a value for distance between the two parts:

   Offset distances are visualized in the graphics area as dashed lines between the Mates, displaying the value and the axis. Enter the distance in the dialog.

   When you click on a Mate, graphics are displayed to indicate the direction of the X, Y, and Z as defined by the Mate along with the offset and the range of limits dimensions (if any).

   Applying an offset should be viewed as moving the entire coordinate system. The offset is relative to the Mate connector selected first.

   ![Diagram showing X, Y, and Z axes](image)

   You can also select to rotate the part about a specific axis: select the axis, then enter the degrees of rotation.
5. Click and drag the various manipulator handles to see which motions are allowed; notice that no motion is allowed. The entity has zero degrees of freedom.

**Fastened Mate: iOS**

Steps

1. Tap 

2. Specify that **Fastened** is selected in the *Mate type* field.

3. Select two Mate connectors to use.

4. Optionally, tap **Offset** to provide an offset distance.

5. Optionally, tap 

6. Optionally, tap to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that no motion is allowed. The part has zero degrees of freedom because of the Fastened mate.
Fastened Mate: Android
Steps

1. Tap 

2. Specify that **Fastened** is selected in the **Mate type** field.

3. Select two Mate connectors to use.

4. Optionally, tap **Offset** to provide an offset distance.

5. Optionally, tap 🔄 to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap 🌀 to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap checkmark.

*Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that no motion is allowed. The part has zero degrees of freedom because of the Fastened mate.*
Revolute Mate

Mate two entities allowing rotational movement about the Z axis. (Rz)

This functionality is available on Onshape's browser, iOS, and Android platforms.

Revolute Mate: Desktop

Mate two entities allowing rotational movement about the Z axis. (Rz)
Begin by creating Mate connectors on each entity, or use the implicit Mate connectors visible upon hover.

Steps

1. Click ☕.

   The first Mate connector selected serves as the rotational point and the second Mate connector selected serves as the stationary point.

2. To limit the movement, check Limits and supply (optional) minimum and maximum values to control the range of motion of the Mate.
Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.

3. To supply an offset distance, check Offset and supply a distance. Revolute Mates are able to offset the entities along the Z axis only. To create an offset between the two entities, click Offset and specify a distance.

Offset distances are visualized in the graphics area as dashed lines between the Mates, displaying the value and the axis. Enter the distance in the dialog.

When you click on a Mate, graphics are displayed to indicate the direction of the X, Y, and Z as defined by the Mate along with the offset and the range of limits dimensions (if any).

Applying an offset should be viewed as moving the entire coordinate system. The offset is relative to the Mate connector selected first.

You can also select to rotate the part about a specific axis: select the axis, then enter the degrees of rotation.

4. Click an entity to access the manipulator.

5. Click and drag the various manipulator handles to see which motions are allowed; notice that only rotational movement about the Z axis is allowed (Rz).

**Revolute Mate: iOS**

Steps

1. Tap 📲.

2. Confirm that **Revolute** is selected in the **Mate type** field.
3. Select two Mate connectors to use.

4. Optionally, tap **Offset** to provide an offset distance.

5. Optionally, tap **Limits** to set distance limits for movement.

6. Optionally, tap ‣ to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap ‡ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only rotational movement about the Z axis is allowed(Rz).
Revolute Mate: Android

Steps

1. Tap 🔄.

2. Confirm that Revolute is selected in the Mate type field.

3. Select two Mate connectors to use.

4. Optionally, tap Offset to provide an offset distance.

5. Optionally, tap Limits to set distance limits for movement.

6. Optionally, tap ⤊ to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap ⤆ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only rotational movement about the Z axis is allowed(Rz).
Slider Mate

Mate two entities allowing translational movement along the Z axis. (Tz)
This functionality is available on Onshape's browser, iOS, and Android platforms.

### Slider Mate: Desktop

Mate two entities allowing translational movement along the Z axis. (Tz)

The first Mate connector selected serves as the sliding point and the second Mate connector selected serves as the stationary point.

Begin by creating Mate connectors on each entity, or use the implicit Mate connectors visible upon hover.

**Steps**

1. Click ![icon](image.png).

2. Select two Mate connectors.
3. To limit the movement, check Limits and supply (optional) minimum and maximum values to control the range of motion of the mate. Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.

4. To supply an offset distance, check Offset and supply a distance. Slider Mates are able to offset the entities along any combination of the X and Y axis.

Offset distances are visualized in the graphics area as dashed lines between the Mates, displaying the value and the axis. Enter the distance in the dialog.

When you click on a Mate, graphics are displayed to indicate the direction of the X, Y, and Z as defined by the Mate along with the offset and the range of limits dimensions (if any).

Applying an offset should be viewed as moving the entire coordinate system. The offset is relative to the Mate connector selected first.

You can also select to rotate the part about a specific axis: select the axis, then enter the degrees of rotation.

5. Click the entity to access the manipulator.

6. Click and drag the various manipulator handles to see which motions are allowed; notice that only translational movement along the Z axis is allowed (Tz).

**Slider Mate: iOS**
Steps

1. Tap 📦.

2. Confirm that Slider is selected in the Mate type field.

3. Select two Mate connectors to use.

4. Optionally, tap Offset to provide an offset distance.

5. Optionally, tap 🚂 to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap ⏳ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap the checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only transitional movement along the Z axis is allowed (Tz).
Slider Mate: Android

Steps

1. Tap 📦.

2. Confirm that **Slider** is selected in the *Mate type* field.

3. Select two Mate connectors to use.

4. Optionally, tap **Offset** to provide an offset distance.

5. Optionally, tap 🔄 to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap 🔄 to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap the checkmark.

**Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only transitional movement along the Z axis is allowed (Tz).**
Planar Mate

Mate two entities allowing translational movement along the X axis and the Y axis, and rotational movement about the Z axis (Ty, Tx, Rz).

This functionality is available on Onshape's browser, iOS, and Android platforms.

Planar Mate: Desktop

Mate two entities allowing translational movement along the X axis and the Y axis, and rotational movement about the Z axis (Ty, Tx, Rz).

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Steps

Begin by creating Mate connectors on each entity, or use the implicit Mate connectors visible upon hover.

1. Click +.

![Planar 1 dialog](image)

The first Mate connector selected (inferred or existing) serves as the translational and rotational movement point and the second Mate connector selected (inferred or existing) serves as the stationary point.

To supply an offset distance, check Offset and supply a distance. Planar Mates are able to offset the entities only along the Z axis.

Offset distances are visualized in the graphics area as dashed lines between the Mates, displaying the value and the axis. Enter the distance in the dialog.
When you click on a Mate, graphics are displayed to indicate the direction of the X, Y, and Z as defined by the Mate along with the offset and the range of limits dimensions (if any).

Applying an offset should be viewed as moving the entire coordinate system. The offset is relative to the Mate connector selected first.

You can also select to rotate the part about a specific axis: select the axis, then enter the degrees of rotation.

2. To limit the movement, check Limits and supply (optional) minimum and maximum values to control the range of motion of the mate. Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.

3. Click the entity to access the manipulator.

4. Click and drag the various manipulator handles to see which motions are allowed; notice that only translational movement along the X and Y axis, and rotational movement about the Z axis is allowed (Ty, Tx, Rz).

**Planar Mate: iOS**
Steps

1. Tap +.

2. Confirm that **Planar** is selected in the *Mate type* field.

3. Select two Mate connectors (inferred or existing) to use.

4. Optionally, tap **Offset** to provide an offset distance.

5. Optionally, tap **Limits** to set distance limits for movement.

6. Optionally, tap to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only transitional movement along the X and Y axiz, and rotational movement about the Z axis is allowed (Ty, Tx, Rz).
Planar Mate: Android

Steps

1. Tap 🔄.

2. Confirm that **Planar** is selected in the *Mate type* field.

3. Select two Mate connectors (inferred or existing) to use.

4. Optionally, tap **Offset** to provide an offset distance.
5. Optionally, tap **Limits** to set distance limits for movement.

6. Optionally, tap 🔄 to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap 🔄 to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

Touch and drag OR use the [Triad manipulator](#) to move one of the parts. Notice that only transitional movement along the X and Y axiz, and rotational movement about the Z axis is allowed (Ty, Tx, Rz).
**Cylindrical mate**

Mate two entities allowing translational movement along the Z axis and rotational movement about the Z axis (Tz, Rz).
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Cylindrical Mate: Desktop**

Mate two entities allowing translational movement along the Z axis and rotational movement about the Z axis (Tz, Rz).

The first Mate connector selected (inferred or existing) serves as the translational and rotational point and the second Mate connector selected (inferred or existing) serves as the stationary point.

Begin by creating mate connectors on each entity, or use the implicit Mate connectors visible upon hover.

**Steps**

1. Click 🍴.
2. Select two Mate connectors (inferred or existing).
3. To impose limits on the movement of the mate, click Limits and supply minimum distances for the axis of both Mate connectors. Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.

4. Click the entity to access the manipulator.

5. Click and drag the various manipulator handles to see which motions are allowed; notice that only translational movement along the Z axis and rotational movement about the Z axis is allowed (Tz, Rz).

**Cylindrical Mate: iOS**

Steps

1. Tap 🕒.

2. Confirm that **Cylindrical** is selected in the **Mate type** field.
3. Select two Mate connectors (inferred or existing) to use.

4. Optionally, tap **Limits** to set distance limits for movement.

5. Optionally, tap 🚂 to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap ⚪️ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only translational movement along the Z axis and rotational movement about the Z axis is allowed (Tz, Rz).
Cylindrical Mate: Android
Steps

1. Tap 🔄.

2. Confirm that Cylindrical is selected in the Mate type field.

3. Select two Mate connectors (inferred or existing) to use.

4. Optionally, tap Limits to set distance limits for movement.

5. Optionally, tap 🔷 to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap 🔷 to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only translational movement along the Z axis and rotational movement about the Z axis is allowed (Tz, Rz).
Pin Slot Mate

Mate two entities allowing rotational movement about the Z axis and translational movement along the X axis. (Rz, Tx)

This functionality is available on Onshape's browser, iOS, and Android platforms.

Pin Slot Mate: Desktop

Mate two entities allowing rotational movement about the Z axis and translational movement along the X axis. (Rz, Tx)
The first Mate connector selected (inferred or existing) serves as the pin and the rotational movement point. The second Mate connector selected (inferred or existing) serves as the translational movement.

Begin by creating Mate connectors on each entity, or use the implicit Mate connectors visible upon hover.

Steps

1. Click 🔄.

2. Select two Mate connectors (inferred or existing), selecting the pin’s Mate connector first.
The first Mate connector you select represents the Pin (and the rotational motion of the mate, the pin rotation), the second mate connector represents the Slot (and the translational motion of the mate, the pin moving in the slot).

3. To limit the movement, check Limits and supply (optional) minimum and maximum values to control the range of motion of the Mate. Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.

4. To supply an offset distance, check Offset and supply a distance. Pin slot Mates can offset the entities only along the Z axis.

   Offset distances are visualized in the graphics area as dashed lines between the mates, displaying the value and the axis. Enter the distance in the dialog.

   When you click on a Mate, graphics are displayed to indicate the direction of the X, Y, and Z as defined by the Mate along with the offset and the range of limits dimensions (if any).

   Applying an offset should be viewed as moving the entire coordinate system. The offset is relative to the Mate connector (inferred or existing) selected first.

   You can also select to rotate the part about a specific axis: select the axis, then enter the degrees of rotation.

5. Click the entity to access the manipulator.

6. Click and drag the various manipulator handles to see which motions are allowed; notice that only rotational movement about the Z axis and translational movement along the X axis is allowed (Rz, Tx).

**Pin Slot Mate: iOS**
Steps

1. Tap 

2. Confirm that **Pin slot** is selected in the *Mate type* field.

3. Select two Mate connectors (inferred or existing) to use.

   The first Mate connector selected (inferred or existing) serves as the pin and the rotational movement point. The second Mate connector selected (inferred or existing) serves as the translational movement.

4. Optionally, tap **Offset** to provide an offset distance.

5. Optionally, tap **Limits** to set distance limits for movement.

6. Optionally, tap 

5. Optionally, tap 

7. Optionally, tap 

8. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only rotational movement about the Z axis and translational movement along the X axis is allowed (Rz, Tx).
**Pin Slot Mate: Android**

**Steps**

1. Tap 🔄.

2. Confirm that Pin slot is selected in the *Mate type* field.

3. Select two Mate connectors (inferred or existing) to use.

   The first Mate connector selected (inferred or existing) serves as the pin and the rotational movement point. The second Mate connector selected (inferred or existing) serves as the translational movement.

4. Optionally, tap Offset to provide an offset distance.

5. Optionally, tap Limits to set distance limits for movement.

6. Optionally, tap ⬇️ to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap ⏰ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

**Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only rotational movement about the Z axis and translational movement along the X axis is allowed (Rz, Tx).**
Ball Mate

Mate two entities allowing rotational movement about the X, Y and Z axis (Rx, Ry, Rz).

This functionality is available on Onshape's browser, iOS, and Android platforms.

Ball Mate: Desktop

Mate two entities allowing rotational movement about the X, Y and Z axis (Rx, Ry, Rz).
Begin by creating Mate connectors on each entity, or use the implicit Mate connectors visible upon hover.

Steps

1. Click ✨.

2. Select two Mate connectors (inferred or existing).

Be aware that you are only able to select the center of the ball.
The first Mate connector selected (inferred or existing) serves as the rotational movement point and the second Mate connector selected (inferred or existing) serves as the stationary point.

3. Click the entity to access the manipulator.

4. Click and drag the various manipulator handles to see which motions are allowed; notice that only rotational movement about the X, Y and Z axis is allowed (Rx, Ry, Rz).

**Ball Mate: iOS**

Steps

1. Tap ☞.

2. Confirm that Ball is selected in the Mate type field.

3. Select the two Mate connectors (inferred or existing) to use.

4. Optionally, tap ◀ to Flip the primary axis, Z orientation of the instances.

5. Optionally, tap ◀ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

6. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that only rotational movement about the X, Y, and Z axis is allowed (Rx, Ry, Rz).
Ball Mate: Android

Steps

1. Tap 

2. Confirm that Ball is selected in the Mate type field.

3. Select the two Mate connectors (inferred or existing) to use.

4. Optionally, tap to Flip the primary axis, Z orientation of the instances.

5. Optionally, tap to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

6. Tap checkmark.

Touch and drag OR use the Triad manipulator to move one of the parts. Notice that only rotational movement about the X, Y, and Z axis is allowed (Rx, Ry, Rz).
Parallel Mate

Mate two entities allowing individual translational movement along any axis, and parallel rotation along any axis.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Parallel Mate: Desktop

Mate two entities allowing individual translational movement along any axis, and parallel rotation along any axis.
Steps

1. Click \( \bullet \). 

2. Select two Mate connectors (inferred or existing).

3. To impose limits on the movement of the Mate, click Limits and supply minimum and maximum distances for the relative displacement of the Mate connectors. Limits are visualized in the graphics area as dashed lines with bars at the ends. The dashed lines represent the direction and distance of the movement and the solid lines represent the limit.
4. Click the entity to access the manipulator.

5. Click and drag the various manipulator handles to see which motions are allowed; notice that the parallel Mate allows the entities to translate freely in the X, Y, and Z directions and that the entities are able to rotate freely around the Z axis while the Mate keeps the Mate connectors parallel.

**Parallel Mate: iOS**

Steps

1. Tap .

2. Confirm that Parallel is selected in the Mate type field.

3. Select two Mate connectors (inferred or existing) to use.

4. Optionally, tap Limits to set distance limits for movement.

5. Optionally, tap  to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap  to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap checkmark.

Touch and drag OR use the "Triad Manipulator" on page 244 to move one of the parts. Notice that the parallel Mate allows the parts to translate freely in the X, Y, and Z directions and that the parts can rotate freely around the Z axis while the Mate keeps the Mate connectors parallel.
Parallel Mate: Android
Steps

1. Tap ![image].

2. Confirm that **Parallel** is selected in the *Mate type* field.

3. Select two Mate connectors (inferred or existing) to use.

4. Optionally, tap **Limits** to set distance limits for movement.

5. Optionally, tap ![image] to Flip the primary axis, Z orientation of the instances.

6. Optionally, tap ![image] to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

7. Tap checkmark.

Touch and drag OR use the **Triad manipulator** to move one of the parts. Notice that the parallel mate allows the parts to translate freely in the X, Y, and Z directions and that the parts are able to rotate freely around the Z axis while the Mate keeps the Mate connectors parallel.
**Tangent Mate**

Mate two entities tangent to the selected faces, edges, or vertices.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Tangent Mate: Desktop**

Mate two entities tangent to the selected faces, edges, or vertices.

Tangent Mates do not require or accept Mate connectors.
Steps

1. Click 🦇.

2. Select a face, edge, or vertex of one entity. In this instance, a face of the box is selected.

3. With focus in the second field in the Mate dialog, select a face, edge, or vertex of the second entity.

Tips

- Tangent Mate doesn't support offset surfaces.
- Flip primary axis works only when two faces are selected; for other selections it is ignored.
- Only swept faces are supported (torus, cones, etc), no generic faces (like splines).
- Tangent Mate does not work with any Relations.

**Tangent Mate: iOS**
Steps

1. Tap 📖.
2. Tap to select a face, edge, or vertex of one entity.
3. In the dialog, tap on the second *Face, edge or vertex* field to activate it.
4. Tap to select a face, edge, or vertex of one entity.
5. Optionally, toggle to Flip the primary axis.
6. Tap checkmark.

Below, both parts share a Planar Mate and the box is fixed in place. When the cylinder is dragged, it hugs the side of the box and follows the face all the way around:

![Diagram showing Planar Mate example](image)

Tips

- Tangent Mate doesn’t support offset surfaces.
- Flip primary axis works only when two faces are selected; for other selections it is ignored.
- Only swept faces are supported (torus, cones, etc), no generic faces (like splines).
- Tangent Mate does not work with any Relations.

**Tangent Mate: Android**
Steps

1. Tap 🔍.
2. Tap to select a face, edge, or vertex of one entity.
3. In the dialog, tap on the second Face, edge or vertex field to activate it.
4. Tap to select a face, edge, or vertex of one entity.
5. Optionally, toggle to Flip the primary axis.
6. Tap checkmark.

Below, both parts share a Planar Mate and the box is fixed in place. When the cylinder is dragged, it hugs the side of the box and follows the face all the way around:

Tips

- Tangent Mate doesn't support offset surfaces.
- Flip primary axis works only when two faces are selected; for other selections it is ignored.
- Only swept faces are supported (torus, cones, etc), no generic faces (like splines).
- Tangent Mate does not work with any Relations.
Group

Use Group to fix the position of selected instances relative to one another within the assembly. It is very convenient when the instances were all modeled in the same Part Studio in the correct locations relative to one another. Group enables you to keep that relative positioning without having to create Mates.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Group: Desktop

Use Group to fix the position of selected instances relative to one another within the assembly. It is very convenient when the instances were all modeled in the same Part Studio in the correct locations relative to one another. Group enables you to keep that relative positioning without having to create Mates.

Steps

1. Click .

2. Select the entities to include in the group; pre-select is available.

   You have the ability to click the entity name in the Feature list, click the entity in the graphics area, or click and drag a selection box around entities in the graphics area.

3. Accept .

Notice that a Group feature is added to the Mate Features list in the Feature list box.

Hiding and showing groups

You are able to hide (and show) groups from the context menu:
1. Right-click a group in the Feature list.
2. Select Hide instances from the context menu.

Follow the same procedure to show the group again, selecting Show instances from the context menu.

**Example**
1. Create the parts you wish to group (in a Part Studio).

2. Insert those parts into an Assembly.
3. Arrange the parts in the desired relationship to each other.
4. Select Group in the toolbar, and then select the parts to group; click ✅.
5. Open another Assembly, click Insert, select Assemblies and notice that the group is listed there as one entity:
6. Select the group and insert it; a Group instance is listed in the Instances list.

**Tips**
- Despite the selection of child entities listed in the Feature list, the group moves and behaves as a group. The child entities cannot be acted upon individually.
- You have the ability to suppress a Group and change the relative position of the entities; when you unsuppress, the Group reflects the new relative positions. (To return to the original group configuration, use Undo.)
- You have the ability to change the relative positions of the entities in a Part Studio, and the Group in the Assembly updates accordingly.
- A single part instance is only able to be selected as a group member if it is not yet contained by a subassembly. If it is contained by a subassembly and selected, then that subassembly is selected automatically instead.
For example, when Part 1 <1> is selected (above), the entire Assembly 1 <1> is selected by default.

- Adding a subassembly instance to a group makes the whole subassembly effectively rigid.
- When members of a group are deleted, they are also deleted from the group. When the last member of a group is deleted the group is also deleted.

**Group: iOS**

**Steps**

1. Tap 
2. Tap to select the parts to include in the group. Select the entity name in the Instance list or select the entity in the graphics area; pre-select is available.
3. Tap the checkmark.

A Group feature is added to the Mate Features list in the Instance list.

![Instances (3) Edit](image)

**Hiding and showing groups**

Hide (and show) groups from the overflow menu in the Feature list.

1. Select a group in the Feature list.
2. Open the overflow menu and select Hide (or Show).

The entire group is hidden or shown.
Tips

- Despite the selection of child parts listed in the Feature list, the group moves and behaves as a group. The child parts are not able to be acted upon individually.

- You have the ability to suppress a Group and change the relative position of the parts; when you unsuppress, the Group reflects the new relative positions. (To return to the original group configuration, use Undo).

- You have the ability to change the relative positions of the parts in a Part Studio, and the Group in the Assembly updates accordingly.

- Only instances of an assembly are able to be selected as group members.

- Adding a subassembly instance to a group makes the whole subassembly effectively rigid.

Group: Android

Steps

1. Tap

2. Tap to select the parts to include in the group. Select the entity name in the Instance list or select the entity in the graphics area; pre-select is available.

3. Tap the checkmark.

A Group feature is added to the Mate Features list in the Instance list.

Hiding and showing groups
You have the ability to hide (and show) groups from the overflow menu in the Feature list.

1. Select a group in the Feature list.
2. Open the overflow menu and select Hide (or Show).

The entire group is hidden or shown.

**Tips**

- Despite the selection of child parts listed in the Feature list, the group moves and behaves as a group. The child parts are not able to be acted upon individually.

- You have the ability to suppress a Group and change the relative position of the parts; when you unsuppress, the Group reflects the new relative positions. (To return to the original group configuration, use Undo).

- You have the ability to change the relative positions of the parts in a Part Studio, and the Group in the Assembly updates accordingly.

- Only instances of an assembly can be selected as group members.

- Adding a subassembly instance to a group makes the whole subassembly effectively rigid.

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**Snap Mode**

This functionality is available on Onshape’s browser and iOS platforms only.

Automatically create a Mate when inserting an entity into an Assembly, using explicitly created Mate connectors on the entity being inserted. Also, drag one entity (by a Mate connector) to snap to a Mate connector on another entity. Both methods result in an open Mate dialog so you are able to fine tune the Mate type and alignment. Snap mode is able to be toggled on or off.

**Snap Mode: Desktop**
Shortcut: Shift+s

Automatically create a Mate when inserting an entity into an Assembly, using explicitly created Mate connectors on the entity being inserted. Also, drag one entity (by a Mate connector) to snap to a Mate connector on another entity. Both methods result in an open Mate dialog so you are able to fine tune the Mate type and alignment. Snap mode is able to be toggled on or off.

**Steps**

When inserting an entity

1. Click 🔄 to toggle on (highlighted when on).

2. Open the Insert dialog, Insert an entity, and with the Insert dialog still open drag the newly inserted entity to hover over a second entity to activate inferred Mate connectors.

3. While inserting a part with more than one explicit Mate connector, use the Ctrl key to cycle through available Mate connectors on the entity being inserted:

   In the images above, the first Mate connector is highlighted upon selecting it to insert, the second Mate connector is selected upon using the Ctrl key, and the third Mate connector is selected upon...
using the Ctrl key again - repeatedly using the Ctrl key will continue to cycle through the Mate connectors

4. Click when the desired Mate connector is active.

5. To tweak the Mate, open the newly created Mate in the Feature list.

Steps with entities already inserted

1. Click 🍋 to toggle on (highlighted when on).

2. Click and drag an entity; when you start dragging, the entity becomes transparent (to aid you in seeing the Mate connectors of the second entity).

3. When you drag to the point of waking up another Mate connector, the cursor changes to show that the entities will snap together at those points when released.

   Upon release, a Mate dialog opens.

   While hovering over an active Mate connector, use the Ctrl key to cycle through available Mate connectors on the entity being inserted.

   Use the ‘A’ and ‘Q’ keys to change alignment during the Snap drag (in place of clicking the Secondary axis icon in the dialog).
4. At this point you are able to select a type of Mate (Fastened, Planar, Revolute, etc) and tweak the orientation of the Mate connector itself using the directional arrows and secondary axis tools. Use the Play button to animate the Mate behavior.

Tips

- You are able to use the 'A' and 'Q' keys to change alignment during the Snap drag (in place of clicking secondary axis icon in the dialog).
- You are able to zoom and rotate the graphics area while the entity is selected and in the process of dragging.
- When selected, entities become transparent so you are able to see where you’re going with them - when dragging an entity near another entity, the second entity’s Mate reference points become active/visible.
- As you are inserting an instance into an assembly, you are able to pan and rotate as usual, even with Snap mode on.
- Use the Ctrl key to cycle through different Mate connectors when using Snap mode while inserting an entity.

Snap Mode: iOS

Steps

1. Tap 🌠.
2. Tap to select a Mate connector (inferred or existing).

3D Rotate Lock activates once you select a Mate connector (inferred or existing), allowing you to drag the selected Mate connector without rotating the view.

3. Drag the Mate connector to the desired Mate connector (inferred or existing) that you want to connect to.

A dialog opens allowing you to fine tune your Mate type and placement using offset and the reorient tools:
• **Offset** - Tap to set an offset distance for defining a fixed space between the parts being mated.

• ⤇ - **Flip the primary axis**, Z orientation of the instances.

• ⚙ - **Reorient the secondary axis**; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

4. Accept the dialog to make and save specifications/changes.

Snap mode remains toggled on until you tap to manually toggle it off.

**Tips**

• Zoom and pan the graphics area while the Mate connector and part is selected.

• Once you have selected a Mate connector, touch and drag anywhere in the graphics area to move the part. Touch and drag with your finger above or below the Mate connector to better see what you are doing.

• Manually toggle 3D Rotate Lock on and off by tapping the icon in the upper right. Try rotating the view if you need a better angle from which to line up your Mate connectors.

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**Replicate**
Replicate takes a seed entity or entities as input, a bolt for instance, and locates geometry identical to that which the seed is mated to (based on an additional selection). The seed is then replicated and mated to that matching geometry.

This functionality is available on Onshape’s browser, iOS, and Android platforms.

**Replicate: Desktop**

Replicate takes a seed entity or entities as input, a bolt for instance, and locates geometry identical to that which the seed is mated to (based on an additional selection). The seed is then replicated and mated to that matching geometry.

This feature makes completing an assembly and BOM very efficient due to the replication of entities and mates that would otherwise be inserted and assembled manually.

**Steps**

1. Open an assembly with relevant entities inserted.
2. You need an entity to act as the seed. Make sure that is already mated as desired.
3. Click 🌊.
4. For the Instances field, select the seed entity.
5. Depending on the entity selected made when mating (edge or face), the Match Scope field may display face options or edge options:
   - Match faces in parts - Replicate the seed instance in the faces of the selected parts.
   - Match faces in features - Replicate the seed instance in the faces of the selected features.
   - Match individual faces - Replicate the seed instance in individually selected faces.
- Match edges on plane - Replicate the seed instance on the edges of the selected plane.
- Match edges on face - Replicate the seed instance on the edges of the selected face.
- Match individual edges - Replicate the seed instance on individually selected edges.

6. For the Parts to find match in field, select a part, face, or individual matches to make (depending on your choice above).

7. Click ✓.

Note that there is no Replicate feature created. If you wish, you are able to use Undo to remove the actions just taken, or you may edit each feature individually.

**Example**

1. Open an assembly with relevant entities installed.

2. Make sure the seed entity is already mated as desired.

   In this example, first the bolt is mated to the hub, by way of a Mate connector on the edge of the bolt and a Mate connector on the edge of the hole on the hub:
3. Click �atism.

4. In the Seed instances field, select the entity you want to replicate (the seed entity).
   In this example, the bolt is the seed entity.

5. Select a Face match scope.
   In this example, the Face match scope is Match face in parts.

6. Select a part to find match in.
   In this example, the hub is the part that the matches will be found in.

7. Notice that all relevant features are created and listed in the Feature list.
No Replicate feature is created in the Feature list. If you wish to make changes, you have the option to use Undo to remove the actions just taken, or you are able to edit each feature individually.

**Face match scopes**

These examples use this model:

Scope: Match faces in parts

Replicate the seed instance in the faces of the selected parts:
Scope: Match faces in features

Replicate the seed instance in the faces of the selected features:

Scope: Match individual faces

Replicate the seed instance in individually selected faces:

Tips

- If you get an error, hover over the orange dialog title for hints at what might be wrong.
Check to make sure you have the proper seed selections; and that those seeds have the desired Mate connectors and mates in place.

Seed instances may have only one external Mate; that is, there may only be one Mate to the entity onto which to replicate the seed instances.

**Replicate: iOS**

**Steps**

1. Open an assembly with relevant entities installed.

2. You need an entity to act as the seed. Make sure this entity is already mated as desired.

3. Tap 📼.

4. For the **Instances** field, select a seed entity.

5. Select a **Face match scope**:

   - **Match faces in parts** - Replicate the seed instance in the faces of the selected parts.
   - **Match faces in features** - Replicate the seed instance in the faces of the selected features.
   - **Match individual faces** - Replicate the seed instance in individually selected faces.

6. For the **Parts to find match in** field, select a part, face, or individual matches to make (depending on your choice above).

7. Tap checkmark.

**Example**

1. Open as assembly with relevant entities installed.
2. Make sure the seed entity is already mated as desired.

   In this example, first the bolt is mated to the hub, by way of a Mate connector on the edge of the bolt and a Mate connector on the edge of the hole on the hub.
3. Tap 📃.

4. For the **Instances** field, select a seed entity.
   In this example, the bolt is the seed entity.

5. Select a **Face match scope**.
   In this example the face match scope is **Match face in parts**.

6. Tap to select **parts to find match in**.
   In this example, the hub is the part that the matches will be found in.
7. Tap checkmark.

Note that there is no Replicate feature created in the Assembly list. If you wish, you have the option to use Undo to remove the actions just taken, or you are able to edit each feature individually.

Example of seeding with multiple parts

This example illustrates seeding with multiple parts (washer/lock/nut):
1. To start, the nut is mated to the lock washer, the lock washer is mated to the washer, and the washer is mated to the knuckle.

There is only one external mate: the one to the entity to which to mate the seed entities. In this case, the Mate between the washer and knuckle is the external Mate.

2. Tap 🗨️.

3. For the **Instances** field, select the entity you want to replicate.

   In this example, the nut, lock washer, and washer are the seed instances (since they are all mated to each other their Mates will also be replicated).
4. Select an **Edge to match scope**.

   In this example, **match edges on face** is selected because the seed's instances is mated by the cylindrical faces of the holes.

5. Select a **part to find match in**.

   In this example, the top face of the knuckle is the face that the matches will be found in.

6. Tap checkmark.
Notice that all relevant features are created and listed in the Assembly list.

No Replicate feature is created in the Assembly list. If you wish to make changes, you have the option to use Undo to remove the actions just taken, or you are able to edit each feature individually.

**Face match scopes**

These examples use this model:

Scope: Match faces in parts

Meaning: Replicate the seed instance in the faces of the selected parts.

Example:
Scope: Match faces in features

Meaning: Replicate the seed instance in the faces of the selected features.

Example:

Scope: Match individual faces

Meaning: Replicate the seed instance in individually selected faces.
Example:

Tips

- If you get an error, look for orange text in the dialog for hints at what might be wrong.

- Check to make sure you have the proper seed selections; and that those seeds have the desired Mate connectors and Mates in place.

- Seed instances may have only one external mate; that is, there may be only one Mate to the entity onto which to replicate the seed instances.

Replicate: Android

Steps

1. Open an assembly with relevant entities installed.

2. You need an entity to act as the seed. Make sure this entity is already mated as desired.

3. Tap 📨.

4. For the Instances field, select a seed entity.
5. Select a **Face match scope**:
   - **Match faces in parts** - Replicate the seed instance in the faces of the selected parts.
   - **Match faces in features** - Replicate the seed instance in the faces of the selected features.
   - **Match individual faces** - Replicate the seed instance in individually selected faces.

6. For the **Parts to find match in** field, select a part, face, or individual matches to make (depending on your choice above).

7. Tap checkmark.

**Example**

1. Open as assembly with relevant entities installed.

   ![Assembly diagram](image)

2. Make sure the seed entity is already mated as desired.

   In this example, first the bolt is mated to the hub, by way of a Mate connector on the edge of the bolt and a Mate connector on the edge of the hole on the hub.
3. Tap 📷.
4. For the **Instances** field, select a seed entity.
   In this example, the bolt is the seed entity.

5. Select a **Face match scope**.
   In this example the face match scope is **Match face in parts**.

6. Tap to select **parts to find match in**.
   In this example, the hub is the part that the matches will be found in.

7. Tap checkmark.

   **Note that there is no Replicate feature created in the Assembly list. If you wish, you have the option to use Undo to remove the actions just taken, or you are able to edit each feature individually.**

**Example of seeding with multiple parts**

This example illustrates seeding with multiple parts (washer/lock/nut):
1. To start, the nut is mated to the lock washer, the lock washer is mated to the washer, and the washer is mated to the knuckle.

There is only one external mate: the one to the entity to which to mate the seed entities. In this case, the Mate between the washer and knuckle is the external Mate.
2. Tap 📺.

3. For the **Instances** field, select the entity you want to replicate.

   In this example, the nut, lock washer, and washer are the seed instances (since they are all mated to each other their Mates will also be replicated).

4. Select an **Edge to match scope**.

   In this example, **match edges on face** is selected because the seed's instances is mated by the cylindrical faces of the holes.

5. Select a **part to find match in**.

   In this example, the top face of the knuckle is the face that the matches will be found in.
6. Tap checkmark.

Notice that all relevant features are created and listed in the Assembly list.

No Replicate feature is created in the Assembly list. If you wish to make changes, you have the option to use Undo to remove the actions just taken, or you are able to edit each feature individually.

**Face match scopes**

These examples use this model:
Scope: Match faces in parts

Meaning: Replicate the seed instance in the faces of the selected parts.

Example:

Scope: Match faces in features

Meaning: Replicate the seed instance in the faces of the selected features.

Example:
Scope: Match individual faces

Meaning: Replicate the seed instance in individually selected faces.

Example:

Tips
- If you get an error, look for orange text in the dialog for hints at what might be wrong.
Check to make sure you have the proper seed selections; and that those seeds have the desired mate connectors and mates in place.

Seed instances may have only one external mate; that is, there may only be one mate to the entity onto which to replicate the seed instances.

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**Replace Instance**

This functionality is currently available only on browser.

Replace instance allows you to replace selected or all instances of a part, surface, sketch, or subassembly within an assembly. Even if the replacement component is slightly different (for example, a derivative of the part), it will be inserted and all previously associated mates reapplied.

**Steps**

1. Open an assembly with relevant components inserted.

2. Click 📡.

3. When the dialog opens, select the instance (or select multiple instances) you wish to replace:

4. Click to activate the **Select replacement component** field.
5. Use the dialog that opens to navigate to the document and Part Studio that contains the replacement instance. You are able to select Current document, or you may select Other documents and select from their features, parts, surfaces, sketches, and subassemblies as well (shown below).

![Select replacement component dialog](image)

6. Use the filters and search bar to find and select a document.

   a. You have the ability to search by instance name in Part Studios or Assemblies, and other properties, including Custom properties.

   b. Search accepts multiple words as well as non-alphanumeric characters, such as punctuation.

   c. The filter to the left of the Search bar shows common properties (or all properties) to search by. You are able to use more than one filter at a time.

   d. Each search result indicates which type of entity fulfills the search criteria by the icon preceding the name, followed by the name of the entity. Below that, the workspace or version icon, the document name and workspace name (or version name, when part of the search criteria), the part number, and the release management state (shown below).

![Part 1](image)

7. Select the replacement instance (and the dialog closes on its own).
8. If you have more than one instance of the component in the assembly and wish to replace all of them, check **Replace all instances**.

9. Click ☑️ in the dialog to accept your change.

**Example**

This example replace the cap of a birdfeeder with a different cap type:

1. Click 📌

2. Select the part instance to replace:
3. Click to activate the Select replacement component field:

4. Select a part (the dialog closes immediately):

5. Click ✓.
Assembly Linear Pattern

Pattern selected entities and arrange them in a row pattern. For information on creating circular patterns, see Assembly Circular Pattern.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Assembly Linear Pattern: Desktop

Pattern selected entities and arrange them in a row pattern. For information on creating circular patterns, see Assembly Circular Pattern.

Steps

1. Click.

2. With the focus in the Instances field, select instances (part, subassembly, sketch or surface) to pattern. Click to select an entity from the workspace or from the Instance list.

3. Set focus in the Direction field, and then set the direction of the linear pattern: Click to select an edge or face of the entity along which to set the pattern.
4. In the *Distance* field, enter the distance between pattern instances.

5. In the *Instance count* field, set the number of instances for your pattern. The minimum number of instances you are able to use is 1.

6. Click ⬤ to toggle your pattern in the opposite direction.

7. Select *Equal spacing* to toggle between the specified distance being the spacing between the beginning of each instance (unchecked), or the specified distance being the spacing between the beginning of the first instance and the end of the last instance (checked).

8. Select *Second direction* to create more instances of the part, subassembly, sketch or surface in a second direction, different from the first direction:
Repeat with *Third direction*, if desired.

9. Click ☑️

**Tips**

- If you create a pattern of an entity that is in a group, the new instances are also in that group. For more information on groups, see [Group](#).

- Once an assembly is patterned, the components/Mates become rigid or locked. There will be no movement in them. You will not be able to move the individual components once the pattern is applied, nor will you be able to animate the sub-assembly components. To move the instances, you are able to suppress the pattern and move the components, at which time the pattern instances will update their positions to match the seed instance.

**Assembly Linear Pattern: iOS**
Steps

1. Tap $\text{}$.

2. With the focus in the *Instances* field, select instances (a part or subassembly) to pattern. Tap to select an entity from the graphics area or from the Instances list.

3. Set focus in the *Direction* field, and then set the direction of the linear pattern: tap to select an edge or face of the part along which to set the pattern.

4. In the *Distance* field, enter the distance between pattern instances.

5. In the *Instance count* field, set the number of instances for your pattern.

6. Select *Equal spacing* to toggle between the specified distance being the spacing between the beginning of each instance (unchecked), or the specified distance being the spacing between the beginning of the first instance and the end of the last instance (checked).

7. Select *Second direction* to create more instances of the part, subassembly, sketch or surface in a second direction, different from the first direction:
8. Tap ⇑ to toggle your pattern in the opposite direction.

9. Tap the checkmark to save your actions.

If you create a pattern of an entity that is in a group, the new instances are also in that group. For more info on groups, see Group.

Assembly Linear Pattern: Android
Steps

1. Tap 📋.

2. With the focus in the _Instances_ field, select instances (a part or subassembly) to pattern. Tap to select an entity from the graphics area or from the Instances list.

3. Set focus in the _Direction_ field, and then set the direction of the linear pattern: tap to select an edge or face of the part along which to set the pattern.

4. In the _Distance_ field, enter the distance between pattern instances.

5. In the _Instance count_ field, set the number of instances for your pattern.

6. Select _Equal spacing_ to toggle between the specified distance being the spacing between the beginning of each instance (unchecked), or the specified distance being the spacing between the beginning of the first instance and the end of the last instance (checked).

7. Select _Second direction_ to create more instances of the part, subassembly, sketch or surface in a second direction, different from the first direction:
8. Tap ⤖ to toggle your pattern in the opposite direction.

9. Tap the checkmark to save your actions.

If you create a pattern of an entity that is in a group, the new instances are also in that group. For more info on groups, see Group.

**Assembly Circular Pattern**

Pattern selected entities about an axis. For information on creating linear patterns, see Assembly Linear Pattern.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Assembly Circular Pattern: Desktop**

Pattern selected entities about an axis. For information on creating linear patterns, see [Assembly Linear Pattern](#).

**Steps**

1. Click 🔄.

2. With the focus in the Instances field, select instances (part, subassembly, sketch or surface) to pattern. Click to select an entity from the workspace or from the Instance list.

3. Set focus in the Axis of pattern field, and then select an edge, face, mate connector, or conic or cylindrical face of the entity about which to place the pattern.

4. In the Angle field, enter the distance between pattern instances, in degrees.

5. In the Instance count field, set the number of instances for your pattern. The minimum number of instances you are able to use is 1.

6. Click the opposite arrow icon to change the direction of the pattern.

7. Click to set equal spacing.

8. Click 🔄.
Once an assembly is patterned, the components become grouped (locked in place). To move the components after the pattern is applied, suppress the pattern, move the components, and then unsuppress the pattern. At this point, the pattern instances update to match the new position.

Note: if you create a pattern of an entity that is in a group, the new instances are also in that group. For more info on groups, see Group.

**Assembly Circular Pattern: iOS**

**Steps**

1. Tap 🔄.

2. With the focus in the **Instances** field, select instances (a part or subassembly) to pattern. Tap to select an entity from the graphics area or from the Instances list.

3. Set focus in the **Axis of pattern** field, and then select an edge, face, mate connector, or conic or cylindrical face of the part about which to place the pattern.

4. In the **Angle** field, enter the distance between pattern instances, in degrees.
5. In the *Instance count* field, set the number of instances for your pattern.
6. Tap *Opposite direction* to change the direction of the pattern.
7. Tap to set equal spacing.
8. Tap the checkmark to save your actions.

If you create a pattern of an entity that is in a group, the new instances are also in that group. For more info on groups, see [Group](#).

**Assembly Circular Pattern: Android**

**Steps**
1. Tap 📋.

2. With the focus in the *Instances* field, select instances (a part or subassembly) to pattern. Tap to select an entity from the graphics area or from the Instances list.
3. Set focus in the *Axis of pattern* field, and then select an edge, face, mate connector, or conic or cylindrical face of the part about which to place the pattern.
4. In the *Angle* field, enter the distance between pattern instances, in degrees.

5. In the *Instance count* field, set the number of instances for your pattern.

6. Tap *Opposite direction* to change the direction of the pattern.

7. Tap to set equal spacing.

8. Tap the checkmark to save your actions.

If you create a pattern of an entity that is in a group, the new instances are also in that group. For more info on groups, see Group.

---

**Relations**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Use relations on mates to constrain degrees of freedom between mates. Onshape currently offers these types of relations:

- **Gear Relation** - Relate two mates with revolute degrees of freedom
- **Rack and Pinion Relation** - Relate a mate with rotational degree of freedom to a mate with linear degree of freedom
- **Screw Relation** - Constrain the rotational degree of freedom in one cylindrical mate to the translational degree of freedom in the same cylindrical mate
- **Linear Relation** - Constrain the linear motion between two mates to change at a constant ratio

**Relations: Desktop**

Use relations on mates to constrain degrees of freedom between mates. Onshape currently offers these types of relations:
Steps
To add a relation:

1. Select one of the relation icons.

2. Select the required mates you want to constrain in the main list of mate features (or in the list of features in any of the current subassemblies).

3. In the relation dialog, confirm that the desired type of relation is selected:

4. Select the required mates either on the model or from the Mate Features list.

5. Specify degree of freedom for the mate, if necessary:

   a. When you select mates with the exact degrees of freedom required by the relation, Onshape displays the degree of freedom in the dialog for each mate.

      The icons indicate the specific degrees of freedom of each mate: linear movement along the Z axis for Slider, and revolving movement about the Z axis for Revolute.

   b. If you select mates with more degrees of freedom than required by the relation, a second dialog appears in which to select the desired degree of freedom for each mate selected.
6. Specify the distance per revolution (and units if not using the default units).

7. Once the appropriate degrees of freedom are selected for both Mates, the dialog is populated and ready to be accepted.

You have the ability to use expressions and trigonometric functions in numeric fields in Assemblies.

**Relations: iOS**

Use relations on mates to constrain degrees of freedom between mates. Onshape currently offers these types of relations:

- "Gear Relation" on page 1669 - creates a constant ratio of angular rotation between mates

- "Rack and Pinion Relation" on page 1673 - relates the rotational degree of freedom of a revolute mate with the linear degree of freedom of a slider mate

- "Screw Relation" on page 1677 - constrains the rotational degree of freedom in one cylindrical mate to the translational degree of freedom in the same cylindrical mate

- "Linear Relation" on page 1680 - constrains the linear motion between two slider mates to change at a constant ratio

**Steps**

To add a relation:

1. Select one of the relation icons.

2. Select the required mates you want to relate in the main list of mate features (or in the list of features in any of the current subassemblies),

3. In the relation dialog, confirm that the desired type of relation is selected and enter the required information.
The relation only succeeds if the mates chosen are appropriate for the type of relation chosen.

**Relations: Android**

Use relations on mates to constrain degrees of freedom between mates. Onshape currently offers these types of relations:

- "Gear Relation" on page 1669 - creates a constant ratio of angular rotation between mates
"Rack and Pinion Relation" on page 1673 - relates the rotational degree of freedom of a revolute mate with the linear degree of freedom of a slider mate

"Screw Relation" on page 1677 - constrains the rotational degree of freedom in one cylindrical mate to the translational degree of freedom in the same cylindrical mate

"Linear Relation" on page 1680 - constrains the linear motion between two slider mates to change at a constant ratio

Steps

To add a relation:

1. Select one of the relation icons.
2. Select the required mates you want to relate in the main list of mate features (or in the list of features in any of the current subassemblies),
3. In the relation dialog, confirm that the desired type of relation is selected and enter the required information.

The relation only succeeds if the mates chosen are appropriate for the type of relation chosen.
**Gear Relation**

Relate two Mates with revolute degrees of freedom. The relation creates a constant ratio of angular rotation between the Mates. If either mated part is moved, the other will move rotationally.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Gear Relation: Desktop**

Relate two Mates with revolute degrees of freedom. The relation creates a constant ratio of angular rotation between the Mates. If either mated part is moved, the other will move rotationally.

**Steps**

To add the Gear relation:

1. Click 🔗.

2. In the dialog, confirm that Gear is selected.

3. Select two Mates in the main list of Mate features (or in the list of features in any of the current subassemblies). Note that Revolute Mates have the exact degrees of freedom required by Gear.
4. Specify a degree of freedom for that Mate, if necessary:
   
a. When you select two Revolute Mates, no further action is needed because each has the exact degree of freedom required by Gear.

b. If you select Mates with more degrees of freedom than required, a second dialog appears in which to select the desired degree of freedom for that Mate.

5. Enter the desired gear ratio.

6. Optionally check the box to reverse the direction.

Once you select a degree of freedom for the relation, you are unable to change it unless you delete the Mate from the dialog, change the Mate type, or delete the Mate and start over.

**Gear Relation: iOS**

Steps

To add the Gear relation:
1. Tap 🔄.

![Gear 1 dialog](image)

- Gear relations require two revolute mates.
  - Relation type: Gear
  - Mate(s): No entities selected
  - Relation ratio: 1.0000
- Reverse direction (on/off)

2. In the dialog, confirm that **Gear** is select in the **Relation type** field.

3. Tap to select two Mates in the main list of Mate features (or in the list of features in any of the current subassemblies). Note that Revolute Mates have the exact degrees of freedom required by Gear.

4. Specify a **degree of freedom** for that Mate, if necessary:
   a. When you select two Revolute Mates, no further action is needed because each has the exact degree of freedom required by Gear.
   b. If you select Mates with more degrees of freedom than required, a second dialog appears in which to select the desired degree of freedom for that Mate.

5. Enter the desired gear ratio.

6. Optionally tap the slider to reverse the direction.
7. Tap the checkmark.

Once you select a degree of freedom for the relation, you are unable to change it unless you delete the Mate from the dialog, change the Mate type, or delete the Mate and start over.

**Gear Relation: Android**

**Steps**

To add the Rack and pinion relation:

1. Tap 🔄.

![Gear Relation Dialog]

2. In the dialog, confirm that **Gear** is select in the **Relation type** field.

3. Tap to select two Mates in the main list of mate features (or in the list of features in any of the current subassemblies). Note that Revolute Mates have the exact degrees of freedom required by Gear.

4. Specify a **degree of freedom** for that Mate, if necessary:
   a. When you select two Revolute Mates, no further action is needed because each has the exact degree of freedom required by Gear.
   b. If you select Mates with more degrees of freedom than required, a second dialog appears in which to select the desired degree of freedom for that Mate.

5. Enter the desired gear ratio.
6. Optionally tap the slider to reverse the direction.

7. Tap the checkmark.

Once you select a degree of freedom for the relation, you are unable to change it unless you delete the Mate from the dialog, change the Mate type, or delete the Mate and start over.

---

**Rack and Pinion Relation**

Relate a Mate with a rotational degree of freedom to a Mate with a linear degree of freedom.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Rack and Pinion Relation: Desktop**

Relate a Mate with a rotational degree of freedom to a Mate with a linear degree of freedom.

**Steps**

To add the Rack and Pinion relation:
1. Click 🔄.

2. In the dialog, confirm that Rack and pinion is selected.

3. Select two Mates in the main list of Mate features (or in the list of features in any of the current subassemblies). Note that selecting a Slider Mate and a Revolute Mate provides the exact degrees of freedom required by Rack and pinion.

4. Specify a **degree of freedom** for that Mate, if necessary:
   
a. When you select a Slider Mate and a Revolute Mate, no further action is needed because each has the exact degree of freedom required by Rack and pinion.

   b. If you select Mates with more degrees of freedom than required, a second dialog appears in which to select the desired degree of freedom for that Mate.

5. Enter the desired linear value.

6. Optionally check the box to reverse the direction.

---

**Rack and Pinion Relation: iOS**

**Steps**

To add the Rack and pinion relation:
1. Tap 🔄.

2. In the dialog, confirm that Rack and pinion is selected in the Relation type field.

3. Tap to select one Revolute Mate and one Slider Mate in the main list of Mate features (or in the list of features in any of the current subassemblies).

4. Enter desired distance per revolution.

5. Optionally, tap to reverse the direction.

6. Optionally, tap 🔄 to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap 🔄 to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.
The relation only succeeds if the Mates chosen are appropriate for the types of relation chosen. Rack and pinion requires one Revolute and one Slider Mate.

**Rack and Pinion Relation: Android**

Steps

To add the Rack and pinion relation:

1. Tap 🏁.

2. In the dialog, confirm that **Rack and pinion** is selected in the *Relation type* field.

3. Tap to select one Revolute Mate and one Slider Mate in the main list of Mate features (or in the list of features in any of the current subassemblies).

4. Enter desired distance per revolution.

5. Optionally, tap to reverse the direction.

6. Tap checkmark.

The relation only succeeds if the Mates chosen are appropriate for the types of relation chosen. Rack and pinion requires one Revolute and one Slider Mate.
Screw Relation

Constrain the rotational degree of freedom in one Cylindrical Mate to the translational degree of freedom in the same Cylindrical Mate.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Screw Relation: Desktop

Constrain the rotational degree of freedom in one Cylindrical Mate to the translational degree of freedom in the same Cylindrical Mate.

Steps

To add the Screw relation:

1. Click .

2. In the dialog, confirm that Screw is selected.

3. Select the Cylindrical Mate you want to relate in the main list of Mate features (or in the list of features in any of the current subassemblies). Note that a Cylindrical Mate is required.

4. Enter the desired ratio.

5. Optionally check the box to reverse direction.

Screw Relation: iOS
Steps

To add the Screw relation:

1. Tap 🔨.

2. In the dialog, confirm that **Screw** is selected in the *Relation type* field.

3. Tap to select a Cylindrical Mate you want to relate in the main list of Mate features (or in the list of features in any of the current subassemblies).

4. Enter desired distance per revolution.

5. Optionally, tap to reverse the direction.

6. Optionally, tap 🔄 to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap 🔄 to Reorient the secondary axis; rotate the quadrant orientation
(in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

The relation only succeeds if the Mates chosen are appropriate for the type of relation chosen. Screw relation requires one Cylindrical Mate.

**Screw Relation: Android**

**Steps**

To add the Screw relation:

1. Tap .

2. In the dialog, confirm that **Screw** is selected in the *Relation type* field.

3. Tap to select a Cylindrical Mate you want to relate in the main list of Mate features (or in the list of features in any of the current subassemblies).

4. Enter desired distance per revolution.

5. Optionally, tap to reverse the direction.

6. Tap checkmark.
Linear Relation

Constrain the linear motion between two Mates to change at a constant ratio. The first Mate will move linearly in one direction as the other Mate is moved linearly in one direction.

This functionality is available on Onshape's browser, iOS, and Android platforms.

Linear Relation: Desktop

Constrain the linear motion between two mates to change at a constant ratio. The first Mate will move linearly in one direction as the other Mate is moved linearly in one direction.

Steps

To add the Linear couple relation:

1. Click .

2. In the relation dialog, confirm that Linear is selected.

3. Select the two Mates in the main list of Mate features (or in the list of features in any of the current subassemblies). Note that two Slider Mates have the exact degrees of freedom required by Linear.
4. Specify a degree of freedom for that Mate, if necessary:

   a. When you select two Slider Mates, no further action is needed because each has the exact degree of freedom required by Linear.

   b. If you select Mates with more degrees of freedom than required, a second dialog appears in which to select the desired degree of freedom for that Mate.

5. Enter the desired linear ratio.

6. Optionally check the box to reverse the direction.

Once you select a degree of freedom for the relation, you are unable to change it unless you delete the Mate from the dialog, change the Mate type, or delete the Mate and start over.

**Linear Relation: iOS**

**Steps**

To add the Linear relation:
1. Tap ⊕.

2. In the dialog, confirm that **Linear** is selected in the *Relation type* field.

3. Tap to select the two slider Mates you want to relate in the main list of Mate features (or in the list of features in any of the current subassemblies).

4. Enter desired Relation ratio.

5. Optionally, tap to reverse the direction.

6. Optionally, tap 🔄 to Flip the primary axis, Z orientation of the instances.

7. Optionally, tap ⬆️ to Reorient the secondary axis; rotate the quadrant orientation (in the XY plane) of the instances by 90 degrees at a tap.

8. Tap checkmark.

The relation only succeeds if the Mates chosen are appropriate for the type of relation chosen. Linear relation requires two slider Mates.
Linear Relation: Android

Steps

To add the Linear relation:

1. Tap 🔄.

2. In the dialog, confirm that **Linear** is selected in the *Relation type* field.

3. Tap to select the two slider Mates you want to relate in the main list of Mate features (or in the list of features in any of the current subassemblies).

4. Enter desired Relation ratio.

5. Optionally, tap to reverse the direction.

6. Tap checkmark.

The relation only succeeds if the Mates chosen are appropriate for the type of relation chosen. Linear relation requires two slider Mates.

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**Named Positions**
This functionality is currently available only on Onshape's browser platform.

Assign a name to a specific position (that is, the Mate degree of freedom values, and absolute transforms for instances with no Mates) of the assembly and switch to a particular named position easily at any time. Keep in mind that the Mate values are relative and are able to be satisfied even if both sides of the Mate have moved.

**Steps**

1. Click 

2. Click and drag to move parts to the positions desired (optionally using mate positioning, limits, etc). Note that Mates are not required in order to create Named positions and the Named position references the position of every part in the Assembly.

3. Enter a name for the position.

4. Click + to add that Named position.

5. Repeat to create more named positions.

For example:
Using named positions

1. Click 📔.
2. Select a named position from the drop down menu:

If the position is unable to be loaded, a message that some Mate values could not be applied appears.

Reasons a position may fail to load might include:
- Deletion or modification of a mate or limit
- Deletion or modification of a part

### Updating or renaming named positions

In the event something has changed about the named position, for instance, a part is no longer in the assembly (and the named position), or you simply want to tweak a position, you can:

1. Click to open the Named position dialog.
2. Select the named position you wish to change.
3. Make the necessary changes in the model.
4. Click to save the new model positions with the selected Named position.

To rename a selected Named position:

1. Select the Named position you wish to rename.
2. In the New position name field, type a new name for the selected position.
3. Click the plus sign button to add the Named position with the new name.
4. Delete the Named position with the old name by clicking (Remove position).
Tips

- Setting up named positions beforehand may aid in effectively changing positions for modeling in context.
- The positioning of Tangent and Fastened Mates are not impacted in named positions. Fastened Mates always continue to work, however, Tangent Mates may rely on a parameter that has changed and may not work in all cases.
- If a named position fails, you may have made a change with which the named position is unable to render. Revert to an earlier point in the document history, or delete the named position and create a new one.

Display States

This functionality is currently available only on Onshape's browser platform.

Assign a name to a specific display state (that is, with certain parts, subassemblies, or mates hidden or in view) of the assembly and switch to a particular display state easily at any time.

Steps

Begin with a desired display already present in the graphics area, and then click the Display states icon in the toolbar, or click the icon first to open the dialog. That is, hide the parts or mates you do not want visible in the display (or make visible those parts or mates that you do want visible):

1. Click.

2. Hide and show parts and/or mates in the Assembly to your satisfaction for a
particular view. Use the Hide and Show commands on the context menu for a particular part or subassembly.

Select a part or mate, right-click either the item or the item name in the Instances list, select Hide (or Show). Or, hover over the item name in the Instances list and click the icon to hide or show.

3. Enter a name for the display.

4. Click + to add the display state.

5. Repeat to create more display states.

If you create a display state and then add new parts with mates or mate connectors, the explicit mate connectors show up by default in previously created display states while the implicit mate connectors and mates are hidden by default.

**Using display states**

To use a display state already created:

1. Click.

2. Select a display state from the drop down menu:

   ![Display states image]

**Updating a display state**

To change a display state to show a part that was previously hidden, or to hide a part that was previously visible:
1. Click  to open the display state dialog.

2. Select the desired display state.

3. In the Instances list, adjust the visibility of the part (click the hide/show icon, or use the context menu to change the visibility of the part instance).

4. In the Display states dialog, click  to update the display state definition.

**Removing a display state**

To remove a display state previously created:

1. Click  to open the Display states dialog.

2. Select a display state from the drop down menu.

3. Click  

The display state is immediately removed from the list.

**Tips**

- When you create a drawing from an assembly, the default display state used in the drawing is 'Show all', regardless of the display state the assembly is in at the time the drawing is created. You have the option to change the display state of any view in the assembly and update the drawing, or select a display state directly in the drawing view context menu.

- Changing the display state of an assembly will cause the Update icon in related drawings to highlight, indicating that there is an update pending for the drawing.

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**Create Part Studio in Context**

Create a new Part Studio in the context of an existing assembly; positioning the assembly specifically for use when modeling the part. The assembly appears in the Part Studio as context graphics to be used for reference.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Create Part Studio in Context: Desktop**

Create a new Part Studio in the context of an existing assembly; positioning the assembly specifically for use when modeling the part. The assembly appears in the Part Studio as context graphics to be used for reference.

**Steps**

1. Click 🎨.

2. Select a Mate connector (inferred or existing) or the Assembly origin as the origin of the new Part Studio sketch planes.

3. Click ✅.

4. A new Part Studio is created and opened.
   
   Notice the references planes appear and align with the point of origin or Mate connector (inferred or existing) you selected. A message is included at the top of the graphics area with a dropdown (Insert and go to assembly/Go to assembly).

5. Begin modeling in the context of the assembly by creating a sketch or selecting another modeling tool. Notice the assembly is shown visually dimmed.

6. Select an in-context reference entity as the basis of your action and notice the entity is highlighted in purple/pink.

   A Context object is created in the Assembly Contexts list at the top of the Part Studio’s Feature list.

   The new feature in the Feature list has a context arrow ➔ indicating the feature was created using a reference from a context of an Assembly.
7. When the part is created, select either:
   a. **Insert and go to assembly** - Insert the part back into the Assembly and open that Assembly:
      i. Select the part (or parts, if more than one was created) to insert back into the Assembly.
      ii. Click.
      iii. Notice the part in the Assembly Instances list has a context arrow indicating it was created within a specific context of the Assembly and is the primary instance of the part in context.
   b. **Go to assembly** - Open the Assembly without inserting the new part.
      If you later insert the part through the Insert part tool in the Assembly, the Context of the part is not associated with the primary instance of the part for positioning the assembly, and is represented by a dashed arrow, instead of the solid arrow representing the primary instance.
      To learn more about in-context modeling, including editing in context, updating a Context, and exiting a Context, see [Modeling in Context](#).

**Create Part Studio in Context: iOS**

**Steps**
1. In an Assembly, open the context menu (three vertical dots) for the Assembly origin (in the Assembly list).
2. Tap.
3. Select the Assembly origin to act as the new Part Studio origin: either the Assembly origin or a Mate connector in the Assembly.
4. Tap the checkmark.
5. A new Part Studio opens.
   A message is included at the top of the graphics area with a dropdown menu (containing Go to assembly/Insert and go to assembly commands).
6. Begin modeling in the context of the assembly by creating a sketch or selecting another modeling tool. Notice the assembly is shown visually dimmed.

7. Select an in-context reference entity as the basis of your action and notice the entity is highlighted. You can use the geometry of the referenced assembly to create new geometry (through the Use tool).

   A Context object is created in the Assembly Contexts list at the top of the Part Studio’s Feature list.

   The new feature in the Feature list has a context arrow icon indicating the feature was created using a reference from a context of an Assembly.

8. When the part is created, select either:

   a. Insert and go to assembly - Insert the part back into the Assembly and open that Assembly:
      i. Select the part (or parts, if more than one was created) to insert back into the Assembly.
      ii. Tap the checkmark.
      iii. Notice the part in the Assembly Instances list has a context arrow icon indicating it was created within a specific Context of the Assembly and is the primary instance of the part in context.

   b. Go to assembly - Open the Assembly without inserting the new part.

      If you later insert the part through the Insert part tool in the Assembly, the Context of the part is not associated with the primary instance of the part for positioning the assembly, and is represented by a dashed arrow icon, instead of the solid arrow representing the primary instance.

To learn more about in-context modeling, including editing in context, updating a Context, and exiting a Context, see Modeling in Context.

**Create Part Studio in Context: Android**

Create a new Part Studio in the context of an existing assembly, positioning the Assembly specifically for use when modeling the part. The assembly appears in the Part Studio as context graphics to be used for reference.
Steps

1. In an Assembly, open the context menu (three vertical dots) for the Assembly origin (in the Assembly list).

2. Tap .

3. Select the Assembly origin to act as the new Part Studio origin: either the Assembly origin or a Mate connector in the Assembly.

4. Tap the checkmark.

5. A new Part Studio opens.

   A message is included at the top of the graphics area with a dropdown menu (containing Go to assembly/Insert and go to assembly commands).

6. Begin modeling in the context of the assembly by creating a sketch or selecting another modeling tool. Notice the assembly is shown visually dimmed.

7. Select an in-context reference entity as the basis of your action and notice the entity is highlighted. You can use the geometry of the referenced assembly to create new geometry (through the Use tool).

   A Context object is created in the Assembly Contexts list at the top of the Part Studio's Feature list.

   The new feature in the Feature list has a context arrow icon indicating the feature was created using a reference from a context of an Assembly.

8. When the part is created, select either:

   a. Insert and go to assembly - Insert the part back into the Assembly and open that Assembly:

      i. Select the part (or parts, if more than one was created) to insert back into the Assembly.

      ii. Tap the checkmark.

      iii. Notice the part in the Assembly Instances list has a context arrow icon indicating it was created within a specific Context of the Assembly and is the primary instance of the part in context.
b. Go to assembly - Open the Assembly without inserting the new part.

If you later insert the part through the Insert part tool in the Assembly, the Context of the part is not associated with the primary instance of the part for positioning the assembly, and is represented by a dashed arrow icon, instead of the solid arrow representing the primary instance.

To learn more about in-context modeling, including editing in context, updating a Context, and exiting a Context, see Modeling in Context.

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**Assembly Measure Tool**

The Onshape measure tool is available in Part Studio, for sketches and parts, and in Assemblies for parts and assemblies.

This functionality is available on Onshape's browser, iOS, and Android platforms.

**Assembly Measure tool: Desktop**

The Onshape measure tool is available in Part Studio, for sketches and parts, and in Assemblies for parts and assemblies. The Measure tool appears automatically in the bottom right corner of the interface. If pre-selections are made, the tool displays some information. For more information, click the icon to open the dialog:
Hover over a measurement in the dialog to see visual representation on the model.

**Steps**

1. Select the part edges, faces, the origin, or Mate connectors (inferred or existing) to obtain measure information about it.

2. Click the Measure icon in the bottom right corner of the window to open the dialog (as shown in image above).

Measuring more than two entities in an assembly will provide you with the total area or length of those entities.

**Using values**

You are able to use the information displayed to enter values elsewhere in the system, for example, as a dimension.

1. With the Measurement dialog expanded, click to highlight the value you want to copy. One click captures the maximum precision value, clicking a second time captures the lower precision.

2. Use keyboard shortcuts to copy the value.

**Interpreting the measure information**

When you hover over measurement information in the flyout, the measurement is visualized in the graphics area, depicting the exact measurement referred to.
Bold dotted lines

Minimum distances between entities:

- Changes in X are shown in red
- Changes in Y are shown in green
- Changes in Z are shown in blue
- Center distances are shown in black

Note that when measuring to the center of a circle, you are able to select a planar face, edge, and edge of a cylinder.
Measure Tool: iOS

Steps

1. Tap the Measure tool to active it.
With the Measure tool active:

2. Tap to select desired entities to measure.

   The relevant measurement information appears in the dialog at the bottom of the screen. Tap the top of the dialog to collapse or expand it. Swipe inside the dialog to scroll up or down.

3. Tap the X in the upper right corner of the Measure dialog to close it and exit the measurement tool.

   ![Measurement Info]

   Note that the measurement tool displays information using the units of measurement that you have set for your document. To change the units of measurement, select Units from the document info panel and select desired unit.

**Measure Tool: Android**

The Onshape Measure tool is available in Part Studios, for sketches and parts, and in Assemblies for parts and assemblies; it appears in the far right of the main toolbar at the top of the screen. It displays measurement dynamically whenever you select an entity. The tool shows all possible measurements for the selected entity/entities including, but not limited to, minimum distance between entities, total surface area, angles, length, parallel distance, etc.
Steps

1. Tap the Measure tool to active it.

With the Measure tool active:

2. Tap to select desired entities to measure.

   The relevant measurement information appears in the dialog at the bottom of the screen. Tap the top of the dialog to collapse or expand it. Swipe inside the dialog to scroll up or down.

3. Tap the X in the upper right corner of the Measure dialog to close it and exit the measurement tool.

Mass Properties Tool

The Onshape Mass properties tool is available in Part Studio and Assemblies for parts and assemblies. Find the Mass properties tool in the bottom right corner of the interface, the scales icon, when you have parts selected.
This functionality is available on Onshape's browser, iOS, and Android platforms.

**Mass Properties Tool: Desktop**

The Onshape Mass properties tool is available in Part Studio and Assemblies for parts and assemblies. Find the Mass properties tool in the bottom right corner of the interface, the scales icon.

You can also use this tool to override properties and supply your own values for mass.

Properties are additive: for each additional part you select, its properties are added to the calculations in the dialog. When you apply materials to parts, the density of the material is used in the calculations in the Mass properties dialog. If a part has no material assigned, no figure for that part is used in the calculation (and a note is displayed in the dialog to that effect).

Results of mass property calculations are approximate. The calculation of the properties may vary in accuracy, depending on the complexity of the geometry.
Enabling Show calculation variance displays the value and the difference between the lower and upper bound of the calculated value. If Show calculation variance is not enabled, the computed value without the bounds is displayed. The computations of the values are not affected by the state of the Show calculation variance checkbox.

Apply Materials to parts through the context menu on a part in the Parts list (or the graphics area).

**Steps**

1. To access the Mass Properties dialog, click the small scale icon in the bottom right corner of the interface.

2. Make your selections in the graphics area, for parts.

   ![Mass Properties Dialog](image)

   Note that the center of mass is marked with on the model when the Mass properties dialog is open, as shown in the image above and in the image below.

3. If you wish to override calculations and enter your own values for Mass, select the entire assembly in the Instances list on the left. Then place a check in the Mass
Override box and enter the desired value. Appropriate recalculations are made once a new value is entered.

For any intersecting parts, the properties are calculated for each individual whole part and added together.

**Using values**

You are able to use the information displayed to enter values elsewhere in the system, for example, as a dimension.

1. With the Mass properties dialog expanded, click the value to view and highlight the max precision, click again to toggle the view to value with default decimal place setting; use shortcut keys to copy to clipboard.

2. The Mass Properties dialog provides the following information, presented from top down as shown in the tool:

   - A list of selected parts - Hover over a part in the list and a small red x appears beside it. Use this x to remove the part and it's properties from the dialog and calculations. Alternately, you can click the selected part in the Parts lists to deselect it.
Select a Mate connector, inferred or existing (optional), to calculate the Moments of Inertia more accurately (instead of to the common centroid of the selected parts), as described below:

- Mass of all parts that have a material applied
- Volume of all selected parts
- Surface area of all selected parts
- Center of mass of all parts that have a material applied
- Moments of inertia - With respect to the common centroid of the selected parts (not the Part Studio origin) and reported using the densities of the materials assigned to the selected parts. Any selected parts without materials assigned are omitted from the calculation. If no materials are assigned to any selected parts, no calculation is made.

**Mass Properties Tool: iOS**

The Onshape Mass properties tool is located in main toolbar in both Part Studios and Assemblies.

Properties are additive: for each additional part you select, its properties are added to the calculations in the dialog. When you apply materials to parts, the density of the material is used in the calculations in the Mass properties flyout. If a part has no mater-
ial assigned, no figure for that part is used in the calculation (and a note is displayed in the flyout to that effect).

**Materials** are applied to parts through the Parts list in a Part Studio.

---

### In a Part Studio

- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.

<table>
<thead>
<tr>
<th>Mass Properties</th>
<th>X</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 entities selected</td>
<td>&gt;</td>
</tr>
</tbody>
</table>

- **Mass:** 1.2 lb
- **Volume:** 14.069 in³
- **Surface area:** 99.893 in²
- **Center of mass:**
  - X: -2.521 in
  - Y: -1.73 in
  - Z: 1.448 in
- **Moments of inertia:** lb in²
  - Lxx: 2.477e0
  - Lxy: 0
  - Lxz: 0
  - Lyy: 3.8e0
  - Lyz: 0
  - Lzz: 4.27e0

The Mass properties dialog provides the following information, presented from top down as shown in the dialog.
Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document). To change the default units for your document, tap the Document menu icon the upper left, select **Units**, and select desired length, angle, and mass units.

- Selected entities field - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.
- Mass of all selected entities.
- Volume of all selected entities.
- Surface area of all selected entities.
- Center of mass.
- Moments of inertia - with respect to the common centroid of the selected parts (not the Part Studio origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in³).

If you select more than one part at a time, the combined properties of the parts are calculated.

**In an Assembly**

- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.
The Mass Properties dialog provides the following information, presented from top down as shown in the dialog.

- **Selected entities field** - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.
- **Mass of all selected entities.**
- **Volume of all selected entities.**
- **Surface area of all selected entities.**
- **Center of mass.**

Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document).

To change the default units for your document, tap the Document menu icon the upper left, select **Units**, and select desired length, angle, and mass units.
Moments of inertia - with respect to the common centroid of the selected parts (not the Assembly origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in³).

- If you select more than one part at a time, the combined properties of the parts are calculated.

**Mass Properties Tool: Android**

![Mass Properties tool shown in Part Studios toolbar on Android application.](image)

![Mass Properties tool shown in Assemblies toolbar on Android application.](image)

**In a Part Studio**

- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.
The Mass properties dialog provides the following information, presented from top down as shown in the dialog.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass</td>
<td>0.134 lb</td>
</tr>
<tr>
<td>Volume</td>
<td>3.503 in³</td>
</tr>
<tr>
<td>Surface area</td>
<td>32.134 in²</td>
</tr>
<tr>
<td>Center of mass</td>
<td></td>
</tr>
<tr>
<td>X</td>
<td>0.274 in</td>
</tr>
<tr>
<td>Y</td>
<td>0 in</td>
</tr>
<tr>
<td>Z</td>
<td>1.077 in</td>
</tr>
<tr>
<td>Moments of inertia</td>
<td></td>
</tr>
<tr>
<td>Lxx</td>
<td>8.4221E⁻²</td>
</tr>
<tr>
<td>Lxy</td>
<td>0</td>
</tr>
<tr>
<td>Lxz</td>
<td>0</td>
</tr>
<tr>
<td>Lyy</td>
<td>1.9177E⁻¹</td>
</tr>
<tr>
<td>Lyz</td>
<td>0</td>
</tr>
<tr>
<td>Lzz</td>
<td>2.3073E⁻¹</td>
</tr>
</tbody>
</table>

Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document).

To change the default units for your document, tap the Document menu icon the upper left, select **Units**, and select desired length, angle, and mass units.

- **Selected entities field** - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.
- **Mass of all selected entities.**
- **Volume of all selected entities.**
- **Surface area of all selected entities.**
- **Center of mass.**
Moments of inertia - with respect to the common centroid of the selected parts (not the Part Studio origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in³).

If you select more than one part at a time, the combined properties of the parts are calculated.

Any intersecting parts have the properties of each individual whole part calculated and added together.

In an Assembly

- To use the Mass properties tool; either select the tool and then select the part(s) OR select the part(s) first and then the tool.
- You are able to select multiple parts at a time.
- Parts may be selected in both the graphics area and the Feature list.

The Mass Properties dialog provides the following information, presented from top down as shown in the dialog.
Note that the information is presented in the default units that you have set for your document. The image above is displaying mass property information in terms of inches (the default unit for this document). To change the default units for your document, tap the Document menu icon the upper left, select **Units**, and select desired length, angle, and mass units.

- Selected entities field - tap to view the list of selected entities. Swipe left on an entity in the list to reveal a delete option. Deleting an entity from the list will remove it from the dialog and calculations.
- Mass of all selected entities.
- Volume of all selected entities.
- Surface area of all selected entities.
- Center of mass.
- Moments of inertia - with respect to the common centroid of the selected parts (not the Assembly origin) and reported assuming density of 1 in current document unit (mass per unit volume, for example: 1 lb/in³).
- If you select more than one part at a time, the combined properties of the parts are calculated.

---

**Creating Exploded Views**

Manually manipulate an assembly into an exploded view using the manipulator to drag components or groups of components from their current positions to a new position. The steps are recorded in the Exploded views panel, complete with the ability to reorder and otherwise edit the view.

This functionality is currently available only on Onshape’s browser platform.

**Steps**

With an assembly visible in the graphics area of an Assembly tab:
1. Click 🎨 to open the Exploded views panel.

2. Click the Add exploded view button at the top of the panel.

The Explode 1 view name appears above a rollback bar in the panel.

A first Explode feature appears (Explode 1); Explode features can contain one or many Explode steps

3. A gold message box appears to indicate you are now in Explode mode. You cannot create typical Assembly features (mates, pattern, relations) while in Explode mode:

   Editing Explode 1  Done

4. Click the part (or parts: you can also use box select) you wish to move to a new (exploded) location; use the manipulator that appears to drag the part to its exploded location:
You can re-position the manipulator by dragging its origin to a new location. This can allow you to drag from a different orientation.

5. After the click-drag of the part is released, a dialog box appears:

   ![Dialog Box]

   This creates the first Explode step; Explode step 1, under the Explode 1 feature

6. Change or leave the selected instance (in the Instances field).
7. Fine tune the new location by selecting the type of movement of the explode step in the dialog box: Translation or Rotation.

a. For translational movement, to use a specific entity as a reference for movement, check Specify direction, then select a face or edge to use as the direction.

b. For rotational movement, to use a specific entity as the axis along which to move the selection, check Specify axis, then select a face or edge to use as the axis.

Keep in mind that the reference to movement is the part instance itself unless a Translational or Rotational entity has been selected.

8. The Explode lines box is checked by default, uncheck the box to hide Explode lines.

9. By default, explode lines are defined on the centroid of all selected entities. To define a different explode line, click in the Edges, vertices, or face for explode lines field, and select the edge, vertex, or face for those entities for which you want explode lines to appear. The explode line will appear at the centroid of the edge, vertex or face selected for each entity.

10. You can use the manipulator that appears to drag the selected instances, or type a distance in the Translation distance field (or the number of degrees for Rotational movement) in the dialog box. You can use negative numbers to change the direction, if necessary.
11. Click the check mark to accept and close the dialog for Explode step 1.

Notice that previously selected instances are still selected after committing the Explode step. To continue creating more steps for Explode 1 with existing selections, simply drag the manipulator to another location.

To create more steps for different selections, click on other instances in the graphics area that you want to move to an exploded location and follow the steps above. Clicking on an instance not previously exploded in the last step clears all previous selections and selects only the new selection.

12. To finish Explode 1, click Done in the gold message box.

To create more Exploded views, click the Add exploded view button at the bottom of the panel, and repeat the steps above. You can create as many exploded views as you wish. Note that explode lines will automatically inherit the selections that are defined in the previous explode step.

Exploded views can be selected (individually) as a view in a drawing. If you want all your exploded steps to be visible in one drawing view, create one Explode with many Explode steps. If you want to select different exploded steps for different purposes or drawing views, create many individual Explode features to select from when creating a drawing. For more information on selecting Exploded views in drawings, see "Insert view" on page 1810.

Notice the explode step is listed in the Exploded views panel. Remember that you can edit the explode steps, as well as rename the explode step and drag it to a new position in the list when you have more steps created or to a different exploded view.
Editing an exploded view
To edit an exploded view, double-click on the Explode instance or right-click on the name in the Exploded views panel. With the context menu you can:

- Activate the exploded view, meaning to make it display in the graphics area and size it to fit the space
- Rename the exploded view
- Reset the starting position of the view: resetting the starting position changes the starting position of the explode to reference the new position of the instance, in the case you have changed the position
- Delete the exploded view

Exploded views can be seen by users who are following in Follow mode. This includes the movement of exploded views, the Exploded view panel, and exploded view activation.

Editing step names and steps
To edit the names of the exploded view’s steps or the exploded view itself, right-click on the name and select Rename. To edit the step, right-click the step name and select Edit (or you can double-click the step name to open the dialog). You can remove and add part instances, reselect an axis, change translation to rotation or vice versa and specify a different translational or rotational distance.

Using the Rollback bar
You can drag the rollback bar to view the explode in different stages (if there is more than one explode step). Any steps beneath the rollback bar are suppressed until the rollback bar is returned to the end. You can also left-click on the rollback bar, then use the up and down arrow keys to walk through the explode steps.

Configuring exploded views
You can configure an exploded view the same way you configure any other feature.
1. Begin a configuration:
   a. Open the configuration panel and click Configure Assembly.
   
b. Enter a name for the configuration.

2. Open the Exploded views panel:
   a. Click Add exploded view.
   
b. Select the parts you wish to explode, and drag the manipulator.
   
c. When the explode step dialog opens, right-click in the field you wish to configure and select Configure.
   
d. The field is outlined in a yellow broken line to indicate it is the field to configure.

3. Return to the Configuration panel to see the field and default values in the table.

When viewing exploded views in the Assembly, you can use the Configuration section at the top of the Instances list to control which configured explode you are viewing:

If you select a configuration that includes parts that are not used in an exploded view, that exploded view will go red in the Exploded view list. Only the exploded steps that specify the missing part instances will result in an error. Selecting another configuration with the parts instances will resolve the exploded views.

For example, the configuration below doesn't include screws, but the explode's Step 1 includes instances of the screw part, therefore, both the step and the Explode StdContent feature fail (seen below):
Using exploded views in drawings

To use an exploded view in a drawing, you only need to have created an exploded view (or views) in an Assembly in Onshape. When inserting a view into a drawing, you have the option to select an exploded view, no matter what view you used when creating the drawing (if any). Exploded drawing views will always show the exploded lines defined in the assembly.

Simply open the Insert view dialog, and specify a value for Explode/Position: expand the dropdown and select one of the named Exploded views to use for that particular view being inserted.
Note that moving the rollback bar in the Exploded tree between explode steps will not affect the drawing view. The drawing views reflect all explode steps in an Exploded view.

For more information about inserting views into a drawing, see Views.
Modeling In-Context

This functionality is available on Onshape's browser, iOS, and Android platforms.

Modeling In-Context: Desktop

Modeling a part in the context of surrounding parts is a powerful way to design top-down. Onshape provides two methods of designing parts top-down. Each method has its own strengths, so you might want to use different methods for different designing scenarios.

- Use Onshape multi-part Part Studios when you have a strong understanding of your design intent at the start of your design process, and you want to use the power of a single parametric history to drive several inter-related parts.

- Use In-Context modeling (explained in this topic) when you need relationships between parts that were created in separate Part Studios, or even in different (linked) documents, or when geometric relationships are dependent on assembly position. This often occurs when your Assembly already exists and you need to make some in-context tweaks to one of the parts. This approach also scales well to large assemblies, when it's not feasible to have one parametric history drive all the parts.

  In addition to editing existing parts, you are also able to create an entirely new part in-context using the Create Part Studio in context tool.

The Edit in context command is initiated in an Assembly. Select the part you wish to edit, and access the command from the context menu.

Upon initiating the command, Onshape:

- Switches focus to the Part Studio of the part selected
- Displays the assembly as visually ghosted in the Part Studio
- Creates the Context object (a snapshot) when a reference is made by selecting a ghosted entity as a reference point during the edit process
Captures all the geometry of the components in the Assembly (and stores the information in the Context object)

Captures the positions of the components in the assembly (and stores the information in the Context object)

Context objects are listed just above the Feature list in the Part Studio, see the illustration of a Part Studio, below:

1. List of Context objects in this Part Studio
2. Button to return to the Assembly
3. Ghosted parts involved in the Assembly, in the articulated position at the time Edit in context was initiated
4. Newly created Feature (Extrude new) referencing the face of the assembled parts
5. Opaque parts originally created in this Part Studio; in the position they were created

When editing In Context, you are able to make as many references as you’d like to any of the parts in the assembly - safe in the knowledge that those references will never be lost or broken, so your part will never fail.

You are also able to create multiple Contexts of the same Assembly in various articulated positions and update a Context, manually, in the Part Studio or Assembly if
desired. Updates are never automatic; you control if and when to update and what to update through the Update context command. This prevents accidental changes to in-context parts that might occur as a result of moving or redefining other parts in the Assembly.

**How it works**

In the Assembly, right-click the part you want to edit (called a primary instance), then select Edit in context to open the Part Studio containing that part. The assembled parts are visualized in the Part Studio (ghosted) around the primary instance in the same spacial relationship as in the Assembly. In addition, all parts originally in the Part Studio are displayed, opaque, placed as they were created.

For example, the shears and handles shown in Main Assembly are defined in separate Part Studios. Notice that the blade and limit plate do not meet:

![Image of shears and handles](image1.png)

To ensure that the limit plate and blade meet, select the limit plate (the part to edit), right-click and select Edit in context to open the Part Studio in which the limit plate was created, visualized with the assembly:

![Image of adjusted parts](image2.png)

Extrude up to face, using the blade as a reference point for the Extrude feature:
Return to the Assembly (click Go to assembly) and see the edits there.

If the design intent was clear at the outset, all the parts could have been designed in a single Part Studio. This example assumes the assembly had already been built with parts from different Part Studios, so Edit in context is the best option.

You have the ability to create a new Part Studio using the present Context of the Assembly with the [Create Part Studio in Context](#) Assembly tool.

**Edit in context steps**

Edit a Part Studio within the context of an Assembly:

1. In an Assembly, insert parts and position them as desired by adding Mates and relations, or by using the triad manipulator.
2. Right-click on a part to use as the reference point (primary instance) and select Edit in context from the context menu and then select the particular context in which to edit. (You also have the option to create a new context from this menu.)

The Part Studio containing the selected part opens with the entire Assembly visualized in a ghosted state.

3. Make edits as desired, referencing faces, edges, or parts of the Assembly as needed. (Note that nothing automatically updates, you can manually update your Part Studio or Assembly when you want, see "Update Context" on the next page, below.)

Selecting a reference point on another part in the Assembly (aside from the primary instance) creates a Context object above the Feature list. You are able to use as many reference points as needed. (Selected reference points are highlighted in purple)

Be aware that you can repeat these steps and create many Contexts of an Assembly in the same Part Studio as well as switch between them, so be sure to rename the Context with a meaningful name.

4. When finished editing, navigate back to the Assembly, in any of the following ways:
   a. Click Go to assembly at the top of the Graphics area.
   b. Select the Assembly tab at the bottom of the window.
   c. Right-click in the empty space and select Go to assembly.

   Note the edits are visible in the Assembly.

Tips

- You are able to turn off a Context to make unrelated edits at any time by selecting None in the Assembly contexts list above the Feature list.

- Parts created without referencing the part being edited in-context are not automatically inserted into the Assembly.

- Best practice is to rename a Context immediately with a meaningful name; many Context objects can be created for a single Part Studio.
Edit in context and select an existing Context when:

- You need to add additional geometric relationships
- You need to edit existing geometric relationships

To edit in context and select an existing Context: open the context menu, select the name of the existing Context you wish to edit and select "Edit in context."

**Update Context**

Once you have created a Context, you can make changes either in the Part Studio or in the Assembly and choose to update that Context in the Part Studio if you wish. This enables you to work in a Context in the Part Studio and not affect your Assembly unless you want to update your Assembly with the changes. The same is true for changing the Context from within an Assembly. You can make changes to the Assembly, then switch to the Part Studio and update the Context there to view the changes if you wish. Updates are never applied automatically.

**To update a Context in an Assembly:** Select the part edited in context, right-click and select Update context and then the context you want to update:

**To update a Context in a Part Studio:** Select the Context in the Assembly context list, click • and then Update context:
Example

The original Context below was created with a straight purple blade:

The blade was then edited to have a curved edge:

In the Assembly, select Update context (of the limit plate) to update the shears with the curved blade. Since the limit plate was extruded up to the face of the blade, when updated, the limit plate is recalculated up the face of the new, curved blade:
Editing a Part Studio in multiple Contexts

Since it's possible to have multiple Contexts in one Part Studio, this example shows a Part Studio with two Contexts: one that references a ball valve in the closed position, and another that references the ball valve in the open position. A stop is modeled in each Context, so the final design has a stop for the open position and the closed position.

In the Part Studio, the ball valve in the closed position, in context:

![Image of ball valve in closed position]

In the Part Studio, the ball valve in the open position, in context:

![Image of ball valve in open position]
The final design of the stop mechanism in the Part Studio:

Set primary instance

The primary instance is created when you edit a part in-context and is indicated by a solid arrow beside the feature in the Feature list (in the Part Studio) and beside the part in the Instance list (in the Assembly). The primary instance defines the anchor part (the part selected for the Edit in context command) for the placement of the ghosted assembly in the Part Studio; all other assembly components appear in the Part Studio in relationship to that primary instance.

You have the ability to change the primary instance of a Context at any time and may wish to do so especially in the case of a broken or missing primary instance:

1. In the Assembly, select a part.
2. Right-click to access the menu, and select <context name> > Set as primary instance.

The new primary instance is marked with a solid arrow in the Parts list, and the previous primary instance (if present) is marked with a dashed arrow.

Rename Context

Context objects are given a default name when they are created. To avoid confusion, rename each Context with meaningful names as it is created.
In the Part Studio that contains the Context object:

1. In the Assembly Contexts list (located above the Feature list), select the Context from the dropdown.

2. Click the that becomes active when the Context name is selected.

3. Select Rename.

4. Type a new name and press Enter or click ✓

Selecting a workspace to edit

When working in one workspace and inserting a part from a version, when you select to Edit in context, the Select a workspace to edit dialog opens (since you cannot edit the part in an immutable version). This allows you to select the specific workspace in which to edit the part:

In the dialog, select the specific workspace (Main or B2, in this case) in which to edit in context.

Exit edit in context

Use this command when you want to end an edit in context session and return to the Part Studio without a Context and without creating a Context object.

While editing In-Context in the Part Studio:

Right-click either in the white graphics area, or on the part being edited and select Exit context or click the X in the gold banner at the top of the graphics area:
Modeling In-Context: iOS

Context objects are listed just above the Feature list in the Part Studio. See the illustration of a Part Studio below:
1. List of Context objects in this Part Studio

2. Icon to return to the Assembly

When editing in context, you are able to make as many references as you'd like to any of the parts in the assembly - safe in the knowledge that those references will never be lost or broken, so your part will never fail.

You are also able to create multiple Contexts of the same Assembly in various articulated positions and update a Context, manually, in the Part Studio or Assembly if desired. Updates are never automatic; you control if, when, and what to update through the Update context command. This prevents accidental changes to in-context parts that might occur as a result of moving or redefining other parts in the Assembly.

**How it works**

In the Assembly, select Edit in context from the context menu of the part you want to edit. The Part Studio containing that part opens. The assembled parts are visualized in the Part Studio (ghosted) around the primary instance in the same spacial relationship as in the Assembly. In addition, all parts originally in the Part Studio are displayed, opaque, placed as they were created.

For example, the shears and handle shown below are defined in separate Part Studios. Notice that the blade and limit plate do not meet:

![Shears and handle](image)

To ensure that the limit plate and blade meet, select the limit plate (the part to edit), open the context menu and select Edit in context to open the Part Studio in which the limit plate was created, visualized with the assembly:
Extrude up to face, using the blade as a reference point for the Extrude feature. Return to the Assembly (tap Go to assembly) and see the edits there.

If the design intent was clear at the outset, all the parts could have been designed in a single Part Studio. This example assume the assembly had already been built with parts from different Part Studios, so Edit in context is the best option.

**Edit in context steps**

Edit a Part Studio within the context of an Assembly:

1. In an Assembly, insert parts and position them as desired by adding Mates and relations, or by using the triad manipulator.

2. Double-tap on a part to use as the reference point (primary instance) and select Edit in context from the context menu.

   The Part Studio containing the selected part opens with the entire Assembly visualized in a ghosted state.

3. Make edits as desired, referencing faces, edges, or parts of the Assembly as needed. (Note that nothing automatically updates, you are able to manually update your Part Studio or Assembly when you want, see "Update Context" on page 1724, below.)
Select a reference point on another part in the Assembly (aside from the primary instance) creates a Context object above the Feature list. You are able to use as many reference points as needed. (Selected reference points are highlighted in purple.)

Be aware that you are able to repeat these steps and create many Contexts of an Assembly in the same Part Studio as well as switch between them, so be sure the rename the Context with a meaningful name.

4. When finished editing, navigate back to the Assembly, in any of the following ways:

a. Tap Go to assembly at the top of the Graphics area.

b. Select the Assembly tab at the bottom of the window

Note the edits are visible in the Assembly.

**Update Context**

Once you have created a Context, you are able to make changes either in the Part Studio or in the Assembly and choose to update that Context in the Part Studio if you wish. This enables you to work in a Context in the Part Studio and not affect your Assembly unless you want to update your Assembly with the changes. The same is true for changing the Context from within an Assembly. You have the ability to make changes to the Assembly, then switch to the Part Studio and update the Context there to view the changes if you wish. Updates are never applied automatically.

To update a Context in an Assembly: Select the part edited in-context, access the context menu and select Update context.

To update a Context in a Part Studio: Select the Context in the Assembly Context list, tap the horizontal-dot menu and select Update context.

**Set primary instance**

The primary instance is created when you edit a part in-context and is indicated by a solid arrow beside the feature in the Feature list (in the Part Studio) and beside the part in the Instance list (in the Assembly). The primary instance defines the anchor part (the part selected for the Edit in context command) for the placement of the ghosted assembly in the Part Studio; all other assembly components appear in the Part Studio in relationship to that primary instance.
You have the ability to change the primary instance of a Context at any time and may wish to do so especially in the case of a broken or missing primary instance:

1. In the Assembly, select a part.

2. Open the context menu and select the Context and then Set as primary instance.

The new primary instance is marked with a solid arrow in the Parts list, and the previous primary instance (if present) is marked with a dashed arrow.

**Rename Context**

Context objects are given a default name when they are created. To avoid confusion, rename each Context with meaningful names as it is created.

In the Part Studio that contains the Context object:

1. In the Assembly Contexts list (located above the Feature list), select the Context from the dropdown.

2. Tap the vertical-dot menu that becomes active when the Context name is selected.

3. Select Rename.

4. Type a new name and tap the checkmark.

**Exit edit in context**

Use this command when you want to end an edit in context session and return to the Part Studio without a Context and without creating a Context object.

While editing in-context in the Part Studio:

Tap the x in the gold banner at the top of the graphics area.

**Selecting a workspace to edit**

When working in one workspace and inserting a part from a version, when you select to Edit in context, the Select a workspace to edit dialog opens (since you are not able to edit the part in an immutable version). This allows you to select the specific workspace in which to edit the part.
Context objects are listed just above the Feature list in the Part Studio. See the illustration of a Part Studio below:

1. List of Context objects in this Part Studio
2. Icon to return to the Assembly

When editing in context, you can make as many references as you'd like to any of the parts in the assembly - safe in the knowledge that those references will never be lost or broken, so your part will never fail.

You are also able to create multiple Contexts of the same Assembly in various articulated positions and update a Context, manually, in the Part Studio or Assembly if desired. Updates are never automatic; you control if, when, and what to update through the Update context command. This prevents accidental changes to in-context parts that might occur as a result of moving or redefining other parts in the Assembly.

**How it works**
In the Assembly, select Edit in context from the context menu of the part you want to edit. The Part Studio containing that part opens. The assembled parts are visualized in the Part Studio (ghosted) around the primary instance in the same spacial relationship as in the Assembly. In addition, all parts originally in the Part Studio are displayed, opaque, placed as they were created.

For example, the shears and handle shown below are defined in separate Part Studios. Notice that the blade and limit plate do not meet:

![Image of shears and handle]

To ensure that the limit plate and blade meet, select the limit plate (the part to edit), open the context menu and select Edit in context to open the Part Studio in which the limit plate was created, visualized with the assembly:

![Image of shears with limit plate meeting]

Extrude up to face, using the blade as a reference point for the Extrude feature. Return to the Assembly (tap Go to assembly) and see the edits there.
If the design intent was clear at the outset, all the parts could have been designed in a single Part Studio. This example assume the assembly had already been built with parts from different Part Studios, so Edit in context is the best option.

**Edit in context steps**

Edit a Part Studio within the context of an Assembly:

1. In an Assembly, insert parts and position them as desired by adding Mates and relations, or by using the triad manipulator.

2. Double-tap on a part to use as the reference point (primary instance) and select Edit in context from the context menu.

   The Part Studio containing the selected part opens with the entire Assembly visualized in a ghosted state.

3. Make edits as desired, referencing faces, edges, or parts of the Assembly as needed. (Note that nothing automatically updates, you are able to manually update your Part Studio or Assembly when you want, see "Update Context" on page 1724, below.)

   Select a reference point on another part in the Assembly (aside from the primary instance) creates a Context object above the Feature list. You are able to use as many reference points as needed. (Selected reference points are highlighted in purple.)

   Be aware that you are able to repeat these steps and create many Contexts of an Assembly in the same Part Studio as well as switch between them, so be sure to rename the Context with a meaningful name.

4. When finished editing, navigate back to the Assembly, in any of the following ways:
a. Tap Go to assembly at the top of the Graphics area.
b. Select the Assembly tab at the bottom of the window

   Note the edits are visible in the Assembly.

**Update Context**

Once you have created a Context, you are able to make changes either in the Part Studio or in the Assembly and choose to update that Context in the Part Studio if you wish. This enables you to work in a Context in the Part Studio and not affect your Assembly unless you want to update your Assembly with the changes. The same is true for changing the Context from within an Assembly. You are able to make changes to the Assembly, then switch to the Part Studio and update the Context there to view the changes if you wish. Updates are never applied automatically.

To update a Context in an Assembly: Select the part edited in-context, access the context menu and select Update context.

To update a Context in a Part Studio: Select the Context in the Assembly Context list, tap the horizontal-dot menu and select Update context.

**Android Set primary instance**

The primary instance is created when you edit a part in-context and is indicated by a solid arrow beside the feature in the Feature list (in the Part Studio) and beside the part in the Instance list (in the Assembly). The primary instance defines the anchor part (the part selected for the Edit in context command) for the placement of the ghosted assembly in the Part Studio; all other assembly components appear in the Part Studio in relationship to that primary instance.

You are able to change the primary instance of a Context at any time and may wish to do so especially in the case of a broken or missing primary instance:

1. In the Assembly, select a part.
2. Open the context menu and select the Context and then Set as primary instance.

The new primary instance is marked with a solid arrow in the Parts list, and the previous primary instance (if present) is marked with a dashed arrow.

**Rename Context**
Context objects are given a default name when they are created. To avoid confusion, rename each Context with meaningful names as it is created.

In the Part Studio that contains the Context object:

1. In the Assembly Contexts list (located above the Feature list), select the Context from the dropdown.

2. Tap the vertical-dot menu that becomes active when the Context name is selected.

3. Select Rename.

4. Type a new name and tap the checkmark.

**Exit edit in context**

Use this command when you want to end an edit in context session and return to the Part Studio without a Context and without creating a Context object.

While editing in-context in the Part Studio:

Tap the x in the gold banner at the top of the graphics area.

**Selecting a workspace to edit**

When working in one workspace and inserting a part from a version, when you select to Edit in context, the Select a workspace to edit dialog opens (since you are unable to edit the part in an immutable version). This allows you to select the specific workspace in which to edit the part.
Drawings

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

You have the ability to create mechanical drawings from within Onshape Part Studios and Assemblies and also of entire Part Studios. All Onshape drawings are based on the .DWG file format (drawing database) and the .DXF file format (Drawing Interchange File) is also supported.

You are able to create a drawing on iOS devices, however, the only actions you are able to take on them in iOS is rename, change the drawing URL, delete, and export.

To learn more about creating drawings in Onshape, follow a self-paced course here: [Detailed Drawings](opens in new tab).

Important

Keep in mind that currently, simultaneous editing is not supported in drawing elements. If you try to activate a drawing element that another user has already activated...
(in a shared document), you will see a message explaining who is currently editing the tab.

### Keyboard shortcuts

<table>
<thead>
<tr>
<th>Shortcut</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>f</td>
<td>Zoom to fit</td>
</tr>
<tr>
<td>w</td>
<td>Zoom window</td>
</tr>
<tr>
<td>Shift-z</td>
<td>Zoom in</td>
</tr>
<tr>
<td>z</td>
<td>Zoom out</td>
</tr>
<tr>
<td>d</td>
<td>Dimension</td>
</tr>
<tr>
<td>Shift-r</td>
<td>Radial dimension</td>
</tr>
<tr>
<td>Shift-d</td>
<td>Diameter dimension</td>
</tr>
<tr>
<td>Shift-q</td>
<td>Toggle on midpoints and quad points</td>
</tr>
<tr>
<td>m</td>
<td>Maximum/minimum dimension</td>
</tr>
<tr>
<td>n</td>
<td>Note annotation</td>
</tr>
<tr>
<td>Ctrl-q</td>
<td>Update drawing</td>
</tr>
<tr>
<td>l</td>
<td>Line</td>
</tr>
<tr>
<td>p</td>
<td>Create Projected view</td>
</tr>
<tr>
<td>s</td>
<td>Display shortcut toolbar (if enabled; Esc key to close)</td>
</tr>
<tr>
<td>Ctrl-s</td>
<td>Display sheet menu</td>
</tr>
<tr>
<td>PgDn</td>
<td>Next sheet</td>
</tr>
<tr>
<td>PgUp</td>
<td>Previous sheet</td>
</tr>
<tr>
<td>Home</td>
<td>First sheet</td>
</tr>
<tr>
<td>End</td>
<td>Last sheet</td>
</tr>
<tr>
<td>Delete</td>
<td>Delete selected entity</td>
</tr>
</tbody>
</table>

### Drawings context menu
Right-click on a Drawing tab to access the context menu:

- **Delete** - Delete the Drawing (or any tab), even if it is active. The last remaining tab cannot be deleted.
- **Open in new browser tab** - Open this Drawing in a new browser tab.
- **Rename** - Access the dialog to rename this Drawing.
- **Properties** - Access the dialog to provide information about the Drawing. In the Properties dialog, you can provide meta data for the entire Drawing. Properties that are grayed out (inactive) are defined and populated through the Company’s properties in Account management. See Manage Companies > Properties for more information.
- **Duplicate** - Copy this Drawing tab and insert the copy into this same document. All references to the original Part Studio are maintained.
- **Copy to clipboard** - Make a copy of this Drawing tab on the clipboard. You can then use the menu in another document and the Paste tab command to add the Drawing tab into that document. When a Drawing tab is copy/pasted into another document, the Part Studio from which it was created is also pasted into the other document. No references to the original document are maintained.
- **Change to version** - Select a different version of this document from which to create this drawing. You can update to the latest version with a click, or handpick an earlier version.
- **Select as document thumbnail** - Use an image of this drawing as the document thumbnail.
- **Update linked document** - Update the document you linked with by inserting parts, Assemblies, drawing views, or derived parts.
- **Move to document** - Move the Drawing to a new document, creating the document during this operation (or selecting an existing document). If any part or assembly is used in any tab of the original document, a link between the two documents is created. Note that, the Assembly tab and the Part Studios from which the part instances are referenced will all move to the new document. This action will be prevented if it would result in a document with no tabs.
bullet **Export** - Export the Drawing in a variety of formats with options of where to download or keep in a separate Onshape tab.

bullet **Release** - Create a Release candidate containing this drawing.

---

**Drawing Basics**

This functionality is currently available only on Onshape’s browser platform.

This functionality is also available on iOS and Android in a limited form.

---

There are three ways to create a drawing:

- Using a part, surface, Assembly, or Part Studio (presented below)
- As an empty drawing, from scratch
- By "Importing a Drawing" on page 1962 a .DWG or .DXF file

**Navigating within drawings**

Navigating within drawings can be customized to accommodate your familiarity with some traditional CAD systems. See [Setting Preferences](#) for more information.

**Basic workflow**

1. **Create a drawing of a part, curve, or surface in a Part Studio, or of a sub-assembly in the Assembly list:**
   a. Right-click on the name of the part, curve, or surface in the Part list or assembly in the Assembly list.
   b. Select Create drawing of <name>.
c. Choose a template.

Notice that you are able to select from Onshape-supplied templates, by selecting the Onshape filter on the left (or Show Onshape drawing templates).

If you are a member of a company or a team, those filters are in the list as well (as the company or team name) under Existing templates.

You are also able to create your own "This functionality is currently available only on Onshape's browser platform." on page 1753.

d. Click OK.

For more details on creating drawings, see "You can view, but not create, drawings on iOS and Android mobile platforms." on page 1748.

2. Create "Views" on page 1807. The drawing is created, by default, with no views but with a view at your cursor. Use the Escape key to cancel the view, or click to place it in the drawing. You are also able to create additional views.
3. **Add dimensions:**

   a. Select a dimension tool.

   ![Tool Icons]

   Notice that some of the tool icons have open circles; these tools use snap points, the other tools use edges.

   b. Snap points appear as open boxes: hover and then click when the appropriate snap points are visible (or select necessary edges).

   ![Dimension Text Box]

   The dimension text box appears on the click of the second snap point or edge.
c. Drag dimension text box to desired location and click to place.

For more details, see "Dimensions" on page 1853.

4. Export to .DXF, .DWG, or PDF:
   a. Right-click on the drawing tab.
   b. Select the preferred format.
      Access the downloaded file on your local drive.

5. Print the file:
   a. Open the downloaded file in a compatible application.
   b. Print the file.

All drawings objects (except view geometry) have context menus: right-click on an object to access the context menu. These menus vary with the type of object.

**Drawings cursors**

It is worth noting that the cursor will change depending upon what type of selection a command requires. The two types of cursors you will see when working with drawings are:

- + - Indicates a requirement to select a position
- ◊ - Indicates a requirement to select an entity

**Drawings grip points**

The grip points that appear on drawings entities indicate how you can use them.
Circular grip points can be used to move an entity. Square grip points can be used to resize an entity. Red grip points indicate an error. More details are shown in the table below:

<table>
<thead>
<tr>
<th>Grip point type</th>
<th>Hover</th>
<th>Selected</th>
<th>Cursor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal resize</td>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>Rotate: Drag these to rotate</td>
<td><img src="image4.png" alt="Image" /></td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>Flip: Click these to flip the orientation</td>
<td><img src="image7.png" alt="Image" /></td>
<td><img src="image8.png" alt="Image" /></td>
<td><img src="image9.png" alt="Image" /></td>
</tr>
<tr>
<td>Point attachment: Drag these to reattach to another point</td>
<td><img src="image10.png" alt="Image" /></td>
<td><img src="image11.png" alt="Image" /></td>
<td><img src="image12.png" alt="Image" /></td>
</tr>
<tr>
<td>Edge attachment: Drag these to reattach to another edge</td>
<td><img src="image13.png" alt="Image" /></td>
<td><img src="image14.png" alt="Image" /></td>
<td><img src="image15.png" alt="Image" /></td>
</tr>
<tr>
<td>Dimension text: The entire dimension is draggable, no grip points are necessary</td>
<td><img src="image16.png" alt="Image" /></td>
<td><img src="image17.png" alt="Image" /></td>
<td><img src="image18.png" alt="Image" /></td>
</tr>
</tbody>
</table>

Inferencing and snap points

Callouts and dimensions in Onshape drawings have automatic inferencing between each other upon creation and before placement in the drawing. Turn off inferencing by holding the Shift key during the operation, if you wish.
Once placed, there is no association among entities; each entity can be dragged on its own, even after snapping alignment.

**Copying and pasting drawing entities**
Certain entities in drawings are able to be copied and pasted for ease of duplication. These entities may then be tweaked for changes, instead of creating similar constructs from scratch. These entities are:

- Notes
- GDT frames
- Datum
- Hole callout dimensions
- BOM tables
- Surface finish symbols
- Weld symbols
- Balloons
- Tables

To duplicate these entities:

1. Hover the cursor over the entity; it will show in highlighting.
2. Use the Alt key and the LMB (left mouse button); the cursor will show a + indication.
3. Continue to hold down the Alt key and LMB and drag the entity to the desired location.
4. Release the mouse button to place the entity.
5. Repeat as desired.

---

**Creating a Drawing**

You can view, but not create, drawings on iOS and Android mobile platforms.
When you create a drawing of a part, curve, or surface in a Part Studio, or in an Assembly, the drawing contains default views. You are also able to create an empty drawing using the menu in the lower left corner of the window and select Create Drawing.

Create drawing with default views

From the Parts list in a Part Studio or Assembly:

Right click on a part, curve, or surface, select Create drawing of <part, curve, or surface name>.

Create empty drawing

From the Create tab menu:

Click on the plus sign icon, then select Create Drawing:
Create from existing drawings files

Import an existing drawing file in .DWG or .DXF format. You are able to import from the Documents page and from within a document.

When importing from the Documents page, a new document is created:

1. Click the Create button in the top left corner of the page:

   ![Create button](image)

2. From the dropdown menu, click Import files...

   ![Import files dropdown](image)

   1. Select the .DWG or .DXF file.
   2. Select an owner for the document (if available).
3. Note the new document listed on the Documents page. (The document name is the same as the file name.)

When importing from within a document:

1. Click +, then ➩ Import.

2. Select the .DWG or .DXF file.

3. Note the new tab in your document, with the same name as the file you imported.

**Selecting templates**

A template must be specified at the creation of the drawing, and determines the drawing's starting system variable values, sheet size, border entities, units, standards and other properties.

Onshape owns and provides publicly available templates with names like Onshape ANSI Drawings Templates, Onshape ISO Drawing Templates, and so on. To view the Onshape templates, select the Onshape filter.
You are also able to create your own custom templates. You are also able to specify whether to create four standard views or begin with no views. To change the background color of a drawing, access the setting under Preferences.

To narrow your search for the desired template, you can use:

- **Filters** - On the left of the dialog, select a filter (which are similar to the filters found of the Documents page). Select a filter to either narrow the list of templates or order the list:
  - **Recently opened** - List the templates in the order of most recently used
  - **Onshape** - List templates provided by Onshape
  - **My templates** - Display all templates created by you or shared with you by another user (select sub-filter Created by me or Shared with me); these include any .DWT files in documents for which you have read permission.
  - **Public** - Display all templates made public by other Onshape users
  - **Teams and companies** - If you belong to a team or company, those names appear in this list; click a name to see available templates
  - **Standards** - Across the top/right of the dialog, select a standards acronym to reduce the list of templates to only those of the selected standards format:
• **All** - List templates in all supported standards

• **ANSI** - List only ANSI standard templates (American National Standards Institute). Note that these templates configure the drawing for third angle view projection with inch or millimeter dimension units.

• **ISO** - List only ISO standard templates (International Organization for Standardization). Note that these templates configure the drawing for first angle view projection and millimeter dimension units.

Only templates of the selected standard and in the selected filter are displayed.

**What's next**

Once you have a drawing (empty or with default views), you are then able to:

1. Add more views, see "Views" on page 1807.

2. Add dimensions, see "Dimensions" on page 1853.

3. Add notes, see "Note" on page 1895.

4. If the underlying geometry changes, you might want to update the drawing, see "Updating a Drawing" on page 1959.


6. Add more sheets, see "Sheets" on page 1763.

---

**Custom Drawing Templates**

This functionality is currently available only on Onshape's browser platform.

In addition to allowing the creation of custom templates from scratch, Onshape also provides a number of public drawing templates for you to use and customize. These templates are typical of what most users would need and may be sufficient used as-is by many users. To use a drawing from a different CAD package as a template in Onshape, see "Using traditional CAD drawings as templates in Onshape" on page 1759.
You have the ability to think of a drawing template as being comprised of the format of a drawing that includes the title block, border, and zones, if desired, as well as the properties that describe the function of the drawing that are found under the drawing Properties panel. A drawing template is nothing more than formatting the desired appearance and defining the drawing properties and behavior, to be available for reuse for every detailed drawing.

For best practices when creating native Onshape drawing templates, see best-practices-for-creating-native-onshape-drawing-templates in the Learning Center.

Customizing a public template

If you need a custom drawing template, to enforce company drawing formats, follow this procedure:

1. Sign in to your Onshape account.
2. On the Documents page, select the Public filter.
3. Type either Onshape ANSI Drawing Templates or Onshape ISO Drawing Templates in the Search box located at the top of the page.
4. The search results will include at least 2 documents owned by Onshape and containing drawing templates:
   You are able to use these links to find the templates as well:
   - Custom ISO templates document (opens in new tab)
   - Custom ANSI template (opens in new tab)
5. Open the document containing the template you want to customize.
6. Once in the document, right-click on the tab containing the template you want to customize and choose Download.
   You now have a file named something like "ANSI_A.dwt" on your local drive.
7. Edit that file with another editor (AutoCAD, Ares, or some other DWG editor) to make changes.
   For example, you could add your company logo or alter the title block (in vector form).
Note while editing:

- There are 2 sheets in the DWT file - one for the first sheet of a drawing and a second sheet for continuation sheets in your drawing. You may need to edit both sheets.
- The template contains many settings that are helpful when creating Onshape drawings. You'll generally see better behavior if you avoid removing items from the template and instead just modify, add, or move items in the template. For example, it's fine to add additional text and areas to the title block.

8. When finished editing the DWT file, save it to your local drive with the current name or another name and be sure it still has the file extension .dwt.

Onshape uses the names of tabs when searching for templates. So if your template has "ANSI" or "ISO" in its tab name, it will be found when the user clicks on the ANSI or ISO filter in the drawing creation dialog.

9. To access your newly created custom template, create or open a new document in Onshape. (You will use this document to hold your custom templates.)

10. Click on the menu in the lower left corner of the Onshape window and choose Import to import the DWT file you just saved:

![](import.png)

This creates a new template tab in the document.

At this point, the next time you create a drawing, when you click on My templates or Created by me, you will see that template tab listed and you are able to choose it as the template for your new drawing.
Creating a custom template

As soon as you begin creating a drawing of a part in Onshape, you have the choice to select an existing template, or to create a custom template:

1. Select Create a drawing from the part's context menu in a Part Studio to access the Create drawing dialog:

![Create Drawing Dialog](image)

Copyright © 2017, Onshape. All rights reserved.
2. At the top of the dialog, select Custom template (shown above, to the left of the blue arrow) to access the *Custom template dialog*:

3. Design your template:
   a. **Standard** - ANSI or ISO
   b. **Size** - Choices are presented based on the Standard selected
   c. **Units** - Inches, Millimeters, or Feet and inches (defaults are by standard, but you can choose whatever you want)
   d. **Decimal separator** - Period or Comma (defaults are by standard, but you can choose whatever you want)
   e. **Projection** - Third angle or First angle (defaults are by standard, but you can choose whatever you want)
   f. **Border** - Include a border, or create the drawing without a border at all.
   g. **Horizontal zones** - Specify the number of horizontal zones in the border.
   h. **Vertical zones** - Specify the number of vertical zones in the border.
   i. **Start zones** - Specify in which corner of the drawing to begin labeling the zones.
   j. **Titleblock** - Include a title block, or do not include a titleblock (you can still create your own titleblock once in drawing mode)

4. Select whether to automatically include 4 standard views or leave the drawing empty (no views)
5. Click OK (or Cancel).

Exporting a drawing to a template

When you have a drawing edited the way you want it, you are able to use it to create a template for future drawings.

When exporting a drawing as a DWT file, keep in mind that:

- Only the first and, if present, second sheets are exported to the DWT file. No other sheets are exported.
- All views are deleted, and only non-view geometry and text remain.
- The format will be DWT; the version will be 2013, the primary template will be the first drawing sheet and if present, the second sheet becomes the continuation template.

Steps

1. Right-click the Drawing tab and select Export to open the Export drawing dialog:
1. To create a drawing template, select DWT as the format (as shown above).

2. Click **Export**.

To use the template:

1. In a document, use the + and select Import.

2. Select the drawing template.

3. When creating a drawing, select the Created by me filter.

4. Select the template you created.

5. Click OK.

**Using traditional CAD drawings as templates in Onshape**

Create an Onshape drawing template as described in the instructions above. Then, in...
Onshape, create an empty drawing:

1. At the template step, select Custom Template.
2. Select Do not include the borders.
3. Select Do not include the title block.
4. Set size, standard, and other characteristics as needed.
5. Click OK and close the open dialogs in the drawing.
6. Import your exported traditional CAD drawing (in DWG/DXF format) into Onshape (through the menu > Import).
7. Open the empty drawing you created.
8. Use Insert DWG/DXF tool to insert the file just uploaded.
9. To tweak the fonts, select the text and change the font to an internally supported font of your choice.
   a. To add Revision Table or Block functions:
      i. Generate a drawing using the newly uploaded DWG/DXF.
      ii. Place a table where needed (for example, the upper right corner).
   b. To include a company logo in the drawing template:
      i. Upload the logo through the Insert Image command.
      ii. Insert the newly uploaded image, properly place and size it in the drawing.
   c. Insert drawing fields as necessary in the appropriate areas of your drawing:
      i. Add a note to the drawing.
      ii. Select the Insert field button.
   iii. Use the Move to command on the View’s context menu (right-click) to add elements to: Title block, Border frame, and Border zones.

Elements will be added to the corresponding layer: title block, border frame, or border zones.

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10. Right-click the drawing tab and select Export.
11. Select a Format of DWT.
12. Select the Option of Store file in a new tab.
13. Click Export.
14. When selecting a template for a future drawing, this template will appear in the Created by me and Shared with me filters in the drawing creation dialog.
15. Test the drawing template before using.

---

**Measure Tool**

Obtain a measurement between points, lines, arc and circles in a drawing.

This functionality is currently available only on Onshape’s browser platform.

**Steps**

1. Click (in the bottom right corner of the drawing) to open the Measure dialog in a drawing.
2. Select the entity to measure, or multiple entities to measure between.

3. Select the unit of measure from the dropdown: the document's default units are indicated by (Drawing). This is especially useful for users with view-only permission, to obtain measurements in the units necessary.

4. Optionally elect to show dual units, if desired. A second measurement appears beside the first measurement:
5. The measurements of the projected geometry are displayed in the dialog box, and axes are indicated.

6. The type of relation is shown, in the above example it is point-to-point.

7. Click the red x to close the dialog.

**Sheets**

This functionality is currently available only on Onshape's browser platform.

Shortcut: Ctrl-s

An Onshape sheet is a page of a drawing which represents a single sheet of paper in a printed version of a drawing. View the sheets of a drawing and their contents with the Sheet flyout, located on the left side of the interface.

Once the sheet flyout is opened, it remains open with the currently displayed sheet selected in the list. To view another sheet, select it in the flyout, or use "Drawings" on page 1739.
Sheets shortcuts

<table>
<thead>
<tr>
<th>Shortcut</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl-s</td>
<td>Open and close Sheet flyout</td>
</tr>
<tr>
<td>PgDn</td>
<td>Display next sheet</td>
</tr>
<tr>
<td>PgUp</td>
<td>Display previous sheet</td>
</tr>
<tr>
<td>Home</td>
<td>Display first sheet</td>
</tr>
<tr>
<td>End</td>
<td>Display last sheet</td>
</tr>
</tbody>
</table>

Viewing and adding sheets

Onshape drawing templates contain multiple sheets: the main sheet and continuation sheets. The main sheet is displayed when the drawing is created. The currently displayed sheet name is located to the upper left of the drawing space.

To view more sheets:
1. Click to open the Sheets flyout:

![Sheets flyout image]

2. Double-click the sheet you want to view; the currently displayed sheet is highlighted in blue with a vertical blue bar to the left. (You can also select multiple sheets, indicated by blue highlighting.)

![Sheets selection image]
For sheets with parts involved in a release process, the part number and state are displayed next to the part name.

To create sheets:

1. Click 📊 in the upper right corner of the dialog.

When adding sheets, the new sheet is added directly after the highlighted sheet in the flyout and is immediately displayed in the drawing area.

Viewing sheets and references

In the Sheet flyout, you can sort by Sheets with the referenced entities and views listed below, or by Referenced entities with the sheets and views listed below.

Click Sort by reference to list out each entity referenced by a sheet with the sheets they are referenced by listed below.

Sort by reference with references collapsed in the tree
Referenced entities are the major category, with sheets they are referenced on listed below with views on each sheet also listed.

Click Sort by sheet to list out each sheet and the entities referenced on each listed below.

Sort by sheet with sheets collapsed in the tree.
Hovering over a reference of a configured part displays a tooltip of the configuration selection:

Sheets are the major category, with each entity referenced on the sheet listed for each sheet, with the views listed below.
You can act on referenced entities in either list, right-click on the entity:

- **Switch to reference** - Opens the Part Studio or Assembly of the referenced entity.
- **Replace** - Replace the entity on every sheet on which it is referenced with another entity of your choosing. This action opens the Insert view dialog. Note that the entity is replaced on all sheets on which it is referenced.
- **Change configuration** - Opens the Change configuration dialog. Select an option from the Configuration dropdown to change the configuration of the selected entity.
  - If you start the command from a parent-child set in the Sort by sheets category, only the configuration for that parent-child set of views will change.
  - If you start the command from the Sort by reference category, the configuration for all views on the drawing of that reference will change.

**Deleting sheets**

To delete a sheet:

1. Right-click the sheet in the Sheet flyout.
2. Select Delete.
Deleted sheets are able to be restored by the Undo command or by restoring a workspace at a point in time before the sheet was deleted.

**Reordering sheets and views**

When you have more than one sheet in a drawing, you have the ability to reorder them by dragging and dropping them in the Sheets flyout or by selecting Move up or Move down in the context menu.

Views can be selected and dragged to another sheet within the Sheet flyout. Just click to select the views. (To unselect, click the selected view again.) Child views do not move automatically with their parent view, select each view you wish to move. You can use the context menu to initiate a move as well. Select the view or views to move, right-click the view and select Switch to sheet, then select the target sheet.

**Renaming sheets**

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Renaming a sheet only renames the sheet in the sheet flyout. This does not affect the title of the sheet as specified in the Title block of a sheet.

1. Click to open the Sheet flyout.
2. Right-click the sheet you want to rename.
3. Select Rename and provide a new name (the name of the sheet is automatically selected so you can just enter the new name).

With a single sheet selected in the Sheet list, you can use Shift+N to open the Rename dialog.

Sheet properties
You can access the active sheet's properties by right-clicking in empty drawing space and selecting Sheet properties:

Or, right-click on the sheet name in the Sheet flyout and select Properties:

Once a sheet is displayed in the drawing, you can double-click the name in the flyout to open the Properties dialog for that sheet.

Edit settings of the sheet:
1. Right-click the sheet name in the Sheet flyout and select Properties.

The Sheet properties modal window opens:

```
+----------+-------+----------+
<table>
<thead>
<tr>
<th>Scale</th>
<th>Format</th>
<th>Size</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Custom</td>
<td>A</td>
</tr>
<tr>
<td>Reference</td>
<td>Assembly 1</td>
<td></td>
</tr>
</tbody>
</table>
```

2. Make changes as desired:
   a. Indicate the scale for the sheet. This is reflected in the title block and in the View properties.
   b. Specify the paper size. Size is a field linked to the Title block and can be selected from the Fields dropdown in the Notes panel.

       To set a custom paper size select Custom from the drop down and specify a width and height.

c. Indicate whether or not to include a border.

d. Indicate the number of horizontal zones.

e. Indicate the number of vertical zones.

f. Select the location of the start zones: bottom right or top left.

g. View the references for that sheet. (All names of parts, Part Studios, and Assemblies used in views on the sheet are listed here.) Click the link icon to switch to the tab or open the document containing the part/assembly.

3. Click OK or Cancel.

**Changing the format of a sheet**

You can swap the drawing format of any sheet in a drawing, at any time, as long as you have a DWT file accessible to you. This can be done on a per sheet basis, if
desired.

1. While in the drawing, on the sheet whose format you want to change, right-click on the drawing and select Sheet properties. (Or with the Sheets flyout open, right-click the name of the sheet and select Properties.)

2. The Sheet properties dialog opens:

   ![Sheet properties dialog](image)

   - Scale
   - Format: Custom
   - Size: A
   - Reference: Assembly 1

3. Click the Select a DWT icon.

4. Use the dialog to find the document and DWT you are looking for:

   ![Select a DWT dialog](image)

5. Select the document that contains the DWT file; then select the file itself.
Once you select the DWT file, the sheet format is applied to the currently displayed sheet in your drawing. At this point you can adjust properties like Scale and Reference in the Sheet properties dialog. When satisfied with your specifications, click to accept and close the dialog.

**Editing title blocks**

All of the Onshape-supplied drawings templates have the following automatic referencing between the drawing’s properties and the title block:

- Property: Nickname = Title block: Drawn, Name
- Property: Created date = Title block: Drawn, Date
- Property: Part Number = Title block: DWG No
- Property: Description = Title block: Title
- Property: Revision = Title block: Rev
- Sheet number - Automatically displayed and updated in the title block
- Total number of sheets - Automatically displayed and updated in the title block

**Tips**

- To access the properties of a drawing, right-click on the Drawing tab and select Properties. Edits made in this Properties panel are automatically reflected in the title block of the drawing.
- You have the ability to edit the fields in the title block as you normally would: drag, copy, and paste. You also have the ability to double-click a field to edit the formatting via the Note panel.
- When the drawing’s properties haven’t been specified, the title block contains dashes in place of information. These dashes will print if you print the drawing. To remove the dashes, just select and delete that note in the title block.
You can copy and paste drawing entities (like notes and symbols) across documents, workspaces within a document or across documents, and also sheets within a drawing.

**Properties**
RMB in drawing and select 'Drawing properties'

This functionality is currently available only on browser.

The icon to access the Drawing Properties panel is located on the right side of the drawings area. Click the icon to open the panel, click again to close the panel. Use these settings to define the defaults for your drawing. Individual drawing entities may be changed separately, but these settings apply to all entities in a drawing as a default. Once you have your drawing properties set to your satisfaction, you are able to save the drawing and its settings as a template. For information on creating a template, see "This functionality is currently available only on Onshape’s browser platform." on page 1753.

The illustration below shows the icon, located partway down the right side of the drawing window, outlined in blue:
The Properties panel has multiple tabs, each explained separately below.

**Units and precision**
Edit the units and precision of a drawing by clicking the first icon, labeled Units and precision, at the top of the Drawing properties panel.

**Primary**

Define the default primary units for the drawing. Edit the format of:

- **Units** - Default units for the drawing. Choose between: Inches, Inches fractional, Millimeters, and Feet and inches.
- **Decimal separator** - Default character to use a decimal separator. Choose between: Comma and Period.

- **Precision** - The precision of all numbers except those listed in the Properties flyout. Choose between: Zero and up to 6 decimal places, or between 0 and 1/256 for fractional numbers.

- **Tolerance precision** - The precision of geometric tolerances. Choose between: Zero and up to 6 decimal places, or between 0 and 1/256 for fractional numbers.

- **Angular precision** - The precision of angular measurements. Choose between: Zero and up to 6 decimal places.

- **Angular tolerance precision** - The precision of the tolerance for an angle. Choose between: Zero and up to 6 decimal places.

**Dual**

Specify whether or not to show dual dimensions and/or dual units in the drawing. You have the ability to specify the location of the dimension in reference to the view, as well as unit type, precision of decimal places, and the tolerance precision.
Check **Show dual dimensions** to display all drawings dimensions in the default document units as well as a second, specified unit. Specify where to display the second dimension, either on top of, or on the bottom of the default units:

![Dual Dimension Example](image)

Check **Show dual unit** to display the units of the dual dimension; otherwise, just the dimension is shown, without the unit of measurement.

- **Dimension location** - Specify where to place the dual dimension: on the top or bottom of the initial dimension
- **Hole callout location** - Select where to place the hole callout dual dimension: top, bottom, left or right of the initial dimension.
- **Units** - Specify the units for the dual dimension: this list will contain all units of measure except the units specified for the initial dimension label.
- **Precision** - The precision of the dual dimension. Choose between: Zero and up to 6 decimal places, or between 0 and 1/256 for fractional numbers.
**Tolerance precision** - The precision of geometric tolerances. Choose between:
Zero and up to 6 decimal places, or between 0 and 1/256 for fractional numbers.

Drawing with *Show dual dimensions* selected, *Show dual unit* selected with *Units* set to 'Inches', and *Precision* set to '0.12' (two decimal places)

**Leading and trailing zeros**
Use these settings to format the leading and trailing zeros in lengths.

- **Length leading zeros** - Select to include leading zeros on lengths.
- **Length trailing zeros** - Select to include trailing zeros on lengths.
- **Length tolerance leading zeros** - Select to include leading zeros for tolerances.
- **Length tolerance trailing zeros** - Select to include trailing zeros for tolerances.
Drawing with *length trailing zeros* set on

**Dimensions**

To access the Dimensions menu, click the second icon in the Drawing Properties panel icon bar.

Use this tab to edit your drawing’s font, text height, color, text alignment, text gaps, geometry gaps, and more. Change each option by clicking the drop down menu arrows on the right side of the menu and selecting an option.
Font - Select the font of choice for dimension text.

Text height - Select the size of choice for dimension text.

Color - To edit colors for any drawing entity, click the color block in the Drawing properties panel to access the color dialog:

- Select Palette to choose a color or enter a hex or RGB codes. Use the Mixer panel to drag to a general color area and then enter a specific hex or RGB code.

- On either the Palette or Mixer panels you are able to click the plus sign under Custom colors to save the currently specified color value as a custom color for later use.

- Arrowhead length - Change the size of arrowheads for dimensions. Arrowhead length can be any value between 0.004 inches and 393.7 inches.

- Arrowhead type - Select no arrowhead, a filled or unfilled arrowhead, or oblique marks.
Drawing with the **Color** of dimensions set to red, the **Font** changed to APMONO, the **Text height** increased to 4.5, and the **Arrowhead length** set to 4.5000. Second image illustrates ‘Oblique’ selected as the Arrowhead type.

**Gaps and extensions**

Use Text alignment, under Gaps and extensions, to specify whether to align the dimension text with either the dimension line, or align it horizontally along the bottom edge of the drawing sheet:

- **Align with dimension line** - Aligns dimension text with the dimension line, whether vertical or horizontal.
- **Horizontal** - Aligns all dimension text with the horizontal bottom edge of the drawing sheet.
Chamfer dimension

For information on using the tool, see Dimensions, Chamfer dimension.

Use these properties to style the chamfer dimensions on the drawing. You are able to specify:

- 45 degree style - Select either a Note or a Dimension to appear
- 45 degree content - Angle x Length; Length x Angle; Length x Length
- Non 45 degree style - Dimension; Note
- Non 45 degree content - Angle x Length; Length x Angle; Length x Length
- Length precision - Number of digits in the length value
- Angle precision - Number of digits in the angle value
Annotations

To access the properties for Annotations, click the third icon at the top of the Drawing properties panel. From there you are able to edit the font, text height, arrowhead length and type, and color of each annotation type in your drawing: You have the ability to supply defaults for all annotations at once, or Notes, Callouts, Datums, Geometric tolerances, Surface finish symbols, Weld symbols, Hole callouts, and Bend notes (for sheet metal) separately.
### Drawing properties

#### All annotations
- **Arrowhead**: 0.1800 in

#### Notes
- **Font**: Noto Sans
- **Text height**: 0.1200 in
- **Color**

#### Callouts
- **Font**: Noto Sans
- **Text height**: 0.1200 in
- **Color**

#### Datums
- **Font**: Noto Sans
- **Text height**: 0.1200 in
- **Datum size**: 0.1200 in
- **Color**

#### Geometric tolerances
- **Font**: Noto Sans
- **Text height**: 0.1200 in
- **Color**

---

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The properties for each drawing annotation entity are similar and explained below. Any properties specific to an annotation type are explained under that subheading, following this section.

- **Font** - Select the font of choice for each annotation type.
- **Text height** - Select the size of choice for each annotation type.
- **Color** - To edit colors for any annotation entity, click the corresponding color block to access the color dialog:

![Color Dialog](image)

- Select Palette to choose a color or enter a hex or RGB codes. Use the Mixer panel to drag to a general color area and then enter a specific hex or RGB code.
- On either the Palette or Mixer panels you are able to click the plus sign under **Custom colors** to save the currently specified color value as a custom color for later use.

**Notes**

Specify the defaults for all note properties, including Font, Text height, and Color.
Note that all exported drawings will, by default, no longer contain Drawing and Sheet Property placeholders ("----").

**Callouts**

Specify the defaults for all callouts, placed in space or attached to geometry, including Font, Text height, and Color.

<table>
<thead>
<tr>
<th>Item No.</th>
<th>QTY</th>
<th>Part number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*The callouts in the drawing correspond to the Item No. column in the BOM*

**Datums**

Specify the defaults for all datum properties, including Font, Text height, Datum size (the size of the triangle), and Color.

**Geometric tolerances**

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Specify the defaults for all geometric tolerances, including Font, Text height, and Color.

**Surface finish symbols**

Specify the defaults for all surface finish symbols, including Font, Text height, and Color.

**Weld symbols**

Change the standard for the Weld symbol at the bottom of the Annotations menu. This menu changes the default standard for the Weld symbols used in the drawing. This standard is separate from the standard of the drawing and does not change that. For example, for an ANSI standard drawing you could change this setting and use all ISO Weld symbols in your drawing.
**Bend notes**

Change the formatting for bend notes attached to sheet metal views, including font, text height and color. Bend notes come into the drawing from the bend specifications made in the Part Studio:

![Bend notes example]

**Hole callouts**

Change the formatting for hole callouts attached to sheet metal views, including font, text height and color. Hole callouts are applied to a drawing of a sheet metal flat pattern through the Dimensions dropdown menu, Hole callout tool 📏.

![Hole callouts example]
Views

To access the Views menu, click the fourth icon at the top of the Drawing properties panel. In this menu, you are able to edit your drawing's projection angle, the color and thickness of your drawing's visible edges, tangent edges, hidden edges, cutting lines, arrowheads, view labels and more.
The properties for each drawing view are similar and explained below. Any properties specific to a view type are explained under that subheading, following this section.

- **Arrowhead: size and type** - Select the size and type of arrowhead to use, choose from: filled or unfilled

- **Labels: View, Cutting line, and Detail circle** - Select the font, size, and font treatment (apply bold and/or italic), and color for each label type

**Color** - To edit colors for any annotation entity, click the corresponding color block to access the color dialog:

![Color Dialog](image)

- Select **Palette** to choose a color or enter a hex or RGB codes. Use the **Mixer** panel to drag to a general color area and then enter a specific hex or RGB code.

- On either the Palette or Mixer panels you are able to click the plus sign under **Custom colors** to save the currently specified color value as a custom color for later use.

**All views**

Specify defaults here for all view types: Section views, Detail views, Break views, and Flat pattern views:

- **Projection angle** - Specify which projection to use

- **Visible edges** - Specify the thickness and color of lines representing visible edges
• **Tangent edges** - Specify the thickness and color of lines representing tangent edges

• **Hidden edges** - Specify the thickness and color of lines representing hidden edges

• **View labels** - Specify the font, size, font treatment and color of all view labels

• **Exploded lines** - Specify the width and color of the exploded lines of the exploded view

**Section views**

Specify defaults here for:

• **Hatches** - Specify the default line size and color for hatch marks

• **Cutting lines** - Specify the default line size and color for cutting lines

• **Arrowhead** - Specify the default size and arrowhead type (filled or unfilled)

• **Cutting line label** - Specify the font, size, font treatment and color of all cutting line labels

In the illustration above, the hatches of the section view were changed to a gold color (which is propagated to the detail view as well) and the arrowhead on the cutting line was increased in size

**Detail views**

Specify details here for:

• **Detail circle lines** - Specify the default line size and the color of the detail circle lines

• **Arrowhead** - Specify the default size and arrowhead type (filled or unfilled)
**Detail circle label** - Specify the font, size, font treatment and color of all detail circle labels

*In the illustration above, the detail circle line was changed from 0.13mm to 0.50mm and the detail circle label was changed from black to teal*

**Break views**

Specify the default line thickness and color for break lines.

*In the illustration above, the break line thickness is 0.50mm and the color is blue*

**Flat pattern views**

For drawings of sheet metal flat patterns, specify the default line thickness, of all bend lines and the colors for up bend lines and down bend lines.
In the illustration above, all bend lines have a value of 0.50mm and the up bend lines have a red color specified.

Construction geometry

To access the Construction geometry menu, click the fifth icon at the top of the Drawing properties panel. In this menu, you are able to edit the color and thickness of your drawing's centerlines, as well as the style, color, and thickness of the drawing's virtual sharps. Use the Part Studio sketches section to edit the format of sketches brought into the drawing.

- **Created in drawing** - Specify defaults for construction geometry you create in the drawing:
  - **Centerlines** - Specify the default line size and color.
  - **Centermarks** - Specify the default line size and color.
- **Virtual sharps** - Specify the visual treatment for all virtual sharps as Centermark or Edge extension, shown below, respectively:

![Virtual sharps example](image)

- **Lines and splines** - Specify the default line size and color for all lines and splines added to the drawing.

- **Part Studio sketches** - Specify defaults for the geometry brought in as a Part Studio sketch.

![Part Studio sketch example](image)

*The illustration above shows the selections made in the dialog for construction geometry; the image to the right shows the virtual sharps are purple and edge extensions.*

**Formats**

To access the Formats menu, click the sixth icon at the top of the Drawing properties panel. This menu enables / disables selection of objects in a drawing's bor-
der frame, border zone, and title block. The default for a new drawing is Locked; you are not able to immediately select objects in the title block, border, or border zones.

Drawings should have this property initially set to unlocked; so the user is able to select objects in the titleblock or border.

If field values change, the content of the titleblock updates when the title block is locked.

If you move an entity to the titleblock, border, or border zone, and the Format is locked, you must move the entity back to Drawings in order to edit it.

- **Locked** - Lock or unlock the title block to disallow editing or allow editing.
- **Border frame** - Specify the color and thickness of the drawing's inner border
- **Border zone** - Specify the color and thickness of the labels inside the drawing's border zone
- **Title block** - Specify the color of lines and content of the title block, and the thickness of the lines
In the title block and border, entities in gray inherit their values from document data or properties. However, right-click on an entity to access the context menu for that entity. Select **Edit note** to edit the entity. Hover over a field (or click on it) to see what the corresponding property is. You have the ability to delete the field (property) and insert your own text if you wish. Use the **Insert drawing property** icon in the Note edit box to insert a different property:

![Illustration of a title block being edited](image1)

*This illustration shows a field in the title block being edited (select Edit note from the context menu), and a new field being inserted via the Insert drawing property command.*

### Tables

To access the Tables menu, click the last icon at the top of the Drawing properties panel. You are able to make changes for Bill of Material (BOM) tables as well as all other tables, like Hole tables, you create in your drawing.
### Drawing properties

**BOM tables**
- **Line thickness**: 0.13 mm
- **Font**: Noto Sans
- **Header row text**: 0.1200 in, Bold, Italic
- **Content row text**: 0.1200 in, Bold, Italic

**Hole tables**
- **Line thickness**: 0.13 mm
- **Font**: Noto Sans
- **Header row text**: 0.1200 in, Bold, Italic
- **Content row text**: 0.1200 in, Bold, Italic
- **Indicator style**: ANSI
- **Indicator arrowhead**: 0.1800 in
- **Indicator lines**: 0.15 mm
- **Tags**: Noto Sans
- **Line thickness**: 0.1200 in, Bold, Italic

**Revision tables**
- **Line thickness**: 0.13 mm
- **Font**: Noto Sans
- **Title row text**: 0.1500 in, Bold, Italic
- **Header row text**: 0.1200 in, Bold, Italic
- **Content row text**: 0.1200 in, Bold, Italic
- **Revision callouts**: Triangle, 1 Character
- **Font**: Noto Sans
- **Line thickness**: 0.1200 in, Bold, Italic

**Tables**
- **Line thickness**: 0.13 mm
- **Font**: Noto Sans
- **Title row text**: 0.1500 in, Bold, Italic
- **Header row text**: 0.1200 in, Bold, Italic
- **Content row text**: 0.1200 in, Bold, Italic

*Update properties from a template...*
BOM tables - Specify default values for Onshape bill of material tables, including:

- Line thickness - Specify the default thickness of the BOM table lines.
- Font - Specify the default font type for BOM tables inserted in the drawing.
- Header row text - Specify the default font size, font treatment, and color for the header row.
- Content row text - Specify the default font size, font treatment, and color for the content rows.

Hole tables - Specify default values for Onshape hole tables, including:

- Line thickness - Specify the default thickness of the hole table lines.
- Font - Specify the default font type for hole tables inserted in the drawing.
- Header row text - Specify the default font size, font treatment, and color for the header row.
- Content row text - Specify the default font size, font treatment, and color for the content rows.
- Indicator style - Select a standard for the hole indicator: ANSI or ISO.
- Indicator arrowhead - Enter or select a size for the indicator arrowhead.
- Indicator lines - Select a thickness and a color for the lines.
- Tags - Specify characteristics of the text for the hole tags: font, size, formatting (bold, italic, or none), and color.

Revision tables - Specify default values for Onshape revision tables, including:

- Line thickness - Specify the default thickness of the revision table lines.
- Font - Specify the default font type for tables created in the drawing.
- Title row text - Specify the default font size, font treatment, and color for the table title row.
- Header row text - Specify the default font size, font treatment, and color for the table header row.
- Content row text - Specify the default font size, font treatment, and color for the table content rows.
- **Revision callout** - Specify the default shape of the callout, the number of characters, and the font, size, color, and treatment.

- **Tables** - Specify default values for tables you create in the drawing, including:
  - **Line thickness** - Specify the default thickness of the lines in all tables created in the drawing.
  - **Font** - Specify the default font type for tables created in the drawing.
  - **Title row text** - Specify the default font size, font treatment, and color for the table title row (if applicable).
  - **Header row text** - Specify the default font size, font treatment, and color for the table header row (if applicable).
  - **Content row text** - Specify the default font size, font treatment, and color for the table content rows.

Edit the cells of the table separately from the table properties by clicking in the empty area of a cell to open the Cell edit panel:

![Cell edit panel](image)

Edit the text in any cell separately from the table properties by double-clicking on text in the table:

![Text edit panel](image)

**Updating and locking drawings properties**
In Onshape, you have the ability to change the properties of a drawing by importing a drawing template you have already created by using the feature at the bottom of the Drawing properties panel (shown below outlined in blue). You can also use the Lock drawing properties checkbox to prevent them from being accidentally edited.

To update properties from an existing Onshape drawing template, open the Drawing properties panel.

Click the Select a DWT icon to the right of Update properties from a template...to open the Select a DWT dialog box:
To select a DWT from the current document

1. Click at the top of the dialog.

2. Type in the *Search files* bar to find a file, or select one from the list below it.

3. To import a DWT from your desktop, click `Import...` at the bottom of the dialog box, select the file you wish to import, and click Open.

Use the three icons at the top of the dialog, under Other documents, as explained below:

- **View released items** - Click the View released items icon to list all released items for you to choose from.

- **Create version** - Click the Create version icon to open the Create version dialog box:
Adjust the name and/or description of your version, and click **Create** to create the version, or **Create version and edit properties** to open the Properties dialog:

Adjust the properties of your version, click **Apply** to apply your changes, or click **Save** to apply the changes and close the Properties dialog.

Click **Close** to close the Properties dialog without applying any changes.
**Version graph** - Click the Version graph icon to select a different version of the document from the version graph.

To select a DWT from other documents

1. Click at the top of the dialog.

2. Use the Search bar, or click one of the options below it, to select a DWT from another document in Onshape.

- **My Onshape** - Select a DWT from a document in your Onshape.
- **Recently opened** - Select a DWT from your recently opened documents.
- **Created by me** - Select a DWT from documents created by you.
- **Shared with me** - Select a DWT from documents shared with you.
- **All Company Users** - Select a DWT from documents created by all company users (when part of a Professional company account).
Release Approval team - Select a DWT from documents created by the Release approval team.

3. To import a DWT from your desktop, click Import...at the bottom of the dialog box, select the file you wish to import, and click Open.

After you import a drawing template, the properties will automatically update.

Views

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

When you create a drawing from a part, curve, surface, or subassembly, you have the ability to create it without any views, by default, or with 4 standard views: top, front, right, and isometric. Typically, the projection of the views depends on the standard chosen: first angle projection for ISO standard and third angle projection for ANSI standard, you can also use a custom template and select the projection, or change the projection after the drawing is created.

For example, a standard ANSI drawing may look like this:
All views of a part, curve, or surface in a drawing are from the same version of the part. When creating a view (drawing, projected, auxiliary, section) the part version used is the same as for all existing views.

Views are placed on sheets and can have relationships with other views.

This table illustrates the types of views and which can be created from which:

### Projected View

<table>
<thead>
<tr>
<th>Can be created from:</th>
<th>Cannot be created from:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base views</td>
<td>Auxiliary views</td>
</tr>
<tr>
<td>Projected views</td>
<td>Detail views</td>
</tr>
<tr>
<td>Section views</td>
<td></td>
</tr>
</tbody>
</table>

Projected views retain settings from the view from which it is created.

### Auxiliary View

<table>
<thead>
<tr>
<th>Can be created from:</th>
<th>Cannot be created from:</th>
</tr>
</thead>
</table>

Copyright © 2017, Onshape. - 1808 -
All rights reserved.
| Linear edge in base views | Section views |
| Linear edge in projected orthographic views | Details views |
| Linear edge in auxiliary views | |
| Linear edge in isometric views | |

### Section View

| Can be created from: | Cannot be created from: |
| Positions in base views | Auxiliary views |
| Positions in projected orthographic views | Detail views |
| Positions in isometric views | Cut line tangent to cylindrical face |
| Section views | |

### Broken-out Section View

| Can be created from: | Cannot be created from: |
| Projected views | Section views |
| Auxiliary views | Detail views |
| Broken views | Crop views |
| Views with other broken-out sections | |

### Detail View

<p>| Can be created from: | Cannot be created from: |
| Positions in base views | Detail views |
| Positions in projected orthographic views | |
| Positions in isometric views | |
| Position in an auxiliary view | |
| Position in a section view | |</p>
<table>
<thead>
<tr>
<th>Break View</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Can be created from:</td>
<td>Cannot be created from:</td>
</tr>
<tr>
<td>Base views</td>
<td>Detail views</td>
</tr>
<tr>
<td>Project views</td>
<td></td>
</tr>
<tr>
<td>Auxiliary views</td>
<td></td>
</tr>
<tr>
<td>Section views</td>
<td></td>
</tr>
<tr>
<td>Isometric views</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Crop View</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Can be created from:</td>
<td>Cannot be created from:</td>
</tr>
<tr>
<td>Projected views</td>
<td>Section views</td>
</tr>
<tr>
<td>Auxiliary views</td>
<td>Broken-out section views</td>
</tr>
<tr>
<td></td>
<td>Detail views</td>
</tr>
<tr>
<td></td>
<td>Break views</td>
</tr>
</tbody>
</table>

**Insert view**

Place a view of the model (part, surface, assembly, sketch, curve, composite part, or flat sheet metal pattern) on the active sheet; use the dialog to select the desired part, sketch or flat sheet metal pattern, including version and orientation. By default, the label and scale are off. To see the scale, double-click the view: the View properties dialog opens to the top left of the drawing.
1. Click

2. In the dialog, either:

   a. Use the drop down to select a part, curve, surface or assembly, and then a view:

   ![Insert view dialog](image)

   The screenshot on the left shows the Insert view dialog for a Part Studio; the one on the right shows the dialog for an Assembly.

   If available, you can also select a named view, listed at the bottom of the View field (where Front appears, above).

   When inserting an Assembly that has Display states defined, the Display state drop down appears below the Assembly view drop down. Here you can select the Display state to include in the drawing.

   b. Click to search the Part Studios and Assemblies in the current (or any document) for parts, surfaces, assemblies, curves, or sketches (or versions thereof):

   i. Select to insert from the Current workspace (in Part Studios or Assemblies in this document):
By default, the selected filter is Parts, but you can also search for an entire Part Studio, Assemblies, surfaces, sketches, curves, composite parts, and sheet metal flat patterns. These filter icons appear only when the selected Part Studio or Assembly contains these types of entities.

3. Or, click **Other documents** to use the familiar filters to search for and then select from a version in a different document:
4. Once a document is selected, if it has versions, click to open the Version graph and select the version or workspace from which to select a part or assembly:

You can also select to view only released items , and to create a version.
5. Use the Scale dropdown (where 1:1 Sheet appears as the default) to select a specific scale for the view.

6. If you are inserting an assembly and have created exploded views, you can select the exploded view to insert:
   a. Select the dropdown beside the Explode/Position field in the dialog.
   b. Select the Exploded view name to insert.

![Insert view dialog]

7. Once you have the intended entity selected, click on the drawing to place the view. A preview appears as you place the view.

**Projected view**

Create a new view by projecting (folding) out an existing view. By default, the label and scale are off. To see the scale, double-click the view: the View properties dialog opens to the top left of the drawing. By default, the Projected view tool is active after a drawing is created and you place the first view. The Projected view tool remains active, even after use, until you click on the icon to turn it off.
1. Click to select an existing view.

2. Drag the cursor in different directions from the original view to see possible projected views.

3. Click to place the new view.

4. You can also double-click on the view to open the "View properties" on page 1833 dialog.

**Auxiliary view**

Create an auxiliary view; an orthographic view that is folded out 90 degrees from a selected edge in the parent view (usually from a slanted edge). By default, the label and scale are off. To see the scale, double-click the view: the View properties dialog opens to the top left of the drawing.

1. Select the edge of the part about which to orient the auxiliary view.

2. Drag the cursor to the location for the auxiliary view.
3. Click to place the view.

4. Double-click to open the "View properties" on page 1833 dialog.

Note that the View properties dialog formats all input into an N:N or N/N format. For user input values, the second digit or denominator is always set to 1, and you can double-click the Scale label to edit it. By default, the scale of an Auxiliary view is always set to Parent (the same scale as the parent view).

Selecting an Auxiliary view also highlights the edge in the parent view.

**Section view**

Create a section view, jogged section view, or partial section view of an existing view (including a section view) by placing a cutting plane line (or lines) and specifying a direction and label. Keep in mind that you are unable to create section views from auxiliary or detail views. By default, the label is on and the scale is off. To see the scale, double-click the view: the View properties dialog opens to the top left of the drawing.

1. Click .

2. Select Vertical, Horizontal, or Angular in the dialog:
3. Optionally supply a label for the view:

![Section view dialog](image)

Labels are automatically applied (you are able to change them) and by default progress from A through Y, omitting I, O, Q, S, X, and Z.

4. Choose to only show cut geometry by clicking the box next to Show cut geometry only, or control the depth of the section by clicking Section depth (shown below):

![Section view dialog with Section depth option](image)

Select whether you wish to use another entity to set the depth (and then select the entity on the drawing) or use Blind and specify a depth.

Click the section view icon in the dialog.

5. Move the cursor over the part for which to create a view. Hover to view snap points:
6. When the dotted section cutting line is in the desired place, or the snap point is visible, click once to place the line.

   For angular section views, click a second snap point to set the angle.

7. If desired, select more snap points to create a jogged section view (up to 4 points total for an angular jogged view):
8. Drag the new section view away from the cutting line and click to place it:

Vertical section view (above)

Angular section view (above)
Horizontal section view (above)

Note that dragging the section view to one side or the other before clicking it into place flips the side of the section:

Click to select the section line and use the snap point at the ‘elbow’ of the arrow to drag the line to shorten it or lengthen it:
Adjust the length of the section line to the inside of the part to create a partial section view. (You cannot create a partial section view with a jogged section view.)

9. Double-click the view to open the View properties dialog.

Selecting a Section view highlights the cut line in the parent view.

The "View properties" on page 1833 dialog formats all input into an N:N or N/N format. For user input values, the second digit or denominator is always set to 1, and
can double-click the Scale label to edit it. By default, the scale of a Section view is always set to Parent (the same scale as the parent view).

Once placed, jogged section views may be adjusted by clicking a snap point and dragging to another point on the drawing view. The jogged section view adjusts respectively.

Creating a section view of a section view

You can use the Section view to create a section view from a section view:

1. Click .
2. Select Vertical, Horizontal, or Angular in the dialog:
3. Optionally supply a label for the view.
4. If you wish to cut from the sectioned geometry (the section view already created), select Cut from section:

   ![Section view dialog]

   If you want to create the view from the base geometry, leave Cut from section unchecked.

5. Click the check mark to create the view, or the x to cancel the operation.

Moving a section line

Once a section line is placed, if it was placed on a snap point, it is possible to move it to a new placement:
1. Select the line:

2. Click and drag the snap point to a new location:

3. Click to place the section line.
4. Notice that the corresponding view changes:

![Diagram showing section lines]

**Flipping a section line**

To flip a section line after you place it:

1. Select the section line.
2. Right-click and select *Flip direction*.

The section line labels change sides and the view regenerates appropriately.

**Editing a section line**

To edit a section line with Onshape:

1. Click on the section line (jogged section line).
2. Right-click and select Edit section from the context menu.
3. A Section view dialog box appears, edit your preferences and click the checkmark in the top right corner to apply your edits. Click the red x in the top right corner to cancel without applying your edits.

**Adding or removing a segment**

1. Select the section line (jogged section line).
2. Right-click and select Edit section from the context menu.
3. To add a segment:
   a. Select a snap point and drag it to the desired location on the view.
   b. Hover over a new snap point at the desired location, click to place the line segment there.
4. To remove a segment:
   a. Hover over the snap point of the segment you want to remove. An orange box appears around the snap point.
   b. Click on the snap point with the orange box; the snap point disappears.
5. End the operation by clicking anywhere away from the view.
6. The view updates to reflect the changes in the section line.

**Broken-out section view**

Create a broken-out section view to cut away a portion of the model in a drawing view by defining a closed profile with the Spline tool.
1. Click

![Image of a button](image)

2. Use the Spline tool to define a closed profile for the view.

   Use the Spline point tool to add points with which to edit the spline.

3. Select an end type, either Blind to a specified depth, or Up to entity (and select the point or edge on another view).

4. Click

You can also double-click on the view to open the "View properties" on page 1833 dialog. Right-click and select Edit section to re-open the dialog and edit the specifications of the view (or Remove section to delete the broken-out section from the drawing).

![Diagram of a view before and after broken-out view](image)

A view before the broken-out view is applied (left), and after the broken-out view is created (right)

Note that, the depth point (defined by the End type, described above) has an association back to the model. Therefore, if the model is modified resulting in a change of
the depth point, the spline and depth point for the broken-out section will update with the update of the drawing. However in some cases the update may result in an error. The broken-out spline will be displayed in red; if a problem occurs, redefine the depth point to fix the issue.

**Detail view**

Use Detail view to select an area of an existing view to enlarge for more detail.

1. Click 📊.
2. Click in the approximate center of the area you wish to enlarge (on an existing view).
3. Drag and click again to define the circumference of the area.
4. Drag and click again to place the detail view:

Note that you have the ability to edit the scale and labels for detailed views through the "View properties" on page 1833 dialog. Selecting a Detail view highlights the detail view circle in the parent view.

**Resizing a Detail view**

To resize a Detail view:
1. Hover over the view to activate highlighting:

2. Select the grip point between the arrows

3. Drag in or out to resize the view smaller or larger.

**Moving a Detail view**

To relocate a Detail view:
1. Hover over the view to activate highlighting:

2. Select the grip point at the center of the view circle.

3. Drag to new location.

**Break view**

Use Break view to shorten an existing view by trimming out a portion.

1. Click ✂️.

2. Specify a horizontal or vertical break.
3. Specify the type of break line and gap distance:

a. Zig zag  
b. Small zig zag  
c. Curve  
d. Straight

4. Click in the view to place the two break lines (indicating where the gap occurs).

Note that you have the ability to double-click to open the "View properties" on page 1833 dialog.

To delete a break, click to select it and press Delete.

**Crop view**

To crop a view, sketch a spline around the area you wish to keep.

1. Click .

2. Use the Spline tool to define a closed profile for the view.

   Use the Spline point tool to add points with which to edit the spline.

3. Click .
Double-click the view to open the View properties dialog.

1. Use the Spline point tool to add points with which to edit the spline.

2. Click ✓.

You can also double-click on the view to open the "View properties" on the next page dialog. Right-click and select Edit crop to re-open the dialog and edit the specifications of the view (or Remove crop to delete the crop view from the drawing).

![Image showing views before and after cropping]

The view (left) before Crop is applied; the spline defining the crop boundary (middle); the resulting crop view (right)

**Deleting views**

1. Select the view to delete using any selection method.

2. Press the Delete key or right-click to activate the context menu and select Delete.

**Moving a view**

1. Select the view.

2. Drag to the desired placement.

**Moving a view to another sheet**

You can move any view to another, pre-existing sheet in your drawing through three ways: use the Move to sheet command on the context menu, select a new sheet in the Sheet dropdown in the Sheet properties dialog, and by dragging the view to another sheet in the Sheets flyout.
When a view is moved to another sheet, all related entities (labels, dimensions, etc) move with it.
When moving an auxiliary view, the parent view is also moved.
When moving a parent view, the auxiliary view is also moved.

**View properties**

Select a view, right-click and select *View properties*. Or, you can double-click a view to open the View properties dialog. For all views except Detail and Section, the View properties dialog is:

You can select a Display state (when available) and the views will update accordingly.

When multiple views with differing values for their properties are selected, the appropriate fields in the dialog display "multiple values", as shown below:
For Detail and Section views, there is an additional property: View label, explained below.

- **Document** - The name of the document the part or assembly resides in.
- **Workspace** or **Version** - The workspace name if the part or assembly is from the current document. The version name if the part or assembly is in a different document (when Part Studios or Assemblies are moved to another document, that document is automatically versioned).
- **Type** - Whether the drawing is of a part or an assembly.
- **Reference** - The name of the part or assembly of the drawing, with a link to open the referencing document and Part Studio or Assembly tab.
- **View** - To change the view from one perspective to another, select from the drop-down: Top, Left, Right, Front, Back, Bottom, or Isometric. The view on the drawing changes immediately.
- **Scale** - Set the scale of the drawing. Input is in an N:N or N/N format. For user input values, the second digit or denominator is always set to 1, and you can double-
click the Scale label to edit it. By default, the scale of a Projected view is always set to Parent (the same scale as the parent view).

- **Rotation angle** - Use this to rotate the angle of view (in default units). The arrow reverses the direction of the angle. All views, when created, have a rotation angle of 0 degrees. You can change this value only if the view has no parent (is not a 'child'), is not a parent (has no 'children'), or if the alignment with a parent is suppressed.

Valid values are between 0 and 360 degrees.

Views that may be rotated may also be "Align view vertically or horizontally" on page 1852 along a selected straight edge.

- **Tangent edges** - Select the visual treatment of tangent edges in the view:
  - **Hidden** - Tangent edges are visually removed from the drawing:
  ![Hidden Tangent Edges](image1)
  - **Solid** - Tangent edges are shown by solid lines:
  ![Solid Tangent Edges](image2)
  - **Phantom** - Tangent edges are shown by broken lines:
  ![Phantom Tangent Edges](image3)
**View render mode** - Select the type of render mode you wish: Best quality or Best performance. Drawings views default to Best performance. If, in certain cases, some edges are not displayed correctly, you should change the View render mode to Best quality. The rendering mode setting applies to both drawing views within Onshape and views in exported drawings.

**View simplification** - This feature allows users to simplify the geometry shown in the drawing by setting a threshold below which features will be hidden.

Automatic - Default. Finds the best view simplification settings based on the geometry of the part and automatically uses those settings to display the part or assembly.

Absolute - Enter a number in the length units of the drawing to indicate that any feature smaller than the value will be simplified within the view. If whole parts are smaller than the threshold value, those parts will be missing from the view. This is useful for removing excessive details not needed for drawing purposes (for example, a large number of very small features or components).

Ratio to studio - Enter a percentage of the size of the Part Studio or Assembly below which the feature will be simplified within the view. If whole parts are smaller than the threshold value, those parts will be missing from the view. This is useful for removing excessive details not needed for drawing purposes (for example, a large number of very small features or components).

Ratio to part - Enter a percentage of the size of the part below which the feature will be simplified within the view. This setting aims to preserve parts in a view, while simplifying the detail within those parts. This is useful if you intend to ensure all the parts are present to facilitate detailing actions, such as placing callouts.

All child views will receive the View simplification setting of their parent view, but any child view can be independently changed later and not affect the rest of the view settings in the parent/child dependency.

**Sheet** - The name of the current sheet shown; use the dropdown to move the view to another sheet.

**Name** - The name of the view in the format <view perspective>-<part name>. Changing the name of the view does not alter the view perspective or the part
With a single view selected in the Sheet/Views list, you are able to use Shift+N to open the Rename dialog.

- **Scale label** - Check to display the scale label below the view.

- **View label** - (For Detail and Section views only) Specify a custom prefix for the label and specify a suffix, creating a multi-line label. Changing the letter also changes the letter of the view referenced (the parent of the Detail view or the cutting line of a Section view, for instance).

- **Move to sheet** - Opens a dialog with a dropdown listing all available sheets. Select a sheet to move the currently selected view to.

- **BOM table reference** - Select the assembly the BOM table should reference.

You also have the ability to open the Part Studio or Assembly that the view is from, and specify a scale, rotation angle, and scale label.

Select Parent scale to link the view’s scale to its parent’s scale or Sheet scale to link the view’s scale to the sheet scale.

**Tips**

- The view rotates around the center of the view rectangle, which changes size as needed. For detail views, the view rotates about the center of the circle surrounding the detail view; the visible geometry stays the same and the circle stays the same size.

- When the Rotation angle is not 0 degrees, then the view properties to reconnect alignment are disabled. Similarly, the commands to reconnect alignment with the
parent are also disabled. You must change the Rotation angle to 0 degrees before the view can reconnect with the parent.

- All dimensions adjust when a Rotation angle changes. Vertical and horizontal linear dimensions remain vertical and horizontal. Aligned and rotated linear dimensions remain aligned and rotated to their view geometry.

- View scale and label location change to be centered below the new view rectangle or detail view circle.

- Use Suppress alignment with parent to remove the alignment of the view to its parent. If there are no dependencies, that is, if the view has no children, then you can use the Rotation angle field once the alignment is broken. However, if there are other issues blocking the view from being rotated (that is, if it has children), then the view cannot be rotated. Keep in mind, that if a view has children it cannot be rotated even if you suppress alignment.

- Copy a view label using Ctrl+C to copy the label and apply the default properties of Notes (from the Property panel). Use Alt+drag to copy a view label keeping the view label properties.

**Modifying views through the Context menu (RMB)**

Use the right-mouse click to open a context menu on any view to access a list of command options for that view. These command options are listed below. Note that not all the commands listed here are available for every type of view.

**Show/Hide hidden lines**

Show or hide the lines of a view that are not visible in current view position (hidden lines).

Select the view, right-click and then select *Show hidden lines* from the context menu:

![Diagram](image.png)

The resulting view:
**Show/Hide bend lines**

Show or hide the bend lines of a sheet metal flat pattern view.

Select the view, right-click and then select *Show bend lines* (or *Hide bend lines*) from the context menu:

**Tangent edges**

Select the visual treatment of tangent lines in a drawing view. Select the view, right-click and select *Tangent edges* and then select from these commands:

- **Hidden** - Tangent edges are visually removed from the drawing
- **Solid** - Tangent edges are shown by solid lines
- **Phantom** - Tangent edges are shown by broken lines

**Show/Hide shaded view**

Show (or hide) a shaded view of the parts. This command is not available for Section views or views that have breaks in them.

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Shaded view:

If a parent view is shaded, then the Detail view will also be shaded. You can change the shading independently of the parent, and also the parent independently of the child (right-click on the view).

**Show/Hide threads**

Show (or hide) threads (lines indicating threaded holes):

Select the view, right-click and select *Show threads* (or Hide threads).

**Show/Hide sketches for Part Studios**

Showing sketches comes in handy show flat pattern sketches on drawing views of those flat patterns for sheet metal.

For views of parts, this command shows or hides selected sketches from within a Part Studio. Select the view, then right-click and select *Show sketches*. When the Show/hide sketches dialog opens, select the sketch from the menu. You can select more than one sketch. This dialog displays all sketches in the Part Studio in which the part was modeled. Selecting the same drawing sketch for each view displays the sketch in each view's perspective.

To hide a sketch, open the Show/hide sketch dialog again and click to un-select the sketch or sketches.

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Show/hide sketches for Assemblies

For views of assemblies, this command shows or hides selected the sketch from within the Assembly (sketches must first be inserted into an Assembly). Select the view, then right-click and select Show sketches. Any sketches inserted into the Assembly become visible in the drawing.
To hide all displayed sketches for a view, open the context menu and select Hide sketches.

**Show/hide sketch points**

If the sketch includes specific sketch points, you can show those points in the drawing (and also hide them). To show sketch points:

1. Right-click on the view and select Show/Hide sketch points.

To customize the appearance and size of the sketch points in the drawing, use the Drawing properties panel, "Construction geometry" on page 1796 tab.

**Insert sketch**

To insert a sketch on a section view:

1. Right-click on the section view and select Show/Hide sketches...to open the Show/Hide sketches dialog box:
2. From here, click on the sketch or sketches you want to insert.

3. Click the ✔️ checkmark in the top right corner of the dialog box to insert the sketch or sketches into your section view.

4. To remove a sketch from a section view, right-click on the section view and select Show/Hide sketches...from the context menu (as shown in Step 1).

5. Click on the sketch or sketches you want to remove (notice they are no longer highlighted in blue).

6. Click the ✔️ checkmark in the top right corner of the dialog box to finish removing the sketch or sketches.

**Show/Hide offset cut lines**

Show (or hide) the offset cut lines.
Select the view, then right-click and select *Show offset cut lines.*

**Show/Hide bend notes**

Show (or hide) bend notes for flattened views of sheet metal.

Select the view, then right-click and select *Hide (or Show) bend notes.*

**Show/Hide part intersections**

Select the view, then select *Show (or hide) virtual edges* to show or hide the virtual edges (curves drawn at the places where parts intersect) where parts intersect. This command defaults to Hide for all new views to improve performance. If an assembly view with more than 20 parts does not display correctly because parts interfere with each other or portions of intersecting edges/faces are misidentified as hidden (or visible) in any view, toggle Show part intersections.

In addition to toggling the display of virtual edges (curves drawn at the places where parts intersect) this command also restores visibility of parts which have...
Display state

Select a view, then right-click and hover over Display state to display a list of available display states for the view. Select one. For all parent views, the default is 'show all.' All child views default to 'follow parent' display state. Section and Detail views will only ever 'follow parent' display state.

If you create a display state after creating the drawing, update the drawing to make use of the display state. If you delete a display state from the assembly, then upon drawing update an error message is displayed:

"Failed to resolve Display state"

Any views using the deleted display state will be empty, but remain on the drawing. The name of the deleted display state no longer appears in the context menu.

Explode/Position

Select a view, then right-click and hover over Explode/Position to display a list of available explode or position states for the view. Select one.

If you create an explode view after creating the drawing, update the drawing to make use of the explode. If you delete an explode view from the assembly, then upon drawing update an error message is displayed:

"Failed to resolve exploded view"

Note that moving the rollback bar in the Exploded tree between explode steps will not affect the drawing view. The drawing views reflect all explode steps in an Exploded view.

Suppress alignment with parent

Select a child view, right-click and select Suppress alignment with parent to disconnect the automatic alignment of views derived from other views in order to place them independently on the drawing.

When suppressing an alignment, you are not breaking the alignment to the view’s children. If the view has children (or any alignments) you will not be able to rotate the view.
Create projected view
Select a view, right-click and select *Create a projected view* (see above) from the currently selected view.

Edit section/Edit crop
Select the broken-out section or crop view, right-click and select *Edit section* (or *Edit crop*). Use the dialog that opens to edit the spline, add points if necessary to drag the spline into a new shape or change specification from depth to entity or vice versa for broken-out section.

Remove section/Remove crop
Select a section or crop view, right-click and select *Remove section* (or *Remove crop*) to remove the broken-out section or crop view and leave the spline for reference. To remove the spline, click to highlight it, then either press Delete, or right-click and select Delete.

View properties
Select a view, right-click and select *View properties*. Or, you can double-click a view to open the View properties dialog. For all views except Detail and Section, the View properties dialog is:
You can select a Display state (when available) and the views will update accordingly.

When multiple views with differing values for their properties are selected, the appropriate fields in the dialog display "multiple values", as shown below:
For Detail and Section views, there is an additional property: View label, explained below.

- **Document** - The name of the document the part or assembly resides in.

- **Workspace** or **Version** - The workspace name if the part or assembly is from the current document. The version name if the part or assembly is in a different document (when Part Studios or Assemblies are moved to another document, that document is automatically versioned).

- **Type** - Whether the drawing is of a part or an assembly.

- **Reference** - The name of the part or assembly of the drawing, with a link to open the referencing document and Part Studio or Assembly tab.

- **View** - When one view is selected, this field allows you to change the orientation of the view: Top, Left, Right, Front, Back, Bottom, Isometric.

- **Scale** - Set the scale of the drawing. Input is in an N:N or N/N format. For user input values, the second digit or denominator is always set to 1, and you can double-click the Scale label to edit it. By default, the scale of a Projected view is always set to Parent (the same scale as the parent view).
Rotation angle - Use this to rotate the angle of view (in default units). The arrow reverses the direction of the angle. All views, when created, have a rotation angle of 0 degrees. You can change this value only if the view has no parent (is not a 'child'), is not a parent (has no 'children'), or if the alignment with a parent is suppressed.

Valid values are between 0 and 360 degrees.

Views that may be rotated may also be "Align view vertically or horizontally" on page 1852 along a selected straight edge.

Tangent edges - Select the visual treatment of tangent edges in the view:

- Hidden - Tangent edges are visually removed from the drawing:

- Solid - Tangent edges are shown by solid lines:

- Phantom - Tangent edges are shown by broken lines:

View render mode - Select the type of render mode you wish: Best quality or Best performance. Drawings views default to Best performance. If, in certain cases,
some edges are not displayed correctly, you should change the View render mode to Best quality. The rendering mode setting applies to both drawing views within Onshape and views in exported drawings.

- **View simplification** - This feature allows users to simplify the geometry shown in the drawing by setting a threshold below which features will be hidden.

  Absolute - Enter a number in the length units of the drawing to indicate that any feature smaller than the value will be simplified within the view. If whole parts are smaller than the threshold value, those parts will be missing from the view. This is useful for removing excessive details not needed for drawing purposes (for example, a large number of very small features or components).

  Ratio to studio - Enter a percentage of the size of the Part Studio or Assembly below which the feature will be simplified within the view. If whole parts are smaller than the threshold value, those parts will be missing from the view. This is useful for removing excessive details not needed for drawing purposes (for example, a large number of very small features or components).

  Ratio to part - Enter a percentage of the size of the part below which the feature will be simplified within the view. This setting aims to preserve parts in a view, while simplifying the detail within those parts. This is useful if you intend to ensure all the parts are present to facilitate detailing actions, such as placing callouts.

  All child views will receive the View simplification setting of their parent view, but any child view can be independently changed later and not affect the rest of the view settings in the parent/child dependency.

- **Sheet** - The name of the current sheet shown; use the dropdown to move the view to another sheet.

- **Name** - The name of the view in the format <view perspective>-<part name>. Changing the name of the view does not alter the view perspective or the part name.

  With a single view selected in the Sheet/Views list, you are able to use Shift+N to open the Rename dialog.

- **Scale label** - Check to display the scale label below the view.
**View label** - (For Detail and Section views only) Specify a custom prefix for the label and specify a suffix, creating a multi-line label. Changing the letter also changes the letter of the view referenced (the parent of the Detail view or the cutting line of a Section view, for instance).

**Move to sheet** - Opens a dialog with a dropdown listing all available sheets. Select a sheet to move the currently selected view to.

**BOM table reference** - Select the assembly the BOM table should reference. You also have the ability to open the Part Studio or Assembly that the view is from, and specify a scale, rotation angle, and scale label.

Select Parent scale to link the view’s scale to its parent’s scale or Sheet scale to link the view’s scale to the sheet scale.

**Tips**

- The view rotates around the center of the view rectangle, which changes size as needed. For detail views, the view rotates about the center of the circle surrounding the detail view; the visible geometry stays the same and the circle stays the same size.

- When the Rotation angle is not 0 degrees, then the view properties to reconnect alignment are disabled. Similarly, the commands to reconnect alignment with the parent are also disabled. You must change the Rotation angle to 0 degrees before the view can reconnect with the parent.

- All dimensions adjust when a Rotation angle changes. Vertical and horizontal linear dimensions remain vertical and horizontal. Aligned and rotated linear dimensions remain aligned and rotated to their view geometry.
- View scale and label location change to be centered below the new view rectangle or detail view circle.
- Use Suppress alignment with parent to remove the alignment of the view to its parent. If there are no dependencies, that is, if the view has no children, then you can use the Rotation angle field once the alignment is broken. However, if there are other issues blocking the view from being rotated (that is, if it has children), then the view cannot be rotated. Keep in mind, that if a view has children it cannot be rotated even if you suppress alignment.
- Copy a view label using Ctrl+C to copy the label and apply the default properties of Notes (from the Property panel). Use Alt+drag to copy a view label keeping the view label properties.

**View orientation**
Change the orientation of a selected view. Right-click on the view and select View orientation from the context menu. The current view orientation will be grayed out in the menu:
- Top
- Left
- Right
- Front
- Back
- Bottom
- Isometric

These commands are also available through the "View properties" on page 1833 dialog as well.

**Align view vertically or horizontally**
For rotated views, you can elect to align them vertically or horizontally, according to a selected straight edge. The commands are found in the view context menu:
1. Select a view.
2. Right-click to activate the context menu.
3. Select **Align view vertically** or **Align view horizontally**.

   The cursor changes to a selection cursor.

4. Select a straight edge with which to align the view.

   The view is aligned and the command mode is exited.

---

**Dimensions**

Shortcut: d

This functionality is currently available only on browser.

When defining dimensions for a drawing, you will notice that orange snap points appear when you hover over a line or point. There are 4 types of snap points:

- Square snap points indicate end points
- Triangle snap points indicate midpoints
- Diamond snap points on a quad point of a circle or arc indicate one of the quadrants of the circle
- Circle snap points indicate an arc or circle's center; once a dimension snap point exists on a circle or arc's center, you can click and drag the point to a quadrant point.
Midpoints and quad points are disabled during dimensioning for ease of selecting appropriate dimension points. However, after a dimension has been placed, editing the dimension provides access to these midpoints and quad points. Use keyboard shortcut Shift-q to quickly toggle on midpoints and quad points for the current command. Shift-q again to toggle them off.

Once the snap point is visible, the point has been snapped to and you are able to click. There is no need to click directly on the point once it is visible. While moving the mouse to place the dimension, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the dimension to; simply click when you see the line appear to align the dimension to that line.

You can dimension to hidden lines (after using the command “Show hidden lines”).

Editing the value of a dimension causes it to be converted to an Overridden dimension. See "Troubleshooting dimensions" on page 1877.

Once a dimension is created, hover over it to see which entities are involved in the dimension. The entities turn blue upon hover:
You are able to edit grip points of an existing dimension, if necessary. Click and drag any grip point to another edge, point, arc, circle, or circle center. Associations are maintained on other grip points. For example, in the illustrations below, the right grip point of the dimension is dragged from the point to the edge:
You are able to drag the dimension text simply by clicking and dragging. There is no need for a grip point on the text.

When an edge is selected, click again (even in the middle of dimensioning) to deselect it and select a different edge.

**Dimension**

Shortcut: d

The Dimension tool for drawings work much like the Dimension tool for sketches. Activate the tool (click the icon or use the "d" shortcut), then:

1. Click a highlighted drawing entity (circle, arc, circle center, line, or point).
2. Click a second highlighted drawing entity.
3. Click to place the dimension in the drawing.

You are able to edit a dimension after placement by clicking and dragging a grip point. As you move the cursor, drawing entities highlight to indicate you are able to dimension to them.

Below are entity-specific dimension tools. For each tool, only specific entity types are highlighted as you move the cursor.

**Chamfer dimension**

Place a note or dimensions on a chamfered edge.

1. Click .
2. Click the chamfered edge and an adjacent edge.
3. Move the cursor away from the edges to preview the dimension.
4. Click to place the note or dimensions.

To alter the style of the chamfer dimension (note or dimension), select the preferred style in the Properties panel, Dimensions tab, Chamfer section. For more information, see "Properties" on page 1775.
To include a prefix for the chamfer dimension, double-click the dimension to open the dialog:

Adjust the chamfer dimension formatting as explained in "Dimension" on the previous page.

**Maximum or minimum dimension**
Shortcut: m

Place a minimum or maximum dimension directly to any combination of arcs, circles and any other geometry type, excluding curves.

1. Click 🟢.
2. Select a combination of one arc/circle and any other geometry type, excluding curves.
3. Drag to move the dimension text to desired location.
4. Click to place the dimension.
Despite clicking on the circle itself, the min-max dimension tool gives you the maximum or minimum dimension depending on where you click the entity; for example, the maximum distance, shown above as 1.420, was acquired by clicking on the top of the top circle and the bottom of the bottom circle.

### 2 point linear dimension
Measure the distance between two points. Create horizontal, vertical, and rotated linear dimensions.

*You must select two points, you cannot select a line.*

1. Click ✿.
2. Hover over the drawing view to activate the snap points.
3. Click when you see an appropriate snap point.
4. Click when you see the second appropriate snap point.
5. Drag to place the dimension box.

### Point-to-line dimension
Measure the distance between a point and a line. Create horizontal, vertical, and
rotated linear dimensions.

You must select one point and one line.

1. Click 🖇️.
2. Hover over the drawing view to activate the snap points.
3. Click when you see an appropriate snap point.
4. Click when you see the appropriate line highlighted.
5. Drag to place the dimension box.

**Line-to-line dimension**

Create dimensions between parallel lines.

1. Click 🖇️.
2. Hover over the drawing view to activate the snap points.
3. Click the first line highlight.
4. Click the second line highlight. Note that only parallel lines will highlight for selection.
5. Drag to place the dimension box.

**Placing dimension text**

After picking two entities the dimension is drawn in a preview mode to allow final placement:

- Dragging the text around during preview allows for the text to move outside of the extension lines, and also switch between horizontal, vertical, and aligned measurement modes:
Drag the text away from the two chosen snap points up or down the drawing creates a horizontal dimension line:

Drag the text away from the two chosen snap points toward the side of the drawing creates a vertical dimension line:
• Dragging the text away in a direction perpendicular to a line through the two chosen snap points creates a dimension line parallel to the two chosen snap points:

• Horizontal and vertical "projected" snaps are also available during text placement. This allows for lining dimensions up with existing text/dimensions and other locations on the drawing:

   Hover over a marker to wake up alignment. This is available in Preview mode only. Pass over other drawing entities to wake up alignment as well, like other views’ entities.
Center marks on circular edges

When the dimension tool is selected, you are able to move the cursor over an edge representing a circular edge to 'wake up' the center mark. Once visible, this mark remains visible.

Upon moving the cursor over an edge, an orange circular snap point appears, with the vertical center marker:
After hover, the orange snap point disappears but the marker remains:

[Diagram]

**Line-to-line angular dimension**

Measure the interior angle between the two legs and the exterior angle formed by two lines.
1. Click  

2. Click two lines.

3. Move the cursor between the lines to preview the inner angle dimension.

Line-to-line angular dimensions have a drag-able grip on the dimension arc for changing the angle to be measured:

Drag the grip point across one of the infinite lines through the ends of the selected edges/points to change the measured value to the supplementary or vertical angle of the angle where the text was first placed.

3 point angular dimension

Measure an angle by selecting 3 points, including a vertex and two points on the legs:
1. Click

2. Click the vertex.

3. Click a point on each leg on the perimeter of the arc.

Angular dimensions have a drag-able grip on the dimension arc for changing the angle to be measured.

Drag that grip point across one of the infinite lines through the ends of the selected edges/points to change the measured value.

On 3-point angular dimensions it changes from the initial angle to the outside angle (360 minus initial angle).

Before drag (below):

After drag (below):
Shortcut: Shift-r

Measure the radial dimension of a circle or arc.

1. Click 🧵.
2. Select the arc or circle.
3. Move the cursor and click to place the dimension.

You have the ability to foreshorten a dimension line on an arc when the center point is at an inconvenient distance from the arc on the drawing. In this case, select the dimension, right-click and select Foreshorten. A jogged line appears, drag to the desired location and click to set the endpoint. To undo the foreshortened line, select the line, right-click and select Remove foreshorten.

Diameter dimension

Shortcut: Shift-d

Measure the diameter of a circle or arc.
1. Click \( \bigcirc \).
2. Select the arc or circle.
3. Move the cursor and click to place the dimension.

**Ordinate dimension**

Create ordinate dimensions (X, Y pairs) for a feature measured from a datum. Ordinate dimensions created as a group move together when one is moved.

1. Click \( \bigcirc \).
2. Click the point to serve as the datum (0, 0).
   
   If the orientation of the ordinate dimension is not correct, move the mouse around the datum point until the 0,0, dimension is in the correct orientation, then click to set its location on the drawing.

3. Click each point in one direction (Y, for example) to associate with that datum point.
4. Press Escape to exit the tool.
   
   At this point, one Ordinate dimension group is created.

5. Click \( \bigcirc \).
6. Click the point to serve as the datum (0,0).
7. Click each point in the other direction (X, for example) to associate with that datum point. This datum is able to be the same as the first datum chosen.
8. Press Escape to exit the tool.

At this point, a second Ordinate dimension group is created.

Working with ordinate dimensions

Right-click on an ordinate dimension access the context menu for the ordinate dimension group:

- **Add to ordinate dimension group** - Select to add another ordinate dimension to the existing group; click a point to act as a datum point.

- **Edit** - Select to open the "Dimension palette" on page 1873 for the ordinate dimension group.

- **Reset ordinates** - If you've moved the ordinate dimension out of its original alignment, select this option to automatically move it to its previous position.

- **Clear selection** - Clear all selected items.

- **Zoom to fit** - Zoom the drawing to fit in the view.

- **Delete** - Delete the selected items.

Tips

- Use the inferencing to make realigning the datum points easier. Click and drag a datum point (it doesn't matter if it's a jogged position or not) to activate inferencing guidelines.
For example, to unjog the .963 datum point below:

Click and drag until the desired inferencing guideline appears, then release to place the datum point.

The bold datum point (.963 to the left, above) is the point being realigned; the red crosses (plus signs) on each datum point are alignment snap indicators.

The guidelines come in especially handy when trying to jog or unjog datum points.
Grouped ordinate dimensions are able to be moved as a group or singly. Drag the middle snap point of the dimension, notice the dashed lines appear on all dimensions in the group, and drag the dimension to a new position and all members of the group move in sync:

![Grouped ordinate dimensions](image)

Use the outmost snap point (below, on the far right) to drag a singular dimension on its own:

![Singular dimension](image)

If you subsequently move any of the group, all are moved in their relative positions, as indicated by the dashed lines:

![Group movement](image)

- Circular grip points at arrow bases flip the arrows to the opposite side of the dimension lines.
- Each direction must have a datum; each time you initiate the command from the toolbar, the first click establishes the datum point (0, 0).
Each direction of dimensions (Y, for example) consists of an ordinate dimension group with a single datum. To add another value pair to that group, select an existing value in the group, right-click and select **Add to ordinate dimension group**. This activates the command and the next click establishes the additional dimension value:

![Diagram of ordinate dimension group with added value]

If the drawing is updated such that the feature an ordinate dimension refers to is removed, the ordinate dimension remains and turns red. You are able to safely delete the dimension (right-click and select Delete, or select and press Delete).
Drag the jog point, the grab point at the point in the leader where it jogs, to reposition the jog point. When you reposition the dimension, the leader will jog at the new point. You can line up all the jog points for a series of ordinate dimensions, then when you move one dimension, the jog points of all of them will stay aligned:

The red line and x, above, show the alignment of the red x jog point to the others

Dimension palette
You have the ability to customize the appearance of a selected dimension with the Dimension palette. Selecting a dimension causes the dimension palette icon to appear.

1. With no tool selected, select the dimension.

2. The Dimension palette icon appears. 

3. Hover over the icon and the palette opens:

   ![Dimension palette](image)

4. You are able to enter, in order from top to bottom of the palette:
   - **Above text** - Enter the text or symbol to appear above the dimension value. 
   And then, for each of the primary dimension units and the dual dimension units, respectively:
   - **Prefix text** - Enter the text to appear as a prefix to the dimension value.
   - **Precision** - Select the depth of unit precision (zero to 8 decimal places).
     Precision defined on a drawing dimension may be linked to the Properties panel through this Dimension palette by selecting the tolerance with “(Drawing)” beside it. Whenever the Properties panel tolerance precisions are updated, any dimension with the “(Drawing)” tolerance selected will also be updated. You are able to choose to link these properties (and unlink them) on a dimension-by-dimension basis.
   - **Tolerance display** - Select None, Symmetrical, Deviation, Limits, or Basic.
   - **Select units** - Select the units of your choice, currently selected units are displayed in the dropdown label:
Choosing any of: Inches, Inches fractional, Millimeters, or Feet and Inches overrides the units for that dimension. If you later change the drawing units, the units for the dimension are not overridden. But you are able to change the units back to "(Drawing)" if you want to inherit the drawing properties again.

When you choose units, you set 2 properties in the drawing or on a dimension - the Units property and the Fractional display property.

- **Show hide units** - Toggle the display of units on and off.
- **Suffix text** - Enter the text to appear as a suffix to the dimension value.
- **Below text** - Enter the text or symbol to appear below the dimension value.
- **Symbol dropdown** - Select a symbol to insert from the dropdown:

- Degree
- Diameter
- Center line
- Counter sink
- Depth
- Counterbore
- Square
- Arc length
- Plus/Minus

- **Reset text position** - Toggle to reset the text to the previous location.
- **Add parenthesis** - Toggle to add or remove parenthesis around the dimension field.
• **Dimension inspection** - Toggle to add or remove an oval frame around the dimension to indicate this is an inspection dimension.

• **Dual dimension** - Toggle to specify whether to have a single unit dimension, single unit as specified for the drawing properties, or a dual unit dimension:

![Dimension options](image)

• **Toggle Radius/Diameter dimension** - Use this to change a radial dimension to diameter or vice versa.

> [•R]

You are also able to copy/paste into all text boxes, in dimensions and notes as well.

**Adding symbols**

In the text box of the Dimension panel, you have the ability to add codes in order to display the symbols of your choice:

**Drafting symbols**

• Degree (°), %°d

• Plus minus (±), %±p

• Diameter (Ø), %Øc
Flipping dimension arrows
Change the position of dimension arrows. Use on any dimensions that display arrows or ticks.

When you select dimensions, a node displays near dimension arrows or ticks. Clicking a node flips the arrows of the dimension.

To flip dimension arrows:
1. In the graphics area, select the dimension to change.
2. Drag the value to the new position (the arrows change accordingly).

Troubleshooting dimensions
At times, you may run across issues that you need to resolve, some of these may include:

- **Dangling dimensions** - A dimension with broken associativity, displayed in red. Drag the dimension snap point to re-associate to geometry. See Dangling entities for more info.

- **Overridden dimension** - A dimension with the text value converted into a non-associative annotation. The text of an overridden dimension is always underlined. Editing the dimension value of a dimension causes it to be converted into an overridden dimension, as such:
  - When a dimension is overridden, you cannot edit any of the other fields in the dimension panel; these fields become frozen and their contents are not shown on the dimension. Only the center and parenthesis commands are available.
  - You have the ability to restore an overridden dimension back to an associative dimension by deleting the characters in the dimension value field and exiting the panel.
  - Underlined dimension values on an engineering drawing indicate the value is not to scale.
This functionality is currently available only on browser.

Apply a hole callout to a hole, automatically inserting the metadata of the hole.

Steps

Creating a hole callout:

1. Click Ø
2. Select a circle that is part of a hole feature.
3. Drag to place the callout.

You can add multiple leaders to more holes, as long as the holes are the same size:

1. Right-click the callout and select Add leader.
2. Click on another hole.
3. Repeat as necessary to add leaders to as many same-size holes as necessary.
While moving the mouse to place the callout, you'll notice thin, red, dashed lines as the cursor passes near other entities (shown below to the right of the blue arrow). These are inferencing lines that you are able to align the dimension to; simply click when you see the line appear to align the dimension to that line.

**Editing a hole callout**

You can edit a hole callout once it is placed: double-click on the callout to open the panel:
Use the panel to edit characteristics such as:

- Prefix
- Units
- Suffix
- Notes
- Symbols
- Dimension inspection
- Single dimension or dual dimension selection - Dual dimensions may be top/bottom or side-by-side

**Hole drawing lines**

Threads on tapped holes are indicated by dashed lines for ANSI drawings, and the appropriate 3/4 outline for ISO drawings. An ANSI drawing is illustrated below:
To show/hide thread lines:

1. Right-click on the view.
2. Select Show/Hide threads (to toggle thread lines on and off).

---

**Datum**

This functionality is currently available only on browser.

Use Datum to create and place associative datum symbols to the drawing view on a surface that appears as a linear or circular edge to identify datum planes in the part:
Steps

Creating a datum:

1. Click A.

2. Enter the necessary label in the dialog.

3. Click to select an edge of a part view and drag away from the edge to establish the datum line.

4. Click to set the datum symbol.

While moving the mouse, you'll notice thin, red, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the dimension to; simply click when you see the line appear to align the dimension to that line.

Tips

- You are able to drag a datum to another location after placement: click to select it, then drag.

- You are able to drag a datum closer to or farther away from its placement point on the drawing and the extension line adjusts appropriately.
You are able to associate a datum with a dimension extension line:

![Dimension Extension Line](image)

To change the label, click to select the datum, then double-click in the highlighted square:

![Datum Label Change](image)

The datum dialog opens and you can change the label. You can also change the triangle in the properties setting for all arrowheads through the Properties panel, Annotations tab, Arrowhead setting.

---

**Geometric Tolerance**

This functionality is currently available only on browser.

Often associated with datum, use Geometric tolerance to create and place basic dimension notations in the drawing, like this:

![Geometric Tolerance](image)
Tolerances can be attached to edges, holes, dimensions (with or without leaders), dimension extension lines, extension lines, and even away from an edge (with no leader).

Creating a tolerance:

1. Click 

2. In the dialog, from corresponding lists, specify the symbols and associated tolerances for your drawing:

3. Complete the specifications by typing tolerance values in the corresponding boxes.

4. Click the plus sign \( \uparrow \) to add additional frames to the geometric tolerance. Change the bottom symbol to "Composite" to create a composite frame.

5. If desired, click the minus sign \( \downarrow \) to remove the last frame added.

6. Click in the graphics area to place the tolerance.

To place tolerance with a leader, hover over drawing view until a snap point appears, click with the desired snap point visible, drag tolerance and click to place. To add another leader, right-click the leader and select "Add leader". Click to place additional leader. Repeat to add more leaders:

To place the tolerance on an edge in the view, place the tolerance first, then drag it to the edge and release when there is no leader visible:
To place the tolerance along an extension line, drag the tolerance away from the snap point, until an extension line appears:

When aligning dimensions and annotations, you can hover over edges, mid-points, or other lines to activate red inference points. Use the inference points to snap the location of the entity when creating or dragging a dimension or annotation. Similarly, you can move the mouse vertically or horizontally over views, lines, dimensions, or annotations to activate pink vertical and horizontal inference lines. Use these inference lines to align the location of the entity vertically or horizontally from the desired referenced annotation.

The tolerance displays in the graphics area.
Tolerance with leader:

You have the ability to drag a geometric tolerance closer to or farther away from its placement point on the drawing, and the extension line adjusts appropriately. You also have the ability to attach and orient the geometric tolerance to other drawing annotations.

When dragging away from a dimension, the geometric tolerance automatically gains a leader and remains inline with the dimension.

Editing a tolerance:

1. Double-click on the tolerance in the graphics area.

2. Make your changes in the dialog that opens.

Add/remove leader nodes:

1. Right click on the leader and click Add node:
A node will appear on the leader. To add another node, right click on the leader and select Add node again.

2. To remove a node from a leader, right click on the node and select Remove node.

3. You are also able to add leaders to nodes by right clicking on the node and selecting Add leader. This will add a leader starting from that node and ending wherever you click next.

**Geometric characteristic symbols**

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<tr>
<th>Symbol</th>
<th>Characteristics</th>
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<td>Location</td>
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<td><img src="symbol2" alt="Concentricity symbol" /></td>
<td>Concentricity or coaxiality</td>
<td>Location</td>
</tr>
<tr>
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<td>Location</td>
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<td>Orientation</td>
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<tr>
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<td>Circularity or roundness</td>
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**Leader symbols**

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**Diameter symbols**

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<tr>
<td>SØ</td>
<td>Spherical diameter</td>
</tr>
<tr>
<td>R</td>
<td>Radius</td>
</tr>
<tr>
<td>SR</td>
<td>Spherical radius</td>
</tr>
<tr>
<td>CR</td>
<td>Controlled radius</td>
</tr>
</tbody>
</table>

**Modifier symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Characteristics</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>At maximum material condition, a feature contains the maximum amount of material stated in the limits</td>
<td>MMC</td>
</tr>
<tr>
<td></td>
<td>At least material condition, a feature contains the minimum amount of material stated in the limits.</td>
<td>LMC</td>
</tr>
<tr>
<td>Symbol</td>
<td>Characteristics</td>
<td>Type</td>
</tr>
<tr>
<td>--------</td>
<td>---------------------------------------------------------------------------------</td>
<td>--------</td>
</tr>
<tr>
<td>☨</td>
<td>Regardless of feature size, indicates that the feature can be any size within the state limits.</td>
<td>RFS</td>
</tr>
<tr>
<td>☪</td>
<td>Free state</td>
<td>F</td>
</tr>
<tr>
<td>☩</td>
<td>Tangent plane</td>
<td>T</td>
</tr>
<tr>
<td>☪</td>
<td>Regardless of feature size, indicates that the feature can be any size within the state limits.</td>
<td>RFS</td>
</tr>
<tr>
<td>☪</td>
<td>Projected tolerance zone</td>
<td></td>
</tr>
</tbody>
</table>

### Surface Finish

This functionality is currently available only on browser.

Place a surface finish symbol on the edge of a circular view or away from edges.
Steps

1. Click ✓.

2. In the dialog, describe the surface finish using the fields.

   ![Surface finish dialog]

3. Click in the drawing to place the symbol and surface finish descriptions:
   - Click anywhere in the white space of the drawing
   - Click a point on the edge of a circular view

4. Enter more descriptions in the dialog, if desired, and place more symbols.

5. Click ✓ when finished (or ✗ to cancel).

To add a leader, right-click the symbol and select "Add leader". Click to place leader. Repeat to add more than one leader.

You are able to associate a surface finish symbol with a dimension extension line:
Add/remove leader nodes:

1. Right click on a leader and click Add node:

   A node will appear on the leader. To add another node, right click on the leader and select Add node again.

2. To remove a node from a leader, right click on the node and select Remove node.

3. You are also able to add leaders to nodes by right clicking on the node and selecting Add leader. This will add a leader starting from that node and ending wherever you click next.

---

**Weld Symbol**

This functionality is currently available only on browser.
Place weld symbols on a drawing. Weld standards default to the standard specified for the drawing. To change the weld standard, access the drawing’s Properties flyout located on the right side of the page.

**Steps**

1. Click ⬇️.
   
   The drawings standards (ISO or ANSI) determines which version of the dialog is displayed.

2. In the dialog, describe the weld symbol using the fields.

![Weld symbol dialog](image)

3. Select either Joint spacer (with accompanying defaults) or a joint type (and specify options).
   
   a. Joint spacer types:
      
      i. Double V groove
      
      ii. Double bevel groove
      
      iii. Double U groove
      
      iv. Double J groove
      
      v. Double flare V
      
      vi. Double flare bevel
b. Joint types:
   i. Square butt joint
   ii. V butt joint
   iii. U butt joint
   iv. J butt joint
   v. Bevel butt joint
   vi. Single bevel butt broad root joint
   vii. V flare butt joint
   viii. Bevel flare butt joint
   ix. Bead joint
   x. Fillet joint
   xi. Plug or Slot joint
   xii. Seam joint
   xiii. Spot joint
   xiv. Seam centered joint
   xv. Spot centered joint

4. Click in the drawing to place the weld symbol and weld descriptions, on a line or vertex.

5. Check Stagger (available on specific ANSI butt joints) to indicate staggered welds.
6. Check Symmetric (available on specific ISO joints) to indicate symmetric welds. Optionally, also select a Second fillet to indicate a fillet with a left vertical line in the symbol.

7. Check Reference to activate the text box and enter notes.

8. Check ‘All around’ to indicate to generically apply an option everywhere it’s appropriate.

9. Select a ‘flag’ to indicate an action to be performed in the field.

10. Click in drawing to place weld symbol.

    To add another leader, right-click the leader and select "Add leader". Click to place additional leader. Repeat to add more leaders.

11. Click ✔️ when finished (or ✗ to cancel).

    There is no need to click directly on the point once it is visible. While moving the mouse to place the weld symbol, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the weld symbol to; simply click when you see the line appear to align the weld symbol to that line.

    Add/remove leader nodes:

    1. Right click on the leader and click Add node:
A node will appear on the leader. To add another node, right click on the leader and select Add node again.

2. To remove a node from a leader, right click on the node and select Remove node.

3. You are also able to add leaders to nodes by right clicking on the node and selecting Add leader. This will add a leader starting from that node and ending wherever you click next.

### Note

Shortcut: n

This functionality is currently available only on browser.

Add single or multi-line text notes to any drawing, wherever you want, and use them to fill in the title blocks as well. You have the ability to define the size of the text box as well as format the text itself and optionally include a leader or many leaders. You also have the ability to rotate the orientation of the note and create additional leaders.

Note that hovering over a note automatically causes a blue highlight on the edges attached to the leaders:
Creating notes:

1. Click A.

2. To create a note with a leader:

   a. Move the cursor near a drawing view until you see the grab point you want the leader to attach to.
   
   b. Click when you see the desired grab point (by default, the start point is the arrow head).
   
   c. Move the cursor to the position at which to place the text box: click again.
      
      A small text box appears at the end point of the leader. A note dialog accompanies it.
   
   d. Enter text and apply formatting and when finished, click to close the dialog.
      
      To add more leaders, right-click the note and select Add leader. A leader automatically appears at the note grip point closest to the mouse location. Drag to a grip point in the drawing and click to place the leader. Repeat as necessary.

3. To create a note with no leader:

   a. Move the cursor to empty white space and click to place the text box.

   b. Enter text and apply formatting and when finished, click to close the dialog.
Note that only notes without leaders can be rotated.

There is no need to click directly on the point once it is visible. While moving the mouse to place the note, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the note to; simply click when you see the line appear to align the note to that line.

To resize or rotate a note:

a. Create the note without a leader.

b. Select the note, or hover over the note.

c. To rotate: Click the center drag point (as shown) and drag in a circular motion. Use the numeric box that appears to estimate the value of the angle:

Rotating an aligned note detaches it from the edge it is aligned with.

d. To resize: Click a square drag point on a side edge and drag to resize.

After creating a note, you are able to indicate to **explode** the note into polylines upon export. Use the context menu and select Explode:
You are also able to **Flip** a note that is aligned to an edge (accessed through the context menu on the note). Flip rotates the note by 180 degrees and moves it to the other side of the edge. Flip is not available for notes that are not aligned.

You are able to enter unicode characters in notes. For example, use \U+00AE to create the registered trademark symbol ®.

You are also able to use shortcuts such as %%d for the degree symbol, %%c for the diameter symbol, and %%p for the plus/minus symbols.

You have the ability to drag a note off of an edge, thereby detaching it: use the middle bottom grip point. You are able to drag to reattach to another position on an edge.

**Adding a leader**

You are able to add a leader to a note created without a leader:
1. Right-click on the note.
2. Select Add leader from the menu:

![Menu with Add leader option highlighted]

3. Click on the grab point to which to attach the leader.

**Removing a leader**

You are able to remove a leader from any note, regardless of whether the leader was created at the time the note was created or after the note was created:

1. Right-click on the note.
2. Select Remove leader from the menu:

![Menu with Remove leaders highlighted]

**Removing leaders and/or text**

1. Click to select the leader and/or text.
2. Press the Delete key.
You are also able to remove a leader and/or text by right-clicking on the leader and/or text and clicking Delete:

Repositioning leader and text
To reposition the leader and text at the same time:
1. With no tool selected, click to select the text.
2. Click anywhere in the text and drag.
3. Upon release, the leader snaps to the text in its new location.

To reposition only the leader:
1. With no tool selected, click to select a grab point on the leader.
2. Drag the leader to its new position.

Add/remove leader nodes
1. Right click on the leader and click Add node:

A node will appear on the leader. To add another node, right click on the leader and select Add node again.
2. To remove a node from a leader, right click on the node and select Remove node.

3. You are also able to add leaders to nodes by right clicking on the node and selecting Add leader. This will add a leader starting from that node and ending wherever you click next.

**Modifying text**

1. With no tool selected, double-click the text.

2. Make changes to the text and/or the Note formatting. Changes to text height become your new default.

3. Click ✓.

> Alignment of text is controlled by the drafting standard of the chosen template; the alignment buttons are disabled in this context.

**Moving notes**

Use the context menu (right-click) on the note to move it to the title block, border zones, border frame, or back to the drawing:

1. Select the note.

2. Right-click and select Move to.

3. Select title block, border zones, or border frame.
When the Format (on the Properties panel) is locked, you are unable to edit Notes that are not on the Drawing layer. To edit when the Format is locked, move the Note back to the Drawing layer.

**Copying notes**

Use the context menu (right-click) on the note:

1. Select the note.
2. Right-click and select Copy.

3. Right-click again and select Paste.

Copy and paste notes and notes with leaders from any drawing in any document (including a specific document workspace) to which you have permission and to any drawing in any document (including a specific document workspace) to which you have permission.

**Formatting notes**

You have the ability to double-click on a note to open the editor or triple-click to open with all text selected. In addition, in an open text box:

- Ctrl-a to select all text in the note
- Double-click to select a word (up to the next space)
• Triple-click to select a line (up to the next line break)
• Use copy/paste shortcut keys with the system clipboard to insert text from other pages or programs

Use the following controls to format your notes:

• **Ruler** - Set paragraph indents and tab stops for Notes. See more information below under *Formatting ruler*.

• **Font** - Specify a typeface using an SHX file or a True Type font file.

• **Text height** - Specify the text height for subsequent or selected text. Text height is measured from the baseline to the top of a regular uppercase glyph (cap line), also known as the Cap Height. This specification becomes your new default.

• **Bold** - Indicate subsequent or selected text is bold; works with True Type fonts only.

• **Italic** - Indicate subsequent or selected text is italic; works with True Type fonts only.

• **Underline** - Indicate subsequent or selected text is underlined; works with True Type fonts only.

• **Strikethrough** - Indicate subsequent or selected text is struckthrough (draws a line through the middle of the text).

• **Line spacing** - Change the spacing between lines of text. This applies to the entire Note. Select an option:
  - **1.0, 1.5, 2.0, 2.5, 3.0** - Set line spacing to one of these factors.
  - **Add space before paragraph / Remove space before paragraph** - Add or remove space before a paragraph; the line spacing is set as mentioned in this list, above.
  - **Add space after paragraph / Remove space after paragraph** - Add or remove space after a paragraph; the line spacing is set as mentioned in this list, above.

• **Horizontal alignment** - Select a type of horizontal alignment of paragraphs: left-aligned text, right-aligned text, centered text, or justified text (aligned evenly along left and right margins).
Note that this option is disabled in Note with leader command; drafting standards dictate alignment of text in that context.

- **List** - Select a type of list formatting to use, including: numbers, bullets, uppercase letters, or lowercase letters.

- **Vertical alignment** - Indicate the type of paragraph justification in relation to the insertion point of the Note: Top, Middle, or Bottom.

Note that this option is disabled in Note with leader command; drafting standards dictate alignment of text in that context.

- **Fractions** - You can use 3 different codes to create fractions formatted in 3 ways:
  - Fractions formatted with a diagonal slash between the numbers - `<number>#<number><space>`
  - Fractions formatted with a horizontal slash between the numbers - `<number>/<number><space>`
  - Fractions formatted with no slash between the numbers, just stacked on each other vertically - `<number>^<number><space>`

If the conversion to a fractional character is not desired, type any character other than `<space>` directly after the second number, then navigate back to it using the arrow keys or cursor, and delete it. (Otherwise, the special character code is not editable.)

- **Symbols** - Select a symbol to insert at the current cursor location. Keep in mind that you can use some shortcuts, such as: `%%d` (degrees), `%%c` (diameter), and `%%p` (plus/minus).

- **Insert drawing property** - Insert the value of a Drawing property automatically, especially useful for filling in the title block, including:
  - Drawing created by - The user name of the user who created the drawing
  - Drawing created date - The date the drawing was initially created
  - Drawing description - The description of the drawing as it exists in the Drawing tab’s Properties (right-click the tab and select Properties)
• Drawing last changed by - The user name of the user who last modified the drawing
• Drawing last changed date - The date of the most recent change to the drawing
• Drawing name - The name of the drawing, as displayed in the title box
• Drawing part number - The part number given to the drawing as it exists in the Drawing tab’s properties (right-click the tab and select Properties)
• Drawing revision - The current revision of the drawing as it exists in the Drawing tab’s properties (right-click the tab and select Properties)
• Drawing Title 1 - Title 1 of the drawing as it appears in the title box; defined in the Document properties
• Drawing Title 2 - Title 2 of the drawing as it appears in the title box
• Drawing Title 3 - Title 3 of the drawing as it appears in the title box
• Sheet number - The number of the currently displayed sheet
• Total sheets - The total number of sheets in the drawing
• Sheet scale - The scale of the views on the sheet. Change this value in Sheet properties.
• Sheet size - The sheet size; also linked to the size property in the Title block.

To view and edit Drawing properties, right-click the drawing tab and select "Properties" from the context menu.

• Insert sheet reference property - Insert a property referencing the entity from which you created the drawing. Default Onshape properties and active custom properties are listed. Active custom properties appear at the bottom of the list in alphabetical order. For more information on metadata and custom properties see Properties.

• Name
• Description
• Part Number
• Revision

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All rights reserved.
- State
- Vendor
- Project
- Product Line
- Title 1
- Title 2
- Title 3
- Material

- **Text format** - Set the text format of an inserted drawing property to one of the following:
  - All uppercase
  - All lowercase
  - Sentence case
  - Title case
  - Unmodified

- **Date time format** - Select a date and time format to use for an inserted drawing property.

To edit a field in the title block:

1. Double-click the property. At this point, the cursor will be at the end of the property:

2. Select the property. It is helpful to use a mouse (and not the Shift-arrow keys). When a property is highlighted you will notice a thin blue line below it:
3. Click and select the property to insert, for example, Drawing name:

4. With the property still highlighted, the text formatting icon is active; click to select text formatting options for capitalization formatting.

5. If a date property was inserted, the Date format icon becomes active. Enter any of the following date formats:

<table>
<thead>
<tr>
<th>Code</th>
<th>Sample</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>yyyy-M-d</td>
<td>2016-3-9</td>
<td>4 digit year, 1-2 digit month, 1-2 digit day</td>
</tr>
<tr>
<td>yyyy-MM-dd</td>
<td>2016-03-09</td>
<td>4 digit year, 2 digit month, 2 digit day</td>
</tr>
<tr>
<td>M.d.yyyy</td>
<td>3.9.2016</td>
<td>1-2 digit month, 1-2 digit day, 4 digit year</td>
</tr>
<tr>
<td>M/d/yyyy</td>
<td>3/9/2016</td>
<td>1-2 digit month, 1-2 digit day, 4 digit year</td>
</tr>
<tr>
<td>M/d/yy</td>
<td>3/9/16</td>
<td>1-2 digit month, 1-2 digit day, 2 digit year</td>
</tr>
<tr>
<td>MM-yy</td>
<td>03-16</td>
<td>2 digit month, 2 digit year</td>
</tr>
<tr>
<td>Code</td>
<td>Sample</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>MM/dd/yyyy</td>
<td>03/09/2016</td>
<td>2 digit month, 2 digit day, 4 digit year</td>
</tr>
<tr>
<td>MMM.d, yy</td>
<td>Mar.9, 16</td>
<td>3 letter month, 1-2 digit day, 2 digit year</td>
</tr>
<tr>
<td>MMMM yy</td>
<td>March 16</td>
<td>Full month, 2 digit year</td>
</tr>
<tr>
<td>MMMM d,yyyy</td>
<td>March 9,2016</td>
<td>Full month, 1-2 digit day, 4 digit year</td>
</tr>
<tr>
<td>d-MMM-yy</td>
<td>9-Mar-16</td>
<td>1-2 digit day, 3 letter month, 2 digit year</td>
</tr>
<tr>
<td>d MMMM yy</td>
<td>9 March 16</td>
<td>1-2 digit day, full month, 2 digit year</td>
</tr>
<tr>
<td>dd.MM.yyyy</td>
<td>09.03.2016</td>
<td>2 digit day, 2 digit month, 4 digit year</td>
</tr>
<tr>
<td>dd/MM/yyyy</td>
<td>09/03/2016</td>
<td>2 digit day, 2 digit month, 4 digit year</td>
</tr>
<tr>
<td>dddd,MMMM d,yyyy</td>
<td>Monday, March 9,2016</td>
<td>Day of the week, full month, 1-2 digit day, 4 digit year</td>
</tr>
</tbody>
</table>

**Formatting ruler**

Use the Note Formatting ruler to set paragraph indents and tab stops for Notes.

The ruler appears with the Note Formatting pop-up toolbar. It is located at the top of the Note bounding box.

By default, there are no paragraph indents or tab stops on the ruler when you start a new Note.
Paragraph indents and tab stops that you set before you start to enter text apply to the entire Note. When you type or edit, place the pointer in the paragraph to format or select multiple paragraphs to adjust indents and tab stops.

This example shows first line indent, left indent, and hanging indent:

![Example paragraph with indent settings](image)

**Setting paragraph indents**

1. Place your cursor in the paragraph to format, or select multiple paragraphs.
2. On the Formatting ruler, slide indent markers:
   a. Slide the First line indent marker to the position you want the first line of the paragraph to begin.
   b. Slide the Left indent marker from the left to the position you want the second and all following lines of a paragraph to begin (also referred to as a hanging indent).
   c. Slide the Right indent marker from the right to the position you want all lines of a paragraph to end.

The indent settings are maintained for subsequent paragraphs as you type.

**Setting tab stops**

1. Place your cursor in the paragraph to format, or select multiple paragraphs.
2. Click the tab selector at the left end of the ruler until it displays the type of tab you want to use:
   - **Left** - Set the start position for subsequent text. The text runs to the right as you type.
   - **Center** - Set the position for the middle of the text. The text centers on this position as you type.
   - **Right** - Set the start position for subsequent text. The text runs to the left as you type.
• **Decimal** - Align numbers around a decimal point. Independent of the number of digits, the decimal point is in the same position. You are able to align numbers around the same type: period, comma, or space.

3. Click the ruler at the location you want to place the tab stop.

   As you click or drag tab stops, tooltips show the exact position from the left (in drawing units).

4. Repeat the steps above as needed.

Note that when multiple paragraphs are selected, only the tab stop from the first paragraph shows on the ruler.

**Relocating tab stops**

Drag existing tab stops left or right along the ruler.

**Removing tab stops**

Drag a tab stop (up or down) off the ruler. When you release the mouse button, the tab stop disappears.

**Completing a title block**

Some fields in **title blocks** are filled in automatically, using document and drawings information. The rest you are able to fill using notes within the boundaries of the title block cells.

For best results:

- Create a text box the size of the cell. You are then able to experiment with text size, font, etc. and see if the text will fit without having to resize the text box.

- You are unable to copy and paste text boxes; but you are able to copy and paste text from one text box to another.

- When copying and pasting text from one text box to another, the formatting is carried over.

- The labels in a title block are completely customizable as well. They are simply multi-line text, just another note.

- You are able to move the lines of the title blocks, or create your own.
Callout (Balloon)

This functionality is currently available only on browser.

Create and edit callouts (balloons) with a leader line. Callouts are associative with the part metadata of the document and a BOM table when a BOM App is used within your Onshape document. When a BOM table is present, the callout defaults to use the Item Number in the table. If the table order is changed, use the Update Drawing feature to update the callouts.

To create a callout:
1. Click 

   a. To place a **callout with a leader**, hover over a view to activate a snap point. Click to anchor the leader to the snap point, then click to set the balloon.

   b. To place a **callout without a leader**, simply click in the white space of the drawing without activating a snap point.

2. Make specifications in the dialog before placing the callout in the drawing.

   ![Callout dialog](image)

   Click ✔️ to save your specifications and close the dialog.

   While moving the mouse to place the callout, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you can align the dimension to; simply click when you see the line appear to align the dimension to that line.
3. Use the options to dictate the style and content of the callout:

- Part property - Select from the metadata for the part to create associative links to that data:

```
<table>
<thead>
<tr>
<th>Callout</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Name</td>
</tr>
<tr>
<td>Description</td>
</tr>
<tr>
<td>Part number</td>
</tr>
<tr>
<td>Revision</td>
</tr>
<tr>
<td>State</td>
</tr>
<tr>
<td>Vendor</td>
</tr>
<tr>
<td>Project</td>
</tr>
<tr>
<td>Product line</td>
</tr>
<tr>
<td>Material</td>
</tr>
<tr>
<td>Title 1</td>
</tr>
<tr>
<td>Title 2</td>
</tr>
<tr>
<td>Title 3</td>
</tr>
</tbody>
</table>

Exclude from BOM
```
Table property - Select from the fields included in your BOM table to create associative links to that table:

- Enter text for above, below, and to the left and right of the callout data.
- Select text formatting, balloon style, and number of characters for the callout text.
- Font size, and border shape and size specifications are retained and used as defaults for the next callout.

Editing a callout

Simply right-click the callout to open the context menu:

- **Edit** - Open the Callout dialog to edit the callout specifications.
- **Copy** - Copy the callout.
- **Add leader** - Add a leader (or another leader) to the callout.
- **Remove leaders** - Remove all leaders from the callout.
- **Group annotation / Remove annotation from group** - Select multiple callouts, right-click and select Group annotation to create a group of annotations that can be moved together. Select one or more callouts in the group and select Remove annotations from group to remove those items from the group so they can be moved singly.
- **Copy/Paste** - Copy, and paste a copied callout; callouts move as one until you attach one to a part in the drawing (via the leader).
• **Move to** - Move a callout to a different layer of the drawing: Border frame, Border zones, or Title block. Once on another zone, you have the option of locking the layers (through Drawings properties, Formats tab, in order to stabilize the position of entities on that layer.

• **Clear selection** - Remove any highlighted items from the selection queue.

• **Zoom to fit** - Zoom appropriately to fit the entire drawing in the field of view.

• **Delete** - Delete selected entities.

To align callouts with each other, select multiple callouts, right-click to access the context menu and select either:

• Align horizontally

• Align vertically

Keep in mind that the callouts are not locked into the place. You can still drag to place them separately, or multi-select and select the alignment commands again.

**Removing callouts**

1. Click to select the callout or leader. Onshape selection is additive so you can select multiple items and then execute a command. (Use right-click *Clear selection* to remove all selections from the queue.)

2. Press the Delete key.

**Grouping callouts**

You can group callouts to dimensions and to other callouts, and the move them as a group, if desired.

To group an existing callout to a dimension:

1. Click to select the callout.

2. Drag the callout to an existing dimension. When the dimension gains a highlight,
release the drag.

To group a callout during creation:

1. Once the Callout dialog is open, make your necessary changes in the dialog.
2. When placing the callout in the drawing area, click on the dimension you wish to group it with.

You can change the position of just the callout in relationship to the dimension or other grouped items, use SHIFT-drag. This moves just the selected callout and not the other items in the group. To move all items in the group together, simply click and drag any item in the group.

**Using callouts with BOMs**

When you are using BOMs with your drawings ("Insert BOM" on page 1925) the Balloon tool adjusts to create automatic identifiers in the balloon, according to the data in the BOM you have inserted into the drawing.

With a BOM already inserted into your drawing:

1. Click \( \text{\textcircled{1}} \).
2. Click the start point of the leader.
3. Click the end point of the leader.
4. The dialog opens for further detail input:

5. Use the icon drop downs at the top to specify which properties in the table to pull the balloon identifier from:
   - ⬤ - Select from a list of all Part properties
   - ⬤ - Select from a list of all columns in the BOM table

6. Supply additional details (optional) for above, below, and to the right and left of the balloon identifier.

7. Select a point size for the text, the shape of the balloon, and how many characters wide you want the balloon to be.

8. Click ✓ to save your specifications.

**Add/remove leader nodes**

1. Right click on the leader and click Add node:
A node will appear on the leader. To add another node, right click on the leader and select Add node again.

2. To remove a node from a leader, right click on the node and select Remove node.

3. You are also able to add leaders to nodes by right clicking on the node and selecting Add leader. This will add a leader starting from that node and ending wherever you click next.

Table

This functionality is currently available only on browser.

Add fully-customizable tables to any drawing.
Steps

1. Click 📋.

2. The cursor becomes a table icon and the Table dialog opens:

   ![Table dialog](image)

3. Before clicking to place the table, you can enter the number of rows and columns in the dialog.

4. Specify whether to include a Title row (a row that spans all columns at the top of the table) and a Header row (an additional row just below the Title row) by clicking one of the checkboxes to the left of the option.

   ![Table dialog with options](image)

5. Select which corner of the table to set as your fixed corner at the bottom of the dialog (as shown below):

   ![Select fixed corner](image)

The default anchor point in Onshape is the upper right corner of tables.
6. To place the table, click on the sheet to set the location (anchored by the fixed corner selected in step 4).

There is no need to click directly on the point once it is visible. While moving the mouse to place the table, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the table to; simply click when you see the line appear to align the table to that line.

1. Click the green check in the dialog.

2. The "Note" on page 1895 opens:

3. Click in a cell to enter text; tab (or press Enter) to move from cell to cell, use Shift-Tab for previous cell.

Double-click in any cell to open the Note panel for editing text.

**Formatting tables**

After a table is created and the Table dialog is closed, you are able to hover over a table to activate grab points:
Drag any of the mid-points (top, bottom or sides) to resize the table; cells are resized relative to each other.

Drag any of the corner points to move the table.

Double-click on the mid-points of a table to automatically size to fit the contents of the table.

Hover over a row or column divider for a grab point and drag to resize the entire row or column at once.

Drag a corner of the table to snap to a corner of the drawing; when you see the red snap point, let go of the mouse button to place the table.

Double-click on a column or row divider to size the column or row to the text within:

Select a cell, right-click and select Size > Columns equally or > Rows equally to resize either an entire row or column equally to be of the same size.
• Single-click in a table cell or row to activate the Edit Table toolbox:

The cell outlined in blue is the selected cell.

• Double-click your previously selected fixed corner to activate the Table properties dialog, where you are able to edit your selected fixed corner by simply clicking on the box of your choice. Click the checkmark in the top right corner of the dialog to set your changes:

Edit Table toolbox

Copyright © 2017, Onshape. - 1922 -
All rights reserved.
Shift-click to select more than one cell
Select multiple cells that are adjacent to each other
Click in a cell to select; use the grab points to resize the cell's row or column

- **Insert row above** - Insert one row above the currently selected row(s)
- **Insert row below** - Insert one row below the currently selected row(s)
- **Insert column left** - Insert one column to the left of the currently selected column(s)
- **Insert column right** - Insert one column to the right of the currently selected column(s)
- **Remove rows** - Remove the currently selected row(s)
- **Remove columns** - Remove the currently selected column(s)

When more than one column is selected, you can also:

- **Size columns equally** - Resize all selected columns to the average width
- **Merge cells** - Merge the selected cells into one cell (horizontally, vertically, or all)

When more than one row is selected, you can also:

- **Size rows equally** - Resize all selected rows to the average height

After merging cells, you may also:

- **Unmerge cells** - Return last-merged cells to previous unmerged state

Note that you are able to access these commands from the context menu when at least one cell is selected:
All of the text formatting commands available in the Note panel are also available in the Table toolbox.

**Table properties**

There are two ways to open the table properties dialog, listed below:

1. Right click on the table and select Table properties...

   ![Table properties dialog](image)

   The table properties dialog opens.

2. Or, simply double click on any corner of the table, and the table properties dialog opens immediately:

   ![Table properties dialog](image)
From the table properties dialog, you are able to adjust which corner of the table is the fixed corner by simply clicking on the box of your choice. Click the checkmark in the top right corner of the dialog to set your changes.

---

**Insert BOM**

This functionality is currently available only on browser.

Insert a Bill of Materials table from an Onshape BOM table, uploaded file, or an authorized app. You are also able to use the Balloon tool to automatically pull identifiers from the BOM to label the drawing. See "Callout (Balloon)" on page 1911 for more information.

**Steps**

1. Click 🗂️.

2. Select to insert a Bill of Materials from the currently active document, or another document.

![Insert BOM Dialog]

If there are Onshape BOM tables in your document, they are automatically listed in the drop down menu (shown with "Master assembly" above); select one, then the parameters:
a. The BOM type: Flattened (no assembly hierarchy indicated), Structured - Top level (assembly hierarchy indicated for one level only), or Structured - Multi-level (all levels of assembly hierarchy indicated).

b. Order: Top to Bottom or Bottom to Top

The three types of BOMs are shown below: Flattened, Structured-Top level, and Structured-Multi-level, respectively:

<table>
<thead>
<tr>
<th>Item No.</th>
<th>QTY</th>
<th>Name</th>
<th>Part number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>Part 2</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>Part 3</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>Part 1</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Item No.</th>
<th>QTY</th>
<th>Name</th>
<th>Part number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>Assembly A</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>Assembly B</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Item No.</th>
<th>QTY</th>
<th>Name</th>
<th>Part number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>Assembly A</td>
<td></td>
</tr>
<tr>
<td>1.1</td>
<td>1</td>
<td>Part 2</td>
<td></td>
</tr>
<tr>
<td>1.2</td>
<td>2</td>
<td>Part 3</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>Assembly B</td>
<td></td>
</tr>
<tr>
<td>2.1</td>
<td>1</td>
<td>Part 1</td>
<td></td>
</tr>
<tr>
<td>2.2</td>
<td>2</td>
<td>Part 3</td>
<td></td>
</tr>
</tbody>
</table>

In the multi-level BOM, immediately above, the sub-assembly (Assembly A) is listed as item number 1 and it consists of two parts, listed as item numbers 1.1 and 1.2 (Parts 2 and 3). Sub-assembly (Assembly B) is listed as item number 2 and it consists of two parts, listed as item numbers 2.1 and 2.2 (Parts 1 and 3). The top-level BOM, the middle BOM above, lists only the top-level of the two sub-assemblies. The flattened BOM, the first BOM above, lists just the parts contained in the Assembly.

Use the Insert icon to select a BOM file or table from an approved app.
From here you are able to use the standard Insert dialog to select from the current document, another document, create a version to work with or select another version of a document. You are able to insert BOMs from versions, workspaces, and releases that don't have a BOM, as well. You also have the ability to select from Assemblies in the current document, use BOM data that you have uploaded in the form of a file, or select a BOM from an approved app you are using.

3. Select whether to insert Bill of Materials data from:

- The current document or Other documents
  - **Assemblies** - Within whatever filter you have selected: Current document or another document.
  - **BOM data** - An uploaded file containing data, select the file or use the Import option at the bottom of the dialog to upload a file (not shown above).
  - **BOM app** - Data created from a BOM app through the Onshape App store, select the Bill of Material.
4. If a document contains released parts, the Release filter is present; use this filter to display only data that has been released.

5. Select which corner of the table to set as your fixed corner at the bottom of the table (as shown below).

   ![Select fixed corner]

   The default anchor point in Onshape is the upper right corner of tables.

   There is no need to click directly on the point once it is visible. While moving the mouse to place the Bill of Materials, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the BOM to; simply click when you see the line appear to align the BOM to that line.

6. Click in the drawing space to place the Bill of Material table.

   **Editing a BOM table**

   An inserted Bill of Materials is able to be edited much like any table in drawings. Click the edge of the table to select the entire table; you may grab one of the middle grab points to move the table. Right-click with the entire table selected to access these commands:

   - **BOM table properties** - To open the properties panel for Bills of Materials
   - **Split BOM table below** - To split the BOM table below the row the cursor was in when you right-clicked for the context menu. This splits the table into two tables. You can then select one of the tables, right-click for the context menu and use *Move to sheet* to move that table to another sheet in the drawing.
   - **Copy** - To place a copy of the table on the clipboard
   - **Move to** - (Border frame, Border zones, Title block, Drawing)
   - **Bring to front** - Bring the selected table to the forefront of the display
   - **Send to back** - Send the selected table to the background of the display
• **Switch to <appropriate tab>** - To open the tab containing the Bill of Materials data

• **Move to sheet** - To move a table (or a split table) to another, existing, sheet of the drawing. You can open the Sheets flyout to view the location of the BOM table and also drag/drop it to another sheet if you wish:

![Sheets (3)](image)

Right-click on the BOM in the list to access the context menu, including Properties, Switch to BOM (opens the Assembly tab in which the BOM originated), and Delete:

![Right-click on BOM](image)

• **Clear selection** - Unselect the currently selected table

• **Zoom to fit** - Zoom the drawing to fit in the window

• **Delete** - Delete the currently selected table

Once the table is inserted you are able to:

• Use Shift-click to select more than one cell.

• To select a row, click the first cell and Shift-click the last cell in the row.
To select a column, click the first cell and Shift-click the last cell in the column.

Click and drag a corner to resize the table.

Moving the table

Click and drag a corner grip point of the table and drag to reposition it.

Resizing the table

You are able to right-click in the table and select Size and choose from resizing Columns equally and resizing Row equally. Click and drag a horizontal or vertical mid-point to resize the table.

To resize a single column, click and drag any vertical edge grab point of any cell in the column.

To resize a single cell, click and drag any edge grab point of the cell.

Formatting

Double-click in a table cell to open the cell formatting panel:

Deleting a BOM table

Right-click the edge of the table and select Delete. You can also delete a BOM table from the Sheet flyout: right-click the BOM in the Sheets flyout, and select Delete.
Replacing a BOM table

Once a BOM table is placed in a drawing, you can replace it with another BOM table if you wish:

1. Open the Sheets flyout and locate the listing for the BOM table you wish to replace:

2. Right-click on the listing and select Replace.

3. In the Insert dialog that appears, Select the Assembly whose BOM you wish to insert:
4. The BOM is replaced with the BOM of the selected Assembly.

5. The views associated with the BOM are also replaced to reference the selected Assembly.

**BOM properties**

Select the table, then right-click and select BOM table properties to open the Properties dialog for that table:

![BOM properties dialog](image)

Click the Reference link to open the tab containing the original BOM data.

Select:

- **BOM type** - For Onshape BOMs:
  - Flattened (no assembly hierarchy shown)
  - Structured (assembly hierarchy shown)
Order:

- Top to bottom - (Default) Header at top of table and rows in the order in which they appear in the BOM app.
- Bottom to top - Header at bottom of table and rows in reverse order as they appear in the BOM app.

To adjust which corner of the table is your fixed corner, click your preferred option at the bottom of the BOM table properties table:

![Select fixed corner]

---

**Hole Table**

This functionality is currently available only on browser.

![Hole Table]

Insert a hole table for a view or multiple views of a common, singular part (and not an entire Part Studio). Style the table as you would any other table, and hover over a table row for cross-highlighting in the drawing.
Steps

1. Click ✅

2. Check the box to indicate that you want the location data shown in the selected view(s).

3. Indicate a precision for the location coordinates.

4. Select a view to register its holes in the table. To select more than one view, click in the Views field again and then select another view. Repeat as necessary until all views are selected.
A table preview appears when you select the tool, and is populated as you select views to be represented in the table.

Note that views are only selectable when the drawing is made of a part and not an entire Part Studio.

5. Select the origin from which to locate the holes.

6. Select the X or Y axis for the hole location by clicking an edge in the drawing.

7. Select any holes you want excluded from the table (by clicking them in the drawing).

8. Specify the order in which the holes are listed in the table, alpha-numerically (top to bottom or bottom to top).

9. Select the icon that matches the corner of the table to set as your fixed corner.

The default anchor point in Onshape is the upper right corner of tables.

There is no need to click directly on the point once it is visible. While moving the mouse to place the hole table, you’ll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the hole table to; simply click when you see the line appear to align the hole table to that line.

10. Click in the drawing space to place the Hole table.

Hover over a row in the table to see cross-highlighting in the drawing:
Editing Hole tables and tags

Edit hole tag

Right-click on an axis (seen highlighted below) for the hole in a view to access the associated context menu. The context menu for a hole in a view is:

- **Hole table properties** - To open the properties panel for the hole table.
- **Flip X direction** - Flip the direction of the X axis arrow.
- **Flip Y direction** - Flip the direction of the Y axis arrow.
Clear selection - Clear selected items.
Zoom to fit - Zoom the drawing to fit the window.

Edit hole table

You can edit a hole table like any table in drawings. Click the edge of the table to select the entire table; grab one of the corner grab points to move the table, drag one of the middle grab points to resize the table. Right-click the table to access the context menu with the following commands:

Hole table properties - To open the properties panel for the hole table.
Split hole table above/below - To split the hole table below the row the cursor was in when you right-clicked for the context menu. This splits the table into two tables. You can then select one of the tables, right-click for the context menu and use Move to sheet to move that table to another sheet in the drawing.

Move to - Border frame, Border zones, Title block, Drawing
Bring to front - Bring the selected table to the forefront of the display
Send to back - Send the selected table to the background of the display
Move to sheet - To move the table (or a split table) to another, existing, sheet of the drawing.
Clear selection - Unselect the currently selected table
Zoom to fit - Zoom the drawing to fit in the window
Delete - Delete the currently selected table

Formatting

Double-click in a table cell to open the cell formatting panel:

You can click on the mid-points of a table to automatically size to fit the contents of the table.

Deleting a table
Right-click the table and select Delete.

**Hole table properties**

Select the table, then right-click and select Hole table properties to open the Properties dialog for that table:

![Hole table properties dialog]

You can indicate whether or not to show the location coordinates in the drawing views, and also whether or not to collapse the tags in the table.

You can also change any other specifications made for the hole table including which views are selected, the order of the tags in the table, and which corner of the table is fixed.

Click the checkmark to register your edits.

---

**Revision Table**

This functionality is currently available only on browser.

Insert a revision table to record the revisions of the drawing created during the [release management](#) process. Style the table as you would any other table and adjust...
properties of the table through the Properties panel. As the drawing moves through the release management process, further revisions and table information is updated automatically.

Steps

1. Click

![Revision table panel]

2. Use the Multi-sheet field to indicate how to display the table when the drawing contains multiple sheets. The Linked option displays the full Revision table on all sheets. The See sheet 1 option displays the full table on the sheet the table was inserted on and a small table indicating "See sheet 1" on every other sheet.
The first sheet of the drawing, with the revision table, above
3. Set the Visible rows field to the number of revisions you wish to list in the table.

4. Select the Order in which to list the revisions: newest at the bottom (Top to bottom) or newest at top (Bottom to top).

5. Select Hide revision callouts no longer shown in the table to hide any revision callouts you have inserted in the drawing that do not have a corresponding revision row in the Revision table.

6. Select the icon that matches the corner of the table to set as your fixed corner.

<table>
<thead>
<tr>
<th>Select fixed corner:</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Icon options]</td>
</tr>
</tbody>
</table>

The default anchor point in Onshape is the upper right corner of tables.
There is no need to click directly on the point once it is visible. While moving the mouse to place the table, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the table to; simply click when you see the line appear to align the table to that line.

7. Click in the drawing space to place the table.

**Editing Revision tables**

Right-click on the table and select Revision table properties to open the properties dialog for the table. Make changes as necessary and click to accept your changes.

When editing a pending revision (a revision marked with an asterisk (*) in the table), the contents of the Description cell of the Revision table for that release becomes available through the Release properties dialog when viewing the Revision history of the release.

**Formatting**

Click in a table cell to open the cell formatting panel:

![Cell Formatting Panel]

Make changes as desired.

You can click on the mid-points of a table to automatically size to fit the contents of the table.

**Editing column headers and title of table**

Double-click a column header or the title of the table in order to edit the text as well as open the formatting panel. Once all changes are satisfactory, click the checkmark to close the formatting panel, or use the x to close without saving changes.

**Deleting a table**

Right-click the table and select Delete.

**Adding revision callouts to the drawing**
To add revision callouts to a drawing, first a Revision table must be inserted into the drawing. Right-click in the drawing space, or right-click a row in the Revision table, and select Add revision callout:

![Table showing revision A and B*]

Click in the drawing space to place the callout. The callout can be used to indicate changes made to the drawing since the previous revision. As shown below, the asterisk listed next to the revision label, B*, indicates the next revision, B, has not yet been released. This suffix (*) is customizable through the Release management settings, under Revisions and Part numbers. See "Setting up Release Management" on page 2174 for more information on customizing the unreleased revision suffix.

![Dimension with B* revision callout]

To hide callouts, right-click on the row in the Revision table for which to hide the callouts, and select Hide revision callouts. The right-click on the row highlights those revision callouts in the drawing. Clicking the command hides those revision callouts in the drawing. To show the callouts again, right-click on the table row and select Show revision callouts.

To group a callout with a dimension, place the dimension first, then when placing the callout, drag it to the dimension until you see the highlight activate. You can place the
callout anywhere and it will be grouped with the dimension and move when the dimension is moved:

![Callout Diagram]

**Setting Revision table default properties**
Open the Properties panel, select the Table tab, adjust the properties for Revision tables. See "Properties" on page 1775 for more information.

**Drawing Tools**

This functionality is currently available only on browser.

Onshape provides tools for creating sheet geometry: drawing entities like lines and centerlines, created on the sheet outside of a view and meant to represent some part of the 3D model.

**Edge-to-edge centerline**
Create centerlines using two edges or two concentric arcs on your drawing.
1. Click.

2. Select two edges or two concentric arcs with which to establish a centerline.

Adjust the length of the centerlines by clicking and dragging the grip points at the end of the centerline.

Removing centerlines
1. With no tool selected, click the centerline (it appears highlighted).

2. Press the Delete key.

Modifying centerlines
1. With no tool selected, click the centerline (it appears highlighted).
2. Click and drag an end point to resize the line:

![Line Resize Diagram]

Note that centerlines may be dragged below the distance between the reference points.

3. Click and drag a snap point to move the line:

![Snap Point Movement Diagram]

Create centerlines using two points on your drawing, including the end points on another 2-point linear centerline.

1. Click.

2. Select two points to establish a centerline. Note that you are able to use snap points, but it is not required.
**Edge-to-edge centerline**

Create centerlines using two edges or two concentric arcs on your drawing.

1. Click 🔄.

2. Select two edges or two concentric arcs with which to establish a centerline.

Adjust the length of the centerlines by clicking and dragging the grip points at the end of the centerline.

Removing centerlines

1. With no tool selected, click the centerline (it appears highlighted).

2. Press the Delete key.
Modifying centerlines

1. With no tool selected, click the centerline (it appears highlighted).

2. Click and drag an end point to resize the line:

   ![Diagram of centerline resizing](image)

   Note that centerlines may be dragged below the distance between the reference points.

3. Click and drag a snap point to move the line:

   ![Diagram of centerline moving](image)

**3 point circle centerline**

Create a circular centerline for a bolt circle diameter.

1. Click 🩣.

2. Click each of 3 points (centers of the holes, end, mid, or quad points). The first illustration shows the centerline in process:
The illustration below shows the centerline selected; you can see which holes help define the centerline:

2 point circle centerline
Create a circular centerline using two points.

1. Click 🔄.

2. Click a point to mark the center of the centerline (this does not have to be an actual circle center, you are able to snap to any point like an end point or midpoint as well).

3. Click a point to mark the circumference of the centerline (like the center of a bolt hole).

The first illustration shows the centerline in process (the orange highlighting represents the selected points):
You are now able to dimension the centerline.

**Centermark**

Place a mark in the centers of circles and arcs for visibility when printing and as a reference point for dimensions.

1. Click.
2. Click the edge of a circle or arc:

To delete a centermark, click to select and press the Delete key.

**Virtual sharp**
Create a virtual sharp associated with two linear edges.

1. Click 🟡.
2. Select first linear edge.
3. Select second linear edge.

Dimension to the intersection of the cross only. To change the visual style of the virtual sharp from Centermark to Edge extension, open the Properties flyout:
### Properties

**Dimensions**
- **Text alignment**
  - Align with dimension line: Horizontal
- **Text size**
  - 0.18

**Dual dimensions**
- **Top**
- **Bottom**
- **Show dual unit**

<table>
<thead>
<tr>
<th>Primary</th>
<th>Dual</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length units</td>
<td>Inches</td>
</tr>
<tr>
<td></td>
<td>Millimeters</td>
</tr>
<tr>
<td>Length precision</td>
<td>0.1234</td>
</tr>
<tr>
<td>Length tolerance precision</td>
<td>0.1234</td>
</tr>
<tr>
<td>Angular precision</td>
<td>0</td>
</tr>
</tbody>
</table>

**Decimal separator**
- Period
- Comma

**Length leading zeros**
- On

**Text gap**
- 0.09

**Geometry gap**
- 0.06

**Extension past line**
- 0.18

**Font**
- ARIsopol

**Virtual sharps**
- Centermark
- Edge extension

**Weld standard**
- ANSI
- ISO

**Format**
- Locked: Unlocked
Shortcut: L

Create lines in your drawing.

1. Click .
2. Click to begin the line.
3. Drag and click to define subsequent line segments.
4. Escape to end the line and exit the tool.

Note that horizontal and vertical inferencing lines appear as appropriate:

```
______________________ X
```

Each segment in a series of connected lines is a separate entity.

As you draw, snap points appear on existing objects to aid you in line placement.
Click once the snap points appears to connect to it automatically.

Create a spline through multiple points.

1. Click .
2. Click to begin the spline.
3. Click to select additional points for the spline to fit to in the view.
4. Double-click or press Escape to end the spline.

As you draw, snap points appear on existing objects to aid you in spline placement.
Click once the snap points appears to connect to it automatically. As with any spline, you can drag the points to reshape the spline.

Add points along a spline.
1. Click

2. Click along the spline to set additional points.

3. Drag and click to define subsequent line segments.

4. Press Escape to exit the tool.

You are able to drag spline points to modify the spline.

As you draw, snap points appear on existing objects to aid you in point placement. Click once the snap points appears to connect to it automatically.

---

**Insert DWG and DXF Files**

This functionality is currently available only on browser.

Insert the contents of a DXF or DWG file as Onshape drawing entities. Use this tool as a method to create a custom template; for more information see "This functionality is currently available only on Onshape's browser platform." on page 1753.

1. Click

2. In the dialog that appears, enter a search phrase into the *Search files* box to locate a file, or select one from the list.

You are also able to click Other documents in the dialog to browse for a document that has an image file already uploaded. Inserting a file from another Onshape document (that you own or has been shared with you) copies the file.
If there is no file listed, use the Import option at the bottom of the dialog to bring one into the document.
**Tips**

- Contents are imported from model space and copied to the drawing.
- Only wireframe geometry (lines, arcs, polylines, etc) and notes from model space are imported (no 3D data).
- All colors are removed and the default color in the Onshape drawing is applied (the appropriate layer color). To change the background color:
  - Navigate to My account > Preferences > Drawings, and select Dark or Light. Click Save drawing settings.
- All blocks are exploded.
- All polylines are exploded.
- All simple notes are converted to Notes and are editable.
- Use the Move to command on the View’s context menu (right-click) to add elements to: Title block, Border frame, and Border zones.
Elements will be added to the corresponding layer: title block, border frame, or border zones.

Insert Image

This functionality is currently available only on browser.

You can insert and scale JPG, PNG, and GIF images on a drawing sheet, such as a logo and QR and bar codes.

1. Click 📷.

2. In the dialog that appears, enter a search phrase to locate an image file, or select one from the list.

You also have the ability to click Browse documents in the dialog to browse for a document that has an image file already uploaded. Inserting an image from another Onshape document (that you own or has been shared with you) copies the image.
If there is no image listed, use the Import option at the bottom of the dialog to bring one into the document.

3. Click and drag to position the image in the graphics area. (The aspect ratio of the image is maintained.)

4. Click and drag to reposition the image.
There is no need to click directly on the point once it is visible. While moving the mouse to place the image, you'll notice thin, dashed lines as the cursor passes near other entities. These are inferencing lines that you are able to align the image to; simply click when you see the line appear to align the image to that inferencing line.

**Tips**

- Images are inserted without visible borders. To show (or hide) borders for all images in the drawings at once, use the image context menu (Show image borders/Hide image borders).

- Right-click to access the Properties for the image, where you can specify on which sheet to include the image, as well as view the references if the image has been inserted from another document.

| Image properties |  
|------------------|---|
| **Document**     | rel-1.121 |
| **Version**      | V2       |
| **Reference**    | flare-icon.png |
| **Sheet**        | Sheet1   |

- Open the document from which an image or drawing file (image or .DXF/.DWG) is inserted: right-click on the image and select Open linked document.

- Deleting an Image tab from a document results in the image being removed from the drawing upon the next update.

- Inserting an image in a drawing that has been imported via the Documents page creates a non-linked image. Deleting the image from the Documents page and then updating the drawing workspace does not remove the image from the drawing.

**Updating a Drawing**

Shortcut: Ctrl-q
This functionality is currently available only on browser.

When an underlying Part Studio or Assembly of a drawing is changed, the drawing may need to be updated, as indicated by the active (the inactive icon is grayed out). Changes that trigger this condition may be seemingly insignificant, like moving a sketch dimension or hiding a construction plane in a Part Studio. To understand better why this button is active, check the History of the document to view recent changes.

Note that this action only updates drawing views, nothing else in the drawing or document, and does not check for updated links to other documents.

**Steps**

1. Click ⏻.
2. Refresh your browser to regenerate the drawing.
3. Check the drawing for any issues.

At times, the update might not work seamlessly and an added entity (a dimension, for example) may turn red because it might be dangling (or broken). See [Dangling Entities](#) for more info.

**Tips**

- Given that a drawing may need to be updated as a result of a small change in a document (see above), you may want to 'lock down' a drawing so the Update button will not highlight. Simply version the document: this freezes the drawing in its current state and you can then mark the drawing's state as Released in the version.

- A drawing of a part from a workspace will show that it needs updating whenever the part in the workspace changes. However, you can work on a drawing without updating it. Simply update whenever you wish to incorporate the current state of the part.
The performance while opening a drawing (and the document containing the drawing) is significantly faster when the drawing references either a version or a revision of a part.

To reference a version or revision, right-click on the drawing tab and select Change to version. Then select a version (or revision) to change to.

The increase in performance when using a version or revision is significant and still allows you to update the drawing to newer versions and revisions as they become available.

If a drawing is out-of-date at the time a new view is placed on the drawing, the drawing will automatically update.

---

**Dangling entities**

This functionality is currently available only on browser.

A dangling entity in a drawing is an entity that has lost one (or more) of its reference points. Dimensions, section lines, note leaders, callouts, and geometric tolerances are all examples of entities that could become dangling (broken) after a drawing update. This is to be expected, especially if the change to the part or assembly was significant.

A dangling entity is indicated by a red grip point indicating which reference point has been lost.
To fix a dangling entity

Assign a new reference point where one is missing. Click and drag the red grip point away from its location and onto a new reference point.

---

**Importing a Drawing**

This functionality is currently available only on browser.

Drawings are able to be imported from:

- The Documents page where they automatically become a new document.
- Within a document, where they automatically become a tab inside that document.
Importing from Documents page

1. Click \( \rightarrow \).
2. Select the file to import.

Onshape automatically translates the file to the proper format and creates a new Onshape document using the imported file name as the document name.

Importing from within a document

1. Click \( + \).
2. Select Import...
3. Select the file to import.

Onshape automatically translates the file to the proper format and creates a new tab for the file, as well as a drawing, using the file name as the file tab name and the drawing tab name.

To set the background color of model space for imported DWG and DXF files go to Account settings > Preferences, and specify a background color under the Drawings section. Click Save drawing settings when done. See Managing Your Onshape Account for more info.

Exporting a Drawing

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

You have the ability to export Onshape drawings to the following file types:
- PDF
- DWG
You have the ability to export Onshape drawings to the following file types:

- PDF
- DWG
- DXF
- DWT (see, "Exporting a drawing to a template" on page 1758)

The export function presents the opportunity to select the desired format; when the translation is finished, the file is also downloaded to your local machine.
1. Right-click on the Drawing tab.
2. Select Export.
3. Specify a name for the export file.
4. Select the desired export format:
   - .DWG
   - .DXF
   - .DWT
   - .PDF
5. If exporting to DXF or DWG, select the version and sheets. If exporting to PDF, select whether to export text normally, or as selectable text.
6. Choose how to treat overridden dimensions: Show underlines or Hide underlines.
7. Indicate which drawing sheets to export, if applicable.
8. Indicate whether to export splines as polylines, available for all formats except PDF.
9. Indicate whether to explode the text upon export, if desired.
10. Select what to do with the export file:
   a. Download the file only
   b. Download the file and store the file in a new tab in the document
   c. Email with file download link
   d. Store the file in a new tab only.
11. For DWG, DWT, and DXF exports, you can check the box to Set z-height to zero and normals to positive. This sets the z-height/direction of any hole normals to be positive (+1) for drawings with view geometry.

**Tips**
- When you initiate the export, a blue message bubble appears at the top of the window:
You also get a notification when the export is complete:

- Drawings exports are simplified output that is readable by most DWG readers.
- To export notes as polylines, you can also use the Explode command on the Note context menu.
- Image properties are not available for embedded images on imported drawings.
- To export a drawing with notes containing strike-through text, select Version 2013 in the dialog.

**Exporting a Drawing: iOS and Android**

In Mobile, you have the ability to export Onshape drawings to the following types of files:

- DXF
- DWG
- DWT
- PDF

The export function presents the opportunity to select the desired format; when the translation is finished, the file is also downloaded to your device.

**Steps**

1. Tap the overflow menu in the drawings tab
2. Tap Export

Use the arrows to the right of each subject to change your export preferences.

3. When you are finished choosing your preferences, tap Export.

Printing a Drawing

This functionality is currently available only on browser.
You have the ability to print your drawings:

1. Expand the Document menu in the top left corner of the interface.
2. Select Print...
   A new tab opens with a print-friendly format of your drawing.
3. Use the controls at the bottom of the window to print or save the drawing.
Feature Studios

This functionality is currently available only on Onshape's browser platform.

A Feature Studio is a tab containing FeatureScript, a programming language used to define your own custom features in Onshape.

FeatureScript is designed by Onshape for writing features, and more generally, working with 3D parametric models. The language is built into Onshape from the ground up, providing the foundation of Part Studio modeling and used to define Onshape standard features (like Extrude, Fillet and Shell).

For detailed information on how to use FeatureScript to create custom features, see Welcome to FeatureScript (opens in new tab).

For detailed information on using custom features within your Onshape account, see Custom Feature.

To customize the toolbar of Part Studios, Assemblies, or Feature Studios, see "Document Toolbar and Document Menu" on page 134.
Importing and Exporting Files

This functionality is available on Onshape's browser, iOS, and Android platforms.

You have the ability to import and export many types of files, not only CAD files, from your hard drive and cloud based sources (such as Google Drive and Dropbox). When importing CAD files, Onshape automatically rewrites it to Onshape's internal format. When exporting files, you are able to export to another CAD format, as well as simply download non-CAD files. For more information, see the links below.

**Importing & Exporting Files: Desktop**

- **Import** - Load any type of file into Onshape, either as its own document, or into an existing document. If the file is a CAD file, it will be automatically converted to Onshape format.

- **Download** - Copy any file that was imported into Onshape back out of Onshape in its current file format to your local machine.

- **Export** - Write an Onshape Part Studio or individual part to another CAD format, or a sketch or planar face to DWG/DXF format and download it to your local machine.

Read more on [Supported file formats](#).

To learn more about importing and exporting files in Onshape, follow the self-paced course here: [Importing and Exporting Data](#) (opens in new tab).

**Importing & Exporting Files: iOS and Android**

You have the ability to import and export many types of files, not only CAD files. When importing CAD files, Onshape automatically rewrites it to Onshape's internal format. When exporting files, you are able to export to another CAD format, as well as simply download non-CAD files. For more information, see the links below. Note that exporting files is available only on the browser/desktop version of Onshape.
Please note that Onshape mobile currently supports importing files from cloud-based sources only.

- **Import** - Load any type of file into Onshape, either as its own document, or as a new tab in an existing document. If the file is a CAD files, it will be automatically translated to Onshape format.

- **Download** - Copy any file that was imported into Onshape back out of Onshape in its current file format to your device.

- **Export** - Write an Onshape Part Studio or individual part to another CAD format, or sketch or planar face to DWG/DXF format and download it to your device.

Read more on [Supported File Formats](#).

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### Supported File Formats

This functionality is available on Onshape's browser, iOS, and Android platforms.

#### Import formats

Onshape automatically imports and translates these CAD file formats for parts and assemblies, including wire bodies and sketch curves.

For more on importing parts and assembly files see, "Importing MCAD files" on page 1985.

Part files

- Parasolid B-rep (.x_t or .x_b) from v10 to v33

  Parasolid is the Onshape preferred import format.

- Parasolid mesh (.xmt_txt or .xmt_bin) from v28 to v29 (view and reference meshes only, unable to edit a mesh)

- ACIS (.sat) up to R21, 2018 1.0
• STEP (.stp or .step) AP203, AP214, and AP242 (geometry only)
• IGES (.igs or .iges) up to 5.3
• CATIA v4 from 4.15 to 4.24 (via an Assembly .zip file)
• CATIA v5 from R7 to R30 (v5-6R2020)
• CATIA v6 R2010x to R2013x, R2015x, R2016X, R2020X
• SOLIDWORKS (.sldprt) 1999 to 2019
• Inventor 9 up to 2021
• Pro/ENGINEER, Creo from Pro/E 2000i to Creo Parametric 6.0
• Creo 7.0
• JT (.jt) up to 10
• Rhino (.3dm) up to 7
• STL (.stl) (view and reference meshes only, unable to edit a mesh)
• OBJ (.obj) (view and reference meshes only; unable to edit a mesh)
• NX UG15.0 through NX12.0.2.9
• NX UG 1911, NX 1915, NX 1919, NX 1926 through 1934
  NX1847, latest version NX 1847.2400, and NX1851
• Solid Edge (.par and .psm) 10 through 2019

Importing Solid Edge sheet metal files (.psm) will not result in sheet metal models in Onshape. You have the ability to reference imported faces in the Sheet metal model and use the Thicken option to create an Onshape sheet metal model.

Assembly files
• Parasolid B-rep (.x_t or .x_b) from v10 to v33
• ACIS (.sat) up to R21, 2018 1.0
• STEP (.stp or .step) AP203, AP214 and AP242 (geometry only)
• CATIA v4 from 4.15 to 4.24 (via a .zip file)
• CATIA v5 from R7 to R30 (v5-6R2020) (via a .zip file)
• CATIA v6 R2010x to R2013x, R2015x, R2016X, v6 2020X (via a .zip file)
• SOLIDWORKS as Pack & Go .zip files from 1999 to 2019 (via a .zip file)
• Inventor (.iam) 9 up to 2021
• Pro/ENGINEER, Creo from Pro/E 2000i to Creo Parametric 6.0 (via a .zip file)
• Creo 7.0
• JT (.jt) up to 9.0
• Rhino (.3dm) up to 7
• NX UG15.0 through NX12.0.2.9 (via a .zip file)
• NX UG 1911, NX 1915, NX 1919, NX 1926 through 1934
• NX1847, latest version NX 1847.2400, and NX1851
• Solid Edge 10 through 2019 (via a .zip file)

For all assembly files, you must place all parts and the assembly in a zip file and import the zip file into an Onshape document. The zip file name must match the name of the top-level assembly. For more information, see "Importing Files" on page 1975.

Drawing files
• AutoCAD (.dwg) up to 2018
• DXF (.dxf) up to 2013, 2018
• DWT (.dwt) 2013, 2018

Non-CAD files
Onshape supports a number of common and native CAD file formats for both import and export. You are also able to import other, non-CAD files into Onshape for reference (for example, viewing or storage only).

Non-CAD files that are able to be imported into Onshape include:
• .pdf
• .mp4
Files that are able to be uploaded but not viewed include:

- Any programming language files (py, json, java, js)
- .mov (this may depend on browsers and allowed plugins)
- .csv
- .doc/docx

**Export formats**

Onshape exports parts, Part Studios, Assemblies, Drawings, and tabs containing other imported CAD files to these CAD formats.

For more on exporting see, "Exporting Files" on page 2013.

Parts and Part Studios

- Parasolid B-rep (.x_t or .x_b) from v25 to v33
- Parasolid mesh (.xmt_txt or .xmt_bin) from v28 to v29
- ACIS (.sat) R21
- STEP (.step) AP214 (geometry only)
- STEP 3D AP242 E2
- IGES (.igs or .iges) 5.3
- SOLIDWORKS (.sldprt) 2004
- STL
- GLTF
Rhino 5 (.3dm)

Collada (.dae) 1.4.1 without joints data (with meters as default units)

Assemblies

Parasolid B-rep (.x_t or .x_b) from v25 to v33

ACIS (.sat) R21

STEP (.step) AP214 (geometry only)

IGES (.igs or .iges) 5.3

STL

GLTF

Collada (.dae) 1.4.1 without joints data (with meters as default units)

Drawings


DWT Template (.dwt) 2013, 2018

PDF

Importing Files

This functionality is available on Onshape’s browser, iOS, and Android platforms.

You can import (upload) any type of file into Onshape, either into an existing and open document, or as its own document (from the Documents page). For a list of supported files, see "Supported File Formats" on page 1971. You can import files from your desktop and also from integrated Dropbox, Google Drive, or Microsoft OneDrive accounts. See "Integrations" on page 2483 for more information on integrating cloud apps with your Onshape account.

Importing Files: Desktop
The Onshape limit for uploads is 2GB per file.

Note that CAD files imported into Onshape from another system do not have a feature tree in Onshape; Onshape provides tools for performing parametric features on simple solids instead. For more information, see "Working with Imported CAD" on page 1996 and the blog post Making Supplier Models Work for You (opens in new tab).

When importing data into Onshape, there are two things to think about:

- The origin of the data
- Where in Onshape you want the data to reside

Where Onshape imports your data depends upon where you initiate the import:

- **To import data into a brand new document, import from the Documents page** -
  
  Click ![Create button](image) and then select **Import** to create one or many new Onshape documents with appropriate tabs from the imported file. The document is given the same name as the file you are importing. The document is listed on the Documents page and a version is automatically created. For more specific instructions and information, see "Importing from the Documents page" on page 1979.

- **To import data into an existing document, import from within an open document** -
  
  Open a document, click the plus sign at the bottom left, then Import. This creates new Onshape tabs in the active document; the tab names reflect the file name, and the file name is reflected by the icon. For more specific instructions, see "Importing from within a document" on page 1983.

The origin and format of the data comes into play with how Onshape translates it:

- Data from most systems imports via a universal format like ACIS, JT, STEP, IGES, Parasolid, OBJ, or STL. For AutoCAD solids the best import format is ACIS, and all 2D data is imported to DWG or DXF.

When importing a SOLIDWORKS Pack and Go file, zip the file and give it the exact same name as the top level assembly. The zip file must be flat-packed, that is, have no folder structure.

**Processing CAD files**
When importing files into Onshape, the following types of files can be imported directly from one system into Onshape:

When Onshape recognizes an imported file as a CAD file (based on its file extension), Onshape automatically presents processing options. Onshape checks the zip file for supported assembly files with the same name as the zip file. When zipping assembly files for import into Onshape, you have the option to zip the files individually, or in an entire directory.

Place all parts and the assembly in a zip file and import that into Onshape. Keep in mind that when zipping an entire directory, the zip file must have the same name as the top-level assembly (minus the extension) and you must not rename the zip file. You must ensure that the file extension is only .zip and does not include the old extension (such as .sldasm.zip or .asm.zip).

When importing a CAD file into Onshape (any Pack and Go .zip file), you have the following processing options:

- **Import all assemblies and parts to a single document** - Part Studio and Assemblies are created as needed, depending on the contents of the imported file and named according to the names in the imported file.

- **Split into multiple documents** - If the file is an assembly, this option creates a separate document for each part or subassembly and places those documents in a new folder whose name is the imported file name. The data structure and its links are preserved.

- **Combine to a single Part Studio** - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio. In this case, the assembly is stored as a set of parts in a single Part Studio. There will be duplicate parts created whenever a part is instanced more than once. Be aware that this choice should only be made if the assembly is small, otherwise, performance will be affected.

Regardless of which option you choose, above, Onshape allows import of parts with faults. If a part doesn’t pass Onshape validation, it is still imported with faults. Faults are indicated in the Feature list and the Parts list by a red name, and also in the Noti-
fications messages. Note that if some geometry in the imported part is bad, some downstream operations may fail.

If Onshape detects that a part may be fractured upon import, then all surfaces and curves that belong to the same (shattered) body are automatically turned into a single closed composite part during the import process.

Also, you have the ability to manually create a composite part when importing a neutral CAD file (.step or .x_t for example) consisting of multiple non-solid bodies, with the option explained below.

- **Create a composite part when importing multiple or non-solid bodies** - When this option is checked, all parts in the imported Part Studio become a single closed composite part, regardless of whether or not the parts belong to the same body. The bodies are combined to form the composite (one single part).

If you miss this option during the import process, after import you can double-click the Import feature in the Part Studio and select Create composite to create the composite part at that time:
Onshape's **up axis** is the Z axis. For CAD files from systems where the up axis is the Y axis, you can elect to have the models imported with the Y axis up; otherwise they are imported with the Z axis up. Note that you can change the up axis from within the Onshape document post-import as well.

Don't worry if you miss the option during the import process, you can open the Import feature in the Part Studio and check the box there:

- **Orient imported models with Y Axis Up** - If the file was created in a system that orients with Y axis up, the models would by default be brought into Onshape (a Z axis up system) with a flipped coordinate system. Check this box to reorient the axis and display the model with the coordinates you expect.

The automatic processing happens only for files that Onshape can translate. All other files are simply imported into a tab.

**Importing from the Documents page**

1. Click ![Create](create.png).

2. To input from you hard drive, click ![Import files](import_files.png). The file explorer opens on your local machine.

To input from your Google Drive ![Google Drive](google_drive.png) or Dropbox ![Dropbox](dropbox.png), click ![Import from](import_from.png).

If you have already integrated your applications, click on your cloud-source. If you have not yet integrated a cloud-source, click **Connect service...**; a wizard
This will lead you through the integration process. See integrations for more information.

3. Select a file (or files) to import.

If you belong to a company plan, Onshape prompts for the desired owner of the document: select yourself or the company.

4. Select your desired processing option:

- **Import all assemblies and parts to a single document** - Part Studio and Assemblies are created as needed, depending on the contents of the imported file and named according to the names in the imported file.

- **Split into multiple documents** - If the file is an assembly, this option creates a separate document for each part or subassembly and places those documents in a new folder whose name is the imported file name. The data structure and its links are preserved.

  - If the file only contains multiple parts (and no assembly), each part will be put into its own document, and a top level assembly will be created that links to those parts.
- Combine to a single Part Studio - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio. In this case, the assembly is stored as a set of parts in a single Part Studio. There will be duplicate parts created whenever a part is instanced more than once. Be aware that this choice should only be made if the assembly is small, otherwise, performance will be affected.

Regardless of which option you choose, above, Onshape allows import of parts with faults. If a part doesn’t pass Onshape validation, it is still imported with faults. Faults are indicated in the Feature list and the Parts list by a red name, and also in the Notifications messages. Note that if some geometry is bad, some downstream operations may fail.

Onshape's up axis is the Z axis. For CAD files from systems where the up axis is the Y axis, you have the option to elect to have the models imported with the Y axis up; otherwise they are imported with the Z axis up. Note that you are able to change the up axis from within the Onshape document post-import as well.

- Orient imported models with Y Axis Up - If the file was created in a system that orients with Y axis up, the models would by default be brought into Onshape (a Z axis up system) with a flipped coordinate system. Check this box to reorient the axis and display the model with the coordinates you expect.

- Create a composite part when importing multiple or non-solid bodies - If the file contains multiple bodies or non-solid bodies, you can elect to have them imported as a composite part. See "Composite Part" on page 1311 for
more information.

5. Click **OK** to initiate the import.

   The automatic processing happens only for files that Onshape has the ability to translate. All other files are simply imported into a tab.

   Onshape displays a list of recently imported files.

6. Now you have the ability to:
   - Click on the file name in the import list to immediately view the file in Onshape, through the document that was automatically created for it.
Upon hovering, click the X on the right side of the Import list to close it and return to the Documents page. The document just created is listed on the Documents page.

**Importing from within a document**

1. Once in an open document, at the bottom of the page, click and select Import.
2. The file explorer opens on your local machine; select a file to import.
3. In the Import dialog make your import type selection:

   ![Import to Onshape dialog]

   - **Import to this document** - Part Studio and Assemblies are created as needed, depending on the contents of the imported file and named according to the names in the imported file.
   - **Combine to a single Part Studio** - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio. In this case, the assembly is stored as a set of parts in a single Part Studio. There will be duplicate parts created whenever a part is instanced more than once. Be aware that this choice should only be made if the assembly is small, otherwise, performance will be affected.
Regardless of which option you choose, above, Onshape allows import of parts with faults. If a part doesn’t pass Onshape validation, it is still imported with faults. Faults are indicated in the Feature list and the Parts list by a red name, and also in the Notifications messages. Note that if some geometry is bad, some downstream operations may fail.

Onshape's up axis is the Z axis. For CAD files from systems where the up axis is the Y axis, you can elect to have the models imported with the Y axis up; otherwise they are imported with the Z axis up. Note that you can change the up axis from within the Onshape document post-import as well.

- **Orient imported models with Y Axis Up** - If the file was created in a system that orients with Y axis up, the models would by default be brought into Onshape (a Z axis up system) with a flipped coordinate system. Check this box to reorient the axis and display the model with the coordinates you expect.

- **Create a composite part when importing multiple or non-solid bodies** - If the file contains multiple bodies or non-solid bodies, you can elect to have them imported as a composite part. See "Composite Part" on page 1311 for more information.

The automatic processing happens only for files that Onshape can translate. All other files are simply imported into a tab.
4. Click **OK** to initiate the import.

You will then receive a notification telling you the status of the import.

The imported file is now in the open Onshape document, as its own tab (listed across the bottom of the document page).

In addition to being able to write directly to and from the Onshape format, you also have the ability to write to and from any of the Onshape supported formats. For supported formats, see "Supported File Formats" on page 1971.

**Importing MCAD files**

These instructions explain the nuances of importing SOLIDWORKS, Inventor, Pro/Engineer and Creo, NX, and Solid Edge parts and assemblies files.

**Assemblies files**

1. Create a .zip file of your top-level assembly.
   
a. Include all parts and subassemblies.
   
b. Flatten the file structure so there are no folders in the .zip file.
   
2. Ensure the top-level assembly and the .zip file have the same name.
   
3. Import the entire .zip file into Onshape.

For SOLIDWORKS assembly files, Onshape needs all of the parts and sub-assembly files alongside the top level .sldasm file. You can use the Pack & Go tool to create a .zip file of your top-level assembly, or follow the steps above. Additionally, Onshape and SOLIDWORKS both run on the Parasolid modeling kernel and saving a SOLIDWORKS assembly as a Parasolid (.x_t) file is another way to import into Onshape.

**Parts files**

There are no special workflows required to import Pro/Engineer, Creo, Inventor, SOLIDWORKS, NX, or Solid Edge parts files into Onshape. Simply select the part file when prompted by the Onshape import dialog.

**Saving a SOLIDWORKS part as a Parasolid (.x_t) file is another way to import it into Onshape.**
Importing Non-CAD files

Files that do not contain any CAD data can also be imported into Onshape: PDFs, image files, literally any kind of file. Onshape handles them the same way as CAD files: when you import from the Documents page, the imported file is placed in a new document; when you import from within a document, the imported file is placed in a tab within the document. When there is no data to display, Onshape displayed information about the file, such as:

![intakeform_050105.doc](image)

*If the file changes after you import into Onshape, use the Update button to update the file. If you wish to download the file, use the Download button.*

Importing Files: iOS

You have the ability to import (upload) any type of file into Onshape, either into an existing and open document, or as its own document (from the Documents page). For a list of supported files, see [Supported File Formats](#).

Onshape mobile currently supports importing files from cloud-based sources only. The cloud-based storage app that you want to use must be installed on your mobile device and you must be signed into it, in order for it to incorporate with Onshape.

How Onshape handles your import depends upon where you initiate the import: from the Documents page or within a document.

**From the Documents Page**

Creates a new Onshape document and appropriate tabs; the document is given the
same name as the file you are importing, as are the tabs.

Steps:

1. Tap the import button in the lower right corner of the Documents Page.

2. Select where you want to import the file from.

   Onshape supports importing files from cloud-based sources such as iCloud and Dropbox, as well as your device camera and photo library.

   If you do not see your preferred cloud-based source, tap More and check the settings that follow.

   Depending from where you would like to import a file, you may have to manually enable the ability to import from your device storage in your device settings.

3. The imported file will appear as a document in the Documents page. Specify the privacy of the document and, if you are importing a CAD file, specify import settings. For more info see Import settings below.

4. The imported file appears as a new document in your Documents list. The name of the new document matches that of the file you imported.

   Note that the Import dialog does not refresh files. You must go to the cloud storage provider app to see an updated list of files to import.

**From within a document**

Creates new Onshape tabs in the active document; the tab names reflect the naming of the file.
Steps:

1. In a document, open the Create Tab menu and select "Import...".

2. Select where you want to import the file from: Import Files, Import from Camera, Import from Photo Library.

Import Files allows you to import from your device and cloud-based sources like iCloud and Dropbox.
If you do not see your preferred cloud-based source, tap More and check the settings that follow.

3. The imported file will appear as a tab within your document. Specify the privacy of the document and, if you are importing a CAD file, specify import settings. For more info see Import settings below.

4. The imported file appears as the appropriate tab(s) in your document. The name of the new tab (or tabs) matches that of the file you imported.

If you import a CAD file and it is successfully translated into Onshape format, a second tab is created for it.

**Import settings**

Every file imported into Onshape becomes its own tab, named with the original file name. If the file is a CAD file, the appropriate Part Studio and Assembly tabs are also created.

When importing a SOLIDWORKS Pack and Go file, the name of the zip file must exactly match the name of the top level assembly and the zip file must be flat-packed, that is, have no folder structure.

When importing a CAD file, you have the following processing options:
• **Imported models are in 'Y Axis Up' coordinates** - If the file was created in a system that orients with Y Axis Up, the models would by default be brought into Onshape (a Z Axis Up system) with a flipped coordinate system. Toggle this switch to reorient the axis system to match Onshape and display the model with the coordinates you expect.

• **Import file to Part Studios only (flatten)** - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio, by toggling on the flatten option. In this case, the assembly is flattened to a set of parts in a Part Studio. There will be duplicate parts created whenever a part is instanced more than once.

• **Allow import of parts with faults** - If a part doesn't pass Onshape validation, it can still be imported with faults. The fault is indicated in the Feature list and the Parts list by a red name, and also in the Notifications messages. The presence of this option does not indicate that a fault has been detected.

  By toggling on 'Allow import of parts with faults', the Import feature will show an error in the Features list, but you will be allowed to import and reference the bad geometry. Note that because the geometry is bad, some downstream operations may fail.

• **Create a composite part when importing multiple or non-solid bodies** - When this option is checked, all parts in the imported Part Studio become a single closed composite part, regardless of whether or not the parts belong to the same body. The bodies are combined to form the composite (one single part).

  If you miss this option during the import process, after import you can double-click the Import feature in the Part Studio and select Create composite to create the composite part at that time:

  The automatic processing happens only for files that Onshape can translate. All other files are simply imported into a tab.

**Processing CAD files**

When Onshape recognizes an imported file as a CAD file (based on its file extension), Onshape automatically presents processing options. You are also able to choose to export to another format from a context menu for an entire Part Studio (including
hidden parts), or for a particular part selected from the parts list. Onshape checks a zip file for supported assembly files with the same name as the zip file. When zipping assembly files for import into Onshape, you have the option to zip the files individually, or zip an entire directory. Keep in mind that when zipping an entire directory, the zip file must have the same name as the assembly (minus the extension) and you must not rename the zip file.

For more information, see Exporting Files.

**Importing Files: Android**

You have the ability to import (upload) any type of file into Onshape, either into an existing and open document, or as its own document (from the Documents page). For a list of supported files, see Supported File Formats.

Onshape mobile currently supports importing files from cloud-based sources only. The cloud-based storage app that you want to use must be installed on your mobile device and you must be signed into it, in order for it to incorporate with Onshape.

How Onshape handles your import depends upon where you initiate the import: from the Documents page or within a document.

**From the Documents Page**

Creates a new Onshape document and appropriate tabs; the document is given the same name as the file you are importing, as are the tabs.

Steps:

1. Tap the Plus sign button in the lower right corner of the Documents Page, and then the Import button:

   ![Plus button](image)

2. Select where you want to import the file from.
   
   Onshape supports importing files form cloud-based sources such as DropBox, Google Drive, and One Note (any provider that implements Storage Access Framework is available).
Depending from where you would like to import a file, you may have to manually enable the ability to import from your device storage in your device settings.

3. Select how you wish to import the file, tap the down arrow below the image, to the right of *Import to a single document*:

**Import to a single document** - Part Studio and Assemblies are created as needed, depending on the contents of the imported file and named according to the names in the imported file.

**Split into multiple documents** - if the file is an assembly, this option creates a separate document for each part of subassembly and places those documents in a new folder whose name is the imported file name. The data structure and its links are preserved.

**Combine to a single Part Studio** - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio. In this case, the assembly is stored as a set of parts in a single Part Studio. There will be duplicate parts created whenever a part is instanced more than once. Be aware that this choice should be made only if the assembly is small, otherwise, performance will be affected.
4. Select whether to **Orient imported models with Y Axis Up** - If the file was created in a system that orients with Y Axis Up, the models would by default be brought into Onshape (a Z Axis Up system) with a flipped coordinate system. Toggle this switch to reorient the axis system to match Onshape and display the model with the coordinates you expect.

5. The imported file appears as a new document in your Documents list. The name of the new document matches that of the file you imported.

Note that the Import dialog does not refresh files. You must go to the cloud storage provider app to see an updated list of files to import.

### From within a document

Creates new Onshape tabs in the active document; the tab names reflect the naming of the file.

**Steps:**

1. In a document, tap the icon at the bottom of the screen.

2. Tap the plus sign icon to open the Create Tab menu and select Import.

3. Select where you want to import the file from. Onshape supports importing files from cloud-based sources such as Google Drive as well as files downloaded to your device.
4. Select the file to import.

5. A folder is created inside the document, CAD Imports. The imported file has a tab in the folder. The imported data appears as the appropriate tab(s) in your document. The name of the new tab (or tabs) matches that of the file you imported.
If you import a CAD file and it is successfully translated into Onshape format, a second tab is created for it.

If you import a non-CAD file like a PDF, JPEG, or another type of binary content, if the source file changes, use the Update command on the tab context menu to update the file in the document.

**Import settings**

Every file imported into Onshape becomes its own tab, named with the original file name. If the file is a CAD file, the appropriate Part Studio and Assembly tabs are also created.

When importing a SOLIDWORKS Pack and Go file, the name of the zip file must exactly match the name of the top level assembly and the zip file must be flat-packed, that is, have no folder structure.

When importing a CAD file, you have the following processing options:

- **Imported models are in 'Y Axis Up' coordinates** - If the file was created in a system that orients with Y Axis Up, the models would by default be brought into Onshape (a Z Axis Up system) with a flipped coordinate system. Toggle this switch to reorient the axis system to match Onshape and display the model with the coordinates you expect.

- **Import file to Part Studios only (flatten)** - If the file is an assembly, or contains an assembly, you have the option to import it as only a Part Studio, by toggling on the flatten option. In this case, the assembly is flattened to a set of parts in a Part Studio. There will be duplicate parts created whenever a part is instanced more than once.

- **Allow import of parts with faults** - If a part doesn't pass Onshape validation, it can still be imported with faults. The fault is indicated in the Feature list and the Parts list by a red name, and also in the Notifications messages. The presence of this option does not indicate that a fault has been detected.

By toggling on 'Allow import of parts with faults', the Import feature will show an error in the Features list, but you will be allowed to import and reference the bad
geometry. Note that because the geometry is bad, some downstream operations may fail.

- **Create a composite part when importing multiple or non-solid bodies** - When this option is checked, all parts in the imported Part Studio become a single closed composite part, regardless of whether or not the parts belong to the same body. The bodies are combined to form the composite (one single part).

  If you miss this option during the import process, after import you can double-click the Import feature in the Part Studio and select Create composite to create the composite part at that time:

  The automatic processing happens only for files that Onshape can translate. All other files are simply imported into a tab.

**Processing CAD files**

When Onshape recognizes an imported file as a CAD file (based on its file extension), Onshape automatically presents processing options. You are also able to choose to export to another format from a context menu for an entire Part Studio (including hidden parts), or for a particular part selected from the parts list. Onshape checks a zip file for supported assembly files with the same name as the zip file. When zipping assembly files for import into Onshape, you have the option to zip the files individually, or zip an entire directory. Keep in mind that when zipping an entire directory, the zip file must have the same name as the assembly (minus the extension) and you must not rename the zip file.

For more information, see [Exporting Files](#).

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**Working with Imported CAD**

- This functionality is currently available only on browser.

- This functionality is also available on iOS and Android in a limited form.

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Designing from scratch in Onshape isn't the only way to build and refine your CAD models. If you have existing CAD data in another system, you can import that data into Onshape and continue designing here.

Onshape accepts data from many other systems, the most popular and well-translated being Parasolid files. For more information on file types accepted and how to import your data into Onshape, see "Supported File Formats" on page 1971 and "Importing Files" on page 1975.

This topic explains how to proceed and what tools are available after you've imported your existing CAD data into Onshape.

Onshape tools

Due to system differences, data imported into Onshape may translate in with faults, or no parametric history, or as a series of surfaces or parts instead of a solid or cohesive body. While you have some options for how data will be handled upon import, there remain some things beyond control. However, Onshape has tools that will help you move forward with your imported data and work with it to continue designing, sharing, and releasing designs.

The tools provided to help you work with imported CAD data and covered in this topic include:

- **Mesh** - Change units for meshes post-import into Onshape
- **Repair imported models** - Identify holes and disconnects in imported models that may require repair.
- **Create composite parts** - Create composite parts out of parts or surfaces fractured during the import process.
- **Edit imported parts with direct edit tools** - Including modify fillet, delete face, move face, and replace face
- **Update imported parts** - Update imported parts from within Onshape when the source file from another system is updated.
- **Replacing one reference with multiple references** - When parts are linked within Onshape, you can replace a referenced part with multiple parts, if desired.
Mesh

Onshape enables you to import three faceted file formats: STL, OBJ and Parasolid Mesh for visualization and referencing. Mesh points are able to be used as vertices for creating planes.

A mesh is imported into Onshape Part Studio, and shown in the Parts lists under Meshes (below Surfaces).

Note that you are able to view and reference meshes, but you are not able to edit them.

You can change the mesh units during the import process and from the units drop-down within the tab menu, shown in the second image below:

During import, above
Within the imported tab, above

What you can do

Once a mesh is imported into an Onshape Part Studio, you have the ability to:

- Create a three point plane using mesh points

- Measure the surface area, distances to and from mesh points, and mass properties for solid meshes

- Project mesh points in a sketch (via the Use tool)
- Create Mate connectors at mesh points

- Reference a mesh point for ‘Up to vertex’ operations (as in Extrude)

Meshes are not supported in Assemblies or Drawings.

Repairing imported models

Onshape provides a tool for identifying holes and disconnects in imported models that may require repair: Highlight boundary edges.

Highlight boundary edges can be found as a menu option in the View tools cube: "Highlight boundary edges" on page 205.
When *Highlight boundary edges* is selected, boundary (or laminar) edges are displayed in solid red lines (for visible edges) and dashed red lines (for hidden edges).

To repair these surfaces, you can use any number or combination of tools in Onshape, for example:

- **Enclose** - Select surfaces, faces, and parts that enclose a region to create a new part.
**Fill** - Create a surface bounded by a closed set of curves or edges.
**Move boundary** - Move boundary edges of a surface in order to extend or trim it.
- **Bridging curve** - Create a curve connecting any two points or vertices.

- **Composite curve** - Create curves from selected edges, sketch entities and other curves.

Creating composite parts

When parts or surfaces are fractured upon import, or when you want separate bodies to be treated as a cohesive unit, create either a closed or open composite part (one which consumes the entities, or does not, respectively).

For example, if your design is fractured upon import, as in this example below where one section of the design is rendered as multiple surfaces, you can select the fractured entities and combine them to form a composite part:
The selected surface, "object_1(13)," is cross-highlighted in the model - but suppose you want all the surfaces to be one part. You can select them and create a composite part, shown below:

The part, after the feature is accepted:
Editing imported parts with direct editing tools

Imported geometry does not carry with it the parametric history so there may be times when you need to use Onshape's direct editing tools to continue modeling an imported design.

For example, after you import a part, the holes might be in the wrong places. You can use the direct editing tools to remove the original holes and then make new holes in the proper locations on the model.

Another example is fixing imported geometry when a face is missing or has an error, or you have to delete a face and recreate it. In the example below, the knob on the top of the cap needs to be reshaped, so the Delete face tool is used:
The selected face to remove, above

The resulting cap with new face, above

Updating imported parts

Parts imported into Onshape can be updated if the source part has changed. The first thing to understand is where Onshape stores that information. Depending on where you initiated the import in Onshape, the part will be in one of two places:

- Import from the Documents page and the part is inserted into a new document bearing the same name as the imported file.
- Import from within an existing document and the part is inserted into a Part Studio bearing the same name as the imported file.

In both cases, you will find a folder by the name of CAD Imports and within that folder, a tab with the same name as the imported file. Also in each case, you will find a Part Studio with the same name as the imported file. You can change the names of the folder as well as the tabs.

After a part is imported, you can update it should the part change in the source file.

To update a previously imported part:
1. Export the part once more from the other CAD system. (Presumably this part has been changed since the first time it was exported.)

2. In the document into which the exported part was previously imported, you can proceed in either of two ways:

a. Open the Part Studio that holds the part, right-click the Import entry in the Features list and select Update.
Select the export file and click Open. The Import feature updates to the updated part.
b. The other option for updating an import feature is to click the folder titled CAD import and select the tab of the imported file. Click Update (shown below), select the export file and click Open. The new file name is displayed and the part is updated in the Part Studio.

The new file name is displayed and the part is updated in the Part Studio.

If a document has not been upgraded after Onshape release 1.93, then this Update button will not upgrade the imported feature but will instead make a new Part Studio containing the new/upgraded part. If the document has been upgraded since Onshape release 1.93, when the Update button is clicked, the imported blob is updated with the new file and the updated geometry propagates to the imported feature.
In both cases, the Part Studio containing the Import feature is updated.

**What is updated**

When a part is updated, all features and changes made to the part in the source system are updated in the target Part Studio. If the part has been modified in the target Part Studio, by applying features to it, those features are reapplied to the updated part where possible.

For example, if a fillet has been applied to the part in the target Part Studio, that fillet is reapplied to the updated part where possible. There may be modeling changes made to the source part that result in applied features in the target Part Studio from regenerating successfully. These features will be shown in an error state in the Features list.

**Deleting imported files**

When deleting imported files, you will see a warning about which Part Studios the deletion will impact:
Replace multiple references with one reference

Sometimes when you import a project into Onshape, commonly used components may become duplicated. Since duplication of components is an undesirable situation, in Onshape you can instead use multiple references to the one component.

For organizational purposes, it's best to move the common component to its own document, so all users can find it easily without disturbing projects in process. To move a component out of one document and into another:

1. Select the name of the component in the Part Studio’s Parts list.
2. Right-click and select Export.
3. On the Documents page, click Create and then Import.
4. Import the file you just exported, which will create a new document.
5. In the original document, locate the part in its original Part Studio.
6. Delete the part from the Part Studio.
   
   Note that in this document, all references to that Part (in assemblies) will 'break' and appear red.

7. In each assembly that references that part, select each reference to the part in the Instances list, right-click and select Replace parts.
8. In the Replace part dialog, navigate to the Created by me document section, and select the new source document containing the common component.
9. Now the component will be inserted where it was referenced in the assembly, but with a link icon next to it. The link icon indicates that the instance is referenced from another document.

The advantage to this workflow is that you now have one component instanced many times. If you want to make a change to the component, do it in the source document and then you can decide if and when to update the instances referenced in other documents. This saves you the trouble of reinserting and mating multiple components unnecessarily, as well as aid you in keeping components up to date easily.

For a more in-depth example, see this Onshape Tech Tip.

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**Exporting Files**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape enables you to export parts and surfaces (from Part Studios), entire Part Studios, entire Assemblies, as well as sketches and planar faces to your hard drive or cloud-source (for example, Google Drive or Dropbox) for use elsewhere.

To have the option to export to a cloud-source, you must give Onshape permission to access your accounts. For information on integrating your cloud accounts with Onshape, see "Managing Your Onshape Professional Subscription" on page 2492.

**Exporting Files: Desktop**

- Surfaces may be exported individually from the list (in Part Studios and Assemblies) or selected in the graphics area (use the context menu, Export option) to:
  - Native or standard formats (Parasolid, ACIS, STEP, IGES, SOLIDWORKS and Rhino)

- Parts may be exported individually from the Parts list (in Part Studios) or selected in the graphics area (use the context menu, Export option) to:
- OBJ, STL
- Native or standard formats (Parasolid, ACIS, STEP, IGES, SOLIDWORKS, Collada, Rhino, and GLTF).
- You can select multiple parts at once in Part Studios. For STL and Parasolid formats, choose to export as one file or as individual files.
- In Assemblies, use the tab Export option to export the entire Assembly at once. For STL and Parasolid formats, choose to export as one file or individual files.

Note that exporting as individual files creates a zip file with multiple files each containing a single part.

- Sketches can be exported to DWG and DXF formats.
- Planar faces can be exported to DWG and DXF formats.
- Flattened views of sheet metal models can be exported to DWG and DXF formats.

The download location is browser-specific. If you'd like to be prompted to save to a different location, look in your browser settings for an option to "Ask where to save each file before downloading."

Note that the downloaded data will not contain features or parametric history. See the topics below for more information.

**Exporting parts from Part Studios**

To export a single part or multiple parts, select the part in the Parts list, then right-click to access the context menu. Select Export from the context menu:

Specify the parameters to use:
When exporting a part to STL format, you have the following options:

- Use Text or Binary for the STL format
- Select from units such as: Centimeter, Foot, Inch, Meter, Millimeter, Yard
- Choose a resolution: Coarse, Medium, Fine, Custom
  - When selecting a Custom resolution, you can then specify:
    - Angular deviation
    - Chordal tolerance
    - Minimum facet width
- Download, Download and store file in a new tab, Email with file download link, or skip the download and Store file in a new tab.

Note that when multiple parts are selected, you can specify whether to export the parts as one file, or as individual files, zipped together.

You can also specify a new name for the file, to replace the default name.

The download location is browser-specific. If you'd like to be prompted to save to a different location, look in your browser settings for an option to "Ask where to save each file before downloading."

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When exporting in OBJ format, the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

Exporting via email with file download link

With every export, you have the option to email the exported file as a download link (that is, with all plan types except Free or Education). For the Option, select Email with file download link:
Enter the recipients' email, a customized subject for the email, and an optional message. Specify how long you want the download link to be available: the default is 3 days.

If you want to password-protect the link you are emailing, check "Require password for download" and supply a password in the next field. Note: Never email a password, a better practice is to specify in the message for the recipient to contact you for the password by other means.

The recipient receives an Onshape-branded email with the link to download the exported part or assembly.

**Exporting Part Studios**

To export an entire Part Studio, access the Export command from the context menu on the Part Studio tab:

Specify the parameters to use:
Check your file downloads location for the file upon completion.

**When exporting in OBJ format**, the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

**Exporting sketches or planar faces**

Sketches are exported in the document's default units, and planar faces are exported with outer solid geometry only, no dimensions or interior geometry. Specify a file name, select the format and version. Optionally check the box next to Set z-height to zero and normals to positive (use this option to ensure that all normal vectors of components with coordinates on the z plane have a positive z component). Click Export.

> Note that splines are exported as splines. If you prefer to explode the spline into polylines, check the **Export splines as polylines** check box in the Export dialog.

**Export a sketch from the Feature list**

Specify a file name, select the format and version. Optionally check the box next to Set z-height to zero and normals to positive (use this option to ensure that all normal vectors of components with coordinates on the z plane have a positive z component).
Click Export.

Note that splines are exported as splines. If you prefer to explode the spline into polylines, check the Export splines as polylines check box in the Export dialog.

Export a planar face from the context menu in the graphics area

When exporting a planar face from the graphics area, use the Export as DXF/DWG option. Specify a file name, select the format and version. Optionally check the box next to Set z-height to zero and normals to positive (use this option to ensure that all normal vectors of components with coordinates on the z plane have a positive z component). Click Export. (Using the Export... option exports the entire part not just the planar face.)

Exporting from Assemblies

You have the ability to export all parts from Assemblies as either one file containing the entire Assembly, or as a zip file of individual files for each part in the Assembly.
using the tab Export option:

1. Select a format (note that only STL and Parasolid formats allow export of individual parts files, zipped together).

2. When selecting STL or Parasolid, indicate how to package the files. Check the box to download one file for each part, zipped together, or leave unchecked to export as one zip file containing one file per part.

**When exporting in OBJ format,** the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

**Exporting a Drawing**

You have the ability to export Onshape drawings to the following file types:

- PDF
- DWG
- DXF
- DWT (see, "Exporting a drawing to a template" on page 1758)

The export function presents the opportunity to select the desired format; when the translation is finished, the file is also downloaded to your local machine.

1. Right-click on the Drawing tab.

2. Select Export.

3. Specify a name for the export file.

4. Select the desired export format:
   - .DWG
   - .DXF
5. If exporting to DXF or DWG, select the version and sheets. If exporting to PDF, select whether to export text normally, or as selectable text.

6. Choose how to treat overridden dimensions: Show underlines or Hide underlines.

7. Indicate which drawing sheets to export, if applicable.

8. Indicate whether to export splines as polylines, available for all formats except PDF.

9. Indicate whether to explode the text upon export, if desired.

10. Select what to do with the export file:

   a. Download the file only

   b. Download the file and store the file in a new tab in the document

   c. Email with file download link

   d. Store the file in a new tab only.

11. For DWG, DWT, and DXF exports, you can check the box to Set z-height to zero and normals to positive. This sets the z-height/direction of any hole normals to be positive (+1) for drawings with view geometry.

**Exporting from a release**

When you have released entities, you can export directly from the revision history:

1. Right-click on the entity that has been released, for example an Assembly tab, and select Revision history.

2. The Revision history dialog opens.
3. Click the Export button for the release containing the entities you wish to export:

![Revision history for part FISH A1307]

4. Select your preferred export options.

**Exporting Files: iOS**

Onshape enables you to export parts and surfaces (from Part Studios), entire Part Studios, and subassemblies and instances (from Assemblies) as well as entire Assemblies.

To export a Part Studio containing a single part or multiple parts:

1. Select the overflow menu in the Part Studio or Assembly tab (make sure the tab is not open when you select the menu):
2. Tap Export.

3. Enter a file name for the export file.
4. Specify a format: Parasolid, ACIS, IGES, SOLIDWORKS, Collada, GLTF, OBJ, Rhino, STL. (Specify a version if necessary.)

5. Select an option:
   - Download
   - Download and store file in a new tab
   - Store file in a new tab

6. When there are multiple parts in the Part Studio, tap the checkbox to Export parts as individual files.

7. Tap Export in the lower right.

**Exporting parts from Part Studios**

To export a single part or multiple parts, select the part in the Parts list, then select Export from the three dot context menu.

Specify the parameters to use.

When exporting a part to STL format, you have the following options:
   - Use Text or Binary for the STL format
   - Select from units such as: Centimeter, Foot, Inch, Meter, Millimeter, Yard
   - Choose a resolution: Coarse, Medium, Fine, Custom
     - When selecting a Custom resolution, you can then specify:
       - Angular deviation
       - Chordal tolerance
       - Minimum facet width
   - Download, Download and store file in a new tab, or skip the download and Store file in a new tab.

Note that when multiple parts are selected, you can specify whether to export the parts as one file, or as individual files, zipped together.

You can also specify a new name for the file, to replace the default name.

**When exporting in OBJ format**, the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip
file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

**Exporting from Assemblies**

You can export all parts from Assemblies as either one file containing the entire Assembly, or as a zip file of individual files for each part in the Assembly using the tab Export option:

1. Select a format (note that only STL and Parasolid formats allow export of individual parts files, zipped together).
2. When selecting STL or Parasolid, indicate how to package the files. Check the box to download one file for each part, zipped together, or leave unchecked to export as one zip file containing one file per part.

**When exporting in OBJ format,** the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

**Exporting a Drawing**

You have the ability to export Onshape drawings to the following file types:

- PDF
- DWG
- DXF
- DWT (see, "Exporting a drawing to a template" on page 1758)

The export function presents the opportunity to select the desired format; when the translation is finished, the file is also downloaded to your local machine.

1. Tap the three dot context menu.
2. Select Export.
3. Specify a name for the export file.
4. Select the desired export format:
   - .DWG
   - .DXF
   - .DWT
   - .PDF

5. If exporting to DXF or DWG, select the version and sheets. If exporting to PDF, select whether to export text normally, or as selectable text.

6. Choose how to treat overridden dimensions: Show underlines or Hide underlines

7. Indicate whether to explode the text upon export, if desired.

8. Select what to do with the export file:
   a. Download the file only
   b. Download the file and store the file in a new tab in the document
   c. Store the file in a new tab only

9. For DWG, DWT, and DXF exports, check the box to Set z-height to zero and normals to positive. This sets the z-height/direction of any hole normals to be positive (+1) for drawings with view geometry.

**Tips**

- Surfaces are able to be exported individually from the Parts list in Part Studios or from the graphics area context menu.
- Parts are able to be exported individually from the Parts list (in Part Studios).
- You have the ability to select multiple parts at once in Part Studios. For STL and Parasolid formats, choose to export as one file or as individual files.
- In Assemblies, use the tab Export option to export the entire Assembly at once or select instances and sub-assemblies for export in the Instances list.

Note that exporting as individual files creates a zip file with multiple files each containing a single part.

Note that the downloaded data will not contain features or parametric history. See the topics below for more information.
For OBJ format, the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

**Exporting Part Studios**

To export an entire Part Studio, access the Export command from the three dot context menu on the Part Studio tab.

1. Specify the parameters to use.
2. Check your file downloads location for the file upon completion.

When exporting in OBJ format, the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

**Exporting sketches or planar faces**

Sketches are exported in the document's default units, and planar faces are exported with outer solid geometry only, no dimensions or interior geometry. Specify a file name, select the format and version. Optionally check the box next to Set z-height to zero and normals to positive (use this option to ensure that all normal vectors of components with coordinates on the z plane have a positive z component). Click Export.

Note that splines are exported as splines. If you prefer to explode the spline into polylines, check the Export splines as polylines check box in the Export dialog.

**Export a sketch from the Feature list**

1. Tap the overflow menu next to the sketch in the Feature list.
2. Select Export as DXF/DWG.

3. Select the format and Tap Export.

**Export a planar face from the context menu in the graphics area**

When exporting a planar face from the graphics area, use the Export as DXF/DWG option. Specify a file name, select the format and version. Optionally check the box next to Set z-height to zero and normals to positive (use this option to ensure that all normal vectors of components with coordinates on the z plane have a positive z component). Click Export.

**Exporting Files: Android**

Onshape enables you to export parts and surfaces (from Part Studios), entire Part Studios, and subassemblies and instances (from Assemblies) as well as entire Assemblies.

To export a Part Studio containing a single part or multiple parts:
1. Select the overflow menu in the Part Studio or Assembly tab:

2. Tap Export.

3. Enter a file name for the export file.

4. Specify a format: Parasolid, ACIS, IGES, SOLIDWORKS, Collada, GLTF, OBJ, Rhino, STL. (Specify a version if necessary.)

5. Select an option:
   - Download
   - Download and store file in a new tab
   - Store file in a new tab

6. When there are multiple parts in the Part Studio, tap the checkbox to Export parts as individual files.

7. Tap Export in the upper right.

Tips
- Surfaces are able to be exported individually from the Parts list in Part Studios.
- Parts are able to be exported individually from the Parts list (in Part Studios).
- You have the ability to select multiple parts at once in Part Studios. For STL and Parasolid formats, choose to export as one file or as individual files.
- In Assemblies, use the tab Export option to export the entire Assembly at once.
Note that exporting as individual files creates a zip file with multiple files each containing a single part.

- Sketches are able to be exported to DWG and DXF formats, in Release 11 format
- Planar faces are able to be exported to DWG and DXF formats, in Release 11 format

Note that the downloaded data will not contain features or parametric history. See the topics below for more information.

- For OBJ format, the export is a .zip containing the .obj and .mtl file. The .obj file is in meters only. To re-import the .obj file, you must first extract the .zip file and only import the .obj file. We do not support importing .zip files containing .obj and .mtl files.

Exporting a Drawing

You have the ability to export Onshape drawings to the following file types:

- PDF
- DWG
- DXF
- DWT (see, "Exporting a drawing to a template" on page 1758)

The export function presents the opportunity to select the desired format; when the translation is finished, the file is also downloaded to your local machine.

1. Tap the Drawing tab.
2. Select Export.
3. Specify a name for the export file.
4. Select the desired export format:
   - .DWG
   - .DXF
   - .DWT
   - .PDF
5. If exporting to DXF or DWG, select the version and sheets. If exporting to PDF, select whether to export text normally, or as selectable text.

6. Choose how to treat overridden dimensions: Show underlines or Hide underlines

7. Indicate whether to explode the text upon export, if desired.

8. Select what to do with the export file:
   a. Download the file only
   b. Download the file and store the file in a new tab in the document
   c. Store the file in a new tab only

9. For DWG, DWT, and DXF exports, you can check the box to Set z-height to zero and normals to positive. This sets the z-height/direction of any hole normals to be positive (+1) for drawings with view geometry.

**Exporting parts from Part Studios**

To export a single part or multiple parts, select the part in the Parts list, then select Export from the overflow menu:

Specify the parameters to use.

When exporting a part to STL format, you have the following options:

- Use Text or Binary for the STL format
- Select from units such as: Centimeter, Foot, Inch, Meter, Millimeter, Yard
- Choose a resolution: Coarse, Medium, Fine, Custom
  - When selecting a Custom resolution, you can then specify:
    - Angular deviation
    - Chordal tolerance
    - Minimum facet width
- Download, Download and store file in a new tab, or skip the download and Store file in a new tab.

Note that when multiple parts are selected, you can specify whether to export the parts as one file, or as individual files, zipped together.

**Exporting Part Studios**
To export an entire Part Studio, access the Export command from the overflow menu on the Part Studio tab.

Specify the parameters to use.

Check your file downloads location for the file upon completion.

**Exporting sketches or planar faces**

Access the Export command from the overflow menu next to the sketch in the Feature list. Access the Export command for a planar face after selecting the face and using the context menu. Sketches are exported in the document's default units, and planar faces are exported with outer solid geometry only, no dimensions or interior geometry.

**Export a sketch from the Feature list**

Select the format and click Export.

**Export a planar face from the context menu in the graphics area**

When exporting a planar face from the graphics area, use the Export as DXF/DWG option (this exports in Release 11 format). Using the Export... option exports the entire part not just the planar face.

**Exporting from Assemblies**

You have the option to export all parts from Assemblies as either one file containing the entire Assembly, or as a zip file of individual files for each part in the Assembly using the tab Export option.

Select a format (note that only STL and Parasolid formats allow export of individual parts files, zipped together).

When selecting STL or Parasolid, indicate how to package the files. Check the box to download one file for each part, zipped together, or leave unchecked to export as one zip file containing one file per part.
Downloading Files

This functionality is available on Onshape's browser, iOS, and Android platforms.

Files that are unable to be processed upon export (primarily non-CAD files) are able to be downloaded through the Onshape tab context menu. You are able to download any tab that is able to be represented as a file.

Downloading Files: Desktop

Download copies the file in its current format to your local machine, giving it the same name and file type. You are able to import and download any non-native file type into and out of Onshape.

The download location is browser-specific. If you’d like to be prompted to save to a different location, look in your browser settings for an option to "Ask where to save each file before downloading."

Downloading Files: iOS and Android

Files that are not able to be processed upon export (primarily non-CAD files) are able to be downloaded through the Onshape tab context menu. You have the ability to download any tab that may be represented as a file.

Tap the menu icon next to the tab and tap Download.
**Download** copies the file in its current format to your local machine, giving it the same name and file type. You are able to import and download any non-native file type into and out of Onshape.
Sharing and Collaboration

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape provides multiple tools for collaborating with other Onshape users, as well as people outside of the Onshape process but still very much a part of your process.

To learn more about sharing documents and collaborating in Onshape, you can follow the self-paced course here: [Sharing and Collaboration](#) (opens in new tab).

Sharing and Collaboration: Desktop

**Share Documents**

Share a document with one or more users, enabling real-time collaboration right in the same document. Share with individuals, lists of individuals, teams, and companies, or make a document publicly available (or private again). You are also able to share the document with Onshape support, if needed.

Set and remove permissions on an individual or team basis to fine-tune document security.

See [Share Documents](#) for more info.

**Collaboration**

Multiple users working in the same document at the same time is referred to as Simultaneous editing or Collaboration. Any and all features added or changes made are displayed in real time to all collaborators.

See [Collaboration](#) for more info.

**Comments**

Collaborating users can communicate with each other in a workspace with comments. Owners of documents and collaborators (with Edit or Comment permission) can
create comments, see each others’ comments, leave replies, and opt to receive email notifications of comments.

See [Comments](#) for more info.

**Follow Mode**

When users are collaborating in a single document, they may choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

A user may follow a collaborator across browser and mobile. This means a user on a browser can follow a collaborator on a browser or mobile device. A user on a mobile device may follow a collaborator on a mobile device or on a browser.

See [Follow Mode](#) for more info.

**Transfer Ownership**

Every document is owned by either a user or a company. Users who are not members of a company automatically own the documents they create, while users who are members of a company can only create company-owned documents. (Even when a user makes a document public, the specified owner still owns the document.)

Owners of documents and owners of companies have these permissions on documents they own: Delete, change sharing privileges, make Public, make Private, and Transfer ownership. Document ownership may be transferred at any time, by the document owner or company admin, by right-clicking on the document in the Documents page and selecting Transfer ownership from the context menu.

See [Transfer Ownership](#) for more info.

**Sharing and Collaboration: iOS**

Onshape provides multiple tools for collaborating with other Onshape users, as well as people outside of the Onshape process but still very much a part of your process.

- Share a document with one or more users, enabling real-time collaboration right in the same document.
- Share a document with non-users; allowing them to view the document in View-only mode.
Set and remove permissions on an individual or team basis to fine-tune document security.

Initiate Follow mode so other users with permission are able to see all of your actions, in real-time, in a document.

Transfer ownership of a document; as the creator you are the owner by default but you are able to transfer ownership to any other Onshape user.

Comment on entities and tag friends so they receive an email with the comment and a link to that comment in Onshape.

Share Documents

Share a document with one or more users, enabling real-time collaboration right in the same document. Share with individuals, lists of individuals, teams, and companies, or make a document publicly available or private. You are also able to share the document with Onshape support, if needed.

Set and remove permissions on an individual or team basis to fine-tune document security.

See Share Documents for more info.

Collaboration

Multiple users working in the same document at the same time is referred to as Simultaneous editing or Collaboration. Any and all features added or changes made are displayed in real time to all collaborators.

The creator of the document must share it with the other Onshape users before they can collaborate.

See Collaboration for more info.

Comments

Collaborating users are able to communicate with each other, in a workspace, with comments. Owners of documents and collaborators (with Edit or Comment permission) are able to create comments, see each others' comments, leave replies, and opt to receive email notifications of comments.
See Comments for more info.

**Follow Mode**

When users are collaborating in a single document, they are able to choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

A user has the ability to follow a collaborator across browser and mobile. This means a user on a browser is able to follow a collaborator on a browser or mobile device. A user on a mobile device is able to follow a collaborator on a mobile device or on a browser.

See Follow Mode for more info.

**Transfer ownership**

Every document is owned by either a user or a company. Users who are not members of a company automatically own the documents they create, while users who are members of a company can only create company-owned documents. (Even when a user makes a document public, the specified owner still owns the document.)

Owners of documents and owners of companies have these permissions on documents they own: Delete, change sharing privileges, make Public, make Private, and Transfer ownership. Document ownership are able to be transferred at any time, by the document owner or company admin, through the Share dialog.

This functionality is available only in Onshape on a browser at this time. Refer to the Browser Help topic Transfer Ownership for more information.

**Collaboration example**

Suppose there is an Onshape document, with two Part Studios that define a total of 3 parts, and one Assembly that contains instances of those parts.

Alice is working in the Frame Part Studio, and it is able to be seen from the social cues that Diana is also working in that Part Studio. Fred is working in the Assembly (Assembly 1) and Nick’s cue is able to be seen from the Documents page, in the Detail panel. Each user knows the other users are in the document, and where, based on the social cues:
The arrows in the images above indicate the various social cues that indicate who is working in each document, Feature, or tab.

The document owner is always able to choose to restrict simultaneous editing by limiting the collaborators and/or the access rights of those collaborators. Document owners decide when, how much, and with whom to collaborate. To learn more about this, read Share Documents.

**Tips**

- A single collaborator is able to have many followers
- A follower may follow only one user at a time
- For tabs that do not support collaboration (drawings, for example, or third party applications), a user who has share permissions to edit the document can "steal" focus on a non-collaborative tab:
  - While on the tab, right-click the tab and select **Take edit permission** to get a lock on the tab, preventing other users from getting focus on that tab.

When trying to access a non-collaborative tab when another user has focus on it, you'll see this message:

Bob Miner is currently editing this tab and only one user may edit this tab at a time.

**Take edit permission from Bob Miner**
Either click the blue button (shown above) or right-click on the tab and select Take edit permission to gain focus and view/edit that tab:

Other users will see the non-collaborative message once you have focus. When you leave the tab, it becomes available to other users again.

Sharing and Collaboration: Android

Onshape provides multiple tools for collaborating with other Onshape users, as well as people outside of the Onshape process but still very much a part of your process.

- Share a document with one or more users, enabling real-time collaboration right in the same document.
- Share a document with non-users; allowing them to view the document in View-only mode.
- Set and remove permissions on an individual or team basis to fine-tune document security.
- Initiate Follow mode so other users with permission can see all of your actions, in real-time, in a document.
- Transfer ownership of a document; as the creator you are the owner by default but you can transfer ownership to any other Onshape user.
- Comment on entities and tag friends so they receive an email with the comment and a link to that comment in Onshape.

Share Documents
Share a document with one or more users, enabling real-time collaboration right in the same document. Share with individuals, lists of individuals, teams, and companies, or make a document publicly available or private. You can also share the document with Onshape support, if needed.

Set and remove permissions on an individual or team basis to fine-tune document security.

See Share Documents for more info.

**Collaboration**

Multiple users working in the same document at the same time is referred to as Simultaneous editing or Collaboration. Any and all features added or changes made are displayed in real time to all collaborators.

The creator of the document must share it with the other Onshape users before they can collaborate.

See Collaboration for more info.

**Comments**

Collaborating users can communicate with each other, in a workspace, with comments. Owners of documents and collaborators (with Edit or Comment permission) can create comments, see each others’ comments, leave replies, and opt to receive email notifications of comments.

See Comments for more info.

**Follow mode**

When users are collaborating in a single document, they can choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

**To follow someone, double-click their social cue icon in your toolbar.**

Double-click the social cue icon (in green) at the top, next to the Share button. Notice the banner at the top of the window, and that the workspace is outlined to indicate that the collaborator is being followed.

**To stop following, click anywhere in your browser window.**

Followers can see:
- The collaborator’s active tab and actions in that tab
- The collaborator’s cursor movements (shown as a hand in the social cue icon color)
- Views and Render modes of parts (accessed from the (icon) menu, including Section view)
- Selections made in the graphics area

What followers do not see:
- Selections made in the Feature list
- Dialog boxes and work done inside dialogs
- Part movement and sketching: you will see the part/assembly in its new location after a collaborator moves it, and a sketch after the Sketch dialog is accepted.

**Transfer ownership**

Every document is owned by either a user or a company. At the time of creation, a user who belongs to a company can specify the owner of the document: that user or the company (the default is company). Users who are not members of a company automatically own the documents they create. (Even when a user makes a document public, the specified owner still owns the document.)

Owners of documents and owners of companies have these permissions on documents they own: Delete, change sharing privileges, make Public, make Private, and Transfer ownership. Document ownership can be transferred at any time, by the document owner or company admin, through the Share dialog.

This functionality is available only in Onshape on a browser at this time. Refer to the Browser Help topic **Transfer Ownership** for more information.

**Collaboration example**

Suppose there is an Onshape document, with two Part Studios that define a total of 3 parts, and one Assembly that contains instances of those parts.

Alice is working in the Frame Part Studio, and it can be seen from the social cues that Diana is also working in that Part Studio. Fred is working in the Assembly (Assembly
1) and Nick’s cue can be seen from the Documents page, in the Detail panel. Each user knows the other users are in the document, and where, based on the social cues:

The arrows in the images above indicate the various social cues that indicate who is working in each document, Feature, or tab.

The document owner always has the option to choose to restrict simultaneous editing by limiting the collaborators and/or the access rights of those collaborators. Document owners decide when, how much, and with whom to collaborate. To learn more about this, read Share Documents.

Tips

- A single collaborator may have many followers.
- (drawings, for example, or third party applications), a user who has share permissions to edit the document can "steal" focus on a non-collaborative tab:
  - While on the tab, right-click the tab and select **Take edit permission** to get a lock on the tab, preventing other users from getting focus on that tab.

When trying to access a non-collaborative tab when another user has focus on it, you’ll see this message:

> Bob Miner is currently editing this tab and only one user may edit this tab at a time.

> Take edit permission from Bob Miner
Either click the blue button (shown above) or right-click on the tab and select **Take edit permission** to gain focus and view/edit that tab:

Other users will see the non-collaborative message once you have focus. When you leave the tab, it becomes available to other users again.

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**Share Documents**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Share a document to collaborate with other designers, change permission levels, or make a document publicly available. You are also able to share the document with Onshape Support, if needed.

**Share Documents: Desktop**

Access the Share dialog from either the Documents page or in a specific document:
Note that the Teams and Companies tabs appear only when you are a member of a team or a company. The Applications tab only appears when you have enabled an app to work with your document. Members of Enterprises will see only Individuals, Teams, Invite guests, and Links tabs. All documents created in an Enterprise are automatically owned by the Enterprise. For more information on Enterprise guest users, see "Adding and Administering Users" on page 2264.

**Sharing a document**

1. Click **Share**.

2. Select the appropriate tab (which may vary based on your subscription type):

   a. **Individuals** - Enter one or more individual user email addresses. You are also able to copy and paste a comma-separated list here. Onshape provides type-ahead support and records new email addresses as you enter them. Previously recorded emails appear in the list, along with a user image and email address.
b. **Teams** - Teams of which you are a member appear in the drop down. Select a team to send a share message to all members of that team. For more information regarding teams, see "Enterprise Teams" on page 2385 or "Managing Your Onshape Professional Subscription" on page 2492 (for Professional subscriptions).

c. **Companies** - Companies of which you are a member appear in the drop down. Select a company to send a share message to all members of that company.

d. **Public** - Makes the document accessible to all Onshape users. Users may not edit a public document, but may make a copy and edit that. To subsequently make a public document private again, open the Share dialog for the document and click the "x" next to the Public entry in the dialog.

e. **Application** - Applications you have purchased or have a subscription to appear in this list. To see this tab, you must have turned the switch (opens in new tab) on in your Account preferences.

f. **Link sharing** - Copy a document-specific URL to the clipboard in order to send a link to another person. The link allows View-only access to this document alone, and does not require signing in to Onshape for viewing. Only Part Studios, Assemblies, and Drawing tabs will be available. The recipient of the link can use an existing Onshape account to view the document, but is not required to have an Onshape account at all. The document is still viewable without an Onshape account.

To share a document through a link, select the Link sharing tab, then click Turn on link sharing:

**Turn on link sharing**

then Click copy to clipboard:

**Copy to clipboard**

Paste the link into an email to another person (Onshape user or not) or otherwise send the link. Be aware that anyone with the link will have access to the document. To revoke all link sharing access to the document, open the Share dialog and click the X next to the **Anyone with link** entry. This action immediately revokes all access to that document through that link.
When you create a Share document link from within an active tab in a document, the link directs the recipient to the default tab of the document. If you want to direct individuals to specific tabs within a document, select the tab you wish to share first, then open the Share dialog, turn on Link sharing, and instead of using the dialog to copy the link to the clipboard, copy the URL at the top of the window and send that to the user. Regardless of which link you send, the user still has access to the entire document.

To enable an Onshape user to export the document once the link is shared with them, check **Allow exporting from the link**.

3. Select (or deselect) additional permissions per user below the email address field.

To enable other users to link to your document from their document (by linking to a part, assembly, FeatureScript, or custom feature, for example) check the Link document permission box. Link document is automatically selected if a user was granted Edit permission (with at least Copy/Export) prior to Release 1.54. Post-Release 1.54, this box must be checked manually.

Collaborators receive an email with your message, and a link to the document in Onshape. Onshape users can click the link to access the document. If the recipient is not an Onshape user, the email includes a link to create an account before accessing your document.

Unshare a document with a user at any time: click the 'x' beside the user name in the Share dialog. Users may also remove themselves from a shared document using the context menu on the Documents page, or through the Share dialog.

Document owner

Only owners of documents and those with Can edit & share permission are able to share a document with another user. Owners are able to be individual users or a company (for those with a Professional or Enterprise subscription account). There may be only one owner of a document. In the case of a company owning a document, the owner permissions are assigned to the owner of that company.
Ownership is the highest level of permissions, giving a user the right to transfer that ownership to another user or to a company. To learn more about transferring ownership in Onshape, see the Documents Page topic.

Listed users

This area lists all users, companies, teams, and applications that the document has been shared with. The current permission is shown to the right of the email address and can be changed by the owner of the document (click the pencil icon). Use the small x further to the right to remove this user, team or company from the share permissions of this document.

Sharing options

Select an option:

- **Individuals** - Enter one email address or paste in a list of email addresses separated by commas or semi-colons (this results in individual entries in the Share list above); note that the address list is not saved. You can add an optional message to be included in the email notification.

- **Teams** - Available for team members, you can select a team in order to share the document with many users at once.

- **Companies** - Available for company members, you can select a company from the list to share the document with all members of that company.

- **Public** - Make the document publicly available as read-only to all Onshape users, enabling them to make a private, editable copy. To subsequently make a public document private again, open the Share dialog for the document and click the "x" next to the Public entry in the dialog.

- **Application** - When you grant an application access to your document, you are effectively sharing the document with it. You can view the share permissions you have granted to applications as well as revoke those permissions. (You can always re-grant permissions.) To allow desktop applications access to your Onshape documents, grant access on a document-by-document basis through this Share dialog.

- **Link sharing** - Copy a link to the document in order to send it to another person (who may or may not be an Onshape user) with read-only permission. Keep in mind that anyone with the link can view the document. This allows a user to view
the document only and possibly export the document if you check the appropriate box.

Permissions

For each share operation, select the document permission level for each user or group of users:

- **Can edit** - Permission to edit, copy, link, export, share, and comment on a document.
- **Can view** - Permission to view, copy, link, export, and comment on a document. Users with this permission will be shared into the document in "Getting Started as a Light User" on page 2226.

Keep in mind that Edit and View permissions include Comment permission. Use the individual check boxes below the email address field to include any of the following permissions:

- **Copy** - Ability to make a copy of the document
- **Link document** - Ability to link to this document from another document (via inserting an assembly, part, image, drawing, etc)
- **Export** - Ability to export the document
- **Share** - Ability to reshare the document with another user
- **Comment** - Ability to make comments within the document
- **Delete** - Ability to delete a document or a workspace on a branch within that document. Note that when deleting a workspace, a warning dialog displays and the action, if taken, cannot be undone. If there are versions on the branch, the branch will remain and you will be able to create a new workspace from any version on that branch, pending permissions.

All permissions allow users to collaborate in the same workspace. A user with view only permission may be in the same workspace as a user with edit permission. The view only user is unable to edit the workspace but they are able to see any changes made to workspace in real time.
Removing permissions

All documents with a company as the owner may be deleted only by the creator of the document or the company owner.

The creator of a document may share it with other users and assign permissions to the document at the time of sharing; permissions explained above.

The Admins of a company may remove the document creator as owner and, if desired, add that user as a collaborator with specific permissions. (By default, all creators of documents have complete permissions, including Delete, of that document.)

To remove a particular permission, simply click the pencil icon next to the Share entry in the Share dialog, then use the check boxes that appear to edit the permission, or remove them entirely.

**Sharing with teams or companies**

Share a document with a team or company you're a member of to collaborate with other members and allow them to access your document's contents:

1. Select the document on the *Documents* page and click (or click it from the open document).
2. Select the appropriate tab: Teams or Companies.
3. Select the team or company name from the drop down list.
4. Select permissions for the team or company (and all members of that organization), as described above.
5. Optionally, add a personal message to be included in the notification email.
6. Click .
7. Repeat steps to share with additional teams or companies. Click when finished.

Unshare a document at any time by clicking the 'x' beside the name.

**Sharing with a Free account user**

Users with a Professional account may share documents with users of Free accounts.

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All rights reserved.
The following restrictions apply:

- Free users are only able to view private documents, regardless of permissions granted on the share action, including Edit permission. They are not allowed to edit private documents.

- Sharing a document with a Free account user does not make the document Public.

- Free account users are able to be granted and perform: Comment, Export, Copy, and Link. They are unable to Share or Edit.

**Making a document public**

If you don’t belong to an enterprise, you have the ability to make a document available to all Onshape users: select the Public tab and click Make public. When a document is public, all Onshape users are able to view and make copies of it, but are unable to edit the original document:

Revoke public access of a document by clicking × next to All Onshape users in the Share dialog.
On the Documents page, public documents appear with a 🌍 badge and also in their title bar when open:

Sharing a document with Onshape support

If you would like help with a document or you have encountered a bug, opt to Share the document with Onshape support by clicking the Share with Onshape support toggle button at the bottom left of the dialog. When shared, the toggle button turns blue.

When a document is shared with Onshape support, the Share toggle button turns blue; at any time you can unshare the document with Onshape support by clicking the toggle link again.

Share Documents: iOS
Share a document with one or more users, enabling real-time collaboration right in the same document. Share with individuals, lists of individuals, teams, and companies, or make a document publicly available or private. You can also share the document with Onshape support, if needed.

Access the Share dialog from:

- **The Documents page** - Tap the icon to the right of the document to open the document info panel.

  Tap the Share icon.

- **In a specific document** - Tap the icon in the upper right corner to open the document info panel.

  Tap the Share icon.

You have the ability to remove yourself from a shared document at any time: access the document info panel from the documents page or from within the document. Then select the **Remove** icon.

**Steps**

1. Tap the Share icon.

The Share dialog opens.
2. Add an individual, team, or company to share your document with:

- **Individual** - Enter one or more individual email addresses. You can also copy and paste a comma-separated list. Onshape provides type-ahead support and records new email addresses as you enter them.

- **Team** - Teams of which you are a member appear in the list. Selecting a team sends a share message to all members of that team.

- **Company** - Companies of which you are a member appear in the drop down. Selecting a company sends a share message to all members of that company.

- **Link sharing** - Copy a link to the document in order to send it to another person (who may or may not be an Onshape user) with read-only permission. *Keep in mind that anyone with the link can view the document.* This allows a user to view the document only and possibly export the document if you check the appropriate box.

3. Select Share permissions for each individual, team, and company you are sharing with:

- **Can edit** - Permission to edit the document

- **Can view** - Permissions to view the document only (read-only)

   For details on specific permissions see [Permissions](#), below.

4. Optionally, toggle to set the document to Public.

5. Optionally, toggle to share the document with Onshape support.

6. Tap Done in the upper right corner of the dialog.

**Document owner**

In the Share dialog, the Owner line specifies the owner of the document. Only owners of documents and those with Can edit & share permission can share a document with another user. Owners can be individual users or a company (for those with a Professional Subscription account). There may be only one owner of a document. In the case of a company owning a document, the owner permissions are assigned to the owner of that company.
Ownership is the highest level of permissions, giving a user the right to transfer that ownership to another user or company.

**Listed users**

This section lists all users, teams, and companies that the document has been shared with. The current permission is shown below the email addresses and can be changed by the owner of the document.

See [Permissions](#) below for more information on editing permissions.

Users, teams, and companies can also be completely removed from a share by the document owner. See [Remove from Share](#) below for more info.

**Permissions**

For each share operation, select the document permission level for each user or group of users:

**Owner** - Full permission to edit the document including Edit, Share, Comment, and Transfer ownership.

**Can edit** - Permission to Edit the document and Comment on it.

**Can view** - Permission to View the document (read-only).

Keep in mind that Edit and View permissions include Comment permissions. Use the individual check boxes to include or leave out Copy, Link, Export, Share, or Comment permission, if applicable.

**Copy** - Permission to make a private copy of the document.

**Link** - Permission to link to the document, or reference details of the document, in other documents.

**Export** - Permission to export the document as a file.

**Share** - Permission to share the document with other users, teams, and companies.

**Comment** - Permission to comment on the document.

Removing permissions

**To remove (or edit) the permissions of an individual, team, or company:**
1. Open the Share dialog.

2. Tap to select the individual, team, or company whose permissions you would like to remove (or edit).

3. Tap to check or uncheck the boxes for the permissions.

All documents with a company as the owner can be deleted by only the creator of the document or the company owner.

The creator of a document can share it with other users and assign permissions to the document at the time of sharing; permissions explained above.

The Admins of a company can remove the document creator as owner and, if desired, add that user as a collaborator with specific permissions. (By default, all creators of documents have complete permissions, including delete, of that document.)

**Remove from share**

To completely remove a user, team, or company from a share (thus removing all of their permissions):

1. Open the Share dialog.

2. Tap on the name of the individual, team, or company that you want to unshare the document from.

3. Tap **Remove** at the bottom of the dialog.

You can also open the Share dialog, then swipe from right to left on the name of the individual, team, or company that you would like to remove. Tap **Remove**.

**Making a document public**

Tap the toggle next to **Public** to make a document available to all Onshape users. When a document is public, all Onshape users can view and make copies of it, but cannot edit the original document:

Revoke public access of a document by tapping the toggle next to **Public**, to switch it to the off position. When the toggle is on, it turns blue.

If you have a Free Subscription, private documents are prohibited, and the **Public** toggle remains on.

**Share with Onshape support**
If you would like help with a document or you have encountered a bug, tap the toggle next to "Shared with Onshape support." When shared, the toggle turns blue. At any time you can unshare the document with Onshape support by clicking the toggle link again.

**Notifications**

Onshape sends notifications if:

- A document is shared with you
- You are specifically mentioned in a comment
- A document that has been shared with you or a document you own has been commented on

Tap on a notification to open the document that has been shared with you.

In your device settings you can specify how, or if, Onshape sends you notifications.

**Receiving an anonymous link**

You can receive and open an anonymous link on your iOS device:

1. Open the email on your iOS device.
2. Click the link (before opening the Onshape app).
3. If you are a registered Onshape user, the app opens and automatically opens the document for which the link was sent.

You can also copy the link from the email, proceed to the Onshape app sign in page, scroll down to see and tap "Open Document", then paste the copied URL into the field and tap "Open."

If you are not a registered Onshape user, the document opens, but with limited permissions and navigation options.

**Share Documents: Android**

Share a document with one or more users, enabling real-time collaboration right in the same document. Share with individuals, lists of individuals, teams, and companies, or
make a document publicly available or private. You can also share the document with Onshape support, if needed.

Access the Share dialog from:

- **The Documents page** - Tap the icon to the right of the document to open the document info panel.

  Tap the Share icon.

- **In a specific document** - Tap the icon in the upper right corner to open the document info panel.

  Tap the Share icon.

You can remove yourself from a shared document at any time: access the document info panel from the documents page or from within the document. Then select the **Remove** icon.

**Steps**

1. Tap the Share icon.

   The Share dialog opens.

2. Add an individual, team, or company to share your document with:

   - **Individual** - Enter one or more individual email addresses. You can also copy and paste a comma-separated list. Onshape provides type-ahead support and records new email addresses as you enter them.
● **Team** - Teams of which you are a member appear in the list. Selecting a team sends a share message to all members of that team.

● **Company** - Companies of which you are a member appear in the drop down. Selecting a company sends a share message to all members of that company.

● **Link sharing** - Copy a link to the document in order to send it to another person (who may or may not be an Onshape user) with read-only permission. *Keep in mind that anyone with the link can view the document.* This allows a user to view the document only and possibly export the document if you check the appropriate box.

3. Select Share permissions for each individual, team, and company you are sharing with:

   ● **Can edit** - Permission to edit the document
   
   ● **Can view** - Permissions to view the document only (read-only)

   For details on specific permissions see [Permissions](#), below.

4. Optionally, toggle to set the document to Public.

5. Optionally, toggle to share the document with Onshape support.

6. Tap Done in the upper right corner of the dialog.

**Document owner**

In the Share dialog, the Owner line specifies the owner of the document. Only owners of documents and those with Can edit & share permission can share a document with another user. Owners can be individual users or a company (for those with a Professional Subscription account). There may be only one owner of a document. In the case of a company owning a document, the owner permissions are assigned to the owner of that company.

Ownership is the highest level of permissions, giving a user the right to transfer that ownership to another user or company.

**Listed users**

This section lists all users, teams, and companies that the document has been shared with. The current permission is shown below the email addresses and can be
changed by the owner of the document.

See Permissions below for more information on editing permissions.

Users, teams, and companies can also be completely removed from a share by the document owner. See Remove from Share below for more info.

**Permissions**

For each share operation, select the document permission level for each user or group of users:

**Owner** - Full permission to edit the document including Edit, Share, Comment, and Transfer ownership.

**Can edit** - Permission to Edit the document and Comment on it.

**Can view** - Permission to View the document (read-only).

Keep in mind that Edit and View permissions include Comment permissions. Use the individual check boxes to include or leave out Copy, Link, Export, Share, or Comment permission, if applicable.

**Copy** - Permission to make a private copy of the document.

**Link** - Permission to link to the document, or reference details of the document, in other documents.

**Export** - Permission to export the document as a file.

**Share** - Permission to share the document with other users, teams, and companies.

**Comment** - Permission to comment on the document.

Removing permissions

**To remove (or edit) the permissions of an individual, team, or company:**

1. Open the Share dialog.

2. Tap to select the individual, team, or company whose permissions you would like to remove (or edit).

3. Tap to check or uncheck the boxes for the permissions.

All documents with a company as the owner can be deleted by only the creator of the document or the company owner.
The creator of a document can share it with other users and assign permissions to the
document at the time of sharing; permissions explained above.

The Admins of a company can remove the document creator as owner and, if desired,
add that user as a collaborator with specific permissions. (By default, all creators of
documents have complete permissions, including delete, of that document.)

**Remove from share**

To completely remove a user, team, or company from a share (thus removing all of
their permissions):

1. Open the Share dialog.
2. Tap on the name of the individual, team, or company that you want to unshare the
document from.
3. Tap **Remove** at the bottom of the dialog.

**Making a document public**

Tap the toggle next to **Public** to make a document available to all Onshape users.
When a document is public, all Onshape users can view and make copies of it, but
cannot edit the original document:

Revoke public access of a document by tapping the toggle next to **Public**, to switch it
to the off position. When the toggle is on, it turns blue.

If you have a Free Subscription, private documents are prohibited, and the **Public**
toggle remains on.

**Copying a link to a public document**

You can send a link to a public document to other people so they can look at the pub-
lc document directly without having to find it on the Documents page. To access and
copy the link:

1. Open the Detail pane of the document (tap the <three dot> icon next to the doc-
ument on the Documents page).
2. In the Detail pane, tap the copy icon on the line stating that the document is public:

![Detail pane screenshot]

This copies a link to the phone clipboard, that you can then insert into a message or email.

**Share with Onshape support**

If you would like help with a document or you have encountered a bug, tap the toggle next to "Shared with Onshape support." When shared, the toggle turns blue. At any time you can unshare the document with Onshape support by clicking the toggle link again.

**Notifications**

Onshape sends notifications if:
A document is shared with you

You are specifically mentioned in a comment

A document that has been shared with you or a document you own has been commented on

Tap on a notification to open the document that has been shared with you.

From the Documents Page, go to Settings to enable or disable notifications from Onshape.

**Receiving an anonymous link**

You can receive and open an anonymous link on your Android device:

1. Open the email on your Android device.
2. Click the link (before opening the Onshape app).
3. If you are a registered Onshape user, the app opens and automatically opens the document for which the link was sent.

You can also copy the link from the email, proceed to the Onshape app sign in page, scroll down to see and tap "Open Document", then paste the copied URL into the field and tap "Open."

If you are not a registered Onshape user, the document opens, but with limited permissions and navigation options.

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**Collaboration**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Multiple users working in the same document at the same time is referred to as Simultaneous editing or Collaboration. Any and all features added or changes made are displayed in real time to all collaborators.

**Collaboration: Destop**
The creator of the document must share it with the other Onshape users before they are able to collaborate.

Users collaborating in the same document have the option to activate Follow mode in which one user is able to see what another user is doing in real time. For more information, see Follow Mode.

Collaboration example

Suppose there is an Onshape document, with two Part Studios that define a total of 3 parts, and one Assembly that contains instances of those parts.

Alice is working in the Frame Part Studio, and it is able to be seen from the social cues that Diana is also working in that Part Studio. Fred is working in the Assembly (Assembly 1) and Nick’s cue is able to be seen from the Documents page, in the Detail panel. Each user knows the other users are in the document, and where, based on the social cues:

The arrows in the images above indicate the various social cues that indicate who is working in each document, Feature, or tab. The bordered rectangle inset into the graphics area, above, shows a social cue as it would appear in the Details pane on the Documents page.

The document owner is always able to choose to restrict simultaneous editing by limiting the collaborators and/or the access rights of those collaborators. Document own-
ers decide when, how much, and with whom to collaborate. To learn more about this, read "Share Documents" on page 2045.

All permissions allow users to collaborate in the same workspace. A user with view only permission may be in the same workspace as a user with edit permission. The view only user is unable to edit the workspace but they are able to see any changes made to workspace in real time.

**Tips**

- For tabs that do not support collaboration (drawings, for example, or third party applications), a user who has share permissions to edit the document can "steal" focus on a non-collaborative tab:
  - While on the tab, right-click the tab and select Open tab, close it for other user to get a lock on the tab, preventing other users from getting focus on that tab.
  - When trying to access a non-collaborative tab when another user has focus on it, you'll see a message explaining who is currently editing the tab.

  Either click the blue button or right-click on the tab and select Open tab, close it for other user to gain focus and view/edit that tab.

  Other users will see the non-collaborative message once you have focus. When you leave the tab it becomes available to other users again.

- If you are a collaborator with view-only permission and you change the position of the rollback bar, you no longer will see real-time updates of any changes made to the rollback bar by another collaborator. To fix this, reload your browser.

**Collaboration: iOS**

Multiple users working in the same document at the same time is referred to as Simultaneous editing or Collaboration. Any and all features added or changes made are displayed in real time to all collaborators.

The creator of the document must share it with the other Onshape users before they can collaborate.
Users collaborating in the same document have an option to activate Follow mode in which one user can see what another user is doing in real time. For more information, see Follow Mode.

**Collaborator icon and list**

The Collaborator icon in the upper right of the screen indicates with a number how many collaborators are currently in that document:

![Collaborator icon](image)

Tap the Collaborator icon to view the list of current collaborators:

<table>
<thead>
<tr>
<th>Collaborators</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diane</td>
</tr>
<tr>
<td>Mary</td>
</tr>
</tbody>
</table>

**Social cues**

Social cues indicate where each collaborator is currently working in the document. They are located in both Tabs and Feature lists:
In the image above, you can see that there are two users currently working in "Part Studio 1" as represented by the two colors in the Part Studio tab. See that the user with the initial "N" is currently in working in "Part Studio 1" and is also currently editing the "Sketch 1" feature.

Note: If there is more than one collaborator in a single tab at one time, the social cue at the bottom changes from one colored and initialed box, to a multicolored box. The colors represent the collaborators in that tab. To check which collaborator is represented by which color, tap the Collaborator icon again.

The document owner can always choose to restrict simultaneous editing by limiting the collaborators and/or the access rights of those collaborators. Document owners decide when, how much, and with whom to collaborate. To learn more about this read "Share Documents" on page 2045.
Collaboration: Android

Multiple users working in the same document at the same time is referred to as Simultaneous editing or Collaboration. Any and all features added or changes made are displayed in real time to all collaborators.

The creator of the document must share it with the other Onshape users before they can collaborate.

Collaborator icon and list

The Collaborator icon in the upper left of the screen indicates with a number how many collaborators are currently in that document:

Tap the Collaborator icon to view the list of current collaborators:

Social cues

Social cues indicate where each collaborator is currently working in the document. They are located in both Tabs and Feature lists:
In the image above, you can see that there are two users currently working in "Part Studio 1" as represented by the circle with the "2" in it. See that the user with the initial "N" is currently editing the "Sketch 1" feature as is represented by the circle with the "N" next to the feature.
Note: If there is more than one collaborator in a single tab at one time, the social cue at the bottom changes from one colored and initialed box, to a numbered box. The number indicates how many collaborators are working in that tab.

The document owner can always choose to restrict simultaneous editing by limiting the collaborators and/or the access rights of those collaborators. Document owners decide when, how much, and with whom to collaborate. To learn more about this read "Share Documents" on page 2045.

Commenting in Workspaces and Versions

This functionality is available on Onshape's browser, iOS, and Android platforms.

Collaborating users have the ability to communicate with each other, in workspaces and versions, with comments and mentions. Owners of documents and those the document is shared with directly (and with Edit or Comment permission) have the ability to create comments, mention another user, see each others' comments, leave replies, and opt to receive email notifications of comments.

Comments: Desktop

You can comment generally on the document or modeling space, and you can explicitly tag certain entities directly. The entities you can tag directly are:

- Features in Feature lists
- Mate connectors, including implicit
- Mates
Entities in a sub-assembly

Drawings (keep in mind, however, that currently there is no collaboration allowed on drawings; one user at a time may view the drawing)

The Comments dialog also offers a markup function which enables you to annotate and markup your design space and include it within the comment.

**Accessing the Comment flyout**

Click 📣 in the document title bar to open the comment flyout:

![Comment flyout](image)

Note that the flyout remains open until you close it. Click ❌ in the upper right corner of the flyout to close it.

**What you can do**

The Comments flyout has mechanisms that allow you to:

- Filter comments according to: All comments, All open tasks, Open tasks assigned to the current user, and Completed tasks.
- Tag an entity by clicking this icon first, then an entity either in the graphics area, the Feature list, or the Parts list (in a Part Studio) or the Instances list (in an Assembly).
- Insert an image that will be included in the comment as an attachment.
- Toggle comment notifications on or off. When notifications is turned on, you will receive email notifications of comments you are mentioned in.
- Initiate the markup tool.

Note that the flyout remains open until you close it. Click ❌ in the upper right corner of the flyout to close it.

**Adding comments**
While you are in a specific tab, open the Comment flyout and enter a comment. Any collaborators who have email notifications turned on will receive an email notifying them of the comment, with a link to that document. The link brings the collaborator to the first tab of the document.

Use the “@” sign to direct a comment towards a specific user. If this is the first time you are directing a comment to this user, first share the document with them. In Enterprise, you can only mention users who are in your company. All users in the company will appear in the suggestion list.

Click the Add button and an email is sent to the user containing the comment text and a link to the document. If the user mentioned does not have permission to the document, you are notified and prompted to share:

If the mentioned user has the document open at the time, the Comment flyout opens automatically.

Once the document is shared, an email is sent to the user containing the comment and a link to the document. An additional email is also sent to notify the user that the document has been shared.

Tagging entities (adding comments on features) in Feature lists

To add a comment on an entity in a Feature list (in Part Studios and Assemblies), select the feature in the list and access the context menu. Select Add comment:
Or you can use the Tag entity icon in the Comments flyout, shown below outlined in blue:

Then select the entity: part, edge, face, etc.
Use the “@” sign to specify that a specific user receive an email directing them to the comment. When you click in the Comment flyout (above), a comment icon appears next to the feature in the list (below) or on the entity in the graphics area (both shown below):
When the Comment flyout is closed, the comment icons in the Feature list disappear. The icons are visible only when the Comment flyout is open.

Features in the Feature list will not have comment icons in the graphics area. Their comment icons are displayed in the Feature list.

Adding comments on implicit mate connectors

You have the ability to attach a comment directly to an implicit mate connector.

1. Hover to activate implicit mate connectors, then right-click to access the context menu.

2. Select Add comment to open a new comment on the Comment flyout.

When you click the Add button in the comment flyout, a comment icon appears on the implicit mate connector:
When the Comment flyout is closed, the comment icons in the graphics area disappear. The icons are visible only when the Comment flyout is open.

To delete the comment, click X in the upper right corner of the comment box; then confirm the action.

**Collaborator icons**

Notice that your collaborator icon is shown (because you are active in the document). If you have shared this document with another user, they also see the collaborator icon, and you see theirs. When a user closes the document, the icon disappears, but the comments remain.

**Working with comments**

Owners of documents automatically have comment permission on their documents. When an owner shares a document directly with another user, the owner may grant Comment permission to that user (and also revoke it). All of the permissions that allow editing automatically also allow commenting. The Can view permission also allows commenting.

When users have Comment permission on a document, they are automatically opted in to receive notification emails when:

- A new comment is made
- A reply to a comment thread they have participated in is posted
Users who receive access to a document through an organizational share are not automatically opted-in for email notifications. You may, however, elect to receive email notifications by checking the box in the comment flyout.

**Assigning tasks to users**

You can assign tasks to other users in Onshape using the Comments flyout. When a user mentions other users in a comment, the mentioned user will be displayed below the comment box preceded with Assign to and a check box, as shown below:

![Comments flyout](image)

To assign a task to that user, click the check box next to Assign to. The action button changes from **Add** to **Assign**, as shown below:

![Assign to task](image)

If there are multiple user mentions in the comment, the first user is shown on the assigned section, and a dropdown arrow to the right allows you to select another mentioned user. You are only able to assign a task to one user at a time.

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Once assigned, the task indicates the assigned user at the top (as shown below). The assigned user gets an email notification, as well as a desktop notification, that a task has been assigned to them.

Once the assigned user finishes the assigned task, they are able to resolve the task by clicking on the check box in the upper right corner.

Filter comments

To filter comments or tasks in the comments flyout, click the filter icon in the top right corner of the flyout �骓.
Click on the option you wish to view, and the Comments flyout list will adjust accordingly.

**Including markup in the comment**

To use markup to annotate and make notes directly in the graphics area and then include that as an image attached to the comment:

1. Once the comment flyout is open, click ![markup tool].
2. The markup area is outlined with a blue dashed line and the markup tools appear:
3. Use the tools as you would any graphics tools, to draw arrows, lines, free form, circles, squares, text. Select a color and a size for the graphics.

a. When a tool is selected, the cursor changes to a cross-hair symbol. Click and drag in the graphics area to draw the selected shape, or to draw a text box.

b. To undo an action in markup, use CTRL+z.

c. While you can click and drag to create a markup symbol anywhere and in any direction, you can also use SHIFT+drag to create the symbol at a 45 degree angle:
To center an ellipse/circle or square/rectangle about the initial click point, use CTRL+click while dragging.

To create a circle or square (instead of an ellipse or rectangle), use SHIFT+click while dragging.

Use CTRL+SHIFT+click while dragging to create a circle or square centered about the initial click point.

d. When Text is selected, you first draw a text box, then click in the box to set the cursor, and then type. You can change the color and size while the text box is active.

4. To cancel without saving your markup, click the X in the Markup dialog.

5. To save your markup, click the check mark in the Markup dialog.

Once saved, the markup is uploaded as an attachment to the currently active comment. Once you have added the comment (clicked Add), the markup appears as a thumbnail in the comment:
Note that the markup icon is only available for new comments, not replies to comments.

Tips

- Click the link in a comment to navigate directly to the tab for which the comment was created.
  
  Once in the tab, the link changes to reflect the entity the comment was created on, if appropriate.
  
  Click the new link to highlight that part in the graphics area.

- Comments are associated with a specific document and one of its workspaces, so the set of comments will vary depending on which workspace is active.

- Click the Comment icon again to close the flyout, or click the small x in the upper right corner of the flyout to close it.

- Click Add comment to add another comment.

- Click x replies to create a reply to a comment; click Reply to save it.

- Hover in the box of a comment to access the edit and delete icons.
• If a user doesn’t have edit rights to the workspace (which is inherently true for versions), then there is no access to the Comments flyout.

• Comments are not recorded in the workspace history.

• When an assembly is moved to a new sub-assembly, the comments follow and remain attached.

Comments: iOS

Collaborating users can communicate with each other in a workspace with comments. Owners of documents and collaborators (with Edit or Comment permission) can create comments, see each others’ comments, leave replies, and opt to receive email notifications of comments.

Comments are labeled with each user’s username.

On mobile you can comment on:

• Documents

With Onshape on a browser, you can comment on:

• Features in Feature lists

• Mate connectors

• Mates, including implicit mates

• Entities in a sub-assembly

• Drawings (keep in mind, however, that currently there is no collaboration allowed on drawings; users may view the drawing one at a time)

Accessing the comment flyout

Tap the Comments icon in the upper right of the screen, the comment flyout opens.
On a smaller screen, you may need to access Comments from the document information panel.

Tap the more icon in the upper right, then select **Comments**.

![Untitled document](image)

**Adding comments**

While you are in a specific tab, open the Comment flyout and enter a comment. Any collaborators who have email notification turned on will receive an email notifying them of the comment, with a link to that document. The link brings the collaborator to the first tab of the document.

Use `@<username>` to look up a specific user to direct a comment to.

**Notifications**

Onshape sends notifications if:

- A document is shared with you
- You are specifically mentioned in a comment
- A document that has been shared with you or a document you own has been commented on

Tap on a notification to open the comment flyout in the document.

In your device settings you can specify how, or if, Onshape sends you notifications.

**Tips**

- All comments show the name of the collaborator and a timestamp of when the comment was sent.
Comments are associated with a specific document and one of its workspaces, so the set of comments will vary depending on which workspace is active.

A user must have any edit or comment permissions to the document or there is no access to the comments flyout.

Comments are not recorded in the workspace history.

**Comments: Android**

Collaborating users have the ability to communicate with each other in a workspace with comments. Owners of documents and collaborators (with Edit or Comment permission) can create comments, see each others’ comments, leave replies, and opt to receive email notifications of comments.

Comments are labeled with each user’s username.

On mobile you have the ability to comment on:

- Documents

With Onshape on a browser, you have the ability to comment on:

- Features in Feature lists
- Mate connectors
- Mates, including implicit mates
- Entities in a sub-assembly
- Drawings (keep in mind, however, that currently there is no collaboration allowed on drawings; users may view the drawing one at a time)

**Accessing the comment flyout**

Tap the Comments icon in the upper right of the screen, the comment flyout opens.
Adding comments

While you are in a specific tab, open the Comment flyout and enter a comment. Any collaborators who have email notification turned on will receive an email notifying them of the comment, with a link to that document. The link brings the collaborator to the first tab of the document.

Use @<username> to look up a specific user to direct a comment to.

Assigning tasks to users

You are able to assign tasks to other users in Onshape using the Comments flyout. When a user mentions other users in a comment, the mentioned user will be displayed below the comment box preceded with Assign to and a check box, as shown below:

Once assigned, the task indicates the assigned user at the top (as shown below). The assigned user gets an email notification, as well as a desktop notification, that a task has been assigned to them.

Once the assigned user finishes the assigned task, they are able to resolve the task by clicking on the check box in the upper right corner.

Notifications

Onshape sends notifications if:

- A document is shared with you
- You are specifically mentioned in a comment
• A document that has been shared with you or a document you own has been commented on

Tap on a notification to open the comment flyout in the document.

From the Documents Page, go to Settings to enable or disable notifications from Onshape.

Tips
• All comments show the name of the collaborator and a timestamp of when the comment was sent.
• Comments are associated with a specific document and one of its workspaces, so the set of comments will vary depending on which workspace is active.
• A user must have any edit or comment permissions to the document or there is no access to the comments flyout.
• Comments are not recorded in the workspace history.
• Tap the Comments icon to open and close the comment flyout.

Follow Mode

This functionality is available on Onshape's browser, iOS, and Android platforms.

When users are collaborating in a single document, they have the ability to choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

Follow Mode: Desktop

A user may follow a collaborator across browser and mobile. This means a user on a browser may follow a collaborator on a browser or mobile device. A user on a mobile device may follow a collaborator on a mobile device or on a browser.

To follow someone, double-click their social cue icon in your toolbar.
Double-click the social cue icon at the top of the window. Notice that the workspace is outlined to indicate you are following the collaborator.

The user being followed sees the follower’s social cue outlined to indicate the follower is using Follow Mode:

To stop following, click anywhere in your browser window.

What followers see

- The collaborator’s active tab and actions in that tab
- The collaborator's cursor movements (shown as a hand in the social cue icon color)
- Views and Render modes of parts (accessed from the [icon] menu, including Section view)
- Selections made in the graphics area
- Movements and activations of exploded views
- Selections made in the Feature list
- Part Studio dialog boxes and work done inside dialogs (for editing features, but not when creating new features)
- Part movement and sketching: you will see the part/assembly in its new location after a collaborator moves it, and a sketch while the Sketch dialog is open.

What followers do not see

- Reference manager dialog
- Sketch Text subdialog

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All rights reserved.
• Derived dialog
• Mass properties and Measure dialogs
• Right-side panel and tables, including: Explode views panel, Bill of materials table, Configuration table, Custom table, Appearance panel, and Hole table

Tips
• A single collaborator may have many followers.
• A follower may follow only one user at a time.

Follow Mode: iOS

When users are collaborating in a single document, they can choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

A user can follow a collaborator across browser and mobile. This means a user on a browser can follow a collaborator on a browser or mobile device. A user on a mobile device can follow a collaborator on a mobile device or on a browser.

To follow someone, tap on their social cue icon in the Collaborators section.
To stop following, tap anywhere in the graphics area.

**What followers see**

- The collaborator's active tab and actions in that tab
- The collaborator's cursor movements (shown as a hand in the social cue icon color)
- Views and Render modes of parts (accessed from the [icon] menu, including Section view)
• Selections made in the graphics area
• Movements and activations of exploded views

What followers do not see
• Selections made in the Feature list
• Dialog boxes and work done inside dialogs
• Part movement and sketching; you will see the part/assembly in its new location after a collaboration moves it, and a sketch after the Sketch dialog is accepted.

Tips
• A single collaborator can have many followers.
• A follower may follow only one user at a time.
• For tabs that do not support collaboration (drawings, for example, or third party applications), a user who has shared permissions to edit the document can "steal" focus on a non-collaborative tab.

Follow Mode: Android
When users are collaborating in a single document, they can choose to follow another collaborator. This allows the follower to see the actions of the other collaborator.

A user can follow a collaborator across browser and mobile. This means a user on a browser can follow a collaborator on a browser or mobile device. A user on a mobile device can follow a collaborator on a mobile device or on a browser.

To follow someone, tap on their social cue icon in the Collaborators section.
To stop following, tap anywhere in the graphics area.

What followers see

• The collaborator’s active tab and actions in that tab
• The collaborator’s cursor movements (shown as a hand in the social cue icon color)
• Views and Render modes of parts (accessed from the [icon] menu, including Section view)
• Selections made in the graphics area
• Movements and activations of exploded views

What followers do not see

• Selections made in the Feature list
• Dialog boxes and work done inside dialogs
• Part movement and sketching; you will see the part/assembly in its new location after a collaboration moves it, and a sketch after the Sketch dialog is accepted.

Tips

• A single collaborator can have many followers.
• A follower may follow only one user at a time.
• For tabs that do not support collaboration (drawings, for example, or third party applications), a user who has shared permissions to edit the document can "steal" focus on a non-collaborative tab.
Using the View Only Toolbar

This functionality is available on Onshape's browser and iOS platforms only.

The View only toolbar is the default presentation for all users that do not have write permissions on a document.

Upon entering a document, a Light user or a user with View only permissions will see this layout *(note the View only toolbar - for an Assembly - outlined in blue)*:

*The View only toolbar as seen on a browser, above*
The View only toolbar on an iPad, above, of a user without Comment permissions

A good place for Light users to start understanding Onshape and their Onshape environment is "Getting Started as a Light User" on page 2226.

A user in Enterprise or any other account type will have this toolbar as well, when shared with the View only permissions. Only a company owner has the ability to change their company’s settings to hide the View only toolbar in all documents. Users with editing permissions on a document will not see the View only toolbar.
In addition to the toolbar, you can also right-click on a sketch or part to access a menu of available commands like Show Dimensions, and other commands as well. Double-click on a feature in the Features list to see the settings and options used to create it.

The View only toolbar is located at the bottom of your Onshape Part Studio or Assembly (shown above outlined in blue) and is able to have up to 18 tools or features, some of which depend on the tab type or the user’s permissions:

1. **Home** - Click the Home icon to automatically restore the initial view of your document.

2. **Rotate** - Click the Rotate icon, then click and drag your cursor in the direction you want your document to rotate.

3. **Pan** - Click the Pan icon, then click and drag your cursor in the direction you want to pan your document.
4. **Zoom to fit** - Click the Zoom to fit icon to automatically zoom your image to fit the middle of the screen. Click the dropdown menu arrow to the right of the icon to see the following features:

- **Zoom to window** - Click Zoom to window, then drag your cursor to create your zoom bounding box around the part of the entity you wish to zoom in on; your image zooms accordingly.

- **Zoom** - Click Zoom, then click and drag your cursor up and down or left and right to zoom in and out.

Note that while using the Onshape application on a desktop, you are able to scroll up or down with your mouse at any time to zoom in or out.

On Windows machines with a standard 3-button mouse, Onshape provides the following scheme for manipulating the 3D model in Part Studios and Assemblies:

**3D Rotate**: Right-mouse-button-click+drag

**Zoom in and out**: Scroll up and scroll down, respectively

**2D pan**: CTRL-right-mouse-button+drag (middle button click+drag)

For more information on customizing your view manipulation in Onshape, see "This functionality is available on Onshape's browser, iOS, and Android platforms." on page 192.

5. **Section view** - Click the Section view icon to open the Section view manipulator:
Select the plane, planar face, or mate connector that you want to view, then click the checkmark in the top right corner of the Section view manipulator to finalize your decision.

6. **Properties** - When you are in a Part Studio, click the Properties icon to open a Properties dialog where you will have the ability to edit your Part Studio's name, description, part number, state, and more:

![Properties dialog](image)

To see the properties of a *specific* part on an entity, click the specific part and then click the Properties icon.

In an Assembly, click the Properties icon to open a Properties dialog.

Light users and full users who lack write permissions do not have the ability to edit properties.

**Toggle Appearance panel** - This tool appears only when a Part Studio tab is selected. Click to open the Appearance panel from the right side of the UI:
You can see the color associated with a particular part or face. For more information about assigning colors, see "Customizing Parts, Faces, and Features: Appearance" on page 302.

**Toggle Custom tables panel**

This tool appears only when a Part Studio tab is selected. Click to open the Custom tables panel from the right side of the UI. If a custom table has been added to the document, it opens:
7. **Toggle Configuration panel** - Click the Configuration table icon to open the Configured part properties panel *(note that this icon appears only when your assembly or Part Studio contains configured properties):*
To learn more about the Configuration panel, see Configurations.

8. **Toggle BOM panel** - BOM Click the BOM icon to open the BOM panel, or close the BOM panel:

![BOM panel](image)

9. **Open Exploded views panel** - Click the Exploded views icon to open the panel in order to select an Exploded view to display:

![Exploded views panel](image)

   *This tool only appears when an Assembly is selected.*

10. **Toggle comments panel** -
Click the Comments icon to toggle the comments panel *(note that this icon appears only when the user has permission to comment)*:

To learn more about the Comments panel, see [Comments on Workspaces](#).

11. **Follow a user...** -  
    Click the Follow a user icon to open a menu with a list of users that are currently in the document *(note that this icon only appears if there are multiple users in a document)*:

Click a user you wish to follow, and your Onshape window will adjust to show you their document view in real time, as shown below:
To stop following a user, re-click the Follow a user icon in the toolbar. To learn more about following a user, see Follow Mode.

12. Export tab - Click the Export icon to open the Export dialog:

Here, you have the ability to edit your document's name and export options. Click OK to finalize your decisions.
- **Export selected** - Click the Export selected icon to export all selected geometry (note that this icon appears only when exportable geometry is already selected). An Export dialog like the one above appears; edit your file name, format, version, and export options, and click [OK] to finalize your decisions.

- **Select and export...** - To export a specific assembly or part, click the dropdown menu arrow to the right of the Export icon and click Select and export...(note that this icon only appears if there no exportable geometry is selected). This opens an Export manipulator; click the part or assembly you want to export, then click the checkmark [✓] in the top right corner. This opens the same Export dialog as shown above. Choose your export settings and click [OK] to finalize your decisions.

13. **Print** - Click the Print icon to open the Print setup dialog:

Here, you have the ability to edit your print settings. When you are ready to print, click the checkmark [✓] in the top right corner. This opens a print-preview of your document, with options to choose more printer-specific settings.

14. **Measure** - Hold your cursor over the Measure icon to read how the Measure tool functions:
The above message explains how the Measure tool is automatically shown in the lower right corner of the Onshape window whenever an entity or part is selected. Click on the Measure icon to see an animated example of the tool's functionality.

15. **Mass Properties** - Click the Mass properties icon to open the Mass properties panel:

   ![Mass Properties Panel]

   In a Part Studio, if there are no parts selected, the Mass properties panel displays the Part Studio properties. If there are parts selected, it displays only the properties for those selected parts.

**Users with release management responsibilities**

When a user has release management responsibilities, (Enterprise or Professional users only), they may be responsible for approving a release candidate. There is no Light user tool on the toolbar for this purpose, however, the user must access the release candidate through the notification they received, which may be:

- Email notification with a link
- Internal notification in the document
- Mobile notification with a link

These notifications will contain a View release link that when clicked takes the user directly to the release candidate dialog.
Transfer Ownership

This functionality is available on Onshape's browser, iOS, and Android platforms.

Every document and folder is owned by either a user or a company. Users who belong to a company (in a Professional or Enterprise subscription) create company-owned documents and folders by default. Users who are not members of a company (that is, users in a Free, Education, or Standard subscription) automatically own the documents and folders they create. For users belonging to more than one paid subscription, a drop down is included in the Create document/folder dialog with which you are able to select the owner at the time of creation.

Keep in mind that ownership can be transferred from one owner to another, and from one company to another. This topic covers that information, and includes a discussion of how transferring ownership can sometimes affect metadata and in particular, release management.

Transfer Ownership: Desktop

Owners of documents/folders and owners of companies have these permissions on documents/folders they own: Delete, change sharing privileges, make Public, make Private, and Transfer ownership. Ownership may be transferred at any time, by the owner or company admin, through the Transfer ownership dialog:
Transferring ownership

Users who own documents or folders are able to transfer that ownership to another user, or to a company of which they are a member. In general, when transferring ownership to a user, that transfer recipient must accept the transfer in order for the action to be complete. Be aware there are "Special cases and notes" on page 2109 in addition to this scenario. For company/enterprise-owned documents, only the administrators of the company/enterprise can transfer ownership.

To transfer ownership to an individual:

1. Right-click on the document or folder (on the Documents page) and select Transfer ownership:
This opens the Transfer ownership dialog:

![Transfer ownership dialog]

2. From here, at the top of the dialog, select *Individual* to transfer ownership to an individual.

3. In the empty space that reads Search users, enter the email of the person you wish to transfer ownership to.

4. Click **Transfer ownership**.

To transfer ownership to a company:

1. Select *Company* to transfer ownership to a company.

2. Select the company to which to transfer ownership from the drop down list.

3. Click **Transfer ownership**.

At this point, the transfer of ownership is pending until that user or company accepts the transfer request.

If both the user who is transferring the ownership of a private document and the user to whom the ownership is being transferred are part of the same company plan, the transfer is automatic and doesn't require explicit acceptance. This is also true when a company user is transferring ownership of a private document to the company. This also means that there is no opportunity to revoke the transfer.
When transferring ownership between companies, remember that all the metadata of a given document is controlled by the document ownership as well. That means that all the company metadata around parts, assemblies, drawings, etc is controlled by the company properties set by a professional or enterprise plan administrator. Note that in order for the company properties to be applied to a document, that document must be owned by the company (not an individual user). Standard, Free, and Education plans do not have company-owned documents and therefore no properties that can be applied to documents on behalf of a company.

Ownership and in particular, company metadata/properties, become an important consideration when releasing objects and transferring ownership. Generally speaking, when working with a consultant, instead of transferring ownership to the user, do the following:

- In a Professional subscription - Simply share the document with edit permissions to the user unless the user needs to create releases - in that case it's best to add them to the company subscription.

- In an Enterprise subscription - Simply add the consultant as a "Full Guest" user to edit the document/data. However, if the user needs to create release candidates, the user must be added to the "Create Releases" global permission.

**Canceling ownership transfer**

When transferring ownership from one individual to another (not members of the same company), that transfer must be accepted by the recipient in order to complete the transfer. During the time between the issue of the transfer and the acceptance, the original owner may revoke the transfer by right-clicking on the document in the Documents page and selecting Cancel ownership transfer from the context menu:
The act of transferring ownership gives the recipient View only permission to the document, and this permission remains even if the transfer has been revoked.

**Accepting transfer request**

Once you click the Transfer button in the dialog, the document/folder appears on the recipient's Documents page with their name and Transfer pending alongside it in the Owner column.

To complete the transfer, the recipient must accept the transfer by right-clicking on the document or folder and selecting Accept Ownership.

**Declining transfer request**

Once you click the Transfer button in the dialog, the document/folder appears on the recipients Documents page with their name and Pending alongside it in the Owner column.

If the user does not want to be the owner of the document, they have the ability to decline the transfer by right-clicking and selecting Decline Ownership.

At the time of the initial transfer request being sent, the recipient of the request receives View only permission on the document. When the user declines the transfer request, the View only permission remains.

**Special cases and notes**

- If you create a document or folder and give ownership to a company, you gain Full
access permission, allowing you to edit, share/change permissions, make Public/Private, and delete. However, since the company is the owner, the company owner is then able to remove your access to the document, or change your permissions.

- Ownership is able to be transferred from:
  - **Individual to individual** - Requires acceptance of the transfer unless both users are members of the same company
  - **Individual to company** - Does not require acceptance of the transfer; and user must be a member of the company
  - **Company to individual** - Requires acceptance of the transfer unless the user is a member of the company
  - **Company to company** - Only if the company owner or admin is a member of both companies

- You are unable to transfer ownership to a user without an active Onshape account.
- Ownership is an implied share with edit and share permissions, however, before the transfer is accepted the pending owner has permission to only view and comment on the document.
- Even when a user declines ownership, they keep their previous share permissions, or retain view and comment only permissions if they were not previously shared on the document.
- Transferring ownership doesn't change any existing Shares or links to the document.
- The user transferring ownership retains any Edit and Share permission on the document; unless this is changed by the new owner.
- Any user shared on a document may remove themselves from the share or be removed by the new owner.

**Transfer Ownership: iOS**

This functionality is available only in Onshape on a browser at this time.
Transferring ownership

Users who own documents can transfer that ownership to another user or to company of which they are a member. In general, when transferring ownership to a user, that transfer recipient must accept the transfer in order for the action to be complete. Be aware there are "Special cases and notes" on page 2109 in addition to this scenario.

To transfer ownership:

1. Click **Share** either on the Documents page with the document selected, or with the document open. (Alternatively, you can use the gear menu for a specific document on the Documents page and select **Share**...):

   ![Share settings](image)

   On the Documents page: select a document, expand the gear menu, and select **Share**:

   ![Recently opened](image)
2. Once the Share dialog is open, enter the user's email address on the *Individuals* tab.

To transfer ownership to a company, select the *Companies* tab and select the company in the drop down.

3. Select the permission level (Owner) from the drop down:

4. Click Transfer:

5. Click Close.

At this point, the transfer of ownership is pending until that user accepts the transfer request.
If both users are part of a the same company plan, and when transferring ownership to a company, the transfer is automatic and doesn't require explicit acceptance. This also means that there is no opportunity to revoke the transfer.

**Revoking transfer request**

When transferring document ownership from one individual to another (not members of the same company), that transfer must be accepted by the recipient in order to complete the transfer. During the time between the issue of the transfer and the acceptance, the original owner of the document can revoke the transfer:
1. Open the Share dialog for the document.

![Share dialog for transferring ownership](image)

2. You can assign a different level of permission through the drop down, or you can click the x to the right of the drop down to remove all permission, including the pending ownership.

The act of transferring ownership gives the recipient View only permission to the document, and this permission remains even if the transfer has been revoked.

**Accepting transfer request**

Once you click the Transfer button in the Share dialog, the document appears in the recipient's Documents list with their name and (Transfer pending) alongside it in the Owner column.

To complete the transfer, the recipient must accept the transfer:
1. Use the gear menu and select Accept Ownership:

![Gear menu with Accept Ownership option]

2. Or click the Share button (from the Documents page or from within the open document) and click Accept:

![Share settings for Free document 8 - Copy]

**Declining transfer request**

Once you click the Transfer button in the Share dialog, the document appears in the
recipients Documents list with their name and (pending) alongside it in the Owner column:

<table>
<thead>
<tr>
<th>Owned by</th>
<th>Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>DiFree Amadeo</td>
<td>4 MB</td>
</tr>
</tbody>
</table>

If the user does not want to be the owner of the document, the transfer can be declined:

1. Use the gear menu and select Decline Ownership:

2. Or click the Share button (from the Documents page or from within the open doc-
(document) and click Decline:

At the time of the initial transfer request being sent, the recipient of the request receives View only permission on the document. When the user declines the transfer request, the View only permission remains.

**Special cases and notes**

- If you create a document and give ownership to a company, you gain Full access permission to that document, allowing you to edit, share/change permissions, make Public/Private, and delete the document. However, since the company owns the document, the company owner can then remove your access to the document through the Share dialog, or change your permissions.

- Document ownership can be transferred from:
  - Individual to individual - Requires acceptance of the transfer unless both users are members of the same company
  - Individual to company - Does not require acceptance of the transfer; and user must be a member of the company
• Company to individual - Requires acceptance of the transfer unless the user is a member of the company

• Company to company - Only if the company owner or admin is a member of both companies

• You cannot transfer ownership to a user without an active Onshape account

• Ownership is an implied share with edit and share permissions even before the transfer is accepted.

• Even when a user declines ownership, they keep their previous share permissions, or retain View only permissions if they were not previously shared on the document

• Transferring ownership doesn't change any existing Shares or links to the document

• The user transferring ownership retains any Edit & Share permission on the document; unless this is changed by the new owner

• Any user shared on a document can remove themselves from the share or be removed by the new owner

Transfer Ownership: Android

This functionality is available only in Onshape on a browser at this time. Refer to the Browser Help topic Transfer Ownership for more information.

Transferring ownership

Users who own documents can transfer that ownership to another user or to company of which they are a member. In general, when transferring ownership to a user, that transfer recipient must accept the transfer in order for the action to be complete. Be aware there are "Special cases and notes" on page 2109 in addition to this scenario.

To transfer ownership:

1. Click Share either on the Documents page with the document selected, or with the document open. (Alternatively, you can use the gear menu for a specific document on the Documents page and select Share...:
2. Once the Share dialog is open, enter the user’s email address on the Individuals tab.

To transfer ownership to a company, select the Companies tab and select the company in the drop down.
3. Select the permission level (Owner) from the drop down:

4. Click Transfer:

5. Click Close.

At this point, the transfer of ownership is pending until that user accepts the transfer request.
If both users are part of a the same company plan, and when transferring ownership to a company, the transfer is automatic and doesn't require explicit acceptance. This also means that there is no opportunity to revoke the transfer.

**Revoking transfer request**

When transferring document ownership from one individual to another (not members of the same company), that transfer must be accepted by the recipient in order to complete the transfer. During the time between the issue of the transfer and the acceptance, the original owner of the document can revoke the transfer:
1. Open the Share dialog for the document.

![Share dialog](image)

2. You can assign a different level of permission through the drop down, or you can click the x to the right of the drop down to remove all permission, including the pending ownership.

   The act of transferring ownership gives the recipient View only permission to the document, and this permission remains even if the transfer has been revoked.

**Accepting transfer request**

Once you click the Transfer button in the Share dialog, the document appears in the recipient's Documents list with their name and (Transfer pending) alongside it in the Owner column.

To complete the transfer, the recipient must accept the transfer:
1. Use the gear menu and select Accept Ownership:

![Gear menu with Accept Ownership option]

2. Or click the Share button (from the Documents page or from within the open document) and click Accept:

![Share settings for document]

Declining transfer request

Once you click the Transfer button in the Share dialog, the document appears in the
recipients Documents list with their name and (pending) alongside it in the Owner column:

<table>
<thead>
<tr>
<th>Owned by</th>
<th>Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>DiFree Amadeo</td>
<td>Transfer pending</td>
</tr>
</tbody>
</table>

If the user does not want to be the owner of the document, the transfer can be declined:

1. Use the gear menu and select Decline Ownership:

2. Or click the Share button (from the Documents page or from within the open doc-
At the time of the initial transfer request being sent, the recipient of the request receives View only permission on the document. When the user declines the transfer request, the View only permission remains.

**Special cases and notes**

- If you create a document and give ownership to a company, you gain Full access permission to that document, allowing you to edit, share/change permissions, make Public/Private, and delete the document. However, since the company owns the document, the company owner can then remove your access to the document through the Share dialog, or change your permissions.

- Document ownership can be transferred from:
  - Individual to individual - Requires acceptance of the transfer unless both users are members of the same company
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- Company to individual - Requires acceptance of the transfer unless the user is a member of the company
- Company to company - Only if the company owner or admin is a member of both companies
- You cannot transfer ownership to a user without an active Onshape account
- Ownership is an implied share with edit and share permissions even before the transfer is accepted.
- Even when a user declines ownership, they keep their previous share permissions, or retain View only permissions if they were not previously shared on the document
- Transferring ownership doesn't change any existing Shares or links to the document
- The user transferring ownership retains any Edit & Share permission on the document; unless this is changed by the new owner
- Any user shared on a document can remove themselves from the share or be removed by the new owner
Document Management

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape captures the state of every tab in a workspace every time an edit is completed (by all users working in that workspace). This information is also preserved for versions. This means that for every document there is an infinite record of states in which it has existed. This is very valuable because you never have to worry about constantly saving your work. You are always able to make changes with confidence that if the changes don't work out, you are able to find and restore any earlier state. In addition, you always have the ability to use version, branching, and merging to explore multiple design variations in parallel, either on your own or with collaborators. Onshape also captures and preserves release package candidates as versions in the Version and History flyout.

Document Management: Desktop Terms

- **Version** - A progress marker in the history of a document. Versions are immutable and capture the complete scope of a workspace at a particular point in time for future reuse or to revert a set of changes.

- **Workspace** - An active modeling/design space within a document.

- **Branch** - A branch is a named fork in the Version Manager graph of a document. Branches fork at a version, end with a workspace, and can have zero to N sequentially stored versions on the branch. The active branch is the branch of the document in which the currently open version or workspace is located.

- **History entry** - A record of a change made to a document workspace at a particular point in time. You can compare history entries and restore the document to a particular history entry (point in time) through the context menu in the Versions and History flyout.
• In progress - The default release state for unreleased workspaces and versions. Objects in progress are either fully editable (in workspaces) or have editable metadata (in versions).

• Pending - The state of a Release candidate (and its revisionable objects) while awaiting approval by one or more approvers; indicated by this icon in the Version and history graph ▲.

• Rejected - The state of a Release candidate (and its revisionable objects) that one or more approvers chose to reject; indicated by this icon in the Version and history graph ▲.

• Released - The state of a Release candidate (and its revisionable objects) that was either successfully approved by one or more approvers or was immediately released by its creator, indicated by this icon in the Version and history graph. This icon also appears beside any released parts (in the Parts lists of a Part Studio) when viewing a released version of the document ▲.

• Observer - Any member of a company who needs to be informed of the status of a release, but whose approval is not required in the release workflow. Any number of observers (or none) can be included on a Release candidate. An observer must have view permission on the document in order to observe the Release candidate.

• Approver - A company member whose approval is required in the release workflow. An approver has the ability to Approve or Reject a Release candidate. An approver must have permission to edit the document in order to approve or reject a Release candidate.

• Revisionable object - Any part, Assembly, Drawing, or other file type in an Onshape document can be revisioned and released in Onshape.

• Release candidate - A user-selected group of revisionable objects that moves through the release workflow together. A Release candidate may contain a single part or an entire product including parts, Assemblies, Drawings and other files.

• Not revision-managed objects - Objects that may need to be included in a Release for reference purposes but whose revisions do not need to be tracked.
To learn more about how document history and versions work in Onshape, you can follow the self-paced course here: Document History and Versions (opens in new tab).

About documents

Onshape documents contain all of your project data and all of your work is automatically recorded. When working in a document, you work in an active workspace. When you create an Onshape document, one version and one workspace are automatically created for you (Start version and Main workspace). The Main workspace is also empty until you begin modeling.

The icons in the image above (from left to right) are: Document menu, Versions and history flyout, and Create a version. The bold name is the Document name and the lighter name is the Workspace name.

The Main workspace and Start version are automatically created for you. These can be renamed.

About versions

A version is the state of an entire document at a particular point in time. The geometric data (and the accompanying properties like Part names, etc.) of that version is unchangeable. You have the ability to, however, change the properties of a version (more on this later). You may create many versions of a document. You may also
branch your work at a version through the context menu command Branch to create a new workspace:

The features shown in the image above include (from top to bottom):

- Create Release icon (+), which opens a dialog from which you can create a Release candidate.

- Create version icon (••), which allows you to create a version from the currently open workspace.

- Compare history entries icon (||), which allows you to select two entries to compare.
The currently open workspace, highlighted in dark blue.

The currently selected workspace, highlighted in light blue.

**Creating versions**

To create a version behind the scenes and still work in the same workspace, use the Create version icon in the title bar (next to the document name).

This creates a version (visible in the Version and history flyout) without moving you away from the current workspace. The default naming convention is Vx: V for version, and x for the incremented number, starting from 1. You are always able to rename the versions.

The top arrow in the image above points to V2, the second version saved through .

The bottom arrow in the image above points to V1, the first version saved through .

Otherwise, open the Manage versions and history flyout and create versions from the current workspace (using the Create version icon ).

**Accessing version and history information**

Managing the workflow around versions and workspaces is performed through the Versions and history flyout, accessed by clicking .

Onshape automatically records the state of each tab (Part Studio, Assemblies, etc) at each persisted change made to all tabs for every workspace by every user. This history of modifications is listed in the flyout. At any time you are able to click Restore (in the context menu) to restore the branch/workspace to a particular point in its change history, click Return to <branch-name> to return to the currently active branch at its current point in history, and click a history entry to visualize the design at that point.

This graph displays all versions, workspaces, and releases of a document, in tree form. The graph is color-coded by branch. Every branch ends with a workspace,
which are depicted as open dots. Versions are depicted as solid dots and are View only. Releases are depicted as triangles (solid for released, open for pending, and solid with a dot for obsoleted).

To simplify the view of a version graph, you can collapse the view to just the branch in which the currently active workspace resides by clicking Active branch at the top of the flyout.

The description of the record in the History list is "Tab-name::Action:Feature-name". In addition, you have the ability to:

- Hover over an entry in the History list to see who made the modification, and when.
- Click after hovering to visualize the document at that history point (including the feature listed).
- "Comparing" on page 2156 two history entries of the document.

**Viewing changes and restoring a document to a specific moment in history**

Onshape keeps a history of all the changes made to every document by every user. This enables you to view those changes in one list in the document history, and also to restore a document to a specific point in history at any time.

When restoring, the workspace you are in is the one being restored. If you wish to restore to another workspace, open that workspace first.

**Listing changes**

To see all the changes made by every user who has worked on the document, since the creation of the document, click the Manage versions and history icon in the title bar of the document. Look for the Show changes entries and click the arrow to display the changes:
Every action is automatically stored and listed and serves as a complete audit trail. Changes are recorded for every action taken on the document since it was created, regardless of user or tab. When a list of changes exceeds more than 25 entries, the first 25 are shown. Click Show more to show more entries.

Restoring to a point in history

At any time, you are able to select a moment in the document's history to restore to. Remember that the Restore command is available only within a workspace. You are
not able to restore a version, since versions are immutable. You are able to, of course, restore a workspace and then create a version of the restored workspace.

To restore:

1. Click to open the Manage versions and history flyout.
2. Click the > arrow next to Changes in the list.
3. Right-click the change entry and select **Restore to**, to restore the workspace to that moment in the document's history.

![Screenshot of Onshape interface showing version management]

4. To view the document at that moment in time before deciding whether to restore or not, click the entry.

   a. If you wish to restore after viewing the change, click the Restore to link above the Manage versions and history flyout:
Restoring is different from undo because you are able to restore all changes made to the document up to that point in the history, by any user on any tab. Undo, by contrast, reverts just the changes you made on the current tab. Right-click on the Undo icon to see a short list of changes you have made and select one to undo to that point without having to click the Undo button several times.

Properties for workspace, versions and releases

You can create and edit properties for parts, Part Studios, Assemblies, and foreign data in order to support your preferred design processes.

Properties are defined and edited through the Properties dialogs found on the context menus:

- **Parts** - In a Part Studio, from the context menu on the part listed in the Feature list; in the Version and history flyout, in the menu of a version or workspace

- **Part Studios and Assemblies** - Through the context menu on the tab; in the Version and history flyout, in the menu of a version or workspace

- **Foreign data files** - Through the context menu on the tab; in the Version and history flyout, in the menu of a version or workspace
Releases - Through the context menu on the release in the Versions and history fly-out.

Document Management: iOS

Onshape documents are uniquely engineered to allow users to:

- **Share documents** - send/receive invitations to work collaboratively and even simultaneously in a document.
- **Manage versions** - use Version Manager to create versions and branches of a document workspace with the ability to merge later.
- **View History** - view all recorded changes in the document's history and restore the document to any point in that list.

When working in a document, you are working in an active **workspace**. When you create an Onshape document, one version and one workspace are automatically created for you (Start version and Main workspace). For new, empty documents the Start version contains an empty Part Studio and an empty Assembly. The Main workspace is also empty until you begin modeling.

A document may have many workspaces; one on each branch.

Share Documents

You can invite other Onshape users to view and make changes to your document. Likewise, you can receive invitations to view and make changes to another user's document. Via the Share feature, you can work simultaneously with other users.

See "Share Documents" on page 2045 for more information.

Version Manager

Onshape provides a graphical representation of versions and workspaces in the Version Manager. As you and your team work in the document, you can at any time mark a new version of the workspace. You can also create a new workspace from a version. Note that marking a version and creating a new workspace are separate actions.

See "Version Manager" on page 2138 for more information.
History

Every persisted change to a tab in a workspace is recorded in the document's History. You can access the list of history records and from there, you can restore your document to any recorded point in its history. Records are listed for all tabs in the currently active workspace.

See "Version Manager" on the next page for more information.

Document Management: Android

Onshape documents are uniquely engineered to allow users to:

- **Share documents** - send/receive invitations to work collaboratively and even simultaneously in a document.

- **Manage versions** - use Version Manager to create versions and branches of a document workspace with the ability to merge later.

- **View History** - view all recorded changes in the document's history and restore the document to any point in that list.

When working in a document, you are working in an active **workspace**. When you create an Onshape document, one version and one workspace are automatically created for you (Start version and Main workspace). For new, empty documents the Start version contains an empty Part Studio and an empty Assembly. The Main workspace is also empty until you begin modeling.

A document may have many workspaces; one on each branch.

Share Documents

You can invite other Onshape users to view and make changes to your document. Likewise, you can receive invitations to view and make changes to another user's document. Via the Share feature, you can work simultaneously with other users.

See "Share Documents" on page 2045 for more information.
**Version Manager**

Onshape provides a graphical representation of versions and workspaces in the Version Manager. As you and your team work in the document, you can at any time mark a new version of the workspace. You can also create a new workspace from a version. Note that marking a version and creating a new workspace are separate actions.

See "Version Manager" below for more information.

**History**

Every persisted change to a tab in a workspace is recorded in the document's History. You can access the list of history records and from there, you can restore your document to any recorded point in its history. Records are listed for all tabs in the currently active workspace.

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**Version Manager**

This functionality is available on Onshape's browser, iOS, and Android platforms.

You can create versions (which are View only) and branch a version to create a new workspace. You can also compare history entries in workspaces and versions, and any combination of the two. For more information, see Comparing.

To learn more about how branching and merging work in Onshape, follow the self-paced course here: Branching and Merging (opens in new tab).

**Versioning and Branching: Desktop**

**Creating a version**

You can create a version from the title bar; click to create a new version without leaving the workspace.
You can also create a version through the Versions and history flyout:

1. With a document open, click § to open the Versions and history flyout.
2. Click the workspace from which to create a version; this makes that workspace active.
3. Click the Create version icon $.
4. In the dialog that appears, enter a name and description for the new version.
5. If any drawings in the document have updates pending, as indicated by a bright yellow Update icon 🔄, a message is included in the dialog:

   ![Create version dialog](image)

   Click the icon in the message 🔄 to update all drawings that are listed so the version will contain the most recent version of the drawings. Note that if you click the Update icon and then cancel the dialog, the drawings are still updated.

6. Click either:
   a. Create - Create the new version and remain in the currently active workspace
   b. Create version and edit properties - Create the new version and also open the Properties dialog for the new version. This Properties dialog includes names and descriptions for each tab and part of the new version.
Note that the new version is shown in the Versions and history graph:

![ Versions and history graph ]

**Creating a branch**

To create a branch of a document:

1. With a document open, click on the Versions and history flyout.
2. Right-click on the version entry from which to create a branch to access the context menu.
3. Select Branch to create workspace:

4. In the Properties dialog that appears, enter the name and description for the branch:

5. Click:

   a. - Create the branch and optionally open the new workspace (check box)

   b. - Create the branch (optionally open it) and also open the Properties dialog for the new workspace. This Properties dialog includes names and descriptions for each tab and part of the new document workspace.

You can compare two workspaces, two versions, or a version and a workspace. You are also able to merge one or more branches.

**Branching example**

A team is working on a bicycle and has reached a stable design base for the frame. Now it's time for the team to experiment with various component designs. Begin by marking the basic frame design document as a version.
1. Click the Manage versions and history icon to open the Versions and history flyout:

![Versions and history flyout](image1)

2. Click the Create Version icon.

3. Name the first version Base Frame; click Create.

![Create version dialog](image2)
Each designer is then able to create their own workspace from the Base Frame version, perhaps labeled Seat, Brakes, Shocks.

4. From the flyout, on Base Frame version, select Branch to create workspace.... from the context menu:
5. Name the new workspace, Seat; click Create:

The Base Frame version still exists as well as the original workspace (Main). In addition, there is a second workspace, Seat, so the designer making changes for the seat won't affect the Base Frame workspace.

6. The Shocks and Brakes designers each create their own workspaces, from the same Base Frame version:
The workspace of each designer is, at this point, identical to the Base Frame version. As they continue to design in their own branches, their designs evolve, independently of the Base Frame version and independently of each other.

As they work, they are free to create versions of the workspaces.

**Version Manager: iOS**

With the Version manager, you can create versions (which are read-only) and branch a version to create a new workspace.

**Open the Version manager**

You can open the Version manager from the Documents page or from within the document:

- From the Documents page:
  
  Select the Information panel icon that corresponds with the document for which you want to open the Version manager. From here tap **Versions**.

- From within the document:

  Tap the Versions icon in the upper left:
Overview

- The Version manager displays the default version graph for a document, notice there is a **Main** workspace, and a "Start" version by default.

- There are two icons in the upper right of the Version manager:
  - ![+] Create a version
  - ![X] Close Version manager

- Tap to switch between viewing **All branches** and the **Active branch** (workspace that is active, or open, in the graphics area):

![Version manager]

The graph displays all versions and workspaces of a document, in tree form. The graph is color-coded by branch. Each branch is one color, and every branch ends with a workspace, which are depicted as open dots. Branch names are shown with bold and italic text. Versions are depicted as solid dots, and normal text, and are View only.
The active branch, which is the currently open workspace, is indicated by a blue box to the left of the title. In the image above, the branch titled *branch sketch* is the active branch as is indicated by the blue box to its left.

**Terms and indicators**

- **Version** - A named and saved state of the document. Versions are immutable and separate from workspaces. Create a version to capture a workspace at a particular point in time. You can open and branch a version.

  Indicated by normal text and a solid dot on the version graph.

- **Workspace** - A modeling/design space.

  The active workspace is indicated by a blue box to the left of it in the version graph list.

- **Branch** - Where the document is split and a new workspace is created. A branch can have many versions, but always has a workspace. You can open and delete a branch.

  Indicated by bold and italic text and an open dot on the version graph.

**Creating a version**

To create a version of a document:

1. Access the Version manager.
2. Open the workspace (or branch) from which to create a version.

   A workspace (branch) is indicated by an open dot.

   To do this, tap to select desired workspace, then tap the Open icon.

3. Tap †+.

4. Add a name and description for the version, then select **Create**.

5. The newly created version appears in the Version manager:

   Tap on a version to see details and options such as Open, Branch, and Restore.
Creating a branch

To create a branch of a document:
1. Access the Version manager.

2. Tap to select the version from which to create a branch:

3. Tap the Branch icon:

4. Add a name and description for the branch, and then select Create.

5. The newly created branch appears in the Version manager.

   Note that both b1 and Main are branches and are indicated as such by the bold and italic text as well as the open dots in the version graph.

   Also note that "v1" and "Start" are saved versions and are indicated by the normal text and the solid dots in the version graph.
The branch titled *Main* has two versions, "v1" and "Start", as is indicated by the blue color of the line. Branch titled *b1* does not yet have any saved versions, it is the only item on the yellow line.

**Restoring a version**

Onshape keeps a history of all the changes made to every document by every user. This enables you to view those changes in one list in the document history, and also to restore a document to a specific point in history at any time.

When restoring, the workspace you are in is the one being restored. If you wish to restore to another workspace, open that workspace first.

To restore a version to the currently active workspace:

1. Open the Versions flyout.
2. Tap the version you wish to restore.
3. Tap the restore icon at the top of the flyout.

For more information about restoring versions, see "Document Management" on page 2127.

**Version Manager: Android**

With the Version manager, you can create versions (which are read-only) and branch a version to create a new workspace.

**Open the Version manager**

You can open the Version manager from the Documents page or from within the document:

- From the Documents page:
  
  Select the Information panel icon that corresponds with the document for which you want to open the Version manager. From here tap Versions.

- From within the document:

  Tap the Versions icon in the upper left:
Overview

- The Version manager displays the default version graph for a document, notice there is a Main workspace, and a "Start" version by default.

- There are two icons in the upper right of the Version manager:
  - Create a version
  - Close Version manager

- Tap to switch between viewing All branches and the Active branch (workspace that is active, or open, in the graphics area):
The graph displays all versions and workspaces of a document, in tree form. The graph is color-coded by branch. Each branch is one color, and every branch ends with a workspace, which are depicted as open dots. Branch names are shown with bold and italic text. Versions are depicted as solid dots, and normal text, and are View only.

The active branch, which is the currently open workspace, is indicated by a blue box around the title. In the image above, the branch titled \textit{b2} is the active branch as is indicated by the blue box around it and to its left.

**Terms and Indicators**

- **Version** - A named and saved state of the document. Versions are immutable and
separate from workspaces. Create a version to capture a workspace at a particular point in time. You can open and branch a version.

Indicated by normal text and a solid dot on the version graph.

- **Workspace** - A modeling/design space.
  
The active workspace is indicated by a blue box to the left of it in the version graph list.

- **Branch** - Where the document is split and a new workspace is created. A branch can have many versions, but always has a workspace. You can open and delete a branch.
  
  Indicated by bold and italic text and an open dot on the version graph.

**Creating a version**

To create a version of a document:

1. Access the Version manager.

2. Open the workspace (or branch) from which to create a version.

   A workspace (branch) is indicated by an open dot.

   To do this, tap to select desired workspace, then tap the Open icon.

3. Tap +.

4. Add a name and description for the version, then select **Create**.

5. The newly created version appears in the Version manager:
   
   Tap on a version to see details and options such as Open and Branch.
Creating a branch

To create a branch of a document:
1. Access the Version manager.

2. Tap to select the version from which to create a branch:

   ![Version manager interface](image)

   **Modified by Nicholas Amadeo**

   **12:59 PM Today**

   **Description:**
   
   version one...

   **Contents:**
   
   - Part Studio 1
   - Assembly 1

3. Tap the Branch icon:

   ![Branch icon](image)

4. Add a name and description for the branch, and then select **Create**.

5. The newly created branch appears in the Version manager.

   Note that both **b2** and **Main** are branches and are indicated as such by the bold and italic text as well as the open dots in the version graph.

   Also note that "v1" and "Start" are saved versions and are indicated by the normal text and the solid dots in the version graph.

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Copyright © 2017, Onshape. All rights reserved.
The branch titled \textit{Main} has two versions, "v1" and "Start", as is indicated by the blue color of the line. Branch titled \textit{b2} does not yet have any saved versions, it is the only item on the yellow line.

\textbf{Restoring a version}

Onshape keeps a history of all the changes made to every document by every user. This enables you to view those changes in one list in the document history, and also to restore a document to a specific point in history at any time.

When restoring, the workspace you are in is the one being restored. If you wish to restore to another workspace, open that workspace first.

To restore a version to the currently active workspace:

1. Open the Versions flyout.
2. Tap the version you wish to restore.
3. Tap the restore icon at the top of the flyout.

For more information about restoring versions, see "Document Management" on page 2127.

\textbf{Comparing}

This functionality is currently available only on browser.

Onshape provides a mechanism for graphically and discretely comparing versions and workspaces at any history point. You are able to compare any combination: workspaces with workspaces or versions, versions with version or workspaces, and at any history entry if desired.

Access the Compare action from the context menus in the Versions and history flyout, or use the icon at the top of the flyout \(\uparrow\). Each method of initiating Compare is explained below.
Using Compare, icon

To compare a workspace with a workspace or with a version, or compare a version with a version or a workspace:

1. With a document open, click to open the Versions and history flyout.
2. Click .
3. Select any two history entries (a workspace or a version, or any history entry within a workspace or a version).

   To select a particular history entry within a version or a workspace, expand the Changes entry before clicking the Compare icon.

4. Select the two history entries to compare. The first selection is highlighted in blue, and the second selection is highlighted in red.

Using Compare, command menu

When you open the Versions and history flyout, the currently active workspace or
version is highlighted (this is referred to as the Base).

To compare to another version or workspace -or a specific history entry- (the Target), open the context menu for the Target and select Compare, as described above.

**What you see**

By default, Compare shows:

- A list of features for each history entry selected; the first selection is displayed on the left side of the flyout and the second selection is displayed on the right side of the flyout.

- A visual representation of the differences between the history entries, with a slider bar to show more or less of each history point. The changes are represented by color: blue for the Base (or first history entry selected) and red for the Target (or second history entry selected):

When a Configuration exists, the Compare dialog includes it and you can select the Configuration to compare:
What you can do

1. Reverse the list to show the Target on the left and the Base on the right (click Reverse compare)

2. Show the differences between only the current Part Studio and the target Part Studio, see the illustration above.

   Comparing Assemblies is not yet supported.

3. List all features of both the Base and Target (instead of just the differences):

4. Control how many of the Base or Target features are displayed graphically through the slider mechanism:
Visualize more of the Base features by sliding the circle towards the Base label (the blue label):

Visualize more of the Target features by sliding the circle towards the Target label (the red label):
5. Select the tab to see a list of all Part Studios in the document from which to select and see the differences within each.

**Interpreting the lists**

The list comparing features or Part Studios between the Base and Target uses the following icons to denote differences:

- No icon - The Base and Target feature, sketch, or Part Studio are identical
- ≠ - The Base and Target feature, sketch, or Part Studio are not identical
- > - The Base has a feature, sketch, or Part Studio that the Target does not have (this feature or Part Studio does not exist in the Target)
- < - The Base is missing a feature, sketch, or Part Studio that the Target has (this feature or Part Studio does not exist in the Base)

**Tip**
In order to see changes involving more than one part more easily, you can use the context menu > Show/Hide Part:

*Reverse compare*

Tweak

BeforeTweak

Hide Part 2
Hide all parts
Clear selection
Zoom to fit
Zoom to selection
View normal to
View material for Part 2...

*Before hiding Part 2, above*
After hiding Part 2, above, you can visualize the changes to Part 1 more easily.

Merging

This functionality is available on Onshape's browser and iOS platforms only.

Onshape provides a mechanism for merging from a document version or workspace (referred to as the Source) into your currently active document workspace (referred to as the Target). Specifically, when you merge a selected Source (workspace or version) into the currently active Target (workspace), all changes made in the Source are merged into the Target, including any additional features, tabs, etc.

Access the Merge action from the context menu in the Versions and history flyout:
How it works

When you open the Versions and history flyout, the currently active workspace is highlighted (this is referred to as the Target).

To merge another workspace or version (the Source) into the active workspace (the Target), open the context menu for the Source and select Merge into current workspace:
In iOS

Tap the Versions and history icon 📚 to open the panel:
Tap the information icon next to the Source workspace and tap Merge:
All changes made in the Source are merged into the currently active workspace (Target). This action is recorded in the Versions and history flyout entries and you can restore from a previous record to reverse the merge action, if necessary. For more information about restoring versions, see Document Management.

When merging workspaces containing drawings, images, PDFs or other tabs that are not Part Studios or Assemblies, if changes have been made to the tab in both Source and Target branches, then the changes in the Source branch overwrite the changes in the Target branch. For example, if you update a PDF tab in both branches (Source and Target) and then merge the branches, the PDF in the Source branch will be in the Target branch after the merge.

Tips
When merging workspaces containing drawings (as in a Drawings tab), the drawing that has changes is favored during the merge, and when the drawing has changes in
both the Source and Target branches, the Source drawing will be favored, specifically:

- If a drawing in the Source has changes that are not in the drawing in the Target, then the drawing in the Source is copied into the Target, replacing the drawing in the Target. Any changes made in the drawing in the Target that are not in the Source will be overwritten.

- If a drawing in the Source workspace has no changes (compared to the drawing in Target), then the drawing in Target is left unchanged.

We recommend that you work in a drawing in one workspace (branch) and merge from that branch into other branches. Working in a drawing on two or more branches simultaneously may result in lost changes when you merge the drawing from one branch into another.
Release Management

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

This functionality is available to users of Professional and Enterprise subscriptions only. Enterprise users, however, can create custom release and obsoletion workflows specific to your own processes. See "Creating a Customized Release Workflow" on page 2320 for more information.

Release management is a set of automated workflows in Onshape used to manage releasing revisions of parts, assemblies, drawings and imported files (translated or not) in a document. This functionality is available only for the Onshape Professional and Enterprise subscriptions and allows you to establish workflows and tools for your entire company.

Overview

Onshape Release management is non-blocking and is built directly into Onshape's design and version control system, requiring no additional products, installs or extensive IT management. All users in your company are automatically provisioned with the latest workflow settings and updating is automatic for all users, so there is never a situation where someone is accessing an out-of-date platform.

A typical workflow starts by identifying objects to release, including parts, assemblies, drawings and other imported file data, and listing them in a Release candidate. The Release candidate may then be submitted for approval or released immediately by its creator, depending upon the workflow rules defined by the administrator.

Administrators have access to company- or enterprise-wide settings to define approval requirements for each release, and to set specific conditions for determining a valid release. In addition to users who need to approve a release, observers may also be assigned to a given release, giving them permission to view the Release
candidate without the ability or responsibility to approve it or reject it. Settings like these may be defined according to the company's needs.

Additional key aspects of Onshape Release management include:

- Users always have access to their designs and are never locked out of editing parts while waiting for a Release candidate to be approved. The team is able to continue to design in parallel, and does not have to wait for the approver to review the design.

- There is never a question of "Do I have the latest changes?" Users receive notifications when a new revision of a released object is approved. Users are not notified when new versions of released objects are created, thereby preventing false notifications of updates.

Enterprise users of custom multi-tier release workflows should keep in mind that notifications are sent within the tier the action takes place in. For example, when an approval in tier 2 of a multi-tier workflow is initiated, all approvers and observers in that tier are notified. No one in any other tier is notified except for the creator of the release; creators receives all notifications.

- Released objects are able to be quickly identified and accessed with native search capabilities on the Documents page, and linked into other documents for use in assemblies, drawings, and Part Studios.

**Terminology**

Onshape Release management depends on a number of terms and concepts that are critical for understanding the details of this powerful set of workflows:

- **Version** - A progress marker in the history of a document. Versions are immutable and capture the complete scope of a workspace at a particular point in time for future reuse or to revert a set of changes.

- **Workspace** - An active modeling/design space within a document.

- **Active branch** - The branch of the document in which the currently open version or workspace is located.

- **History entry** - A record of a change made to a document workspace at a particular point in time. You can compare history entries and restore the document to a
particular history entry (point in time) through the context menu in the Versions and history flyout.

- **In progress** - The default release state for unreleased workspaces and versions. Objects in progress are either fully editable (in workspaces) or have editable metadata (in versions).

- **Pending** - The state of a Release candidate (and its revisionable objects) while awaiting approval by one or more approvers.

- **Rejected** - The state of a Release candidate (and its revisionable objects) that one or more approvers chose to reject.

- **Released** - The state of a Release candidate (and its revisionable objects) that was either successfully approved by one or more approvers or was immediately released by its creator.

- **Observer** - Any member or members, or team of a company who needs to be informed of the status of a release, but whose approval is not required in the release workflow. Any number of observers (or none) can be included on a Release candidate. An observer must have view permission on the document in order to observe the Release candidate.

- **Approver** - A company member, members, or team whose approval is required in the release workflow. An approver has the ability to Approve or Reject a Release candidate. An approver must have permission to edit the document in order to approve or reject a Release candidate.

- **Revisionable object** - Any part, Assembly, Drawing, and any file in an Onshape document can be revisioned and released in Onshape.

- **Release candidate** - A user-selected group of revisionable objects that moves through the release workflow together. A Release candidate may contain a single part or an entire product including parts, Assemblies, Drawings, and other files.

- **Not revision-managed objects** - Objects that may need to be included in a Release for reference purposes but whose revisions do not need to be tracked. You can mark entire Part Studios, Assemblies, Drawings, parts and files as Not revision-managed. Keep in mind that setting the entire document as Not revision-managed includes all parts.
Workspaces, Versions and Releases

Onshape Release management is integrated directly into Onshape's underlying version control system. The Release workflow starts in a workspace where all parts, Assemblies, Drawings and other revisionable objects are editable and have, by default, an *In progress* state.

During the design process, you may create any number of versions to mark your progress.

Creating a Release from an in-progress workspace or a from version prompts you to fill out a Release candidate. Once completed, you either submit the Release candidate for approval or release it immediately, depending on the company's defined workflow settings.

When a Release candidate is submitted for approval, a version is created, the version is automatically named for the Release and marked with an open triangle icon ▲ in the Version and history graph. The state of the revisionable objects within the Release candidate is also set in the version, to *Pending*.

When a Release candidate is approved (released), the version is marked with a solid triangle, ▲, and the state of the revisionable objects within the Release candidate in the version are set to *Released*. 

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Similarly, if one or more approvers rejects the Release candidate, the version is marked with an open triangle, ▲, and the revisionable objects within the Release candidate reflect a state of Rejected.

If one or more objects in the Release are obsoleted, the version is marked with a solid triangle with a dot, ⬤, and the obsoleted objects within the Release reflect a state of Obsolete.

It is critical to note that state changes (eg. from Pending to Released or Rejected) take place on a single version. That version contains the immutable source of truth for the released revision of each object in the Release.

During this process, the team can continue to edit any workspaces in the document, create versions or even create additional releases. None of the workflow actions described above ever block progress.
Setting up Release Management

This functionality is currently available only on browser.

Only a company or Enterprise administrator is able to set up the details of the Onshape Release management workflow for the company, but any user in the account is able to use these tools in the company documents as long as they have edit permissions on the document (or documents) containing the objects for release.

To set up Onshape Release management tools:

1. In the User menu, select **Company settings**.
2. Select the **Release management** tab in the left pane.
3. To use the Onshape Release workflow and tools, select **Enable managed workflows**.
4. Proceed through each of the next sections (below) to define the conditions that govern the release workflow and preferences that work for your company.
5. When you have made your selections, click **Save release settings** (at either the top of the page or the bottom).

You have the option to choose to forgo using Onshape's Release management workflows and tools by *not* selecting **Enable managed workflows**. This turns Release management tools off and you use your own procedures and processes to manually change revision properties and metadata. To learn more about manual release workflows, see the technical briefing titled [Release Workflow & Data Management. (opens in new tab)]

The Onshape Release management workflow is illustrated by the diagram on the Company settings > Release management page:
See "Viewing Revision History and Obsoleting Parts" on page 2217 for an explanation of using Onshape's workflow for obsoleting parts.

Setting revision and part number preferences
Under Revision Scheme, select the automatic revision labeling scheme that suits your company's needs:

- **Alphabetical** - Use the alphabet, sequentially, starting at A and omitting I, O, Q, S, X, Z
- **Numerical** - Use numerals, sequentially, starting at 1
- **New custom revision scheme** - Use a text file containing your own revision scheme. The text file must list one revision per line and contain a minimum of 50 lines. Once you specify a file, it is maintained in this list and is selectable as a custom revision scheme until you delete it. Selecting a custom revision scheme marks the beginning of the use of the scheme and no previously-used revision scheme is replaced. No previously-marked revisions are ever changed, despite selection of a
different or new revision scheme.

Note that when using the alphabetical scheme, certain letters are skipped automatically, and prohibited from use, because they can be confused with some numerals.

**Revision scheme id** is the identifier of the revision scheme. This becomes important when creating your own release management workflows. See "Creating a Customized Release Workflow" on page 2320 for more information.

**Unreleased revision suffix** is the character or characters that follow the current revision in the Drawing Revision table. The default character is an asterisk (*) and you can specify any character or set of characters. See "Revision Table" on page 1938 for more information about revision tables in drawings.

Select the preferred method of part number generation:

- **Manual entry** - Manually enter your own part numbers.
  - Onshape automatically prevents duplicate revision labels or part numbers and displays an error message if you try to use an existing part number of a previously released revision. For example, if you release Revision A for part
number 01, you cannot then release a Revision A for part number 01 for that part or any other part.

- Onshape does not track the sequence of numbers for this scheme and allows you to skip numerals.

- **Sequential part number generation** - Onshape generates part numbers on request.
  
  - Use the fields in the flow chart to define prefixes for each object type, minimum length, and the next numeral in the starting sequence.
  
  - The Next field indicates the next part number to be generated, with successive part numbers increasing in the series, never decreasing even if numbers have been skipped or releases have been discarded or rejected.
  
  - A preview of the generated part number is shown according to the specified parameters:

    ![Part number generation diagram](image-url)

    - Upon selecting this option, no previously manually entered part numbers in the system change. Changes apply only to newly created part numbers.
    
    - Selecting this option does not prohibit you from manually entering part numbers if you wish to do so.
    
    - Once this option is selected, automatic part number generation is available in all properties dialogs for objects that can have a part number.
    
    - The Prefix field is optional.

- **Part number uniqueness** - Select whether or not to allow the use of duplicate part numbers for drawings (duplicates of the part or assembly represented in the
All part numbers in a release must be unique - Do not allow any part number of a part or assembly to be used as the part number of a drawing.

Drawing can reuse part number from an assembly or part in the release - Allow the part number of a part or assembly to be used as the part number of its drawing.

**Part number propagation** - Select whether to propagate part numbers from a released item in a version to a corresponding item in an associated workspace. Select:

- **Never** - To never propagate the released item's part number to a corresponding item in a workspace
- **One workspace** - To propagate the released item's part number to a corresponding item in a workspace when the version the item was released in has only one corresponding workspace. If the version has more than one corresponding workspace, then do not propagate the part number.
- **All workspaces** - To propagate the released item's part number to a corresponding item in all workspaces when the version the item was released in has one or more corresponding workspaces.

How and when part numbers are applied:

- For items that aren’t configured assemblies, the part number is applied to the items with the same internal id in the workspace, if it exists.
- For parts, the part number is only applied if the part is found in the workspace as exactly one part.
- For configured Part Studio and Assemblies, the part numbers are applied only if the configuration parameters are the same between the version and the workspace. If any parameters have been added or removed, the part number is not applied.
- Part number propagation occurs on the first transition of a release package, that is, the Submit action.

**Access**
Administrators may apply rules governing who has access and permissions to objects and data during the release process. You are able to select more than one:

- Only administrators can edit the properties of released objects, see "Editing properties of objects" on page 2197.
- Only administrators can mark objects as not revision-managed, see "Marking objects as not revision-managed" on page 2203.
- Only administrators can delete documents containing released objects.

**Actions**

Administrators are able to specify that a watermark appear for all unreleased drawings by selecting the empty box next to the Show watermark for unreleased drawings option. The watermark is a simple "In Progress". It is removed when the drawing is released, and when a drawing is obsoleted an "Obsolete" watermark is placed on the drawing.

You also have the ability to hide the watermark for drawings that are not revision managed by selecting the empty box next to that option (shown below).

![Actions](image)

**Release conditions**

Administrators are able to define the conditions that, when true, will prohibit a Release candidate from being created. You are able to select more than one:

- **When releasing a part if its Part Studio Feature list contains errors** - Select this to prohibit the release of a part if the Part Studio Feature list has any errors.
- **When releasing a part if its Part Studio Feature list rollback bar is not at the end (of the Feature list)** - Select this to prevent the release of any part with the rollback bar not at the end of the Feature list. Best practice is to delete any unwanted or unused Features from the Feature list and keep the rollback bar at the end of the Feature list to avoid confusion in the future.
• **When releasing an Assembly if its Assembly list contains errors** - Select this to prevent the release of any Assembly if the Assembly list contains errors.

• **When releasing an Assembly if it has pending reference updates** - Select this to prevent the release of any Assembly if any object within it has a pending reference update. For more information on updating references, see [Updating References](#).

• **When releasing a Drawing if it has a pending update** - Select this to prevent the release of any Drawing if the drawing is pending Update.

• **When releasing an item whose part number is included in another pending release candidate** - Select this to prevent the release of any item whose part number is included in another pending release candidate.

• **When releasing an item which has a pending task** - Select this to prevent the release of any item within a Part Studio when any item within that same Part Studio has a pending task. When this condition exists, a message is displayed in the Release candidate dialog; the Release candidate can be created, but the Release action is unavailable until the assigned task has been addressed. If this condition is not checked on the Release conditions page, a message is still displayed when the condition exists, but the item can be released.

**Release dialog**

Define the supporting details that must be present in the Create Release candidate dialog before a release is able to be submitted or released. You are able to select more than one:

• **Require an approver** in the Create Release candidate dialog:
  • Check this to make the Approver field mandatory in the Create Release candidate dialog. An email address will then be required in this field before the Release candidate can be submitted or released.
  • Leave this unchecked to allow the creator of the Release candidate to immediately create a Release, skipping the Release candidate and approval step.

• **Do not allow creator as approver** - When you require an approver on the release, you can also add this option so the creator's name doesn't appear in the list when
entering required approvers. This prevents the creator of the release candidate from simply going ahead and approving the candidate as the approver.

• **Require approval from all approvers:**
  • Check this to require that all approvers listed in the Create Release candidate dialog actually approve the Release candidate in order to proceed with the Release. If one approver rejects the Release candidate, then the candidate is rejected, even if all other approvers have approved it.
  • This does not enforce the number of approvers listed for the Release candidate.
  • Leave this unchecked to allow the Release candidate to be approved by any one of the listed approvers.

• **Require a note in the Release dialog:**
  • Check this to require the creator of the Release candidate (and any person who approves/rejects the release candidate) to enter notes about the release in this field before the Release candidate can be created or released.
  • Leave this unchecked to allow the Release candidate to be submitted or released without any text in this field.

• **Restrict approvers to these users or teams:**
  • Professional users see this option and can check this to create a list of approved approvers. Enter the names or email addresses of specific users and/or teams to create a list of users from which approvers must be selected when approvers are entered in the Release dialog.
  • Enterprise users can also make these specifications, but through the Approve releases Global permission. Select Global permissions on the Company settings page.

• **Allow adding items from other documents:**
  • Check this to allow the creator to add items that are not from the original document into the Release candidate.
  • Leave this unchecked to prevent the creator from adding items from other documents to the Release candidate.

**Obsoletion**
Check this to automatically obsolete previously released versions. When you re-release a previously released object, the previous release is automatically obsoleted upon approval of the re-release. Right-click on the tab of your drawing or object, and click Revision history to view the object's automatic obsoletion:

Delegating approval permission

Users who may be asked to approve releases can delegate that approval. This is
convenient for planned absences as any user can specify another user to be notified of pending release candidate approval requests and given permission to act on the request. Upon specification of a delegate, the user or team members named as delegate are notified of the assignment. The user who is delegating the approval responsibility sees a message every time they sign in or refresh an Onshape session:

You have delegated your approval rights to another user or team. You can change this setting in your account preferences.  

Delegated approval requests cannot be delegated to another user. For example, if User-A delegates approval to User-B, and User-B delegates approval to User-C, User-C will get approval requests that specified User-B, but not approval requests that specified User-A.

To specify a user as your delegate:

1. Open the User menu in the top right corner of the window.
2. Select My account.
3. On the My account page, select Preferences.
4. Scroll down to Release management.
5. Check the box next to Delegate approval to another user or team.
6. Supply the team name or email of the user or users (or click in the box to select from a list of known users and teams).
When something is released, both the user who delegated and the user or users who are delegates receive a notification.

---

**Specifying Approvers**

![Keyboard icon] This functionality is currently available only on browser.

By default, you are not required to list anyone as an approver on a release; the creator of a release may also be the approver. You are able to, of course, require an approver and also require that the approver must not be the creator, through the Release management settings under the Company settings page.

Also available is the ability to restrict who may be specified as an approver (whether an approver is required or not).

**Creating a list of approved approvers**

How to specify your list of approvers varies with the type of Onshape subscription you have.

- **Onshape Professional users, use the Release management settings on the Company settings page** - Look under the Create Release candidate dialog heading for:

  **Restrict approvers to these users or teams** - Check this box to access a text field. Enter the email addresses of each user (or select from the list that appears) exactly who is able to be specified as an approver on releases. You are also able to specify a team as an approver. When a team is specified, this allows the team or any of the team members to be listed in the Approver field of a release.

- **Onshape Enterprise users, use the Global permissions on the Company settings page** - Look for the global permission:

  **Approve Releases** - Right-click and select Manage users to specify individuals and/or teams that can be included in the Approvers field for Release candidates.
If you have a project role under which you want users to approve release candidates, you can enter the project role name in the "Approvers" field of the Create release candidate dialog:

The field outlined in light blue, above, called Approve releases is a project role

- When using project roles to specify approvers for a release candidate, the following standards apply:
  
  - The first user (in the project role) to approve the release, approves for the entire project role. Only one approval per project role is sufficient to represent all users in the project role.
  
  - If a member of the project role is also listed singly as an approver, then their approval counts only as an individual approver and someone else with the project role must approve for the project role.
If a team or project role are listed as approvers, and a user is a member of both, the team approval is accounted for first and another user with the project role must approve for the project role.

Delegation of approval doesn’t affect project roles: if a member of a role delegates to another user, that user approves it only for that individual, not the project role.

**Typical Release Workflow**

This functionality is currently available only on Onshape’s browser platform.

Creating a release can be done only on the browser, but the rest of the steps (reviewing, approving, rejecting, etc) can be done on any Onshape platform.

These instructions present the basic workflow of releasing any object, demonstrated with a single part.

There are four ways you are able to start the release process which is to open the Create Release candidate dialog for a selected entity, illustrated here with a single part:

- **In a Part Studio, from the Parts list:** Right-click the part you want to release and select Release:
This method pre-populates the Create Release candidate dialog with the selected part.

- **For any tab (Part Studio, Assembly, Drawing, or imported file):** Right-click on the tab and select Release:

  ![Tab Menu](image)

  This method pre-populates the Create Release candidate dialog with the contents of the tab. In the case of a Drawing, it pulls in the part or assembly the drawing is of as well.

- **From any Properties dialog:** Right-click on a part, Part Studio, Assembly, Drawing or File tab and select Properties. In the top-left corner of the dialog, click ![Properties Button]. This method pre-populates the Create Release candidate dialog with the part selected or the contents of the tab selected.

- **From the open Versions and history flyout:** Click ![Versions Button]. This method pre-populates the Create Release candidate dialog with objects from the currently active tab.

Upon starting the release process with the methods explained above:
1. The Create Release candidate dialog opens:

![Create Release candidate dialog]

You can resize this dialog and its sections by clicking and dragging an edge or corner.

New or modified fields are marked with a yellow triangle in the upper left corner of the field. While in this dialog, you can use the Undo and Redo buttons or your operating system hot keys to undo or redo changes.

2. The part you selected in the Parts list is already listed in this dialog. (You are able to use the plus sign + to add more objects.) If you selected an Assembly, all of the parts in the Assembly are listed.

You are able to check any object's design before submitting by clicking the object's name in the list. The appropriate tab will open in another browser tab.

You have the ability to remove any objects not necessary to the Release by clicking the red x next to the object's name.

3. Notice that the Revision is automatically supplied, based on the choices made during set up (alphabetic, numeric or custom) and any prior releases of this object.

The Revision is incremented automatically for new releases. Note that revisions are applied only to objects in the release process, which are never objects in a workspace. As soon as a release candidate is created, a version is captured and all the objects in the release candidate are objects in that version. The objects in
the workspace do not have Revision labels unless specifically manually assigned in the Part properties.

You can change the Revision, but you are not allowed to go backward in order. For example, if the previous revision is B, then Onshape will suggest C. You may enter any letter after C in the alphabet as well, but you may not change to A.

4. The object's current State is also automatically supplied:
   - *In progress* reflects objects being released for the first time.
   - *Pending* reflects objects whose release is pending approval.
   - *Released* reflects previously-released, linked objects.

5. Supply a unique part number for the part:
   a. Onshape tracks part numbers across Releases and prevents the re-release of a lower revision of an existing part number.
   b. If you have Sequential part number generation turned on in Release management settings click the 🔄 icon to generate a part number. Clicking 🔄 at the top of the dialog generates part numbers for all of the objects in the Release candidate.

6. Based on how the administrator set up the rules, some fields may be mandatory and some may not be. Supply information, if required or desired, in each of the following fields:
   a. **Release Name** - This is required by Onshape and will be displayed as the version name in the Versions and history flyout (where you can later view the release information).
b. **Release Notes** - This may or may not be required by your company administrator and is a good place for specific information or instructions regarding the Release.

i. **Approvers** - Enter the email addresses of users or the names of teams of users who must approve this Release candidate before the objects are released. (This may or may not be required by the company administrator; if this field remains empty then the objects are immediately released upon clicking Release.) If an email address or team is present, then a Release candidate is created, approvers are notified and the objects are not released until an approver indicates approval.

When listing a team in the Approvers field, only one member of the team is required for approval, even if *Require approval from all approvers* is selected (in the release management work flow set up).

c. **Observers** - Enter the email addresses of users or the names of teams of users to whom you want to give View only rights to the Release candidate (or Release). You may enter observers even if you do not require an approval. Observers receive notifications.

7. Click one of the following based on the desired outcome:

- **Close** to close the dialog without recording any changes or taking any actions.
- **Apply** to record the information entered, but without creating a Release or closing the dialog.

This is usually done to record any properties entered via the Properties icon next to each object in the top half of the dialog.

- **Submit** (visible when approvers are listed) to create a version, and set the version icon, Release candidate, and its objects as *Pending*.

A notification is sent to each approver and observer listed. A blue message bubble also appears at the top of the graphics area stating the release has been submitted (or other action) and contains a link to the *Action items* page where you can monitor and act on the release process.
The message stating you can view the status on the Action items page

The Action items page

To move from Pending to Released and complete the release workflow, the Release candidate must be approved.

- **Release** (visible when no approvers are listed) to create a version and set the version icon, the Release candidate, and its objects as Released, completing the release workflow.

**Best practices**

- The user who creates the Release candidate should not be the only required approver; best practice is to require an additional or different person in the Approver field.

- Pay attention to good design practices (especially the limited or temporary use of suppression and the rollback option) to keep parts from being generated improperly.

- Make sure drawings are updated before being included in a release. If an updated drawing is available, you will be notified at the time you create the release candidate that the "Drawing has a pending update."

- Be careful with Assemblies to make sure there are no errors before including in a release candidate.
Releasing assemblies

To release an Assembly:

1. Right-click on the Assembly tab and select Release to open the Create Release candidate dialog.

   The Create Release candidate dialog is populated with everything listed in the Assembly's Bill of Materials, including all parts, subassemblies, and standard content referenced by the Assembly.

2. Released parts linked from other documents, in the Assembly, are reflected in the Create Release candidate dialog and no new revisions of the parts are created when the Assembly is released:

   ![Create Release candidate dialog](image)

   If you reference a part in a workspace, the Revision label of that part will bump up when releasing the Assembly. If you reference a part that is already released, then the Revision will not be bumped when releasing the Assembly.

   Keep in mind that Revisions only bump up when the candidate is actually released / approved.

   If there are two pending release candidates that contain the same part with the same pending revision value, the first approved one wins and takes the new
revision value. Release candidates are never automatically discarded for you, nor are you required to take action on the pending one. The pending release indicates that the part's pending revision has already been taken and the only action is to either discard or reject.

3. If you are using Onshape's sequential part number generator, click 🌠 at the top of the dialog to generate part numbers for all of the objects in the Release candidate.

4. Fill in the remaining necessary fields, as described in "Typical Release Workflow" on page 2187, above, to complete the workflow steps.

Note that any *standard content* parts in the Assembly are included but will not have a revision.

**Releasing drawings**

To release a Drawing:

1. Right-click on the Drawing tab and select **Release** to open the Create Release candidate dialog:
The Create Release candidate dialog automatically populates with all of the parts or assemblies referenced by the Drawing.

Released parts linked from other documents in the Drawing are reflected in the dialog as *Released*, and no new revisions of the parts are created when the Drawing is released.

2. Proceed filling out the dialog as described above in the "Typical Release Workflow" on page 2187 instructions to complete the workflow steps.

When you are releasing drawings of an assembly and the drawings of its parts, the release dialog includes a listing of all the parts involved in all the drawings. This results in a release candidate dialog with parts repeated. These parts are not being released multiple times, they are simply listed for reference. For example, Part 1 in the example below is in both the drawing of the assembly and its own drawing, so it is listed twice:
If the drawing has a pending update, you are notified at the time of creating the release candidate. This is for informational purposes only, you are not required to update the drawing before creating the release candidate.

**Searching and filtering the object list**

The Create Release candidate dialog allows you to search and filter the objects list, and you are able to search for parts, Assemblies, Drawings and other objects by name:
Filter the list by object type:

1. Click the Filter icon ‣ to expand the filter list.
2. Click a type icon (Assembly, Part) to toggle it on and filter the list for objects of that type.
3. Click multiple icons to filter for several types at once.
4. To clear the filter, click the type icons again to deselect, or click the Clear button to remove all filters.
5. Click the Filter icon ‣ again to collapse the filter list.

**Editing properties of objects**

The Create Release candidate dialog has a mechanism to view and modify the Properties for any given object:
1. While in the Create Release candidate dialog, click the Properties icon to open the Properties view for that object:

Once in the Properties view, you are also able to select other objects in the list to view and modify their properties.

2. Click Apply to save changes to all objects without leaving the Properties view.

3. Click Back or the Back arrow icon (on the lower left) to return to the main Create Release candidate dialog view. This saves metadata changes while you continue to work in the dialog.

4. Click Apply or Submit/Release to save all changes, including those made in the Properties view.

By default, configured parts properties in released objects cannot be edited. However, if the Only administrators can edit the properties of released objects checkbox is checked (during the set up of Release management), then Admins can edit configured parts properties of released objects.

**Refreshing objects within a release candidate**

While creating a release candidate, you can make sure you have the latest iteration of
the objects in your release candidate. Use the Refresh button to the left of the dialog title.

Create Release candidate

When creating a release candidate, you can visually check the objects included in the candidate to make sure everything looks right. If something looks like it might be out of date, you can click the link that is the object name (in the list of objects), to open a new tab with the appropriate Part Studio, Assembly, or Drawing. At this point, you can simply ensure that everything is correct, or make any necessary changes. Once you are satisfied with the object in its tab, return to the tab with the Release candidate dialog open and click the Refresh button.

Onshape applies the changes made to the objects in the Release candidate dialog and refreshes the interface to reflect those changes.

For example:

The release dialog is all set to go, but something doesn't look quite right:

The image of Part 1 is questioned, it seems to be missing fillet features
Click on Part 1 in the list of objects to open a new tab with that part's Part Studio open so you can inspect the object. In this example, the expected fillet features are missing, so they are added to the part.

Now return to the tab containing the Release candidate dialog. The thumbnail of the part still has no fillets, so click the Refresh icon to update the part to include the changes that were just made:

![Create Release candidate dialog](image)

The part is reloaded and contains the fillet features created in the other tab. Even if the changes made are not visible in the Release candidate dialog, once you click the Refresh button, they are included. When you click Apply or Submit, a version is created and all currently updated features are included in that version. If another user is simultaneously making changes, the changes made for the Release candidate take precedence and are recorded in the version that is created. Other changes are recorded in the Version and history, as normally done.

Keep in mind that the Refresh button is available only when creating a Release candidate, and when the Release candidate dialog is open and only for the creator of the release candidate.
You don't have to refresh the dialog for every change. There may be times when you want to ignore a change to the model and create a release candidate that doesn't reflect latest modification.

Similarly, you can use the Refresh button to update a release candidate after fixing errors. For example:

![Create Release candidate dialog](image)

*A part belonging to a Part Studio with a Feature list that is in a rolled-back state cannot proceed through the release candidate creation process*

Click the name of the Part to open a new tab with the Part Studio in it and correct the rollback bar. Return to the tab containing the Release candidate dialog box and click Refresh to update the part in the candidate:
The problem is resolved and this release candidate can be submitted for review and approval.

Releases and linked documents

Part Studios, Assemblies, and Drawings in one document can all reference objects in versions of other documents, thereby creating linked objects. Linked objects are automatically included in a Release candidate, but how all objects are handled during the release process is affected by whether or not they are linked.

- If an object is in the current workspace (an unlinked object), when the Release candidate is created the Release state is initially In Progress since workspaces are always editable by definition. Releasing objects from the current workspace creates a version in the current document, and marks the object and version as Released (or Pending approval if approval is required).

- If an object is linked from an unreleased version in another document, when the Release candidate is created the Release state is initially In Progress. Releasing linked objects from unreleased versions operates no differently from releasing objects in the current workspace, except that the version in the linked document
will also be marked as Released and the Release attached to the version will be the one from the current document.

- If a linked object is linked to a release that is Pending approval, it will have a **Pending** state. Since the linked object is Pending rather than Released, you will be able to modify its metadata and Revision. Releasing the linked object changes its state to **Released** and an additional Release will be attached to the linked version, overriding the previous **Pending** Release candidate.

- If a linked object is linked to a Released version, then that object will have a Released state. Including a Released object in a new Release does not change the object’s State or Revision. State and Revision are included for your reference.

**Marking objects as not revision-managed**

You may occasionally have an object (or even an entire document) that you don’t want to mark with a revision. These objects may need to be included in a Release for reference purposes but there is no need to track their revisions. Onshape provides a way to mark the object as not revision-managed (not receiving a revision during any release process):

1. Access the Properties dialog for the object (Assembly, Document, Drawing, or File) through any of:
   a. The Workspace menu > Properties command
   b. The Versions and history flyout context menu > Properties command
   c. The Create Release candidate dialog.

2. Click the Properties icon. To mark the object as not revision-managed, click the **Not revision-managed** checkbox towards the bottom of the Properties dialog.

Objects that can be marked as not revision managed include: Part Studios (including all parts within the Part Studio), Assemblies, Drawings, Files, and individual parts.
Selecting Parts for a Release Candidate

This functionality is currently available only on browser.

Once a Release candidate has been created, you can include additional parts, Assemblies, Drawings, or files from the document you are currently in that exist in the currently active workspace. Parts derived from another document in a Part Studio or linked from another document in an Assembly are included automatically in the Release candidate. Users (Enterprise only) can also add objects from another document to a release candidate.

Adding objects to a Release candidate

Create a Release candidate and when the Create Release candidate dialog is open, you may begin to select additional objects for release:

1. Click the plus sign 🔄 at the top left of the dialog (indicated above).
2. Select Part Studios, Assemblies, files, or Drawings:
3. You are able to use the search box to enter names or partial names of the selected object type. Note that if you are part of an enterprise, you also have the option to insert items from outside the current document.

Selecting an object on the left adds it to the list of objects to be included in the release, on the right.

When you add a top-level item from either a different version of the same document, or a version of another document, an icon appears beside the added item indicating it is linked from another version:

4. Click **Add** when you are ready to add the listed items (on the right side of the dialog) to the Release candidate in the Create Release candidate dialog.

Use the red X to remove an object from the Release candidate, if necessary.

If you add a part that is part of another release with a state of Pending, the dialog notifies you:
5. Fill out the rest of the Create Release candidate dialog as necessary and dictated by the release rules your company administrator has set up.

Reviewing, Approving, Rejecting Candidates

This functionality is available on Onshape's browser, iOS, and Android platforms.

When you create a Release candidate, you can require that it is reviewed and approved before being released by entering email addresses for a particular user or team of users in the Approver field. (Note that enterprise users can also enter Roles in this Approver field, if that has been set up by their administrator.) These users receive email notifications, and notifications within Onshape, containing links to view the Release.

You can also notify other users of a release by entering email addresses in the Observers field. These observers, however, can only view the Release candidate and
cannot release, reject, approve, or discard it. Users named as approvers must be shared into the document with at least View permission to approve releases.

In the notification, the link to **view release** is the first of the light blue links at the bottom of the specific notification and opens the Review Release dialog. You can also use the **view action items** link to navigate directly to the [Action items](#) page for a list of release tasks.

Only one notification per Release is sent and subsequent actions on the Release result in the notification being updated to reflect the new status. Emails, however, are sent with every action, including when a comment is added to a release candidate.

Users with permissions to the document may also view release status through the Versions and history flyout. Status is indicated by icons. The flags shown are visible in the context sub-menu:

- ▲ = Released [Released](#)
- ▼ = Pending [Pending](#)
- ▼ = Rejected [Rejected](#)
- ⚗ = Discarded releases become regular Onshape versions
Reviewing release candidates

There are multiple ways to view a Release candidate:

- Use the link in the notification
- Use the link in the email
- Open the Versions and history flyout (use the icon), then use the context menu to select Releases, then the name of the release.

A Release candidate that is submitted for review and approval is marked as Pending in the Versions and history flyout's context sub-menu, and the State of any unreleased objects in the Release candidate will also be Pending.
When reviewing:

- Make sure all information in the dialog is correct.

- Check the Revision label to make sure the appropriate revision of the object is listed. Objects receive an automatic revision label, but you are able to override it and supply a label that has not been used yet (you also cannot use revision labels that have been skipped).

- To review the model, drawing, or other object type directly, click its name (which is a link), to open the Part Studio, Assembly or other appropriate Onshape tab in another browser tab.

- Use the to open the Properties view for that object in order to review or edit the properties.

- Use the appropriate button to Approve (Release) or Reject an entire Release:
  - Release marks the objects as released (in the Versions and history flyout, all subsequent Release dialogs, and in the Revision history dialog).
- Reject marks the Release candidate, previously listed as Pending in the Versions and history flyout (with an open triangle icon), as Rejected (still with an open triangle icon).

- Discard removes the Revision and discards the Release; the Release is marked as a normal Onshape version in the Versions and history flyout. This option is available only to the user who created the Release candidate.

  When discarding an obsolete release, the release is still marked as Released in the Versions and history flyout.

- Note that in the Review Release dialog the Part number column refers to the part number for the specified object: part, assembly, drawing, etc. "Part" is used generically here, in a company-wide context.

- If Require a note in the Release dialog is selected in the Release management settings, then the reviewer must enter text in the Release notes box.

- Anyone can enter a comment to be associated with the release candidate, and other users listed on the candidate are notified of the comment via email and also in the Notifications area in the document. Click the comment history link to view all comments entered by all reviewers. Comments can be entered at any time in the release cycle, even after a release has been approved, rejected, or obsoleted.

- In the Approvers field, an approver’s action is reflected by the color of their email address:

  - Green indicates approval and release
  - Red indicates rejection
  - Gray indicates a pending status
  - Since only the creator of the Release has the ability to discard, there is no color change for that

**Approving release candidates**

When a release candidate is approved, automatic notifications are sent to users listed in the Release dialog as approvers and observers. Click Release to mark the object as released in the Version and history flyout, all subsequent Release dialogs, and in the Revision history dialog.
In the Approvers field, an approver's action is reflected by the color of their email address:

- Green indicates approval and release
- Red indicates rejection
- Gray indicates a pending status
- Since only the creator of the Release has the ability to discard, there is no color change for that; only the creator sees the Discard button in the dialog.

When you mark a release as Approved, drawings properties are also updated with the name of the approver as well with the date of approval. To learn more about using properties in drawings, see "This functionality is currently available only on Onshape's browser platform." on page 1753

**Rejecting release candidates**

When a release candidate is rejected, it is flagged as *Rejected* in the Versions and history context sub-menu and has an open Release icon. Rejected release candidates may be used just as any other version in Onshape.

**Discarding release candidates**

The named Approver of a Release candidate is able to discard the Release candidate when it is a Pending Release candidate, through the Review Release dialog. Discarding a Release candidate deletes the Release candidate but keeps a history of it, marking it as a version in the Versions and history flyout. Keep in mind that only the user who created the Release candidate has the ability to discard it. This option is not visible to other users (non-creators of the Release candidate).

**Branching from a release**

You are able to branch from a Release, using the [Versions and history flyout](#) as you normally would.
Released, Pending, and Rejected Releases are marked as such in the Versions and history flyout context sub-menu. Discarded Release candidates are not marked as discarded; the Release candidate is removed and a history is kept, marked as a version. Note that only the creator of the Release candidate has the option of discarding it.

When a workspace is created by branching a Version, meta data related to that particular release is not propagated to the branch, for example Revision, Date approved, and Date released will be blank in the new workspace.

---

**Cloning a Release Package**

This functionality is currently available only on browser.

Cloning a release package that has been released or rejected is an easy way to create a new release package once changes have been made to a part or parts within a workspace.
Clone allows you to use the existing release package as a starting template for the new package.

Keep in mind that in order to clone release packages, you need the appropriate permissions on the document. You also must be viewing the workspace on the branch from which the previous release package was created. (If you are not in the workspace of that branch, you will not see the option to Clone the release package.)

**Steps**

To clone a released or rejected release package:

1. Open the workspace on the branch from which the previous release package was created.

2. Open the Versions and history graph and right-click the release version.

3. Hover over Releases> to display the releases available.
4. Select the release to clone:

Above, the blue highlighting indicates that B1 is the workspace currently being viewed; the release candidate (rel-candidate-1) is being accessed from the version rel-candidate-1, and the release package has been previously rejected.

The Review Release dialog opens.
5. Click Clone to begin recreating a release package from the current workspace. If the Clone button is absent, verify which workspace you are currently viewing and open the one on the branch that the previous release was created from.

6. When the Create Release candidate dialog opens, the parts previously released are now replaced with the parts in the workspace, most likely updated per the requirements of the release manager.

   You can, of course, make any changes necessary: add more parts or remove parts.

7. When the Review Release dialog opens you can make any edits you want:
   a. Change the parts listed by deleting and adding parts, if desired
   b. Alter the Revision or Part number
   c. Alter the release name, include release notes

8. Supply the necessary approvers and observers.

9. Note that all comments from the previous release are still present (click the comment history link to access the comments for the release package):
10. If the release candidate required fields are complete, you can act on the Release candidate depending on your role (Apply, Release, Reject, or Discard) from the comment dialog (shown above). If any of the fields are incomplete, you must return
to the Review Release dialog.

Click the arrow icon at the bottom of the dialog, or the Comments link at the top to return to the Review Release dialog and complete the fields.

Then you can act on the release (Apply, Release, Reject, or Discard) depending on your role.

**Tips**

- Clone is available while viewing a workspace in the same document as the release, and will clone the release into that workspace.

- Comments from the original releases are shown on the Comments page of the Release dialog, with links to view the original releases. You get comments all the way down the chain, so if you clone release B from release A, and release C from release B, C’s comments will show comments from both release A and release B.

- When cloning, any top-level item that can’t be found in the clone’s workspace will not be included in the cloned release.

---

**Viewing Revision History and Obsoleting Parts**

This functionality is available on Onshape's browser, iOS, and Android platforms.

There are times in a Release process when you need to view the revision history of an object and there may also be times when you need to make an object obsolete.

The Onshape obsoletion workflow is illustrated in the company account settings, under Release management:
Access to viewing Revision History and also Obsoleting a part are both available through the context menu on an object (part in a Parts list, Assembly or Drawing tab):

1. Right-click the part or tab. (On iOS or Android platform, tap the Assembly tab for an Assembly or the overflow menu next to a part in the Parts list of a Part Studio for a released part.)

2. Select Revision history. The Revision history dialog opens:
3. This dialog lists all the revisions of the selected part or object. The details provided include:

- Revision label (B and A as shown above, top to bottom) - Each time the object is released, a new revision label is applied or specified (alphabetically, not reusing any skipped labels in the sequence) and the previous revision is automatically obsoleted
- The user who created the Release
- The Release name provided
- The date the revision was released
- The version of the document in which the release was released
- An icon to indicate an obsolete revision, with the opportunity to click View obsolescence
- A button to export the entities involved in the release; this opens the Export dialog box. See "Exporting Files" on page 2013 for more information on exporting.

4. You are able to click View release to open the Review release dialog, described in "Reviewing, Approving, Rejecting Candidates" on page 2206.
5. Use **Obsolete revision** to open the Obsolete dialog to mark a revision as obsolete:

![Screenshot of the Obsolete dialog](image)

If the release was a mistake and you wish to reuse the revision label for the next release of the item, check the box **Mark revision as re-releasable**. Checking this box allows you to obsolete this revision of this item and re-use the revision label for the next release of the same item.

6. Supply the required information. Click **Submit** if an approver's email is supplied, otherwise click **Obsolete** to directly mark the object revision as obsolete, removing it from production and preventing anyone from including that revision in any other release.

An email notification and a notification within Onshape is sent to the required approver and the user initiating the obsoletion.

To see that it's been obsoleted, reopen the Revision history for the object (right-click on the object). The icon, ![icon](image), indicates the item has been obsoleted. If the obsoletion was automatic (caused by a re-release of an existing revision), this icon ![icon](image) also appears.
Obsoleted revisions are never available for insertion into an Assembly or Drawing, so they are never shown in those dialogs.

Comments can be entered at any time in the release cycle, even after a release is approved, rejected, or obsoleted. This can be done on browser and iOS platforms.

Releasing a Configuration

This functionality is currently available only on browser.

Onshape allows you to release configured parts and treats each configuration as a unique revisionable object.

To release a configuration directly, set the desired configuration parameters in the Part Studio, then right-click on the configured part and select Release. The currently active configured part is displayed by default in the Create Release candidate dialog. You are able to choose to release only that configuration or add additional configured
parts by clicking the Plus sign at the top of the dialog to access the Add Items to Release dialog:

This dialog works the same as the Insert parts and assemblies dialog in an Assembly. In the dialog:

1. Select and modify the desired configuration parameters to create a configured part.
2. Click Generate to build the configured part.
3. Click on the newly-generated part to add it to the Release candidate. You can generate and include multiple configurations of each part in a single Release.
4. Click Add to finish the workflow and return to the Create Release candidate dialog.

To remove undesired objects from the Release candidate, simply click the red X next to the object.

When you release a configuration (or multiple configurations) of a part, only those specific, released configurations will be available for reuse from the Released version. Other configurations of that part will need to be released separately. Similarly, when you insert a released version of a configured part, you are inserting a specific, released object and will not be permitted to change configuration parameters.
Assemblies containing configured parts

Releasing an Assembly that contains configured parts releases only the specific configurations of each part referenced by the Assembly. No additional configurations are released unless you add them as described above.
Enterprise

This functionality is available on Onshape's browser, iOS, and Android platforms.

What is Onshape Enterprise?

Onshape Enterprise is Onshape's premium product offering designed for sophisticated product teams who need to work fast without losing control of data.

Onshape Enterprise is a specific plan type available from Onshape which enables a company to purchase a plan for multiple users with consolidated billing. When you purchase Onshape Enterprise, you receive a particular domain where the users in your company can access Onshape. Your Onshape domain is specific to you and inaccessible by other, unrelated Onshape users. Only the users specified and paid for by your company are allowed access to your enterprise.

When working within an Enterprise in Onshape, be aware that you are in a separate, managed environment that has a subtle difference from other types of Onshape subscriptions. The Enterprise environment is owned and managed by an Enterprise Administrator and can include many users, called Members of the Enterprise.

All users belonging to a specific Enterprise plan access Onshape through a single URL, specifically for their Enterprise.

Every Onshape Enterprise user has access to their particular Enterprise’s account as well as an individual account. If you sign in using cad.onshape.com in lieu of your Enterprise URL, you'll find yourself in your own account. You are able to use this account to create Onshape documents that are separate from the Enterprise and not seen or managed by the Enterprise. Regardless of what platform you use, when you sign into Onshape you access this individual account first unless you use your specific Enterprise URL. Use the User menu > Switch to <enterprise name> to switch your sign in to the Enterprise account.

Once signed in to your Enterprise account, you see primary areas of access:
• **Activity** - Browse all Onshape activity of your Enterprise users from simple changes made to documents, to viewing and responding to comments, and even viewing the differences ( diffs) between versions of documents. If you are not an administrator of your Enterprise account, you see only your own Activity stream. See "Monitoring User Activity" on page 2333.

• **Documents** - Access the Documents page on your mobile device. See "Documents Page" on page 140.

• **Analytics** - Onshape provides dashboards for a variety of categories of information. Each user sees and has access to only the reports that their Global permissions allow. The main page lists off the dashboards available to you depending on the Global permissions associated with your account. See "Accessing Analytics" on page 2351.

**Signing in on a mobile device**

The first time you sign into your Onshape Enterprise account on a mobile device, you land in your individual Onshape account. To access your Enterprise:

1. Tap the hamburger menu next to Home in the upper-left corner of the device screen.

2. In the menu that opens, select Switch to <enterprise name>.
   
   A notification appears asking if you mean to sign out of Onshape and switch to your domain.

3. Tap OK.
   
   Your Enterprise URL is displayed.

4. Tap Next.

5. Supply the email address and password associated with your Enterprise account.

6. Tap Sign In.

   **Onshape will remember this state the next time you open the app.**

Once you are signed in to your Onshape Enterprise account, the Activity page is displayed.
At the bottom of the page are icons/links to the additional Onshape Enterprise areas: Activity, Documents, Analytics, Notifications, and Settings.

Reaching Support
Onshape is here to help you. Contact our Support team the same way you normally would:

1. Click the 🎉 in the top right corner of the interface and select Contact Support. You are able to ask a question, suggest an enhancement, or report an issue.
2. If you choose to share your document with Support, the Onshape Support team will be able to jump in and help you in real time.

Getting Started as a Light User

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

What is a Light user?

Light user accounts are typically used by members of the internal supply chain, managers or executives, and can even be provided to external suppliers or clients. All Onshape users have access to the one single source of their enterprise data. There are times, however, when some users don’t need all the CAD functionality but rather just need to access, view, and export data. To better serve the needs of different users, Onshape provides two types of user accounts:
• Full users - Full users have the ability to access all of Onshape’s functionality to create, edit, and share data. Typically, Full user accounts are used by engineers, designers, or CAD specialists.

• Light users - Light users can view, comment on, and export enterprise data that has been shared with them.

When you sign in to Onshape as a Light user, you experience a trimmed-down version of the Onshape user interface called the View only toolbar. Many features and toolbars are removed, leaving only the commands necessary for finding, opening, viewing, commenting on, and exporting data. These commands are conveniently accessed through the View only toolbar.

In addition to the toolbar, you can also right-click on a sketch or part to access a menu of available commands like Show Dimensions, and other commands as well. Double-click on a feature in the Features list to see the settings and options used to create it.

The default entry-point for a Light user upon sign-in is the Activity stream on the dashboard:
A typical entry-point for a Light user, see "Navigating the interface" on page 2232 below for more information on this illustration

An activity stream lists all the actions taken on all documents to which you have permissions.

The only time you will not land on the Activity stream page, is when you are shared directly into a document and access that document via the share link (sent via a notification in the product, an email, or a mobile notification if this type of notification is turned on).

**Signing in**

When working within an Enterprise in Onshape, be aware that you are in a separate, managed environment that has a subtle difference from other types of Onshape.
subscriptions. The Enterprise environment is owned and managed by an Enterprise Administrator and can include many users, called Members of the Enterprise.

All users belonging to a specific Enterprise plan access Onshape through a single URL, specifically for their Enterprise.

Every Onshape Enterprise user has access to their particular Enterprise's account as well as an individual account. If you sign in using cad.onshape.com in lieu of your Enterprise URL, you'll find yourself in your own account. You are able to use this account to create Onshape documents that are separate from the Enterprise and not seen or managed by the Enterprise. Regardless of what platform you use, when you sign into Onshape you access this individual account first unless you use your specific Enterprise URL. Use the User menu > Switch to <enterprise name> to switch your sign in to the Enterprise account.

The structure of Onshape documents

An Onshape document is a version-controlled container for potentially many types of data. A single document may contain parts, assemblies, drawings, and even uploaded files like PDFs, images, or videos. Every organization uses documents differently, but understanding how documents are structured and how to find different types of information will help you work with them effectively.

Tabs

Each Onshape document may contain different types of information, broken down into different kinds of tabs which can be seen along the bottom of the window:

![Tabs](image)

Examples of the different types of tab data

Hover over a tab for a thumbnail of its contents. Click on a tab to access its contents. You are also able to click the Tab manager button in the bottom left to see a vertical list of tabs, as well as search and filter for specific types or instances of data:

1. Select a tab type filter to sort by tab type.
2. Enter search criteria if you wish.
3. View the results of your search.
The Tab manager with the Part Studio filter selected

Tabs fall into 9 categories:

1. **Part Studio** -

   Parts in Onshape are created in Part Studio tabs. A Part Studio may contain one part or multiple parts, depending on how the designer constructed them.

2. **Drawing** -

   Drawing tabs will be familiar if you have used drawing files in another system. Onshape drawings can have multiple sheets and reference multiple parts or assemblies.
3. **Feature Studio** -

A Feature Studio is a tab containing FeatureScript, a programming language used to define your own custom features in Onshape. (See [Feature Studios](#) for more information.)

4. **Application** -

Third-party applications from the Onshape App store may also appear as tabs in an Onshape document. These may be less relevant to your use of Onshape, but are nonetheless important to recognize if you encounter them.

5. **Image** -

Images uploaded into Onshape are stored in a tab and are displayed both in the tab and the thumbnail of the tab.

6. **Zip** -

A zip file is an import of compressed (or zipped) files for the purpose of storing information in Onshape along with the project data. See [Importing Files](#) for more information.

7. **Material library** -

Users are able to upload their own company material libraries for use in Onshape. See [Customizing Materials](#) for more information.

8. **PDFs** -

PDFs generate a preview and may be viewed directly in the Onshape document.
9. **Assemblies** - (including BOMs)

Assemblies are created in Assembly tabs, very similar to the assembly files you may be familiar with from other systems. One key difference is that in Onshape, the assembly simultaneously contains the Bill of Materials (BOM).

10. **X-T**

X-T is translated or un-translated CAD data from another system. See Importing files for more information.

For more information on using the tab manager, see "Document Tabs" on page 122.

**Navigating the interface**

Typical navigation for a Light user might include the following (note that the numbers coincide with the numbers in the image above):

1. **Go to Documents icon** - Your company icon is located in the top far left corner of the Onshape homepage and displays the Documents page. Click this icon from anywhere in Onshape.

2. **Activity tab** - This tab is located in the top left corner of the Light user homepage and allows you to navigate to your Activity stream (a listing of the actions you and your colleagues have taken within Onshape).

3. **Documents tab** - The Documents tab is located in the top left corner of the Light user homepage and allows you to navigate to your Documents page (identical to clicking your company logo).
4. **Action items tab** - The Action items tab is located along the top left of the Light user homepage and allows you to navigate directly to your Action items (tasks and release management assignments) page.

5. **Search bar** - The Search bar on the Documents page allows you to search through all documents. When a filter is selected (on the far left: Recently opened, Created by me, Shared with me), the Search is conducted on documents within that filter.

6. **App Store button** - Onshape’s App Store button, located at the top of your Light user homepage to the right of the Search bar, allows you to navigate to Onshape's App Store. Here you can purchase third-party apps to work with your Onshape data.

7. **Learning Center button** - Onshape’s Learning Center button is located at the top of the Light user homepage to the right of the App Store button, and allows you to navigate to Onshape’s Learning Center. Here you can go through self-paced courses and recorded webinars for help learning how to use Onshape.

8. **Help icon** - Onshape’s Help menu icon is located at the top right of the page, to the right of the Learning Center button. Click on the icon to open the Help menu drop down, where you can choose from a number of Help options. Select the top option, Help, to directly access Onshape's Help center.

This menu also contains a way to contact support: **Contact Support** opens a dialog in which you can enter a question, optionally add a screenshot of your problem, and select whether or not to share your document directly with an Onshape Support representative (this permission can be revoked at any time).
9. **Account user icon** - The right-most icon at the top of your Onshape Light user page is the Account icon, usually followed by the name of the user. Click on this icon, or your name, to open a drop down menu with options to navigate to your account, your company settings, as well as an option to sign out:

```
My account
Company settings
View support tickets
Sign out
```

For more information on getting started with some common setup preferences and accessing commands, see "Setting Your Preferences" on page 40.

**Using the Activity page**

Click **Activity** at the top of the window.

The Activity page is your Onshape home page and is your default starting point in Onshape. On this page you see document-related activity in a stream. Use the 'ellipsis' menu to access commands related to the activity entry.

For more information, see "Monitoring User Activity" on page 2333.
Using the Documents page

Click **Documents** at the top of the window to open the Documents page, a listing of all the documents to which you have permission.

The Documents page is where you'll likely start most of your work in Onshape. Click your company logo in the top left corner of the window to return to the Documents page from anywhere in Onshape.

The Documents page lists all of the documents you have access to, sometimes listed singly, sometimes organized into projects and folders (shown below, outlined in red). This page also contains filters for narrowing the list of documents (outlined in orange),
and a search box (outlined in yellow). Click on a project, folder, or document name to open it.

In the above example, the icons outlined in blue represent Labels, Document details, and List and Grid view options.

For more information about using the Documents page, see "Documents Page" on page 140.

**Searching for information**

You are able to access Onshape's advanced search bar in your Light user Documents page, shown above outlined in red. To navigate there, click the Go to documents icon (your company logo), both located in the top left corner of the page. Onshape's advanced search tools may be used to find specific parts, assemblies and drawings by part number, release status, version and revision.

**Click the down arrow in the search bar** to open the advanced search dialog. From this dialog you can specify the different parameters for your search:
As an example, say you receive a work order specifying part number 00203 rev B and you need both the drawing and the 3D model of the part to complete the order. To find both, open the advanced search dialog and fill it out as shown:
Click **Search** to display the results (shown in the second example above), and click on a result to bring you directly to the drawing and the 3D model. From there you are able to perform any of the actions described previously to comment on, measure, or export objects as needed.

For more information using the Onshape search functionality, see "This functionality is available on Onshape's browser, iOS, and Android platforms." on page 181.

**Using the View only toolbar**

The View only toolbar is the default presentation for all users who do not have write permissions on a document.

Upon entering a document, a Light user or a user with View only permissions will see this layout (*note the View only toolbar outlined in blue*):

![View only toolbar](image)

A user in Enterprise and any other account will have this toolbar as well, when shared with the View only permissions. Only a company owner has the ability to change their company's settings to hide the View only toolbar in all documents. Users with editing permissions on a document will not see the View only toolbar.

For more information on how to operate the View only toolbar, see [Using the View Only Toolbar](#).

The View only toolbar is located at the bottom of your Onshape Part Studio or Assembly (shown above outlined in blue) and is able to have up to 18 tools or features, some of which depend on the tab type or the user's permissions:
1. **Home** - Click the Home icon to automatically restore the initial view of your document.

2. **Rotate** - Click the Rotate icon, then click and drag your cursor in the direction you want your document to rotate.

3. **Pan** - Click the Pan icon, then click and drag your cursor in the direction you want to pan your document.

4. **Zoom to fit** - Click the Zoom to fit icon to automatically zoom your image to fit the middle of the screen. Click the dropdown menu arrow to the right of the icon to see the following features:

   - **Zoom to window** - Click Zoom to window, then drag your cursor to create your zoom bounding box around the part of the entity you wish to zoom in
on; your image zooms accordingly

- **Zoom** - Click Zoom, then click and drag your cursor up and down or left and right to zoom in and out.

Note that while using the Onshape application on a desktop, you are able to scroll up or down with your mouse at any time to zoom in or out.

On Windows machines with a standard 3-button mouse, Onshape provides the following scheme for manipulating the 3D model in Part Studios and Assemblies:

- **3D Rotate**: Right-mouse-button-click+drag
- **Zoom in and out**: Scroll up and scroll down, respectively
- **2D pan**: CTRL-right-mouse-button+drag (middle button click+drag)

For more information on customizing your view manipulation in Onshape, see "This functionality is available on Onshape's browser, iOS, and Android platforms." on page 192.

5. **Section view** - Click the Section view icon to open the Section view manipulator:

Select the plane, planar face, or mate connector that you want to view, then click the checkmark ✓ in the top right corner of the Section view manipulator to finalize your decision.

6. **Properties** - When you are in a Part Studio, click the Properties icon to open a Properties dialog where you will have the ability to edit your Part Studio's name, description, part number, state, and more:
To see the properties of a specific part on an entity, click the specific part and then click the Properties icon.

In an Assembly, click the Properties icon to open a Properties dialog.

Light users and full users who lack write permissions do not have the ability to edit properties.

**Toggle Appearance panel** - This tool appears only when a Part Studio tab is selected. Click to open the Appearance panel from the right side of the UI:
You can see the color associated with a particular part or face. For more information about assigning colors, see "Customizing Parts, Faces, and Features: Appearance" on page 302.

**Toggle Custom tables panel** This tool appears only when a Part Studio tab is selected. Click to open the Custom tables panel from the right side of the UI. If a custom table has been added to the document, it opens:
7. **Toggle Configuration panel** - Click the Configuration table icon to open the Configured part properties panel (*note that this icon appears only when your assembly or Part Studio contains configured properties*):

![Configuration panel](image)
To learn more about the Configuration panel, see Configurations.

8. **Toggle BOM panel** - **BOM** Click the BOM icon to open the BOM panel, or close the BOM panel:

![BOM panel](image)

9. **Open Exploded views panel** - Click the Exploded views icon to open the panel in order to select an Exploded view to display:

![Exploded views panel](image)

   *This tool only appears when an Assembly is selected.*

10. **Toggle comments panel** -

![Comments panel](image)
Click the Comments icon to toggle the comments panel (*note that this icon appears only when the user has permission to comment*):

![Comments Panel](image)

To learn more about the Comments panel, see [Comments on Workspaces](#).

11. **Follow a user…** - Click the Follow a user icon to open a menu with a list of users that are currently in the document (*note that this icon only appears if there are multiple users in a document*):

![Follow Users](image)

Click a user you wish to follow, and your Onshape window will adjust to show you their document view in real time, as shown below:
To stop following a user, re-click the Follow a user icon in the toolbar. To learn more about following a user, see Follow Mode.

12. **Export tab** - Click the Export icon to open the Export dialog:

Here, you have the ability to edit your document's name and export options. Click **OK** to finalize your decisions.
- Export selected - Click the Export selected icon to export all selected geometry *(note that this icon appears only when exportable geometry is already selected)*. An Export dialog like the one above appears; edit your file name, format, version, and export options, and click OK to finalize your decisions.

- Select and export... - To export a specific assembly or part, click the dropdown menu arrow to the right of the Export icon and click Select and export... *(note that this icon only appears if there no exportable geometry is selected)*. This opens an Export manipulator; click the part or assembly you want to export, then click the checkmark in the top right corner. This opens the same Export dialog as shown above. Choose your export settings and click OK to finalize your decisions.

13. Print - Click the Print icon to open the Print setup dialog:

![Print setup dialog](image)

Here, you have the ability to edit your print settings. When you are ready to print, click the checkmark in the top right corner. This opens a print-preview of your document, with options to choose more printer-specific settings.

14. Measure - Hold your cursor over the Measure icon to read how the Measure tool functions:
The above message explains how the Measure tool is automatically shown in the lower right corner of the Onshape window whenever an entity or part is selected. Click on the Measure icon to see an animated example of the tool's functionality.

15. **Mass Properties** - Click the Mass properties icon to open the Mass properties panel:

In a Part Studio, if there are no parts selected, the Mass properties panel displays the Part Studio properties. If there are parts selected, it displays only the properties for those selected parts.

**Links and Sharing**

Everything in Onshape has a unique URL. Many organizations post links to specific versions of Onshape objects (Part Studios, Assemblies, Drawings, etc) in work orders or ERP software, allowing you to get to the exact version you need. These links can be easily copied from and to the address bar of your browser.

Using the Share button is not available to Light users at this time as Light users are not permitted to share documents.

**Selection**

Selection in an Onshape document is additive. That is, when you left click to select a piece of geometry and then click again on another piece of geometry, Onshape adds the second selection (and subsequent selections) rather than switching to it. This takes some getting used to, but you are able to easily clear your selection by tapping the spacebar on your keyboard (or clicking in empty space). The first example below
is of the view of your document in a Part Studio with just one part selected. The second example shows how your document would look with three parts selected simultaneously, the parts all highlight in yellow when selected.
Measurement and Mass Properties

You can use the Measurement and Mass properties tools in the View only toolbar, but also, the Onshape measure tool is automatic in Part Studios and Assemblies when any entity is selected. Measurement information appears automatically in the bottom right corner of the interface when you make a selection or selections. For more information on measuring, see "Measure Tool" on page 344.

The measurement tool in a Part Studio displayed the area of the selected part.

The same is true for mass properties. Select a part or assembly and the mass properties icon appears in the bottom right corner of the interface:
Click the icon to display the properties for the selection.

For more information on mass properties, see "Mass Properties Tool" on page 349

Commenting

You may be given permission to Comment on documents, which allows you to converse with other members of your organization directly in Onshape. To add a comment, open a document and click the Comment button in the View only toolbar to open the comment flyout, shown below:
The Comment button will not appear if you do not have permission to comment.

If you have permission to comment on the document, you can add comments to model geometry and features by right-clicking on it and selecting Add comment from the context menu, shown in the first example below. Selecting **Add comment** opens a Comments dialog with the part selected tagged to the comment, shown in the second example below. You can also mention other users in a comment simply by typing @ before their username.
For more information on commenting, see "Commenting in Workspaces and Versions" on page 2071

**Exporting**

You have the ability to export parts, Assemblies and Drawings to a number of different file formats, which allows you to, for example, easily create the STEP files that your supplier needs to make your parts. If you have permission to export data, right-click on a part in the Parts list or graphics area of a Part Studio, an Assembly tab or a Drawing tab and select Export from the context menu (shown in first example below). An Export dialog will appear (second image below):
For more information on exporting data and supported file formats, see "Exporting Files" on page 2013

**Accessing command context menus**

Right-click, or context, menus appear everywhere in Onshape and contain many of the commands needed to complete your tasks. Right-click to activate context menus for entities in the graphics area (such as parts, surfaces, and sketches), entities in Feature lists (such as Extrude and Revolve features), Parts lists in both Part Studios and Assemblies (individual parts, surfaces, curves, and assemblies), and entities in Drawings. Key command like Export are found in context menus, so be sure to right-click throughout the interface to access the commands needed to accomplish your goals.

*Example of a context menu generated by right-clicking on a document on the Documents page*
Users with release management responsibilities

When a user has release management responsibilities, (Enterprise or Professional users only), they may be responsible for approving a release candidate. There is no Light user tool on the toolbar for this purpose, however, the user must access the release candidate through the notification they received, which may be:

- Email notification with a link
- Internal notification in the document
- Mobile notification with a link

These notifications will contain a View release link that when clicked takes the user directly to the release candidate dialog.

Getting Started as an Enterprise User

This functionality is currently available only on browser.

This functionality is also available on iOS and Android in a limited form.

If you are the administrator, or have administrative permissions, see "Getting Started as an Enterprise Administrator" on page 2259 for more specific information on setting up permissions and security for your Enterprise account.

Signing in

When working within an Enterprise in Onshape, be aware that you are in a separate, managed environment that has a subtle difference from other types of Onshape subscriptions. The Enterprise environment is owned and managed by an Enterprise Administrator and can include many users, called Members of the Enterprise.

All users belonging to a specific Enterprise plan access Onshape through a single URL, specifically for their Enterprise.
Every Onshape Enterprise user has access to their particular Enterprise’s account as well as an individual account. If you sign in using cad.onshape.com in lieu of your Enterprise URL, you’ll find yourself in your own account. You are able to use this account to create Onshape documents that are separate from the Enterprise and not seen or managed by the Enterprise. Regardless of what platform you use, when you sign into Onshape you access this individual account first unless you use your specific Enterprise URL. Use the User menu > Switch to <enterprise name> to switch your sign in to the Enterprise account.

**Transferring privately-owned documents**

If your user account was upgraded from a company or free account to an Enterprise account, any privately-owned documents you had are still accessible through www.cad.onshape.com (as opposed to signing in using the Enterprise domain URL).

To transfer any privately-owned documents to your Enterprise account:

1. Navigate to cad.onshape.com.
2. Sign in using your Enterprise credentials.
3. Locate the data you want to transfer.
   - If there are multiple documents, place them in a folder.
4. Right-click on the document or folder and select Transfer, then enter the email address of the administrator of the enterprise.
   - Once the ownership of the documents are transferred to the administrator of the Enterprise, that administrator can transfer ownership to the Enterprise account.

**Working in your Enterprise domain**

As a user of an Enterprise account, your permissions and security have been pre-defined by an administrator. Depending on the permissions you have been granted, the functionality you have access to may differ from other users.

One thing you have in common with all Enterprise users is that all documents you create are owned by the Enterprise and the Enterprise administrator.

When working in Onshape, your access to certain Projects, and your permissions to the documents within those Projects are controlled by the settings the administrator
has put into place. If you have questions about your access rights, it is best to contact your administrator.

**Activity, Documents, Analytics, and Action Items**

Once signed in to your Enterprise account, you see these primary areas of access:

- **Activity** - Tap or click to browse all Onshape activity of your Enterprise users from simple changes made to documents, viewing and responding to comments, and even viewing the differences ( diffs) between versions of documents. If you are not an administrator of your Enterprise account, you see only your own Activity stream. For more information on the functionality of the Activity stream, see the "Monitoring User Activity" on page 2333 topic.

- **Documents** - Tap or click to access the Documents page. For more information, see the Documents page topic.

- **Action items** - Tap or click to navigate directly to your Action items (tasks and release management assignments) page. For more information, see "Monitoring Releases and Tasks: Action Items" on page 2349.

- **Analytics** - Onshape provides dashboards for a variety of categories of information. Each user sees and has access to only the reports that their Global permissions allow. The main page lists off the dashboards available to you depending on the Global permissions associated with your account. For more information, see the "Accessing Analytics" on page 2351 topic.

All users have access to their own Activity stream (a list of all actions taken within Onshape) and some reports. You can access these areas from the links to the right of your company logo in the Onshape title bar.
As seen in iOS platform of Onshape

As seen in Android platform of Onshape: starting from left the icons represent Activity, Documents, Analytics, Notifications, Account

You have the ability to choose which of these primary areas your account will open to upon signing in to Onshape. To see more information on how to choose your Startup page, see Enterprise Subscription.

Personal account administration

To set your own preferences and view settings, see "Managing Your Onshape Enterprise Subscription" on page 2551. Among the preferences you can set are:

- **Profile** - Specify a nickname for display purposes and upload a photo.
- **Emails** - Add alternate emails to your Onshape account.
- **Preferences** - Specify preferences for language, units, manipulating views, and more.
- **Security** - Enable two-factor authentication for your account or change your password.
- **Devices** - View a list of all devices associated with your account.
- **Applications** - Purchase, view and manage the apps that you've authorized to work with your Onshape account.
- **Early visibility** - Request participation in an Onshape early visibility program to test pre-release functionality.
- **Subscription** - View (or manage) the subscriptions of which you are a member (or admin).
- **Payment options** - View and update payment information.
Getting Started as an Enterprise Administrator

This functionality is available on Onshape's browser, iOS, and Android platforms.

As an Enterprise Administrator, you are responsible for defining permissions for users. You have, by default, full access and full permission to your company's resources. Optionally, you are able to grant similar permissions to other users using Global permissions. For more information, see Understanding Global Permissions.

Getting Started as an Enterprise Administrator: Desktop

Step 1: Add users

If you are upgrading from a Professional subscription, you are able to skip this step because all existing users are automatically part of your Enterprise account.

If this Enterprise account is your first Onshape account, or if you need to add more users to an existing account, see "Adding and Administering Users" on page 2264 for instructions on adding users.

Step 2: Create a project

Use Projects to group users for the purpose of assigning specific permissions for a specific set of users. Once the project is created, you are able to start creating documents within the project, or move documents to the project.

1. After signing in to your account, proceed to the Documents page: Click the logo in the upper left corner of the window, or the Documents link to the right of the logo:
2. Select **Create > Project.**

   a. Enter a name for the Project; using a highly descriptive name is ideal.

   b. Enter a description of the Project.

3. Select the **Default** Permission scheme from the dropdown list.

4. Select **Project Administrators** as the Project role to grant all permissions to a select group of users to act as administrators of the account.

5. Select from the email list the users you want to act as administrators of the account.

6. Click **Add**.
Managing your enterprise settings

As Administrator, you have the permission set to manage your enterprise settings, navigated to through the User menu. Below is a list, a short explanation, and a link to more information for each setting.

- **Users** - Invite or add new users to your Enterprise, view the list of current users, search for a particular user, delete selected users, and see whether a user is a Guest user.

- **Teams** - Create teams of users to group users together for the purpose of making sharing more efficient; once the team is created, you can select the team name instead of entering many users' individual email addresses during a Share operation. All team members must be users in the Enterprise. For more information see "Enterprise Teams" on page 2385.

- **Global permissions** - Control user access to Enterprise-level operations and information. For more information, see "Understanding Global Permissions" on page 2271.

- **Authentication** - Change your Onshape system password, and also enable (or disable) two-factor authentication.

- **Project roles and Permission schemes** - Project roles and Permission schemes work in tandem to apply access permission to documents grouped together within a Project. For more information, see "Understanding and Administering Project Roles and Permission Schemes" on page 2276.

- **Custom properties** - Access the metadata definitions for Onshape objects, view these metadata definitions, and create new custom properties for use in company-owned documents, as well as provide Display names for existing Onshape properties. For more information, see Custom Properties.

- **Categories** - Create categories in Onshape to extend the properties of the standard Onshape object types in the system to include more targeted and relevant metadata to be applied based on design, engineering, and manufacturing processes. For more information, see the Categories heading in "Managing Your Onshape Enterprise Subscription" on page 2551.
• **Release management** - A set of automated workflows in Onshape used to manage releasing revisions of parts, Assemblies, Drawings and imported files (translated or not) in a document. For more information, see [Release Management](#).

• **Preferences** - Specify whether users are required to use only approved drawing templates when creating drawings, and allow (or disallow) users to add company-owned shared material libraries within their accounts. For more details on drawing templates, see Custom Drawing Templates and for more information on shared material libraries, see [Customizing Parts: Materials](#).

• **Integrations** - Grant or remove access to Microsoft OneDrive, Dropbox, and Google Drive accounts to Onshape.

• **Details** - View and update the details of the company information and change the logo image.

**Users and permissions matrix**

Onshape allows you to create the following types of users:

<table>
<thead>
<tr>
<th></th>
<th>Member Status</th>
<th>Guest Status</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Full user</strong></td>
<td>Up to full editing abilities on projects, documents and folders. Company admin access.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Engineers &amp; Designers</td>
<td>Curated access to company data with up to full editing abilities. No admin access.</td>
</tr>
<tr>
<td></td>
<td>• CAD Admins</td>
<td>• Contractors</td>
</tr>
<tr>
<td>Member Status</td>
<td>Guest Status</td>
<td></td>
</tr>
<tr>
<td>---------------</td>
<td>--------------</td>
<td></td>
</tr>
<tr>
<td><strong>Light user</strong></td>
<td>Curated access to company data with maximum view, export and comment abilities. No admin access.</td>
<td></td>
</tr>
</tbody>
</table>

- Sales & Marketing
- Supply chain
- Executive stakeholders

**Administering your personal account**

To set your own preferences and view settings:

- **User Profile** - Specify a nickname for display purposes and upload a photo.
- **Email Addresses** - Add alternate emails to your Onshape account.
- **User Preferences** - Specify preferences for language, units, manipulating views, and more.
- **Two-factor Authentication** - Enable two-factor authentication for your account or change your password.
- **Devices** - View a list of all devices associated with your account.
- **Applications** - Purchase, view and manage the apps that you’ve authorized to work with your Onshape account.
- **Early Visibility Program** - Request participation in an Onshape early visibility program to test pre-release functionality.
- **Payment Options** - View and update payment information.

**Getting Started as an Enterprise Administrator: iOS and Android**

As an Enterprise Administrator, you are responsible for defining permissions for users. You have, by default, full access and full permission to your company’s resources.
Any other user added to the Administrative global permission also has full access and full permission to the company’s resources. Global permissions define how much access a user has to the company resources. For more information, see Understanding Global Permissions.

When you upgrade from a Professional subscription, all users specified as part of your subscription are transferred to your Enterprise plan, so there’s no immediate need to add users. However, if you are creating a new account with Onshape, you will have to add users.

Add users by accessing your Onshape account on a browser. This functionality is not currently available on our mobile platform.

For information on viewing Administrative settings on a browser, see the following topics:

- "Understanding and Administering Project Roles and Permission Schemes" on page 2276
- "Understanding Global Permissions" on page 2271

For information on setting up two-factor authentication or configuring integration with identity providers, see the following topics:

- Security
- Configuring Integration with Identity Providers

Adding and Administering Users

This functionality is currently available only on Onshape's browser platform.

As administrator of an enterprise you are responsible for adding and removing users from your enterprise account. If you have upgraded from a Professional account to an Enterprise account, the users defined in the Professional account were all moved to your Enterprise account.
If your company uses an SSO application, like Okta, even better. Onshape works with other single sign on providers, too. See Configuring Integration with Identity Providers for more information.

Adding users to the account

There are two types of users you can add to an Enterprise account: Light and Full. **Full users**, with permission, can access all functionality including editing, sharing, and releasing data. **Light users**, by contrast, with permission, can only open and view data, leave comments, and export data.

**EDU enterprise users are always Full users, there is no Light user capability within an EDU enterprise.**

In addition to being either a full or light user, a user may also have a status of either Member or Guest. **Members** are typically members of your account, included in the billing for that account. **Guests** are limited-time users, usually meant to have temporary access to your documents and resources. Guest users are added to your enterprise (with access only to data that is shared with them) and billed to your account. Guest users are not members of the "All enterprise users" team, but what they have permission to do in Onshape is determined by the enterprise administrator and users with Share permissions on documents. A user with Share permission can share documents with Guest users and provide certain permissions to that Guest user for that document.

To add users to your enterprise account:

1. Click your name in the upper right corner of the window to open the menu.
2. Select **Enterprise settings**.
3. Click **Users** in the left tab pane.
4. In the top right corner, click **Invite users**.

![Invite users dialog](image)

For EDU Enterprise users, this dialog is different, and allows only Full users to be invited (and added) to the EDU Enterprise.

![Invite users dialog](image)

5. In the dialog, enter user email addresses, select a User type and a status if necessary.

   Use semi-colons (;) to separate multiple email addresses.

6. Click **Send invitation**.

You can also add users using a CSV file:
1. Follow steps 1-3 from above.

2. In the top right corner of the page, click Import from CSV file button.

3. Click Download CSV template.

4. Open template.

5. Edit the template to add users and their specifications. Do not edit the header row.
   a. Email - Email address of the user as it exists in Onshape
   b. Role - Member or admin
   c. Team - The name of a pre-existing Onshape team (See Teams topic for more information on Teams.)
   d. Type - Type of user can either be Full or Light user; for EDU Enterprise plans, this field must be "Full"
   e. Guest - Enter yes or no accordingly

6. Save the CSV file.

7. In the Import dialog, select Choose File to select your CSV import

8. Click Import, a Results notification will appear. Click OK to accept. The users will automatically be added to the enterprise.

The users are added to your account and become available for inclusion in Permission scheme role maps.

**Adding guest users to your enterprise**

You can give users access to your enterprise documents without giving them access to all of your data. Guest users have access only to the documents that are explicitly shared with them, cannot be included in any global permission and never have access to any enterprise account settings. You might want to let certain vendors,
suppliers, contractor or even clients have access to design data. You may let them view design data, but have scoped access, curated to specific projects, folders or documents.

If a guest doesn't have an Onshape account, they receive an email with a link to create an account. Upon signing in, Onshape opens to a private account. The guest must then access the User menu and execute the Switch to <enterprise name> command to access the enterprise account.

If a user already has an Onshape account, an email is sent notifying them that they have been added to an enterprise account in addition to their other account. They can access the enterprise account through the Switch to <enterprise name> command in the User menu.

Guest users are not added to the All enterprise users team automatically nor can they be added manually.

Adding a guest user through Enterprise settings

1. From the Enterprise settings, select Users.
2. Click Invite Users to open the Add users dialog.
3. Enter the email address of the user you wish to add as a guest (or more than one email address, comma-separated).
4. Select the checkbox in front of "This user is a guest and has access only to data that is shared with them."
5. Click Add.

Adding a guest user through the Share dialog

1. From within a document, click the Share button.
2. Select the Invite guests tab.
3. Enter the email or emails of the users you wish to add as guests (use comma separators).
4. Select the permissions to allow for the active document.
5. Click Share.
6. Click Close to close the dialog.

The newly invited guest is displayed as a Guest user on the Users tab in Enterprise settings. Once a user is invited as a guest, you can type their email address directly in the Individuals tab of the Share dialog.

Adding a guest user through the Project dialog

1. Access the context menu on a Project and select Edit project.
2. At the bottom of the dialog are the Share tabs; select the Invite guests tab.
3. Enter the email or emails of the users you wish to add as guests (use comma separators).
4. Select the permissions to allow for the active document.
5. Click Share.
6. Click Close to close the dialog.

The newly invited guest is displayed as a Guest user on the Users tab in Enterprise settings. Once a user is invited as a guest, you can type their email address directly in the Individuals tab of the Share dialog.

Changing a user’s type or status

To change a user status to or from Guest, or change a Full or Light user to the other type:

1. From the Enterprise settings, select Users.
2. Right-click the user name in the Users list and select Change user type.
3. In the dialog, make the necessary changes and click Update.

No notifications are sent regarding this action. When a member is switched to Guest status, any permissions on documents not shared explicitly with the user are removed. If the member was assigned to a Permission scheme this assignment and the associated permissions are removed from the user when switched to Guest status.

Guest users have a Guest tag next to their name whenever it appears in the Analytics console.
Permissions for guest users

Guest users have access to only the specific documents shared with them. You can manage their permissions (and thus the actions they can take on documents) by adding them to Project roles. Permissions on Projects are according to the Project Role the guest is assigned to.

You are able to add guest users to teams, but if the team is added to any Global permissions, the guests users are filtered out of the permissions and those permission to do not apply to them.

All actions taken regarding users and guest users are reflected in the Activity area.

Guest users cannot access Enterprise settings, that option is not available in the User menu.

Removing users from the account

There may be times when you want to remove a user from your Onshape Enterprise domain and disable their access to all data within that domain. The steps below accomplish both tasks:

1. Sign in to your enterprise domain as an enterprise Admin.
2. Click your name in the upper right corner of the window to open the User menu.
4. Click the Users tab in the left pane.
5. Right-click on the user name to remove and select Remove user.
6. Review the dialog describing that this user will no longer be able to sign in to the Enterprise domain, then click Remove.

The user is removed from the Enterprise and prevented from being able to sign in to the domain. Any access to enterprise data is also removed within the Enterprise domain. Within the Enterprise domain, Onshape maintains all of the removed users’ activity and analytics data, so it is still visible to your team. When looking for a removed user, you will see “(inactive)” next to that users name.
Understanding Global Permissions

This functionality is currently available only on browser.

Global permissions control user access to Enterprise-level operations and information. Only the enterprise administrator or another user with administrator rights can grant Global permissions to users. The user who creates the Enterprise account becomes the first administrator of the account and is assigned the Enterprise administrator global permission.

Global permissions dictate what access to (and control over) the enterprise account is available to specific users. These permissions are for users who should have some control over the account, and not intended for non-administrative users. Typically, what non-administrative users are able to do to a document is determined by the properties of Projects and Permission schemes.

To set up your enterprise account quickly using the Onshape-supplied defaults, review "Getting Started as an Enterprise Administrator" on page 2259.

Assigning global permissions

Global permissions are not attached to any particular resource (e.g. document), instead they generally apply to certain operations (like adding and removing users).

1. Select Enterprise settings from the User menu.

   With the Users tab selected on the left, notice that any users you had in your Professional subscription are still present. If you are not upgrading from a Professional subscription with users, see "Adding and Administering Users" on page 2264 for instructions on adding users.

2. Select Global permissions from the left menu pane.

3. Right-click on a Global permission and select Manage users.

4. In the dialog, select the email addresses of the users you wish to associate with those particular permissions and click Add. You can also select teams of users when you have teams set up.
The Global permissions include:

- **Enterprise administrator** - Grants the users assigned to this global permission full control and full permission over all of the enterprise’s resources. Users with this Global permission will see the following tabs when accessing Enterprise settings: Users, Teams, Global permissions, Authentication, Project roles, Permission schemes, Custom properties, Preferences, and Details.

- **Manage users and teams** - Grants the users assigned to this global permission the ability to add and remove users from the enterprise account as well as view and modify teams. Users with this Global permission will see the following tabs when accessing Enterprise settings: Users, Details, and Teams. Also allows transitioning enterprise members to guest users and guests users to enterprise members.

- **Invite guest users** - Grants the users assigned to this global permission the ability to add and remove guest users in the enterprise. This does not, however, allow transitioning enterprise members to guests or guests to enterprise members. For
more information about guest users, see "Adding and Administering Users" on page 2264.

- **Manage role based access control** - Grants the users assigned to this global permission the ability to view and modify Project roles and Permission schemes. Users with this Global permission will see the following tabs when accessing Enterprise settings: Users and Details.

- **Permanently delete** - Grants the users assigned to this global permission the ability to permanently delete any resources that have been placed in Trash and also to permanently delete a workspace. Users with this Global permission will see the following tabs when accessing Enterprise settings: Users and Details.

  Note that when deleting a workspace, a warning dialog displays and the action, if taken, cannot be undone. If there are versions on the branch, the branch will remain and you will be able to create a new workspace from any version on that branch.

- **Analytics administrator** - Grants the users assigned to this global permission the ability to fully control reports and analytics, including creating reports and sharing those reports with other users.

- **Create projects** - Grants the users assigned to this global permission the ability to create projects; the All enterprise users team is automatically assigned to this Global permission. Add and remove users (even guest users) and teams as desired.

- **Create documents and folders in "My Onshape"** - Allows users in the enterprise to create documents and folders within the "My Onshape" filter on the Documents page from within their enterprise account.

- **Create Releases** - Grants the users assigned to this global permission the ability to create Release candidates. The default users assigned to this global permission is the All enterprise users team. Enterprise admins can edit this permission.

- **Approve Releases** - Grants the users assigned to this global permission the ability to approve or reject release candidates and be listed in the Approver field of the release candidate form. You can add Guests users to this global permission.
**Allow access to the App store** - Grants the users assigned to this global permission the ability to sign in to the App store and subscribe to applications.

**Allow access to public documents** - Allow users in the enterprise to view, open, and copy public documents from within their enterprise account.

**Share for anonymous access** - Grants the users assigned to this global permission the ability to share documents by an anonymous link to people outside of the enterprise and outside of Onshape. These links provide read-only access to the document shared. When a user is associated with this global permission, they have an additional option in the Share dialog. For more information, see "Share Documents" on page 2045.

Users not associated with any global permissions see only the Users tab and the Details tab when accessing Enterprise settings.

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**Two-factor Authentication**

This functionality is currently available only on browser.

As the Admin of an Enterprise, you decide whether or not to require two-factor authentication use by company members. In the Company settings, you are able to configure your company's single sign-on and require two-factor authentication for Enterprise members.

For information on integrating Onshape with your single sign on identify provider, see "Configuring Integration with Identity Providers" on page 2301.
Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.

For information on how users set up and use two-factor authentication, see "Managing Your Onshape Professional Subscription" on page 2492.

Enforcing two-factor authentication enforcement

1. Select Authentication on the Company settings page.

2. Under Two-factor authentication (2FA), click **Enable two-factor authentication enforcement**.

3. Follow the instructions for setting up 2FA, in "Managing Your Onshape Professional Subscription" on page 2492
Understanding and Administering Project Roles and Permission Schemes

This functionality is currently available only on browser.

Project roles and Permission schemes work in tandem to apply access permission to documents grouped together within a Project. A Project references only one Permission scheme, which is comprised of Project roles specifying the permissions.

Understanding and Administering Project Roles and Permission Schemes: Desktop

In a Permission scheme, you'll associate permissions with Project roles. For example, in one Permission scheme the Engineers role may be associated with Edit, View and Comment permissions. In the same Permission scheme the Vendor role may be associated with View and Comment permissions only.

Project roles are used to group users into functional roles with associated permissions. For example, you may have Project roles like Designer, Engineer, Reviewer, and Vendor. These roles act as buckets that you will later group specific users into when you're creating a Project.

For example, Onshape provides these default Permissions schemes:

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<tr>
<th>Permission Scheme</th>
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</table>

Additional (advanced) permissions are also supplied for your convenience, enabling you to customize Roles with these permissions (or create your own roles):

- Export
- Link document
Permission schemes and their Project roles are associated with Projects and users during Project creation (or editing). For more information about Projects and associating Permission schemes, see "Understanding and Managing Projects" on page 2293.

**Creating Project roles**

Project roles are simply a named group within Onshape. When creating a Project role, you provide the name and description. For example, if your company employs engineers who design and model within Onshape, and project managers who review designs, you might create a “Designer” Project role and a “Reviewer” Project role.

Project roles have no meaning or effect until associated with permissions and users in a Permission scheme that is applied to a particular project.

Project roles are containers meant to be used to group users and pair them with permissions in a Permission scheme. Create Project roles according to expected job functions of your users.

A Project role corresponds to a job function - for example, Designer, Manager, Drafter - but are just names. They acquire meaning and function in the context of Permission Schemes and Projects.

Onshape provides these default Project roles:

- Project administrators
- Managers
- Engineers
- Reviewers
- Suppliers

You are able to edit the names and descriptions of these default Project roles to suit your needs, and also create your own.

1. Select **Enterprise settings** from the User menu.
2. Select **Project roles** tab.
3. Click Create project role.

4. Enter a name for the Project role and a description (optional).

5. Click Save to create the Project role, or click Cancel to quit without creating the Project role.

**Editing Project roles**

1. Select Enterprise settings from the User menu.

2. Select Project roles tab.

3. Right-click on a selected Project role and select Edit.

4. Make desired changes to the name or description.

5. Click Save to save changes or click Cancel to quit without saving the changes.

Notice at the bottom of the dialog is the Role id field: you can use this value in a customized release or obsoletion workflow to indicate that a user acting in a particular role can perform a particular function in the workflow. For more information on customized release workflows, see "Creating a Customized Release Workflow" on page 2320.

**Creating Permission schemes**
Permission schemes are a named table of Role-Permission set pairs. For example, the ProjA Permission scheme could look like this:

In and of themselves, Permission schemes have no effect on who has what access to which documents.

All Permission schemes have a Project Administrator role by default. This role has all of the permissions automatically to prevent accidentally locking everyone out of a Project.

Permission schemes enable you to group permissions together in logical collections so you can assign them to users when defining a Project. Permission schemes reference one or more roles (described above) paired with one or more permissions.

1. Select **Enterprise settings** from the User menu.
2. Select **Permission schemes** tab.
3. Click **Create scheme**.
4. Enter a name and (optional) description for the new Permission scheme.
5. Click **Create** to create the permission scheme (or click **Cancel** to close the dialog without creating a Permission scheme).

The Project Administrators role is a standard Onshape Project role that is automatically added to all Permissions schemes upon creation. This action may not be undone or edited.

At this point, you should add the desired Project roles to the newly created Permission scheme:

a. Click **Add project role**.

b. In the dialog, select the Project role to add and then select the group of permissions to go along with it.

c. You are able to select a group of permissions, or select **Advanced** to specify a customized group of permissions.

d. Click **Add** to save the Role/Permission assignment to the Permission scheme.

6. Add more Project roles to the Permission scheme, if needed.
**Editing Permission schemes**

1. Select **Enterprise settings** from the User menu.
2. Select **Permission schemes** tab:
   a. Right-click on the selected Permission scheme and select **Edit details** to edit the name or description.
   b. Right-click on the selected Permission scheme and select **Manage project roles and permissions** to add Project roles, edit role permissions, or remove a Project role from the Permission scheme.
3. To edit role permissions, right-click on the Role and select **Edit role permissions** or double-click on the role name.
4. Use the drop down to select a new set of permissions or select **Advanced...** to display additional permissions from which to select:

![Edit project role and permissions dialog](image)

The permissions already associated with that role are checked. Make any changes you wish, remembering that changes affect all Projects that use that Permission scheme. Permissions required for other permissions are automatically checked (for example: Share permission requires Edit, and Delete permission also requires Edit).

5. Click **Save** to save your changes, or click **Cancel** to close the dialog without making any changes.

**Using Project roles and Permission schemes**
Project roles and Permission schemes are used in the definition of a Project. For information on creating and editing Projects, see "Understanding and Managing Projects" on page 2293

**Tips**

- If a Permission scheme is being used by (has been assigned to) any project, you are not able to delete that Permission scheme. When you change a Permission scheme, the change affects all the projects that reference the Permission scheme (and all the documents in those Projects); you are able to remove a Permission scheme from a project and assign a new one.

- We recommend that you set up your ideal matrix for all roles and permissions you want to put users in eventually, but use the quick setup instructions and the Onshape defaults to get up and running quickly.

### Understanding Project Roles and Permission Schemes: iOS

Project roles and Permission schemes work in tandem to apply access permission to documents grouped together within a Project. A Project references only one Permission scheme, which is comprised of Project roles specifying the permissions.

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- Export
- Link document
- Share
- Delete

Permission schemes and their Project roles are associated with Projects and users during Project creation (or editing). For more information about Projects and associating Permission schemes, see [Understanding and Managing Projects](#).

**Creating Project roles**

Project roles are simply a named group within Onshape. When creating a Project role, you provide the name and description. For example, if your company employs engineers who design and model within Onshape, and project managers who review designs. You might create a “Designer” Project role and a “Reviewer” Project role.

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Onshape provides these default Project roles:

- Project administrators
- Managers
• Engineers
• Reviewers
• Suppliers

You can edit the names and descriptions of these default Project roles to suit your needs, and also create your own.

These tasks must be performed on the browser platform of Onshape.

1. Select **Enterprise settings** from the User menu.
2. Select **Project roles** tab.
3. Click **Create project role**.

4. Enter a name for the Project role and a description (optional).
5. Click **Save** to create the Project role, or click **Cancel** to quit without creating the Project role.

**Editing Project roles**

1. Select **Enterprise settings** from the User menu.
2. Select **Project roles** tab.
3. Right-click on a selected Project role and select Edit.

4. Make desired changes to the name or description.

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Notice at the bottom of the dialog is the Role id field: you can use this value in a customized release or obsoletion workflow to indicate that a user acting in a particular role can perform a particular function in the workflow. For more information on customized release workflows, see "Creating a Customized Release Workflow" on page 2320.

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Permission schemes enable you to group permissions together in logical collections so you can assign them to users when defining a Project. Permission schemes reference one or more roles (described above) paired with one or more permissions.

1. Select **Enterprise settings** from the User menu.
2. Select **Permission schemes** tab.
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4. Enter a name and (optional) description for the new Permission scheme.
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At this point, you should add the desired Project roles to the newly created Permission scheme:
a. Click **Add project role**.

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3. To edit role permissions, right-click on the Role and select **Edit role permissions** or double-click on the role name.

4. Use the drop down to select a new set of permissions or select **Advanced...** to display additional permissions from which to select:
The permissions already associated with that role are checked. Make any changes you wish, remembering that changes affect all Projects that use that Permission scheme. Permissions required for other permissions are automatically checked (for example: Share permission requires Edit, and Delete permission also requires Edit).

5. Click **Save** to save your changes, or click Cancel to close the dialog without making any changes.

**Using Project roles and Permission schemes**

Project roles and Permission schemes are used in the definition of a Project. For information on creating and editing Projects, see [Understanding and Managing Projects](#).

**Tips**

- If a Permission scheme is being used by (has been assigned to) any project, you can’t delete that Permission scheme. When you change a Permission scheme, the change affects all the projects that reference the Permission scheme (and all the documents in those Projects); you can remove a Permission scheme from a project and assign a new one.

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A Project role corresponds to a job function - for example, Designer, Manager, Drafter - but are just names. They acquire meaning and function in the context of Permission Schemes and Projects.

Onshape provides these default Project roles:

- Project administrators
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- Suppliers

You can edit the names and descriptions of these default Project roles to suit your needs, and also create your own.

These tasks must be performed on the browser platform of Onshape.

1. Select **Enterprise settings** from the User menu.
2. Select **Project roles** tab.
3. Click **Create project role**.
4. Enter a name for the Project role and a description (optional).
5. Click **Save** to create the Project role, or click **Cancel** to quit without creating the Project role.

**Editing Project roles**

1. Select **Enterprise settings** from the User menu.
2. Select **Project roles** tab.
3. Right-click on a selected Project role and select **Edit**.

4. Make desired changes to the name or description.

5. Click **Save** to save changes or click **Cancel** to quit without saving the changes.

Notice at the bottom of the dialog is the Role id field: you can use this value in a customized release or obsoletion workflow to indicate that a user acting in a particular role can perform a particular function in the workflow. For more information on customized release workflows, see "Creating a Customized Release Workflow" on page 2320.

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2. Select **Permission schemes** tab.
3. Click **Create scheme**.
4. Enter a name and (optional) description for the new Permission scheme.
5. Click **Create** to create the permission scheme (or click Cancel to close the dialog without creating a Permission scheme).

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a. Click **Add project role**.

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c. You can select a group of permissions, or select **Advanced** to specify a customized group of permissions.

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### Using Project roles and Permission schemes

Project roles and Permission schemes are used in the definition of a Project. For information on creating and editing Projects, see [Understanding and Managing Projects](#)

### Tips

- If a Permission scheme is being used by (has been assigned to) any project, you can’t delete that Permission scheme. When you change a Permission scheme, the change affects all the projects that reference the Permission scheme (and all the documents in those Projects); you can remove a Permission scheme from a project and assign a new one.

- We recommend that you set up your ideal matrix for all roles and permissions you want to put users in eventually, but use the quick setup instructions and the Onshape defaults to get up and running quickly.

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**Understanding and Managing Projects**

This functionality is available on Onshape's browser, iOS, and Android platforms.

### Understanding and Managing Projects: Desktop

Projects enable you to group documents and apply permissions to those documents for individual users and groups of users (called Teams). The permissions are
assigned at the Project level through the association of a Permission scheme along with the specification of a role map (one per Project) that contains a list of user/role pairs. (The user may be an individual user or a Team of users.)

The role map plus the Permission scheme determines the level of access users (or Teams) have to the project and the documents within. Permission schemes may be edited at any time and the edits directly and immediately affect any project that refers to that permission scheme. Editing Permission schemes may render some entries in a role map ineffectual or change the permission level of users or Teams in the Project's role map.

One document is only able to belong to one Project at a time and documents may be moved into and out of Projects (by users with Edit permission on the Project).

Documents (and Folders) may be created inside of a Project, moved into a Project, moved out of a Project, and deleted altogether.

When a project is created, the creator is automatically placed into the Project Administrator role, which grants full permission to that user. A project must always have a project admin (or someone with a similar full-permission role).

Steps

On the Documents page:

1. Select then in the dropdown, select Project.

   The creator of the Project is automatically assigned the role of Project Administrator upon creation of the Project. This may not be undone.

2. Supply a name and description for the Project.

3. Select a Permission scheme from the list.

4. Create the Role map by selecting a User or Team and then an associated Role from the drop down menus:
5. Click "Add" to add each User/Role entry. The entry will appear in the lower portion of the dialog.

To enter individual users (via email address), the user must first be a part of the company domain as defined under Company settings > Users. Teams must also be pre-defined under Company settings > Teams.

6. Repeat steps 4 and 5 to create additional entries in the Role map.

7. Click "OK" to save the Project state, (or click "Cancel" to close without creating the Project).
A change to a Permission scheme affects all Projects that reference that Permission scheme.

Cloning existing projects

The option to clone an existing project provides an easy way to create a new project with the same permission scheme, settings, users, and role map as an already-existing document. On the documents page:

1. Under Projects, right-click on the project you wish to clone.

2. A context menu appears:

   - Open
   - Open in new browser tab
   - Edit project...
   - Clone project...
   - Hide details
   - Send to trash

3. Click Clone project...

4. A dialog box appears, already filled out with the same properties from your previous project.

5. Click OK in the bottom right corner to create your cloned project.

Modifying a Project state

Modifying a project’s Permission scheme, role map, name, and description requires that the user have Share permission.

Removing users or teams

On the Documents page:

1. Right-click the Project in the list and select Edit project.

2. Click the X next to the role in the Role map to delete an unwanted entry from the Role map. Add role map entries by selecting users and/or teams and the associated Role as described above.
Changing permission scheme

On the Documents page:

1. Right-click the Project in the list and select **Edit project**.

2. Use the drop down to select a new Permission scheme.
   
   When a new Permission scheme is selected, all users and teams are removed from the Project role map (except the original Project Administrator) thus resetting the role map.

3. Make a new role map by specifying users and roles as described above.

4. Click **OK** to save changes, or click **Cancel** to close the dialog without saving any changes.

Moving documents into and out of Projects

Projects are basically containers for documents allowing you to group them; the permissions you apply to the Projects affect all documents inside the Project as well. Folders also hold documents and other folders, but cannot hold Projects. Projects are always created at the root (My Onshape) level. Projects are able to include folders and sub-folders. The permissions on folders within Projects are inherited from the permissions assigned to the Project.

Since permissions are applied to Project via Permission schemes and those permissions apply to all documents and folders within the Project, when you move documents or folders into and out of Projects, the permission scheme assigned to the Target project becomes the permissions assigned to the document and folders moved.

The action of moving documents and folders into and out of Projects itself requires the appropriate permissions:

- To move a document or folder within a Project requires Edit permission on the document or folder's Project. One example of moving a document within a Project is moving it from one folder to another within the same Project.

- To move a document or folder into or out of a Project requires full permission on both the source Project and the target project. Full permission includes: Delete, Share, Edit, View, Link, Copy, Export, and Comment.
Since the permissions to the documents within a Project are assigned at the Project level, moving a document into or out of a Project may result in a change of permissions on the document. (See below for more information.)

1. Select a document in the Documents list.
2. Right-click on the document name on the Documents page. (Or select multiple documents and then right-click.)
3. Select Move to... from the menu.
4. Select a project.
   Click Show details to see which document/s you are moving.
5. Click Move here to move the document/s, or click Cancel without executing the move operation.

To move a document out of a project:
1. Select the document (or documents) within the project.
2. Right-click and select Move to... from the menu.
3. Use the breadcrumbs at the top of the dialog to navigate to the desired level, project, or folder.
4. Click Move here to move the document/s, or click Cancel without executing the move operation.

**Copying document workspaces in projects**

When copying a workspace in a project (thereby creating a new document), you need edit permission on the project in order for the copy to be placed in the same project as the original document. If you do not have edit permission on the project, the copy is placed at the root level of your Onshape account which is My Onshape. A blue notification is displayed at the top of the window upon successful copy that is has been created in My Onshape.

**Tips**

- Use the drag and drop method of moving documents and folders into and out of other folders and projects.
• There is no way currently for administrators to channel users into creating documents in certain projects.
• Projects are listed in the My Onshape page filter.
• You are only able to move a document out of a Project if you have Share permission on the Project and into another Project if you have Edit permission on that Project.

**Understanding Projects: iOS**
Projects are collections of documents that are associated with Permission schemes (Project roles and Permission pairs) which apply permissions to groups of users. The permissions associated with a group of users for a project apply to all documents inside the project.

Users with the Edit permission are free to create documents: inside of or outside of Projects with which they have been associated as long as they have the proper permissions to the Project. Users are associated with Projects by way of the Permission scheme that is associated with the Project.

One document can belong to, at most, one Project and documents may be moved into and out of Projects (by users with Edit permission on the document).

Project definitions contain this information:
• Name
• Description
• Permission scheme and associated users
• Users and the roles they may act in

A change to a Permission scheme affects all Projects that reference that Permission scheme.

Modifying a project’s Permission scheme, Project roles, name, and description requires that the user have Reshare permission. Adding a document to or removing a document from a Project requires that the user have Edit permission.

Projects definitions can be administered or otherwise edited only on a browser.
Permissions

When a Project is created, the creator is automatically placed into the Project Administrator role, which grants full permission to that user. A project must always have a project admin (or someone with a similar full-permission role) to prevent users from getting into a situation where they are locked out of their project.

Changing from one Permission scheme to another on a project automatically remaps users whose roles exists in both schemes (the original Permission scheme and the new one being assigned).

The dialogs in this process are ‘draft’ which means you must confirm the action in order for it to be executed. If you close the dialog without saving or exiting properly, all changes are discarded.

Understanding Projects: Android

Projects are collections of documents that are associated with Permission schemes (Project roles and Permission pairs) which apply permissions to groups of users. The permissions associated with a group of users for a project apply to all documents inside the project.

Users with the Edit permission are free to create documents: inside of or outside of Projects with which they have been associated as long as they have the proper permissions to the Project. Users are associated with Projects by way of the Permission scheme that is associated with the Project.

One document can belong to, at most, one Project and documents may be moved into and out of Projects (by users with Edit permission on the document).

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Projects definitions can be administered or otherwise edited only on a browser.

Permissions

When a Project is created, the creator is automatically placed into the Project Administrator role, which grants full permission to that user. A project must always have a project admin (or someone with a similar full-permission role) to prevent users from getting into a situation where they are locked out of their project.

Changing from one Permission scheme to another on a project automatically remaps users whose roles exist in both schemes (the original Permission scheme and the new one being assigned).

The dialogs in this process are 'draft' which means you must confirm the action in order for it to be executed. If you close the dialog without saving or exiting properly, all changes are discarded.

Configuring Integration with Identity Providers

This functionality is available on Onshape's browser, iOS, and Android platforms.

Before starting the integration process, you must have requested, and been approved for, an Onshape Enterprise account or trial, and have an Onshape Enterprise domain name.

An example of an Enterprise domain name might be: MyCompanyName.onshape.com.

Note that you can use only one single sign-on (SSO) provider at a time.

Onshape supports the following identity providers for single sign-on (SSO) purposes:
• Okta
• Microsoft Azure AD
• Microsoft ADFS
• Google
• PingOne
• OneLogin

The set up for each of these identity providers varies and is explained in separate topics, but the overall steps are similar to these:

1. Add Onshape to your single sign-on account.

2. Download the required configuration file from your single sign-on account.

3. Upload the configuration file into Onshape.

4. In your Onshape administrator account, enable the single sign-on provider for your users.

5. Do a hard refresh of your Onshape sign in page, then sign in with your SSO credentials.

Administrators can enforce Enterprise users to sign in to Onshape with only the configured SSO method and prevent signing in to the non-enterprise domain by toggling "Disable Onshape password sign in."

For more information on integrating with a specific identity providers see the topic explaining the particular provider you use.

---

Integrating with Okta

This functionality is available on Onshape's browser, iOS, and Android platforms.

Single sign on is available on all Onshape platforms including iOS and Android.
Before starting the integration process, you must have requested, and been approved for, an Onshape Enterprise account or trial, and have an Onshape Enterprise domain name.

An example of an Enterprise domain name might be: MyCompany\-\companyName.onshape.com.

Note that you are only able to use one SSO provider at a time.

This configuration process might fail without parameter values customized for your organization. Use your Okta single-sign on and record the values that are specific for your organization. You need those values for the following procedure.

Adding Onshape to your Okta single sign on account

To enable single sign on for your company, you must first add Onshape to your Okta account:

1. Sign in to your Okta account.
2. Click **Applications** in the menu ribbon.
3. Click the **Add Application** button.
4. Type ‘Onshape’ in the search field.
5. When Onshape appears, click the **Add** button.
6. The ‘Add Onshape’ page appears.
7. Enter your domain prefix for your Enterprise. (For example, MyCompany\-\companyName from the URL mentioned above.)
8. Click **Next**.

At this point, you may assign Onshape to users in your account. On the **Assign Onshape to People** page, follow your usual procedure to add more users to the Onshape application.

**Download the single sign on file**

Once you are finished adding users to the Onshape application in your Okta account, download the Identity Provider metadata file (referred to in Onshape as the configuration file):
1. From the Onshape application page within your Okta account, click Sign On from the menu ribbon.

2. In the SAML configuration area, click the Identity Provider metadata link to download the necessary file.

Upload the configuration file in Onshape

After downloading the metadata file from Okta:

1. Sign in to your Onshape Enterprise account, using your specialized domain name, as an administrator.

2. Select Company settings from the User menu.

3. Select Authentication from the left menu.

4. In the Single sign on (SSO) section:

   a. Click Upload configuration file.

   b. Select the metadata file you previously downloaded and click Open.

   c. In the dialog, enter a name for the SSO Provider and check the Enable SSO provider checkbox.
d. Click `OK`.

![Create SSO Provider](image)

5. You see the Authentication section of Company settings listing the newly integrated Single Sign on:


7. Do a hard refresh of the Onshape account page; notice the page has a new **Sign in** link at the bottom, (see the example of **Sign in with Okta** link below):
In order to sign in to Onshape, administrators must provision their users (in their single sign on account) to use the Onshape application.

**Requiring Onshape sign in through Okta**

Once you have signed in to Onshape as administrator, if you'd like to require your users to sign in only through the identity provider, you can return to the Company settings > Authentication page and check 'Disable Onshape password sign in':

To disable the typical Onshape password sign in for your users (and show just the SSO provider sign in), check the Disable Onshape password sign in checkbox.
Note that you can disable the typical Onshape password sign in for your users and show just the SSO provider sign in prompt for the Onshape URL. However, do not perform this step at this time. Make sure you can sign in to Onshape yourself (as administrator) before disabling this additional sign in option. You can return here later, once sign in through your SSO provider has been verified to work correctly.

Choosing to enforce signing in to Onshape via SSO also results in users not being able to sign in to non-enterprise domains directly, such as cad.onshape.com.

With Onshape password sign in disabled, users will not see an Onshape sign in, they will see only an Okta sign in, like the one below:
Integrating with Microsoft Azure AD

This functionality is available on Onshape's browser, iOS, and Android platforms.

Single sign on is available on all Onshape platforms including iOS and Android.

Before starting the integration process, you must have requested, and been approved for, an Onshape Enterprise account or trial, and have an Onshape Enterprise domain name.

An example of an Enterprise domain name might be: MyCompany-Name.onshape.com.

Note that you are only able to use one (single sign-on) SSO provider at a time.

This configuration process might fail without parameter values customized for your organization. Use your Microsoft Azure AD single-sign on dashboard to add Onshape as an application and record the values that are specific for your organization. You need those values for the following procedure.
Add Onshape to your Azure AD single sign on account

To enable single sign on for your company, you must first add the Onshape application to your Azure AD single sign on account:

1. Sign in to the Microsoft Azure portal.

2. Select *Azure Active Directory* in the left pane.

3. Select *Enterprise applications* in the new left pane that opens (next to the pane you just clicked in).
4. At the top of the portal, click *New application*. 
5. On the Add an application page, click Non-gallery application.

![Add an application](image1)

6. The Add your own application pane opens.

![Add your own application](image2)

7. In the Name field on the Add your own application pane, enter "Onshape".

8. Click the Add button at the bottom of the pane.

   The page reloads and the name you supplied, Onshape, is at the top of the page.
9. Under Manage in the left pane, select "Single sign-on".

![Image of Microsoft Azure interface showing Single sign-on option]

10. Since Onshape supports only SAML authentication, select SAML as the single sign-on method.

![Image of SAML explanation]

The Set up Single Sign-On with SAML page loads.

Now it's time to edit the configuration. Each box has an edit icon (a pencil) in the top right corner of the box. Click this icon to edit the values in each box as explained below.
SAML configuration

These steps are performed on the page that displays at the end of the steps above, the Set up Single Sign-On with SAML page:

Note that in order to edit each subsection, click the small pencil icons in the right corner of each subsection.

In the **Basic SAML Configuration** subsection:

1. Click the small pencil icon to activate editing.
2. Specify the **Identifier (Entity ID)**: com.onshape.saml2.sp
The Reply URL, above, must be "cad.onshape.com" and not the URL of your Onshape enterprise.

4. Click Save at the top of the panel.

5. Close the panel.

6. If asked to validate, you can click "Validate later".

**User attributes and claims**

In the **User Attributes & Claims** subsection:

1. Click the small pencil icon to activate editing. The User Attributes & Claims > Add new claim panel appears:
2. Click the small pencil icon to the right of Name identifier value to activate editing and open the Manage user claims panel:

3. Change Name identifier value to "user.mail" in the format emailAddress, as shown above.

4. Click the Save button at the bottom of the panel.

5. Add three new claims, one for each of the substeps (a - c) below:

   a. Click Add new claim: In the panel that appears, add Name as "firstName" with the Source attribute as user.givenname.
Click the Save button at the bottom of the panel. Close the panel.

b. Click Add new claim: Add Name as "lastName" with the Source attribute as user-.surname.

Click the Save button at the bottom of the panel. Close the panel.

c. Click Add new claim: Add Name as "companyName" with your Onshape enterprise domain prefix (the string before the ".onshape.com" in the URL you use to access Onshape) as the Source attribute. For example, if your URL for your Onshape enterprise is "fishbowl.onshape.com", enter just "fishbowl" here.

6. Close the Claim panel (click the X in the top right corner).

The SAML set up pane is displayed again.
7. In the pane, scroll to subsection 3 **SAML Signing Certificate**.

![SAML Signing Certificate](image)

8. At the bottom of this subsection, click "Federation Metadata XML" download.

A message appears in the top right upon successful download of this file.

**Set up users and groups**

In the left pane under Manage:

1. Select **Users and groups**.

![Users and groups](image)

The Add user pane appears.
2. Click Add user at the top.

3. The add Assignment pane appears.

4. Click in the Users and groups field to open the Users and group right-hand pane.

5. In this pane, you can select users and/or groups, as well as search for members and enter email addresses in order to invite users.

6. Once all selected members are listed in the pane, click the Select button.

At this point, you can leave the Azure portal and open your Microsoft active directory application dashboard.

Upload the configuration file in Onshape

After downloading the metadata file:
1. Sign in to your Onshape Enterprise account, using your specialized domain name, as an administrator.

2. Select **Company settings** from the User menu.

3. Select **Authentication** from the left menu.

4. In the Single sign on (SSO) section:
   a. Click **Upload configuration file**.
   b. Select the metadata file you previously downloaded and click **Open**.
   c. In the dialog, enter a name for the SSO Provider and check the **Enable SSO provider** checkbox.
   d. Click OK.

   Note that you can disable the typical Onshape password sign in for your users and show just the SSO provider sign in prompt for the Onshape URL. However, do not perform this step at this time. Make sure you can sign in to Onshape yourself (as administrator) before disabling this additional sign in option. You can return here later, once sign in through your SSO provider has been verified to work correctly.

   Choosing to enforce signing in to Onshape via SSO also results in users not being able to sign in to non-enterprise domains directly, such as cad.onshape.com.

5. Sign out of the Onshape account.

6. Do a hard refresh of the Onshape account page; notice the page has a new **Sign in** link at the bottom for your SSO provider.

   In order to sign in to Onshape, administrators must provision their users (in their single sign on account) to use the Onshape application.

**Troubleshooting**

If you see the following error:
An error occurred during external sign-in, possibly due to how long it's been since you signed in to your external identity provider. Please try signing out from your external identity provider and then signing in to Onshape again.

Close all tabs, fully sign out, and sign back into Microsoft.

Sign into Onshape again.

Note that the "test connection" button in the AD admin panel will not work.

Creating a Customized Release Workflow

This functionality is currently available only on browser.

Administrators of enterprises can create customized release and obsoletion workflows specific to the enterprise needs. Onshape provides a JSON file of our release and obsoletion workflows as a starting place for customizing the workflow to suit your company's needs. You can also use our JSON as a starting point for creating your own JSON file.

Bear in mind that the currently selected release and obsoletion workflow (chosen by the administrator) governs newly created release candidates through the selected process. If you change the workflow while a release is in-process, that release follows the workflow under which it was created. The newly selected workflow will govern only release candidates created after it was selected.

One more thing: Onshape customized workflows currently do not allow cyclical processes.
At this time, you are not able to delete a workflow, but you are able to add as many as you want to your workflow repertoire to select from. You are also able to replace an existing customized workflow, if necessary.

We recommend keeping all of your workflow definition files in one Onshape document for ease of access.

The customization process has these major steps, each explained in detail below:

1. Download the Onshape default workflows to your hard drive or other easily accessible location.

2. Create a new Onshape document and import the workflow files.

3. Edit and publish the workflows.

**Automatic transition for two-tiered workflows**

When you set up a custom release workflow that includes two tiers, and the second tier is not required (that is, the second tier is optional) Onshape will automatically transition from Pending to Released if there is no approver specified in the release package. No setup is required for this, the only requirements are that you have a two-tier workflow, the second tier is optional and there are no approvers specified in the Approver field in the release candidate.

**Download the Onshape default workflows**

To download the Onshape default workflow files:

1. Navigate to your Company settings and select Release management.

2. Confirm that Enable managed workflows is checked, then click View in document.

   ![View in document](image)

   The workflow file opens in a public document in another tab.

3. Click Download, enter a new name for the file.

   You can close that tab once the tab is downloaded.
Create a new Onshape document and import the workflow files

1. In your enterprise, navigate to the Documents page, select Create > Document, then select OK. (In this example, the document name is Company Customized Workflows)

2. In the new document, click the Plus icon at the bottom and select Import:

3. Select the workflow file you just downloaded and click Open.

4. Another tab appears in your document, with the name of the file you imported. Select that tab to open it:
Edit and publish the workflows

Edit the custom workflow according to the guidelines below:

- As you edit, the diagram on the right will reflect the changes made in your JSON.
- The diagram is dynamic and draggable, so if the structure does not reflect your intended workflow, use your cursor to adjust it.
- Cyclical workflows are not allowed.
- Once you are satisfied with your edits and want to make your workflow available for use in the enterprise, click to open the Publish custom workflow dialog:

   ![Publish custom workflow dialog]

This dialog will alert you of any errors that may have to be fixed before publishing. (After you click Publish, if there are errors in your workflow code they will be displayed at the top of this dialog. Cancel the dialog, fix the errors and click Publish again to open this dialog). Once your workflow is error-free: provide a name, description, and workflow type. If this is a brand new workflow, not replacing another workflow, select None in the Replace workflow dropdown. If you wish to replace an already-existing workflow, select the one you wish to replace.
Click **Publish**.

Enabling multiple active workflows

Once you have published a custom workflow, you can enable multiple active workflows in your enterprise if you wish. When multiple active workflows are available to your users, they can select which workflow to follow when creating a release candidate.

It is the responsibility of the administrator to educate users on which workflows should be used and under what conditions.

To enable multiple workflows:

1. Follow the instructions above to create more than one custom workflow and publish them in Onshape.

2. In the Release management page of Company settings, select one of the custom workflows from the Managed workflows drop down:

3. Once a custom workflow is selected, a check box for enabling multiple workflows appears below the workflow diagram:
4. Check the box to access the additional workflows and select as many as you want to make available to users:

Since 01Review workflow is already selected (above) as the enabled workflow, its checkbox cannot be unchecked.
5. Make sure to click **Save release settings** to save the changes you've made to the Release management settings for the enterprise.

When your users create a release candidate, they are presented with an opportunity to select a workflow at that time (from the dropdown):

```
Syntax

The syntax for customizing the release or obsoletion workflow is presented below. Note that capitalization is important.

Keep in mind the following rules.
```

**Properties**
Format

- name: string
- propertyId: string
- valueType: string|ENUM
- defaultValue?: any
- usersOnly?: boolean
- teamsOnly?: boolean
- enumValues?: list

Restrictions

- Property IDs must be unique (and not match Onshape's hard-coded Name, Description, or Comment property ID)
- valueType must be the name of a BTMetadataValueType (excluding BLOB and OBJECT)
- If valueType is ENUM, enumValues must be specified, and the property must contain one or more enumValues. Each of the enumValues must have a mandatory value element. The label element is optional.
- If the defaultValue is provided, it should match the value field of one of the enumValues.
- usersOnly and teamsOnly are mutually exclusive and only valid for USER properties
- defaultValue, if present, must match the correct type indicated by valueType
  - STRING: string
  - BOOL: boolean
  - INT: integer
  - DOUBLE: decimal
  - USER: string[]}
DATE: ISO date string

BLOB and OBJECT are currently not supported

USER-type property values are a list of user, team, and/or role IDs

**States**

**Format**

- name: string
- displayName: string
- approverSourceProperty?: string
- notifierSourceProperty?: string
- entryActions?: Action[]
- exitActions?: Action[]
- editableProperties?: string[]
  - Values may include:
    - "approvers"
    - "observers"
- requiredProperties?: string[]
- requiredItemProperties?: string[]

**Restrictions**

- Names must be unique
- approverSourceProperty and notifierSourceProperty, if present, must be the property IDs of the workflow and must point to USER-type properties
- At most, one state may use a given property as its approvers.
- Any number of states may use a given property as their notifiers.
- editableProperties and requiredProperties must be property IDs in the same workflow.
requiredItemProperties must be valid metadata property IDs for your company (Onshape built-in properties or custom properties, excluding computed properties).

For more information on IDs, refer to "Understanding and Administering Project Roles and Permission Schemes" on page 2276.

Transitions

Format

- name: string
- displayName: string
- type: string
- sourceState: string
- targetState: string
- uiHint?: string (default: “primary”)
- actions?: Action[]
- requiredProperties?: string[]

Restrictions

- Names must be unique.
- Type must be one of “APPROVE,” “REJECT,” or “SUBMIT”.
- sourceState and targetState must be two different states within the workflow.
- There may be, at most, one APPROVE-type transition from any source state.
- For an APPROVE-type transition, sourceState must have an approver-SourceProperty.
- uiHint, if present, must be either “primary,” “success,” or “danger” (these are the colors of the boxes and other entities in the release dialog and are hard-coded by Onshape).
Transition types
Submit

- Performs various “initialization” on the release as it goes out of set up and into the workflow, such as kicking off thumbnail generation for configured parts in the release, connecting linked document/version IDs, etc.
- Only the release creator has the ability to execute SUBMIT transitions.
- Transitions out of the initial state should be Submits.
- A Submit that isn’t an initial transition does not have any special abilities

Approve

- Marks the user as having approved (turns the token green in the release dialog).
- If company policies are set to Require all approvers, the transition will not be performed until all approvers have approved.
- Generates assembly/drawing content references for items.
- Only one transition of this type is allowed out of a state.
- Only an approver of the current state (or an admin) has the ability to approve.

Reject

- Marks the user as having rejected (turns the token red in the release dialog).
- Only an approver of the current state (or an admin) has the ability to reject.

Actions

Format

- name: string
- params?: {[key: string]: any}

Restrictions

- Name must match one of our predefined action names (see below, under Allowed Actions)
Some actions may be allowed only on states or on transitions, some may have ordering restrictions.

Allowed Actions

- **sendEmailNotifications** - Send email notifications for a transition (including mobile push notifications).
  - Allowed on: transitions only
  - Parameters: none

- **sendUserNotifications** - Send user notifications for a transition.
  - Allowed on: transitions only
  - Parameters: none

- **markItemsPending** - Change the metadata state of all items in the release to Pending.
  - Allowed on: states or transitions
  - Parameters: none
  - Not allowed on obsoletion workflows.
  - In release workflows, not possible after `markItemsReleased` or `markItemsRejected`.

- **markItemsRejected** - Change the metadata state of all items in the release to Rejected.
  - Allowed on: states or transitions
  - Parameters: none
  - Not allowed in obsoletion workflows.
  - In release workflows, not possible after `markItemsReleased` or before `markItemsPending`

- **releaseItems** - Change the metadata state of all items in the release to Released and create revisions for them (auto-obsoleting other revisions if specified in company policy).
Allowed on: states or transitions
Parameters: none

Not allowed on obsoletion workflows.

- **obsoleteItems** - Change the metadata state of all items in the package to Obsolete and obsolete their corresponding revisions.
  - Allowed on: states or transitions
  - Parameters: none
  - Not allowed on release workflows.

**Overriding company release settings**

When defining a custom workflow, you can include an "options" section designed to override specific company release settings defined on the Company settings > Release management page. These options for overriding are explained below:

```
"options": {
    "revisionSchemeId": string,
    "requireApprover": boolean,
    "requireAllApprovers": boolean,
    "disallowCreatorAsApprover": boolean,
    "requireNote": boolean,
    "autoObsolete": boolean,
    "errorOnFeatureListErrors": boolean,
    "errorOnRolledBack": boolean,
    "errorOnAssemblyErrors": boolean,
    "errorOnAssemblyRefsOutOfDate": boolean,
    "errorOnDrawingOutOfDate": boolean
    "errorOnPartNumberPending": boolean
    "errorOnPendingTask": boolean
```
This section and all fields are optional. You can pick any subset with which you want to override a company setting. Any fields not included in the code will default to the specification in the Company settings.

The revision Scheme Id can be retrieved from the read-only text box on the Company settings > Release Management page:

![Revisions and Part Numbers](image)

**Monitoring User Activity**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Upon signing in to your Enterprise, the Activity page immediately loads. Here you are able to browse all Onshape activity of your Enterprise users from simple changes made to documents to viewing and responding to comments, and even viewing the differences (diffs) between versions of documents. If you are not an administrator of your company subscription, you see your own Activity stream.

**Monitoring User Activity: Desktop**
Here's a rundown of the information you have access to and the actions you are able to take.

**Filter Activity stream**

The panel on the left lists filters. These are primarily Project names so you may zero in on seeing all the activity for one Project at a time. Use the folder icon on the border of the Filter list to expand and collapse the list. You are also able to click the border and drag to resize the width of the list.

- **All activity** - View all activity on all documents in the Enterprise.
- **By project** - Click a project to see activity only on the documents included in that project. A selected project is indicated by a blue bar to the left of the name, and the name is also blue and bold.

Within each activity stream you see tiles for each user action. The tiles contain information like:

- **User name** - Click to display an information pane with user-specific information like email address, status, last time signed in, and a scrolling list of that user's recent activity.
• Document acted upon with date and time - Click the document name to display an information pane with document-specific information like a thumbnail, the document description, document labels, sharing specifications, and the folder location (click this link to open the folder and display the contents on the Documents page (this action redirects the page). Also shown is the name of the user who created the document and when it was last modified and by whom. The default workspace is also shown, as well as a list of recent activity.

• Overflow menu (top right of the tile) - Use this menu to Open the document in a new tab.

• When a document has been edited (as opposed to simply opened), you see a Show changes option below the thumbnail. Click this option to expand the tile to include a list of the changes made during that edit session.

• When there is release management activity, the status of the release is displayed along with the note and the name of the workflow used.

View user details

Click the user name in the tile to open the information pane (to the right).

This panel displays general user details as well as a listing of all user-specific activity on all documents. When comments are shown, you are able to reply directly from this panel.

If a document name is listed in the user’s activity, click the document name to open it in another tab.
Notice the User details icon (shown above outlined in blue). To view a user's recent activity, click the icon.

**View document details**

Click the document name in the tile to open the document-specific information pane (to the right).

This pane displays document details as well as all activities performed on the document by various users. Use the link below the document thumbnail to open the document in a new tab. When comments are shown, you are able to reply directly from this panel. You are also able to open the document from the overflow menu in the main activity stream:
Notice the icons outlined above in blue: click the Details icon to display more details about the document. Click the Share details icon to display the permissions on the document and the folder or project (if any) it resides within. Click the Recent activity icon to display the recent activity of the document.

**View and reply to comments**

When the action performed on a document is a comment, that comment is shown along with a link to Show all replies to that comment. You are able to reply directly from the tile. Type your reply in the box and click Add.
If you mention a user in a reply to a comment (using @<email address>) and that user does not have any permissions on the document, a notification opens:

![Notification showing missing permissions]

Longer comments may get truncated in the Activity stream, but will have a "Show more" link below them for the option to expand the comment to its full content.
The truncated comment before 'Show more' is clicked, above

The expanded comment, with a 'Show less' link to collapse the comment, above

**View changes**

When the action performed on a document is a modeling action (creation or edit of a sketch, feature, assembly, etc), a link to **Show changes** is displayed below the document thumbnail.
View diff

When the action performed on a document affects a version of the document, a link to View diff is shown below the thumbnail. Click View diff to open the View diff popup. Cycle through the document's tabs here, viewing the difference between the version thumbnails. Click Open 3D compare to open the document in a new tab, in the Compare window.
Open 3D compare

When comparing your document with 3D compare, you view the differences between the 3D view of the selected Part Studio. See Comparing for more information.
You are also able to choose to show the 2D compare which displays the model in black and white with changes shown in color codes (red=previous and blue=newer) of the versions (or workspace and version).
**Monitoring User Activity: iOS**

Upon signing in to your Enterprise account the Activity page immediately loads. Here you are able to browse all Onshape activity of your Enterprise users (if you are an Administrator) or yourself (if you are not an Administrator). You are able to see a list containing simple changes made to documents, and you are also able to view and respond to comments, and even view the diffs between versions of documents.

Here’s rundown of the information you have access to and the actions you are able to take.

**Anatomy of the Activity stream**

Tap the funnel icon in the top left of the screen to open the filter panel:
Filter Activity stream

- **All activity** - View all activity on all documents in the Enterprise.

- **By project** - Tap a project to see activity only on the documents included in that project.

Open document

Once in a project, tap the overflow menu and select Open document to open the document in a new tab.

View user details

Tap the user name in an activity listing to open the user-specific information panel (to the right).

This panel displays general user details as well as user-specific activity on all documents. When comments are shown, you have the ability to reply directly from this panel.

View document details

Tap the document name in an activity item to open the document-specific information panel (to the right).
This panel displays document details as well as all activities performed on the document by various users. Use the Open link below the document thumbnail to open the document in a new tab. When comments are shown, you have the ability to reply directly from this panel. You also have the ability to open the document from the overflow menu in the main activity stream.

**View and reply to comments**

When the action performed on a document is a comment, that comment is shown along with a link to Show all replies to that comment, if appropriate.
View changes

When the action performed on a document is a modeling action (creation or edit of a sketch, feature, assembly, etc), a link to Show changes is displayed below the document thumbnail.

View diff

When the action performed on a document affects a version of a document, a link to
View diff is shown below the thumbnail. Click View diff to open the View diff popup. Cycle through the document’s tabs here, viewing the difference between the version thumbnails. Click Open 3D compare to open the document in a new tab, in the Compare window.

For more information on comparing versions of documents, see Comparing.

**Open 3D compare**

When comparing your document with 3D compare, you view the differences between the 3D view of the selected Part Studio. See Comparing for more information.

**Monitoring User Activity: Android**

Upon signing in to your Enterprise account the Activity page immediately loads. Here you are able to browse all Onshape activity of your Enterprise users (if you are an Administrator) or yourself (if you are not an Administrator). You are able to see a list containing simple changes made to documents, and you are also able to view and respond to comments, and even view the diffs between versions of documents.

Here’s rundown of the information you have access to and the actions you are able to take.

**Anatomy of the Activity stream**

Tap the funnel icon in the top left of the screen to open the filter panel

**Filter Activity stream**

- **All activity** - View all activity on all documents in the Enterprise.
- **By project** - Tap a project to see activity only on the documents included in that project.

**Open document**

Once in a project, tap the overflow menu and select Open document to open the document in a new tab.

**View user details**

Tap the user name in an activity listing to open the user-specific information panel (to the right).
This panel displays general user details as well as user-specific activity on all documents. When comments are shown, you are able to reply directly from this panel.

**View document details**

Tap the document name in an activity item to open the document-specific information panel (to the right).

This panel displays document details as well as all activities performed on the document by various users. Use the Open link below the document thumbnail to open the document in a new tab. When comments are shown, you have the ability to reply directly from this panel. You also have the ability to open the document from the overflow menu in the main activity stream.

**View and reply to comments**

When the action performed on a document is a comment, that comment is shown along with a link to Show all replies to that comment, if appropriate.

**View changes**

When the action performed on a document is a modeling action (creation or edit of a
sketch, feature, assembly, etc), a link to Show changes is displayed below the document thumbnail.

View diff

When the action performed on a document affects a version of a document, a link to View diff is shown below the thumbnail. Click View diff to open the View diff popup. Cycle through the document’s tabs here, viewing the difference between the version thumbnails. Click Open 3D compare to open the document in a new tab, in the Compare window.

For more information on comparing versions of documents, see Comparing.

Open 3D compare

When comparing your document with 3D compare, you view the differences between the 3D view of the selected Part Studio. See Comparing for more information.

Monitoring Releases and Tasks: Action Items

This functionality is currently available only on browser.

View and filter lists of releases and task activities on the Action items tab.

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1. **Action Items** - To return to the Action Items page from the Activity, Documents, or Analytics page, click Action Items at the top of the page, shown above.

2. **Filter column** - In the filter column on the left side of the page, you have the ability to use the selections to filter through releases and/or tasks in the center of the page. The filter options include:
   
   a. **Type** - Choose between filtering through release activities, task activities, or both, by selecting the check boxes to the left of the filter options.
   
   b. **Role** - Choose between filtering through releases or tasks assigned to you or created by you by selecting one of the radio buttons to the left of the filter options.
   
   c. **Status** - Choose between filtering through open releases or tasks or closed releases or tasks by selecting one of the radio buttons to the left of the filter options.

3. **Action Items list** - As you select different filters, the results for the corresponding selections will appear in the Action Items list located in the center of the page.

4. **Sort** - Sort your filtered results by Oldest releases and/or tasks first or Newest first by clicking the dropdown arrow under Sort and selecting your preference (Onshape defaults to sorting by Oldest first).
Working with the Action items list

In the list of action items, there are many tools to interact with and obtain information from:

- The blue bullet at each item: when open (white center) it indicates a task you have not yet expanded. A closed (blue) bullet indicates an item that has been expanded previously.
- Click an item to expand it:

![Image of action item list]

The name of the document associated with the action item is shown along with the document thumbnail. Click the name of the document to open it.

For release tasks, the release notes are included, the last comment made, and the action associated with this workflow notification. The current state of the release is also noted, along with the names of the approvers and a color-coded indication of their action: Red for rejected or discarded, white for pending, and green for approved.

There is also a link to view the release (View release). This does not open the document, but opens the Review release dialog.

Accessing Analytics

This functionality is available on Onshape's browser, iOS, and Android platforms.

Onshape provides dashboards for a variety of categories of information.
Accessing Analytics: Desktop

Each user has access to all of the reports available to the enterprise, as long as they have been granted access to the Access analytics global permission by the Enterprise administrator:

Select a dashboard from the major list, or through one of the filters on the left. When viewing a dashboard, you can download the data through the gear menu in the upper right, or use the key combination Ctrl+Shift+D (Windows) or Cmd+Shift+D (Mac). You may download in CSV or PDF format. When viewing in a browser, be aware that the back end of the Analytics is run by Looker Data Sciences, Inc.

Controls available within dashboards

All data is presented in dashboards with similar filtering, sorting, and refining capabilities. In general:

- This icon (available on hover) on a data group indicates that you are able to download the data:
Click the icon and click the Download Data... command.

- Filter data by expanding the FILTERS dropdown under the dashboard title. When filter criteria is selected, click Run on the right to generate and display the new report:

![Activity Overview](image)

*The FILTERS dropdown is shown on the left and opens the parameters available for filtering (seen below the left-facing arrow); the Run button is on the right.*

Click the plus sign to the right of the filter parameter to add another parameter:

![Release Audit](image)

*When you click the plus sign, another parameter line is available and an X appears; use the X to remove the parameter.*

Click Run to generate the report using the specified filter parameters.

You can save a filter for use later as a dashboard. After selecting your filter options and running the report, click ![icon] to create a new dashboard:
The dashboard is saved both in My analytics and in the category under which you created it.

You can **share** a saved dashboard with other users, if you have permission, and if the dashboard is one that has been saved (that is, not a default dashboard). Click Share (next to Save), specify the individuals, or teams, with whom to share, select the desired permissions, and click Share:

```
is any value   Workflow is a  Run
```

---

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Click an ellipsis next to a specific entity to access a dashboard specific to that entity:

For example, clicking the ellipsis (called out in blue below) next to a release name on the Release Activity dashboard, opens the dashboard regarding that particular release (or release candidate) as shown in the second image below:
Controls available within reports

In the resulting report you are able to perform actions on the data, explained below by report data display entity.

Tables

You are able to sort by any column with a caret next to the name (^ for instance): click the column name or the caret to sort by that column. You are also able to act on the data in table cells:
Click the underlined name (or the ellipsis) to open a menu of related dashboards; select one to display that dashboard, or select the Open in Onshape link to open the document in Onshape (in another tab).

Click an underlined (on hover) data point in a column to open a detailed information panel.

For instance, click on a Modeling time to open a detail panel showing a breakdown of users and their time spent modeling:

<table>
<thead>
<tr>
<th>Project</th>
<th>Documents</th>
<th>Modeling time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Piston Pump</td>
<td>1</td>
<td>28:43:41</td>
</tr>
<tr>
<td>Landing Gear</td>
<td>1</td>
<td>3:42:48</td>
</tr>
<tr>
<td>Kiosk for Customer XYZ</td>
<td>1</td>
<td>0:06:11</td>
</tr>
<tr>
<td>Throttle Body</td>
<td>2</td>
<td>22:24:59</td>
</tr>
</tbody>
</table>

Click the X in the upper right corner of the detail panel to close it.

Charts

Hover over segments of the pie charts to see detailed information:
Hover over a line in the legend to see the coordinating section of the chart highlight.
Click the hover information to access a menu of related dashboards; click to select one or to open the referenced document.
Click an item (in a bar chart) or hover (in a pie chart) in the legend to toggle the display of that data (or the remaining data)
Maps

Maps show the worldwide locations of activities. Click and drag to reposition the map for access to specific regions. Use the zoom icons on the map to zoom in and out. Click a point on the map to display information about that location's activities. Click the information to open a panel with more detailed information.
Click the X in the upper right corner of the detail panel to close it.

Webs display summary information; hover over a hub point to display summary information. Click to reduce the web to information relating directly to the hub selected. Click again to revert to all data.
Accessing Analytics: iOS

Onshape provides dashboards for a variety of categories of information.

Each user sees and has access to only the reports that the Global permissions grant access to. The main page lists all the dashboards available to you, depending on the Global permissions associated with your account.

Export any dashboard through the gear menu in the upper right. You can export as a PDF or a Single Column PDF. When viewing in a browser, be aware that the backend of the Analytics is run by Looker (Looker Data Sciences, Inc.).

All data is presented in dashboards with similar filtering, sorting, and refining capabilities. In general:

- The three dot icon (available on hover) on a data group indicates that you are able to download the data.
- Filter data by expanding the dropdown under the dashboard title. When filter criteria is selected, tap Run on the right of the section to run and display the new report:
In tables:

- Sort by almost any column; select the column name to sort by that column.
- Act on the data in cells:
  - Click to open a menu of related dashboards; select one to display that dashboard
  - Click to open the referenced document in another tab
  - Click to open a detailed information panel

In charts:

- Hover to see detailed information
- Click the hover information to display a menu of related dashboard; click to select one or to open the referenced document
- Click an item (in a bar chart) or hover (in a pie chart) in the legend to toggle the display of that data (or the remaining data)

In maps:

- Use the zoom icons to zoom in and out.
- Click and drag to reposition the map
- Display information about a map point; click to display

In webs:

- Display summary information; hover over a hub point to display summary information. Click to reduce the web to information relating directly to the hub selected. Click again to revert to all data.

**Accessing Analytics: Android**

Onshape provides dashboards for a variety of categories of information.

Each user sees and has access to only the reports that the Global permissions grant access to. The main page lists all the dashboards available to you, depending on the Global permissions associated with your account.
Export any dashboard through the gear menu in the upper right. You can export as a PDF or a Single Column PDF. When viewing in a browser, be aware that the backend of the Analytics is run by Looker (Looker Data Sciences, Inc.).

All data is presented in dashboards with similar filtering, sorting, and refining capabilities. In general:

- The three dot icon (available on hover) on a data group indicates that you are able to download the data.
- Filter data by expanding the dropdown under the dashboard title. When filter criteria is selected, tap Run on the right of the section to run and display the new report:
- In tables:
  - Sort by almost any column; click the column name to sort by that column.
  - Act on the data in cells:
    - Click to open a menu of related dashboards; select one to display that dashboard
    - Click to open the referenced document in another tab
    - Click to open a detailed information panel
- In charts:
  - Hover to see detailed information
  - Click the hover information to display a menu of related dashboard; click to select one or to open the referenced document
  - Click an item (in a bar chart) or hover (in a pie chart) in the legend to toggle the display of that data (or the remaining data)
- In maps:
  - Use the zoom icons to zoom in and out.
  - Click and drag to reposition the map
  - Display information about a map point; click to display
In webs:

- Display summary information; hover over a hub point to display summary information. Click to reduce the web to information relating directly to the hub selected. Click again to revert to all data.

Audit Reports

This functionality is currently available only on browser.

Audit reports summarize the events occurring on a specific document, or for a specific user, in a specified timeframe.

Access Audit reports by:

1. Select Analytics from the title bar.
2. Select the Audit filter in the left pane.
3. Select the specific report in the list.

Audit reports include:

- **Activity Overview** - The overview of users' activity and the activity taken on documents.
- **Audit Trail** - Lists of all events happening on each specific document for a specific user in a specific timeframe.

**Activity overview**
Across the top of the dashboard are the statistics for all documents:

- **Active users** - The current number of active users for documents in the projects fulfilling the filter criteria
- **Created** - The number of documents created for the projects fulfilling the filter criteria
- **Versions** - The number of document versions for documents in the projects fulfilling the filter criteria
- **Releases** - The number of releases within the documents in the projects fulfilling the filter criteria
- **Shares** - The number of times the documents in the projects fulfilling the filter criteria have been shared
- **Imports** - The number of import operations in the projects fulfilling the filter criteria
- **Exports** - The number of export operations in the projects fulfilling the filter criteria
• **Drawings** - The number of drawings within the documents in the projects fulfilling the filter criteria

Below the statistics are individual reports, including:

• **Projects** - A list of all projects, including the name, the number of documents in each project, and the total modeling time to date.

• **Project activity** - A bar graph showing modeling activity by date and color-coded by project. Click a project name label to turn the information for that project on or off. Hover over a section of the graph for more information.

• **Most active documents** - A pie chart of percent of time spent in each document. Hover over a label or a section to dim the rest of the chart so that section stands out. Click to access actions like: open the document, access additional dashboards, see a list of all activity.

• **Release status** - A pie chart of percent of documents with any release status. Hover over a label or a section of the chart to dim the rest of the chart so that section stands out and see a status and the number of releases with that status. Click to access additional actions. Status shown include: Release, Rejected, and Pending.

• **Login locations** - The locations of all signins to Onshape. Click a locator on the map to see details. Click and drag the map to reorient and view more areas. Click the plus and minus signs to zoom in and out.

• **User activity** - A bar graph of time spent modeling by each user. Click a user name label to turn the information for that user on or off. Hover over a section of the graph for more information. Click on the graph to see more detailed information.

Audit trail
The Audit trail displays every event on a specific document or for a specific user for a specific timeframe, click on a column title to sort by that column:

- **Event time** - The time the event occurred.
- **Document name** - The name of the document.
- **Element** - The Onshape tab in the document: Part Studio, Assembly, etc.
- **User** - The name of the user involved in the event.
- **Description** - A general description of the event.

### Document Reports

This functionality is currently available only on browser.
Document reports provide information about relationships between projects, users and documents, lists of documents available in the enterprise and reports showing lists of permissions for a document or a user.

Access Document reports by:

1. Select **Analytics** from the title bar.
2. Select the **Documents** filter in the left pane.
3. Select the specific report in the list.

Document reports include:

- **Document Access** - A table of relationships between projects, users and documents. Documents are linked to the users who have access to the document.

- **Document List** - A list of documents created within the enterprise. Documents may be searched.

**Document access**
The key to the web is at the right side of the window. Click the double arrow to open the key and click again to close the key.
The web is interactive:

- Hover over a point in the web to obtain information about that point (document, user, project or collapsed project).
- Click a point to reduce the web to entities correlating with that point. Click again to extend the web again.
- Click and drag an entity for better visibility of other entities and to reorient the web.
- Right-click on an entity to access a context menu and commands to view additional dashboards for the entity, and to open the Onshape document that contains the entity.

Below the web is a list of all projects that fulfill the search criteria along with associated document name, user name, the permissions the user has on the document, and the time spent modeling in the document. Click the three dots to the right of any entry to access a list of dashboards to view.

**Document list**
The list contains all documents that fulfill the search criteria and includes information for:

- **Project name** - The name of the project for which other information is displayed.
- **Folder path** - The folder path of the document.
- **State** - The state of the document, which may include: Active, Deleted, or Trashed.
- **Last modified** - The date the documents was last modified.
- **Modified by** - The name of the last user to modify the document.
- **Created** - The date and time the document was created.
- **Created by** - The username of the person who created the document.
- **Users with access to the document** - The number of users how have access to the document.
- **Modeling time** - The total time spent modeling by all users with permissions on the document.
Projects Reports

This functionality is currently available only on browser.

The Project Dashboard report illustrates the events within a single project. Select a project to see project-specific data, including: total modeling time, number of documents in the project, file imported and modeling time for the project.

Access Project reports by:

1. Select Analytics from the title bar.
2. Select the Projects filter in the left pane.
3. Select the specific report in the list.

Project List

Lists of all projects in the Enterprise and their summary information. Use the filter to specify a particular Permission scheme for which to provide the information, including:
• Project name - The name of the project fulfilling the search criteria.

• State - The state of the document: Active or Deleted.

• Permission scheme - The name of the permission scheme that allows access to this project.

• Created by - The user who created the project.

• Created on - The date the project was created.

• Modified on - The date on which the project was last modified.

• Documents - The number of documents in the project.

• Users - The number of users with permissions to the document.

• Modeling time - The total amount of modeling time performed in the project (all users, on all documents).

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**Release Reports**

This functionality is currently available only on browser.

Release reports provide both summary and detail information about release activity and contributors to the release process. You are also able to search for release candidates and parts.

Access Release reports by:

1. Select **Analytics** from the title bar.

2. Select the **Release** filter in the left pane.
3. Select the specific report in the list.

Release Activity

Display a dashboard summary of all release activity plus a list of contributors and pending releases.

This report contains information about the releases fulfilling the search criteria, including:

- **Releases** - The number of releases (in progress, pending release, and released) fulfilling the search criteria.
- **Approved** - The number of releases that have been approved.
- **Rejected** - The number of releases that have been rejected.
- **Obsoleted** - The number of releases that have been obsoleted.
- **Status** - The percentage of releases in each category: Released, Pending, and Rejected.
- **Top Contributors** - A table of users and the number of times they have taken action in releases: submitted, approved, rejected, obsoleted, and discarded.
- **Pending for action** - A list of releases that have actions pending, which actions, and how long (in days) the action has been pending.

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Use the ellipsis (in the light gray oval) next to an entity name (for example, Part Number above), to access a menu that includes an option to access a dashboard specific to that entity. For example, next to Fastener Assembly in the Release column, as seen below.

The Release Candidate dashboard lists statistics about the release candidate, including the name of the approver. If the approver has delegated that responsibility to another user, that user is listed.

Release Audit

Search for release candidates, listing details about the object in the candidate. Use filters to narrow your search results.
Search for particular parts, or within particular releases to list parts and details, including:

- **Part number** - The part number assigned to the part, within the release.
- **Part name** - The name of the part within the release.
- **Revision** - The revision assigned to the part.
- **Type** - The Onshape type of the part, including: part, assembly, drawing, or file.
- **Workflow** - The name of the workflow as stated on the Accounts Release management page.
- **State** - The state of the part, including: released, in progress, pending, rejected, approved, obsoleted, or discarded.
- **Release** - The name assigned to the release.
- **Modified date** - The date the part was last modified.

Use the ellipsis (in the light gray oval) next to an entity name (for example, Part Number above), to access a menu that includes an option to access a dashboard specific to that entity.

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**Resource Reports**

This functionality is currently available only on browser.

Resource reports provide information on daily and detailed document and modeling activity.
Access Resource reports by:

1. Select **Analytics** from the title bar.

2. Select the **Resource** filter in the left pane.

3. Select the specific report in the list.

**Document Activity**

The time spent on a document for a given user. Use the filter to specify a project and date range to display information like:

- **Project** - The name of the project for which details are displayed.
- **Folder** - The name of the folder.
- **Document** - The name of the document.
- **Modeling date** - The date for which the information is displayed.
- **User** - The name of the user for which details are displayed.
- **Modeling time Linked** - The amount of time the named user spent modeling in the named document.
Modeling Activity

The Modeling Activity dashboard displays an overview of modeling activity across the entire company. Display activity by project, documents, and users for a particular duration as well as view trends over time, including:

- **Project activity** - The number of active users and the total modeling time per project. The (No Project) row contains information for all documents not in a project.

- **Project activity trend** - The amount of modeling time per project over time.

- **Active documents** - The percentage of time spent modeling for each active document.

- **Document activity trend** - The amount of time spent modeling in each document over time.

- **Active users** - The percent of time each user has spent modeling, showing up to 10 users.

- **User activity trend** - The amount of time each user has spent modeling over time.

As with all reports, you are able to drill into any data on the dashboard.
Users Reports

This functionality is currently available only on browser.

List all the users (including members, administrators, and guests) in the Enterprise and their summary information in these reports.
Access User reports by:

1. Select **Analytics** from the title bar.
2. Select the **Users** filter in the left pane.
3. Select the specific report in the list.

**Team List**

Display all the teams in the Enterprise and their summary information, including:

- Team name - The name of the team for which information is displayed.
- Created by - The user who created the team.
- Created on - The date the team was created.
- State - The state of the team: Active or Deleted.
- Members - The number of users assigned to the team.
- Shares - The number of times documents have been shared to the team directly or through a project. Click the number to open a list of the documents along with their permissions.
- Projects - The number of projects assigned to the team.
User List

Display the users in the Enterprise and their summary information, filtering on email address, admin status, guest status, to display information like:

- **User name** - The name of the user for whom the information is displayed.
- **Email address** - The user's email address.
- **Added on** - The date the user was added to the enterprise.
- **Is Admin** - Yes, the user is an administrator of the enterprise or no, the user is not.
- **User type** - The user designated type: Light or Full.
- **Documents** - The number of documents accessible to the user.
- **Projects** - The number of projects accessible to the user.
- **Releases** - The number of releases the user has created (in any state).
- **Modeling time** - The total amount of time the user has spent modeling.
For more information regarding a particular user, click the overflow menu button (three horizontal dots) to the right of their name in the list.

Access the User dashboard:
Access the Audit trail:
Public Documents and Custom Features

This functionality is currently available only on browser.

Enterprise admins and users can add custom features (and public custom features) to the toolbar in the Enterprise account and also access public documents. Enterprise admins can add public custom features to the toolbar for the entire enterprise. Users within the enterprise can add public custom feature easily once the enterprise admin has given them the "Allow access to public documents" global permission.

Public documents

Enterprise admins can make public documents accessible to their users through the "Allow access to public documents" global permission. Every user who is granted this global permission has access to public documents.

For users without rights to the "Allow access to public documents permission", the enterprise domain prevents them from making enterprise data public or directly accessing public documents. To bring a public document into your enterprise domain without the global permission, follow these steps:

2. Sign in with your Enterprise account information.
3. Browse the Public document account filter as you did previously and copy any documents of interest.
4. Follow the steps under Enterprise URL or cad.onshape.com to use those documents in your Enterprise.

FeatureScript

You may add custom features (written in Onshape's FeatureScript) to your toolbar in your enterprise domain. Enterprise admins may use this procedure to add public custom features to the toolbar for the entire enterprise.

1. Navigate to cad.onshape.com.
2. Sign in with your Enterprise account information.
3. Locate the custom feature you want to add. If the custom feature is within a public document, you must have been given permission to the "Allow access to public documents" by your enterprise admin, or have followed the instructions above to bring a public document into your enterprise domain.

4. Add the feature to your toolbar by clicking the + button on the toolbar.

5. In the User menu, select Switch to <enterprise domain name>.

When you access your enterprise domain the feature appears on your toolbar and will update as expected.

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**Enterprise Teams**

This functionality is available on Onshape's browser, iOS, and Android platforms.

Teams in Enterprises are defined and managed at the enterprise level only. Users need permission to create teams and assign other users to teams. Specifically, a user needs the Manager users global permission and the Enterprise admin global permission in order to create teams and assign other users to a team. In Professional and Standard subscriptions, by contrast, anyone is able to create teams.

**Enterprise Teams: Desktop**

Teams may be found under Enterprise settings on the User menu. At the creation of an Enterprise, all members of the enterprise are automatically added to the All enterprise users team.

**How teams work**

You are able to create teams in order to group users together for the purpose of sharing documents more efficiently; once the team is created, you are able to select the team name instead of entering many users' individual email addresses during a Share operation.
It is not required that the members of a team have anything in common, just that they are part of the same enterprise.

An admin creates a team (thereby becoming the initial administrator of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Team members receive notification emails when they are added and removed from a team, and users may belong to more than one team at a time.

Any user has the ability to share a document with a enterprise-owned team, even if they are not a member of the team. However, in order to share a document with a team external to the enterprise, the user must be a member of that team.

Sharing a document with a team does not give any team member additional permissions on the document other than those the owner/creator of the document allows during the Share operation.

At any point, the admins of a team may remove any member from the team, thereby removing any Share permissions previously made through the team (but not Share permissions made on an individual basis).

When a user is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions are able to be assigned during the Share operation:

- **Owner** - Full permission to the document including: Edit, Share, Comment, and Transfer ownership

- **Can edit** - Permission to edit the document and comment on it.

- **Can view** - Permission to only view the document (read-only).

Keep in mind that Edit and View permissions include Comment permission. Use the individual check boxes below the email address field to include any of the following permissions:
- **Copy** - Ability to make a copy of the document
- **Link document** - Ability to link to this document from another document (via inserting an assembly, part, image, drawing, etc)
- **Export** - Ability to export the document
- **Share** - Ability to reshare the document with another user
- **Comment** - Ability to make comments within the document

All permissions allow users to collaborate in the same workspace. A user with view only permission may be in the same workspace as a user with edit permission. The view only user is unable to edit the workspace but they are able to see any changes made to workspace in real time.

Following are instructions for:
- Creating teams and adding members
- Removing members and admins
- Deleting a team
- Additionally, see information about [Sharing and assigning permissions to documents](#)

**Creating teams and adding members**

1. Expand the menu under your user name in the top right corner of the page and select Enterprise settings:

2. On the page that appears, select Teams from the left panel and click Create team.
3. Enter a name for the team, and a description, or statement of purpose:

4. Click **Save**. The team is created, but there are no team members yet.

5. Click Add users.

6. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses) and click **Add**:

7. When finished adding team members and assigning roles, click Teams to the left of the specific team name you just created (at the top of the page) to return to the list of teams.

You see the new team listed on the Teams page.

Creating a team also adds a filter for that team in each member’s Documents filters on their Documents page. These filters list all documents shared with a particular team.
Each team is assigned a Team id that can be used in a customized release or obsolescence workflow. Edit the team details to see this id and copy it to the clipboard for use in a custom workflow. For more information on customizing release and obsolescence workflows, see "Creating a Customized Release Workflow" on page 2320.

Removing members and admins

Only Admins are able to remove themselves or a member from a team, as long as they are not the only admin user left on the team. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under the user name in the top-right corner of the page and select Enterprise settings:

2. Select Teams in the left panel to access the list of teams.

3. Select the team in the list from which you wish to remove yourself or another member.

4. Right-click on a user name and select Remove user.

Deleting a team

Any Admin of the team is able to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list. Right-click on the team name and select Delete.

Enterprise Teams: iOS

Teams in Enterprises are defined and managed at the enterprise level only. Users
need permission to create and assign other users to teams. Specifically, a user needs the Manage users and Enterprise admin global permissions in order to create teams and assign other users to a team. In Professional and Standard subscriptions, by contrast, anyone is able to create teams.

Find Teams under Enterprise settings on the User menu. At the creation of an Enterprise, all members of the enterprise are automatically added to the All enterprise users team.

**How teams work**

Create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you are able to select the team name instead of entering many users’ individual email addresses during a Share operation.

> It is not required that the members of a team have anything in common, just that they are part of the same enterprise.

An admin creates a team (thereby becoming the initial administrator of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Team members receive notification emails when they are added and removed from a team, and users are able to belong to more than one team at a time.

Sharing a document with a team does not give any team member additional permissions on the document other than those the owner/creator of the document allows during the Share operation.

At any point, the admins of a team is able to remove any member from the team, thereby removing any Share permissions previously made through the team (but not Shares made on an individual basis).

When a user is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions are able to be assigned during the Share operation:
• **Owner** - Full permission to the document including: Edit, Share, Comment, and Transfer ownership.

• **Can edit** - Permission to Edit the document and Comment on it.

• **Can view** - Permission to View the document only (read-only).

Keep in mind that Edit and View permissions include Comment permission. Use the individual check boxes below the email address field to include any of the following permissions:

• **Copy** - Ability to make a copy of the document

• **Link document** - Ability to link to this document from another document (via inserting an assembly, part, image, drawing, etc)

• **Export** - Ability to export the document

• **Share** - Ability to reshare the document with another user

• **Comment** - Ability to make comments within the document

All permissions allow users to collaborate in the same workspace. A user with view only permission is able to be in the same workspace as a user with edit permission. The view only user does not have the ability to edit the workspace but they are able to see any changes made to workspace in real time.

Following are instructions for:

• Creating teams and adding members

• Removing members and admins

• Deleting a team

• Additionally, see information about Sharing and assigning permissions to documents

**Creating teams and adding members**

1. Expand the menu under your user name in the top right corner of the page and select Enterprise settings:
2. On the page that appears, select Teams from the left panel and click Create team.

3. Enter a name for the team, and a description, or statement of purpose:

![Create team dialog](image)

4. Click Save. The team is created, but there are no team members yet.

5. Click Add users.

6. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses) and click Add:

![Add users dialog](image)
7. When finished adding team members and assigning roles, click Teams to the left of the specific team name you just created (at the top of the page) to return to the list of teams.

You see the new team listed on the Teams page.

Creating a team also adds a filter for that team in each member's Documents filters on their Documents page. These filters list all documents shared with a particular team.

**Removing members and admins**

Only Admins have the ability to remove themselves or a member from a team, as long as they are not the only admin user left on the team. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under the user name in the top-right corner of the page and select Enterprise settings:

   - My account
   - Enterprise settings
   - View support tickets

   Sign out

2. Select Teams in the left panel to access the list of teams.

3. Select the team in the list from which you wish to remove yourself or another member.

4. Right-click on a user name and select Remove user.

**Deleting a team**

Any Admin of the team has the ability to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list. Right-click on the team name and select Delete.

**Enterprise Teams: Android**
Teams in Enterprises are defined and managed at the enterprise level only. Users need permission to create and assign other users to teams. Specifically, a user needs the Manage users and Enterprise admin global permissions in order to create teams and assign other users to a team. In Professional and Standard subscriptions, by contrast, anyone is able to create teams.

Find Teams under Enterprise settings on the User menu. At the creation of an Enterprise, all members of the enterprise are automatically added to the All enterprise users team.

**How teams work**

Create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you have the option to select the team name instead of entering many users' individual email addresses during a Share operation.

It is not required that the members of a team have anything in common, just that they are part of the same enterprise.

An admin creates a team (thereby becoming the initial administrator of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Team members receive notification emails when they are added and removed from a team, and users are able to belong to more than one team at a time.

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**Can view** - Permission to View the document only (read-only).

Keep in mind that Edit and View permissions include Comment permission. Use the individual check boxes below the email address field to include any of the following permissions:

- **Copy** - Ability to make a copy of the document
- **Link document** - Ability to link to this document from another document (via inserting an assembly, part, image, drawing, etc)
- **Export** - Ability to export the document
- **Share** - Ability to reshare the document with another user
- **Comment** - Ability to make comments within the document

All permissions allow users to collaborate in the same workspace. A user with view only permission is able to be in the same workspace as a user with edit permission. The view only user does not have the ability to edit the workspace but they are able to see any changes made to workspace in real time.

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- Creating teams and adding members
- Removing members and admins
- Deleting a team
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7. When finished adding team members and assigning roles, click Teams to the left of the specific team name you just created (at the top of the page) to return to the list of teams.

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4. Right-click on a user name and select Remove user.

**Deleting a team**

Any Admin of the team has the ability to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list. Right-click on the team name and select Delete.
Managing Your Onshape Plan

This functionality is currently available only on browser.

Onshape offers different types of accounts based on different levels of need regarding data management and collaboration requirements. Each type of account is presented below.

- **Enterprise** - Onshape Enterprise is Onshape’s premium product offering designed for sophisticated product teams who need to work fast without losing control of data.

  Onshape Enterprise is a specific plan type available from Onshape which enables a company to purchase a plan for multiple users with consolidated billing. When you purchase Onshape Enterprise, you receive a unique domain where the users in your company are able to access Onshape. Your Onshape domain is specific to you and inaccessible by other unrelated Onshape users. Only the users specified and paid for by your company are allowed access to your enterprise.

  When working within an enterprise in Onshape, be aware that you are in a separate, managed environment that has a subtle difference from other types of Onshape subscriptions. The enterprise environment is owned and managed by an Enterprise Administrator and is able to include many users, called Members of the Enterprise.

  All users belonging to a specific enterprise subscription access Onshape through a single URL, specifically for their Enterprise. For more information about enterprises in Onshape, see "Enterprise" on page 2224.

- **Professional** - The Onshape Professional subscription enables a company to consolidate billing for multiple users, thereby creating a company within Onshape: a named, user-visible Onshape entity with consolidated billing, ownership and sharing for a set of Professional subscription users. Documents created by company members are owned by the company and all company-owned documents are automatically shared with all company members. The Professional company subscription differs from an Enterprise subscription in many ways, most notably that...
there is no unique domain for users to access, and no enterprise dashboard with analytics and activity reports.

If an existing Free user is listed as belonging to a Professional subscription, that user's plan is automatically upgraded to Professional and included in the company's subscription billing. Any Onshape user may be paid for and included in a Professional subscription, and even multiple Professional subscriptions.

All Professional subscriptions are billed per user (at an annual rate of $2,100) and include:

- Automated release management tools
- Custom properties for company metadata
- Company-wide material libraries
- Company-based sharing

**Standard** - Onshape Standard includes all of Onshape’s best-in-class parametric modeling functionality along with core data management tools. Standard subscriptions are billed per user (at an annual rate of $1,500), and include all Onshape features with the *exception* of the following:

- Automated release management tools
- Custom properties for company metadata
- Company-wide material libraries
- Company-based sharing
- Consolidated billing

For the addition of release management and company-wide controls, see Onshape Professional.

**Education** - Education subscriptions are for current faculty members, volunteers, or degree- or certificate-seeking students at accredited education institutions. Students must be at least 13 years of age. This plan is solely for classroom instruction, student learning projects, school clubs or organizations, and academic research. This plan is not for government, commercial, or other organizational use.
Education subscriptions allow the same working environment as the Standard subscription, and expire after one year of use resulting in automatic downgrade of the account. As long as the user still qualifies according to the criteria stated above, the Education subscription may be renewed. When the user no longer qualifies, the subscription must be downgraded to a Free subscription.

- **Free** - Onshape's Free subscription enables you to create an Onshape account and use Onshape at no cost. There is no time limit imposed and no credit card information collected.

The Free subscription allows you to create as many public documents as you want. You are unable to create any private documents. If a private document is shared with you, you are able to open it in View only mode (non-editable). If you attempt to create a private document, you are prompted to request a trial version of our Professional subscription in order to do so.

Trial versions are of our Professional subscription and features, including private documents. Using a trial version gives you a more realistic feel for all the features of Onshape, and includes the ability to more easily set up a company through a Professional subscription when you decide to subscribe to Onshape Professional.

Documents created in the free plan is able to be viewed and copied by all Onshape users, and there is no assumed copyright on any public document you create. You are also able to share a public document with specific users and give them edit rights.

Free subscribers may belong to only one Onshape subscription at a time (per email address). To change from a Free subscription to a Professional subscription, click [Upgrade to Professional](#) at the bottom of the left pane. If a free user is included in a Professional company subscription, their account is automatically upgraded to Professional.

All account maintenance is done through the browser/desktop platform. You are able to view some account information on mobile devices but you only have the ability to edit default dimension settings as well as turn on/off Touch and Face ID for mobile devices that support those methods of security. For all other account management, sign in to Onshape on a browser (not on a mobile device).

**Creating an account**
To create an Onshape account, open cad.onshape.com and click Sign up. Supply the required information and click Get Started. If you are an educator or student, you are directed to create a free account. You will receive an email with a link to setup your Onshape free account.

For all other users, you will see a button labeled 'Request Trial Account'. This enables Onshape to aid you in selecting the appropriate account type for you, while you evaluate Onshape at the Professional level with the ability to manage releases, create private data, and collaborate with other users.

For specific information regarding creating accounts for each type of account, see that topic (using the links above).

**Signing in to an account**

Once you create a free account, or request a trial account, you receive an email with a link to Onshape to verify your account and create your sign-in credentials. Once you have created your credentials, to sign in to Onshape at any time on a browser, open cad.onshape.com and use your credentials to sign in. The only exception is if you are part of an Enterprise subscription.

Enterprise accounts are created with a unique URL. Use the URL for your enterprise, supplied to your administrator by your Onshape representative. If you sign in without the URL to your enterprise domain, you sign in to a personal Professional account. Use the link in the User menu to "switch to" your enterprise account. You will be required to enter sign in credentials again.

**Signing in to an enterprise account on a mobile device**

The first time you sign into your Onshape enterprise account on a mobile device, you land in your individual account. To access your enterprise:

1. Tap the hamburger menu next to home in the upper left corner of the device screen.
2. In the menu that opens, select Switch to <enterprise menu>. A notification appears asking if you mean to sign out of Onshape and switch to your domain.
3. Tap OK. Your Enterprise URL is displayed
4. Tap Next.
5. Supply the email address and password associated with your Enterprise account.

6. Tap Sign in. Onshape will remember this state the next time you open the app. Once you are signed in to your Onshape enterprise account, the activity page is displayed. At the bottom of the page are three icon/links to the additional Onshape enterprise areas: Activity, Documents, and Analytics.

**Belonging to multiple accounts**

Users have the ability to belong to multiple accounts using one set of sign in credentials. When a user is a member of more than one Professional account, the companies they belong to (who are paying for their account) are listed in the Filter list on the left side of the Documents page, once signed in to Onshape. If you are part of an Enterprise, but also part of another Professional account or have your own personal account, when you sign in to Onshape through cad.onshape.com (versus the Enterprise domain), you are signing in to your non-Enterprise account. To access the Enterprise account, access the User menu and select ‘Switch to \<enterprise name\>’. You will not be prompted to sign in again, but will automatically access the enterprise.

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**Managing Your Free Onshape Subscription**

This functionality is available on Onshape's browser, iOS, and Android platforms.

All of the management and maintenance of your Onshape account can be done through the My account page, accessed through the User menu at the top right of the Onshape window. The icon for the user menu may look like this. You can also upload an image of your choosing to take the place of this icon.

The Account menu, located under your name in the upper-right corner of the interface, allows you to access:

- **My account** - Where you can manage and maintain your Onshape account, set preferences, notifications settings, security and more, as explained below.
**View support tickets** - View any support tickets you have submitted. If you would like to submit a support ticket, look in the Help menu (the icon to the right of your name in the upper-right corner of the interface).

**Sign out** - Sign out of and close your Onshape session.

**My account**

**Profile**

To navigate to your Onshape profile, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This automatically brings you to your Onshape user profile.

Onshape automatically records the first and last names you specify during sign up; here you may also enter a personal nickname for display in the system (in the upper right-hand corner of the user interface). Upload a photo to be used next to your user name and on comments, in the Share dialog, and generally wherever lists of user information exists.

**Username** is the name to be used as your Onshape forum name.

**Nickname** is the name seen by other users when you collaborate and is also displayed in the upper right corner of your Onshape window.
Click the Update profile button at the bottom of the page to save your changes.

**Update profile**

**Emails**

To navigate to your Emails settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your profile. From there, click Emails in the list on the left side of the page.

This brings you to your Emails settings.

Specify up to three email addresses with which to access your Onshape account. One address functions as your primary email, used for all Onshape notifications and communications. Change the primary designation at any time after adding at least one more email address to your account.

All email addresses added to the system must be verified. Check the email address for a verification notice from Onshape.

Any email address associated with an account (even those not designated as primary) may not be used to create another Onshape account.

Remove an email from your account by clicking the small "x" next to the email listing (shown above).

You may use any of the verified email addresses on your account to request a reset for a forgotten password.
Preferences
To navigate to your Preferences settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.
This brings you to your Onshape profile. From there, click Preferences in the left pane.
You can specify your preference for the following settings in Onshape.

Language
Select your preferred language from the dropdown. When you click Save language, Onshape automatically signs you out and you must sign in again to view the language change.

This is an ongoing effort; you may see terms that are not yet translated.

Units
The units of measurement and precision set here are used in all your Onshape documents, unless specifically overridden in a dialog (by entering units of choice) or by setting the default units per workspace through the workspace's Document menu > Workspace units. You can also select from a 12 hour or 24 hour time format here.
Onshape defaults to inch, degree, pound, and three decimal places for units of measure for all documents and encompasses all measurements in Part Studios, Assemblies, and Drawings; all values displayed in sketch dimensions as well as the default input units for all features.

The decimal place settings:

- Are currently available on browser only
- Are currently applied to the feature dialogs, sketch dimensions, and manipulator dialogs
- Work with the Measure tool and Mass properties tool
  - The Measure tool will display values in scientific notation when the display precision is not sufficient.
  - The Mass properties tool will display error in measurement; see "Mass Properties Tool" on page 349 for more information.
- Impacts the display only; values are rounded internally
- Are not used for computation
- Are used internally to determine the number of decimal places to display, regardless of how many places are entered; if more than the specified number are entered, they will be visible when the field is selected for edit.

- Do not affect any external files imported

**Overriding default settings**

In addition to setting default units for all documents you create (through this Preferences tab), you may also change and specify default units for a specific workspace in a document through the "Document Toolbar and Document Menu" on page 134 in a document.

Despite default settings, Onshape allows you to specify a different unit of measure in any numeric field; the value is converted to the default unit automatically. For example, if the default unit is inches, you may still specify a different unit type (for example "10mm") in a numeric field.

**Mouse controls**

You have the option to keep the default settings for mouse mappings, or select a familiar traditional CAD system's default settings. These settings also control mouse mappings for Drawings.
Onshape supports 3Dconnexion devices including the SpaceMouse. See your SpaceMouse instructions for information on how to set up your mouse with Onshape.

**Environment profile settings**

Create device profile preferences here, including rendering at high resolution pixel density. This profile may be used on any device.

Associate the profile and the browser/device by selecting a particular profile on a particular machine or browser through this interface. Select a profile for each machine and each browser used on a machine.

- **Match pixel density:**
  - **Automatic (default)** - Onshape determines the resolution needed for rendering.
  - **On** - Render at the resolution of the display.
  - **Off** - Do not render at the resolution of the display. Graphics will be rendered at a lower resolution.

Creating a profile:

1. Click **Create profile**.
2. Enter a name for the profile and click **Create**.
3. Select the preferred setting for matching pixel density.
4. Click **Save profile settings**.

Deleting a profile:

1. Select the profile from the dropdown.
2. Click **Delete profile**.
   
   Note that this action may not be undone.
3. Click **OK** to confirm the deletion, or **Cancel**.

**Shortcut toolbars**
Customize the shortcut toolbars available for Sketch tools, Feature tools, Assembly tools, and Drawing tools. Select the toolbar to customize; check the tools to include and uncheck the tools to exclude from the menu. If you do not customize the toolbar, each time Onshape is updated, the default toolbar may change. Once customized, your customizations take precedence over any defaults.

There are no limits to the number of tools you are able to include.

- The order of tools in the toolbar is determined by the order in the list (currently).
- Use the S key to invoke the toolbar; use the Esc key to close the toolbar. The toolbar appears near the mouse pointer.

Toolbars

Click **Reset to defaults** to restore toolbars to the default settings.

**Drawings**

Set background color of model space for imported DWG and DXF files.

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Material libraries

Create and add custom material libraries, remove unnecessary libraries, and make libraries available to all users within a company. For more detailed information, see “Customizing Parts: Materials” on page 322.

Notifications

This functionality is also available on Onshape’s iOS and Android platforms.

To navigate to your Notifications settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Notifications in the list on the left side of the page.

This brings you to your Onshape Notifications settings, where you have the option to edit details such as your first name, last name, username, nickname, and biography.

In your Notifications settings, there are two sections: Email notifications for shares and comments, and Mobile notifications for shares and comments. Both sections have various radio buttons you may or may not select in order to make notification changes, and a Save notifications button to save your changes.

Email notifications for shares and comments

The top section of your Onshape Notifications page is where you make changes to your Email notifications for shares and comments. In this section, you have three options to choose from:

- **All new shares and comments** - When selected, you will receive email notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive email notifications for shares directly to you, comments mentioning you, and documents you have specifically marked to receive comment emails.
Nothing - When selected, you will not receive any email notifications for shares or comments.

Mobile notifications for shares and comments

The bottom section of your Onshape Notifications page is where you make changes for receiving notifications on mobile devices for shares and comments. Similar to the Email notifications for shares and comments section, in this section you have three options to choose from:

- **All new shares and comments** - When selected, you will receive mobile notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive mobile notifications for shares directly to you, comments mentioning you and documents you have specifically marked to receive comment emails.

- **Nothing** - When selected, you will not receive any mobile notifications for shares or comments.

Security

If you have forgotten your password and need it reset, proceed to the Onshape sign in page and click the "Forgot your password?" link to access a page on which you are able to request a password reset link via email.
To navigate to your Security settings, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Security in the list on the left side of the page.

This brings you to your Onshape Security settings. Change your Onshape system password, and also enable (or disable) two-factor authentication.

**Resetting password**

1. Expand the menu under your user name and select My account:
2. Select the Security tab.

3. Click Change password and enter the old password, the new password, and re-enter the new password.

   The list of guidelines leads you through creating a password. Each requirement is marked when your password fulfills the requirement.

4. Click Update password.

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**Two-Factor Authentication**

Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.
How it works

Download a two-factor authentication app (like Google Authenticator) to your phone and set it up with Onshape through the Onshape user interface. This enables the app to generate a one-time code that Onshape is able to recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You are able to allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign in to Onshape from a specific device, you won't need a 2FA code to sign in on that device for 30 days.

Enabling and using two-factor authentication

1. Download a two-factor authentication app to your device.
   
   Google Authenticator is one example.

2. Sign in to your Onshape account.

3. In the menu under your username, select My account.

4. In the list on the left side of the page, click Security.

5. To the right of Two-factor Authentication, click Enable.

6. Click Set up two-factor authentication.

7. Confirm password.

8. Click OK.

Configuring the app to work with Onshape

Continuing from the instructions above:

1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface.

   Once registration is complete, the phone app will list a code for each registration you create. It is these codes that you enter into Onshape when presented with the 2FA sign in page.
If you are not able to use the QR code, click the enter this text code link provided in the Onshape interface to obtain a code.

2. Enter either the six-digit code that the 2FA app generates or the code supplied by Onshape.

3. Click Enable.

4. When the recovery codes are displayed, copy them to a safe place; you need access to them in the event you do not have your phone or the authentication app.

5. Click OK.

Onshape provides you with 5 active recovery codes at a time. Keep these codes in a place accessible to you separate from your device or the authentication app.

Onshape will not be able to help you should you delete the app or lose your phone.

Note that you are able to generate these Recovery codes at any time through the Onshape interface, but only the most recently generated series are active at any one time. Once you use a code it is no longer valid. When you generate a new list of codes, all previous codes (used or unused) become invalid.

**Signing in to Onshape with code**

When two-factor authentication is enabled, Onshape prompts you for a code upon sign in:

1. After you enter the password to your Onshape account, you are prompted for the authentication code.

2. Open the two-factor authentication app on your device to view the code; enter the code in Onshape.
3. Click Verify.

In the event that you don't have access to the app, click the Enter a two-factor recovery code link to enter one of your current recovery codes.

Disabling two-factor authentication in Onshape
You may disable (and re-enable) two-factor authentication at any time.

1. On the Security tab of the User Profile page in Onshape click Manage, and then Disable:

2. Confirm password.

3. Click OK.

Replacing a device with 2FA enabled
Should you need to replace a device on which you have 2FA enabled for Onshape:

1. Before replacing the device, disable 2FA through the Onshape interface.

2. Enable 2FA once the new device is online.

Note that Onshape doesn’t support the Replace 2FA option.

Devices
To navigate to your Devices settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Devices in the list on the left side of the page.

This brings you to your Onshape Devices settings, a list of all mobile devices associated with and authorized to use this account. Once you access your Onshape account on a mobile device, that mobile device is listed here.
To remove a device from the list, click Forget on the right of the window.

**Applications**

To navigate to your Onshape Applications settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there click My account.

This brings you to your Onshape profile. From there, click Applications in the list on the left side of the page.

This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store, navigate to [http://appstore.onshape.com](http://appstore.onshape.com) and sign in with your Onshape account credentials.

Here’s a list of frequently asked questions ("App Store FAQs" on page 2613).

Once signed in to the App store, you are able to browse the available apps and make purchases. As the owner of a company or enterprise, you may purchase multiple seats for other users in your company or enterprise and assign users to those seats.

**Types of apps**

Onshape third-party apps are of the following types:

- **Integrated Cloud Application** - Accessible from within an Onshape document
- **Connected Desktop Application** - Downloaded from the third-party website and installed on your physical machine
- **Connected Cloud Application** - Accessible from a cloud-based service

After purchasing an app, you may have to grant it access to your Onshape documents.
1. While in a document, click the and select Add application. (Select the desired application.)

2. Review the permissions you are about to authorize, then click Authorize application (or Deny if you no longer wish to use the app).

3. To view the permissions an application has on a document, or to give the application access to another document:
   a. Click to open the Share dialog and settings for that document.
   b. On the Application tab, select the application from the drop down and click Allow.

4. To revoke an application's access, click the x next to the application name at the top of the dialog.

Purchased apps that are authorized to access your Onshape data are listed in three places in your Onshape documents:

- Applications tab in the user profile (My account page) - Shows all apps you have authorized to access your Onshape data.
- Subscriptions tab in the user profile - Shows all apps for which you have a subscription.
- On the Add application command from the menu at the bottom of your Onshape window.

**Purchasing and managing multiple seats for an application**

An owner of a company or enterprise may purchase multiple seats for an application, to make available to users of their organization. An owner may add and remove users at any time for any given application.

**Adding seats during purchase**

To add seats while purchasing an application:

1. Click the app button to purchase or subscribe, as you normally would, in the App store.

2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).
3. Click the purchase button.
4. Enter your Onshape password.
5. A confirmation dialog appears, providing more information about your purchase.
6. Click Close to dismiss the dialog.

Adding seats and managing users after purchase

Once you purchase multiple seats for an app, you can manage the users who are allowed to use that app. You can change users who may use the app and change the number of seats.

1. On the Applications tab of the Company settings page, select the app you wish to manage.
2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.
   
   A confirmation dialog appears and your changes take effect immediately.
3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the company.
4. To remove a user's access to the application, click the X next to their name in the list.

User actions on apps

Through the My account area and Applications tab, users are able to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the $+$ icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.
Control application access to my documents individually through the Share dialog? - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.

If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the Share dialog to allow a specific application access to a specific document.

**Integrations**

To navigate to your Onshape Integrations settings to set up access to your Dropbox, Google Drive, or Microsoft OneDrive account, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Integrations in the list on the left side of the page.

This brings you to your Onshape cloud-source settings. You have the ability to grant Onshape access to your Dropbox, Google Drive, or Microsoft OneDrive account (separately), and also remove that access by clicking the red Remove button on the right side of the page.
To grant access for Onshape to import documents from (and export documents to) a Dropbox, Google Drive, or Microsoft OneDrive account, click the appropriate "Add" button and follow the steps. You'll need the credentials to your account in order to proceed through the wizard. For example:

To continue, Google will share your name, email address, and profile picture with Onshape - Connector. Before using this app, you can review Onshape - Connector's privacy policy and terms of service.

**Early Visibility**

To navigate to your Onshape Early visibility settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your Onshape profile. From there, click Early visibility in the list on the left side of the page.

This brings you to your Onshape Early visibility settings. The Onshape Early Visibility program offers you an opportunity to test functionality that is still under development. Due to the nature of features in development, we recommend you create specific documents for use with any Early Visibility feature. (Feel free to copy existing documents for this purpose.)

Please do not use documents you create under the Early Visibility feature for business critical or production use.

**Viewing programs**

Find the Early Visibility program sign up page under My account:

Once in the Account management area, select the Early visibility tab.

**Signing up**

1. Click to the right of the feature of interest (or features; you are able to request access to multiple features).
   
   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click if you agree and wish to continue.
   
   Click if you are no longer interested.
Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

To navigate to your Onshape Early visibility settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:

```
My account
Rachel’s Company company settings
View support tickets
Switch to Fishbowl
Sign out
```

This brings you to your Onshape profile. From there, click Early visibility in the list on the left side of the page:

```
Profile
Emails
Preferences
Notifications
Security
Devices
Applications
Early visibility
Subscriptions
Payment options
Teams
```

This brings you to your Onshape Early visibility settings. The Onshape Early Visibility program offers you an opportunity to test functionality that is still under development. Due to the nature of features in development, we recommend you create specific documents for use with any Early Visibility feature. (Feel free to copy existing documents for this purpose.)
Please do not use documents you create under the Early Visibility feature for business critical or production use.

**Viewing programs**

Find the Early Visibility program sign up page under My account:

Once in the Account management area, select the Early visibility tab.

**Signing up**

1. Click **Add** to the right of the feature of interest (or features; you are able to request access to multiple features).
   
   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click **Accept** if you agree and wish to continue.
   
   Click **Cancel** if you are no longer interested.

Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

**Teams**

To navigate to your Onshape Teams settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:
This brings you to your Onshape profile. From there, click Teams in the list on the left side of the page:

This brings you to your Onshape Teams settings. Create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you have the option to select the team name instead of entering many users' individual email addresses during a Share operation.

It is not required that the members of a team have anything in common; not even an Onshape subscription.

One user creates a team (thereby becoming the initial admin of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Members receive notification emails when they are added and removed from a team, and users are able to belong to more than one team at a time.
Sharing a document with a team does not give any team member additional permissions on the document than the owner/creator of the document allows during the Share operation.

At any point, the admins of a team is able to remove any member from the team, thereby removing any Share permissions previously made through the team. Any Shares made on an individual basis remain in place, as well as the permission they grant.

Team members are able to remove themselves from a team, unless they are the last admin member of the team. (A team must have at least one admin.) When a member is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions are able to be assigned during the Share operation:

- **View** - Permission to open for read-only access; no editing allowed
- **Edit** - Permission to open and edit (make changes)

Following are instructions for:

- Creating teams and adding members
- Removing members and admins
- Deleting a team
- Additionally, see information about [Sharing and assigning permissions to documents](#)

### Creating teams and adding members

1. Expand the menu under your user name in the top right corner of the page and select My account.
2. On the page that appears, select Teams from the left panel and click Create:

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Members</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assembly</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>Body</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>Modeling</td>
<td></td>
<td>1</td>
</tr>
</tbody>
</table>

3. Enter a name for the team, and a description, or statement of purpose:
4. Click Create team:

![Create team interface](image)

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).

   Use the Search bar to search for team members.  
   Note that you are able to return to this page and change a team member's role.

6. Click Add.

7. When finished adding team members and assigning roles, click the arrow to the left of the team name (at the top of the page) to return to the Accounts page.

8. You see the new team listed on the Teams page.
   
   Creating a team also adds a filter for that team in each member's Documents filters on their Documents page. These filters list all documents shared with a particular team.
Removing members and admins

Members are able to remove themselves from a team, and any member with an Admin role is able to remove users including themselves as long as they are not the only administrative user left. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under your user name in the top right corner of the page and select My account (for Standard and Free subscriptions) or Company settings (for Professional and Enterprise subscriptions).

2. Select Teams in the left panel to access the list of teams of which you are a member.

3. Select the team in the list from which you wish to remove yourself or another member.
   - To remove yourself (as a member): Click the Leave team button.
   - To remove yourself (as an admin): Click the X to the left of your name (Note this only works if there is another admin still on the team).
   - To remove another user: Click the X to the left of the user name (Note this only works if you are an admin).

Deleting a team

Any Admin of the team has the ability to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list.

Payment Options

To navigate to your Onshape Payment options settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:
This brings you to your Onshape profile. From there, click Payment options in the list on the left side of the page:

This brings you to your Onshape Payment options settings. If you are the owner of the account, you have the ability to change credit card information, or remove a card from the listing. Note that a credit card may be removed from the account only if it is not associated with a subscription:

To change the credit card, click Change and enter the new card information.
Upgrading to a Standard Account

To upgrade to a Standard subscription from a Free or Education plan, click one of the "Upgrade" buttons on the Documents page or the My account page (not the Try Professional button).

You will be directed to a new tab with Onshape plans explained. Click the "Select Standard" button. Fill out the form and click "Confirm payment information", then "Review purchase". Confirm your purchase.

Upgrading to a Professional Account

Onshape's Professional subscription allows you to create unlimited private documents. The Professional subscription is a company subscription (you are able to pay for one or multiple users).

- If you are new to Onshape and do not yet have an account, click the Sign up link on the Onshape home page and follow the instructions.
- If you already have an Onshape account, click the Upgrade button on your account page and follow the instructions below (or navigate to http://cad.onshape.com/upgrade (opens in new tab)).

When upgrading from a Free to a Professional subscription, any View only documents you previously saw on the Documents page are now editable documents. Onshape automatically makes all of your documents accessible to you.

Deleting Your Free Account

Users with Free Onshape accounts have the ability to delete their account, removing all personal data as well as document data from Onshape on their own, without the need to contact Onshape. If you have another type of Onshape account, you must first downgrade to Free before you can delete your account.

If you choose to remove your Onshape account, expect the following:

- Your account will be deleted within 30 days and during this time your account and the documents therein will be inaccessible.
Once the request is completely processed, your account and all data will no longer be recoverable. Documents owned by you or your company (if applicable) will no longer be available to you or anyone those documents were shared with.

- Any existing links to (internal to a document, such as a linked part) and any copies of these documents will remain active.

To delete your Onshape account:

1. Proceed to the **My account** page, **Profile** tab.

2. At the bottom of the window, click **Delete my Onshape account**:
3. Read the modal window carefully, it explains what actions will be taken on your behalf and what will happen to your documents:
4. When you understand the consequences of your actions, enter “delete my account” and then your password:
5. The **Delete my account button** becomes active. Click it.
A summary of events is displayed. We recommend you take a screenshot of this information for future reference as it explains what actions are taken while we delete your account, and as well as what happens to your documents.

Managing Your Onshape Education Standard Subscription

This functionality is available on Onshape's browser, iOS, and Android platforms.

Education subscriptions are for current faculty members, volunteers, or degree- or certificate-seeking students at accredited education institutions. Students must be at least 13 years of age. This plan is solely for classroom instruction, student learning projects, school clubs or organizations, and academic research. This plan is not for government, commercial, or other organizational use.
Education subscriptions allow the same working environment as the Standard subscription, but expire after one year of use. As long as the user still qualifies according to the criteria stated above, the Education subscription may be renewed. When the user no longer qualifies, the subscription must be downgraded to a Free subscription.

The Account menu, located under your name in the upper-right corner of the interface, allows you to access:

- **My account** - Where you can manage and maintain your Onshape account, set preferences, notifications settings, security and more, as explained below.
- **Action items** - View any tasks that have been assigned to you through the Comments dialog. More on tasks, below.
- **View support tickets** - View any support tickets you have submitted. If you would like to submit a support ticket, look in the Help menu (the icon to the right of your name in the upper-right corner of the interface).
- **Sign out** - Sign out of and close your Onshape session.

**Action items**

When you select Action items from the account menu, you see page with the following filters on the left:

- **Role** - Choose between filtering through tasks assigned to you or created by you by selecting one of the radio buttons to the left of the filter options.
- **Status** - Choose between filtering through open or closed tasks by selecting one of the radio buttons to the left of the filter options.
- **Sort** - Sort your filtered results by Oldest tasks first or Newest first by clicking the dropdown arrow under Sort and selecting your preference (Onshape defaults to sorting by Oldest first).

As you select different filters, the results for the corresponding selections will appear in the Action Items list located in the center of the page.

**My account**

**Profile**

To navigate to your Onshape profile, click your Account user icon in the top right
corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This automatically brings you to your Onshape user profile.

Onshape automatically records the first and last names you specify during sign up; here you may also enter a personal nickname for display in the system (in the upper right-hand corner of the user interface). Upload a photo to be used next to your user name and on comments, in the Share dialog, and generally wherever lists of user information exists.

**Username** is the name to be used as your Onshape forum name.

**Nickname** is the name seen by other users when you collaborate and is also displayed in the upper right corner of your Onshape window.

Click the Update profile button at the bottom of the page to save your changes.

**Emails**

To navigate to your Emails settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your profile. From there, click Emails in the list on the left side of the page.

This brings you to your Emails settings.

Specify up to three email addresses with which to access your Onshape account. One address functions as your primary email, used for all Onshape notifications and communications. Change the primary designation at any time after adding at least one more email address to your account.

All email addresses added to the system must be verified. Check the email address for a verification notice from Onshape.

Any email address associated with an account (even those not designated as primary) may not be used to create another Onshape account.

Remove an email from your account by clicking the small "x" next to the email listing (shown above).

You may use any of the verified email addresses on your account to request a reset for a forgotten password.

Preferences

To navigate to your Preferences settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Preferences in the left pane.

You can specify your preference for the following settings in Onshape.

Language
Select your preferred language from the dropdown. When you click Save language, Onshape automatically signs you out and you must sign in again to view the language change.

This is an ongoing effort; you may see terms that are not yet translated.

**Units**

The units of measurement and precision set here are used in all your Onshape documents, unless specifically overridden in a dialog (by entering units of choice) or by setting the default units per workspace through the workspace's Document menu > Workspace units. You can also select from a 12 hour or 24 hour time format here.

Onshape defaults to inch, degree, pound, and three decimal places for units of measure for all documents and encompasses all measurements in Part Studios,
Assemblies, and Drawings; all values displayed in sketch dimensions as well as the default input units for all features.

The decimal place settings:

- Are currently available on browser only
- Are currently applied to the feature dialogs, sketch dimensions, and manipulator dialogs
- Work with the Measure tool and Mass properties tool
  - The Measure tool will display values in scientific notation when the display precision is not sufficient.
  - The Mass properties tool will display error in measurement; see "Mass Properties Tool" on page 349 for more information.
- Impacts the display only; values are rounded internally
- Are not used for computation
- Are used internally to determine the number of decimal places to display, regardless of how many places are entered; if more than the specified number are entered, they will be visible when the field is selected for edit.
- Do not affect any external files imported

Overriding default settings

In addition to setting default units for all documents you create (through this Preferences tab), you may also change and specify default units for a specific workspace in a document through the "Document Toolbar and Document Menu" on page 134 in a document.

Despite default settings, Onshape allows you to specify a different unit of measure in any numeric field; the value is converted to the default unit automatically. For example, if the default unit is inches, you may still specify a different unit type (for example "10mm") in a numeric field.

Mouse controls

You have the option to keep the default settings for mouse mappings, or select a familiar traditional CAD system’s default settings. These settings also control mouse mappings for Drawings.
Onshape supports 3Dconnexion devices including the SpaceMouse. See your SpaceMouse instructions for information on how to set up your mouse with Onshape.

**Environment profile settings**

Create device profile preferences here, including rendering at high resolution pixel density. This profile may be used on any device.

Associate the profile and the browser/device by selecting a particular profile on a particular machine or browser through this interface. Select a profile for each machine and each browser used on a machine.

- **Match pixel density:**
  - **Automatic (default)** - Onshape determines the resolution needed for rendering.
  - **On** - Render at the resolution of the display.
  - **Off** - Do not render at the resolution of the display. Graphics will be rendered at a lower resolution.

Creating a profile:

1. Click **Create profile**.
2. Enter a name for the profile and click **Create**.

3. Select the preferred setting for matching pixel density.

4. Click **Save profile settings**.

Deleting a profile:

1. Select the profile from the dropdown.

2. Click **Delete profile**.

   Note that this action may not be undone.

3. Click **OK** to confirm the deletion, or **Cancel**.

**Shortcut toolbars**

Customize the shortcut toolbars available for Sketch tools, Feature tools, Assembly tools, and Drawing tools. Select the toolbar to customize; check the tools to include and uncheck the tools to exclude from the menu. If you do not customize the toolbar, each time Onshape is updated, the default toolbar may change. Once customized, your customizations take precedence over any defaults.

- There are no limits to the number of tools you are able to include.
- The order of tools in the toolbar is determined by the order in the list (currently).
Use the S key to invoke the toolbar; use the Esc key to close the toolbar. The toolbar appears near the mouse pointer.

Toolbars

Click \[\text{Reset to defaults}\] to restore toolbars to the default settings.

Drawings

Set background color of model space for imported DWG and DXF files.

<table>
<thead>
<tr>
<th>Drawings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Background color of model space (imported DWG and DXF files)</td>
</tr>
<tr>
<td>Dark (default)</td>
</tr>
<tr>
<td>[\text{Save drawing settings}]</td>
</tr>
</tbody>
</table>

Material libraries

Create and add custom material libraries, remove unnecessary libraries, and make libraries available to all users within a company. For more detailed information, see "Customizing Parts: Materials" on page 322

Notifications

This functionality is also available on Onshape's iOS and Android platforms.

To navigate to your Notifications settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Notifications in the list on the left side of the page.

This brings you to your Onshape Notifications settings, where you have the option to edit details such as your first name, last name, username, nickname, and biography.
In your Notifications settings, there are two sections: Email notifications for shares and comments, and Mobile notifications for shares and comments. Both sections have various radio buttons you may or may not select in order to make notification changes, and a Save notifications button to save your changes.

**Email notifications for shares and comments**

The top section of your Onshape Notifications page is where you make changes to your Email notifications for shares and comments. In this section, you have three options to choose from:

- **All new shares and comments** - When selected, you will receive email notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive email notifications for shares directly to you, comments mentioning you, and documents you have specifically marked to receive comment emails.

- **Nothing** - When selected, you will not receive any email notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save email notifications button at the bottom of the section, when you have finished editing your Notifications, to save your changes.

**Mobile notifications for shares and comments**

The bottom section of your Onshape Notifications page is where you make changes for receiving notifications on mobile devices for shares and comments. Similar to the Email notifications for shares and comments section, in this section you have three options to choose from:

- **All new shares and comments** - When selected, you will receive mobile notifications for all new shares, including to your teams or companies, and comments in documents you have access to.
• **Direct shares, mentions, and marked documents** - When selected, you will receive mobile notifications for shares directly to you, comments mentioning you and documents you have specifically marked to receive comment emails.

• **Nothing** - When selected, you will not receive any mobile notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save mobile notifications button at the bottom of the section when you have finished editing your Notifications to save your changes.

**Security**

If you have forgotten your password and need it reset, proceed to the Onshape sign in page and click the "Forgot your password?" link to access a page on which you are able to request a password reset link via email.
To navigate to your Security settings, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Security in the list on the left side of the page.

This brings you to your Onshape Security settings. Change your Onshape system password, and also enable (or disable) two-factor authentication.

### Resetting password

1. Expand the menu under your user name and select My account:

2. Select the Security tab.

3. Click Change password and enter the old password, the new password, and re-enter the new password.

   The list of guidelines leads you through creating a password. Each requirement is marked when your password fulfills the requirement.

4. Click...
Two-Factor Authentication

Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.

How it works

Download a two-factor authentication app (like Google Authenticator) to your phone and set it up with Onshape through the Onshape user interface. This enables the app to generate a one-time code that Onshape is able to recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You are able to allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign in to Onshape from a specific device, you won’t need a 2FA code to sign in on that device for 30 days.

Enabling and using two-factor authentication

1. Download a two-factor authentication app to your device.

   Google Authenticator is one example.

2. Sign in to your Onshape account.
3. In the menu under your username, select **My account**.

4. In the list on the left side of the page, click **Security**.

5. To the right of Two-factor Authentication, click **Enable**.

6. Click **Set up two-factor authentication**.

7. Confirm password.

8. Click **OK**.

**Configuring the app to work with Onshape**

Continuing from the instructions above:

1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface.

   Once registration is complete, the phone app will list a code for each registration you create. It is these codes that you enter into Onshape when presented with the 2FA sign in page.

   If you are not able to use the QR code, click the enter this text code link provided in the Onshape interface to obtain a code.

2. Enter either the six-digit code that the 2FA app generates or the code supplied by Onshape.

3. Click Enable.

4. When the recovery codes are displayed, copy them to a safe place; you need access to them in the event you do not have your phone or the authentication app.

5. Click **OK**.

   Onshape provides you with 5 active recovery codes at a time. Keep these codes in a place accessible to you separate from your device or the authentication app.

   **Onshape will not be able to help you should you delete the app or lose your**
phone.

Note that you are able to generate these Recovery codes at any time through the Onshape interface, but only the most recently generated series are active at any one time. Once you use a code it is no longer valid. When you generate a new list of codes, all previous codes (used or unused) become invalid.

**Signing in to Onshape with code**

When two-factor authentication is enabled, Onshape prompts you for a code upon sign in:

1. After you enter the password to your Onshape account, you are prompted for the authentication code.

2. Open the two-factor authentication app on your device to view the code; enter the code in Onshape.

3. Click Verify.

   In the event that you don’t have access to the app, click the **Enter a two-factor recovery code** link to enter one of your current recovery codes.

**Disabling two-factor authentication in Onshape**

You may disable (and re-enable) two-factor authentication at any time.

1. On the Security tab of the User Profile page in Onshape click Manage, and then Disable:

2. Confirm password.

3. Click OK.

**Replacing a device with 2FA enabled**

Should you need to replace a device on which you have 2FA enabled for Onshape:

1. Before replacing the device, disable 2FA through the Onshape interface.

2. Enable 2FA once the new device is online.

Note that Onshape doesn’t support the Replace 2FA option.
Devices

To navigate to your Devices settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:

This brings you to your Onshape profile. From there, click Devices in the list on the left side of the page:

This brings you to your Onshape Devices settings, a list of all mobile devices associated with and authorized to use this account. Once you access your Onshape account on a mobile device, that mobile device is listed here.
To remove a device from the list, click Forget on the right of the window.

**Applications**

To navigate to your Onshape Applications settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there click My account.

This brings you to your Onshape profile. From there, click Applications in the list on the left side of the page.

This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store, navigate to [http://appstore.onshape.com](http://appstore.onshape.com) and sign in with your Onshape account credentials.

Here's a list of frequently asked questions ("App Store FAQs" on page 2613).

Once signed in to the App store, you are able to browse the available apps and make purchases. As the owner of a company or enterprise, you may purchase multiple seats for other users in your company or enterprise and assign users to those seats.

**Types of apps**

Onshape third-party apps are of the following types:

- **Integrated Cloud Application** - Accessible from within an Onshape document
- **Connected Desktop Application** - Downloaded from the third-party website and installed on your physical machine
- **Connected Cloud Application** - Accessible from a cloud-based service

After purchasing an app, you may have to grant it access to your Onshape documents.
1. While in a document, click the and select Add application. (Select the desired application.)

2. Review the permissions you are about to authorize, then click Authorize application (or Deny if you no longer wish to use the app).

3. To view the permissions an application has on a document, or to give the application access to another document:
   a. Click to open the Share dialog and settings for that document.
   b. On the Application tab, select the application from the drop down and click Allow.

4. To revoke an application's access, click the x next to the application name at the top of the dialog.

Purchased apps that are authorized to access your Onshape data are listed in three places in your Onshape documents:
- Applications tab in the user profile (My account page) - Shows all apps you have authorized to access your Onshape data.
- Subscriptions tab in the user profile - Shows all apps for which you have a subscription.
- On the Add application command from the menu at the bottom of your Onshape window.

**Purchasing and managing multiple seats for an application**

An owner of a company or enterprise may purchase multiple seats for an application, to make available to users of their organization. An owner may add and remove users at any time for any given application.

Adding seats during purchase

To add seats while purchasing an application:

1. Click the app button to purchase or subscribe, as you normally would, in the App store.
2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).
3. Click the purchase button.
4. Enter your Onshape password.
5. A confirmation dialog appears, providing more information about your purchase.
6. Click Close to dismiss the dialog.

Adding seats and managing users after purchase

Once you purchase multiple seats for an app, you can manage the users who are allowed to use that app. You can change users who may use the app and change the number of seats.

1. On the Applications tab of the Company settings page, select the app you wish to manage.
2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.
   
   A confirmation dialog appears and your changes take effect immediately.
3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the company.
4. To remove a user's access to the application, click the X next to their name in the list.

User actions on apps

Through the My account area and Applications tab, users are able to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.
Control application access to my documents individually through the Share dialog? - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.

If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the Share dialog to allow a specific application access to a specific document.

Integrations
To navigate to your Onshape Integrations settings to set up access to your Dropbox, Google Drive, or Microsoft OneDrive account, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Integrations in the list on the left side of the page.

This brings you to your Onshape cloud-source settings. You have the ability to grant Onshape access to your Dropbox, Google Drive, or Microsoft OneDrive account (separately), and also remove that access by clicking the red Remove button on the right side of the page.
To grant access for Onshape to import documents from (and export documents to) a Dropbox, Google Drive, or Microsoft OneDrive account, click the appropriate "Add" button and follow the steps. You'll need the credentials to your account in order to proceed through the wizard. For example:

Choose an account to continue to Onshape - Connector

Diane Amadeo
diana@onshape.com

Use another account

To continue, Google will share your name, email address, and profile picture with Onshape - Connector. Before using this app, you can review Onshape - Connector's privacy policy and terms of service.

Early Visibility

To navigate to your Onshape Early visibility settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your Onshape profile. From there, click Early visibility in the list on the left side of the page.

This brings you to your Onshape Early visibility settings. The Onshape Early Visibility program offers you an opportunity to test functionality that is still under development. Due to the nature of features in development, we recommend you create specific documents for use with any Early Visibility feature. (Feel free to copy existing documents for this purpose.)

**Please do not use documents you create under the Early Visibility feature for business critical or production use.**

**Viewing programs**

Find the Early Visibility program sign up page under My account:

Once in the Account management area, select the Early visibility tab.

**Signing up**

1. Click to the right of the feature of interest (or features; you are able to request access to multiple features).

   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click if you agree and wish to continue.

   Click if you are no longer interested.
Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

**Viewing programs**

Find the Early Visibility program sign up page under My account:

Once in the Account management area, select the Early visibility tab.

**Signing up**

1. Click to the right of the feature of interest (or features; you are able to request access to multiple features).
   
   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click if you agree and wish to continue.

   Click if you are no longer interested.

Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

**Teams**

To navigate to your Onshape Teams settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your Onshape profile. From there, click Teams in the list on the left side of the page.

This brings you to your Onshape Teams settings. You are able to create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you are able to select the team name instead of entering many users' individual email addresses during a Share operation.

It is not required that the members of a team have anything in common; not even an Onshape subscription.

One user creates a team (thereby becoming the initial admin of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Members receive notification emails when they are added and removed from a team, and users are able to belong to more than one team at a time.

Sharing a document with a team does not give any team member additional permissions on the document than the owner/creator of the document allows during the Share operation.

At any point, the admins of a team has the ability to remove any member from the team, thereby removing any Share permissions previously made through the team. Any Shares made on an individual basis remain in place, as well as the permission they grant.

Team members are able to remove themselves from a team, unless they are the last admin member of the team. (A team must have at least one admin.) When a member is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions are able to be assigned during the Share operation:

- **View** - Permission to open for read-only access; no editing allowed
- **Edit** - Permission to open and edit (make changes)
Following are instructions for:

- Creating teams and adding members
- Removing members and admins
- Deleting a team
- Additionally, see information about Sharing and assigning permissions to documents

**Creating teams and adding members**

1. Expand the menu under your user name in the top right corner of the page and select My account.

2. On the page that appears, select Teams from the left panel and click Create:

   ![Create Team Table](image)

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Members</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assembly</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>Body</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>Modeling</td>
<td></td>
<td>1</td>
</tr>
</tbody>
</table>

3. Enter a name for the team, and a description, or statement of purpose:

   ![Create Team Form](image)
4. Click Create team:

![Team creation interface](image)

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).

   Use the Search bar to search for team members.

   Note that you are able to return to this page and change a team member’s role.

6. Click Add.

7. When finished adding team members and assigning roles, click the arrow to the left of the team name (at the top of the page) to return to the Accounts page.

8. You see the new team listed on the Teams page.

   Creating a team also adds a filter for that team in each member’s Documents filters on their Documents page. These filters list all documents shared with a particular team.
Removing members and admins

Members have the ability to remove themselves from a team, and any member with an Admin role is able to remove users including themselves as long as they are not the only administrative user left. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under your user name in the top right corner of the page and select My account (for Standard and Free subscriptions) or Company settings (for Professional and Enterprise subscriptions).

2. Select Teams in the left panel to access the list of teams of which you are a member.

3. Select the team in the list from which you wish to remove yourself or another member.
   - To remove yourself (as a member): Click the Leave team button.
   - To remove yourself (as an admin): Click the X to the left of your name (Note this only works if there is another admin still on the team).
   - To remove another user: Click the X to the left of the user name (Note this only works if you are an admin).

Deleting a team

Any Admin of the team has the ability to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list.

Upgrading to Professional

Onshape's Professional subscription allows you to create unlimited private documents. The Professional subscription is a company subscription (you are able to pay for one or multiple users).

- If you are new to Onshape and do not yet have an account, click the Sign up link on the Onshape home page and follow the instructions.
If you already have an Onshape account, click the Upgrade button on your account page and follow the instructions below (or navigate to http://cad.onshape.com/upgrade (opens in new tab)).

When upgrading from a Free to a Professional subscription, any View only documents you previously saw on the Documents page are now editable documents. Onshape automatically makes all of your documents accessible to you.

**Canceling Education Subscription**

The Education plan allows free subscriptions to students and teachers. All documents created with an Education subscription are marked with a badge 🎓 forever. When a student is no longer a student, it's prudent to cancel your Education subscription and move to the Free subscription, and then optionally upgrade to the Professional subscription.

Education subscriptions expire after a year, at which point you are automatically downgraded to the Onshape Free subscription. However, at that point you are able to upgrade to the Education subscription again, provided you still meet the criteria.

To cancel the Education subscription and move to the Free subscription:

1. Expand the user menu under your user name and select My account.
2. Select the Subscriptions tab.
3. Follow the instructions to contact Onshape via email or phone call.

Your subscription is immediately downgraded to the Free subscription. Your private documents remain private, but you will not be able to edit them. Likewise, any private documents shared with you will be view-only (non-editable). You will still be able to view, export, and download your private documents. You are able to upgrade to the Professional subscription at any time and once again edit your private Onshape documents.

Note that all documents created through an Education subscription will always have a badge 🎓 attached regardless of transfer of ownership. If the document is made public, it will get a Public badge in addition to the EDU badge, and the EDU badge will mark any copies made and the document if it is made private again.

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All rights reserved.
Managing Your Standard Onshape Subscription

This functionality is available on Onshape's browser, iOS, and Android platforms.

Standard subscriptions are directed towards single professional users who do not have a need for company-wide settings and functionality. Standard subscriptions are billed per user (at an annual rate of $1,500), and include all Onshape features with the exception of the following:

- Automated Release management tools
- Custom properties (company metadata)
- Company-wide material libraries
- Company-based sharing
- Consolidated billing

The Account menu, located under your name in the upper-right corner of the interface, allows you to access:

- **My account** - Where you can manage and maintain your Onshape account, set preferences, notifications settings, security and more, as explained below.

- **Action items** - View any tasks that have been assigned to you through the Comments dialog. More on tasks, below.

- **View support tickets** - View any support tickets you have submitted. If you would like to submit a support ticket, look in the Help menu (the icon to the right of your name in the upper-right corner of the interface).

- **Sign out** - Sign out of and close your Onshape session.

**Action items**

When you select Action items from the account menu, you see page with the following filters on the left:
• **Role** - Choose between filtering through releases or tasks assigned to you or created by you by selecting one of the radio buttons to the left of the filter options.

• **Status** - Choose between filtering through open releases or tasks or closed releases or tasks by selecting one of the radio buttons to the left of the filter options.

• **Sort** - Sort your filtered results by Oldest releases and/or tasks first or Newest first by clicking the dropdown arrow under Sort and selecting your preference (Onshape defaults to sorting by Oldest first).

As you select different filters, the results for the corresponding selections will appear in the Action Items list located in the center of the page.

**My account**

All of the management and maintenance of your Onshape account can be done through the My account page, accessed through the User menu at the top right of the Onshape window. The icon for the user menu may look like this. You can also upload an image of your choosing to take the place of this icon.

**Profile**

To navigate to your Onshape profile, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This automatically brings you to your Onshape user profile.
Onshape automatically records the first and last names you specify during sign up; here you may also enter a personal nickname for display in the system (in the upper right-hand corner of the user interface). Upload a photo to be used next to your user name and on comments, in the Share dialog, and generally wherever lists of user information exists.

**Username** is the name to be used as your Onshape forum name.

**Nickname** is the name seen by other users when you collaborate and is also displayed in the upper right corner of your Onshape window.

Click the Update profile button at the bottom of the page to save your changes.

**Emails**

To navigate to your Emails settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your profile. From there, click Emails in the list on the left side of the page.

This brings you to your Emails settings.

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Specify up to three email addresses with which to access your Onshape account. One address functions as your primary email, used for all Onshape notifications and communications. Change the primary designation at any time after adding at least one more email address to your account.

All email addresses added to the system must be verified. Check the email address for a verification notice from Onshape.

Any email address associated with an account (even those not designated as primary) may not be used to create another Onshape account.

Remove an email from your account by clicking the small "x" next to the email listing (shown above).

You may use any of the verified email addresses on your account to request a reset for a forgotten password.

Preferences

To navigate to your Preferences settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Preferences in the left pane.

You can specify your preference for the following settings in Onshape.

Language

Select your preferred language from the dropdown. When you click Save language, Onshape automatically signs you out and you must sign in again to view the language change.
This is an ongoing effort; you may see terms that are not yet translated.

**Units**

The units of measurement and precision set here are used in all your Onshape documents, unless specifically overridden in a dialog (by entering units of choice) or by setting the default units per workspace through the workspace's Document menu > Workspace units. You can also select from a 12 hour or 24 hour time format here.

Onshape defaults to inch, degree, pound, and three decimal places for units of measure for all documents and encompasses all measurements in Part Studios, Assemblies, and Drawings; all values displayed in sketch dimensions as well as the default input units for all features.

The decimal place settings:
• Are currently available on browser only
• Are currently applied to the feature dialogs, sketch dimensions, and manipulator dialogs
• Work with the Measure tool and Mass properties tool
  • The Measure tool will display values in scientific notation when the display precision is not sufficient.
  • The Mass properties tool will display error in measurement; see "Mass Properties Tool" on page 349 for more information.
• Impacts the display only; values are rounded internally
• Are not used for computation
• Are used internally to determine the number of decimal places to display, regardless of how many places are entered; if more than the specified number are entered, they will be visible when the field is selected for edit.
• Do not affect any external files imported

Overriding default settings
In addition to setting default units for all documents you create (through this Preferences tab), you may also change and specify default units for a specific workspace in a document through the "Document Toolbar and Document Menu" on page 134 in a document.

Despite default settings, Onshape allows you to specify a different unit of measure in any numeric field; the value is converted to the default unit automatically. For example, if the default unit is inches, you may still specify a different unit type (for example "10mm") in a numeric field.

**Mouse controls**
You have the option to keep the default settings for mouse mappings, or select a familiar traditional CAD system’s default settings. These settings also control mouse mappings for Drawings.
Onshape supports 3Dconnexion devices including the SpaceMouse. See your SpaceMouse instructions for information on how to set up your mouse with Onshape.

Environment profile settings

Create device profile preferences here, including rendering at high resolution pixel density. This profile may be used on any device.

Associate the profile and the browser/device by selecting a particular profile on a particular machine or browser through this interface. Select a profile for each machine and each browser used on a machine.

- Match pixel density:
  - **Automatic (default)** - Onshape determines the resolution needed for rendering.
  - **On** - Render at the resolution of the display.
  - **Off** - Do not render at the resolution of the display. Graphics will be rendered at a lower resolution.

Creating a profile:

1. Click **Create profile**.
2. Enter a name for the profile and click **Create**.

3. Select the preferred setting for matching pixel density.

4. Click **Save profile settings**.

Deleting a profile:

1. Select the profile from the dropdown.

2. Click **Delete profile**.

   Note that this action may not be undone.

3. Click **OK** to confirm the deletion, or **Cancel**.

**Shortcut toolbars**

Customize the shortcut toolbars available for Sketch tools, Feature tools, Assembly tools, and Drawing tools. Select the toolbar to customize; check the tools to include and uncheck the tools to exclude from the menu. If you do not customize the toolbar, each time Onshape is updated, the default toolbar may change. Once customized, your customizations take precedence over any defaults.

- There are no limits to the number of tools you are able to include.
- The order of tools in the toolbar is determined by the order in the list (currently).
Use the S key to invoke the toolbar; use the Esc key to close the toolbar. The toolbar appears near the mouse pointer.

**Toolbars**

Click [Reset to defaults](#) to restore toolbars to the default settings.

**Drawings**

Set background color of model space for imported DWG and DXF files.

<table>
<thead>
<tr>
<th>Drawings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Background color of model space (imported DWG and DXF files)</td>
</tr>
<tr>
<td>Dark (default)</td>
</tr>
<tr>
<td>Save drawing settings</td>
</tr>
</tbody>
</table>

**Material libraries**

Create and add custom material libraries, remove unnecessary libraries, and make libraries available to all users within a company. For more detailed information, see "Customizing Parts: Materials" on page 322

**Material libraries**

Create and add custom material libraries, remove unnecessary libraries, and make libraries available to all users within a company. For more detailed information, see "Customizing Parts: Materials" on page 322

**Notifications**

This functionality is also available on Onshape's iOS and Android platforms.

To navigate to your Notifications settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.
This brings you to your Onshape profile. From there, click Notifications in the list on the left side of the page.

This brings you to your Onshape Notifications settings, where you have the option to edit details such as your first name, last name, username, nickname, and biography.

In your Notifications settings, there are two sections: Email notifications for shares and comments, and Mobile notifications for shares and comments. Both sections have various radio buttons you may or may not select in order to make notification changes, and a Save notifications button to save your changes.

Email notifications for shares and comments

The top section of your Onshape Notifications page is where you make changes to your Email notifications for shares and comments. In this section, you have three options to choose from:

- **All new shares and comments** - When selected, you will receive email notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive email notifications for shares directly to you, comments mentioning you, and documents you have specifically marked to receive comment emails.

- **Nothing** - When selected, you will not receive any email notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save email notifications button at the bottom of the section, when you have finished editing your Notifications, to save your changes.

Mobile notifications for shares and comments

The bottom section of your Onshape Notifications page is where you make changes for receiving notifications on mobile devices for shares and comments. Similar to the Email notifications for shares and comments section, in this section you have three options to choose from:
• **All new shares and comments** - When selected, you will receive mobile notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

• **Direct shares, mentions, and marked documents** - When selected, you will receive mobile notifications for shares directly to you, comments mentioning you and documents you have specifically marked to receive comment emails.

• **Nothing** - When selected, you will not receive any mobile notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save mobile notifications button at the bottom of the section when you have finished editing your Notifications to save your changes.

[Save mobile notifications]

**Security**

If you have forgotten your password and need it reset, proceed to the Onshape sign in page and click the "Forgot your password?" link to access a page on which you are able to request a password reset link via email.
To navigate to your Security settings, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Security in the list on the left side of the page.

This brings you to your Onshape Security settings. Change your Onshape system password, and also enable (or disable) two-factor authentication.

**Resetting password**

1. Expand the menu under your user name and select My account:
2. Select the Security tab.

3. Click Change password and enter the old password, the new password, and re-enter the new password.

   The list of guidelines leads you through creating a password. Each requirement is marked when your password fulfills the requirement.

4. Click Update password.

   **Two-Factor Authentication**

   Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

   We highly recommend you use 2FA for Onshape and for all websites you use that support it.
How it works

Download a two-factor authentication app (like Google Authenticator) to your phone and set it up with Onshape through the Onshape user interface. This enables the app to generate a one-time code that Onshape is able to recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You are able to allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign in to Onshape from a specific device, you won't need a 2FA code to sign in on that device for 30 days.

Enabling and using two-factor authentication

1. Download a two-factor authentication app to your device.

Google Authenticator is one example.

2. Sign in to your Onshape account.

3. In the menu under your username, select My account.

4. In the list on the left side of the page, click Security.

5. To the right of Two-factor Authentication, click Enable.

6. Click Set up two-factor authentication.

7. Confirm password.

8. Click OK.

Configuring the app to work with Onshape

Continuing from the instructions above:

1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface.

Once registration is complete, the phone app will list a code for each registration you create. It is these codes that you enter into Onshape when presented with the 2FA sign in page.
If you are not able to use the QR code, click the enter this text code link provided in the Onshape interface to obtain a code.

2. Enter either the six-digit code that the 2FA app generates or the code supplied by Onshape.

3. Click Enable.

4. When the recovery codes are displayed, copy them to a safe place; you need access to them in the event you do not have your phone or the authentication app.

5. Click OK.

Onshape provides you with 5 active recovery codes at a time. Keep these codes in a place accessible to you separate from your device or the authentication app.

**Onshape will not be able to help you should you delete the app or lose your phone.**

Note that you are able to generate these Recovery codes at any time through the Onshape interface, but only the most recently generated series are active at any one time. Once you use a code it is no longer valid. When you generate a new list of codes, all previous codes (used or unused) become invalid.

**Signing in to Onshape with code**

When two-factor authentication is enabled, Onshape prompts you for a code upon sign in:

1. After you enter the password to your Onshape account, you are prompted for the authentication code.

2. Open the two-factor authentication app on your device to view the code; enter the code in Onshape.
3. Click Verify.

In the event that you don't have access to the app, click the Enter a two-factor recovery code link to enter one of your current recovery codes.

Disabling two-factor authentication in Onshape
You may disable (and re-enable) two-factor authentication at any time.
1. On the Security tab of the User Profile page in Onshape click Manage, and then Disable:
2. Confirm password.
3. Click **OK**.

Replacing a device with 2FA enabled
Should you need to replace a device on which you have 2FA enabled for Onshape:
1. Before replacing the device, disable 2FA through the Onshape interface.
2. Enable 2FA once the new device is online.

Note that Onshape doesn't support the Replace 2FA option.

Devices
To navigate to your Devices settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Devices in the list on the left side of the page.

This brings you to your Onshape Devices settings, a list of all mobile devices associated with and authorized to use this account. Once you access your Onshape account on a mobile device, that mobile device is listed here.
To remove a device from the list, click Forget on the right of the window.

Applications

To navigate to your Onshape Applications settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there click My account.

This brings you to your Onshape profile. From there, click Applications in the list on the left side of the page.

This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store, navigate to [http://appstore.onshape.com](http://appstore.onshape.com) and sign in with your Onshape account credentials.

Here's a list of frequently asked questions ("App Store FAQs" on page 2613).

Once signed in to the App store, you are able to browse the available apps and make purchases. As the owner of a company or enterprise, you may purchase multiple seats for other users in your company or enterprise and assign users to those seats.

Types of apps

Onshape third-party apps are of the following types:

- **Integrated Cloud Application** - Accessible from within an Onshape document
- **Connected Desktop Application** - Downloaded from the third-party website and installed on your physical machine
- **Connected Cloud Application** - Accessible from a cloud-based service

After purchasing an app, you may have to grant it access to your Onshape documents.
1. While in a document, click the + and select Add application. (Select the desired application.)

2. Review the permissions you are about to authorize, then click Authorize application (or Deny if you no longer wish to use the app).

3. To view the permissions an application has on a document, or to give the application access to another document:
   a. Click Share to open the Share dialog and settings for that document.
   b. On the Application tab, select the application from the drop down and click Allow.

4. To revoke an application's access, click the x next to the application name at the top of the dialog.

Purchased apps that are authorized to access your Onshape data are listed in three places in your Onshape documents:

- Applications tab in the user profile (My account page) - Shows all apps you have authorized to access your Onshape data.
- Subscriptions tab in the user profile - Shows all apps for which you have a subscription.
- On the Add application command from the + menu at the bottom of your Onshape window.

**Purchasing and managing multiple seats for an application**

An owner of a company or enterprise may purchase multiple seats for an application, to make available to users of their organization. An owner may add and remove users at any time for any given application.

**Adding seats during purchase**

To add seats while purchasing an application:

1. Click the app button to purchase or subscribe, as you normally would, in the App store.

2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).
3. Click the purchase button.
4. Enter your Onshape password.
5. A confirmation dialog appears, providing more information about your purchase.
6. Click Close to dismiss the dialog.

Adding seats and managing users after purchase
Once you purchase multiple seats for an app, you can manage the users who are allowed to use that app. You can change users who may use the app and change the number of seats.

1. On the Applications tab of the Company settings page, select the app you wish to manage.
2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.
   A confirmation dialog appears and your changes take effect immediately.
3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the company.
4. To remove a user's access to the application, click the X next to their name in the list.

User actions on apps
Through the My account area and Applications tab, users are able to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.
Control application access to my documents individually through the Share dialog? - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.

If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the Share dialog to allow a specific application access to a specific document.

**Integrations**

To navigate to your Onshape Integrations settings to set up access to your Dropbox, Google Drive, or Microsoft OneDrive account, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Integrations in the list on the left side of the page.

This brings you to your Onshape cloud-source settings. You have the ability to grant Onshape access to your Dropbox, Google Drive, or Microsoft OneDrive account (separately), and also remove that access by clicking the red Remove button on the right side of the page.
To grant access for Onshape to import documents from (and export documents to) a Dropbox, Google Drive, or Microsoft OneDrive account, click the appropriate "Add" button and follow the steps. You'll need the credentials to your account in order to proceed through the wizard. For example:

Choose an account

to continue to Onshape - Connector

Diane Amadeo
diane@onshape.com

Use another account

To continue, Google will share your name, email address, and profile picture with Onshape - Connector. Before using this app, you can review Onshape - Connector's privacy policy and terms of service.

Early Visibility

To navigate to your Onshape Early visibility settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your Onshape profile. From there, click Early visibility in the list on the left side of the page.

This brings you to your Onshape Early visibility settings. The Onshape Early Visibility program offers you an opportunity to test functionality that is still under development. Due to the nature of features in development, we recommend you create specific documents for use with any Early Visibility feature. (Feel free to copy existing documents for this purpose.)

Please do not use documents you create under the Early Visibility feature for business critical or production use.

**Viewing programs**

Find the Early Visibility program sign up page under My account:

Once in the Account management area, select the Early visibility tab.

**Signing up**

1. Click **Add** to the right of the feature of interest (or features; you are able to request access to multiple features).

   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click **Accept** if you agree and wish to continue.

   Click **Cancel** if you are no longer interested.
Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

**Teams**

To navigate to your Onshape Teams settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Teams in the list on the left side of the page.

This brings you to your Onshape Teams settings. You have the ability to create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you have the option to select the team name instead of entering many users' individual email addresses during a Share operation.

It is not required that the members of a team have anything in common; not even an Onshape subscription.

One user creates a team (thereby becoming the initial admin of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Members receive notification emails when they are added and removed from a team, and users are able to belong to more than one team at a time.

Sharing a document with a team does not give any team member additional permissions on the document than the owner/creator of the document allows during the Share operation.

At any point, the admins of a team are able to remove any member from the team, thereby removing any Share permissions previously made through the team. Any Shares made on an individual basis remain in place, as well as the permission they grant.

Team members are able to remove themselves from a team, unless they are the last admin member of the team. (A team must have at least one admin.) When a member is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.
A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions are able to be assigned during the Share operation:

- **View** - Permission to open for read-only access; no editing allowed
- **Edit** - Permission to open and edit (make changes)

Following are instructions for:

- Creating teams and adding members
- Removing members and admins
- Deleting a team
- Additionally, see information about [Sharing and assigning permissions to documents](#)

### Creating teams and adding members

1. Expand the menu under your user name in the top right corner of the page and select My account.

2. On the page that appears, select Teams from the left panel and click Create:
3. Enter a name for the team, and a description, or statement of purpose:

4. Click Create team:

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).
Use the Search bar to search for team members. 
Note that you are able to return to this page and change a team member’s role.

6. Click Add.

7. When finished adding team members and assigning roles, click the arrow to the left of the team name (at the top of the page) to return to the Accounts page.

8. You see the new team listed on the Teams page.

Creating a team also adds a filter for that team in each member’s Documents filters on their Documents page. These filters list all documents shared with a particular team.

Removing members and admins

Members have the ability to remove themselves from a team, and any member with an Admin role is able to remove users including themselves as long as they are not the only administrative user left. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user’s Documents page.

1. Expand the menu under your user name in the top right corner of the page and select My account (for Standard and Free subscriptions) or Company settings (for Professional and Enterprise subscriptions).

2. Select Teams in the left panel to access the list of teams of which you are a member.

3. Select the team in the list from which you wish to remove yourself or another member.

   - To remove yourself (as a member): Click the Leave team button.
   - To remove yourself (as an admin): Click the X to the left of your name (Note this only works if there is another admin still on the team).
   - To remove another user: Click the X to the left of the user name (Note this only works if you are an admin).

Deleting a team

Any Admin of the team has the ability to delete the team at any time. This immediately
removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list.

**Payment Options/Setting up Payment**

**Setting up Payment**

When signing up for an Onshape Standard subscription, you enter credentials and credit card information to finish signing up:

1. Click your Onshape Account icon in the top right corner of your page. A dropdown menu opens, click My account:

   ![My account dropdown menu](image)

2. Click Payment options in the list on the left side of the page, then enter your credit card information in the spaces provided.

3. Click Confirm payment information and review your information.

4. Use the Change or Remove links to change the information or remove the credit card from your account.

**Payment Options**

To navigate to your Onshape Payment options settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:
This brings you to your Onshape profile. From there, click Payment options in the list on the left side of the page:

This brings you to your Onshape Payment options settings. If you are the owner of the account, you have the ability to change credit card information, or remove a card from the listing. Note that a credit card may be removed from the account only if it is not associated with a subscription:

To change the credit card, click Change and enter the new card information.
Upgrading to Professional

Onshape's Professional subscription allows you to create unlimited private documents. The Professional subscription is a company subscription (you are able to pay for one or multiple users).

- If you are new to Onshape and do not yet have an account, click the Sign up link on the Onshape home page and follow the instructions.
- If you already have an Onshape account, click the Upgrade button on your account page and follow the instructions below (or navigate to http://cad.onshape.com/upgrade (opens in new tab)).

When upgrading from a Free to a Professional subscription, any View only documents you previously saw on the Documents page are now editable documents. Onshape automatically makes all of your documents accessible to you.

Canceling Your Standard Subscription

Please contact Onshape if you wish to cancel your subscription.

Managing Your Onshape Professional Subscription

This functionality is available on Onshape's browser, iOS, and Android platforms.

The Onshape Professional subscription enables you to consolidate billing for multiple users, thereby creating a Company within Onshape: a named, user-visible Onshape entity with consolidated billing, ownership and sharing for a set of Professional subscription users. All company-owned documents are automatically shared with all company members.

The Professional company account also includes access to release management functionality, as well as the ability to create and use custom meta data.
If an existing Free user is listed as belonging to a Professional subscription, that user's plan is automatically upgraded to Professional and included in the company subscription billing. Any Onshape user may be paid for and included in a Professional subscription, and even multiple Professional subscriptions.

All users in a company have view access to the company settings. The owner has the ability to make changes to the company subscription (and app subscriptions) and admins of the company may edit some company settings.

The Account menu, located under your name in the upper-right corner of the interface, allows you to access:

- **My account** - Where you can manage and maintain your Onshape account, set preferences, notifications settings, security and more, as explained below.

- **Action items** - View any tasks that have been assigned to you through the Comments dialog. More on tasks, below.

- **View support tickets** - View any support tickets you have submitted. If you would like to submit a support ticket, look in the Help menu (the icon to the right of your name in the upper-right corner of the interface).

- **Sign out** - Sign out of and close your Onshape session.

**Action items**

When you select Action items from the account menu, you see page with the following filters on the left:

- **Role** - Choose between filtering through releases or tasks assigned to you or created by you by selecting one of the radio buttons to the left of the filter options.

- **Status** - Choose between filtering through open releases or tasks or closed releases or tasks by selecting one of the radio buttons to the left of the filter options.

- **Sort** - Sort your filtered results by Oldest releases and/or tasks first or Newest first by clicking the dropdown arrow under Sort and selecting your preference (Onshape defaults to sorting by Oldest first).

As you select different filters, the results for the corresponding selections will appear in the Action Items list located in the center of the page.

**My account settings**
Click your name or Account user icon in the upper right corner on your Onshape window, then click **My account** to access your Onshape account information.

If you navigated to your account with a document open, click the **Return to document link** in the upper right corner of the page to return to that document at any time.

**Profile**

To navigate to your Onshape profile, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click **My account**.

This automatically brings you to your Onshape user profile.

Onshape automatically records the first and last names you specify during sign up; here you may also enter a personal nickname for display in the system (in the upper right-hand corner of the user interface). Upload a photo to be used next to your user name and on comments, in the Share dialog, and generally wherever lists of user information exists.

**Username** is the name to be used as your Onshape forum name.

**Nickname** is the name seen by other users when you collaborate and is also displayed in the upper right corner of your Onshape window.
Click the Update profile button at the bottom of the page to save your changes.

**Emails**

To navigate to your Emails settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your profile. From there, click Emails in the list on the left side of the page.

This brings you to your Emails settings.

Specify up to three email addresses with which to access your Onshape account. One address functions as your primary email, used for all Onshape notifications and communications. Change the primary designation at any time after adding at least one more email address to your account.

All email addresses added to the system must be verified. Check the email address for a verification notice from Onshape.

Any email address associated with an account (even those not designated as primary) may not be used to create another Onshape account.

Remove an email from your account by clicking the small "x" next to the email listing (shown above).

You may use any of the verified email addresses on your account to request a reset for a forgotten password.
Preferences

To navigate to your Preferences settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Preferences in the left pane. You can specify your preference for the following settings in Onshape.

Language

Select your preferred language from the dropdown. When you click Save language, Onshape automatically signs you out and you must sign in again to view the language change.

This is an ongoing effort; you may see terms that are not yet translated.

Units

The units of measurement and precision set here are used in all your Onshape documents, unless specifically overridden in a dialog (by entering units of choice) or by setting the default units per workspace through the workspace's Document menu > Workspace units. You can also select from a 12 hour or 24 hour time format here.
Onshape defaults to inch, degree, pound, and three decimal places for units of measure for all documents and encompasses all measurements in Part Studios, Assemblies, and Drawings; all values displayed in sketch dimensions as well as the default input units for all features.

The decimal place settings:

- Are currently available on browser only
- Are currently applied to the feature dialogs, sketch dimensions, and manipulator dialogs
- Work with the Measure tool and Mass properties tool
  - The Measure tool will display values in scientific notation when the display precision is not sufficient.
  - The Mass properties tool will display error in measurement; see "Mass Properties Tool" on page 349 for more information.
- Impacts the display only; values are rounded internally
- Are not used for computation
● Are used internally to determine the number of decimal places to display, regardless of how many places are entered; if more than the specified number are entered, they will be visible when the field is selected for edit.

● Do not affect any external files imported

Overriding default settings

In addition to setting default units for all documents you create (through this Preferences tab), you may also change and specify default units for a specific workspace in a document through the "Document Toolbar and Document Menu" on page 134 in a document.

Despite default settings, Onshape allows you to specify a different unit of measure in any numeric field; the value is converted to the default unit automatically. For example, if the default unit is inches, you may still specify a different unit type (for example "10mm") in a numeric field.

**Mouse controls**

You have the option to keep the default settings for mouse mappings, or select a familiar traditional CAD system’s default settings. These settings also control mouse mappings for Drawings.
Onshape supports 3Dconnexion devices including the SpaceMouse. See your SpaceMouse instructions for information on how to set up your mouse with Onshape.

**Environment profile settings**

Create device profile preferences here, including rendering at high resolution pixel density. This profile may be used on any device.

Associate the profile and the browser/device by selecting a particular profile on a particular machine or browser through this interface. Select a profile for each machine and each browser used on a machine.

- **Match pixel density:**
  - **Automatic (default)** - Onshape determines the resolution needed for rendering.
  - **On** - Render at the resolution of the display.
  - **Off** - Do not render at the resolution of the display. Graphics will be rendered at a lower resolution.

Creating a profile:

1. Click **Create profile**.
2. Enter a name for the profile and click **Create**.
3. Select the preferred setting for matching pixel density.
4. Click **Save profile settings**.

Deleting a profile:

1. Select the profile from the dropdown.
2. Click **Delete profile**.

   Note that this action may not be undone.
3. Click **OK** to confirm the deletion, or **Cancel**.

**Shortcut toolbars**
Customize the shortcut toolbars available for Sketch tools, Feature tools, Assembly tools, and Drawing tools. Select the toolbar to customize; check the tools to include and uncheck the tools to exclude from the menu. If you do not customize the toolbar, each time Onshape is updated, the default toolbar may change. Once customized, your customizations take precedence over any defaults.

- There are no limits to the number of tools you are able to include.
- The order of tools in the toolbar is determined by the order in the list (currently).
- Use the S key to invoke the toolbar; use the Esc key to close the toolbar. The toolbar appears near the mouse pointer.

Click **Reset to defaults** to restore toolbars to the default settings.

### Drawings

Set background color of model space for imported DWG and DXF files.
Material libraries

Create and add custom material libraries, remove unnecessary libraries, and make libraries available to all users within a company. For more detailed information, see "Customizing Parts: Materials" on page 322

Notifications

To navigate to your Notifications settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Notifications in the list on the left side of the page.

This brings you to your Onshape Notifications settings, where you have the option to edit details such as your first name, last name, username, nickname, and biography.

In your Notifications settings, there are two sections: Email notifications for shares and comments, and Mobile notifications for shares and comments. Both sections have various radio buttons you may or may not select in order to make notification changes, and a Save notifications button to save your changes.

Email notifications for shares and comments

The top section of your Onshape Notifications page is where you make changes to your Email notifications for shares and comments. In this section, you have three options to choose from:

- **All new shares and comments** - When selected, you will receive email notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive email notifications for shares directly to you, comments mentioning you, and documents you have specifically marked to receive comment emails.

- **Nothing** - When selected, you will not receive any email notifications for shares or comments.
Click the radio button to the left of the option in order to select it. Click the Save email notifications button at the bottom of the section, when you have finished editing your Notifications, to save your changes.

Mobile notifications for shares and comments

The bottom section of your Onshape Notifications page is where you make changes for receiving notifications on mobile devices for shares and comments. Similar to the Email notifications for shares and comments section, in this section you have three options to choose from:

- **All new shares and comments** - When selected, you will receive mobile notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive mobile notifications for shares directly to you, comments mentioning you and documents you have specifically marked to receive comment emails.

- **Nothing** - When selected, you will not receive any mobile notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save mobile notifications button at the bottom of the section when you have finished editing your Notifications to save your changes.

**Security**

If you have forgotten your password and need it reset, proceed to the Onshape sign in page and click the "Forgot your password?" link to access a page on which you are able to request a password reset link via email.
To navigate to your Security settings, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Security in the list on the left side of the page.

This brings you to your Onshape Security settings. Change your Onshape system password, and also enable (or disable) two-factor authentication.

**Resetting password**

1. Expand the menu under your user name and select My account:
2. Select the Security tab.

3. Click Change password and enter the old password, the new password, and re-enter the new password.

The list of guidelines leads you through creating a password. Each requirement is marked when your password fulfills the requirement.

4. Click Update password.

**Two-Factor Authentication**

Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.
How it works

Download a two-factor authentication app (like Google Authenticator) to your phone and set it up with Onshape through the Onshape user interface. This enables the app to generate a one-time code that Onshape is able to recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You are able to allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign in to Onshape from a specific device, you won't need a 2FA code to sign in on that device for 30 days.

Enabling and using two-factor authentication

1. Download a two-factor authentication app to your device.

   Google Authenticator is one example.

2. Sign in to your Onshape account.

3. In the menu under your username, select My account.

4. In the list on the left side of the page, click Security.

5. To the right of Two-factor Authentication, click Enable.

6. Click Set up two-factor authentication.

7. Confirm password.

8. Click OK.

Configuring the app to work with Onshape

Continuing from the instructions above:

1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface.

   Once registration is complete, the phone app will list a code for each registration you create. It is these codes that you enter into Onshape when presented with the 2FA sign in page.
If you are not able to use the QR code, click the enter this text code link provided in the Onshape interface to obtain a code.

2. Enter either the six-digit code that the 2FA app generates or the code supplied by Onshape.

3. Click Enable.

4. When the recovery codes are displayed, copy them to a safe place; you need access to them in the event you do not have your phone or the authentication app.

5. Click OK.

Onshape provides you with 5 active recovery codes at a time. Keep these codes in a place accessible to you separate from your device or the authentication app.

**Onshape will not be able to help you should you delete the app or lose your phone.**

Note that you are able to generate these Recovery codes at any time through the Onshape interface, but only the most recently generated series are active at any one time. Once you use a code it is no longer valid. When you generate a new list of codes, all previous codes (used or unused) become invalid.

**Signing in to Onshape with code**

When two-factor authentication is enabled, Onshape prompts you for a code upon sign in:

1. After you enter the password to your Onshape account, you are prompted for the authentication code.

2. Open the two-factor authentication app on your device to view the code; enter the code in Onshape.
3. Click Verify.

In the event that you don't have access to the app, click the **Enter a two-factor recovery code** link to enter one of your current recovery codes.

**Disabling two-factor authentication in Onshape**

You may disable (and re-enable) two-factor authentication at any time.

1. On the *Security* tab of the User Profile page in Onshape click Manage, and then Disable:

2. Confirm password.

3. Click **OK**.

**Replacing a device with 2FA enabled**

Should you need to replace a device on which you have 2FA enabled for Onshape:

1. Before replacing the device, disable 2FA through the Onshape interface.

2. Enable 2FA once the new device is online.

Note that Onshape doesn't support the Replace 2FA option.

**Devices**

To navigate to your Devices settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Devices in the list on the left side of the page.

This brings you to your Onshape Devices settings, a list of all mobile devices associated with and authorized to use this account. Once you access your Onshape account on a mobile device, that mobile device is listed here.
To remove a device from the list, click Forget on the right of the window.

**Applications**

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This brings you to your Onshape profile. From there, click Applications in the list on the left side of the page.

This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store, navigate to [http://appstore.onshape.com (opens in new tab)] and sign in with your Onshape account credentials.

Here's a list of frequently asked questions ("App Store FAQs" on page 2613).

Once signed in to the App store, you are able to browse the available apps and make purchases. As the owner of a company or enterprise, you may purchase multiple seats for other users in your company or enterprise and assign users to those seats.

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2. Review the permissions you are about to authorize, then click Authorize application (or Deny if you no longer wish to use the app).

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   a. Click Share to open the Share dialog and settings for that document.
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2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).
3. Click the purchase button.

4. Enter your Onshape password.

5. A confirmation dialog appears, providing more information about your purchase.

6. Click Close to dismiss the dialog.

Adding seats and managing users after purchase

Once you purchase multiple seats for an app, you can manage the users who are allowed to use that app. You can change users who may use the app and change the number of seats.

1. On the Applications tab of the Company settings page, select the app you wish to manage.

2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.

   A confirmation dialog appears and your changes take effect immediately.

3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the company.

4. To remove a user's access to the application, click the X next to their name in the list.

User actions on apps

Through the My account area and Applications tab, users are able to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.
Control application access to my documents individually through the Share dialog? - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.

If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the Share dialog to allow a specific application access to a specific document.

**Integrations**

To navigate to your Onshape Integrations settings to set up access to your Dropbox, Google Drive, or Microsoft OneDrive account, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click **Integrations** in the list on the left side of the page.

This brings you to your Onshape cloud-source settings. You have the ability to grant Onshape access to your Dropbox, Google Drive, or Microsoft OneDrive account (separately), and also remove that access by clicking the red Remove button on the right side of the page.
To grant access for Onshape to import documents from (and export documents to) a Dropbox, Google Drive, or Microsoft OneDrive account, click the appropriate "Add" button and follow the steps. You’ll need the credentials to your account in order to proceed through the wizard. For example:

**Early visibility**

To navigate to your Onshape Early visibility settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your Onshape profile. From there, click Early visibility in the list on the left side of the page.

This brings you to your Onshape Early visibility settings. The Onshape Early Visibility program offers you an opportunity to test functionality that is still under development. Due to the nature of features in development, we recommend you create specific documents for use with any Early Visibility feature. (Feel free to copy existing documents for this purpose.)

Please do not use documents you create under the Early Visibility feature for business critical or production use.

Viewing programs

Find the Early Visibility program sign up page under My account:

Once in the Account management area, select the Early visibility tab.

Signing up

1. Click Add to the right of the feature of interest (or features; you are able to request access to multiple features).

   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click Accept if you agree and wish to continue.

   Click Cancel if you are no longer interested.
Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

**Subscriptions**

To navigate to your Onshape Subscriptions settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:

![My account menu](image)

This brings you to your Onshape profile. From there, click Subscriptions in the list on the left side of the page:

![Subscriptions menu](image)

This brings you to your Onshape Subscriptions settings. You may belong to one or many Professional subscriptions, per email address. You are able to have one set of Onshape credentials per email address.
Professional subscriptions are considered company subscriptions but are appropriate for any professional user, within a company or on their own. Professional subscriptions are billed per user (at an annual rate of $2100), and include all features of Onshape including private documents, Release Management, Custom properties, company-wide material libraries, consolidated billing, and company-based sharing.

On the Subscriptions page and you have the ability to:

- View a list of applications you or the company have subscribed to
- Edit the membership of the company (if you are the owner or an admin of the company), including managing the company; there is a subscription cost for each user
- Update your credit card information
- Cancel a subscription that you own; transitioning all users to Free subscriptions immediately (users with other Professional subscriptions are not transitioned to a Free subscription)
- Click View for a printable invoice

**Payment Options/Payment History**

To navigate to your Onshape Payment options settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account:

This brings you to your Onshape profile. From there, click Payment options in the list on the left side of the page:
This brings you to your Onshape Payment options settings. If you are the owner of the account, you have the ability to change credit card information, or remove a card from the listing. Note that a credit card may be removed from the account only if it is not associated with a subscription:

To change the credit card, click Change and enter the new card information.

Payment History

This area lists a chronological history of all payments made for the account. Click View to access a print-friendly invoice.

Only Company account Owners and Admins are able to see this information.

Teams

To navigate to your Onshape Teams settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.
This brings you to your Onshape profile. From there, click Teams in the list on the left side of the page.

This brings you to your Onshape Teams settings. You have the ability to create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you have the option to select the team name instead of entering many users’ individual email addresses during a Share operation.

It is not required that the members of a team have anything in common; not even an Onshape subscription.

One user creates a team (thereby becoming the initial admin of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Members receive notification emails when they are added and removed from a team, and users are able to belong to more than one team at a time.

Sharing a document with a team does not give any team member additional permissions on the document than the owner/creator of the document allows during the Share operation.

At any point, the admins of a team are able to remove any member from the team, thereby removing any Share permissions previously made through the team. Any Shares made on an individual basis remain in place, as well as the permission they grant.

Team members are able to remove themselves from a team, unless they are the last admin member of the team. (A team must have at least one admin.) When a member is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions are able to be assigned during the Share operation:

- **View** - Permission to open for read-only access; no editing allowed
- **Edit** - Permission to open and edit (make changes)
Following are instructions for:

- Creating teams and adding members
- Removing members and admins
- Deleting a team
- Additionally, see information about Sharing and assigning permissions to documents

**Creating teams and adding members**

1. Expand the menu under your user name in the top right corner of the page and select My account.

2. On the page that appears, select Teams from the left panel and click Create:

3. Enter a name for the team, and a description, or statement of purpose:
4. Click Create team:

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).

   Use the Search bar to search for team members.
   Note that you are able to return to this page and change a team member’s role.

6. Click Add.

7. When finished adding team members and assigning roles, click the arrow to the left of the team name (at the top of the page) to return to the Accounts page.

8. You see the new team listed on the Teams page.

    Creating a team also adds a filter for that team in each member’s Documents filters on their Documents page. These filters list all documents shared with a particular team.
Removing members and admins

Members have the ability to remove themselves from a team, and any member with an Admin role is able to remove users including themselves as long as they are not the only administrative user left. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under your user name in the top right corner of the page and select My account (for Standard and Free subscriptions) or Company settings (for Professional and Enterprise subscriptions).

2. Select Teams in the left panel to access the list of teams of which you are a member.

3. Select the team in the list from which you wish to remove yourself or another member.
   - To remove yourself (as a member): Click the Leave team button.
   - To remove yourself (as an admin): Click the X to the left of your name (Note this only works if there is another admin still on the team).
   - To remove another user: Click the X to the left of the user name (Note this only works if you are an admin).

Deleting a team

Any Admin of the team has the ability to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list.

Company, Owner and Admin

A company is created when a user signs up for the Professional subscription (or upgrades to Professional).

The user who signs up and agrees to pay for the subscription becomes the billing Owner of the company and acting Admin. This user adds more users to the company and assigns roles: Member or Admin. Users receive an email notification upon being added to a company.
Only users with the Owner and Admin roles are able to add and remove users, and reassign roles.

**Documents, company ownership and permissions**

For all company members, all documents created are owned by the company. If you are a member of more than one company, you should select as the owner of the document at creation time, one of the companies of which you are a member.

Only the company owner, users with the Admin role, and the user who created a company-owned document may delete the document. Users with full permissions to the document are able to see the document in their Trash and may restore the document or empty the Trash.

All users in a company have the ability to share all company-owned documents they have access to.

At any point, the company Owner and Admins may completely remove all document permissions from the user who created the document and reshare the document with them with a new set of permissions. Permissions may be:

- **View** - Open for read only access; you have the option to add or remove Copy, Link document, Export, and Comment

- **Edit** - Open and make changes; you have the option to add or remove Copy, Link document, Export, Share, and Comment

Additional permissions may be added or removed, including:

- **Copy** - Make a copy of the document.

- **Link document** - Use features that result in the document being referenced from another document.

- **Export** - Translate and download parts, Part Studios, Drawings, and Assemblies from a document.

- **Share** - Give another user permission to access the document.

- **Comment** - Provide a comment on the document in the Comment flyout.

- **Delete** - Move the document to Trash.

**Transferring ownership of public documents**
Once your account is upgraded to a professional account, you then have the ability to make all of your previously public documents private by going to the Share settings and clicking the x next to the public entry in the Share dialog (shown below to the right of the blue arrow). These documents are still owned by you after they are made private (company-owned documents are not able to be transferred, documents and folders must be owned by an individual). To use Release management, Custom properties, Items for BOM, and other company-level settings, you must transfer document ownership to the company.

Check the ownership of the document by looking at the top right corner of the Share dialog (shown below outlined in blue).

For information on how to transfer ownership, see the Transfer Ownership topic.

Company settings
To navigate to the company settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, select Company settings.

You have the ability to adjust the settings for your company, as described below.

**Users**

Available to Professional and Enterprise admins only, view and edit user details and roles, add and remove users. Only the company Owner and the Admin have the ability to add users to and remove users from the subscription.

The company owner has the ability to adjust the number of users paid for with the subscription (seats), as well as add and remove users from the subscription.

A company has two types of users:

- Admin
- Member

The company **Owner** is the initial account used to create the company. This user has full administrative permissions of a company admin, as well as the ability to manage payment information. A company **Admin** has administrative permissions to manage the company and may access all documents owned by the company and also has the ability to: add, remove, manage users; manage the company details; create and manage properties to be used throughout the company; define company preferences such as custom material libraries; establish the release management workflow. A company **Member** has an account within the company and all documents created by that user are owned by that company. Company members may access documents shared with the company.

Navigate to Company settings under the User menu, then select Users.

**Add seats**

You may add seats to your subscription if you are the owner of the company and your purchase is on a credit card. If you have a purchase order, please contact your Customer Success representative or Onshape Support for assistance.
1. Select <your company name> **company settings** from the User menu:

![User menu screenshot]

2. Select **Users** on the left pane:

![Users menu screenshot]

3. Enter the new **number of users you are paying for**.

4. Click **Update**.

Review and confirm the number of users in the dialog that appears; click **OK**.

Editing the number of users (seats) for your subscription is immediately reflected in the credit card charge.

For further assistance on adding users or other purchase-related activities, please contact your Customer Success representative.

**Add users**

To add users to a subscription:

1. Under **Add users**, enter the email address of the user to add and select a role (Member or Admin):

![Add users screenshot]
2. Click Add.

3. Repeat steps 1 and 2 to add additional users.

**Remove users**

1. Search for the user you wish to remove, if necessary.

2. Click the x at the end of the appropriate user row to open the Remove user dialog box:

   ![Remove user dialog box](image)

   The options will automatically be selected. Make your preferred changes and click OK to save them and close the dialog box.

**Change user role**

1. Search for the appropriate user, if necessary.

2. Click the current role (in blue, with a dashed underline).

3. Use the dropdown to select a new role.

4. Click the check to save. (Click the x to cancel.)

**Teams**

You have the ability to create teams in order to group users together for the purpose of
increasing the efficiency of sharing; once the team is created, you can select the team name instead of entering many users' individual email addresses during a Share operation.

It is not required that the members of a team have anything in common; not even an Onshape subscription.

One user creates a team (thereby becoming the initial admin of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Members receive notification emails when they are added and removed from a team, and users may belong to more than one team at a time.

Any user can share a document with a company-owned team, even if they are not a member of the team. However, in order to share a document with a team external to the company or enterprise, the user must be a member of that team.

Sharing a document with a team does not give any team member additional permissions on the document than the owner/creator of the document allows during the Share operation.

At any point, the admins of a team may remove any member from the team, thereby removing any Share permissions previously made through the team. Any Shares made on an individual basis remain in place, as well as the permission they grant.

Team members can remove themselves from a team, unless they are the last admin member of the team. (A team must have at least one admin.) When a member is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions may be assigned during the Share operation:

- **Can view** - Permission to open for read-only access
- **Can edit** - Permission to open and edit (make changes)

Following are instructions for:
Creating teams and adding members

- Creating teams and adding members
- Removing members and admins
- Deleting a team
- Additionally, see information about [Sharing and assigning permissions to documents](#)

**Creating teams and adding members**

1. Expand the menu under your user name/Account user icon in the top right corner of the page and select Company settings:

2. On the page that appears, select Teams from the left panel and click **Create**:

3. Enter a name for the team, and a description, or statement of purpose:
4. Click Create team:

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).

   Use the Search bar to search for team members.
   Note that you can return to this page and change a team member's role.

6. Click Add.

7. When finished adding team members and assigning roles, click the arrow to the left of the team name (at the top of the page) to return to the Accounts page.

8. You see the new team listed on the Teams page.
Creating a team also adds a filter for that team in each member's Documents filters on their Documents page. These filters list all documents shared with a particular team.

**Removing members and admins**

Members may remove themselves from a team, and any member with an Admin role may remove users including themselves as long as they are not the only administrative user left. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under your user name or Account user icon in the top right corner of the page and select My account (for Standard and Free subscriptions) or Company settings (for Professional and Enterprise subscriptions).
2. Select Teams in the left panel to access the list of teams of which you are a member.
3. Select the team in the list from which you wish to remove yourself or another member.
   - To remove yourself (as a member): Click the Leave team button.
   - To remove yourself (as an admin): Click the X to the left of your name (Note this only works if there is another admin still on the team).
   - To remove another user: Click the X to the left of the user name (Note this only works if you are an admin).

**Deleting a team**

Any Admin of the team can delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list.

**Properties**

The properties on this page define the metadata available for Onshape entities and apply only to documents that are owned by the company. Company owners and admins can create new properties, activate them for use by users, and also deactivate properties.
To navigate to your Onshape Custom properties settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click Company settings:

![Dropdown menu with options](image)

This brings you to your Onshape company settings Users page. From there, click Properties in the list on the left side of the page:

![Properties list](image)

This brings you to your Onshape Properties settings. Available for Professional and Enterprise users only, Onshape provides access to the metadata definitions for Onshape objects. These definitions drive the data displayed in the Properties dialogs and bill of materials for Onshape objects. Professional and Enterprise users can view these metadata definitions, and Admins and Owners can create new custom
properties for use in company-owned documents, as well as provide Display names for existing Onshape properties.

Owners and users with the Admin role can add, modify and retire custom properties. Onshape metadata definitions may be made Inactive (retired) but may not be edited.

Users with permission to create and edit custom properties see a Create button at the top of this page:

Properties may be associated with these Onshape objects:

- Onshape Part
- Onshape Assembly
- Onshape Drawing
- Onshape Part Studio
- Onshape File
- Onshape Application
- Onshape Item

**Creating and activating new custom properties**

On the Properties page:
1. Click Create custom property.

2. Specify a name for the property. The name should be unique among Onshape and custom properties. When testing for uniqueness, Onshape uses the Company name and the name of the property.

3. Select a property type:
   - Text
   - Boolean
   - Integer
   - Double
   - Date
   - List
   - User

4. The initial Publish state is Pending (not yet available to users). When you want this property to be available to users, select Active (available to users and values entered are recorded in the database). To retire this property from use, select Inactive. Inactive values are available only through the Onshape API.

5. Supply the rest of the property’s attributes, a red asterisk indicates a required field. Depending on the property type you selected above, different attributes may be available.
   
a. **Default value** - Enter a default value here, if desired. This value serves as the default if the user doesn’t specify a value.

   When entering values for the User property type, it must be a valid Onshape user’s email who already belongs to the company in which the property was created.

b. Enter an optional Description for the property.

c. Indicate where the property may be edited and whether it is required:
   - **Required** - Indicates that the property value is required.
   - **Edit value in version** - Indicates that this property value may be edited in a document version; allow the user to supply a value for this property in any document
version. Also indicate if the value should be edited only through the API and not through the Onshape user interface.

- **Edit value in workspace** - Indicates that this property may be edited in a workspace; allow a user to supply a value for this property in any document workspace. Also indicate if the value should be edited only through the API and not through the Onshape user interface.

6. Enter optional **validation** criteria for properties, including those that can have a pattern:
   a. Check Multiline if the value can have multiple lines of text (for example, a Description may require multiple lines of text).
   b. Enter a minimum length and/or a maximum length, if desired.
   c. Indicate the pattern in the Pattern field, such as any regular expression including:
      - `[A-Z]+` which requires 1 or more uppercase alphabetical characters
      - `[0-9]+` which requires 1 or more numeric characters
      - `[a-z]+` which requires 1 or more lowercase alphabetical characters
      - `ONS-[0-9]+` which requires the prefix ‘ONS-’ followed by 1 or more numeric characters. You could put the required prefix in the Default value attribute so it appeared automatically. Users receive an error notification specifying the required pattern if the value is invalid.

7. Select a category (or categories) in which the property will be available. The category must already be defined through the Categories tab by an administrator of the company or enterprise. You may select more than one category, and also remove a category by clicking the X to the far right of the category line. For more information on Categories, see below.

8. To make the property available to all users in the Company, make sure the Publish state is set to Active.

9. Click **Create**.
10. A message appears reminding you that if you make this property active, all users in the company or enterprise will be able to see it. Click Confirm publish to continue to activate the property. You can publish later instead.

You can also use FeatureScript to create custom properties. These custom properties are also displayed on the Custom properties page, by name and property ID.

**Modifying properties**

Onshape metadata and custom properties are only able to be edited by Owners and users with the Admin role.

On the Properties page, you have the ability to:

- Search for properties. Use the search box to enter a property name or partial name; check the box to search for inactive properties.
  
  The list presents Onshape metadata properties first, then custom properties in alphabetical order.

- Click a Property name to open it for editing. You may make any changes desired to a property with a Publish state of Pending, including deleting that property. The only changes you are able to make to a property with a Publish State of Active is to change the state to Inactive and change the Display name. Inactive properties are not visible in any Properties dialogs, but are still associated with the objects. Any other changes made to an Active property are immediately available and effective upon Save. An Inactive property is able to be made Active again.

  See "Creating and activating new custom properties" on page 2531 for information on modifying fields.

**Using properties**

When using the User custom property type, when a user fills the field, the dropdown will list the current user at the top of the list. However, searching on the current user will yield no results.

**Retiring properties**

When a property becomes obsolete, or an error in the definition is discovered after the property was made Active, you are able to retire the property. Retiring a property removes it from all Properties dialogs but keeps the property associated with the
objects and preserves the data in the database. To retire a property:

On the Properties page:

1. Search to locate the property, if necessary.
2. Click the property name to open for editing.
3. Select the dropdown for Publish state and select Inactive.
4. Click Save changes.

Inactive properties may be made Active again.

**Categories**

Creating categories in Onshape provides the ability to extend the properties of the standard Onshape object types in the system to include more targeted and relevant metadata to be applied based on design, engineering, and manufacturing processes.

Before creating categories for the first time, you can prepare by checking your PLM, or other appropriate system, for important metadata you want to be able to represent within Onshape.

Categories work in conjunction with custom properties, to group those custom properties into reasonable and useful information. Once categories are defined, you can see the custom properties within your Properties dialogs for all Onshape objects to which the category is applied.

An example

A company manufactures a product that is a combination of a part designed and produced in-house and parts ordered from a third party vendor. When working with off-the-shelf parts from a vendor, you may care about only a handful of properties like Vendor name, Vendor part number, and perhaps things like weight and cost.

However, when working with parts manufactured in-house, the list of properties will likely be entirely different. These properties may include information like Start date, Approved date, Manufacturing method, Length, and more.

This is where Onshape categories come into play. With categories, you can assemble the properties you really care about and need, so they are grouped together in all Properties dialogs within Onshape as "newly created category". Once this is selected,
the properties within the Category are listed first in the dialog, making them easier to find and fill out.

Creating a category

To create a category:

1. Navigate to Company settings.

2. Select Categories. Onshape has standard existing categories for Onshape objects, including: Global, Part, Assembly, Drawing, Part Studio, File, Application, Version, Workspace, and Item. The Scope of these (and all) categories are listed in the next column. The Scope refers to the Onshape objects the properties within the categories apply to.

3. Click the Create category button at the top of the page.

4. Select a Scope, that is, the types of Onshape objects to which this category may be applied. Note that selecting a Scope is not required.

5. Assign a Name for the category.

6. Select a Publish state: Active = visible by and available to all users; Pending = Saved but unavailable for use; Inactive = No longer available for use. You can mark a category as Pending, then go back later and make it Active.
7. Supply a **Description** of the category, if desired.

8. Select any **Parent categories**, existing categories from which you want this new category to inherit properties. Assigning a parent category to a category carries properties assigned to the parent category over to the category. This is useful when the new category uses all of the properties that are already assigned to the parent category, but requires a few extra properties specific to the new category.

9. Select the Properties (custom properties), that you want this category to include. These are the properties that will display whenever Properties of the object are displayed or edited.

10. Click **Create** to create the new category.

The new category is listed on the main Categories page. Categories are also available in the Advanced search dialog, in the Add criteria dropdown.

**Sub-categories and inheritance**

Categories can also inherit from a parent category, if you wish. These nested categories inherit the properties of the parent, and also allow you to add properties that are unique to that sub-category. One example for using inherited properties would be a category for parts that are manufactured in-house. This parent category could have all the common properties across multiple parts like part number, release state, and date approved. You can then create sub-categories under that "in-house" category for each type of part you manufacture, like: sheet metal, injection molded, machined, and
welded, and then have those categories contain properties unique to those types of products and their processes.

The only difference when creating a sub-category, is to select a Parent category:

![Image of Onshape interface for creating a sub-category]

The Parent category of the new category being created, above, is "In-house manufacturing" and its properties are listed as "Inherited properties" under Properties.

![Image of Onshape interface showing inherited properties]

You would also select specific properties for the sub-category, shown above as Product line and Vendor.

Once you click Create, the sub-category is listed on the Categories page. You can view it in the list, just as any other category, and you can also expand the parent category and view it there.
The Welded category shown indented below “In-house manufacturing” is the same category shown last in the list, above.

You can select one or the other of the entries for a sub-category and the other entry is also highlighted; this makes subcategories easier to find, as shown below:

```
<table>
<thead>
<tr>
<th>Category</th>
<th>Scope</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>Part</td>
<td>Part</td>
<td>Active</td>
</tr>
<tr>
<td>Assembly</td>
<td>Assembly</td>
<td>Active</td>
</tr>
<tr>
<td>Drawing</td>
<td>Drawing</td>
<td>Active</td>
</tr>
<tr>
<td>Part Studio</td>
<td>Part Studio</td>
<td>Active</td>
</tr>
<tr>
<td>File</td>
<td>File</td>
<td>Active</td>
</tr>
<tr>
<td>Application</td>
<td>Application</td>
<td>Active</td>
</tr>
<tr>
<td>Version</td>
<td>Version</td>
<td>Active</td>
</tr>
<tr>
<td>Workspace</td>
<td>Workspace</td>
<td>Active</td>
</tr>
<tr>
<td>Item</td>
<td>Item</td>
<td>Active</td>
</tr>
<tr>
<td>In-house manufacturing</td>
<td>Part</td>
<td>Active</td>
</tr>
<tr>
<td>Welded</td>
<td>Part</td>
<td>Active</td>
</tr>
<tr>
<td>Welded</td>
<td>Part</td>
<td>Active</td>
</tr>
</tbody>
</table>
```

To view the list of categories without sub-categories listed, place a check in the Hide sub categories in list checkbox at the top of the interface.

**Preferences**

To navigate to your Onshape Company preferences, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu,
from there, click Company settings.

This brings you to your Onshape company settings Users page. From there, click Preferences in the left pane:

![Preferences panel](image)

This brings you to your Onshape Preferences settings, where you have the ability to supply company-owned material libraries for other users in their own accounts and/or show a view only toolbar to all users of a document with view only permissions:

![Material libraries settings](image)

For information on the simplified user interface, see "Using the View Only Toolbar" on page 2093.

For more information on custom material libraries, see "Customizing Parts: Materials" on page 322.

**Items**

Here you can add items that you will want to add to your bill of material (see "Bill of Materials" on page 1509 for information) but that do not require being modeled (non-geometric entities). Adding those items here allows you to make use of them in any
company or enterprise-owned documents.

Before creating items for the first time, you can prepare for it by ensuring you have your desired categories of materials or supplies already present in your company settings.

To check your company categories:

1. Navigate to **Properties** under Company settings.
2. Find the **Category** property, click the name to edit it.
3. Under **Existing list values** you can browse the Categories that have already been added.
4. To add new categories (for example, "Adhesives"), enter the category names in the "Add new list values" text box:

   ![Existing list values](image)

   ![Add new list values (Enter one list value per line)](image)

   *The category names must be entered one value per line, and each value may contain commas (as seen in the "Existing list values" box, above)*

5. To save your new Category values, click **Save changes** at the bottom of the page.

To create **items** for company or enterprise-owned documents to make use of:
1. Click **Create**.

2. Click **Item** to create an individual item:
a. Enter information for the metadata, such as: Name, Description, Part number, Unit of measure, Category (the Category must have previously been added through the Custom properties for Category; see above).

b. Assign a Publish state (Pending= not yet published; no users can access the item, Active = immediately available to users, Inactive = unavailable to users, retired)

c. Click **Create**.

3. To import a comma-separated value list (CSV):

a. Create the CSV file with column headers to match the fields of the items, pay attention to capitalization (Name, Description, Part number, Unit of measure).

   To enter the correct unit of measure, look at the list shown in the dialog above, and enter them as shown.

b. Enter a Status. If you leave status out, it defaults to active - if you specify a status of Pending you can check the values once imported into Onshape. Then, when you are satisfied, you can change the status in the CSV and reimport over the old one with a status of Active.

c. In Onshape, select Create > Import CSV.

d. Select your CSV file and click Open.

e. Once imported, the table is displayed in the Onshape interface and color-coded as to whether the fields are valid or invalid (red for invalid, green for valid). Hover over a row for a hint at what could have gone wrong for invalid rows. If the import was successful, the Import button will be active. Click it to finalize the import.

**Applications**

To navigate to your Onshape Applications settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click Company settings.

This brings you to your Company settings page. From there, click Applications in the list on the left side of the page.

This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store,
navigate to http://appstore.onshape.com (opens in new tab) and sign in with your Onshape account credentials.

Here's a list of frequently asked questions ("App Store FAQs" on page 2613).

Once signed in to the App store, you are able to browse the available apps and make purchases. As the owner of a company or enterprise, you have the ability to purchase multiple seats for other users in your company or enterprise and assign users to those seats.

**Types of apps**

Onshape third-party apps are of the following types:

- **Integrated Cloud Application** - Accessible from within an Onshape document
- **Connected Desktop Application** - Downloaded from the third-party website and installed on your physical machine
- **Connected Cloud Application** - Accessible from a cloud-based service

After purchasing an app, you may have to grant it access to your Onshape documents.

1. While in a document, click the and select Add application. (Select the desired application.)

2. Review the permissions you are about to authorize, then click Authorize application (or Deny if you no longer wish to use the app).

3. To view the permissions an application has on a document, or to give the application access to another document:
   a. Click Share to open the Share dialog and settings for that document.
   b. On the Application tab, select the application from the drop down and click Allow.

4. To revoke an application's access, click the x next to the application name at the top of the dialog.

Purchased apps that are authorized to access your Onshape data are listed in three places in your Onshape documents:

- Applications tab in the user profile (My account page) - Shows all apps you have authorized to access your Onshape data.
Subscriptions tab in the user profile - Shows all apps for which you have a subscription.

On the Add application command from the menu at the bottom of your Onshape window.

**Purchasing and managing multiple seats for an application**

An owner of a company or enterprise is able to purchase multiple seats for an application, to make available to users of their organization. An owner is able to add and remove users at any time for any given application.

**Adding seats during purchase**

To add seats while purchasing an application:

1. Click the app button to purchase or subscribe, as you normally would, in the App store.
2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).
3. Click the purchase button.
4. Enter your Onshape password.
5. A confirmation dialog appears, providing more information about your purchase.
6. Click Close to dismiss the dialog.

**Adding seats and managing users after purchase**

Once you purchase multiple seats for an app, you are able to manage the users who are allowed to use that app. You have the ability to change users who are able to use the app and change the number of seats.

1. On the Applications tab of the Company settings page, select the app you wish to manage.
2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.
   
   A confirmation dialog appears and your changes take effect immediately.
3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the company.

4. To remove a user's access to the application, click the X next to their name in the list.

**User actions on apps**

Through the My account area and Applications tab, users have the ability to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.

- **Control application access to my documents individually through the Share dialog?** - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.

  If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the Share dialog to allow a specific application access to a specific document.

**Release management**

Set up an Onshape release management workflow for your company, including automatic part numbers, if desired. For more information, see [Setting Up Release Management](#).

**Details**

Available to Professional and Enterprise users only; view and edit a company name, description and address (according to roles). The company Owner and user with the Admin role has the ability to edit these details. Other users may view only.
Steps

1. Log into your account.

2. Click your Account user icon in the top-right corner.

3. Click your company name or Company settings.

4. Click Details in the left panel.

5. Make desired changes:
   - Company name
   - Description
   - Address

6. Click Update.
Upgrading to Enterprise

If you have an existing Onshape account, with existing Onshape documents, you are easily able to upgrade to an Enterprise account. To upgrade your existing account to an Enterprise account, contact us (opens in new tab).

Once your account has been upgraded, you receive a Customer Success contact with whom you'll work closely. The following scenarios apply to Professional accounts, specifically with many users, folders, and documents. You may assume the same process with other types of accounts and objects; any differences between account types are noted.

All users are migrated to the new account

All existing account users receive an email with the Enterprise URL. When signing in to Onshape using the Enterprise URL, you automatically land in your Enterprise environment. All members are still members and all admins are still admins.

What happens

All company members, owners, and admins in a Professional plan are moved to the new Enterprise account, and receive assignment of default user permissions. Company owners and admins retain their status in the new Enterprise domain.

Any Free users with whom you have shared documents may be added to the Enterprise as Guest users after the upgrade. These Free users are indicated by an exclamation point beside their names in the Share dialog. Once they are added to the Enterprise, all previous access is reinstated, unless you choose to remove some or all of it. These Share permissions are accessed and modified on a document-basis.

During the upgrade, only documents owned by the company are moved to the Enterprise domain. These documents are now only accessible through the Enterprise domain.

- Any documents owned by individual users (versus owned by the company) must be transferred to company ownership prior to the upgrade in order to be moved to the Enterprise domain.

- Documents not transferred prior to the upgrade will still be accessible via cad.onshape.com and may still be moved to the Enterprise as described in Transfer any non-company owned documents of folders, below.
Folder structures and sharing permissions are all maintained during the upgrade.

- Folders owned by the company are migrated to the new account.
- Company owned documents are migrated to the new account.

**Signing in**

When you sign in from the cad.onshape.com sign in page, however, you are signing in to a personal Professional account; you have access to any privately-owned documents you had previous to upgrading.

To learn how to transfer privately-owned objects to your Enterprise account, see [Getting Started as an Enterprise User](#).

You are able to switch from the Professional account to the Enterprise account through the User menu > Switch to command.

**User account settings**

User account settings remain identical in Enterprise, including emails, preferences, security settings and any custom features added to toolbars. It is the same user account, all the way down to sign in credentials.

**Company-owned documents are migrated to the new account**

All company-owned documents are migrated to the new Enterprise account. The internal shares on those documents remain intact.

**Transfer any non-company-owned documents or folders**

If you individually owned any data in your company account, you are able to transfer that data (documents and folders, for example) to your Enterprise account:

1. Navigate to cad.onshape.com.
2. Sign in with your Enterprise account information.
3. Locate the data you want to transfer.
4. If there are multiple documents, place them in a folder.
5. Right-click on the document or folder and select “Transfer to <enterprise name>”.
6. Return to your Enterprise and move the items to the correct locations.

**Canceling Professional Subscription**
To cancel the Professional subscription and move to the Free subscription:

1. Expand the user menu under your Account user icon and select My account:

2. Select the Subscriptions tab in the list on the left side of the page:

3. If you have more than one subscription, click the subscription you want to cancel.

4. Follow the instructions for contacting Onshape via email or phone call.

Note that on the date specified that your subscription expires, you are downgraded to the Free subscription and maintain access to your pre-existing data. Your private documents remain private, but you will not be able to edit them. Likewise, any private documents shared with you will be view-only (non-editable). You will still be able to view, export, and download your private documents. You may upgrade to the Professional subscription at any time and once again edit your private Onshape documents.

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All rights reserved.
You may also make your private documents public and have edit access to them again.

Managing Your Onshape Enterprise Subscription

This functionality is available on Onshape's browser, iOS, and Android platforms.

All users in an enterprise have access to User account settings and also have view access to the enterprise settings. The administrator has the ability to make changes to the subscription (and app subscriptions) and edit some enterprise/company settings.

This topic is divided into two main sections: the settings you access as a user or member of the enterprise, and the settings you access as an enterprise administrator on behalf of the enterprise.

User account settings

Click your name or Account user icon in the upper right corner on your Onshape window, then click My account to access your Onshape account information.

Profile

To navigate to your Onshape profile, click your Account user icon
in the top right corner of your Onshape window. This opens a dropdown menu, from there, click **My account**.

This automatically brings you to your Onshape user profile.

Onshape automatically records the first and last names you specify during sign up; here you may also enter a personal nickname for display in the system (in the upper right-hand corner of the user interface). Upload a photo to be used next to your user name and on comments, in the Share dialog, and generally wherever lists of user information exists.

**Username** is the name to be used as your Onshape forum name.

**Nickname** is the name seen by other users when you collaborate and is also displayed in the upper right corner of your Onshape window.

Click the Update profile button at the bottom of the page to save your changes.

**Emails**

To navigate to your Emails settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your profile. From there, click Emails in the list on the left side of the page:

This brings you to your Emails settings. Specify up to three email addresses with which to access your Onshape account. One address functions as your primary email, used for all Onshape notifications and communications. Change the primary designation at any time after adding at least one more email address to your account.

All email addresses added to the system must be verified. Check the email address for a verification notice from Onshape.

*Any email address associated with an account (even those not designated as primary) may not be used to create another Onshape account.*
Remove an email from your account by clicking the small "x" next to the email listing (shown above).

You may use any of the verified email addresses on your account to request a reset for a forgotten password.

**Preferences**

To navigate to your Preferences settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.

This brings you to your Onshape profile. From there, click Preferences in the left pane.

You are able to specify your preference for the following settings in Onshape.

**Language**

Select your preferred language from the dropdown (shown in the image below). When you click Save language, Onshape automatically signs you out and you must sign in again to view the language change.

This is an ongoing effort; you may see terms that are not yet translated.
**Startup Page**

Between these four options, you are able to change the start-up page to whichever one you prefer.

![Activity Documents Analytics Action items]

Select your preferred startup page from the dropdown:

![Startup page]

**Units**

Units of measurement and precision used in all your Onshape documents, unless specifically overridden in a dialog (by entering units of choice). You can also specify default units for a specific workspace using that workspace's Document menu

> Workspace units.
Defaults to inch, degree, pound, and three decimal places for units of measure for all documents and encompasses all measurements in Part Studios, Assemblies, and Drawings; all values displayed in sketch dimensions as well as the default input units for all features.

The decimal place settings:

- Are currently available on browser only
- Are currently applied to the feature dialogs, sketch dimensions, and manipulator dialogs
- Work with the Measure tool and Mass properties tool
  - The Measure tool will display values in scientific notation when the display precision is not sufficient.
  - The Mass properties tool will display error in measurement; see "Mass Properties Tool" on page 349 for more information.
- Impacts the display only; values are rounded internally
- Are not used for computation
- Are used internally to determine the number of decimal places to display, regardless of how many places are entered; if more than the specified number are entered, they will be visible when the field is selected for edit.
- Do not affect any external files imported

Overriding default settings

In addition to setting default units for all documents you create (through this Preferences tab), you may also change and specify default units for a specific workspace in a document through the "Document Toolbar and Document Menu" on page 134 in a document.

Despite default settings, Onshape allows you to specify a different unit of measure in any numeric field; the value is converted to the default unit automatically. For example, if the default unit is inches, you may still specify a different unit type (for example "10mm") in a numeric field.

**Mouse controls**

Keep the default settings for mouse mappings, or select a familiar traditional
CAD system’s default settings. These settings also control mouse mappings for Drawings.

Onshape supports 3Dconnexion devices including the SpaceMouse. See your SpaceMouse instructions for information on how to set up your mouse with Onshape.

**Environment profile settings**

Create device profile preferences here, including rendering at high resolution pixel density. This profile may be used on any device.

Associate the profile and the browser/device by selecting a particular profile on a particular machine or browser through this interface. Select a profile for each machine and each browser used on a machine.

- **Match pixel density:**
  - **Automatic (default)** - Onshape determines the resolution needed for rendering.
  - **On** - Render at the resolution of the display.
  - **Off** - Do not render at the resolution of the display. Graphics will be rendered at a lower resolution.

Creating a profile:
1. Click **Create profile**.

2. Enter a name for the profile and click **Create**.

3. Select the preferred setting for matching pixel density.

4. Click **Save profile settings**.

Deleting a profile:

1. Select the profile from the dropdown.

2. Click **Delete profile**.

   Note that this action may not be undone.

3. Click **OK** to confirm the deletion, or **Cancel**.

**Shortcut toolbars**

Customize the shortcut toolbars available for Sketch tools, Feature tools, Assembly tools, and Drawing tools. Select the toolbar to customize; check the tools to include and uncheck the tools to exclude from the menu. If you do not customize the toolbar, each time Onshape is updated, the default toolbar may change. Once customized, your customizations take precedence over any defaults.

![Shortcut toolbars](image)

- There are no limits to the number of tools you are able to include.
- The order of tools in the toolbar is determined by the order in the list (currently).
- Use the S key to invoke the toolbar; use the Esc key to close the toolbar. The toolbar appears near the mouse pointer.

**Toolbars**

Click **Reset to defaults** to restore toolbars to the default settings.

**Drawings**

Set background color of model space for imported DWG and DXF files.

![Drawings settings](image)

**Material libraries**

Create and add custom material libraries, remove unnecessary libraries, and make libraries available to all users within an enterprise. For more detailed information, see "Customizing Parts: Materials" on page 322

**Release management**

When an enterprise is making use of Onshape release management tools, individual users who may be asked to approve releases are able to delegate that approval. This is convenient for planned absences as any user has the ability to specify another user to be notified of pending release candidate approval requests and given permission to act on the request. Upon specification of a delegate, the user or team members named as delegate are notified of the assignment. The user who is delegating the approval responsibility sees a message every time they sign in or refresh an Onshape session:

> You have delegated your approval rights to another user or team. You can change this setting in your account preferences.
Delegated approval requests are not able to be delegated to another user. For example, if User-A delegates approval to User-B, and User-B delegates approval to User-C, User-C will get approval requests that specified User-B, but not approval requests that specified User-A.

To specify a user as your delegate:

1. Open the User menu in the top right corner of the window.
2. Select My account.
3. On the My account page, select Preferences.
4. Scroll down to Release management.
5. Check the box next to Delegate approval to another user or team.
6. Supply the team name or email of the user or users (or click in the box to select from a list of known users and teams).

When something is released, both the user who delegated and the user or users who are delegates receive a notification.

Notifications

To navigate to your Notifications settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a drop down menu, from there, click My account.
This brings you to your Onshape profile. From there, click Notifications in the list on the left side of the page.

This brings you to your Onshape Notifications settings, where you have the option to edit details such as your first name, last name, username, nickname, and biography. In your Notifications settings, there are two sections: Email notifications for shares and comments, and Mobile notifications for shares and comments. Both sections have various radio buttons you may or may not select in order to make notification changes, and a Save notifications button to save your changes.

Email notifications for shares and comments

The top section of your Onshape Notifications page is where you make changes to your Email notifications for shares and comments. In this section, you have three options to choose from:

- **All new shares and comments** - When selected, you will receive email notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

- **Direct shares, mentions, and marked documents** - When selected, you will receive email notifications for shares directly to you, comments mentioning you, and documents you have specifically marked to receive comment emails.

- **Nothing** - When selected, you will not receive any email notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save email notifications button at the bottom of the section, when you have finished editing your Notifications, to save your changes.

Mobile notifications for shares and comments

The bottom section of your Onshape Notifications page is where you make changes for receiving notifications on mobile devices for shares and comments. Similar to the Email notifications for shares and comments section, in this section you have three options to choose from:
• **All new shares and comments** - When selected, you will receive mobile notifications for all new shares, including to your teams or companies, and comments in documents you have access to.

• **Direct shares, mentions, and marked documents** - When selected, you will receive mobile notifications for shares directly to you, comments mentioning you and documents you have specifically marked to receive comment emails.

• **Nothing** - When selected, you will not receive any mobile notifications for shares or comments.

Click the radio button to the left of the option in order to select it. Click the Save mobile notifications button at the bottom of the section when you have finished editing your Notifications to save your changes.

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**Security**

If you have forgotten your password and need it reset, proceed to the Onshape sign in page and click the "Forgot your password?" link to access a page on which you are able to request a password reset link via email.
To navigate to your Security settings, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Security in the list on the left side of the page.

This brings you to your Onshape Security settings. Change your Onshape system password, and also enable (or disable) two-factor authentication.

**Resetting password**

1. Click Change password and enter the old password, the new password, and re-enter the new password.
The list of guidelines leads you through creating a password. Each requirement is marked when your password fulfills the requirement.

2. Click Update password.

2 Factor Authentication

Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.

How it works

Download a two-factor authentication app (like Google Authenticator) to your phone and set it up with Onshape through the Onshape user interface. This enables the app to generate a one-time code that Onshape is able to recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You are able to allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign in to Onshape from a specific device, you won't need a 2FA code to sign in on that device for 30 days.

Enabling and using two-factor authentication

1. Download a two-factor authentication app to your device.
Google Authenticator is one example.

2. Sign in to your Onshape account.
3. In the menu under your username, select My account.
4. In the list on the left side of the page, click Security.
5. To the right of Two-factor Authentication, click Enable.
6. Click [Set up two-factor authentication].
7. Confirm password.
8. Click [OK].

Configuring the app to work with Onshape

Continuing from the instructions above:

1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface.

Once registration is complete, the phone app will list a code for each registration you create. It is these codes that you enter into Onshape when presented with the 2FA sign in page.

If you are not able to use the QR code, click the enter this text code link provided in the Onshape interface to obtain a code.

2. Enter either the six-digit code that the 2FA app generates or the code supplied by Onshape.
3. Click Enable.
4. When the recovery codes are displayed, copy them to a safe place; you need access to them in the event you do not have your phone or the authentication app.
5. Click OK.

Onshape provides you with 5 active recovery codes at a time. Keep these codes in a place accessible to you separate from your device or the authentication app.

Onshape will not be able to help you should you delete the app or lose your phone.

Note that you are able to generate these Recovery codes at any time through the Onshape interface, but only the most recently generated series are active at any one time. Once you use a code it is no longer valid. When you generate a new list of codes, all previous codes (used or unused) become invalid.

**Signing in to Onshape with code**

When two-factor authentication is enabled, Onshape prompts you for a code upon sign in:

1. After you enter the password to your Onshape account, you are prompted for the authentication code.

2. Open the two-factor authentication app on your device to view the code; enter the code in Onshape.

3. Click Verify.

In the event that you don't have access to the app, click the Enter a two-factor recovery code link to enter one of your current recovery codes.

**Disabling two-factor authentication in Onshape**

You may disable (and re-enable) two-factor authentication at any time.

1. On the Security tab of the User Profile page in Onshape click Manage, and then Disable:

2. Confirm password.

3. Click OK.
Replacing a device with 2FA enabled

Should you need to replace a device on which you have 2FA enabled for Onshape:

1. Before replacing the device, disable 2FA through the Onshape interface.
2. Enable 2FA once the new device is online.

Note that Onshape doesn’t support the Replace 2FA option.

Devices

To navigate to your Devices settings in Onshape, click on your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Devices in the list on the left side of the page.

This brings you to your Onshape Devices settings, a list of all mobile devices associated with and authorized to use this account. Once you access your Onshape account on a mobile device, that mobile device is listed here.

To remove a device from the list, click Forget on the right of the window.

Applications

To navigate to your Onshape Applications settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there click My account.

This brings you to your Onshape profile. From there, click Applications in the list on the left side of the page.

This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store,
navigate to http://appstore.onshape.com (opens in new tab) and sign in with your Onshape account credentials.

Here's a list of frequently asked questions ("App Store FAQs" on page 2613).

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3. To view the permissions an application has on a document, or to give the application access to another document:
   a. Click to open the Share dialog and settings for that document.
   b. On the Application tab, select the application from the drop down and click Allow.

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Subscriptions tab in the user profile - Shows all apps for which you have a subscription.

On the Add application command from the menu at the bottom of your Onshape window.

Purchasing and managing multiple seats for an application

An owner of a company or enterprise may purchase multiple seats for an application, to make available to users of their organization. An owner may add and remove users at any time for any given application.

Adding seats during purchase

To add seats while purchasing an application:

1. Click the app button to purchase or subscribe, as you normally would, in the App store.

2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).

3. Click the purchase button.

4. Enter your Onshape password.

5. A confirmation dialog appears, providing more information about your purchase.

6. Click Close to dismiss the dialog.

Adding seats and managing users after purchase

Once you purchase multiple seats for an app, you are able to manage the users who are allowed to use that app. You are able to change users who may use the app and change the number of seats.

1. On the Applications tab of the Enterprise settings page, select the app you wish to manage.

2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.

A confirmation dialog appears and your changes take effect immediately.
3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the enterprise.

4. To remove a user's access to the application, click the X next to their name in the list.

**User actions on apps**

Through the My account area and Applications tab, users are able to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the + icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.

- **Control application access to my documents individually through the Share dialog?** - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.

  If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the [Share dialog](#) to allow a specific application access to a specific document.

**Integrations**

To navigate to your Onshape Integrations settings to set up access to your Dropbox, Google Drive, or Microsoft OneDrive account, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Integrations in the list on the left side of the page.
This brings you to your Onshape cloud-source settings. You have the ability to grant Onshape access to your Dropbox, Google Drive, or Microsoft OneDrive accounts (separately), and also remove that access by clicking the red Remove button on the right side of the page.

To grant access for Onshape to import documents from (and export documents to) a Dropbox, Google Drive, or Microsoft OneDrive account, click the appropriate "Add" button and follow the steps. You'll need the credentials to your account in order to proceed through the wizard. For example:
Early Visibility

To navigate to your Onshape Early visibility settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click My account.

This brings you to your Onshape profile. From there, click Early visibility in the list on the left side of the page.

This brings you to your Onshape Early visibility settings. The Onshape Early Visibility program offers you an opportunity to test functionality that is still under development.
Due to the nature of features in development, we recommend you create specific documents for use with any Early Visibility feature. (Feel free to copy existing documents for this purpose.)

Please do not use documents you create under the Early Visibility feature for business critical or production use.

**Signing up**

1. Click ![Add](add.png) to the right of the feature of interest (or features; you have the ability to request access to multiple features).

   When you click Request access, you are directed to an End User license agreement page.

2. Read the agreement.

3. Click ![Accept](accept.png) if you agree and wish to continue.

   Click ![Cancel](cancel.png) if you are no longer interested.

Clicking Accept sends a message to Onshape that you are interested in a particular feature. Your request is reviewed and when approved, you receive an email confirmation.

**Subscriptions**

To view the details of your subscription, click the **Subscription** tab in the left pane.

This page presents the details of your subscription including:

- Number of users on the account and being paid for, including the distinction between Full and Light users.

- The details of the subscription type as well as credit card information for the account

- Status of the account: Active or Inactive

**Payment Options**

To navigate to your Onshape Payment options settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu,
from there, click My account.

This brings you to your Onshape profile. From there, click Payment options in the list on the left side of the page.

This brings you to your Onshape Payment options settings. If you are the owner of the account, you have the ability to change credit card information, or remove a card from the listing. Note that a credit card may be removed from the account only if it is not associated with a subscription:

To change the credit card, click Change and enter the new card information.

Changing the enterprise owner and credit card

These actions must take place before the renewal date.

This process will only work if the email address of the new Owner does not exist in the Onshape database yet. If you aren’t sure if the email exists, please let your Customer Success Manager know and they will confirm for you.

1. The current Owner of the Enterprise must log into their Onshape account.

2. Once logged in, click your name in the top right corner.

3. Select “My Account” from the drop-down menu.

4. Click “Emails” in the left column.

5. Add the email address of the user you would like to be the new Owner as an additional email address on your account (directions on how to do that may be found above under "Emails" on page 2552).

6. When you are ready to make the switch, make the Primary Email Address the email of the new Owner.
7. Once the new email has been made the Primary, that user is now considered the Owner and they can:

a. Change the password if they want.

b. Delete the old Owner’s email address (now listed under Other Emails).

c. Remove any users, other than the Owner, they don't want to move forward with at their renewal time.

If you want to change the owner but you use a purchase order, call your Onshape Customer Success Manager and ask for their assistance.

Payment History

This area lists a chronological history of all payments made for the account. Click View to access a print-friendly invoice.

Only Enterprise account Owners and Admins are able to see this information.

Setting preferences on iOS

To access the area in which you change account preferences for all users:

1. Tap the button hamburger menu in the upper-left corner of the screen, then tap My account. This brings you to your account settings page:
2. From there, tap Preferences to open your account Preference settings page.

There are a few preference settings you are able to make on behalf of the Enterprise account:

- To require users to use specific, approved drawing templates that you have loaded/created in Onshape.

- To enable an enterprise-owned, shared materials library to be accessible by all enterprise users.

For information and instruction on creating your own drawings templates and making them available to your users, see the Learning Center's Technical Brief [Best Practices for Creating Native Onshape Drawing Templates](#).

For information on adding custom materials libraries, see [Customizing Parts: Materials](#).

**Enterprise account settings**
Click your name or Account user icon in the upper right corner on your Onshape window, then click **Enterprise settings** to access your Onshape account information.

**Users**

The enterprise admin has the ability to adjust the number of users paid for with the subscription, as well as add and remove users from the subscription. This page also displays the number of Full and Light users currently in the enterprise in relationship to how many seats for each have been purchased.

To make it easier to find specific users you can search for users as well as sort the user list. Searching for users will find matches or partial matches with user names and email. This can be used to find all users of a specific domain (ex. @ptc.com). Sorting can be done on the 'Name', 'User type', or 'Date added' columns by clicking on the corresponding header.

For more information about types of users in an enterprise account, see the "Enterprise" on page 2224 topic.

Navigate to Enterprise settings under the User menu, then select Users.

For information on adding users, see the "Adding and Administering Users" on page 2264 topic.

**Teams**

You have the ability to create teams in order to group users together for the purpose of making sharing more efficient; once the team is created, you are able to select the team name instead of entering many users' individual email addresses during a Share operation.
It is not required that the members of a team have anything in common; not even an Onshape subscription.

One user creates a team (thereby becoming the initial admin of the team) and then adds other users to it, assigning either a user role or an admin role to each team member. Members receive notification emails when they are added and removed from a team, and users may belong to more than one team at a time.

Any user has the ability to share a document with an enterprise-owned team, even if they are not a member of the team. However, in order to share a document with a team external to the company or enterprise, the user must be a member of that team.

Sharing a document with a team does not give any team member additional permissions on the document than the owner/creator of the document allows during the Share operation.

At any point, the admins of a team may remove any member from the team, thereby removing any Share permissions previously made through the team. Any Shares made on an individual basis remain in place, as well as the permission they grant.

Team members are able to remove themselves from a team, unless they are the last admin member of the team. (A team must have at least one admin.) When a member is removed from a team, any document shared with that user through the team becomes unshared and removed from their Documents list.

A team admin may delete the team at any time. Upon deletion of the team, all documents shared with the team become unshared from the team members and removed from their Documents lists.

As with all sharing operations, the following permissions is able to be assigned during the Share operation:

- **Can view** - Permission to open for read-only access
- **Can edit** - Permission to open and edit (make changes)

Following are instructions for:

- Creating teams and adding members
- Removing members and admins
Deleting a team

Additionally, see information about [Sharing and assigning permissions to documents](#).

Creating teams and adding members

1. Expand the menu under your user name/Account user icon in the top right corner of the page and select Enterprise settings:

2. On the page that appears, select Teams from the left panel and click Create:

3. Enter a name for the team, and a description, or statement of purpose:
4. Click Create team:

![Create team screen](image)

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).

Use the Search bar to search for team members.
Note that you are able to return to this page and change a team member's role.

6. Click Add.

7. When finished adding team members and assigning roles, click the arrow to the left of the team name (at the top of the page) to return to the Accounts page.

8. You see the new team listed on the Teams page.
Creating a team also adds a filter for that team in each member's Documents filters on their Documents page. These filters list all documents shared with a particular team.

**Removing members and admins**

To make it easier to find specific teams, you can sort by the name column (click 'Name' to toggle the sort alphabetically). You can also enter search terms for team names.

Members may remove themselves from a team, and any member with an Admin role may remove users including themselves as long as they are not the only administrative user left. Users removed from a team receive an email notification and are removed from the Share list of any document shared with the team. Those documents are removed from the user's Documents page.

1. Expand the menu under your user name or Account user icon in the top right corner of the page and select My account (for Standard and Free subscriptions) or Company / Enterprise settings (for Professional and Enterprise subscriptions).
2. Select Teams in the left panel to access the list of teams of which you are a member.
3. Select the team in the list from which you wish to remove yourself or another member.
   - To remove yourself (as a member): Click the Leave team button.
   - To remove yourself (as an admin): Click the X to the left of your name (Note this only works if there is another admin still on the team).
   - To remove another user: Click the X to the left of the user name (Note this only works if you are an admin).

**Deleting a team**

Any Admin of the team are able to delete the team at any time. This immediately removes the share permissions for all documents shared with the team and removes the documents from each member's Documents list.

**Global permissions**

Global permissions control user access to Enterprise-level operations and
information. Only the enterprise administrator or another user with administrator rights are able to grant Global permissions to users. The user who creates the Enterprise account becomes the first administrator of the account and is assigned the *Enterprise administrator* global permission.

Global permissions dictate what access to (and control over) the enterprise account is available to specific users. These permissions are for users who should have some control over the enterprise account, and not intended for non-administrative users. Typically, what non-administrative users are able to do to a document is determined by the properties of Projects and Permission schemes.

For detailed information about global permissions, see the "Understanding Global Permissions" on page 2271 topic.

**Authentication**

Onshape highly recommends taking advantage of our two-factor authentication functionality. Two-factor authentication (2FA) allows you to configure your Onshape account to require more than a single password to sign in. Using one password to sign into a website makes you more susceptible to security threats because one piece of static information may be easy to guess or acquire. With 2FA, a second piece of information is required, and that second piece of information is generated dynamically during the sign in process, and may be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.

**How it works**

Download a two-factor authentication app (like Google Authenticator) to your phone and set it up with Onshape through the Onshape user interface. This enables the app to generate a one-time code that Onshape is able to recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You are able to allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign in to Onshape from a specific device, you won’t need a 2FA code to sign in on that device for 30 days.

**Enabling and using two-factor authentication**
1. Download a two-factor authentication app to your device.
   Google Authenticator is one example.

2. Sign in to your Onshape account.

3. In the menu under your username, select My account.

4. In the list on the left side of the page, click Security.

5. To the right of Two-factor Authentication, click Enable.

6. Click Set up two-factor authentication.

7. Confirm password.

8. Click OK.

Configuring the app to work with Onshape

Continuing from the instructions above:

1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface.

   Once registration is complete, the phone app will list a code for each registration you create. It is these codes that you enter into Onshape when presented with the 2FA sign in page.

   If you are not able to use the QR code, click the enter this text code link provided in the Onshape interface to obtain a code.

2. Enter either the six-digit code that the 2FA app generates or the code supplied by Onshape.

3. Click Enable.

4. When the recovery codes are displayed, copy them to a safe place; you need access to them in the event you do not have your phone or the authentication app.
5. Click OK.

Onshape provides you with 5 active recovery codes at a time. Keep these codes in a place accessible to you separate from your device or the authentication app.

Onshape will not be able to help you should you delete the app or lose your phone.
Note that you are able to generate these Recovery codes at any time through the Onshape interface, but only the most recently generated series are active at any one time. Once you use a code it is no longer valid. When you generate a new list of codes, all previous codes (used or unused) become invalid.

Signing in to Onshape with code

When two-factor authentication is enabled, Onshape prompts you for a code upon sign in:

1. After you enter the password to your Onshape account, you are prompted for the authentication code.

2. Open the two-factor authentication app on your device to view the code; enter the code in Onshape.

3. Click Verify.

In the event that you don't have access to the app, click the Enter a two-factor recovery code link to enter one of your current recovery codes.

Disabling two-factor authentication in Onshape

You may disable (and re-enable) two-factor authentication at any time.

1. On the Security tab of the User Profile page in Onshape click Manage, and then Disable:

2. Confirm password.

3. Click OK.

Replacing a device with 2FA enabled
Should you need to replace a device on which you have 2FA enabled for Onshape:

1. Before replacing the device, disable 2FA through the Onshape interface.
2. Enable 2FA once the new device is online.

Note that Onshape doesn't support the Replace 2FA option.

**Project roles**

Projects enable you to group documents and apply permissions to those documents for individual users and groups of users (called Teams). The permissions are assigned at the Project level through the association of a Permission scheme along with the specification of a role map (one per Project) that contains a list of user/role pairs. (The user may be an individual user or a Team of users.)

The role map plus the Permission scheme determines the level of access users (or Teams) have to the project and the documents within. Permission schemes may be edited at any time and the edits directly and immediately affect any project that refers to that permission scheme. Editing Permission schemes may render some entries in a role map ineffectual or change the permission level of users or Teams in the Project's role map.

One document is only able to belong to one Project at a time and documents may be moved into and out of Projects (by users with Edit permission on the Project).

Documents (and Folders) may be created inside of a Project, moved into a Project, moved out of a Project, and deleted altogether.

When a project is created, the creator is automatically placed into the Project Administrator role, which grants full permission to that user. A project must always have a project admin (or someone with a similar full-permission role).

For detailed instructions about creating and using projects, see the "Understanding and Managing Projects" on page 2293 topic and the "Understanding and Administering Project Roles and Permission Schemes" on page 2276 topic.

**Permission schemes**

Project roles and Permission schemes work in tandem to apply access permission to documents grouped together within a Project. A Project references only one Permission scheme, which is comprised of Project roles specifying the permissions.
For detailed information about permission schemes and project roles, see the "Understanding and Administering Project Roles and Permission Schemes" on page 2276 topic.

**Custom Properties**

The properties on this page define the metadata available for Onshape entities and apply only to documents that are owned by the enterprise. Enterprise owners and admins are able to create new properties, activate them for use by users, and also deactivate properties.

To navigate to your Onshape Custom properties settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click Enterprise settings.

This brings you to your Onshape Enterprise settings Users page. From there, click Custom properties in the list on the left side of the page.

This brings you to your Onshape Properties settings. Available for Professional and Enterprise users only, Onshape provides access to the metadata definitions for Onshape objects. These definitions drive the data displayed in the Properties dialogs and bill of materials for Onshape objects. Professional and Enterprise users are able to view these metadata definitions, and Admins and Owners are able to create new custom properties for use in enterprise-owned documents, as well as provide Display names for existing Onshape properties.

Owners and users with the Admin role are able to add, modify and retire custom properties. Onshape metadata definitions may be made Inactive (retired) but may not be edited.

Users with permission to create and edit custom properties see a Create button at the top of this page:
Creating and activating new custom properties

On the Properties page:

1. Click Create custom property.

2. Specify a name for the property. The name should be unique among Onshape and custom properties. When testing for uniqueness, Onshape uses the Company name and the name of the property.

3. Select a property type:
   - Text
   - Boolean
   - Integer
   - Double
   - Date
   - List
   - User

4. The initial Publish state is Pending (not yet available to users). When you want this property to be available to users, select Active (available to users and values...
entered are recorded in the database). To retire this property from use, select Inactive. Inactive values are available only through the Onshape API.

5. Supply the rest of the property’s attributes, a red asterisk indicates a required field. Depending on the property type you selected above, different attributes may be available.

a. **Default value** - Enter a default value here, if desired. This value serves as the default if the user doesn’t specify a value.

   When entering values for the User property type, it must be a valid Onshape user’s email who already belongs to the company in which the property was created.

b. Enter an optional Description for the property.

c. Indicate where the property may be edited and whether it is required:
   - **Required** - Indicates that the property value is required.
   - **Edit value in version** - Indicates that this property value may be edited in a document version; allow the user to supply a value for this property in any document version. Also indicate if the value should be edited only through the API and not through the Onshape user interface.
   - **Edit value in workspace** - Indicates that this property may be edited in a workspace; allow a user to supply a value for this property in any document workspace. Also indicate if the value should be edited only through the API and not through the Onshape user interface.

6. Enter optional **validation** criteria for properties, including those that can have a pattern:

a. Check Multiline if the value can have multiple lines of text (for example, a Description may require multiple lines of text).

b. Enter a minimum length and/or a maximum length, if desired.

c. Indicate the pattern in the Pattern field, such as any regular expression including:
   - `[A-Z]+` which requires 1 or more uppercase alphabetical characters
   - `[0-9]+` which requires 1 or more numeric characters
   - `[a-z]+` which requires 1 or more lowercase alphabetical characters
• ONS-[0-9]+ which requires the prefix ‘ONS-’ followed by 1 or more numeric characters. You could put the required prefix in the Default value attribute so it appeared automatically. Users receive an error notification specifying the required pattern if the value is invalid.

7. Select a category (or categories) in which the property will be available. The category must already be defined through the Categories tab by an administrator of the company or enterprise. You may select more than one category, and also remove a category by clicking the X to the far right of the category line. For more information on Categories, see below.

8. To make the property available to all users in the Company, make sure the Publish state is set to Active.

9. Click Create.

10. A message appears reminding you that if you make this property active, all users in the company or enterprise will be able to see it. Click Confirm publish to continue to activate the property. You can publish later instead.

You are also able to use FeatureScript to create custom properties. These custom properties are also displayed on the Custom properties page, by name and property ID.

Modifying properties

Onshape metadata and custom properties is only able to be edited by Owners and users with the Admin role.

On the Properties page, you have the ability to:

• Search for properties. Use the search box to enter a property name or partial name; check the box to search for inactive properties.

  The list presents Onshape metadata properties first, then custom properties in alphabetical order.

• Click a property name to open it for editing. You may make any changes desired to a property with a Publish state of Pending, including deleting that property. The only changes you are able to make to a property with a Publish State of Active is to
change the state to Inactive and change the Display name. Inactive properties are not visible in any Properties dialogs, but are still associated with the objects. Any other changes made to an Active property are immediately available and effective upon Save. You can make an Inactive property Active again.

See "Creating and activating custom properties" above, for information on modifying fields.

**Using properties**

When using the User custom property type, when a user fills the field, the dropdown will list the current user at the top of the list. However, searching on the current user will yield no results.

**Retiring properties**

When a property becomes obsolete, or an error in the definition is discovered after the property was made Active, you are able to retire the property. Retiring a property removes it from all Properties dialogs but keeps the property associated with the objects and preserves the data in the database. To retire a property:

On the Custom properties page:

1. Search to locate the property, if necessary.
2. Click the property name to open for editing.
3. Select the dropdown for Publish state and select Inactive.
4. Click Save changes.

Inactive properties may be made Active again.

**Categories**

Creating categories in Onshape provides the ability to extend the properties of the standard Onshape object types in the system to include more targeted and relevant metadata to be applied based on design, engineering, and manufacturing processes.

Before creating categories for the first time, you can prepare by checking your PLM, or other appropriate system, for important metadata you want to be able to represent within Onshape.

Categories work in conjunction with custom properties, to group those custom properties into reasonable and useful information. Once categories are defined, you can
see the custom properties within your Properties dialogs for all Onshape objects to which the category is applied.

An example

An enterprise manufactures a product that is a combination of a part designed and produced in-house and parts ordered from a third party vendor. When working with off-the-shelf parts from a vendor, you may care about only a handful of properties like Vendor name, Vendor part number, and perhaps things like weight and cost.

However, when working with parts manufactured in-house, the list of properties will likely be entirely different. These properties may include information like Start date, Approved date, Manufacturing method, Length, and more.

This is where Onshape categories come into play. With categories, you can assemble the properties you really care about and need, so they are grouped together in all Properties dialogs within Onshape as "newly created category". Once this is selected, the properties within the Category are listed first in the dialog, making them easier to find and fill out.

**Creating a category**

To create a category:

1. Navigate to Enterprise settings.

2. Select **Categories**. Onshape has standard existing categories for Onshape objects, including: Global, Part, Assembly, Drawing, Part Studio, File, Application, Version, Workspace, and Item. The Scope of these (and all) categories are listed in the next column. The Scope refers to the Onshape objects the properties within the categories apply to.
3. Click the Create category button at the top of the page.

4. Select a **Scope**, that is, the types of Onshape objects to which this category may be applied. Note that selecting a Scope is not required. If no scope is assigned, the category will not be selectable but can still be inherited by other categories which have an assigned scope.

5. Assign a **Name** for the category.

6. Select a **Publish state**: Active = visible by and available to all users; Pending = Saved but unavailable for use; Inactive = No longer available for use. You can mark a category as Pending, then go back later and make it Active.

7. Supply a **Description** of the category, if desired.

8. Select any **Parent categories**, existing categories from which you want this new category to inherit properties. Assigning a parent category to a category carries properties assigned to the parent category over to the category. This is useful when the new category uses all of the properties that are already assigned to the parent category, but requires a few extra properties specific to the new category.

9. Select the **Properties** (custom properties), that you want this category to include. These are the properties that will display whenever the category is selected for an object type.
10. Click **Create** to create the new category.

The new category is listed on the main Categories page. Categories are also available in the Advanced search dialog, in the Add criteria dropdown.

Sub-categories and inheritance

Categories can also inherit from a parent category, if you wish. These nested categories inherit the properties of the parent, and also allow you to add properties that are unique to that sub-category. One example for using inherited properties would be a category for parts that are manufactured in-house. This parent category could have all the common properties across multiple parts like part number, release state, and date approved. You can then create sub-categories under that "in-house" category for each type of part you manufacture, like: sheet metal, injection molded, machined, and welded, and then have those categories contain properties unique to those types of products and their processes.

The only difference when creating a sub-category, is to select a Parent category:
The Parent category of the new category being created, above, is "In-house manufacturing" and its properties are listed as "Inherited properties" under Properties.
You would also select specific properties for the sub-category, shown above as Classification and Size.

Once you click Create, the sub-category is listed on the Categories page. You can view it in the list, just as any other category, and you can also expand the parent category and view it there:

The Welded category shown indented below "In-house manufacturing" is the same category shown last in the list, above.

You can select one or the other of the entries for a sub-category and the other entry is also highlighted; this makes subcategories easier to find, as shown below:

To view the list of categories without sub-categories listed, place a check in the Hide sub categories in list checkbox at the top of the interface.

**Items**

Here you can add items that you will want to add to your bill of material (see "Bill of..."
Materials" on page 1509 for information) but that do not require being modeled (non-geometric entities). Adding those items here allows you to make use of them in any company or enterprise-owned documents.

Before creating items for the first time, you can prepare for it by ensuring you have your desired categories of materials or supplies already present in your enterprise settings.

To check your company categories:

1. Navigate to Custom properties under Enterprise settings.
2. Find the Classification property, click the name to edit it.
3. Under Existing list values you can browse the classifications that have already been added.
4. To add new classifications (for example, "Adhesives"), enter the names in the "Add new list values" text box:

```
Existing list values ?

Adhesives
Instructions
adhesives, glue
adhesives, resin

Add new list values (Enter one list value per line) ?
```

*The classification names must be entered one value per line, and each value may contain commas (as seen in the "Existing list values" box, above)*

5. To save your new Classification values, click Save changes at the bottom of the page.
To create items for company or enterprise-owned documents to make use of:

1. Click **Create**.

2. Click **Item** to create an individual item:
a. Enter information for the metadata, such as: Name, Description, Category (the Category must have previously been added through the Custom properties for Category; see above) Part number, and Revision.

b. Supply any other pertinent specifications, if necessary, such as Vendor, Project, and Product line.

c. Select Classifications, if desired. (You can add list values to your enterprise Classification property in order to see those values in this Classification list.)
d. Assign a Publish state: Pending = not yet published; no users can access the item, Active = immediately available to users, Inactive = unavailable to users, retired

e. Click Create.

3. To import a comma-separated value list (CSV):

a. Create the CSV file with column headers to match the fields of the items, pay attention to capitalization (Name, Description, Part number, Unit of measure).

To enter the correct unit of measure, look at the list shown in the dialog above, and enter them as shown.

b. Enter a Status. If you leave status out, it defaults to active - if you specify a status of Pending you can check the values once imported into Onshape. Then, when you are satisfied, you can change the status in the CSV and reimport over the old one with a status of Active.

c. In Onshape, select Create > Import CSV.

d. Select your CSV file and click Open.

e. Once imported, the table is displayed in the Onshape interface and color-coded as to whether the fields are valid or invalid (red for invalid, green for valid). Hover over a row for a hint at what could have gone wrong for invalid rows. If the import was successful, the Import button will be active. Click it to finalize the import.

Applications

To navigate to your Onshape Applications settings, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click Enterprise settings.

This brings you to your Enterprise settings page. From there, click Applications in the list on the left side of the page:
This brings you to your Onshape Applications settings. Onshape offers third-party applications for use with your Onshape account. To access the Onshape App Store, navigate to http://appstore.onshape.com (opens in new tab) and sign in with your Onshape account credentials.

Here's a list of frequently asked questions ("App Store FAQs" on page 2613).

Once signed in to the App store, you are able to browse the available apps and make purchases. As the owner of a company or enterprise, you have the ability to purchase multiple seats for other users in your company or enterprise and assign users to those seats.

**Types of apps**

Onshape third-party apps are of the following types:

- **Integrated Cloud Application** - Accessible from within an Onshape document
- **Connected Desktop Application** - Downloaded from the third-party website and installed on your physical machine
- **Connected Cloud Application** - Accessible from a cloud-based service

After purchasing an app, you may have to grant it access to your Onshape documents.

1. While in a document, click the + and select Add application. (Select the desired application.)
2. Review the permissions you are about to authorize, then click Authorize application (or Deny if you no longer wish to use the app).

3. To view the permissions an application has on a document, or to give the application access to another document:
   a. Click [Share] to open the Share dialog and settings for that document.
   b. On the Application tab, select the application from the drop down and click Allow.

4. To revoke an application's access, click the x next to the application name at the top of the dialog.

Purchased apps that are authorized to access your Onshape data are listed in three places in your Onshape documents:
- Applications tab in the user profile (My account page) - Shows all apps you have authorized to access your Onshape data.
- Subscriptions tab in the user profile - Shows all apps for which you have a subscription.
- On the Add application command from the + menu at the bottom of your Onshape window.

### Purchasing and managing multiple seats for an application

An owner of a company or enterprise is able to purchase multiple seats for an application, to make available to users of their organization. An owner is able to add and remove users at any time for any given application.

### Adding seats during purchase

To add seats while purchasing an application:

1. Click the app button to purchase or subscribe, as you normally would, in the App store.

2. On the confirmation page is a field to enter the number of users you are paying for (include yourself, if appropriate).

3. Click the purchase button.

4. Enter your Onshape password.
5. A confirmation dialog appears, providing more information about your purchase.

6. Click Close to dismiss the dialog.

**Adding seats and managing users after purchase**

Once you purchase multiple seats for an app, you are able to manage the users who are allowed to use that app. You have the ability to change users who are able to use the app and change the number of seats.

1. On the Applications tab of the Enterprise settings page, select the app you wish to manage.

2. On the page that appears, increase or decrease the number of seats you wish to pay for: enter a value in the field and click Update.

   A confirmation dialog appears and your changes take effect immediately.

3. In the Add users field, enter the email addresses of the users who should have access to the application, and click Add. Note that these users must already be members of the enterprise.

4. To remove a user's access to the application, click the X next to their name in the list.

**User actions on apps**

Through the My account area and Applications tab, users have the ability to take action on the apps used with their Onshape account.

- **Revoke** - Remove an app's access to Onshape data. This does not remove the app from Onshape. If you use this app again, you will be prompted to allow access to your Onshape data.

- **Authorize Application** - Authorize the purchased app to access your Onshape data. You see this option in an Onshape document: Click the 📜 icon > Add Application > application-name. A new tab opens and becomes active in your Onshape document.

- **Control application access to my documents individually through the Share dialog?** - Some applications prompt you to allow the app access to all your Onshape documents. If you would like to have control on a per document basis, turn this option on.
If you have granted access prior to turning this switch on, that access is still granted. If you turn this switch off, all access previously granted is still granted. When this switch is on, you must use the Share dialog to allow a specific application access to a specific document.

**Release management**

Set up an Onshape release management workflow for your enterprise, including automatic part numbers, if desired. For more information, see "Release Management" on page 2169.

**Preferences**

To navigate to your Onshape enterprise preferences, click your Account user icon in the top right corner of your Onshape window. This opens a dropdown menu, from there, click Enterprise settings.

This brings you to your Onshape enterprise settings Users page. From there, click Preferences in the left pane.

This brings you to your Onshape Preferences settings, where you have the ability to require that users use approved drawing templates, make use of enterprise-owned material libraries in their own accounts, or show a view only toolbar to all users of a document with view only permissions:
For information on drawing templates, see "This functionality is currently available only on Onshape's browser platform." on page 1753.

For more information on custom material libraries, see "Customizing Parts: Materials" on page 322.

For information on the simplified user interface, see "Using the View Only Toolbar" on page 2093.

For information on how to edit the watermark in your drawing, see Setting Up Release Management.

Documents

Use this field to enter regex (regular expressions) by which to enforce document name syntax (or prevent duplication of names) enterprise-wide. You can enter any valid java regex. For example:
To prevent any duplication of document names, enter ^.*$ in the field and click Save.

To ensure that all document names are unique unless the names starts with 'test', enter ^(?!test).*$

To enforce that all document names contain only numbers, enter ^[0-9]+$.

If you enable this and there are already existing documents with duplicate names, you will see a message to that effect when you open the document. You always have the option to change the document name to a unique name; Onshape will not change existing document names automatically.

**Enterprise custom features**

Administrators of an enterprise can add custom features to the toolbar for all enterprise users if this box is checked. Checking this box activates the display of the Add enterprise custom features button on the Part Studio toolbar for all administrators of the enterprise.

To add custom feature icons to your enterprise and make them available for all users:

1. Check the box as described above to enable this ability for all administrators of an enterprise.

2. Navigate to a Part Studio in a document owned by the enterprise.
3. Click to open the dialog for selecting custom features:

4. In the dialog, locate the document in which the desired custom feature is defined:
   a. Search in the **Current document** or **Other documents**, by selecting the tab at the top of the dialog.
   b. **FeatureScript samples** - This filter lists all Onshape-supplied documents containing FeatureScript samples for you to try.
   c. **Onshape standard filters** - Use these filters as you do on the Documents page to locate a specific document.
   d. **Search box** - Enter the name of a document or paste the URL of a document containing FeatureScript (usually acquired through a Share action).
Custom features are linked from specific versions of other documents; the latest version is selected by default. If there is no version, you can request that the document owner (or someone with write access) create one.

Clicking on any document name displays the custom features defined in it.

5. Select the top level icon to insert all custom features inside it (each represented by its own icon on your toolbar), or select one feature.

The custom feature icon appears on the Feature toolbar.

To remove the custom feature before closing the dialog, select the custom feature in the dialog again. (This toggles the custom feature in and out of the toolbar.)

6. If there is more than one version of the document, the latest version is displayed by default. Click \( \text{מידד} \) to access the version graph and select a different version.

7. Click the X in the upper-right corner of the dialog to close it.

When you, as the administrator, add custom features to your Feature toolbar, the custom features are added to enterprise users' toolbars upon signin to their account, for all users with Link permissions to the document containing the custom features.

When there is a newer version of the tool available, it will show as out-of-date for you (the admin), but not for other users in the enterprise.

If a feature in a user's Feature list corresponds to a tool added by the admin, the feature will not show as out-of-date (even if the tool is) but if it is not from the same version as the tool it will show as out-of-sync \( \text{מידד} \).

The Reference manager Update all command will update out-of-sync features to the version in the toolbar, even if that version is lower than the latest version or lower than the version the feature is currently from.

Any user can move custom features around when customizing the toolbar, the same as standard feature tools. The one restriction is that standard tools can't be added to the toolsets where the custom features are initially added, and enterprise custom tools can't be added to standard tools toolsets.
Adding custom features on Android

On your Android device, as an administrator of the enterprise:

1. Sign in to the enterprise domain.
2. Open the Part Studio feature toolbar.
3. Click on the "Add enterprise custom features" option.
4. Navigate to the custom feature and select it.

Selecting the custom feature adds it to the toolbar for all enterprise members that have Link permission to the document that contains the custom feature.

**Integrations**

Available to Enterprise administrators only; indicate whether to allow users to integrate with Google Drive, Dropbox, or Microsoft OneDrive, and if so, which users may do so.

Integration settings

Select the corresponding radio button to:

- Allow users to integrate with Google Drive, Dropbox, and Microsoft OneDrive
- Do not allow integration with either service
- Allow integration with either Google Drive, Dropbox, or Microsoft OneDrive (select the allowed integration)

Integration users

Select the corresponding radio button to:

- Allow all users to integrate with the above-specified service.
- Allow only the specified users and/or teams to integrate with the above-specified service.

**Details**

Available to Professional and Enterprise users only; view and edit a company or enterprise name, logo (enterprise only), description and address (according to roles). The account Owner and user with the Admin role has the ability to edit these details. Other uses may view only.
Steps

1. Log into your account.

2. Click your Account user icon in the top-right corner.

3. Click your company name or Company / Enterprise settings.

4. Click Details in the left panel.

5. Make desired changes:
   - Logo (enterprises only).
   - Enterprise name
   - Description
   - Address

6. Click Update.
Subscriptions and Payment FAQs

How much do the Onshape subscriptions cost?

Onshape's Professional subscription is $175/month/user, billed annually. The Standard subscription is $125/month/user, billed annually. Onshape's Free and Education subscriptions are $0/month.

Am I able to try the Professional subscription for free before committing?

Yes, you are able to request a Professional trial. The Professional trial is free and allows you 14 days to experience Onshape's Professional subscription before making any payments. Request a free trial at any time.

What is the difference between the Professional subscription and the Free subscription?

The Professional subscription allows users to create private documents and also use Onshape's release management workflow. Otherwise, the Free subscription includes all of the same CAD and data management functionality as the Professional subscription.

With the Professional subscription you are able to create as many private documents as you like. No Onshape users have any access to your private documents unless you specify who, and what type of access, is allowed.

With the Free subscription you are unable to create any private documents. All of your documents will be public to all Onshape users. No Onshape user has the ability to edit your public documents, but they are able to view or make copies of your documents. You are able to share your public documents as you would a private document, and specify who, and what type of access, is allowed. Please read our Terms of Service for more details.

What is the difference between the Professional subscription and the Standard subscription?

The Professional and Standard subscriptions mainly differ in that Professional is geared towards users who need and want company-wide features like: Release management, company-wide metadata, company-wide material libraries, and company-based sharing features. Both subscriptions allow users to create unlimited private
documents and include all the same CAD functionality.

**Does Onshape store my credit card information?**

No, Onshape never stores your credit card information.

**How do I change my credit card information?**

You are able to change your credit card and other payment information through the My account option on the User menu located in the drop down of your user name in the user interface. Select Payment options in the left pane to access your Onshape payment information.

**When is my credit card charged?**

Your credit card is charged when you sign up for a subscription, when you add users to your subscription, and at the beginning of every payment cycle.

**Why did my credit card transaction fail?**

Declines happen for a variety of reasons, and in many cases only your card-issuing bank is able to tell you definitively why your attempted charge was declined. Banks use automated systems to determine whether or not to accept a charge. These automated systems may take various pieces of data into account, such as your spending patterns, account balance, and card-specific information like the expiration date and CVC.

It may be that you entered one of the required pieces of information incorrectly or perhaps the decline was the result of a fraud protection program. Please contact your card-issuing bank for more information; and if the problem persists, feel free to contact us.

**Am I able to cancel my Professional subscription?**

You are able to change your Onshape subscription to a Free subscription at any time, for as long as you like. Any private documents you created and those shared with you remain available and in View-only mode.

Your Professional subscription remains active for the duration of the payment period, and you are converted to a Free subscription at the conclusion of the active payment period.

Exceptions to this include:
If you belong to more than one Professional subscription, being removed from one subscription means you are still on the other Professional subscription so you are not downgraded to the Free subscription in this case.

If you belong to a Professional company subscription, you must request to be removed by the company owner. Upon removal from the company, you are immediately downgraded to Free unless you are a member of another Professional subscription.

**Do I get a refund when I cancel my Professional subscription?**

No, your subscription becomes transitioned to Free at the end of the current payment cycle. The exception to this is if you are being removed from a Professional subscription that is your only Professional subscription. In this case you are immediately downgraded to Free.

**If I cancel, what happens to my documents?**

You are able to change your Onshape subscription to a Free subscription at any time, for as long as you like. Any private documents you created and those shared with you remain available and in View-only mode.

**Am I able to centralize payment for several users?**

Yes, sign up for a Professional subscription to pay for multiple users and create a company account. When you sign up for this subscription, you designate a company owner. Once the sign-up process is completed, the company owner adds company members to the subscription through the Account page in the user interface.

**How does payment work for multiple users on the same subscription?**

If you add users to your account who are already part of a Professional subscription, you are agreeing to pay for them as well. Users may belong to more than one subscription, but not a Free subscription and a Professional subscription at the same time with the same account. If a user wishes to have a Free subscription and also be part of a paid subscription, they must use two separate Onshape accounts (indicated by different email addresses when they sign up).

**What happens to my documents if a company owner drops me from a company subscription?**

Nothing happens to your documents. If your Onshape account is not associated with
any other Professional subscription, then you are downgraded to a Free subscription. Onshape never deletes your documents and never makes your private documents public.
App Store FAQs

Some commonly asked questions about the Onshape App Store include:

**How do I find an app?**

Use the search box in the app store: type in any part of an app name or type, and the Elasticsearch will return, for example “MasterCAM” when you type “CAM”.

**How do I get help with an app I purchased?**

For help with an application, check the specific app provider's link provided at the bottom of their app store listing. If you have billing questions, contact Onshape Support.

**How do I submit a request for app functionality that I would like to see in the App Store but that isn't yet offered?**

Use the Improvement request feature of the Onshape Forum to request a particular app or type of app.

**How do I integrate my own app into Onshape?**

If you are creating applications or programatic functions that use Onshape, we provide an application programming interface (API) as a means for integration. First, navigate to our [developer portal](https://developer.onshape.com) where you must sign a developer agreement. Once the agreement is signed, you will have access to the API explorer through the Onshape App store.

For examples on how the Onshape API works though JavaScript or Python, see the Onshape public repositories on GitHub.

If you have more questions about the Onshape API, you can contact the [Onshape Partner Development team](mailto:partnerdevelopment@onshape.com).

**How do I submit my own app to be featured on the Onshape App Store?**

Contact the Onshape Partner Development team, [https://www.onshape.com/partners/apply](https://www.onshape.com/partners/apply) (opens in new tab).

**How do I view all the apps I've purchased?**

Sign in to the Onshape App Store ([https://appstore.onshape.com/](https://appstore.onshape.com/) (opens in new tab)) and select the My Apps filter (on the left) to see which apps you’ve purchased.

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You are also able to sign in to your Onshape account (https://cad.onshape.com/ (opens in new tab)), click the plus sign menu at the bottom left corner of the page (in a document). From there, select Add Application to view the list of applications you have purchased. Alternatively, navigate to My account > Applications and view your applications there.

Am I able to purchase apps for my Onshape company?

Yes, the owner of a company in Onshape has the ability to purchase multiple seats of an app subscription and then specify the users who are able to use those seats. One-time purchase apps, however, must be purchased separately per user.

When an Onshape document that uses an integrated app is shared with another user who also has that app, both users are able to see the tabs related to that app.

Am I able to cancel an app subscription?

Yes, once canceled through the Onshape App Store, your subscription will be canceled at the end of the current billing cycle. You are also able to immediately revoke Document access from an app.

How do I instantly revoke Document access from any app?

In your Onshape account, navigate to My account > Applications and click Revoke for the app.

Am I able to resubscribe to an app I’ve canceled?

Yes, in the App store, click the Reactivate button below the canceled app.

How do I access a purchased app in my Onshape document?

The first thing to understand is that only Integrated apps are visible inside the Onshape interface. After you purchase an Integrated app in the Onshape App Store (or sign up for a free app):

1. Sign in to your Onshape account (https://cad.onshape.com/ (opens in new tab)).
2. Open the document with which you want to use the app.
3. Click the plus menu at the bottom of the window and select Add Application.
4. If you have more than one app available, select the one to use with the currently opened document.
5. The first time you use an app, you'll be prompted to authorize the app to access the data in your Onshape document.

After authorizing access, the Integrated app is visible in your document, as a tab, and that tab automatically opens.

For Connected Cloud and Connected Desktop apps, you will need to access Onshape through those apps by first signing in to the app in a separate browser window (or opening the app on your machine).

**Am I able to use apps if I am an Onshape Free plan user?**
Yes.

**What happens if I add an app tab to a document and share the document with another Onshape user who has not purchased or authorized that app?**

The user with whom you shared the document will see a message explaining why they are not able to see the app data and suggesting they purchase it, if desired.

**Am I able to see my app on the Onshape mobile platform?**

Not at this time.

**What if I have questions about an app?**

Contact the provider of that app.

**What if I have questions about the App Store itself?**

Contact Onshape Support directly from the Contact Support tool on the Documents page, or from within a document.

**Does Onshape share my personal information with app providers?**

Onshape will not share your personal information with an app provider unless you have explicitly agreed to provide such information to that provider. To find out how an app provider shares your information with other parties, refer to the terms and policies for that provider's product. You are given an opportunity to review those terms and policies before purchasing or otherwise acquiring the app.
Onshape Mobile Touch Interface: Video

This functionality is also available on Onshape’s iOS and Android platforms.

**Touch Interface**

This video covers:

- Navigating the Onshape workspace
- Using gestures
- View Cube
- Context Menu
- Selections and the Precision selector
- Dialog navigation and actions
- Feature list navigation and actions
- Sketching gestures and methods
- Constraints
- Assembly navigation
- Using the manipulator

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Sketching on Mobile

This video covers:

- Sketching
- Sketch tools
- Sketch dialogs
- Context Menu
- Precision selector
- Sketching methods
- Inferencing
- Constraints
- Dimensions
Part Design on Mobile
This video covers:
- Using Feature tools
- Extrude
- Revolve
- Sweep
- Shell
- Fillets & Chamfers
- Feature dialogs and actions

Assemblies on Mobile
This video covers:
• Creating a new Assembly
• Inserting parts
• Moving parts
• Fixing parts
• Feature list navigation and actions
• Dialog navigation and actions
• Using the Triad Manipulator
• Using Mates
• Mate connectors
• Context Menu
Contact Support

This functionality is available on Onshape's browser, iOS, and Android platforms.

If you have specific CAD-related questions with regard to using Onshape, you always have the ability to look through questions and answers within the greater Onshape Community via our Forums. If you can't find a relevant answer to your question in the Forums, feel free to submit a support ticket.

Please be aware that it is in your best interest to submit a support ticket for each issue you have, and not to bundle multiple issues in one ticket.

File a support ticket

1. Sign in to Onshape.

   Either from the Documents page or from within a document, click the ? in the upper right corner of the screen.

2. Click Contact Support (if you have a Professional subscription) or Report a bug (if you have a free or EDU subscription).

3. Optionally capture a screenshot and mark up the screenshot.

4. Enter a title and provide a description.

5. Optionally check the box to share your document with Onshape Support.

6. Click Send to submit to Onshape, or Cancel to close without sending.

You are also able to use the Contact Us (opens in new tab) link on our home page.

Sharing your document with Onshape Support allows access to only Onshape Support staff, and only for a limited time. Sharing your document enables the Support staff to see first-hand what the issue is and in most cases, to solve the problem more quickly. You can revoke the share at any time. For more information on sharing documents in general, see "Share Documents" on page 2045.
Note that when sharing your document with Onshape Support, any Onshape employee in the document will have the Onshape icon next to their name in the upper right corner of the page.

Find Onshape, a business unit of PTC

PTC World Headquarters
121 Seaport Boulevard
Boston, MA 02210
Glossary

A

administrator
A user with the ability to add and remove users to an organization and to change permissions for users within an organization.

assembly
A collection of instances of parts, sketches, surfaces, or subassemblies that defines both position and movement.

assembly toolbar
A series of tools for creating mate connectors, mates, patterns, and relations for use in assembling parts.

B

branch
A named fork in the Version Manager graph of a document. Branches fork at a version, end with a workspace, and can have zero to N sequentially stored versions on the branch.

branched editing
The ability for multiple users to edit two different branches of the same document without impacting each others' work. When desired, two branches can be merged.

C

collaboration cue
The social cue icon with a user identifier that appears at the top of the page in a document, and on the tab or feature when more than one user is editing a document.

collaborator
A user viewing or editing a document that other users are also viewing or editing.
D

direct edit

Editing a feature directly in the 3D form; especially necessary when the part is imported (uploaded and translated) or the existing parametric history does not support the change needed.

document

A collection of design data organized in Onshape tabs. Tabs may contain Part Studios, Assemblies, Drawings, Tutorials, Applications, Feature Studios, and imported files like CAD files, PDF files, Word files, etc.

Documents page

The Onshape page that lists documents and allows the user to open, create, filter, label, and import documents.

drag manipulator

A manipulator used to resize resulting geometry.

E

entity

An Onshape system object or an item built in Onshape: mates, mate connectors, sketch curves, parts, edges, and faces are all examples of entities.

F

face

A portion of a part, surface, or closed sketch region having area and bounded by edges; a simple rectangular part has six faces.

Feature list

The parametric history contained in a Part Studio.

Feature toolbar

The series of tools available for creating features of a part.
features
The operations that are used to build parts, such as Extrude, Fillet, Shell, Revolve, Sweep, Chamfer, Draft, Patterns, Mirror, Modify fillet, and Move face.

fix
To make a sketch entity or an assembly instance unmoveable.

graphics area
The large rectangular portion of the user interface in which a Part Studio or Assembly is displayed.

inference
An automatic indication during sketching that a constraint may be applied; appears as an orange dotted line as well as orange boxes highlighting related entities.

instance
A part, sketch, surface, or subassembly used in an Assembly.

mate
An Onshape feature used to position instances in an assembly and define how they move.

mate connector
A local coordinate system entity located on or between entities (parts or solid models) that can be used within a mate to locate and orient instances with respect to each other.

merge
The ability to apply edits to a workspace from one branch into a workspace on another branch of the same document.

navigation bar
The top bar of the user interface window that contains the Document name and User ID/profile menu.
owner

State of user or company who has full access to a document. Ownership can be transferred to and from users and companies.

part

A single, simply closed solid body created by Onshape features or by uploading and translating (also referred to as importing) another CAD file.

Part Studio

A parametric, feature-based geometric model that creates parts.

parts list

The list of parts created in the current Part Studio. They are listed in the bottom portion of the Feature list.

permissions

Control over the actions that users can perform on a document.

planes

Planar construction geometry created using the Plane feature.

Preview slider

A slide bar on feature dialogs that allows you to vary the opacity of the edited feature between the state before the feature was added to after it was added.

private

The state of a document that is not shared, or shared only with specified users.

properties

Sometimes called meta data, properties are a way of attaching important information to design entities, such as parts, assemblies, and versions. Properties include: Part Number, Description, Revision, State, Comments, and more. The Property command is available on context and actions menus.
public
The state of a document that is visible to all Onshape users. Public documents are view-only if users do not have edit permissions.

region
A finite area in a sketch defined by a bounding set of sketch curves. Sketch regions are used in features like Extrude, Revolve, and Sweep to create or edit parts in a Part Studio.

rollback
The ability to see and edit an earlier state of the Part Studio's parametric history. This is done by repositioning (by click+drag) the rollback bar in the Feature list.

rollback bar
The rollback bar in the Feature List enables you to temporarily revert to an earlier state in the feature history. You can also add new features or edit existing features while the model is rolled back.

share
The action of giving other users access to an individual document with a specified permission level.

simultaneous editing
The ability for multiple users to edit an active workspace of a document at the same time.

sketch
A set of curves drawn on a plane with sketch constraints on those curves.

sketch constraints
Relations between sketch entities that define their shape and behavior, such as Dimension, Coincident, Concentric, Parallel, Tangent, Horizontal, Vertical, Perpendicular, Normal, Equal, Midpoint, and Fix.

sketch curve
A line, arc, circle, or spline in a sketch.
sketch toolbar
The series of tools available for creating a sketch.

sketch tools
Tools in the Sketch toolbar such as Line, Corner rectangle, Center point rectangle, Center point circle, 3 point circle, Tangent arc, 3 point arc, Spline, Point, and Construction.

social cue icon
The icon with a user identifier that appears at the top of the page in a document, or on a tab or feature when more than one user is editing a document.

surface
An Onshape entity that may have one or many faces but no volume. Surfaces are listed independently of parts in the Feature list and are not parts; surfaces can be inserted into an Assembly. In some traditional CAD systems, Onshape surfaces are similar to Sheet Bodies.

tab
An entity in Onshape that can contain a Part Studio, Assembly, Drawing, image file, PDF file, document file (PDF, Word), Applications, Tutorials, Feature Studios, and even Gcode. Tabs are displayed at the bottom of an Onshape document in the tab bar.

tab bar
The bottom bar of the Onshape document that contains all Onshape tabs.

toolbar
A set of tools displayed at the top of the Onshape document tab.

traditional CAD
Older desktop CAD systems like SOLIDWORKS, Pro/ENGINEER, CATIA, and Inventor.

triad manipulator
A manipulator that appears in a Part Studio Transform operation and in an Assembly when an instance is selected. Use the manipulator to move the part in any direction and angle in relation to the selected face(s) or edge(s).
**U**

**user**

An individual account that provides access to Onshape; the user name can be seen in Navigation bar, in the right corner.

**V**

**version**

A snapshot of a document at a particular time. A version is created using the Create version command and appears in the Version Manager. Versions are immutable and can never be changed. Versions may have properties (meta data) assigned to them.

**Version Manager**

A graphical representation of the document's versions and workspaces in a branch/tree diagram. There is a menu from which to choose actions such as: open, properties, compare, delete, merge, and branch to create a new workspace.

**View cube**

The cube appearing in the top right corner of the graphics area when the user opens a document. Click on a face to view the model from that perspective. Click the arrows to turn the model in increments.

**virtual edges**

Curves in a drawing that are drawn at the places where parts intersect.

**W**

**workspace**

The editable iteration of an Onshape document. There can be multiple workspaces for a document and a branch can end in either a version or a workspace.
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