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# Welcome to Onshape Help

Onshape is available conveniently through your browser and on your mobile devices (iOS and Android) as well. With Onshape mobile, you can access your existing Onshape account any time, anywhere. To learn more about Onshape for mobile devices, see "Onshape Mobile Devices" on page 19.



This is the help system for Onshape on a browser. If you need help with Onshape mobile apps, please access the help system available through those apps.

## About Onshape help

Keep in mind that Onshape help is context-sensitive.



When you click with a dialog open, Onshape displays the relevant help topic.

If there is no active context, you land here.



Click the Help menu icon in the upper right corner of the interface, , for more learning resources.



## Some options to start with

- "Beginning Tutorial" on page 567
- Instructional videos
- "User Interface Basics" on page 30
- "Part Studios" on page 89
- "Assemblies" on page 304
- "Drawings" on page 393
- "Document Management" on page 501

## Navigating the help system

- **Table of Contents** - Use the Table of Contents in the left pane to browse through available topics.
- **Index** - Browse through the index looking for terms that are familiar to you; the link will take you to the appropriate Onshape topic.
- **Search** - Enter whole words or partial words, for example **Part Studio** or **Par**. Results are displayed in the content pane of the help system.

## To make things easier, follow these conventions

What it looks like	What you should do	What to expect
<b>bold text</b>	Take note and scan for actions	Instructions
<i>italic text</i>	Names of fields, dialogs, objects in UI	References to what you see in the UI
blue background	Read for more information	Tips, helpful, background or expansive information
<u>blue and underlined text</u>	Click on it	A jump to another topic for more detailed information

What it looks like	What you should do	What to expect
<b>triangle with blue text</b>  This would be the drop down expanded text	Click on it	An expanded block of more information in place; click again to collapse the information

# Onshape Mobile Devices

CAD is no longer bound by the power of your computer.

Now you can design, wherever you are. Imagine being able to access and edit your models on the same device. Since Onshape is based entirely in the cloud, you don't need a powerful workstation. The heavy computation is done elsewhere so you can use Onshape on a lightweight device.

We designed the mobile interface specifically for using fingers on smaller screens. The same familiar icons open the same powerful editing tools. All the functionality is right there. You can even work simultaneously on mobile, while a friend edits the same model on a desktop.

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—whenever and wherever inspiration strikes

To begin using Onshape mobile with your existing Onshape account, just download from the App store or Google Play.



## Supported OS and devices

Onshape minimally requires:

- iOS 8 or later
- iPad Mini 2 or later (note the first generation of iPad mini is not supported)
- iPad 4th generation or later
- iPhone 5 or later
- Devices running Android 4.2 Jelly Bean or later, inclusive of Lollipop 5.0 and Marshmallow 6.0
  - Such as Nexus 9, Nexus 6p, Nexus 5x, Nexus 6, Nexus 5, and Nexus 7 2013

When considering the Onshape mobile apps, please consider using the following devices to get the most out of your Onshape experience:

- iPad Air 2 Wi-Fi, iPad Air 2 Wi-Fi + Cellular
- iPad Mini 3 Wi-Fi, iPad Mini 3 Wi-Fi + Cellular
- iPhone 6 and iPhone 6 Plus
- iPhone 5s
- iPad Air Wi-Fi, iPad Air Wi-Fi + Cellular
- iPad Mini with Retina display Wi-Fi, iPad Mini with Retina display Wi-Fi + Cellular
- Any devices running Android 4.2 Jelly Bean or later, inclusive of Lollipop 5.0 and Marshmallow 6.0
  - Such as Nexus 9, Nexus 6p, Nexus 5x, Nexus 6, Nexus 5, and Nexus 7 2013

# Graphics Performance Recommendations

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

## Browsers

Onshape currently supports these tested and approved browsers:

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

Internet Explorer is currently not supported.

## WebGL

Onshape requires WebGL. To ensure that you are taking advantage of the highest performing configuration, make sure your preferred browser has WebGL enabled. On Chrome, it should be enabled by default, unless your graphics card does not support WebGL (check the black list: <https://www.khronos.org/webgl/wiki/BlacklistsAndWhitelists>). On Safari or Firefox, you may have to enable it manually:

Safari	Firefox
<ol style="list-style-type: none"><li>1. Click <b>Safari</b> menu, select <b>Preferences</b>.</li><li>2. Click the <b>Security</b> tab.</li><li>3. Select <b>Allow WebGL</b>.</li></ol>	<ol style="list-style-type: none"><li>1. In the address bar, type <b>about:config</b>.</li><li>2. Click through the warnings.</li><li>3. Search for <b>webgl</b>.</li><li>4. Confirm <b>webgl.disabled</b> is set to false (or set it to false; this will not affect any warranties).</li></ol>

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

## Graphics cards

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.

If you do not have NVIDIA or NVIDIA with Optimus technology, you can skip this topic.

To get the most out of your graphics cards:

- Make sure the 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> 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 can download and use a utility such as **Speccy** (for Windows) or **gfxCardStatus** (for Mac) to discover what is installed on your machine. You can also use <chrome://gpu> to see which GPU is used.

You want to use the faster, discrete NVIDIA GPU (when available) for Onshape, always. For applications that don't require high performance graphics or require longer battery life, you can 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.

## Additional information

More resources include:

- <http://alteredqualia.com/texts/optimus/> -- more information and specific instructions
- <http://alteredqualia.com/tmp/webgl-maxparams-test/> -- for immediate and install-less detection of graphics cards on your machine using WebGL

# Getting Started

Getting started with Onshape is as simple as:

1. [Create an account.](#)
2. [Sign in.](#)
3. [Create a document.](#)
4. [Begin modeling.](#)

Onshape doesn't require any commitment on your part: you can sign up, create documents, make models, export files, all for free. If you want to keep large numbers of documents private, it's best to upgrade to a [Professional subscription](#).

To become familiar with modeling in Onshape, read more about our concepts and [user interface](#).

# Creating an Account

Using Onshape requires that you have an Onshape account. Creating an account registers your name and details with an email address that you then use to sign in to Onshape from anywhere, on any device. An Onshape account can be associated with one or more Onshape [subscriptions](#).

## If you already have an account

Enter your credentials and click "Signing In" on the next page.

## If you do not have an existing account

1. Navigate to <http://cad.onshape.com>.
2. Click **Sign up** (beneath and to the right of the Sign in button) to create an account.
3. Answer a few questions and click Create Free Account.
4. Check your email for a confirmation email and confirm your email address.
5. Supply the rest of your information and click Sign Up.

# Signing In

1. Navigate to <http://cad.onshape.com>.
2. Enter the email address you used during the sign-up process.
3. Enter your Onshape password.
4. Click **Sign in**.

The *Documents* page appears:

- If you have previously created Onshape documents, the **Recently opened** filter is active.
- If you have not created any documents, the **Tutorials & Samples** filter is active.

## Forgot your password?

Click the link to receive an Onshape email containing a link to reset your password. Onshape doesn't save or record your password.

## Don't have an account yet?

Click **Sign up** to fill out a simple form to create a free account. See "Creating an Account" on the previous page for more information.

# Creating a Document

1. Click **Create**.
2. Enter a name for your document and click **OK**.

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

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.

Onshape opens a new Part Studio tab in the document you just created; Part Studios are where you create Parts. See "Part Studios" on page 89 for more information on modeling in Onshape.

The **Documents** page is the first page displayed upon subsequent Sign ins. While on any other page, click the Onshape logo to return here.

# Modeling in Onshape

In Onshape, there are two tab types for modeling - [Part Studios](#) and [Assemblies](#).

## 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 can be instanced multiple times in assemblies and each instance can move independently in the assembly.

In a Part Studio, there are two tool sets: "Sketch Tools" on page 103 and "Feature Tools" on page 179.

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

If these concepts are unfamiliar, watch the video titled Welcome to Onshape, or work through the "Beginning Tutorial" on page 567.

## 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 subassemblies), 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 sub-assembly defined in another Assembly.

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

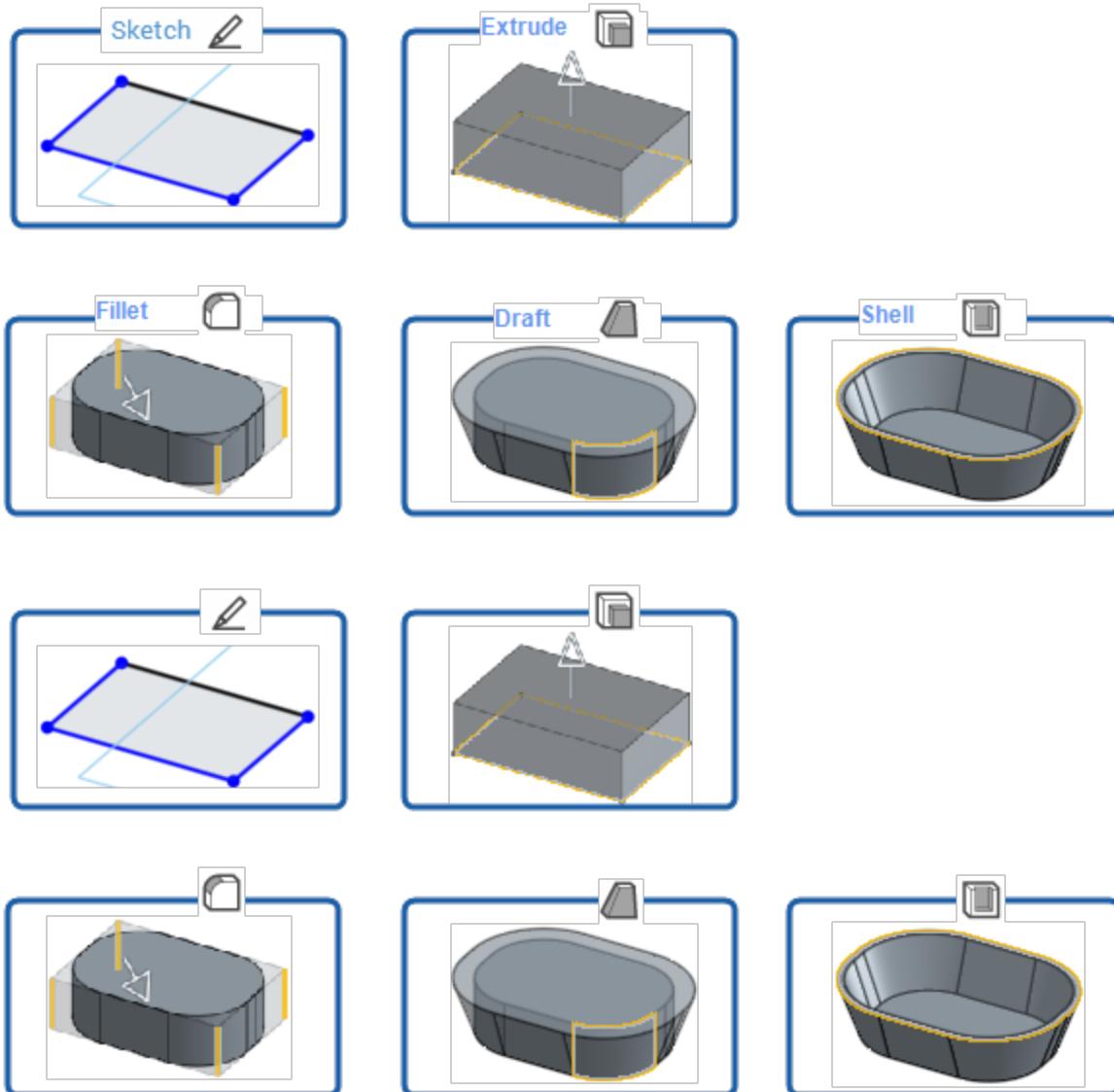
It's important to understand how Onshape mates differ from mates in other CAD systems. In traditional CAD systems, mates are low-level assembly constraints like making two planar faces coincident. As a result, positioning two instances usually requires two or three mates.

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 can define a pin slot relationship.

See the videos titled Getting Started with Assemblies and Assembly Mates to learn more about building assemblies and mating in Onshape.

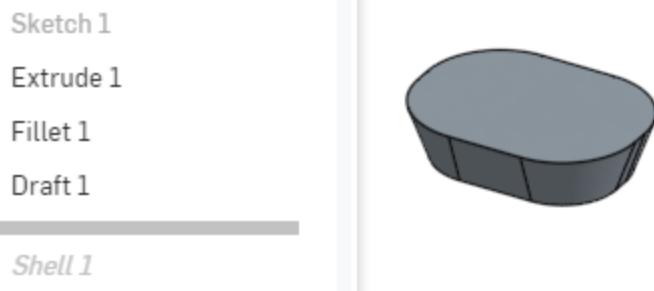
# Simple Modeling Example

The small design example below demonstrates how to use the Preview slider and the Final button in a Feature dialog. This example shows the basic steps involved in creating a simple tub:

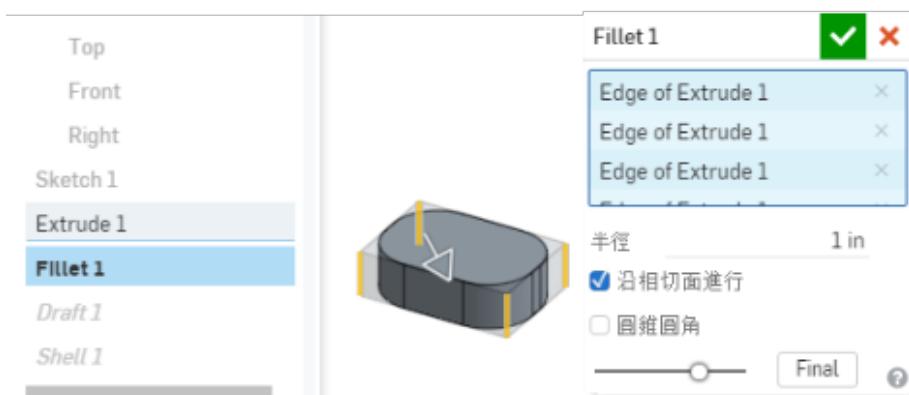
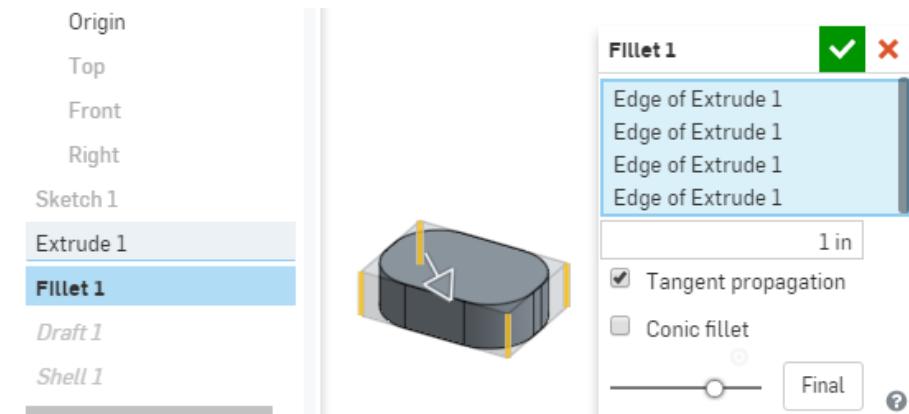


The features are shown in the Feature list in the order they were created, followed by the Rollback bar.

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:



Edit a feature by double-clicking on it or through the context menu > **Edit <feature>** selection. This illustration shows the Fillet 1 feature being edited. The system automatically rolls back the model display to before Fillet 1 was added and opens the Fillet dialog:



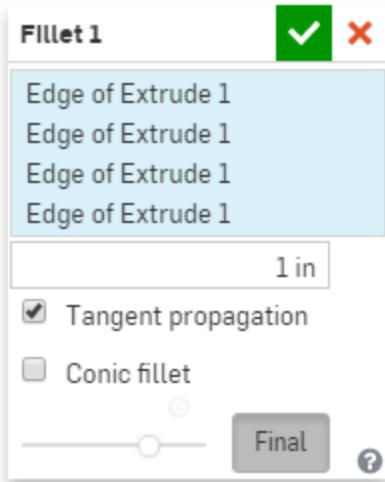
Notice the Preview slider and the Final button in the dialog. By default (as shown above), the Preview slider is at 70%. The image of the model is a blended image of before the feature is added (30%) and after the feature is added (70%). Sliding to the left increases the 'before' image; sliding to the 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.

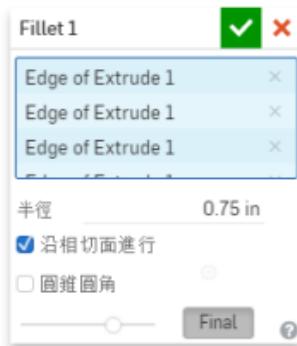
The Final button shows the impact of the feature on the final visualization of the model (with the later features fully regenerated).

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.

Origin  
Top  
Front  
Right  
Sketch 1  
**Extrude 1**  
**Fillet 1**  
Draft 1  
Shell 1



Origin  
Top  
Front  
Right  
Sketch 1  
**Extrude 1**  
**Fillet 1**  
Draft 1  
Shell 1

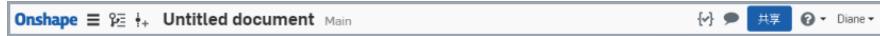


# User Interface Basics

## Toolbars

Located at the top of the page, these change based on the current work flow. There are 4 main toolbars:

- **The Document toolbar**



- **The Feature toolbar**



Select Sketch to open the Sketch toolbar.

- **Sketch toolbar** Open by selecting Sketch on the Feature toolbar.



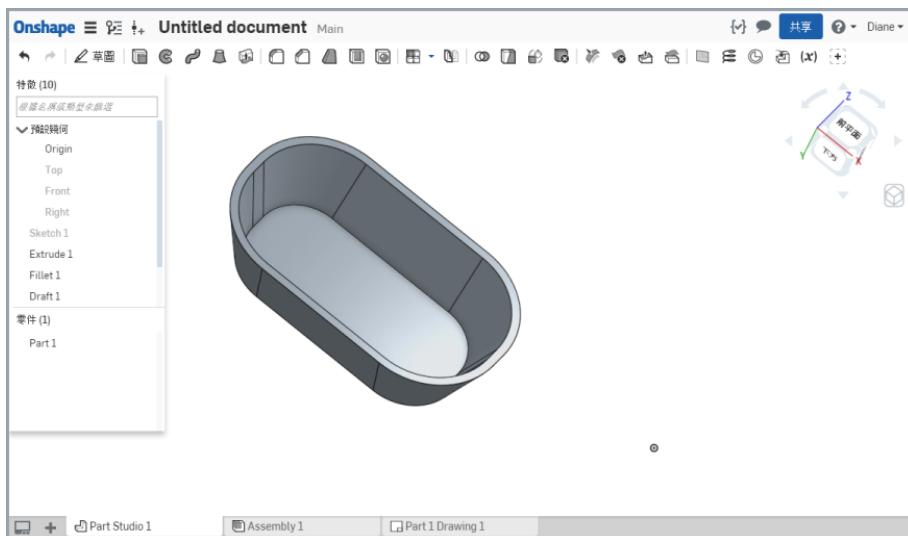
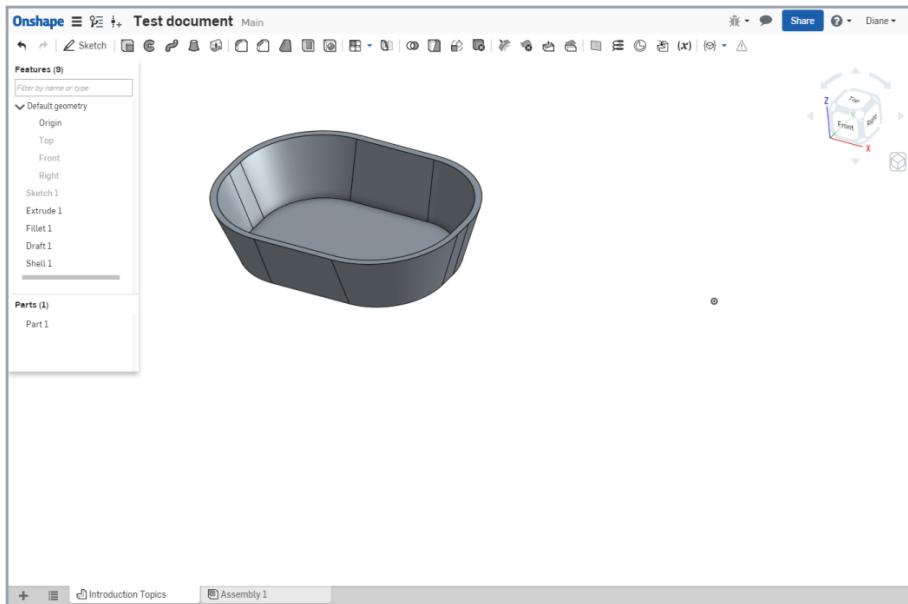
- **Assembly toolbar**



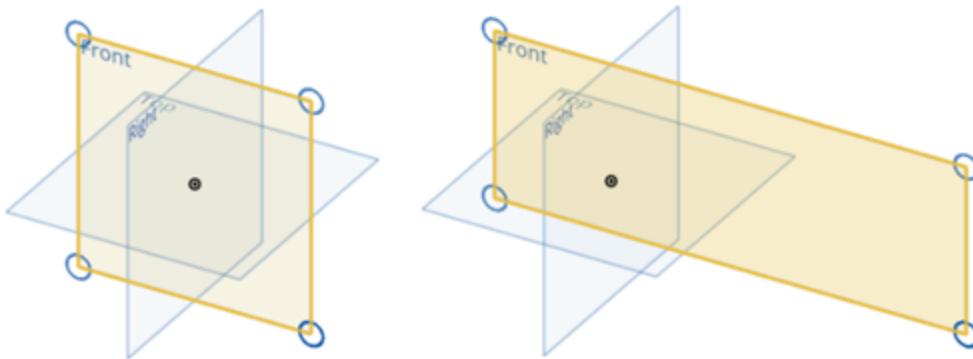
- **Drawings toolbar**



## 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 connectors. See "Part Studios" on page 89 and "Assemblies" on page 304 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.
- **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.
- **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).
- "Error Indicators" on page 70 - Color-coded feedback, messaging, constraint icons.

## Keyboard shortcuts

Activate the keyboard shortcuts map right in the user interface by pressing the Question mark key "?" on your keyboard when in a document. You can even pop it out of the window for continuous display:

Keyboard shortcuts			
General	Part Studio	Sketch	Drawings
shift ? Keyboard shortcuts	s Sketch	shift Disable inferencing	shift z Zoom in
ctrl / ⌘ z Undo	shift e Extrude	l Line	z Zoom out
ctrl / ⌘ y Redo	shift f Fillet	g Corner rectangle	f Zoom to fit
delete Delete selection	ctrl / ⌘ m Mate connector	r Center rectangle	w Zoom window
space bar Clear selection	shift Lock mate inference	c Circle	p Projected view
shift s Save a version	3D view	a Arc	d Linear dimension
esc Cancel command	shift z Zoom in	shift f Fillet	shift r Radial dimension
enter Accept command	z Zoom out	m Trim	shift d Diameter dimension
shift enter Accept & repeat command	f Zoom to fit	x Extend	n Note annotation
shift click Open in new window	w Zoom to window	o Offset	ctrl q Update drawing
ctrl / ⌘ click Open in new tab	← → ↑ ↓ Rotate	u Use	l Line
ctrl u Feedback	shift ← → ↑ ↓ Pan	d Dimension	ctrl s Display sheet menu
Assembly	shift 1 Front view	i Coincident	pg dn Next sheet
shift Lock mate inference	shift 2 Back view	b Parallel	pg up Previous sheet
ctrl / ⌘ c Copy	shift 3 Left view	t Tangent	home First sheet
ctrl / ⌘ v Paste	shift 4 Right view	h Horizontal	end Last sheet
m Mate	shift 5 Top view	v Vertical	
ctrl / ⌘ m Mate connector	shift 6 Bottom view	e Equal	
i Insert dialog	shift 7 Isometric view		
s Enable snap mode	shift 8 Section view		
a Flip alignment	n Normal to		
q Change quadrant	p Hide/show planes		
j Hide/show mates			
k Hide/show mate connectors			



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

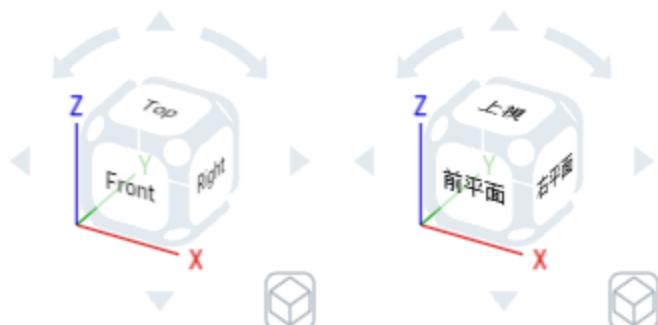
## View navigation

Windows	Mouse	<b>3D Rotate:</b> Right-mouse-button-click+drag <b>Zoom in and out:</b> Scroll up and scroll down, respectively <b>2D pan:</b> CTRL-right-mouse-button+drag (middle button click+drag)
	Touchpad	<b>3D Rotate:</b> Right-mouse-button-click+drag <b>Zoom in and out:</b> Pinch out and pinch in, respectively <b>2D pan:</b> CTRL-right-mouse-button+drag
Apple Mac	Mouse	<b>3D Rotate:</b> Right-mouse-button-click+drag <b>Zoom in and out:</b> Scroll down and scroll up, respectively <b>2D pan:</b> CTRL-right-mouse-button+drag (middle button click+drag)

**Rotate the view in 45 degree increments:** Click arrows around the View Cube.

Return to the **Trimetric view:** Click one of the small bubbles at the corners of the View Cube.

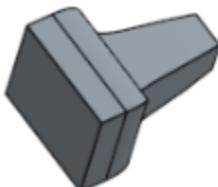
View a particular **plane view** of the cube: Click one of the sides of the View Cube (Top, Bottom, Front, Back, Right, Left)



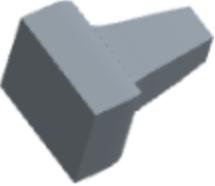
## View tools

The small cube, View Tools, offers these viewing options:

Shaded



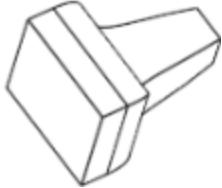
Shaded without edges



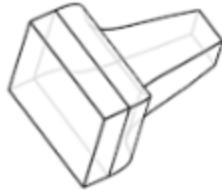
Shaded with hidden edges



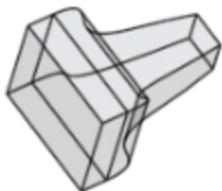
Hidden edges removed



Hidden edges visible



Translucent



Curvature visualization



## Zooming

The mouse wheel direction for zoom is configurable in user account [preferences](#).

To zoom:

- Zoom to fit (shortcut: f) - 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.

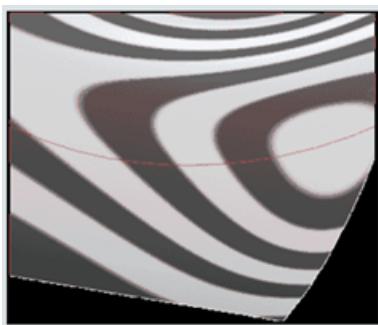
## Curvature visualization

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:

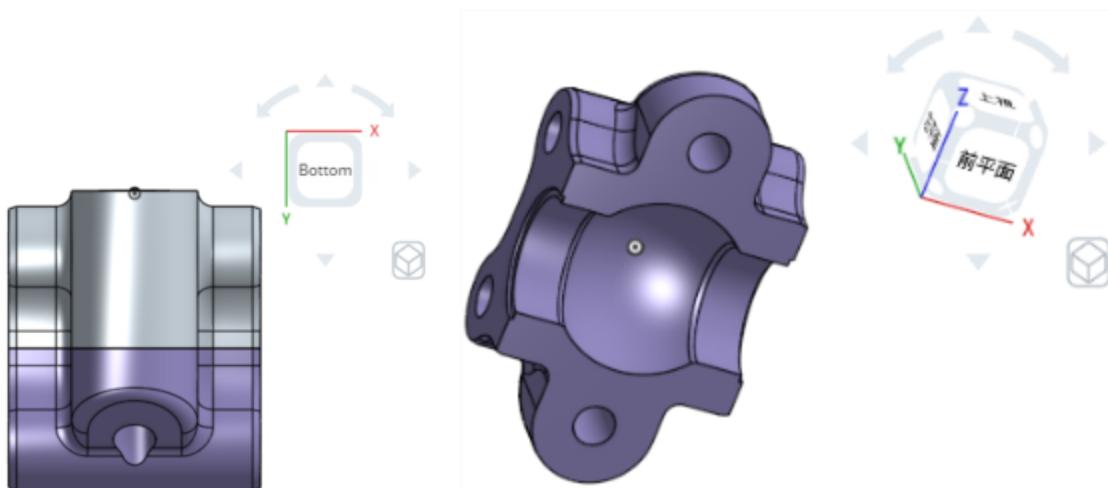


## Creating named views

You can 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:

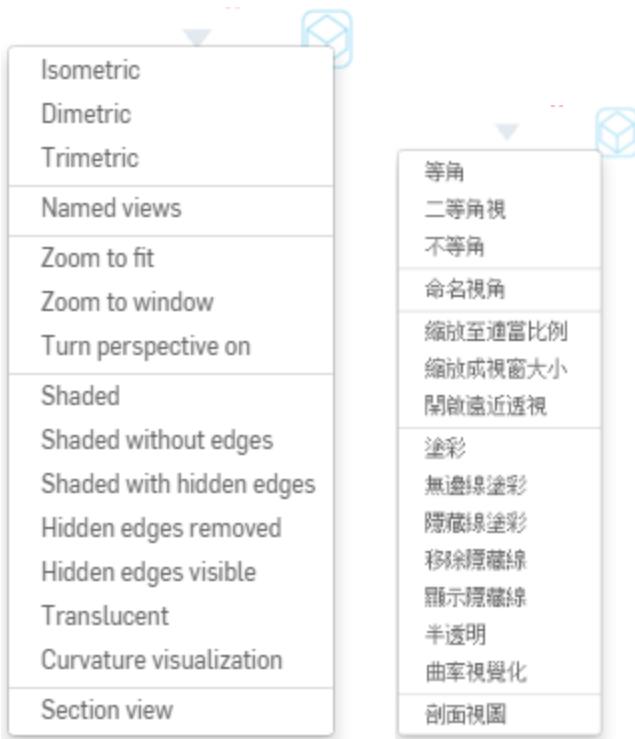
- Rotate your model into the desired view. For example:



2. Optionally, select **Turn Perspective on**, and/or **Zoom to fit**.

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**:



4. In the dialog, enter a name for the view in the first field:



5. Click the binocular icon with the plus sign

6. Notice the view (name) is saved in the next field:



You can create as many named view are you want per workspace.

To delete a named view:

1. Select the view in the second field:



2. Click the binocular icon with the x.

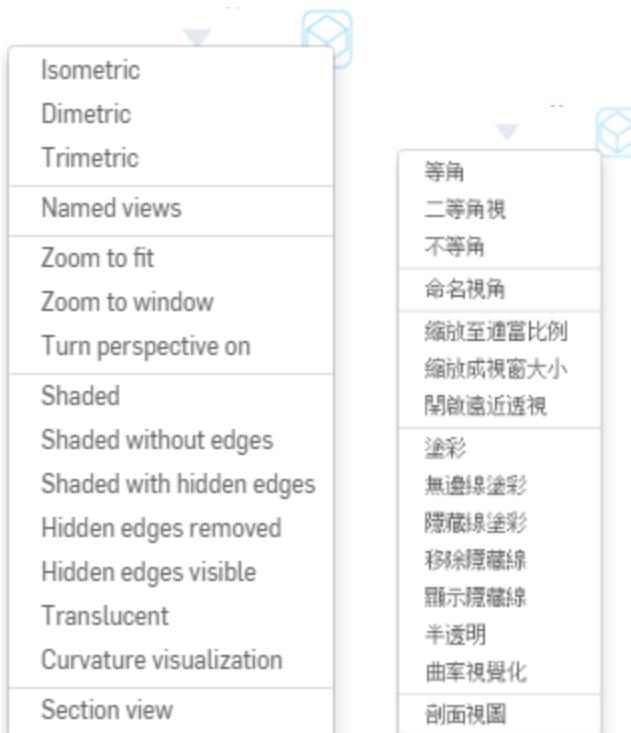
## Setting transparency via the context menu

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:Appearance" on page 64 for more information.

## View parts sectioned with Section view

You can view sectioned parts in both Part Studios and Assemblies:

1. Select one plane, mate connector, or planar face on the part.
2. Expand the menu on the View Tools cube and select **Section view**.



3. The part is sectioned at the point you chose in step 1 above (planar face, plane, or mate connector).

Portions rendered in red represent intersecting parts.

4. Use the manipulator to change the depth and/or angle of the section.

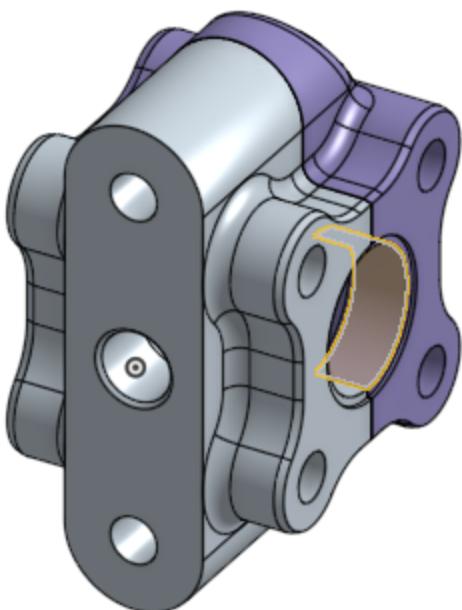
5. Select **Turn off Section view** when you're finished.



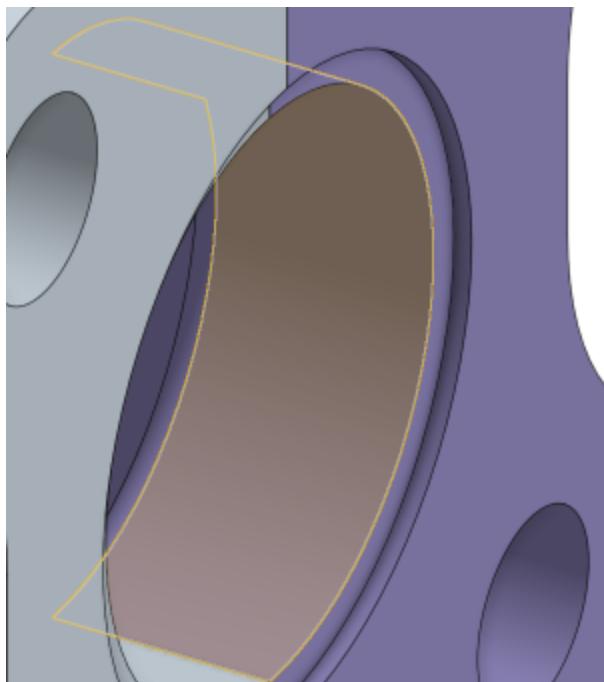
## Zoom to selection

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:



# Toolbars and Document Menu

## Document toolbar



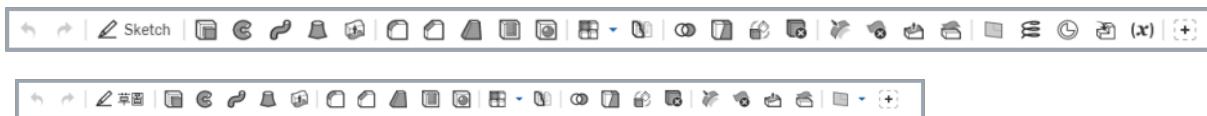
The Document toolbar is available in all Onshape documents (aligned with the Onshape logo) and you can:

- From the Document menu :
  - **Rename** a document
  - **Document description** - Enter a description for the document; this description displays on the Documents page, in the detail pane.
  - **Copy** a workspace
  - 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 affect on imported files.
  - View and edit the active workspace's **properties** (including a list of tabs, parts within the tabs, descriptions, part numbers, revision numbers, and states)
  - **Print** the graphics area of the active tab.
- Access the Versions and history flyout
  - Save a version of the document
  - Create a new document workspace (branch)
  - Enter properties (meta data) for the document or version
  - View the individual points in history of the document workspace
  - Restore the workspace to a previous point in time
- Access comments for the workspace
  - Open the comment flyout to create comments for a workspace
  - Edit comments
  - Delete comments
  - Reply to comments

For more information about commenting on document workspaces, see "Comments on Workspaces and Features" on page 486.

## Part Studio toolbars

### Feature toolbar:



For more information on the Feature toolbar and tools, see "Feature Tools" on page 179.

**Sketch toolbar:** the Sketch toolbar collapses when the browser is resized.



Note the arrows beside some of the icons; click to expand the menu and view additional tools.

For more information on sketching and constraints, see "Sketch Tools" on page 103.

## Assembly toolbar



For more information assembling parts and subassemblies, see "Assemblies" on page 304.

## Drawings toolbar



# Selection

## Graphics area

Onshape selection works like a toggle. Click to select, click again to deselect. You can also use Alt+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.

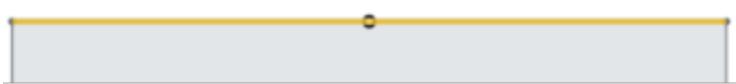
The cursor displays a count of selected entities; the displayed cursor count is accurate up to 5 entities (after 5, the cursor maxes out at 5+).

- Clicking (or Alt+clicking) additional entities adds them to the selection set.
- Clear selections by clicking in empty space or by choosing **Clear selections** from the context menu.
- 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 can 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, double-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 can hover over sketch entities and model edges and visualize the midpoints:



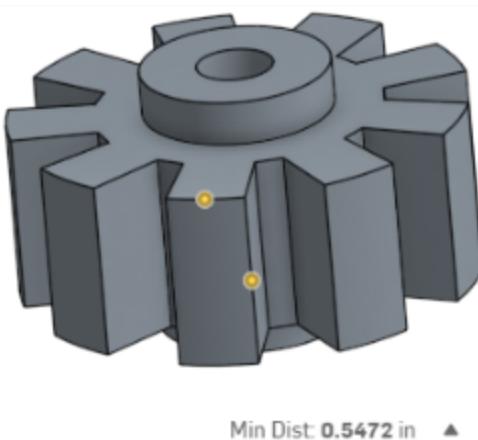
*Hovering to see a midpoint in a sketch*



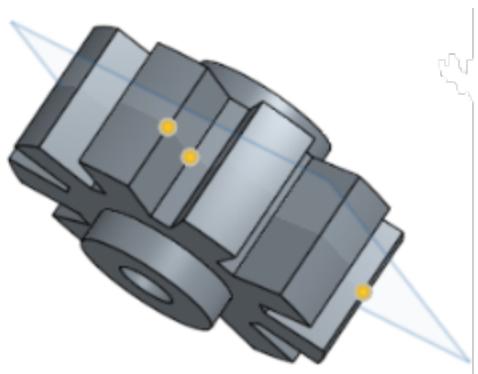
*Hovering to see a midpoint on a model edge*

Use these midpoints for:

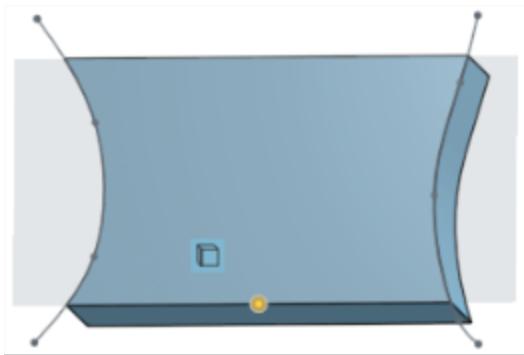
- Measuring - Select two points to get measurement information in the Measure tool in the right bottom corner of the interface:

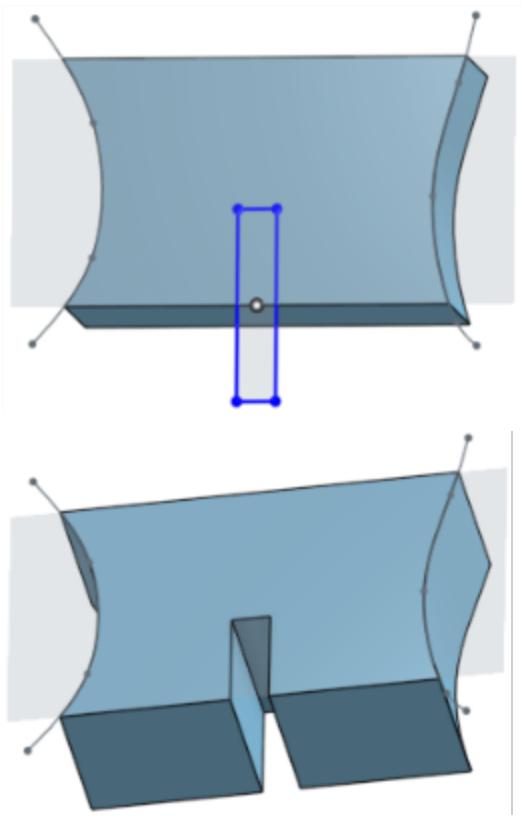


- Creating planes - Select midpoints as points to define planes:



- 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:

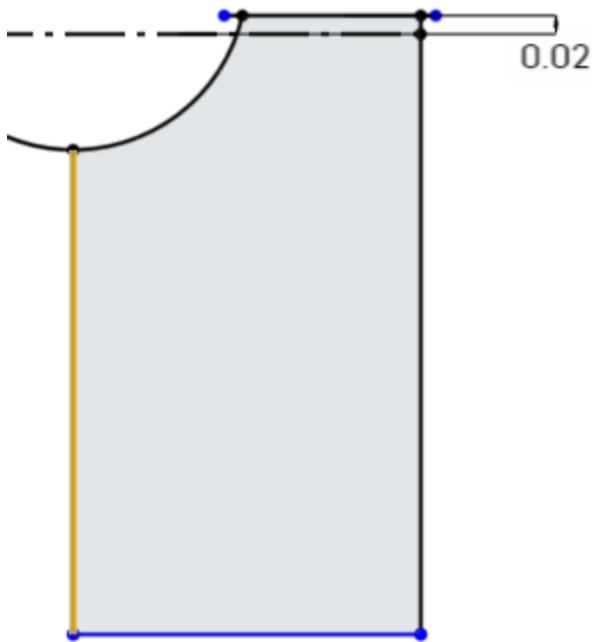




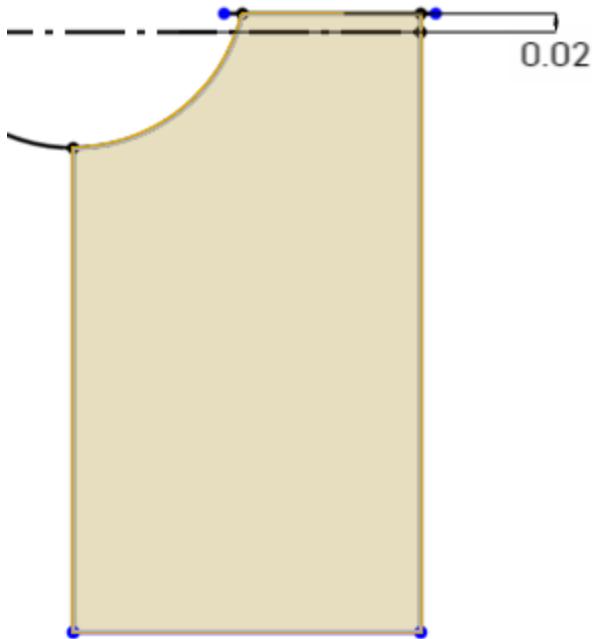
Note that midpoints do not appear for entities in the active sketch. When creating or editing a sketch, you can select the midpoint of an entity in another (inactive) sketch and use the Use tool to incorporate that point in the active sketch.

## Cursor selection examples

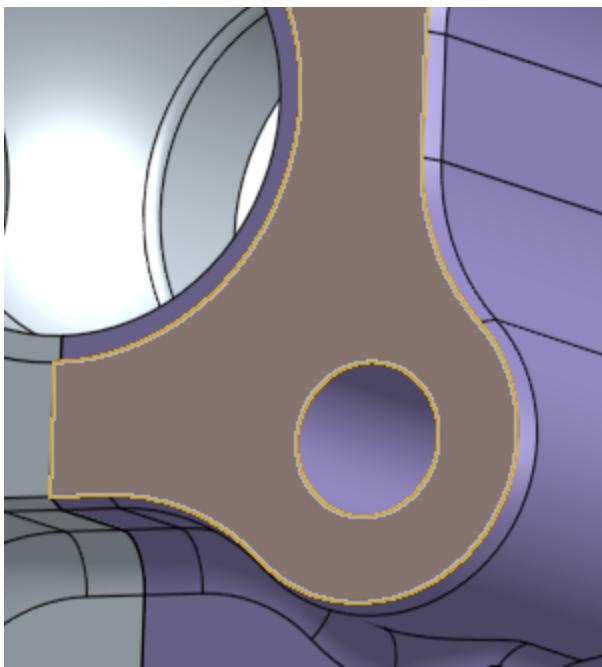
- Sketch curve highlighted



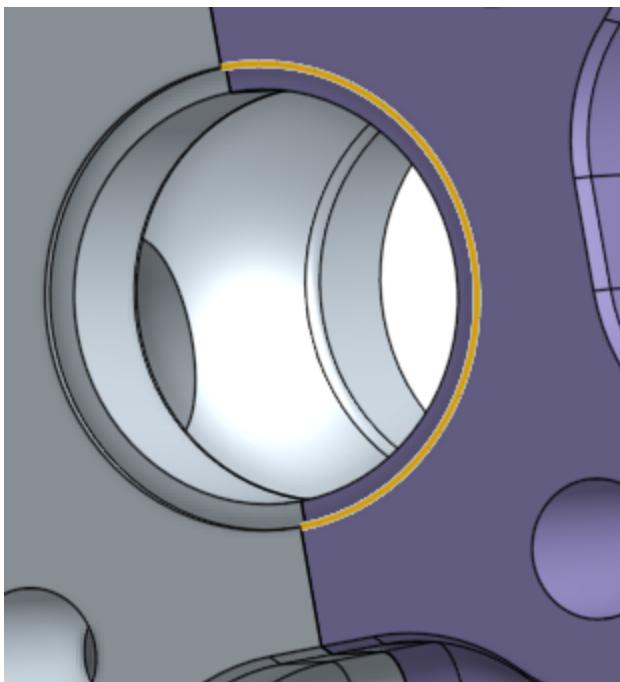
- Region highlighted



- Face highlighted

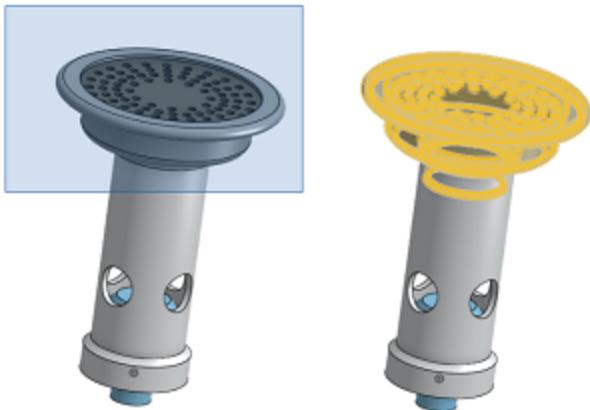


- Edge highlighted



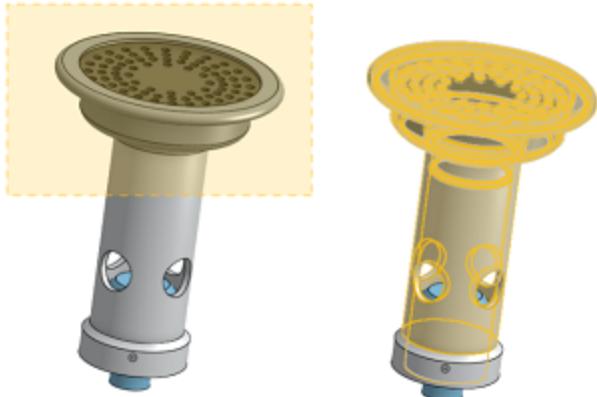
## Box selection examples

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 in both Part Studios and in Assemblies.

# Dialogs

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:

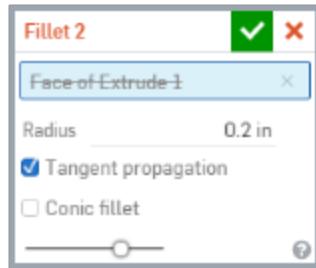
- Fields that are highlighted in blue are populated when you make a selection in the graphics area and in the Feature list.
- Fields that are outlined in blue (and not highlighted) are populated with keyboard input, usually numeric values.

Onshape provides visual representation of the possible states of selected entities. For example:

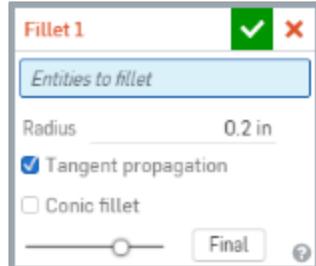
Healthy selection input



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.

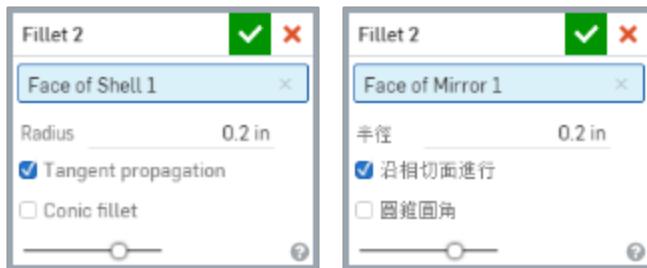
Use the Enter key to accept the dialog and close it, use Shift-Enter to accept the dialog, close it, and re invoke the same function with the dialog empty.

## Example of Active selection field

Before a selection is made in graphics area:

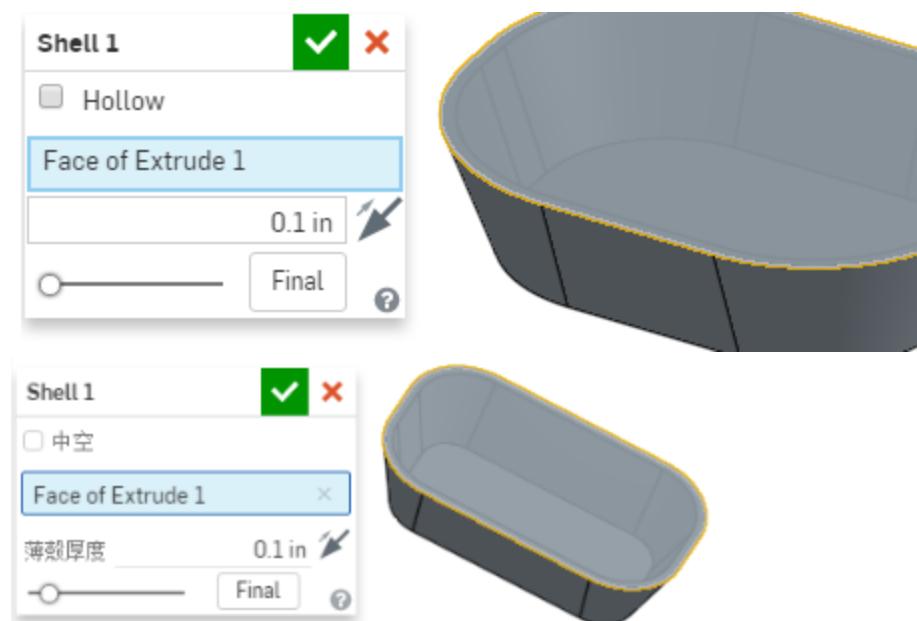


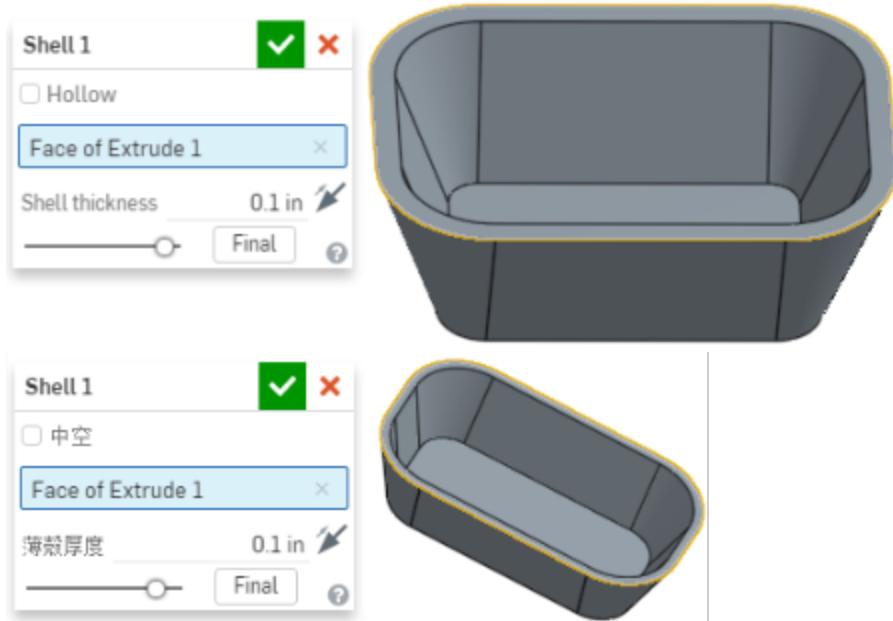
After a selection is made in graphics area:



## Preview slider and Final button examples

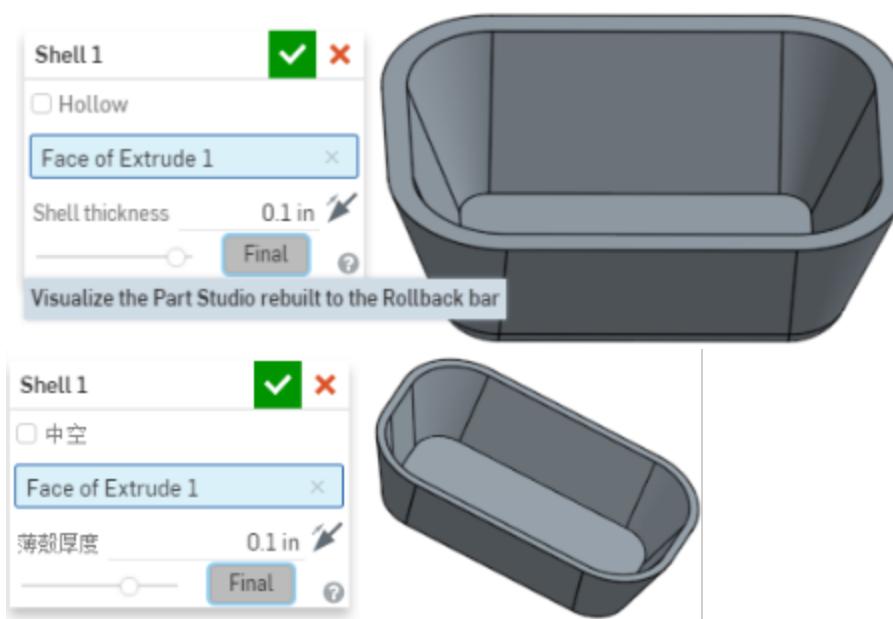
When creating or editing a feature, the preview (the model in the graphics area) is usually displayed as a blend of the model before and after the feature. The **Preview slider** is an opacity control that lets you adjust the display opacity of the feature along a scale of 0% (before the feature is applied) 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.

Let's continue with the example above. Clicking the Final button shows the part in its Final state with the current editing applied:



For a more detailed example of the Preview slider and Final button, see "Simple Modeling Example" on page 27 or watch the video titled Extrude.

# Numeric Fields

When entering values and expressions in numeric fields throughout Onshape, you can use the keyboard and also the mouse scroll wheel:

Scroll+Key	Result
scroll wheel default	increments of 0.1
Ctrl-scroll wheel	increments of 0.01
Shift-scroll wheel	increments of 1.0

Numeric value fields throughout Onshape **Part Studios and Assemblies** 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 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

Keyword type	Keywords accepted	Examples
Length	mm, millimeter, cm, centimeter, m, meter, in, inch, ft, foot, yd, yard	5mm 10meters 3ft
Angle units	deg, degree, rad, radian	7deg (or 7 degree) 14rad (or 14 radian)
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 functions	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 atan2(4, 5) (Give the polar angle of (5,4) in as an angle)
Constants	pi, PI, Pi	(3*pi) in

## 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):

Valid	Invalid
3in + 2.5in	3 + 2.5in

3mm + 2.5in	3mm + 2deg
3 + 2	(2*3)(1/3)
(2*3)*(1/3)	sqrt(16m)
sqrt(16)m	30°
cos(30deg)	

- 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:  $3\text{mm} + (2\text{mm}^*\text{3mm})/(6\text{mm})$ , 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  $3[\text{unit}] * 3[\text{unit}]$  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

- $3\text{in}^*\text{3in}$
- $3 + 3\text{in}$  (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:  $[\text{atan}(1)/\text{deg}]$ .
- Plurals of all length and angle units are allowed (for example: feet, radians, etc.).

- Use parentheses when necessary. For example,  $(2*3)*(1/3)$ .
- 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.

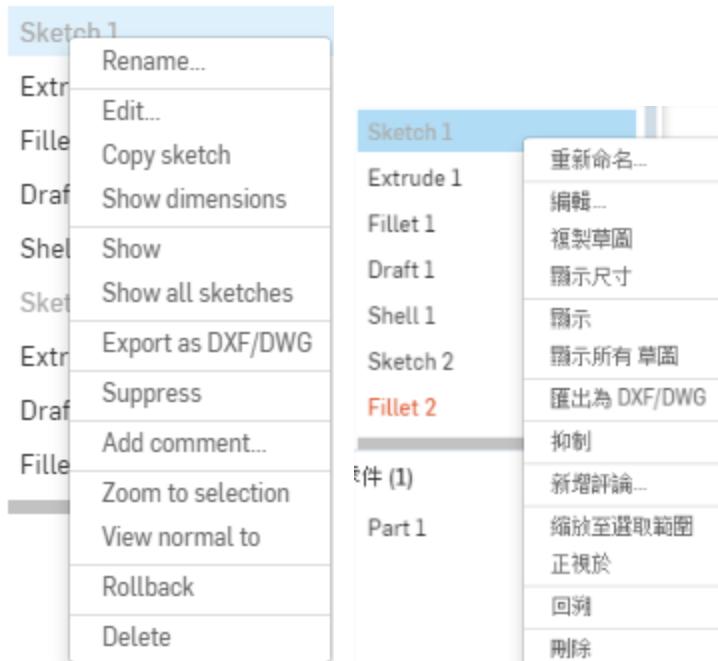
# Context Menus

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, Drawings, as well as Onshape constructs such as tabs. Right-click throughout the interface to discover context menus.

Some of the situations in which you can 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.
- **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.
- **Rollback** - Roll the Feature list back to the selected sketch.
- **Delete** - Remove the selected sketch from the Feature list.

# Create Selection

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 can be used with extrusions, pockets, hole, fillets, tangent connected faces, and bounded faces as the selection criteria. You select one or more faces that the system uses to propagate to select other faces based on the selection criteria. These selections can 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:

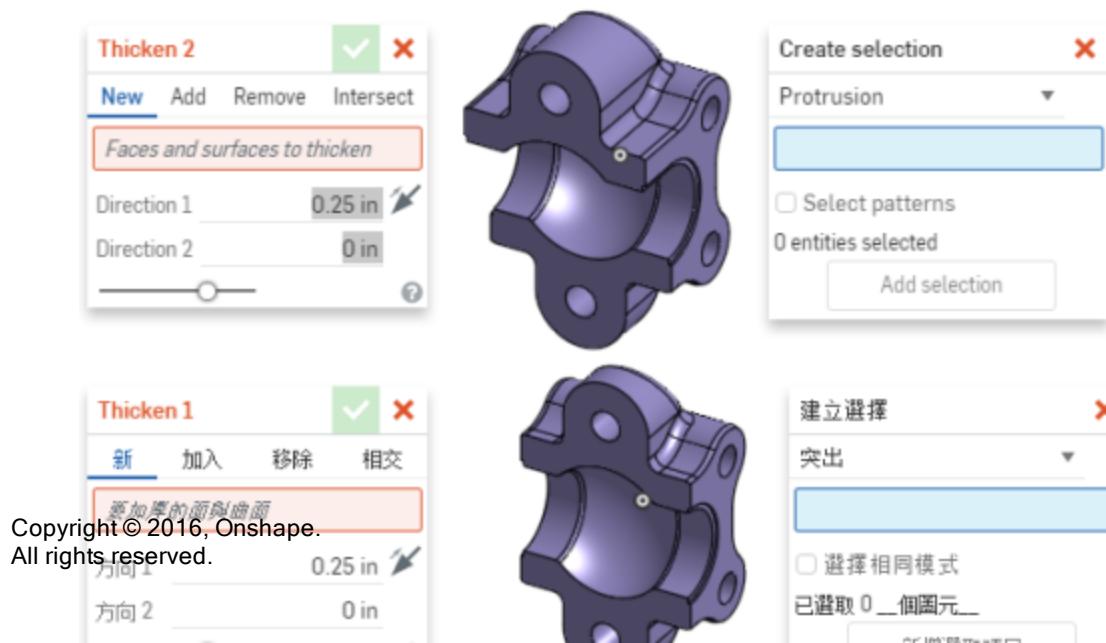
- **Protrusion** - Selects all faces that are connected to the selected face by a convex edge.
- **Pocket** - Selects all faces that are connected to the selected face by a concave edge.
- **Hole** - Selects all faces that are connected to the selected face as part of the same round hole.
- **Fillets** - Selects all faces on a part which form a constant radius fillet.
- **Tangent connected** - 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.

**Select patterns** option - Select all other faces on the part which match the same criteria specified.

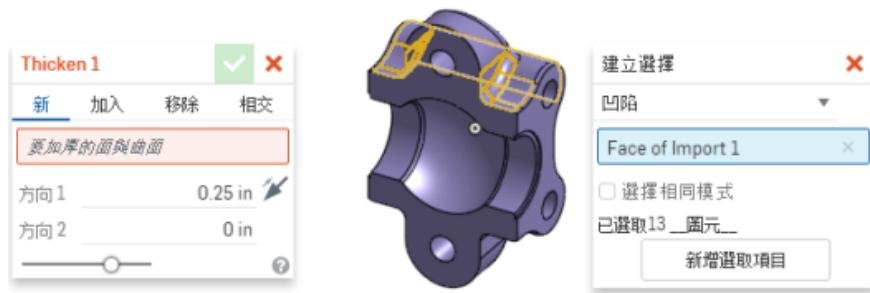
## Example

The example below demonstrates using **Create selection** within the **Thicken** feature.

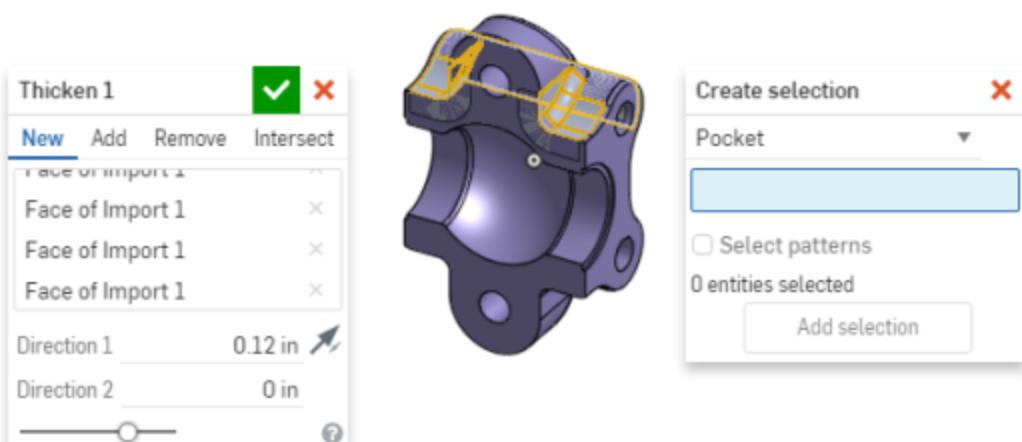
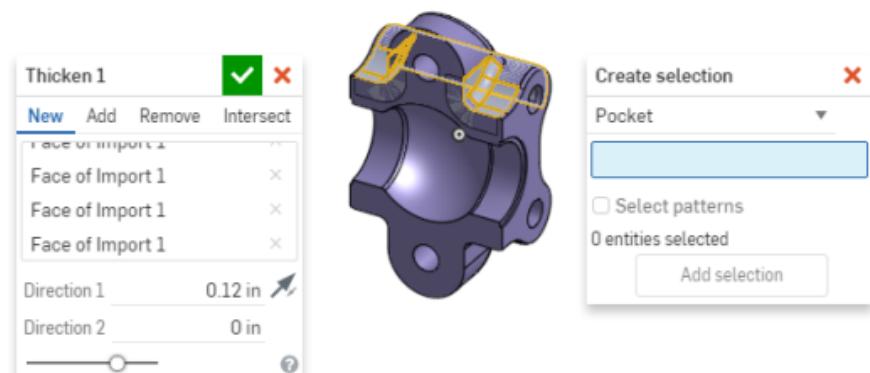
1. Click the Thicken tool.
2. Right-click and select Create selection:



3. Select the type of selection (here, Pocket), select a face, and Onshape automatically makes the appropriate selections.



4. Click **Add selection** to transfer the selected components to the Thicken dialog.



5. Enter the remaining required specifications for the feature.

For more examples, watch the video titled Direct Edit.

There are other selections you can make in addition to Fillet:

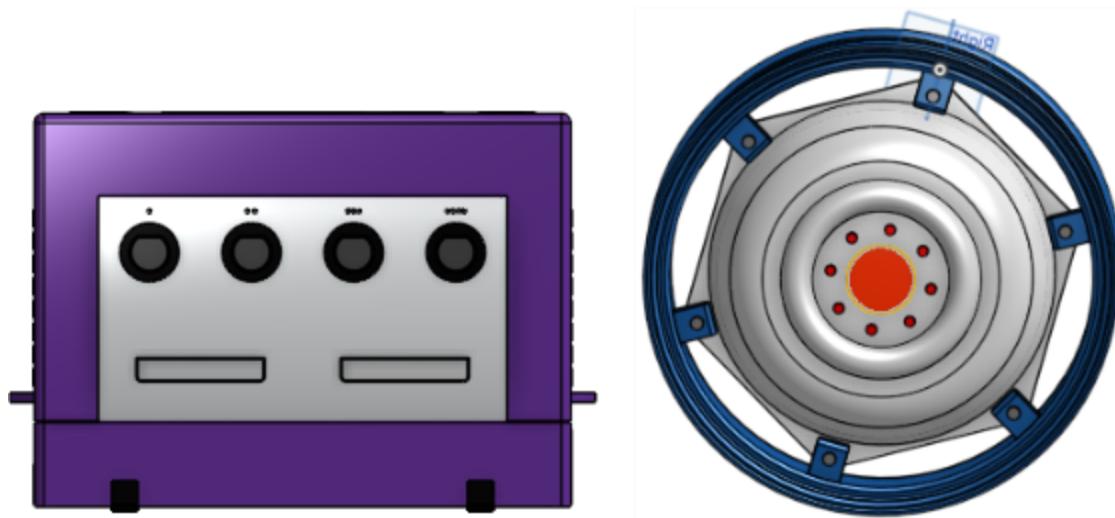
- Protrusion
- Pocket
- Hole
- Tangent connected
- Bounded faces

# Select Other

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.

For example:

The part shown below has many faces that you can't see from this perspective.

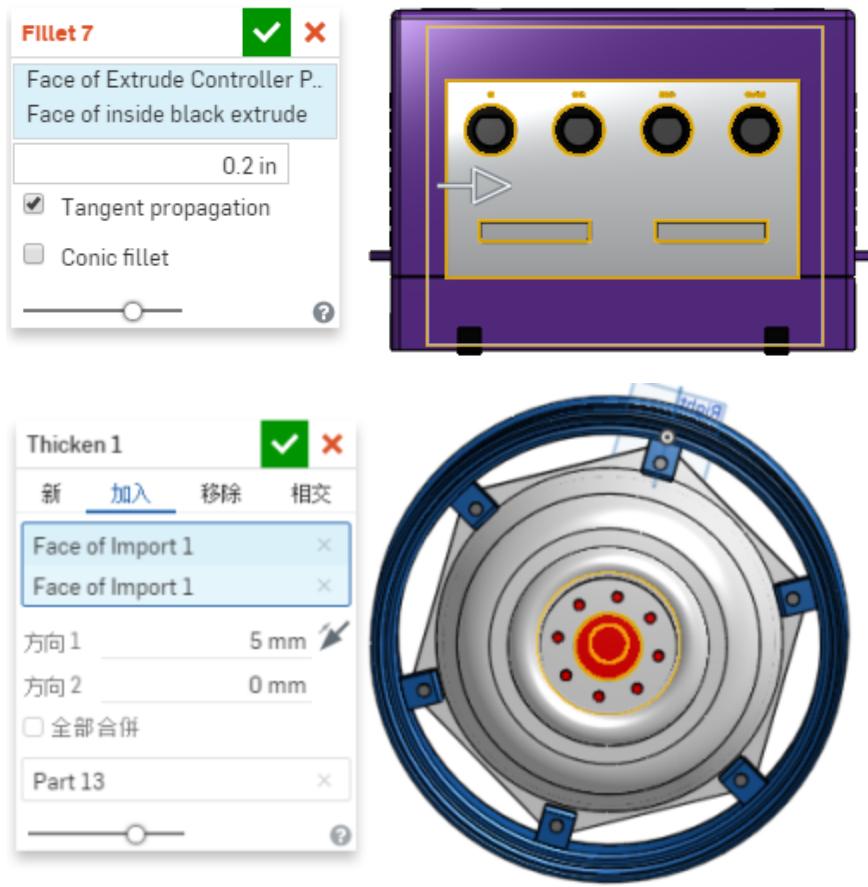


1. Select a face and then from the context menu, then **Select other**:



2. The Select other value list is populated with all faces and edges, working from the one selected down into the part (farther away from your perspective), as shown above.
3. Select the desired entity from the list; the Select other dialog closes.

When you open a Feature dialog, the selections you made in the Select other dialog automatically populate the Feature dialog value list.



You can also open the Feature dialog first, then the Select other dialog.

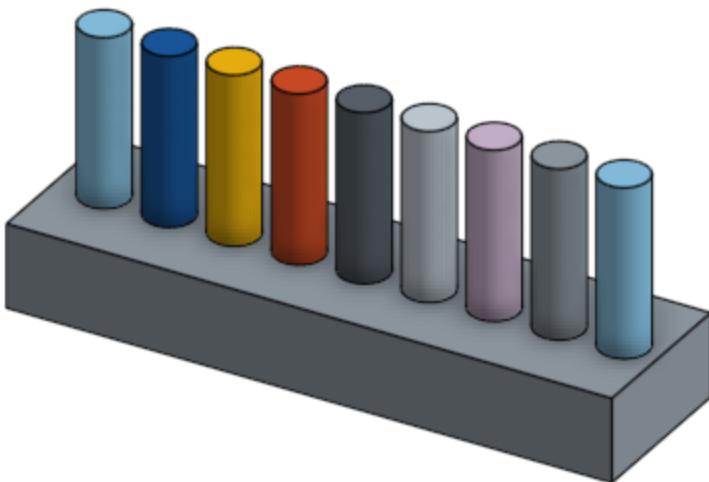
# Customizing Parts:Appearance

Using the context menu for a specific part (or group of selected parts) you can customize not only the color of the part, but also assign a [materials](#) (and thereby, a density) as well.

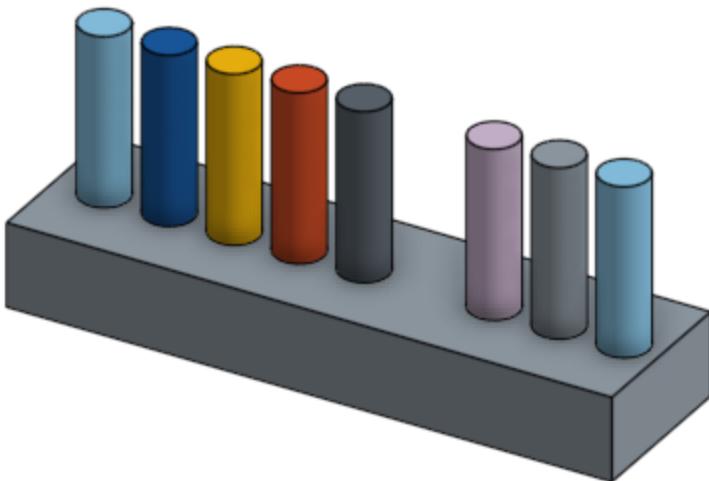
## Default Part Colors

Onshape has a predetermined color palette and rotation of color assignments as parts are created. (You can also assign custom colors to parts, explained below.)

As parts are created, they are rendered in a sequence of eight colors, shown from left to right, with the sequence starting over with the 9th part:



When a part is deleted, the color sequence remains intact with existing parts retaining their color:



## Customizing part colors with the Appearance editor

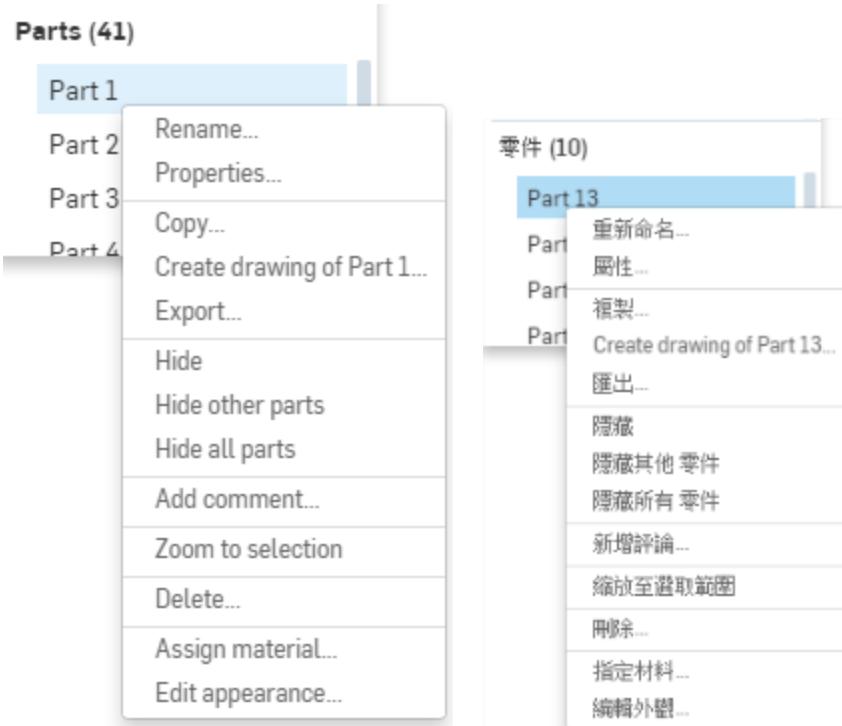
The Onshape Appearance editor enables you to manually assign specific colors to specific parts. Once a color is

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 can 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 can also select multiple parts from the graphics area or from the Parts list to assign appearance characteristics to more than one part at a time.

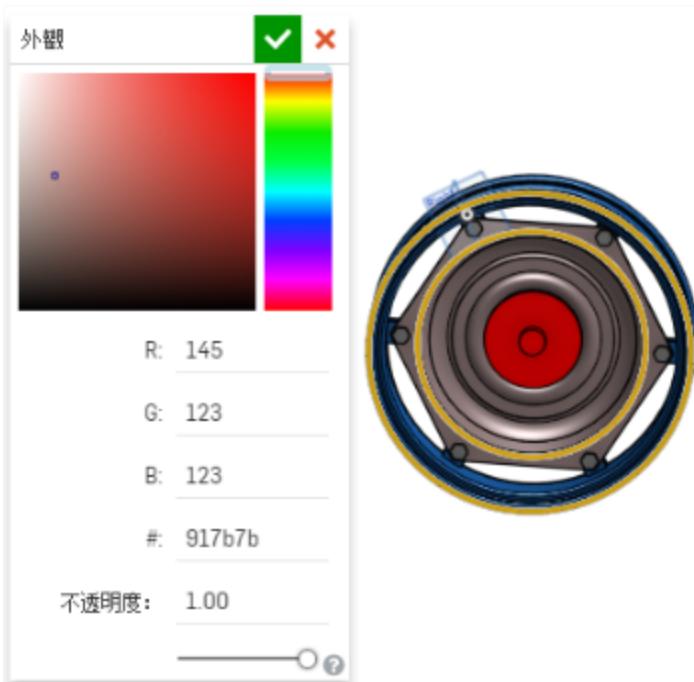


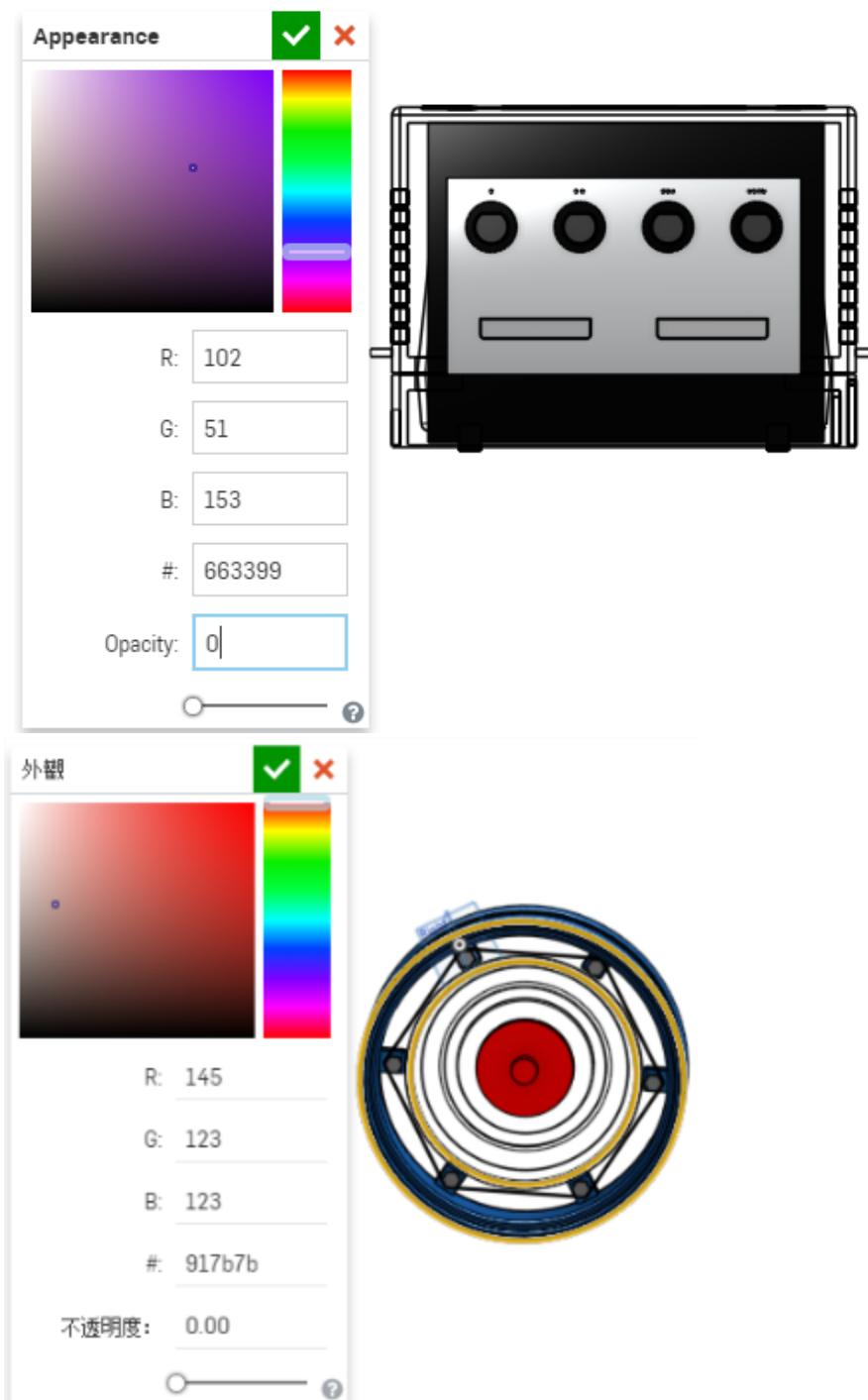
2. Select **Edit appearance**.



3. Specify the RGB values or the hex value for desired colors.

4. Use the Opacity: field to control transparency (on a scale from 0 - 1; use the slider to specify a value):





5. Accept

# Customizing Parts: Materials

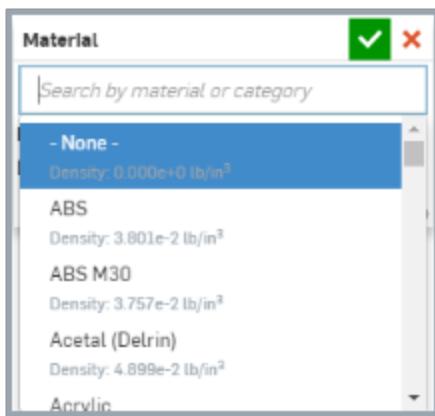
Using the context menu for a specific part (or group of selected parts) you can customize not only the "Customizing Parts:Appearance" on page 64, but also assign a material (and thereby, a density) as well.

## Assigning materials to parts

You can 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 391 then also displays density-related information.

To assign material to a part:

1. Select a part (or group of parts), open the context menu and select Assign material.
2. Select a material from the dialog drop down:

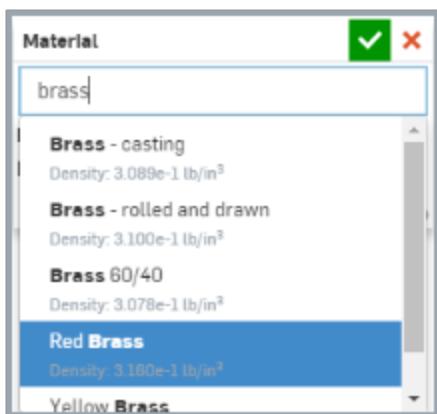


Note that each material has a density value listed with it.

3. Accept .

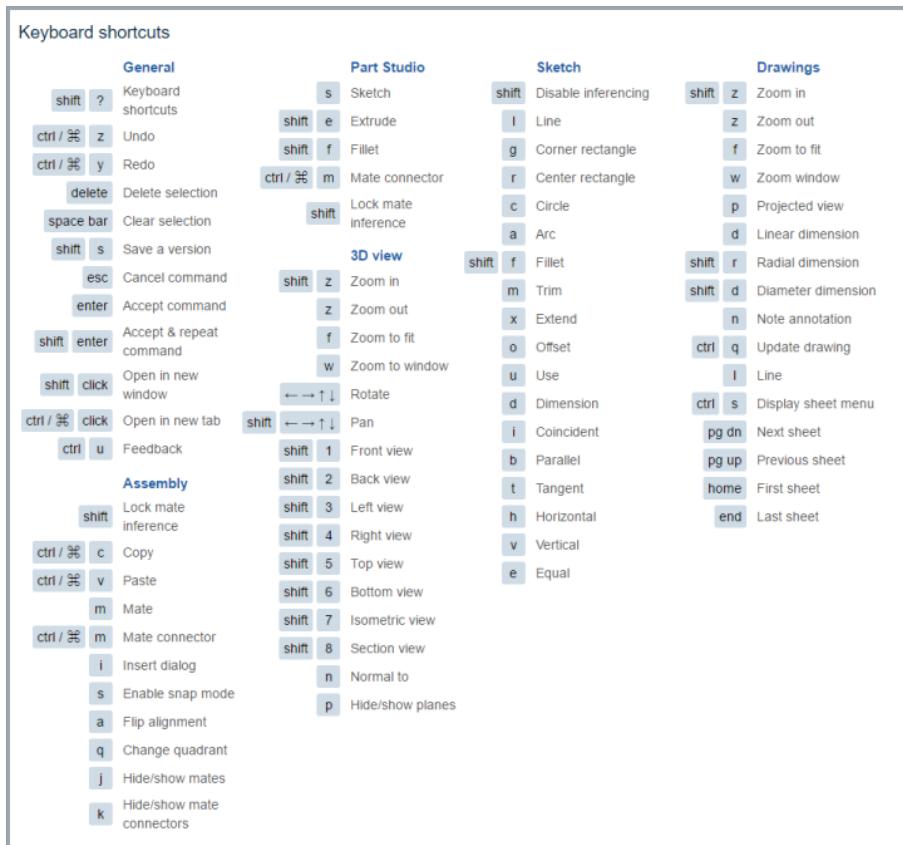
When assigning materials, note that:

- Parts with no material assigned have zero mass.
- Units are shown in the current document units.
- You can search for materials by entering the name or category in the search box:



# Keyboard Shortcuts

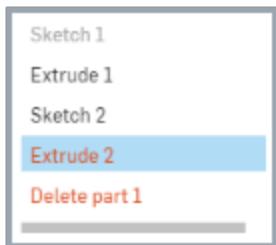
Activate the keyboard shortcuts map right in the user interface by pressing the Question mark key on your keyboard while in a document. You can even pop it out of the window for continuous display:



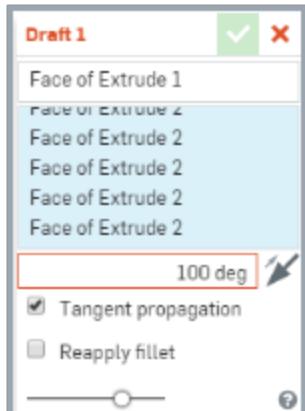
# Error Indicators

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:



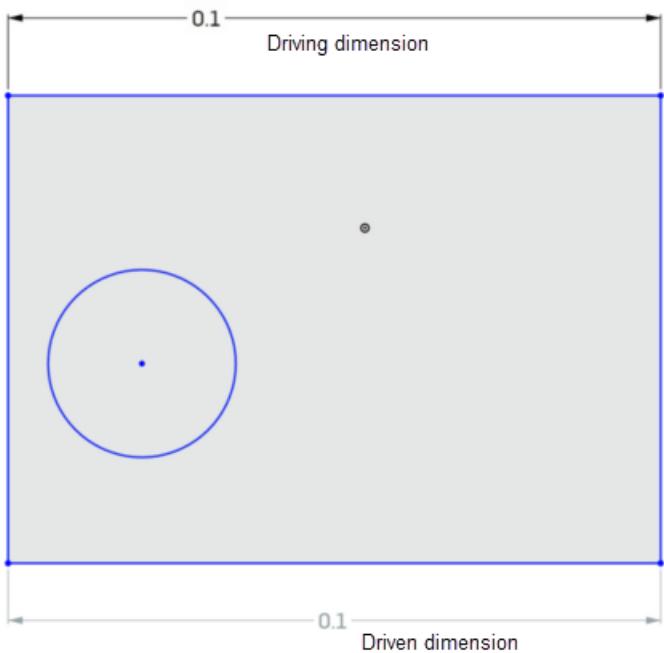
- When the problem lies with a single field, like an invalid entry in a numeric field, that field is outlined in red:



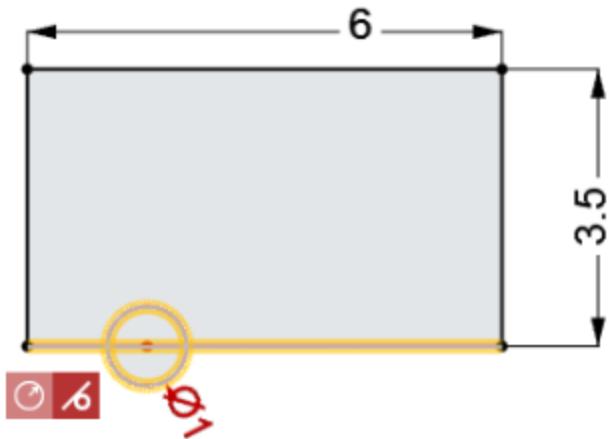
- **Hover information** - When you see orange text in the Feature list, hover over it for a summary of the issue.



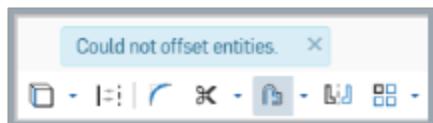
- **Dimension highlighting** - Color is used to indicate the constraint status of dimension: black indicates a driving dimension, and gray indicates a driven dimension.



- **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.



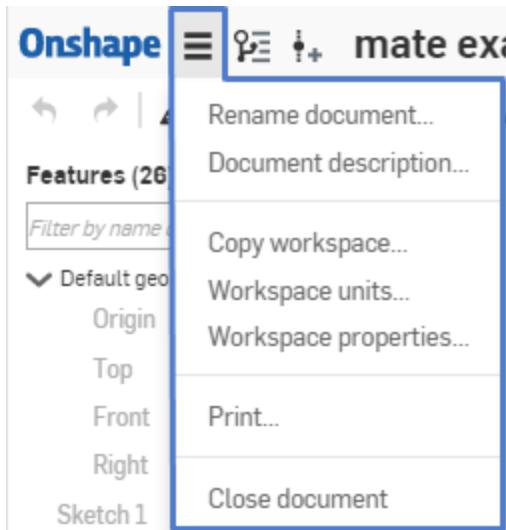
- **Notifications** - When a general system error occurs, a notification appears in a bubble at the top of the user interface window.



# Printing Part Studios and Assemblies

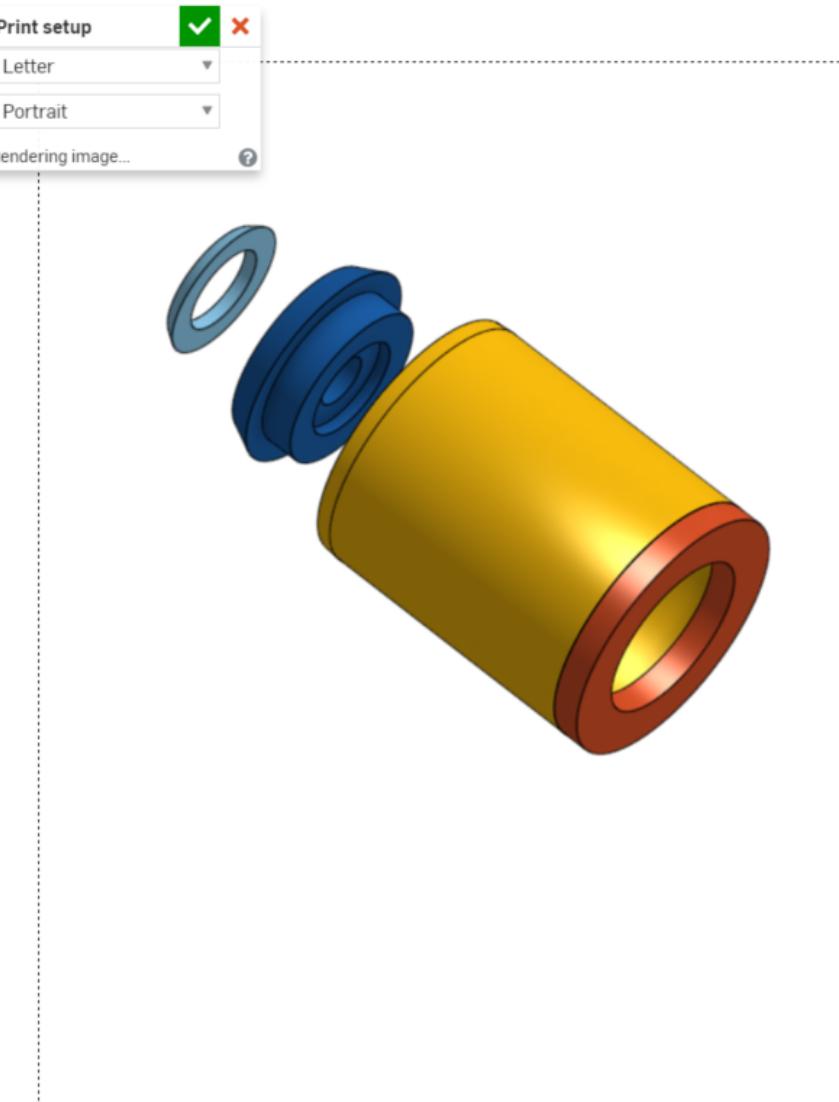
You can print any Part Studio or Assembly in Onshape.

1. Open the tab you wish to print.
2. Expand the Document menu :



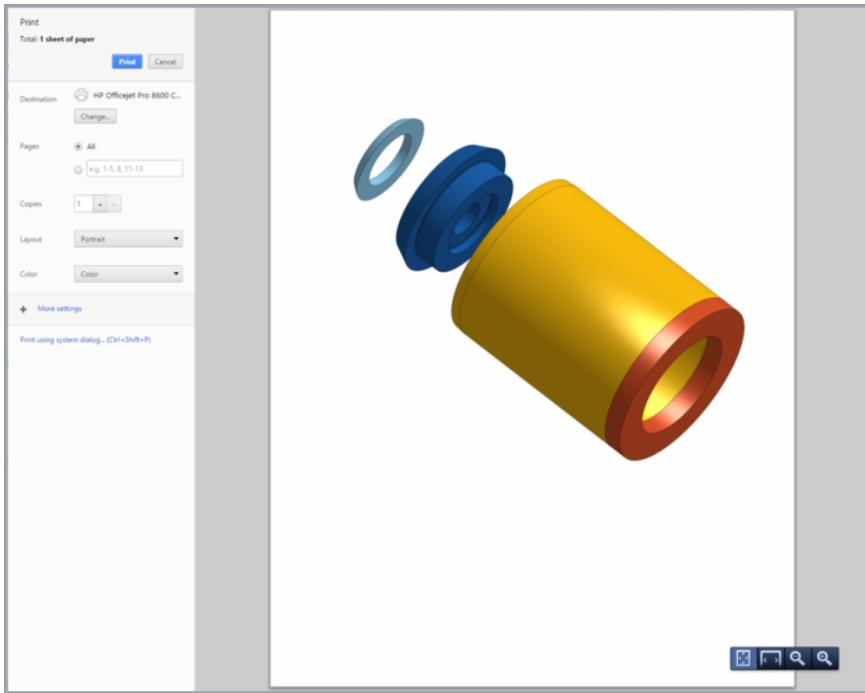
3. Select **Print...**.

A Print dialog opens and a dotted line appears, providing a preview of the printed page border:



4. You can 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):

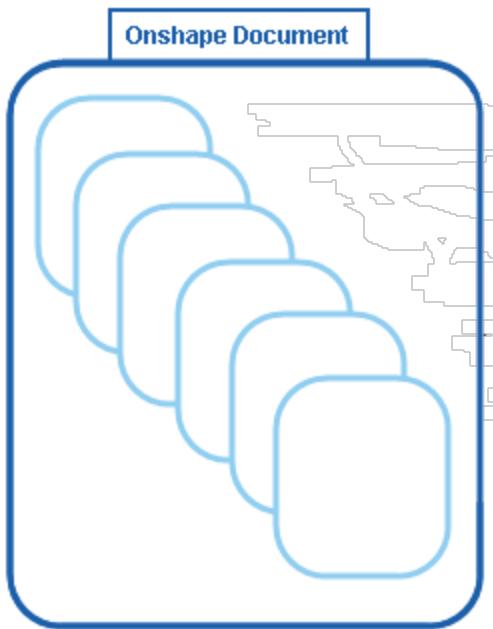


8. Make specifications and click Print.

# Onshape Documents

Onshape has created a new document concept within the CAD industry. Some highlights are:

- **Sketch, and build and assemble parts (solid bodies) in the same document** - All of your work can be done in a single document with complete parametric history.
- **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 document you want to import (including CAD data from another system). All of these elements are shown in separate tabs in an Onshape document.



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



*Jim can share his document with multiple users; all users can be viewing and editing the same document (even the same parts) simultaneously.*

*If needed, the document owner (Jim in this case) can also assign permissions to each user for this document, and also revoke those permissions at any time.*

## Create documents

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 [public](#), and Onshape's own [Tutorial & Samples](#) documents.

Name	Workspace	Modified	Modified by	Owned by	Size
FS Test 2	Main	9:34 AM Today	me	me	961 K...
Document 8b_Perf - Copy	Main	3:57 PM May 13	me	me	163 M...
Untitled document	Main	10:59 AM May 13	me	Documentation L...	363 K...
Untitled document	Main	3:45 PM May 11	me	me	1 MB
Untitled document	Main	3:20 PM May 4	me	Documentation L...	384 K...
DMA-FS-TEst	Main	1:46 PM Apr 29	me	me	4 kB
Redback Spyder Chassis - Copy ...		12:35 PM Apr 28	me	me	2 MB
Redback Spyder Chassis	Main	11:11 AM Apr 29	Philip	Philip	69 MB
Document 8b_Perf	Main	5:34 AM May 16	Aravind	Aravind	163 M...
Sharing test	Main	4:09 PM Apr 26	me	me	12 MB
rel-1.44	Main	10:36 AM Apr 26	me	me	3 MB
2 Test Document	Main	10:58 AM Apr 15	me	me	6 MB
version example	Main	2:44 PM Apr 15	me	me	1 MB
drawing export sheet	Main	10:40 AM Apr 14	me	me	822 K...
Bobber bike	Main	2:27 AM Mar 23	5@t\$4+7	apsar shaik	65 MB
History example	Main	1:56 PM Apr 3	me	me	347 K...
ONSHAPE1	Main	11:09 AM Mar 17	Jon McIntyre	jramsey	25 MB
For use on APS1 - Diane Amadeo..	Main	4:18 PM Mar 17	me	Onshape	3 KB
Sketch Examples	Main	2:06 PM Sep 16	System	me	0 byt...

Subscription: Professional © 2013 - Present, Onshape Inc. All Rights Reserved. Terms & Privacy (1.47.15090.24e03770e)

Click **Create** to create a new documents.

Click the Onshape logo  in the top left corner of the browser window (anywhere in the user interface) to return to the Documents page.

## Keep project information in one document

You can keep all of your project data in one Onshape document if you wish. By default, documents contain a "Part Studios" on page 89 and an "Assemblies" on page 304 (you can create as many as you like 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 first Part Studio in the row 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 can create many parts in one Part Studio and Assemblies can contain subassemblies as well. In addition to these types of data, you can also [import](#) other files into Onshape which will appear each in their own tab, some examples are:

- PDFs
- CAD files
- Images
- Drawings

Within a Part Studio or other document element, you can:

- Duplicate a tab
- Copy a Part Studio and paste it into another document
- "Exporting Files" on page 471 an element (sketch, planar face, part, Part Studio)
- Create a drawing of a particular part or entire Part Studio
- Delete a tab
- Control the order of the tabs (drag and drop)

## Manage documents

Onshape's "Document Management" on page 501 model allows branched editing, and merging, and the ability to restore a document to any point in its editing history through the [Versions and history flyout](#) feature.

Revert a document to a previous point in its history; every action made in a document is saved in the history of the document. Preview a point in a document's history before restoring to that point. Easily reverse the action since the entire history is always available.

## Collaborate

Onshape is designed specifically with collaboration in mind. Documents you create can 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

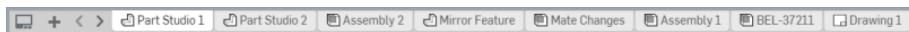
Sharing and permissions can be reversed; all documents that are shared can be unshared and all document you make public can be made private again.

You can also delete documents you own, and also restore them from Trash, or permanently delete them from Trash.

When you create a document, you become the owner. However, when you create a document as a member of a company you can choose whether to own the document yourself, or create it on behalf of the company with the company admins as the owners. Owners of documents can transfer that ownership to other users, pending acceptance of the transfer.

# Document Tabs

Parts Studios, Assemblies, and non-native files imported into Onshape documents are represented in tabs in the user interface. To work in one, click its tab to make it active. Only one is active at a time. Right-click on a tab to access the context menu.

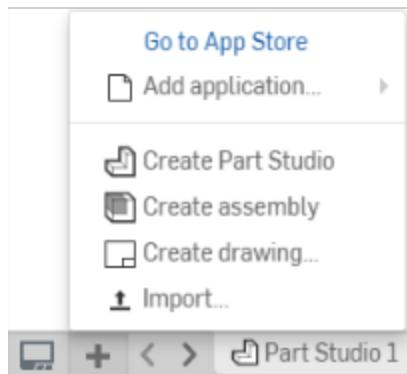


- If there are many tabs to scroll through, use Search tab  to open the Search tabs flyout.
- Click and drag tabs to reorder.

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.

- 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.
- The first tab (leftmost tab) is always the active tab when a workspace is opened.
- A newly created tab is placed directly to the right of the currently open tab and is made active immediately.
- When scrolling through tabs, the active tab is always kept in view.

Click  to create another Onshape tab or "Importing Files" on page 467.

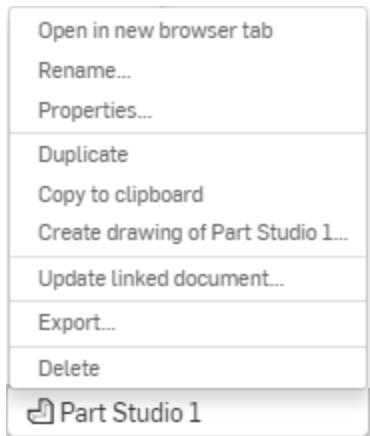


## Acting on tabs

Using the context menu of a tab, you can:

- Open in new browser tab - Open that tab in a new browser tab
- Rename the tab
- Access the Properties for the tab, including Description
- 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
- Export the tab
- Delete the tab, even if it is the currently active tab

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



## Searching tabs in a document

Click  to open the Search tab flyout. The toolbar and Feature list (or Parts list in an Assembly) move to the right and the Tabs flyout opens:

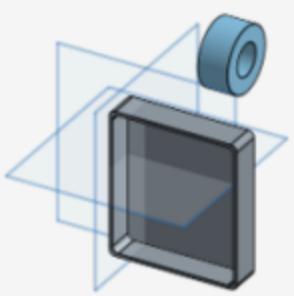
**Tabs** | Search tabs | ▼

Part Studio 1  
**Part Studio 2**  
Assembly 2  
Mirror Feature  
Mate Changes  
Assembly 1  
BEL-37211  
Drawing 1  
Assembly 2 Drawing 1

**Instances (2)**

- Origin
- Part 1 <1>
- Part 2 <1>
- Mate Features (1)**
- > Fastened 1

**Part Studio 2**

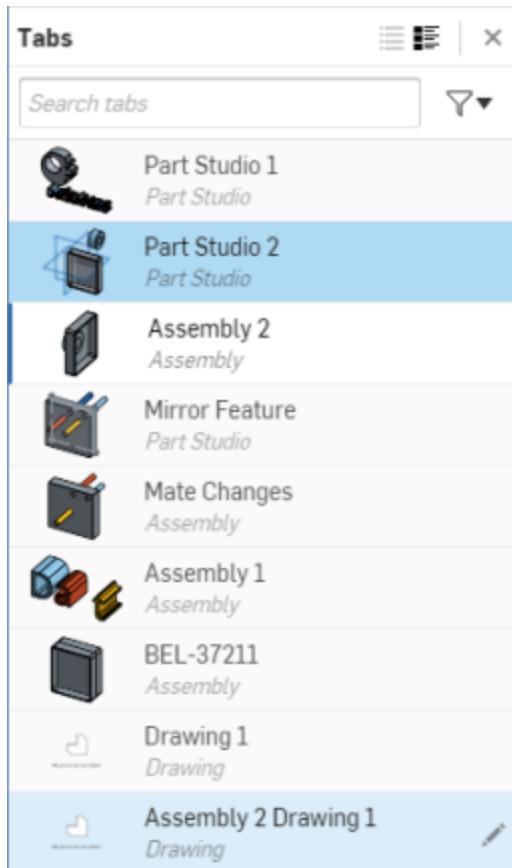


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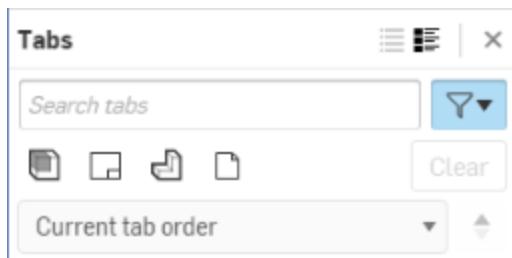
tab or + to paginate to open | +<> | Part Stu

You can:

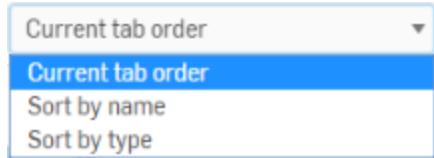
- Enter a partial or complete tab name to find an existing tab. (Onshape employs a type-ahead feature for your convenience.)
- Use to view the tabs as a list, as shown above, with the tab icon indicating the type of tab and a thumbnail preview.
- Use to view the tabs in detail view, shown below, with each line item including a thumbnail preview:



- Use to toggle filters on/off:



- Use the tab icons (shown above) to limit your search to a specific type of tab (Assembly, Drawing, Part Studio, file, respectively).
- User the Current tab order drop down to select how to sort the tabs in the list:



- Click on a tab in the list to open it or use the Tab or arrow keys to navigate to the desired tab in the list and press Enter to open.

# Documents Page

The screenshot shows the Onshape Documents Page. On the left, there's a sidebar with navigation links like 'Create', 'Recently opened', 'My documents', 'Public', 'Tutorials & Samples', and 'Trash'. The main area is titled 'Recently opened' and lists various documents with columns for Name, Workspace, Modified, Modified by, Owned by, and Size. One document, 'FS Test 2', is selected and shown in a detailed view on the right. This view includes a thumbnail of the gear part, owner information ('me'), a description field ('Not shared'), sharing details ('Sharing'), creation date ('Created by me on 149 PM Yesterday'), last modification ('Last modified by me on 9:34 AM Today'), size ('961 KB'), and workspace ('Default workspace Main').

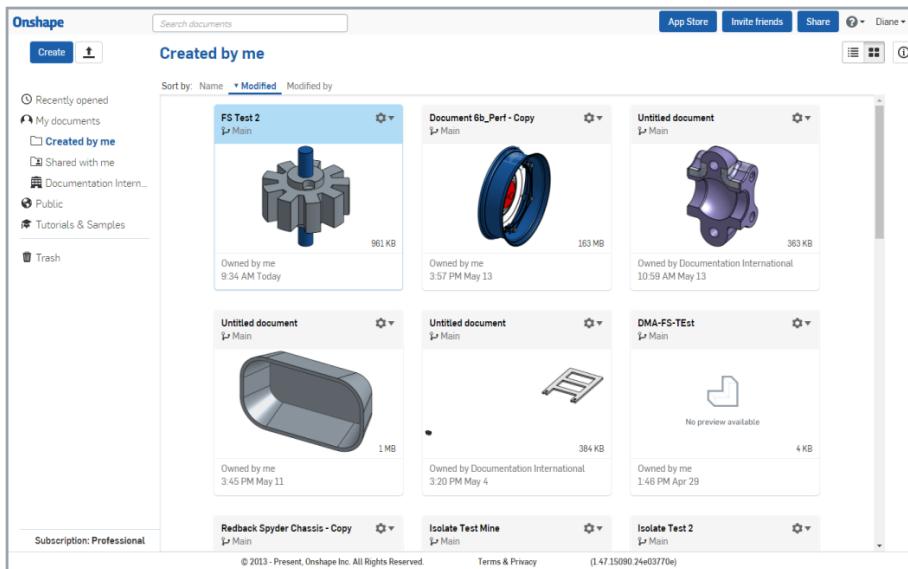
Name	Workspace	Modified	Modified by	Owned by	Size
<u>FS Test 2</u> Main	Main	9:34 AM Today	me	me	961 K...
Document 6b_Perf - Copy	Main	3:57 PM May 13	me	me	163 M...
Untitled document	Main	10:59 AM May 13	me	Documentation I...	363 K...
Untitled document	Main	3:45 PM May 11	me	me	1 MB
Untitled document	Main	3:20 PM May 4	me	Documentation I...	384 K...
DMA-FS-TEst	Main	1:46 PM Apr 29	me	me	4 KB
Redback Spyder Chassis - Copy ...	Main	12:35 PM Apr 28	me	me	2 MB
Redback Spyder Chassis	Main	11:11 AM Apr 29	Philip	Philip	69 MB
<u>Document 6b_Perf</u>	Main	5:34 AM May 16	Aravind	Aravind	163 M...
1 sharing test	Main	4:09 PM Apr 26	me	me	12 MB
rel-1.44	Main	10:36 AM Apr 26	me	me	3 MB
2 Test Document	Main	10:58 AM Apr 15	me	me	6 MB
version example	Main	2:44 PM Apr 15	me	me	1 MB
drawing export sheet	Main	10:40 AM Apr 14	me	me	822 K...
<u>Bobber bike</u>	Main	2:27 AM Mar 23	S@tIS#+7	apsar shaik	65 MB
History example	Main	1:56 PM Apr 3	me	me	347 K...
ONSHAPE1	Main	11:09 AM Mar 17	Jon McIntyre	jramsey	25 MB
For use on APS1 - Diane Amadeo..	Main	4:18 PM Mar 17	me	Onshape	3 KB
Sketch Examples	Main	2:06 PM Sep 16 ...	System	me	0 byt...

- The **Documents** page is the first page displayed upon Sign in. While on any other page, click the Onshape logo to return here.
- If you subscribed to the Free plan, you will see a banner containing information about the Free plan limitations at the top of the page.

## Views

The Documents page offers two types of views :

- List view - (Default, shown above) Presents documents by name in a list view and includes the Detail pane to the right. Click the name to open the document (underlined upon hover), or click anywhere else in the line to select. Hover in line to access the Gear menu to execute actions (see *Actions* below).
- Grid view - (Shown below) Presents documents in a thumbnail view (thumbnails are of the last Part Studio accessed). The Details pane is closed but you can use the icon to open it again. Each thumbnail includes: Document name, owner, time last modified, size and Gear menu. To open a document, click the name. To select a document, click anywhere in the thumbnail.



## Document filters

On the left is a list of pre-defined filters to make finding documents easier. Click one to filter the list of documents:

- **Recently opened** lists documents most recently opened by you or another user with permissions to the document.
- **My documents** lists all documents you have created as well as all documents shared explicitly with you.
  - If you are part of any companies, those companies are listed in this area, by company name, under **My documents**.
- **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.
- If the user is a member of a [company](#) plan or a [team](#), those filters are inserted at this point in the list.
- **Public** lists all documents made publicly available to all Onshape users by another Onshape user.
- **Tutorials & Samples** lists all tutorials and samples 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. Private documents in Trash still count towards your private document limit. Note that any documents you delete from the Trash, and all those present in Trash when you click **Empty Trash**, are deleted forever.
- Click **Details**  to toggle the document details area on and off. You can also **view details** about a specific document, access your **user account**, and **sign out of Onshape**.

## ⚙️ Actions on a document

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 to select a range of documents.

When a document is highlighted, you can use the **Delete** key to move the document to Trash. You can also use the Actions 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](#).
- **Rename document** - Provide a new name for the document.
- **Copy workspace** - Make a copy of the document default workspace.
- **Share** - Share a private document with other users, and assign permissions per user. For more information, see "Share Documents" on page 482.
- **Move to trash / Move selected to trash / Move this document and selected to trash** - This option changes depending on selection of documents:
  - **Move to trash** - Move the single currently selected document to trash.
  - **Move selected to trash** - Move the multiple-selected documents to trash.
  - **Move this document and selected to trash** - Move the document currently highlighted and other multiple-selected documents to trash.

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.

- **Transfer** - Transfer ownership of a document to another user, pending the acceptance of that user, via email notification.

The actions available to you may change based on permissions.

You can also "Share Documents" on page 482, "Invite Friends" on page 500 to try Onshape, and "Importing Files" on page 467:

- **Share** **Share documents** - 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, they are sent an invitation to sign up for an account and automatic Free plan and open your document. You can also make a document public, that is, available to all Onshape users.
- **Invite friends** **Invite friends** - Send an email invitation to a friend explaining how to sign up for an account and automatic Free plan .
- **Import files** - Import other CAD files, as well as any other type of file, into Onshape.

# Set Default Units

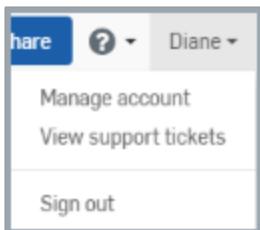
Onshape defaults to inch/degree for units of measure for all documents and workspaces; this encompasses all measurements in Part Studios and Assemblies, all values displayed in sketch dimensions, and the default input units for all features as well. (These default units do not affect any external files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

In addition to setting default units for all documents you create (through this Settings tab), you can also change and specify default units for a specific *workspace* in a document through the "Toolbars and Document Menu" on page 42 in a document.

You can also 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.

## Set default units for *all documents* you create

1. Expand the User menu and select **profile**:



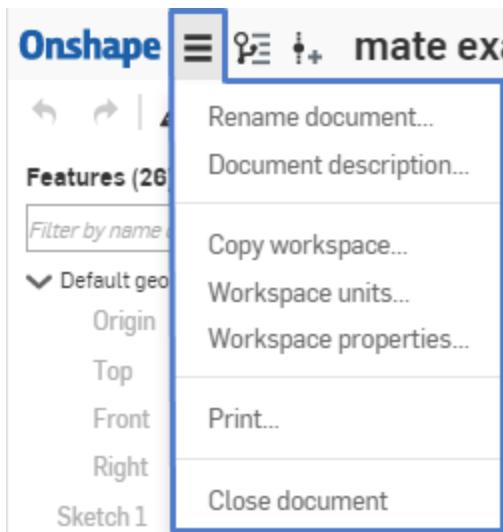
2. Make appropriate selections:

A screenshot of the 'Account settings' page. The left sidebar has a 'Preferences' tab selected, which is highlighted in blue. The main content area shows the 'Units' section with three dropdown menus: 'Length units' set to 'Inch', 'Angle units' set to 'Degree', and 'Mass units' set to 'Pound'. At the bottom right of the section is a blue 'Save units' button.

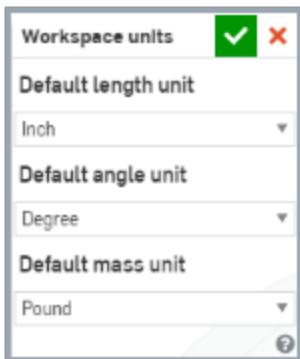
Use the browser Back button to return to the graphics area or click the Onshape logo to return to the Documents page.

## Set the default units for a *specific* workspace in a document

1. Open the Document menu  and select Workspace units.



2. Make appropriate selections (as above).
3. Click  to save changes, or  to exit without saving.



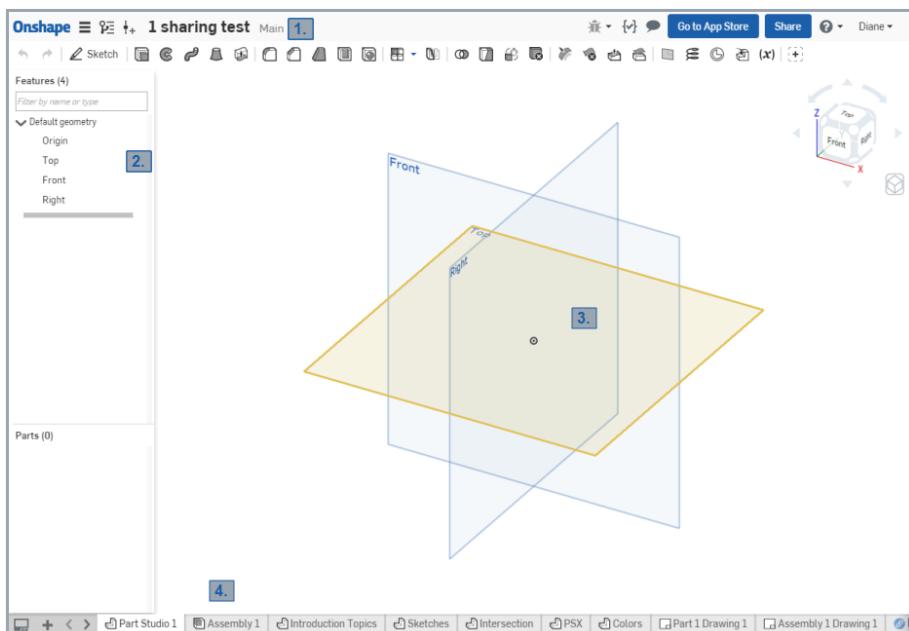
# Part Studios



When you first enter a Part Studio, the Feature toolbar is shown. Most new parts begin with a sketch.

To start sketching, first select the **Create new sketch** tool in the **Feature** toolbar.

An Onshape Part Studio is used to define Parts. It is named and identified as an Onshape tab, located in a tab in the Onshape system. You can see an annotated Part Studio below.



1. 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.
2. The Feature list contains the default geometry as well as any features you create. The thick bar is the Roll-back bar and can be repositioned to generate the Feature list up to its location in the list.
3. The default geometry (origin and planes).
4. The Tab manager area: 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).

When the window is smaller, the Sketch tool may be resized to .

For more information about sketching, see "Sketch Tools" on page 103.

## Part Studio context menu

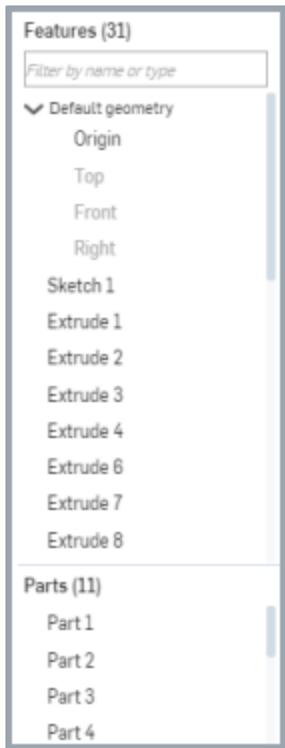
Right-click on a Part Studio tab to access the context menu:

- Open in new browser tab - Open this Part Studio in a new browser tab
- Rename - Access the dialog to rename this Part Studio
- Properties - Access the dialog to provide information about the Part Studio:
  - Description, Part number, and Revision.

- 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  (Create tab) 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.
- 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.

# Feature List

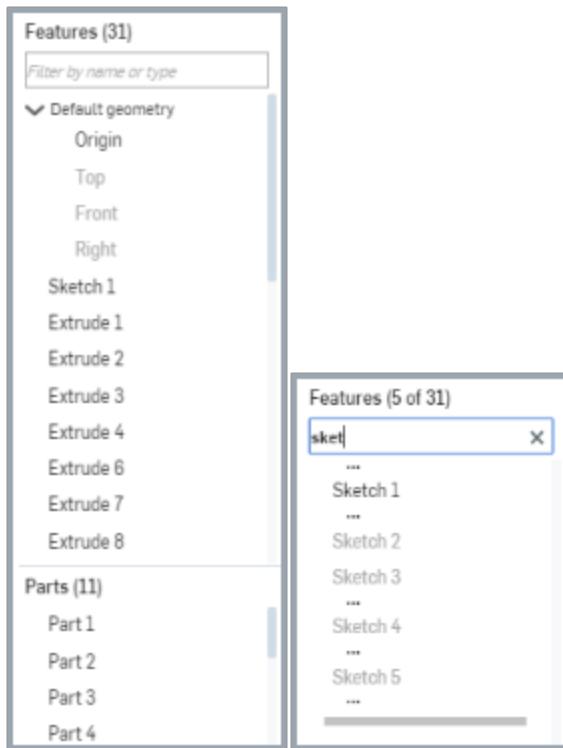
The Part Studio Feature list consists of a list of features and a list of parts:



## 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:

- **Search for features** - Use the search box at the top of the Feature list to filter the list; the ellipsis that appears indicates there are more entries, hover over the ellipsis to view those filtered out features:

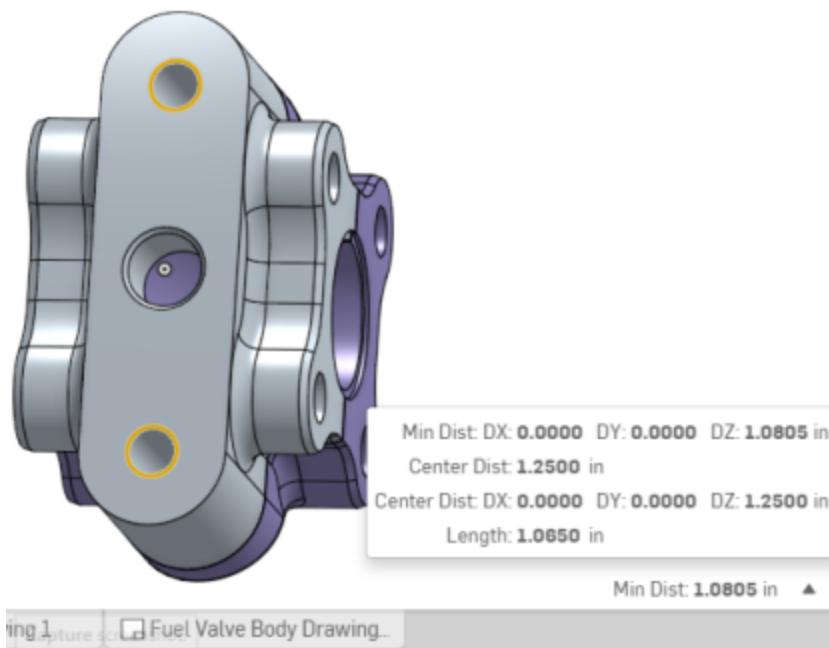


Search terms you can use are:

- **:name myname** - Find all features with the name that matches *myname*
- **:type mytype** - Find all features of the type *mytype* (for example: fillet, extrude, etc)
- **:part mypart** - Find all features contributing to part *mypart*
- **:allparts** - Find all features that generate or affect a part in some way (for example: fillets, splits, transforms would be found and sketches and construction planes would not be found)
- **Drag the rollback bar** - Visualize a model at the point of the rollback bar; all features listed beneath the rollback bar become temporarily suppressed.
- **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 a feature/sketch name in the 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 can also hover next to the name and click the icon

# Measure Tool

The Onshape measure tool is available in Part Studios, for sketches and parts, and in Assemblies for parts and assemblies; it appears in the bottom right corner of the interface when a selection is made:



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, as shown above.

Note the triangle on the measure tool. Clicking it shows more measurements.

You can use the information displayed to enter values elsewhere in the system, for example, as a dimension.

With the Measurement dialog expanded, highlight the value you want to copy and use hot keys or context menu to copy.

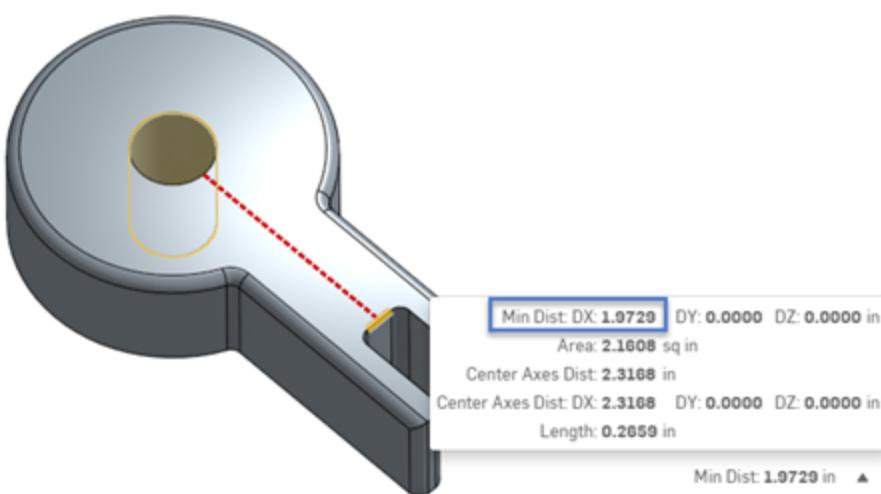
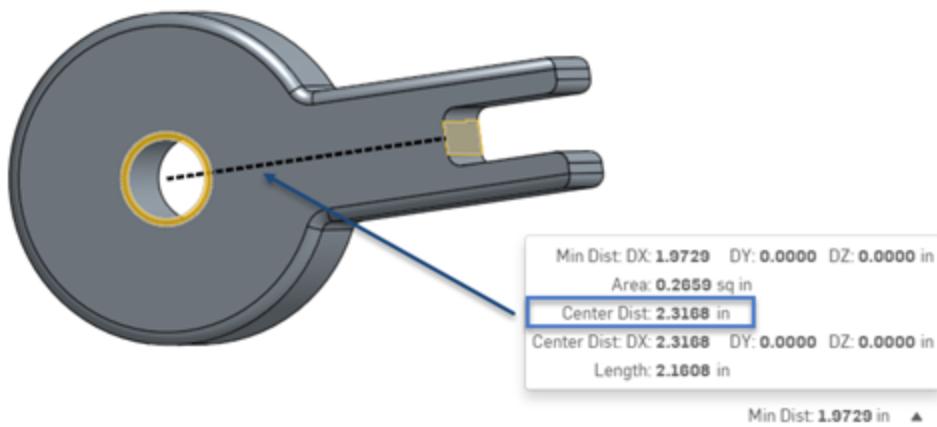
## Interpreting the measure information

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:

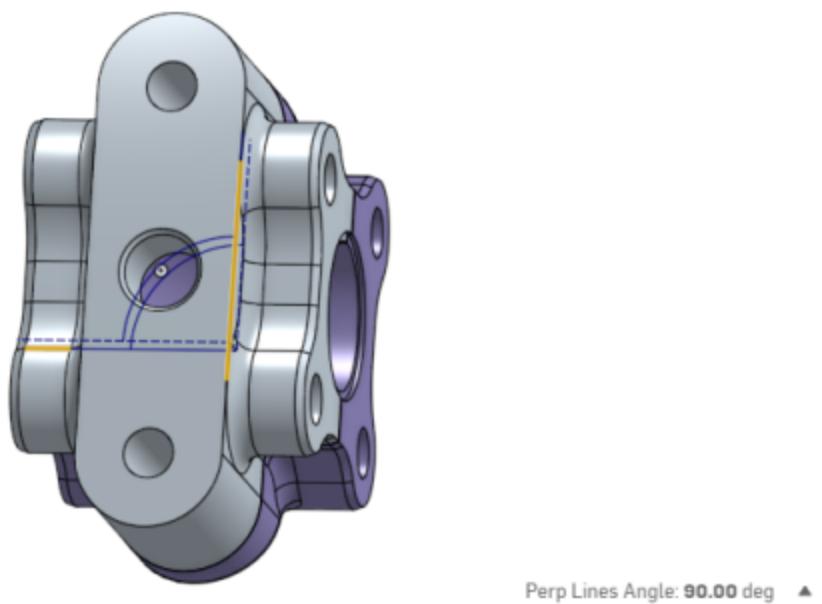
Minimum distances between entities are shown as bold dotted lines:

- 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:



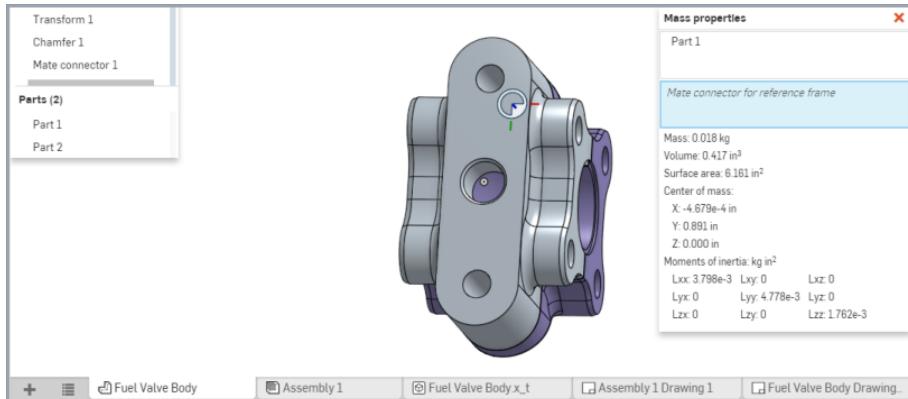
# Mass Properties Tool

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.

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 material assigned, no figure for that part is used in the calculation (and a note is displayed in the flyout to that effect).

Materials can be applied to parts through the context menu on a part in the Parts list (or the graphics area)

To access the Mass Properties dialog, select a part in the Parts list and click the small scale icon that appears in the bottom right corner of the interface:



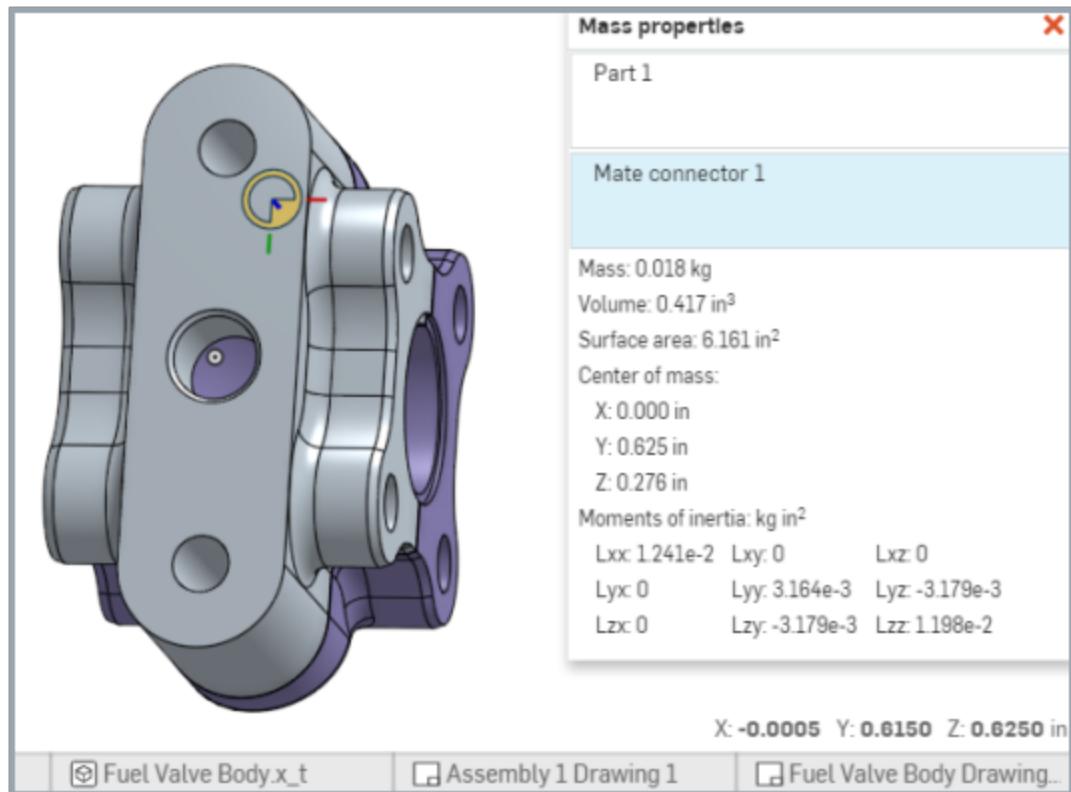
For any intersecting parts, the properties are calculated for each individual whole part and added together.

You can use the information displayed to enter values elsewhere in the system, for example, as a dimension:

With the Mass properties dialog expanded, double-click the value to copy and use shortcut keys to copy to clipboard.

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.

## Steps

1. While in a Part Studio (or Assembly), select a part (or many parts) from the Parts list (or Instances lists).
2. Click the Mass properties icon in the lower-right corner of the user interface to access the information in the flyout.

# Sketch Basics

When creating sketches in Onshape, you use this [Sketch tools](#) toolbar:



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 can rigidly transform geometry in an active sketch simultaneously through the context menu once sketch entities are selected.

You can 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 .

## Basic workflow

You can 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  and notice the Sketch dialog opens:



2. Select the plane to sketch on (you can sketch on only one plane at a time).
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).

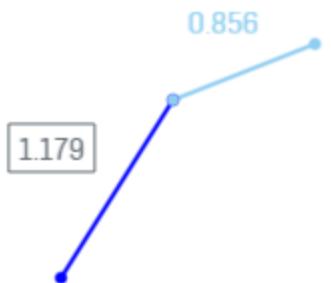
Some tools allow you to specify dimensions while you sketch, for example:

Sketch dimension appears as the sketch curve is drawn.

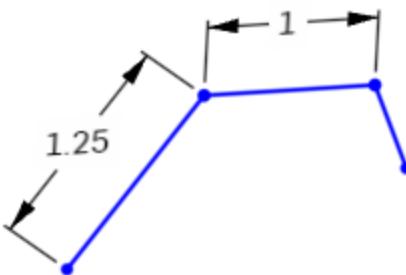


When the sketch curve is drawn the suggested dimension value appears in a box. Type a value (or expression) to dimension the sketch curve.

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 dimension a sketch at a later time using the [Dimension tool](#).
5. Use [automatic inferencing](#) to apply constraints while sketching.
  6. Add manual constraints as appropriate.
  7. Accept the sketch and close the dialog with

### Inferencing while sketching

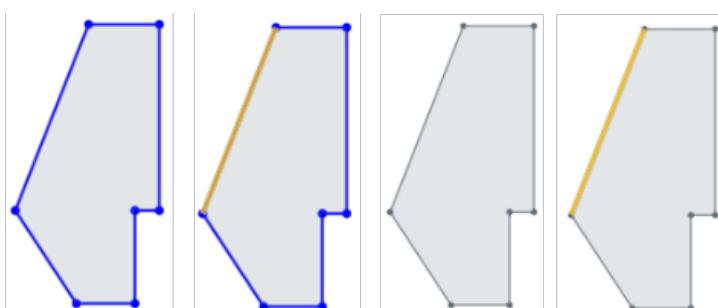
As you sketch and pass over points or lines, you may awaken inferences.

## Line styles

As you sketch and then create models, you 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.

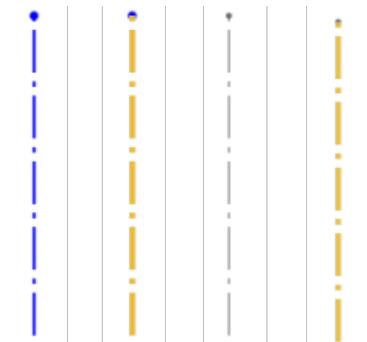
### Sketch lines

Active sketch	Selected line, active sketch	Inactive sketch	Selected line, inactive sketch
---------------	------------------------------	-----------------	--------------------------------



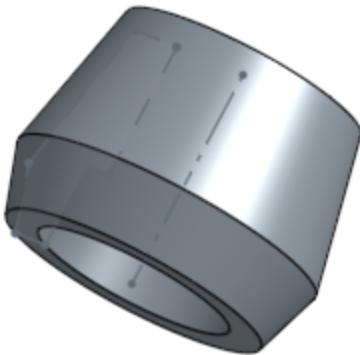
## Construction lines

Active	Active, selected	Inactive	Inactive, selected
--------	------------------	----------	--------------------



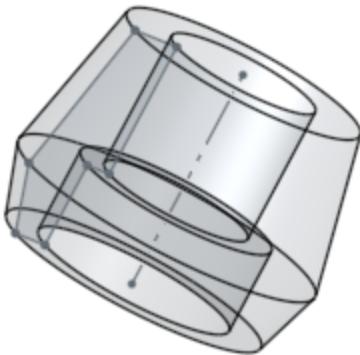
## Sketch lines obstructed by model geometry

The single line in the middle of the part, below, is a construction line.



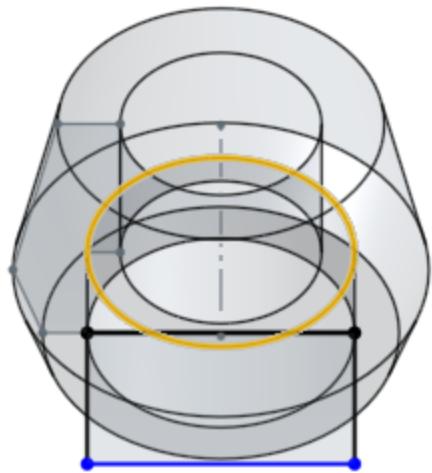
## Hidden lines

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).



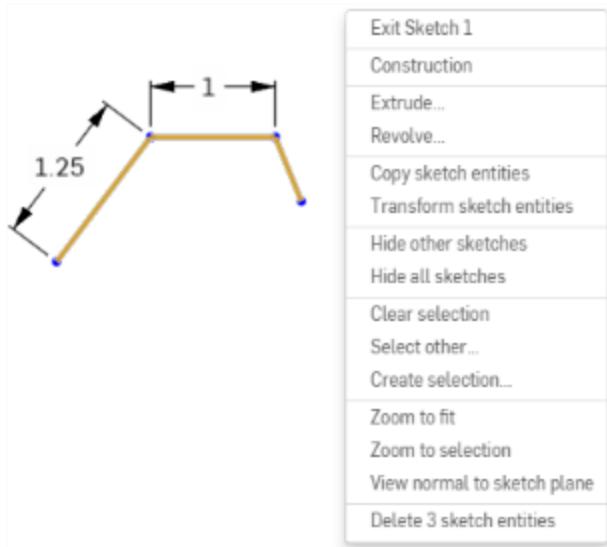
## Used edges (projected edges)

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:

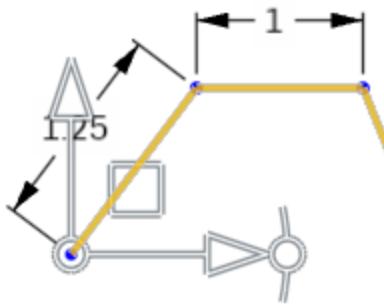


## Transforming sketches

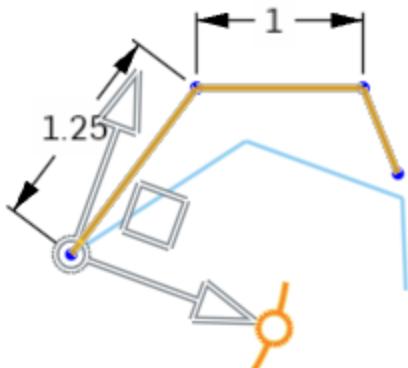
Use the context menu > Transform sketch entities command (available when at least one sketch entity is selected) to move sketch entities simultaneously.



The manipulation triad appears, drag to manipulate selected sketch entities:

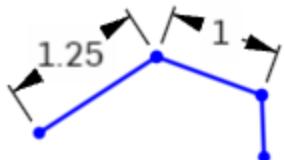


- The center of the triad is used for free drag, allowing for repositioning of the triad without changing the transform operation. Free drag snaps to sketch inferences, and normal drag does not.

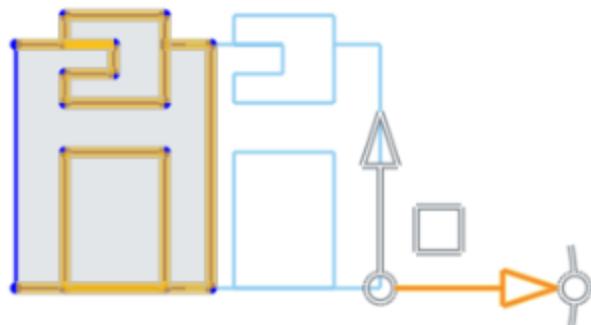


Drag the highlighted (above) angle indicator to rotate the sketch.

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 :



For more information, see " Transform Sketch" on page 145.

## 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.
2. Make Part Studio B active.
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 can also indicate that you want to receive email notifications of other user's 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 Tools

The **Sketch toolbar** appears when you:

- Create a new sketch by clicking the **Create new sketch tool**  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.

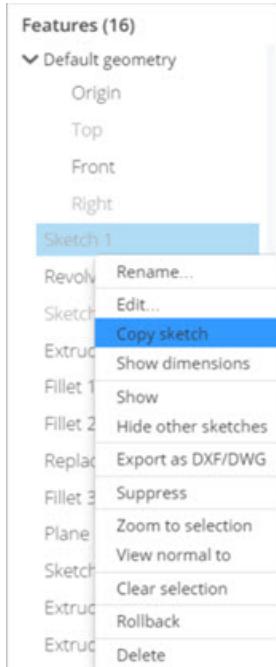
A small arrow beside a tool icon indicates a drop-down menu:



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.

## Tips

- The Escape key exits a tool selection.
- You can apply constraints (including dimensions) between sketch curves and planes.
- You can 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:



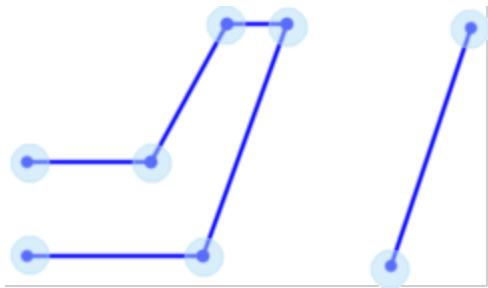
For more information, see "Context Menus" on page 57.



Shortcut: L



Sketch a line segment or series of line segments. Click to begin and end line segments, continuing to create attached segments or click and drag to start one segment and release to end.



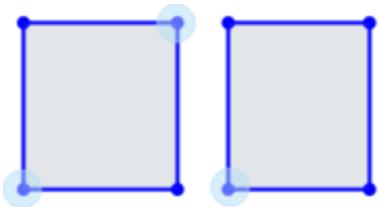


## Corner Rectangle

Shortcut: g



Sketch a rectangle starting with a corner point. Click to start a corner, click to end at diagonal corner or click and drag from corner to diagonal corner and release.



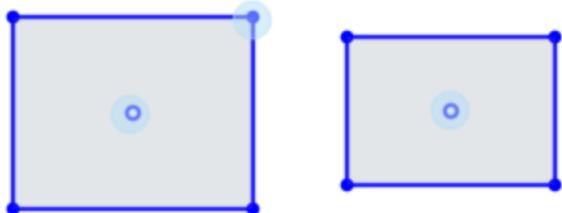


# Center Point Rectangle

Shortcut: r



Sketch a rectangle starting with its center point. Click to set the center point, click again to set a corner or click and drag from center point to corner and release.



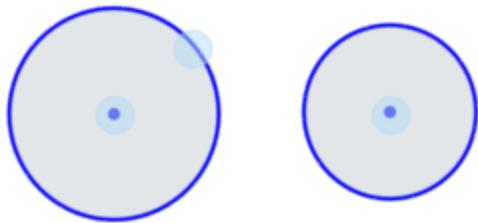


## Center Point Circle

Shortcut: c



Sketch a circle starting with its center point. Click to set the center point and click again to set the radius or click and drag to set the center point, release to set the radius.

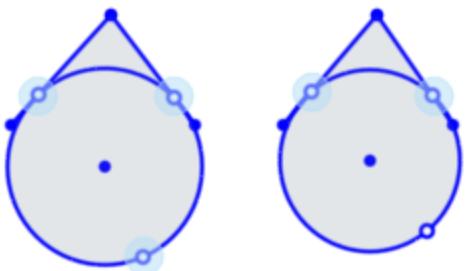




## 3 Point Circle

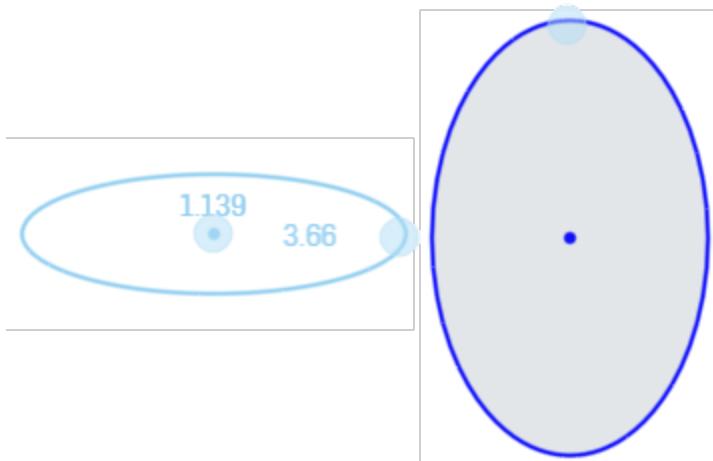


Sketch a circle by defining three points along its circumference. Click to set start point, click to set second point and click to set diameter or click and drag to set start point, release to set diameter and click to set third point.



 **Ellipse**

Sketch an ellipse using a center point, major axis, and minor axis. Click to initiate ellipse, then drag and click to set desired major axis, then drag and click to set minor axis.





## 3 Point Arc

Shortcut: a

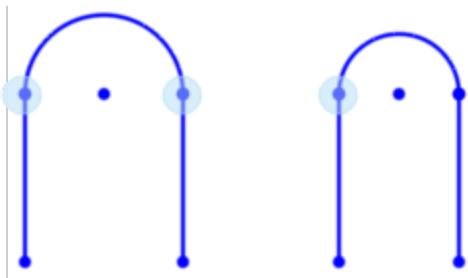


Sketch an arc by defining the two end points and then the radius point. Click to set start point, click to set second point and click to set radius or click and drag to set start point, release to set second point and click to set radius.



 **Tangent Arc**

Sketch an arc at the end of a line. Click to start and click to end or click and drag to start, release to end.

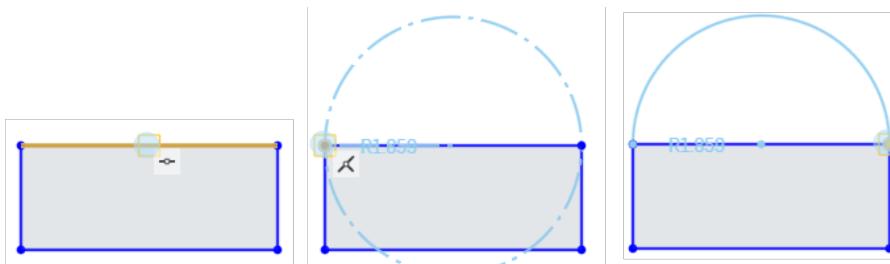




## Center Point Arc



Sketch an arc by defining center, start, and end points. Click a center point on a sketch entity; click start point, click end point.

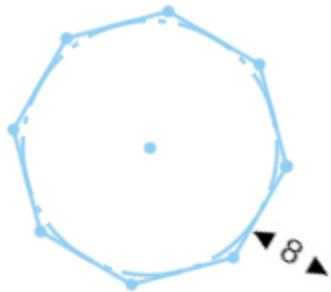




# Inscribed Polygon



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.

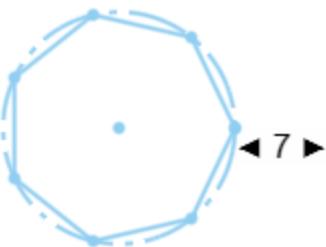
Drag away from the polygon to increase the number of sides, click to set. Maximum sides = 50.



# Circumscribed Polygon



Sketch a **circumscribed polygon** (polygon on the inside of the drawn circle).



## Steps

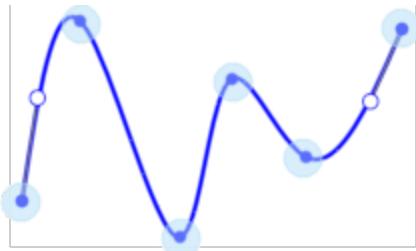
1. Click .
2. Click to start.
3. Drag to set the circumference.  
Notice that the circle drawn for the polygon uses the construction flag.
4. At this point you have two options:
  - a. Use the keyboard to enter the number of sides. Then click to lock the circumference and a value field appears for the 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.

 **Spline**

Sketch a multiple point curve with points along its length. Click and drag any point along the spline to make adjustments. Click to start, click to establish points, double-click to end.

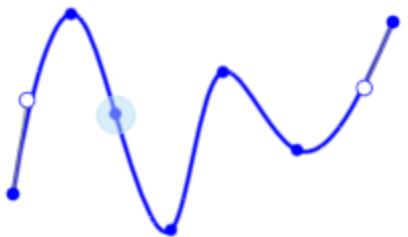


# Spline Point



Add points along a spline.

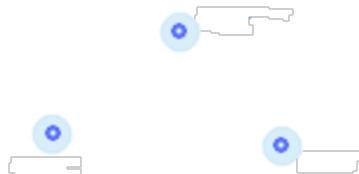
You can click anywhere along a spline to add points to it. Drag the points to modify the spline.



## ○ Point



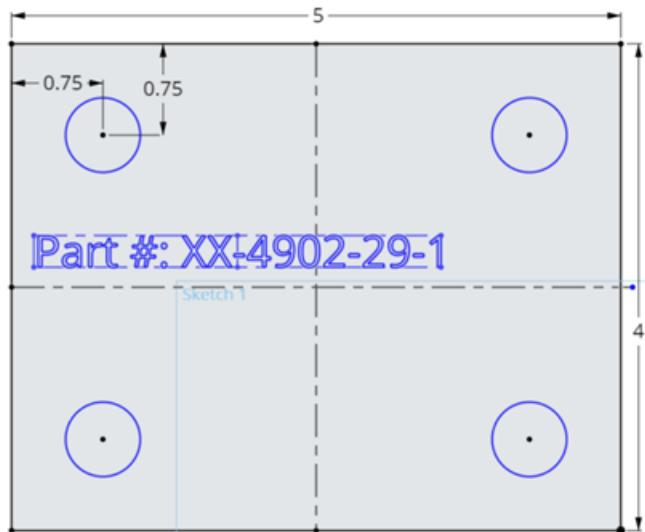
Create points. You can apply many of the same sketch constraints that you can to other sketch entities.



# Text

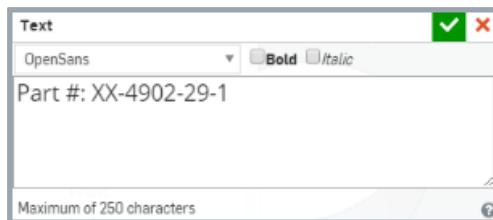


Add up to 250 characters of text to a sketch (you can copy and paste into the text dialog). Treat sketch text as most other sketch entities: extrude, dimension, and apply constraints.



## Steps

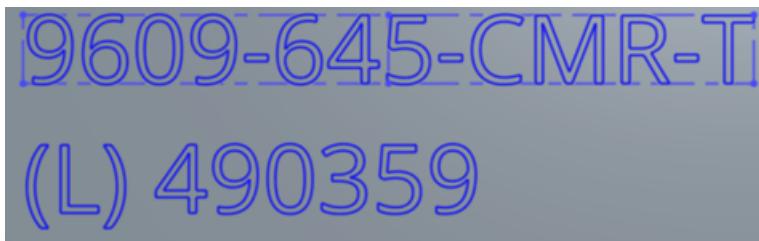
1. Click .
2. Click and drag to establish the position and size of the text box. (The lower left corner and the height define the text position and size of the first line of text.)  
Note that a horizontal constraint is applied to the lower edge of the box by default.
3. In the dialog that appears:
  - a. Enter the text as you wish it to appear.
  - b. Select a font from the drop-down menu.
    - Be sure to select a font that supports your language.
  - c. Optionally, select styling: Bold, Italic.





## 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 can 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 can not 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.



# Use

Shortcut: u

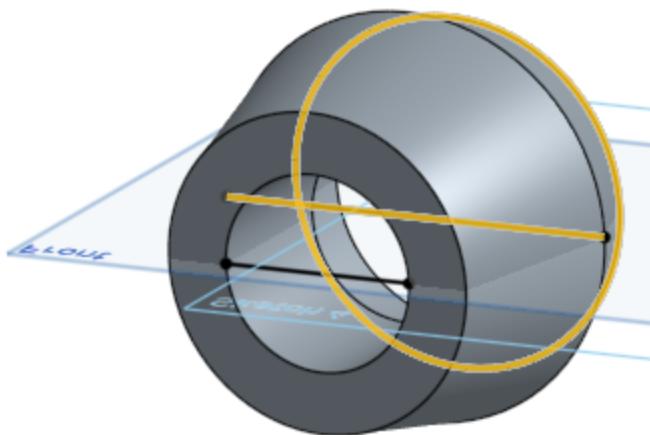


Project (or convert) an edge, edges and silhouettes of a part or sketch onto the active sketch plane.

## Steps

1. Create a sketch or part.
2. Start another sketch.
3. Click Use, then an edges, 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.

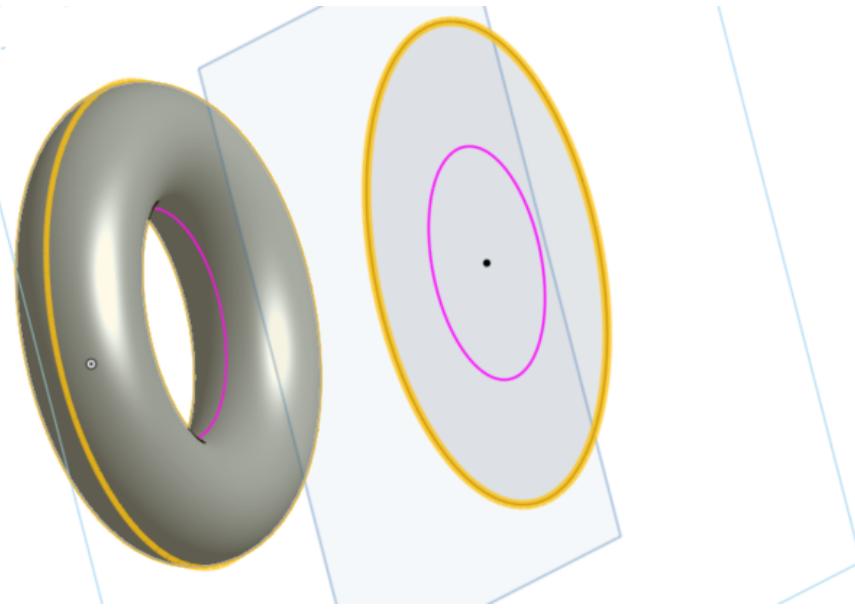


## Using silhouettes

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.

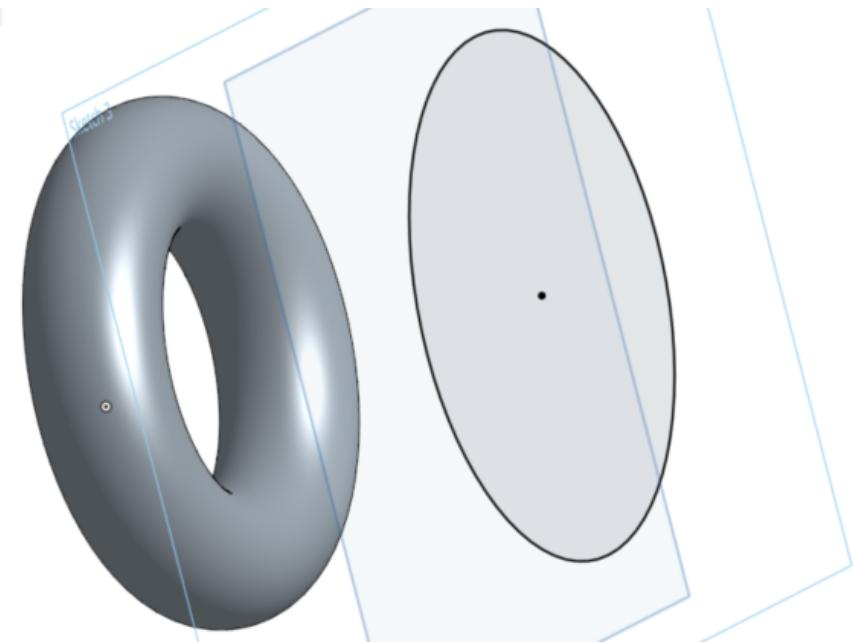
## How does using a silhouette work?

1. Click
- 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.



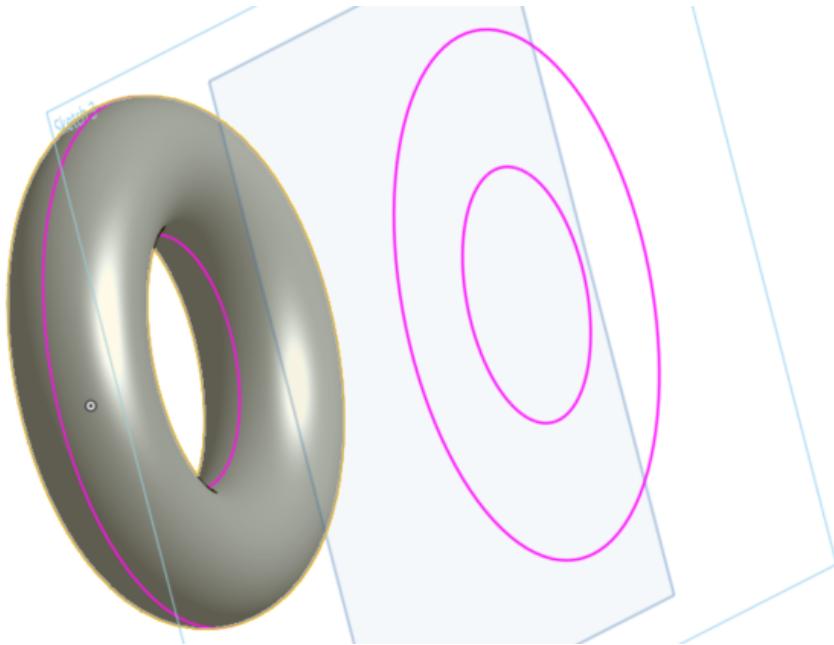
If no highlights are visible, you may be running into a limitation, see *Tips* below.

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.)

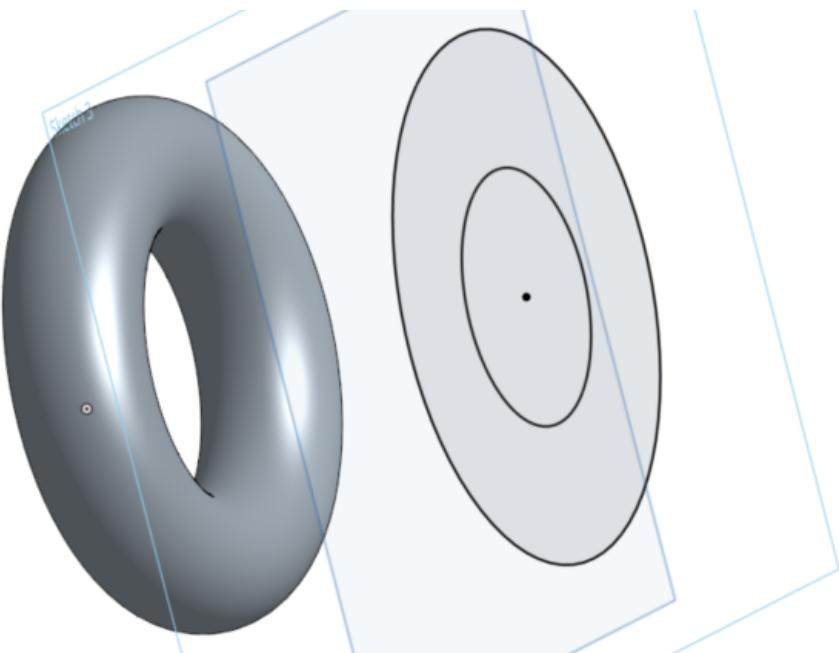


Note that when multiple silhouettes are available, you can click a face to select all silhouettes, or hover over an individual silhouette and click to select just the ones you wish to use:

This is what the highlight looks like during hover:



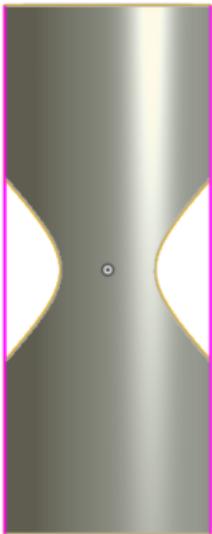
After selection and projection onto another plane:



## 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 can 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 enables a level of certainty 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.



# Intersection

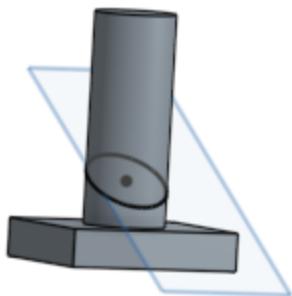


Project (or convert) the intersection of a surface or face and the active sketch plane onto the sketch plane.

## Steps

1. Create a sketch using an intersecting plane as the sketch plane.
2. Select Intersection from the Use drop down menu.
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

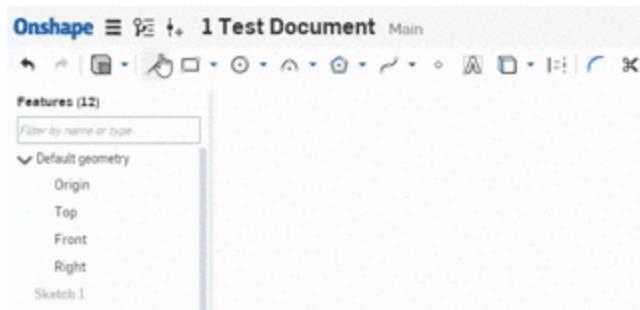
Shortcut: q (to toggle Construction state on and off)



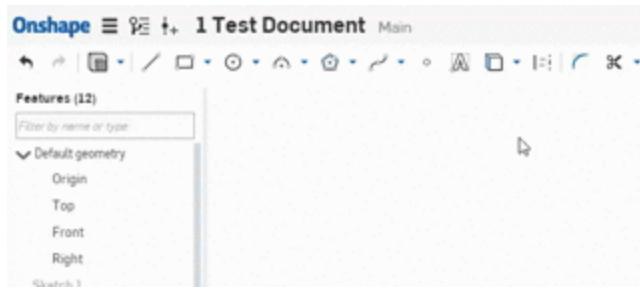
Sketch new construction geometry or convert existing geometry into construction geometry. Construction geometry are sketch entities used in creating other geometry but not used in creating features.

You can take two approaches to drawing construction geometry:

- Draw the sketch entities first, select the sketch entities to toggle, then select the Construction tool:



- Select the Construction tool, then a sketch tool and draw the sketch entities in Construction mode:



## Tips

- Select the Construction tool and then a sketch tool to create construction geometry.
- Select sketch entities and then the Construction tool to toggle construction mode on and off.

# Fillet (Sketch)

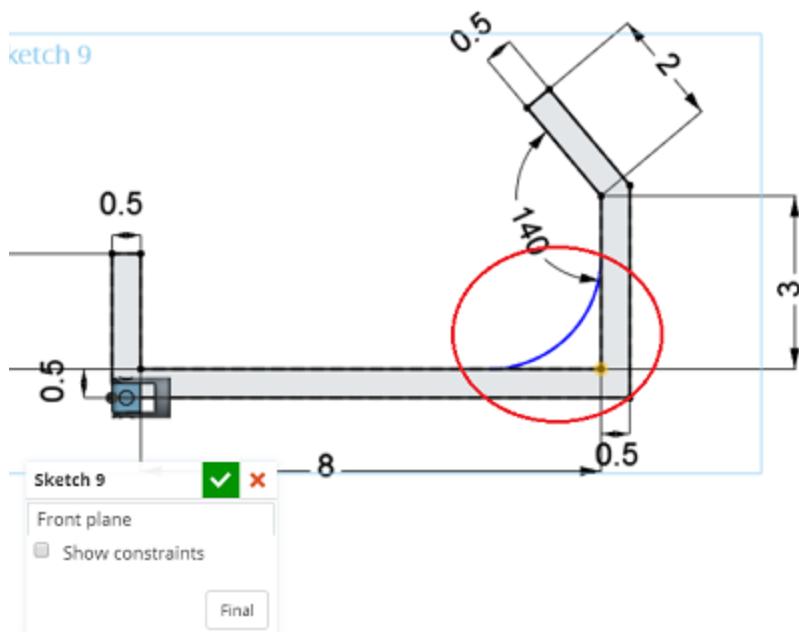
Shortcut: Shift-f



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

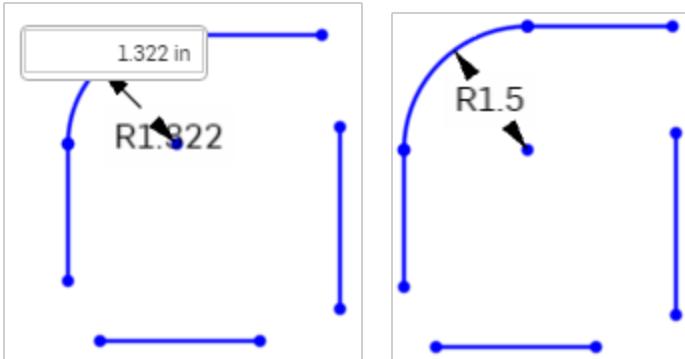
- To apply more than one fillet of the same size, make the first selection, enter the radius, then click the other points to fillet. The fillet will be applied to all selected points. You can change the one value to change all values.



## Examples

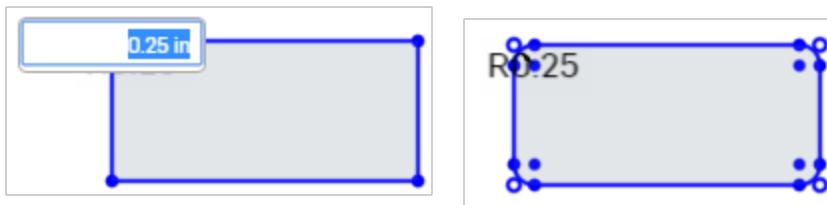
## Line, make selections

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.



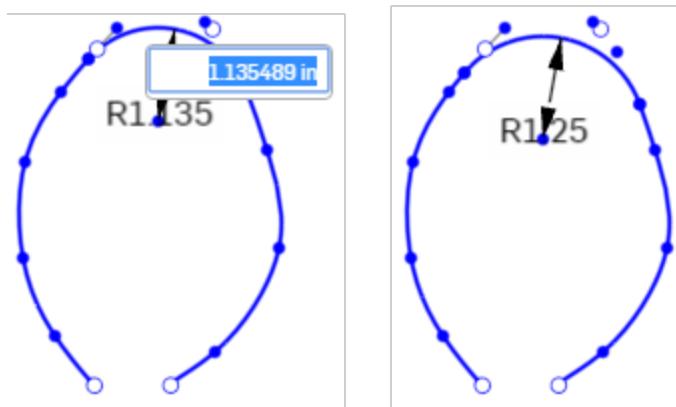
## Vertex, make selections

1. Click Fillet icon.
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 Fillet tool.
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

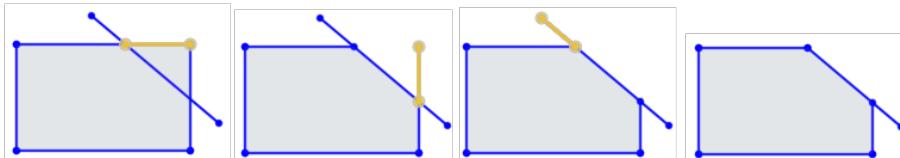
two lines, as well as a dimension (radius of the fillet). You may want to use this as a reference point for adding constraints, for example. You can also choose to simply ignore it. (See the "Vertex, make selections" example above to see the virtual sharp.)

 **Trim**

Shortcut: m



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. Click Trim tool, click entities to trim away.



# ---| Extend

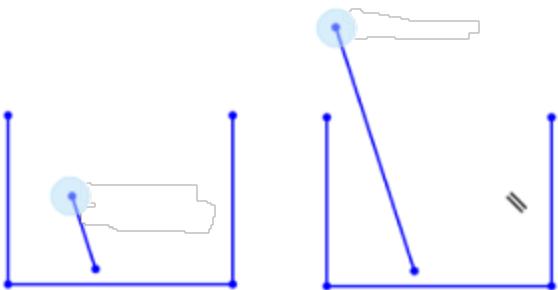
Shortcut: x



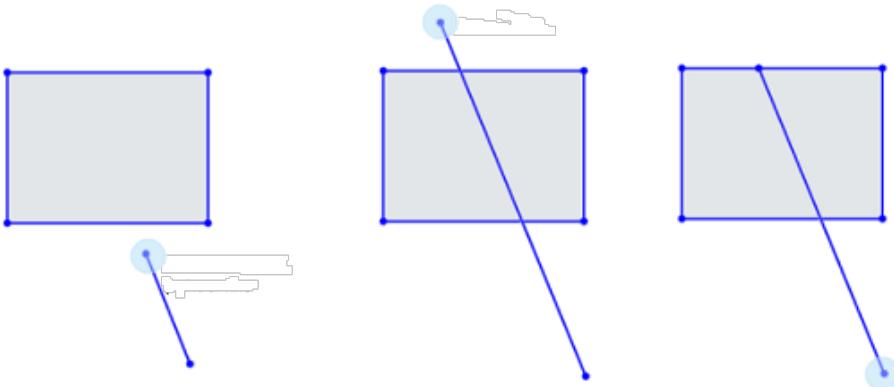
Extend a line to the first intersecting point or bounding geometry. If no intersection or bounding geometry is found, then the line ends at the release point.

#### No intersecting or boundary geometry:

1. Click the point to extend.
2. Click new location for the point.



#### Intersecting geometry:



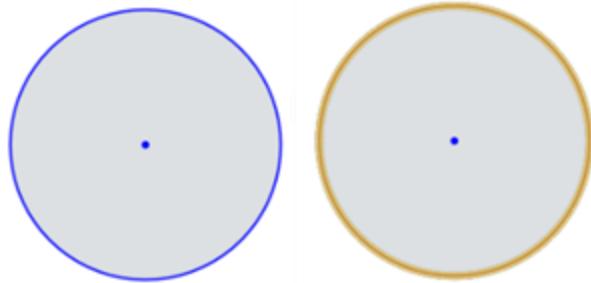


# Sketch Split

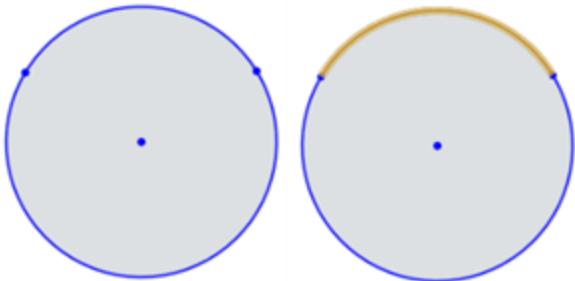


Split open or closed sketch curves into multiple segments. Open curves require one or more points to split with; closed curves require two or more points.

Before the split, there is one sketch curve to select:



After the split, there are two sketch curves to select:



## Tips

- Click on one or more points to split an open curve.
- Click two or more points to split a closed curve.

# Offset

Shortcut: o

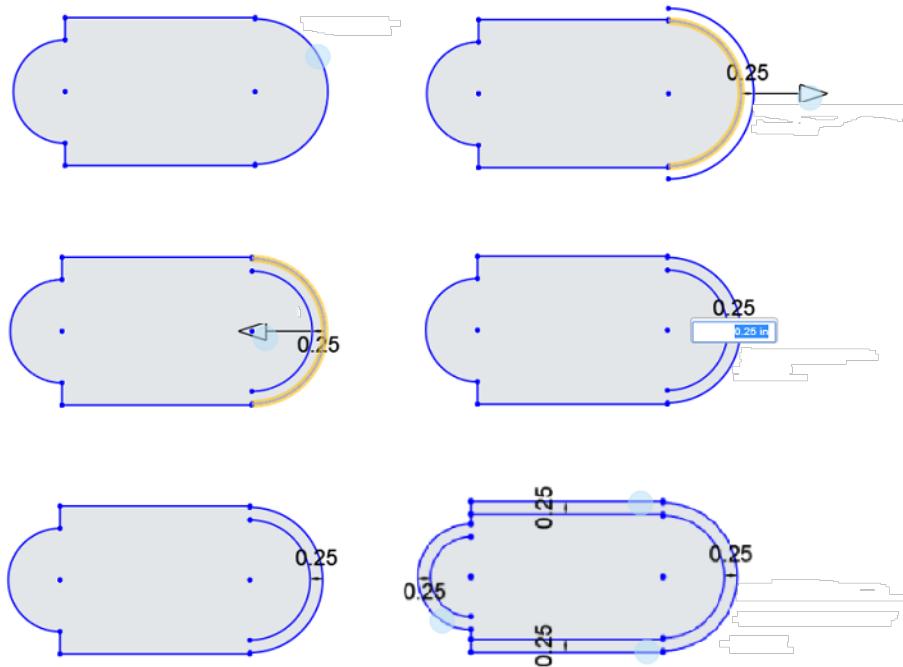


Offset the selected curve or loop at a specified distance and direction from the original.

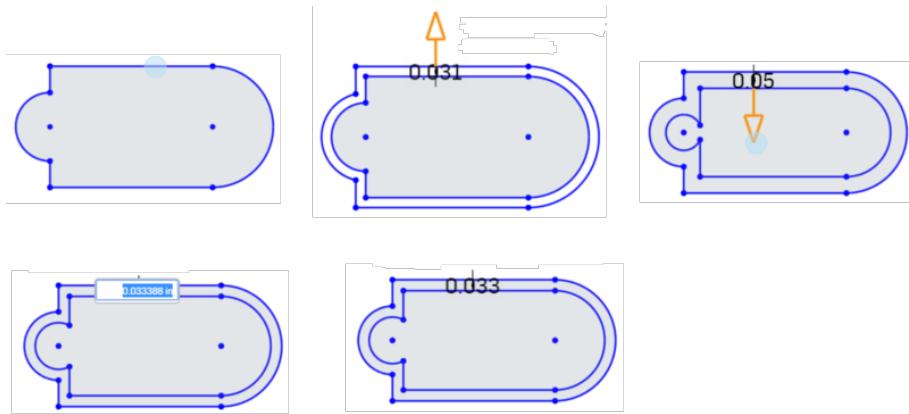
## Steps

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.  
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



## ⊖ Slot



Create a slot around selected sketch curves (including splines, lines, arcs but no closed profiles).

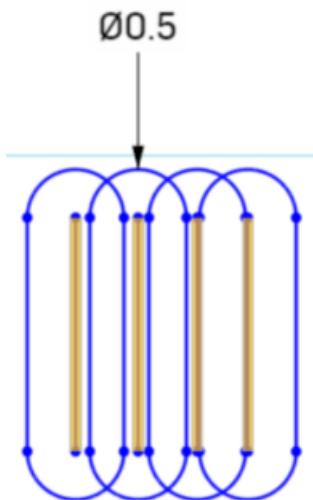
### Steps to creating Slots

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):



2. Click ⊖.

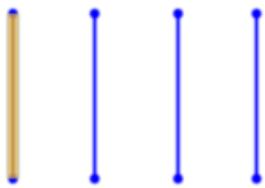


3. Double-click the dimension to edit it.

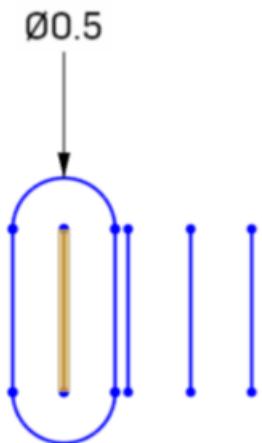
### Chain selection

Applying the Slot command to individual sketch curves in sequence, links the dimensions of the slots:

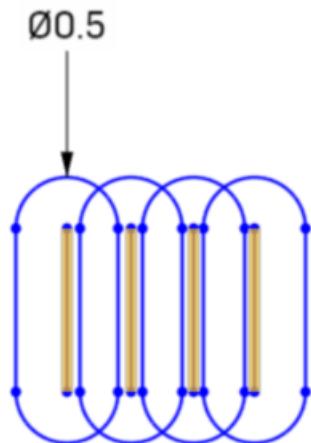
1. Select one sketch curve:



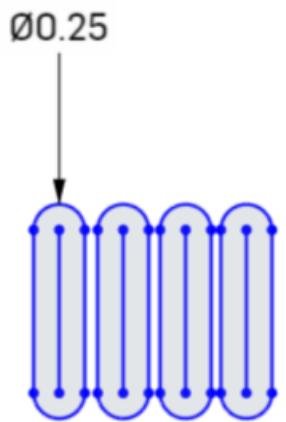
2. Click .



3. With the Slot command still selected, click more sketch curves to apply the same (unlinked) dimension:



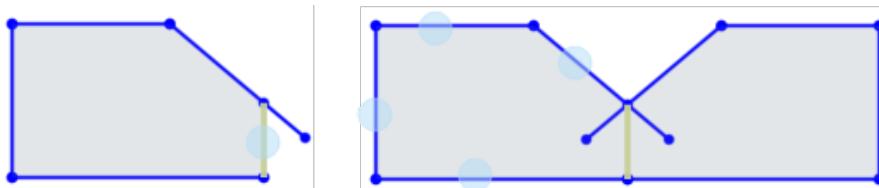
4. When you edit the dimension, all slots are changed:



# Mirror (Sketch)



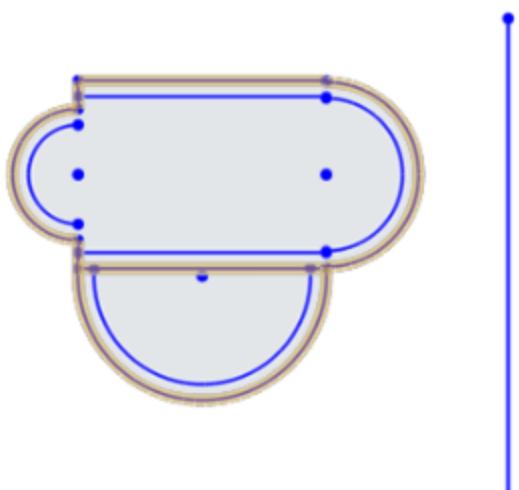
Create the reflection of one or more selected sketch entities about a specified line.



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.

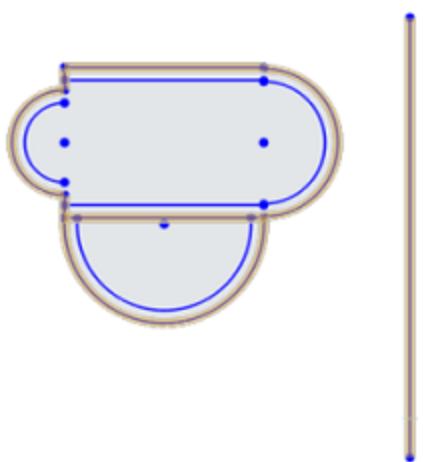
## Pre-selected entities

1. Select one or more sketch entities.

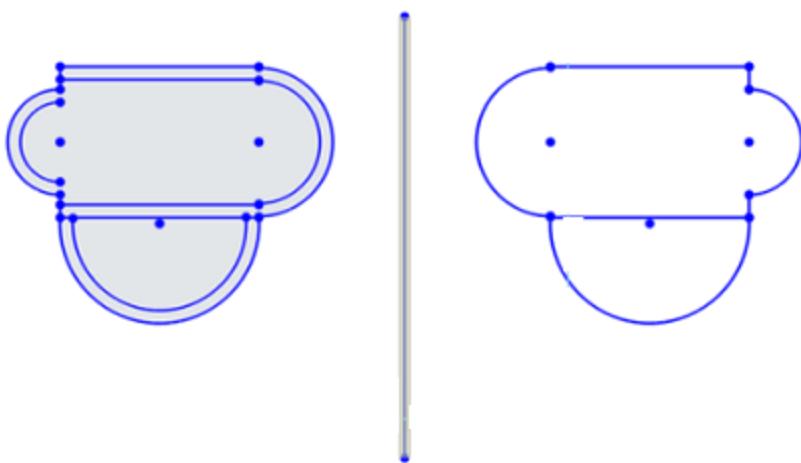


2. Click . (You are prompted to select the mirror line.)

3. Select the line across which to mirror.

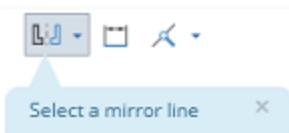


4. As soon as you click the mirror line, the sketch resolves:

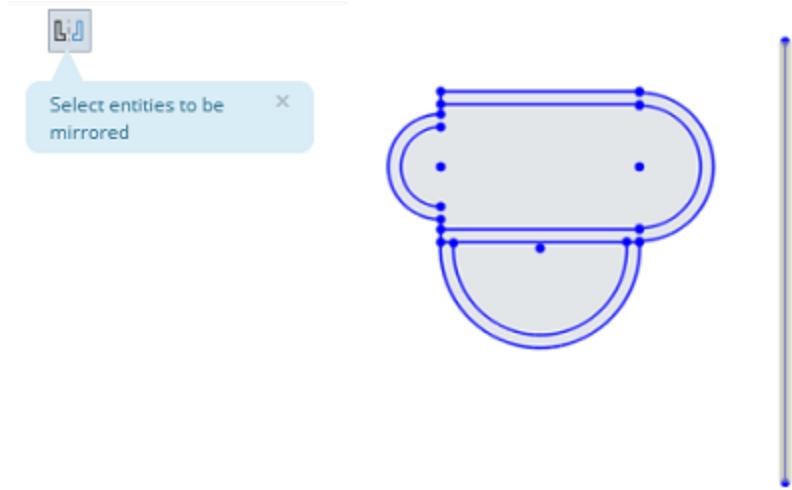


## No pre-selected entities

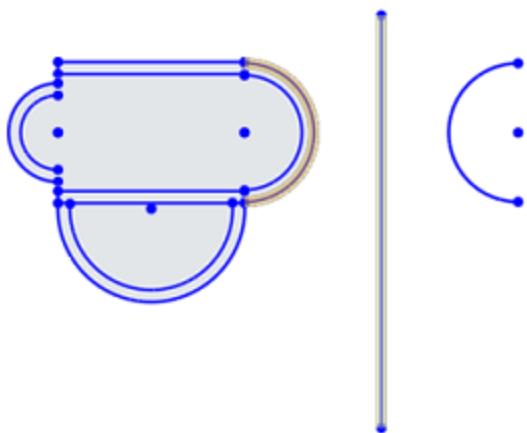
1. Click . (You are prompted to select a mirror line.)



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.



# □□ Linear Sketch Pattern

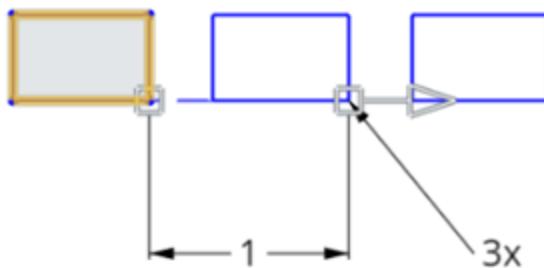


Create multiple instances of sketch entities uniformly in one or two directions. Patterns can be open or closed, as described below.

Select the sketch to pattern and then click the Linear pattern tool icon:



The initial pattern created is:



Double-click to enter the number of entities.

1x



Click and drag the arrow head to change the distance between entities; drag the arrow base to move the pattern at an angle.



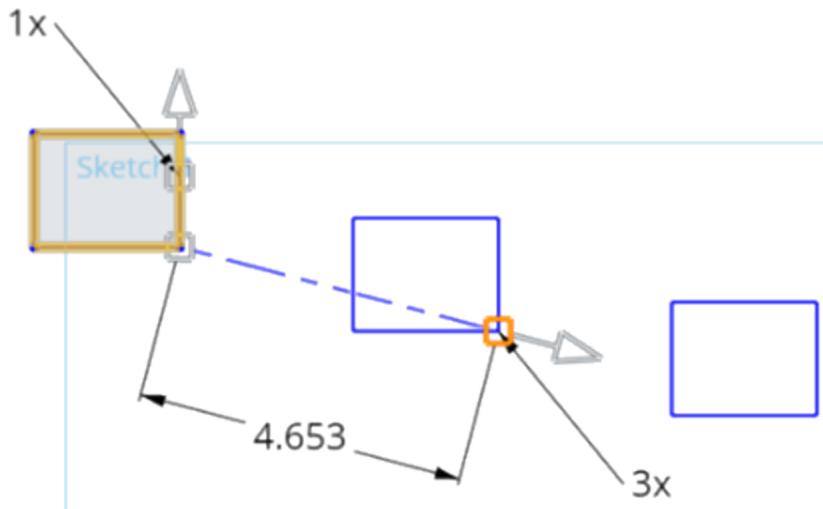
Double-click to specify the distance between entities.

## Tips

- You can 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 can drag the arrow manipulator's base to position the pattern at an angle (the base is shown highlighted below):



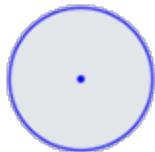
For more information, see "Circular Sketch Pattern" on the next page.

# Circular Sketch Pattern

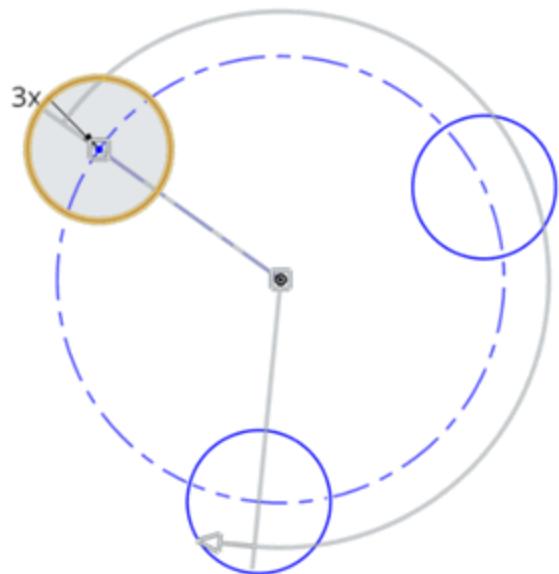


Create multiple instances of sketch entities uniformly about an axis. Circular patterns can be open or closed, as described below.

Entity to pattern radially:



Initial pattern created:



3x

Double-click to enter the number of entities to pattern.



Click and drag to specify the angle of an open pattern.

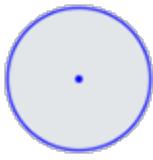
177.224°

Double-click to enter an angle value of an open pattern (visible after clicking the arrow icon).

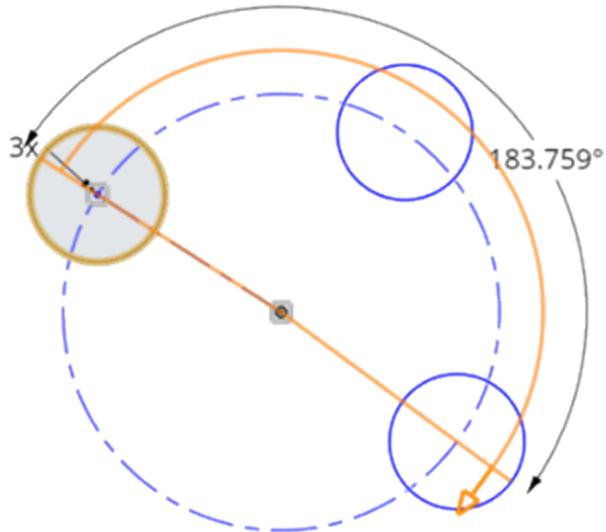


Double-click to specify the distance between entities.

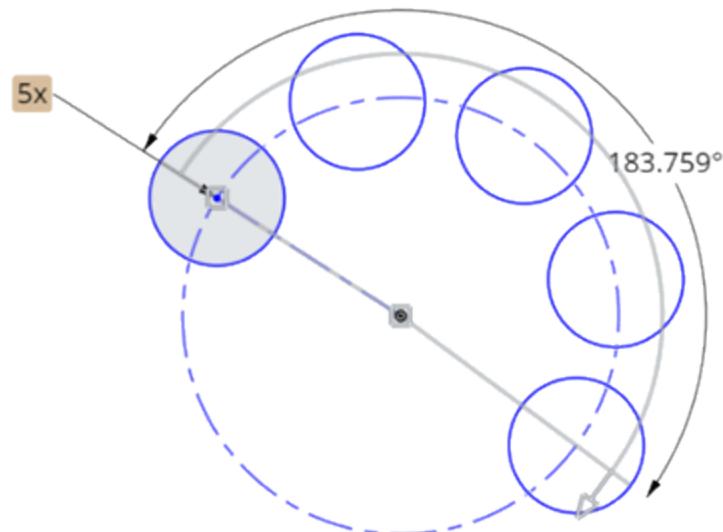
Entity to pattern radially (open pattern):



After initial pattern created, click and drag the arrow head to reduce the angle dimension and open the pattern:



Once the pattern is accepted, when you change the instance count, it keeps the pattern open:



## Tips

- Circular patterns default to closed patterns ( $360^\circ$ ). However, you can 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 can 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 140.

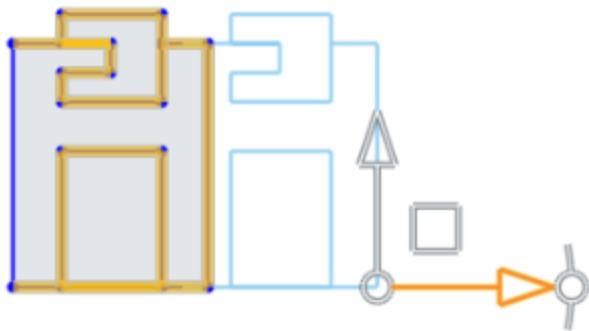
# Transform Sketch



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.



For more information, see "Transforming sketches" on page 100

## 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 supported for images, text, DWG, and DXF .



# Insert DXF and DWG as Sketch Entities



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 **Sketch** 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 or use **Browse documents** to locate a file in another document that you have created or that has been shared with you).

Selecting the file to insert automatically closes the dialog.

## Tips

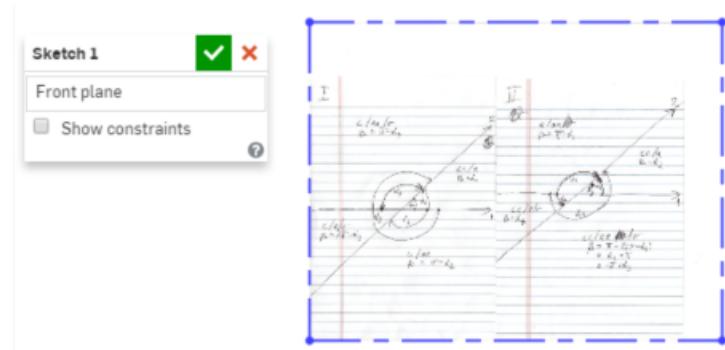
- You can insert DXF/DWG files that have already been imported into your document or another document that you have created or has been shared with you. These show up as tabs and also in the Insert DXF dialog.
- Make sure to select the units in the dialog first; selecting the file automatically closes 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.



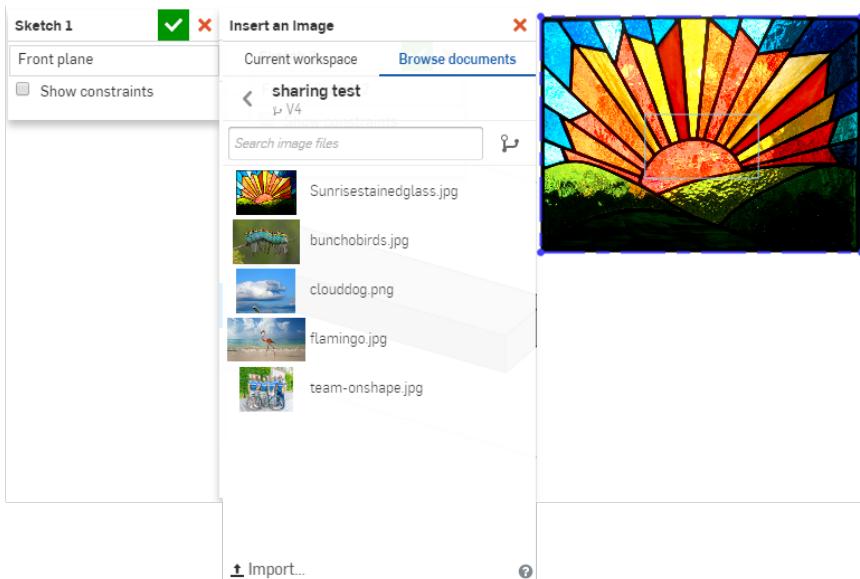
## Steps

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 can also 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 can link only 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.)

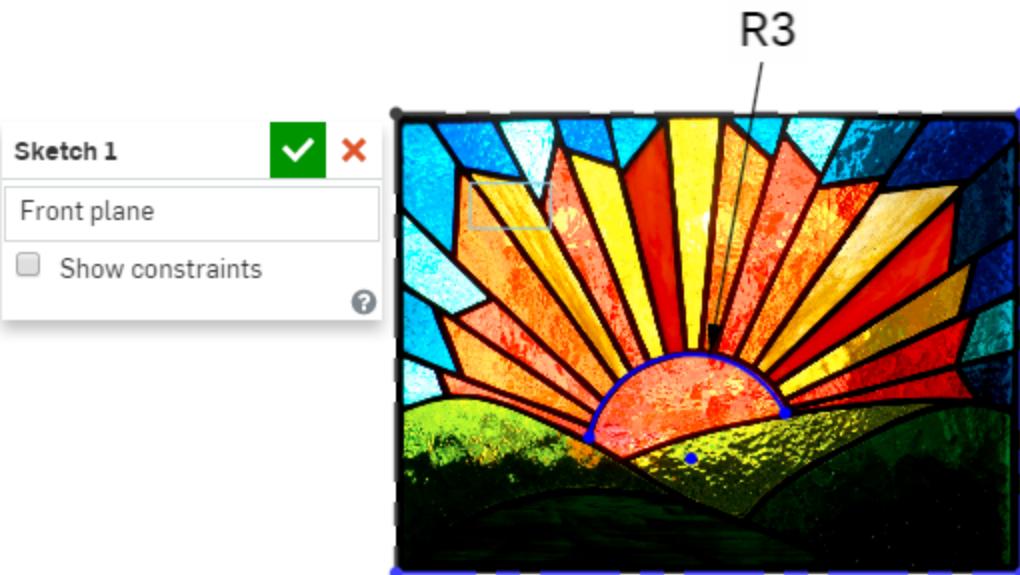
Note that a horizontal constraint is applied to the lower edge of the box by default.



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.  
 6. 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.

# Dimension

Shortcut: d



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 can 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.

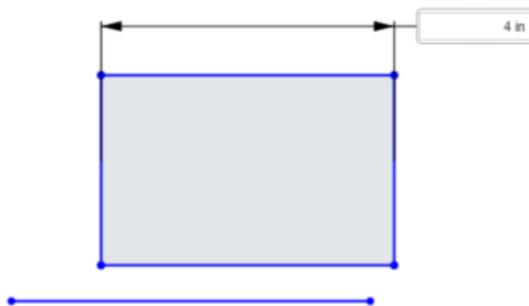
## Steps



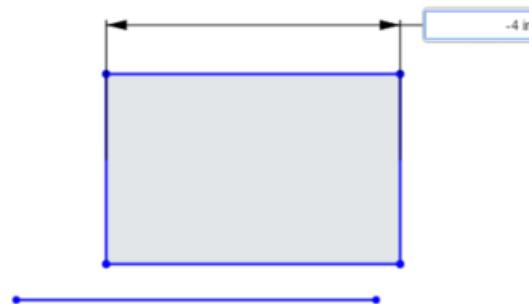
1. Click the Dimension tool icon or press the **d** key.
2. Select the entity (or entities between which) to dimension and the location of the dimension.  
The dialog opens on the placement of the dimension.
3. Enter a value and press **Enter** to accept the value, or use **Shift-Enter** to accept the value and keep the dialog open.

Note that you can enter negative values for dimensions (length, linear distance, and angles), thereby flipping the direction of the entity.

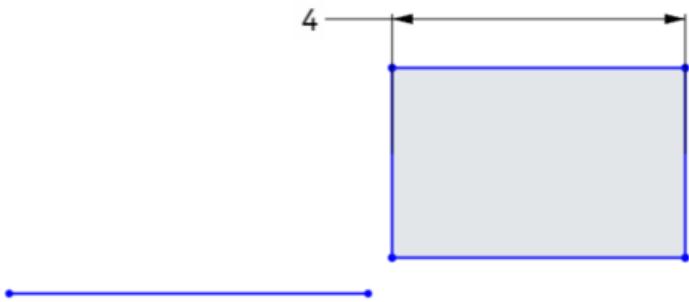
The image below illustrates a positive dimension, notice the position of the rectangle in relation to the horizontal line:



The following image illustrates a negative dimension value in the dialog:



When the dialog is accepted, the rectangle flips direction based on the negative dimension value:

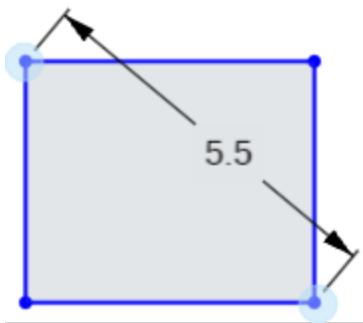


You can use [expressions and trigonometric functions](#) in numeric fields in Part Studios.

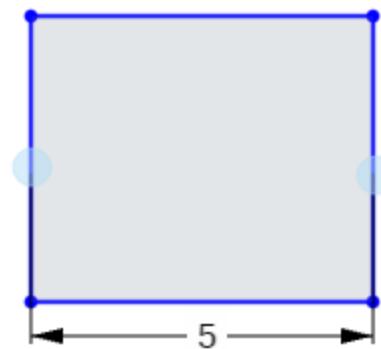
Delete a dimension by selecting it and pressing the Delete key, or select it and select Delete from the context menu.

## Diagonal distance

1. Click corner points diagonal to each other.
2. Drag to visualize the dimension.
3. Click again to access the numeric value field.
4. Type value and press Enter.

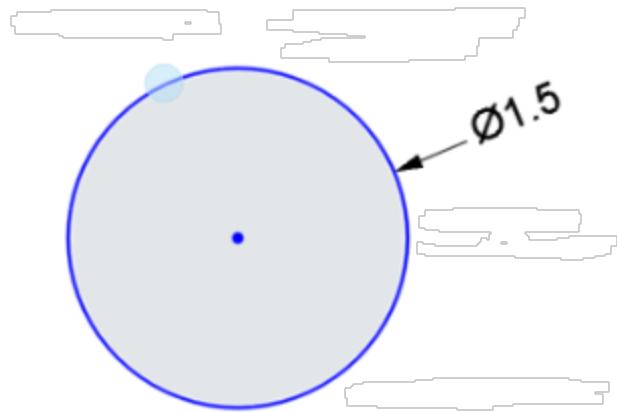


## Length or height



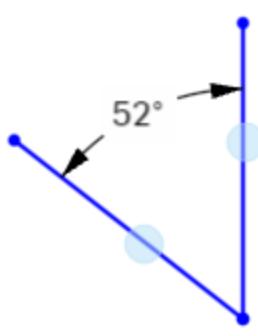
## Diameter

1. Click the edge of the circle.
2. Drag cursor into or away from the circle.
3. Click to activate numeric value field.
4. Type value and press Enter.

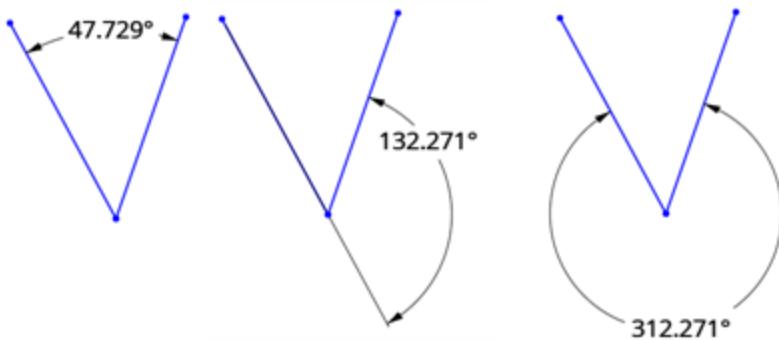


## Angle

1. Click each line.
2. Move cursor into angle.
3. Click to activate numeric value field.
4. Type value and press Enter.

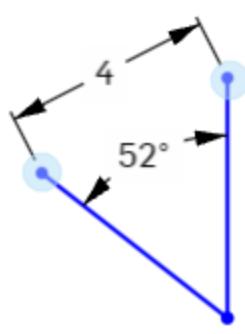


You can also drag the label to the quadrant for which you want to define the angle:



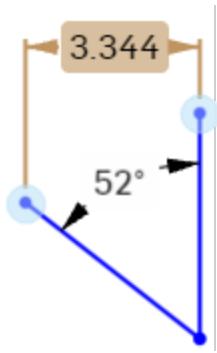
## Direct distance

1. Click each endpoint of the lines.
2. Move cursor away at an angle to get shortest distance between the points.



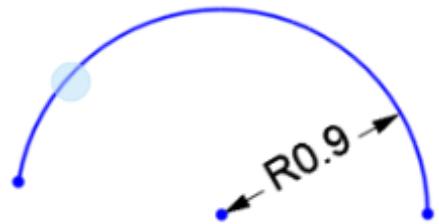
## Linear distance

1. Click each endpoint of the lines.
2. Move cursor straight up for linear distance.



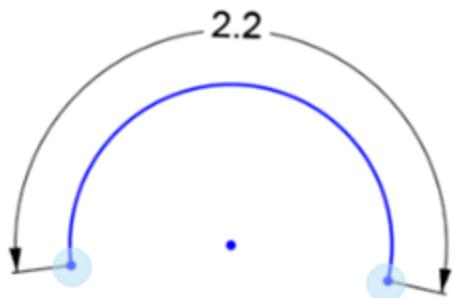
## Radius

1. Click the edge of the arc.
2. Move cursor into or away from arc.
3. Click to enter numeric value field.
4. Type value and press Enter.

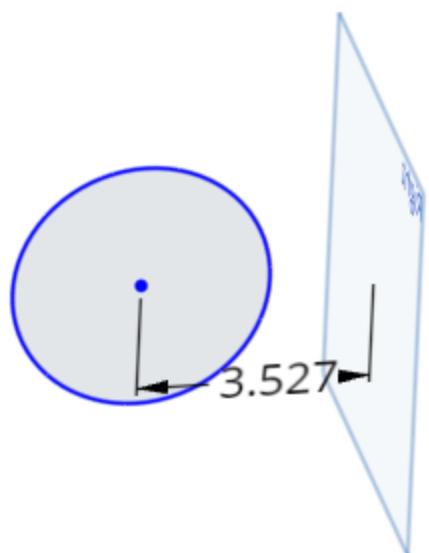


## Arc length

1. Click each arc endpoint.
2. Move cursor to arc line.
3. Click to activate numeric value field.
4. Type value and press Enter.



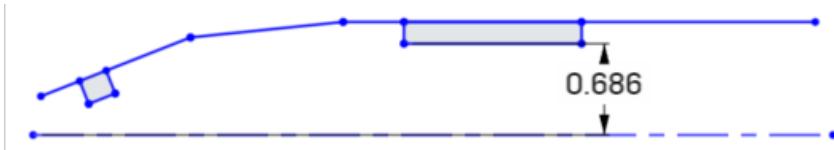
## Between sketch geometry and plane



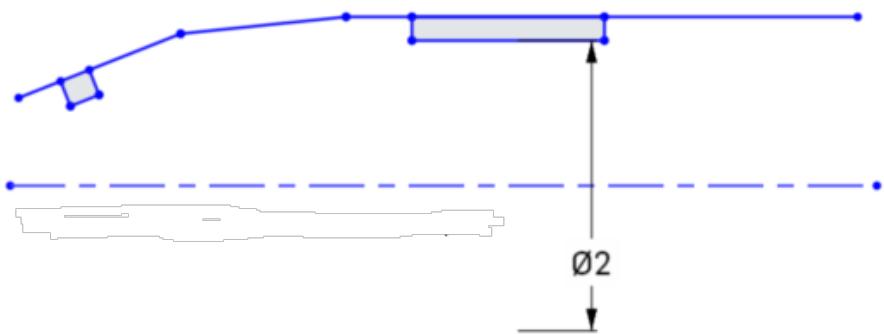
## Centerline dimensions

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:



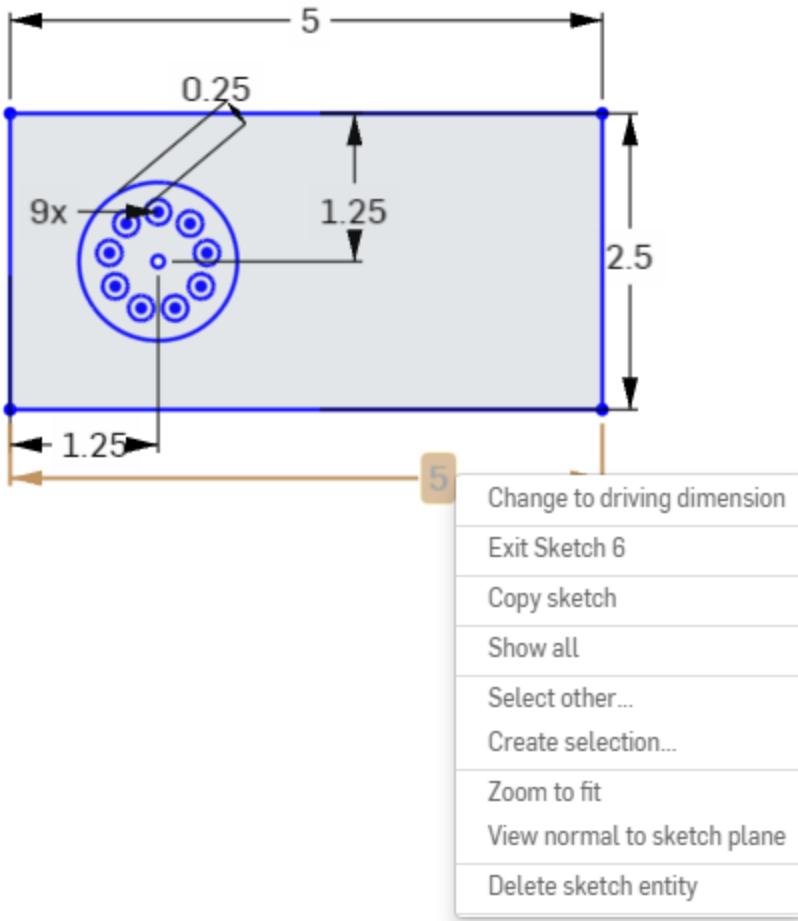
2. Move the mouse to the opposite side of the construction line to toggle the state to a centerline dimension.
3. Enter the value and press Enter.



## 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 can 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.

# Coincident



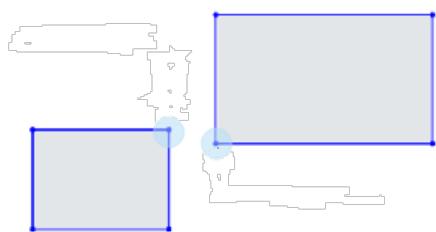
Shortcut: i



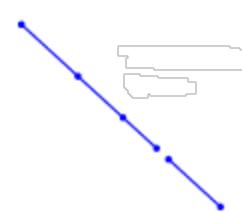
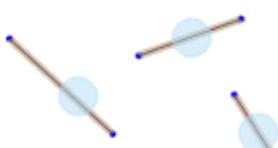
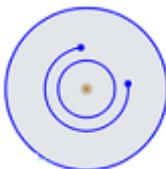
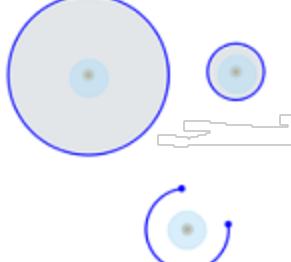
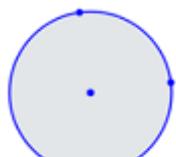
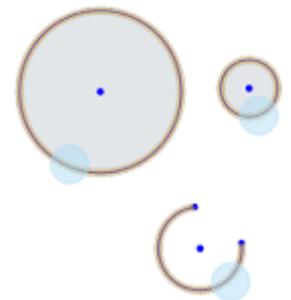
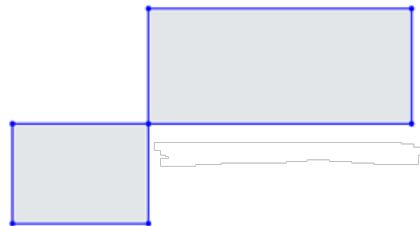
Makes two or more entities coincident, including a sketch entity and a plane.

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.

Select the entities:



Click the Coincident constraint tool:



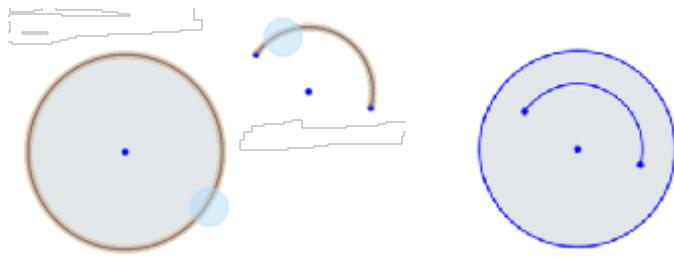
The infinite, underlying geometry of the two entities is made coincident.

# ○ Concentric



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

1. Select the circle.
2. Select the arc.
3. Select the Constraint tool icon.



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.

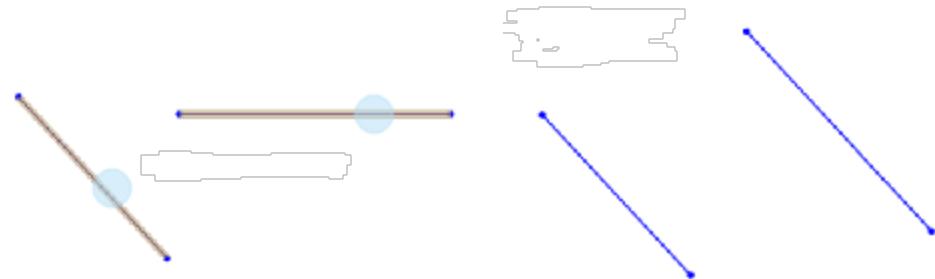
# Parallel

Shortcut: b



Make two or more lines parallel.

1. Select each line.
2. Click Parallel constraint tool icon.



## 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.

# Tangent

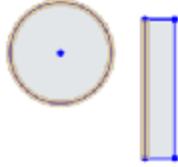
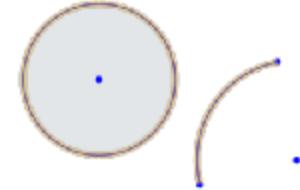
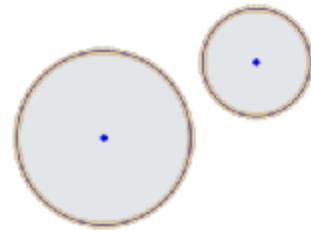
Shortcut: t



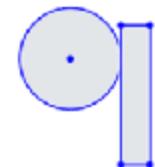
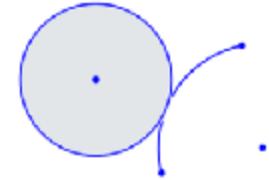
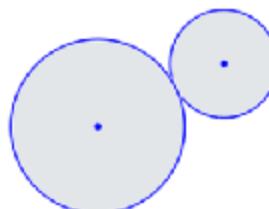
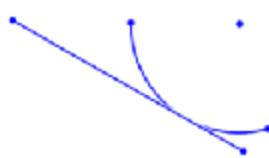
Form a tangent relation between two curves, or between a curve and a plane.

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.

Select two or more curves:



Click the Tangent constraint tool:





## Steps

1. In the graphics area, select two or more curves.
2. Click or press the T key.

# Horizontal

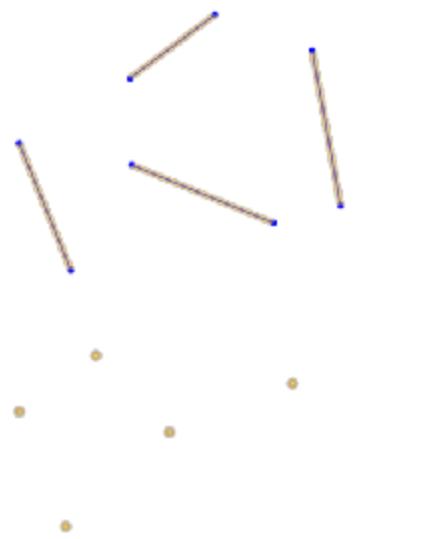
Shortcut: h



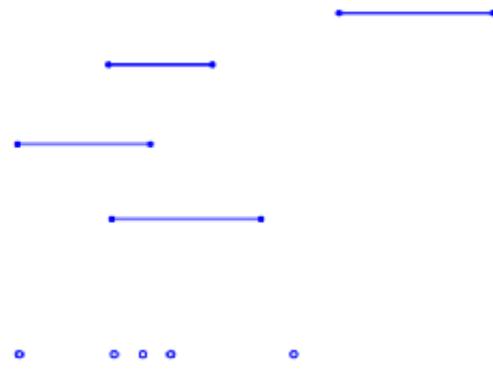
Make one or more lines, or sets of points, align horizontally.

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.

Select one or more lines or points:



Click the Horizontal constraint tool:



# Verticals

Shortcut: v



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.

Select two of lines or points:



Click the Vertical constraint tool:



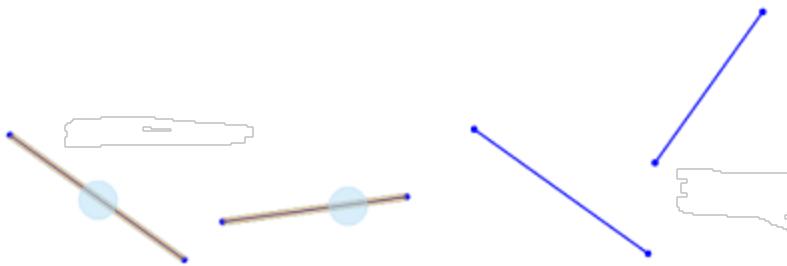


# Perpendicular



Form a right angle between two lines.

1. Select two lines.
2. Click Perpendicular constraint tool icon.



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.

# = Equal

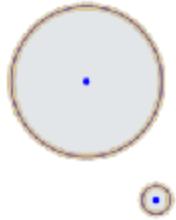
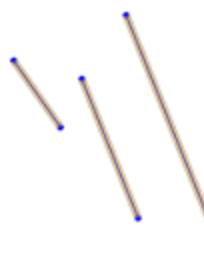
Shortcut: e



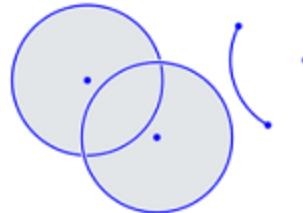
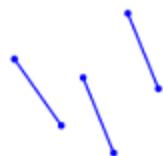
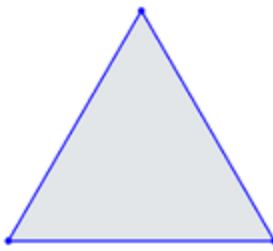
Make two or more sketch curves of the same type equal in size.

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.

Select two or more sketch curves:



Click the Equal constraint tool:



If one sketch curve is dimensioned, that size is used.

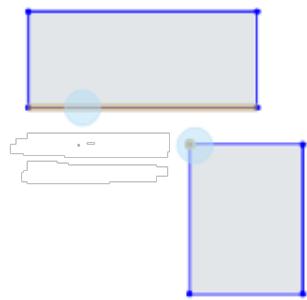
## —○— Midpoint



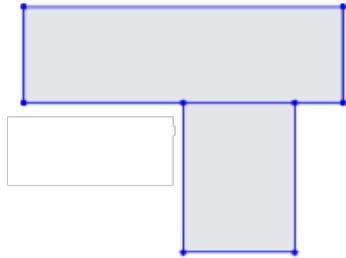
Constrain a point to the midpoint of a line or arc.

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.

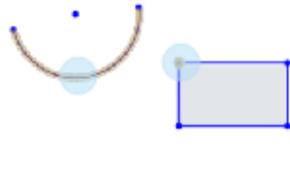
Select a line and a point:



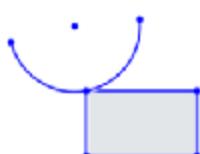
Click the Midpoint constraint tool:



Select an arc and a point:



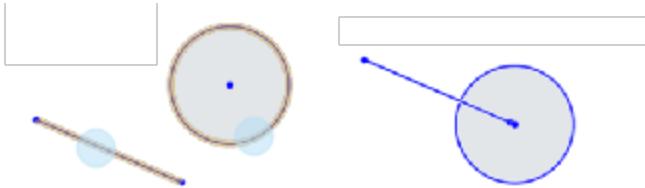
Click the Midpoint constraint tool:



The point will be constrained to the midpoint of the line or arc.

 **Normal**

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

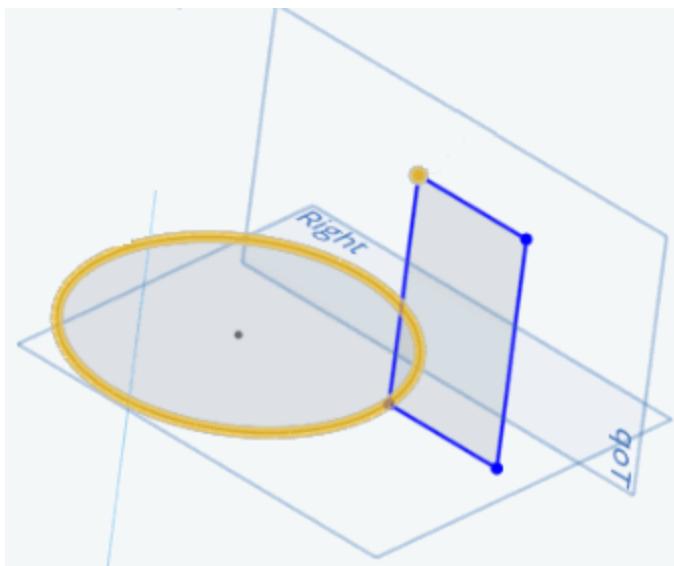
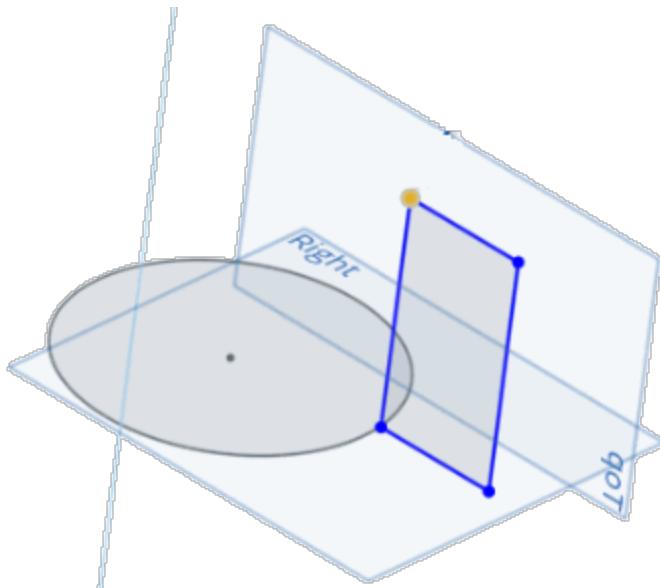


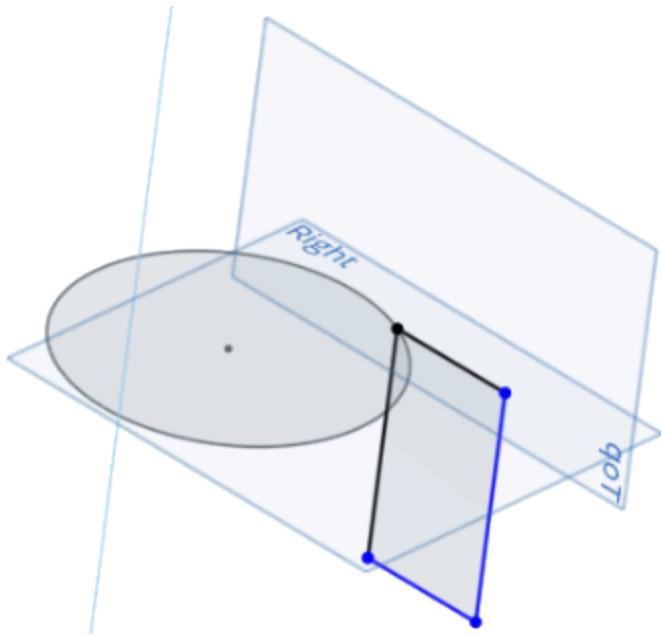
Constraints can be toggled on while you make selections. Toggle Normal on and each pair of entities you select are constrained to each other. Click Normal again to toggle off, or select another tool to toggle off automatically.

 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.

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.

## Steps

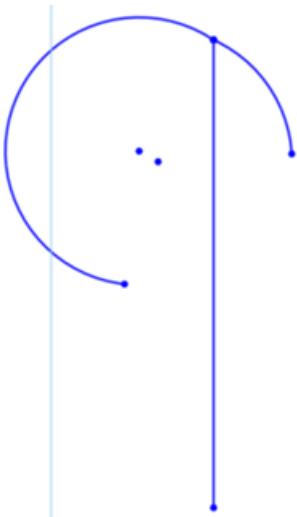
1. Select a sketch point or curve and an edge outside of the sketch (intersecting with the sketch plane).
2. Click **Pierce** 

# $\Sigma \mid \Sigma$ Symmetric



Constrain two geometries (of the same type) to be symmetric relative to a line:

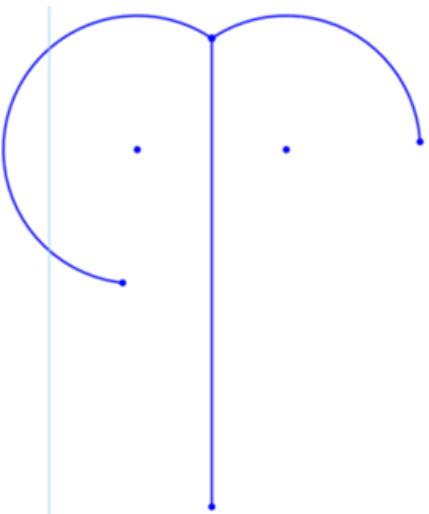
Sketch:



Selections:



Result:



## Steps

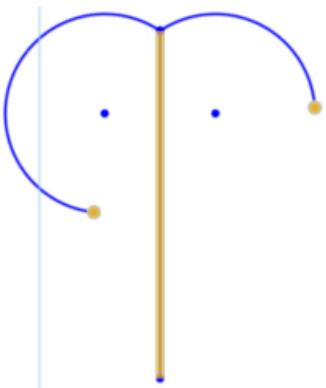
1. Pre-select a line, or linear edge.
2. Select two other geometries (of similar type to each other).
3. Click .

## Tips

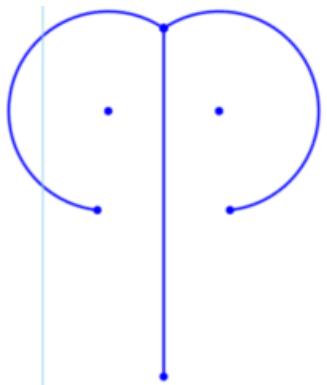
Constraints can be toggled on while you make selections. Toggle Symmetric on and each pair of entities you select are constrained to each other. Click Symmetric again to toggle off, or select another tool to toggle off automatically.

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:

Before Symmetric:



After Symmetric:



# Fix



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

Constraints can be toggled on while you make selections. Toggle Fix on and each pair of entities you select are constrained to each other. Click Fix again to toggle off, or select another tool to toggle off automatically.

## Steps

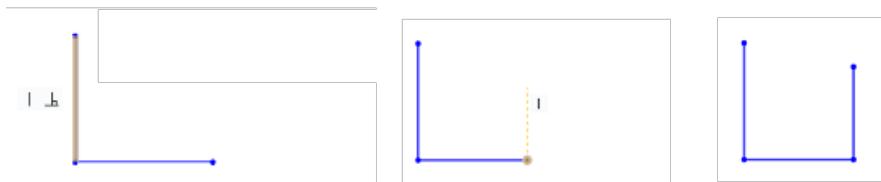
1. Select a sketch entity.
2. Click .

# Automatic Inferencing

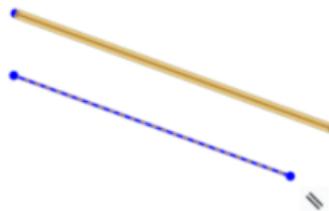
The Onshape sketch editor can assign constraints to certain entities automatically. For example, create a line and hover one of the endpoints above the origin and a dotted line appears indicating a vertical inference between that endpoint and the origin.

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.

1. Create two lines with a perpendicular constraint between them.
2. Move cursor near line until inferencing 'wakes up'.
3. Draw a line.



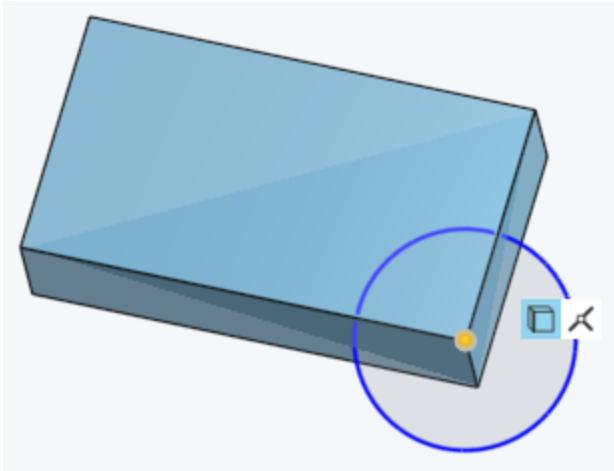
When sketching, Onshape indicates relationships with other sketch entities. In the illustration below, the bottom (blue) line is the one being drawn. When it is parallel to the other line it turns to a dotted line and the other is highlighted in orange to show that there is a relation present. (The parallel 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. 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. Upon hover, the referenced constraint's background is a darker blue:

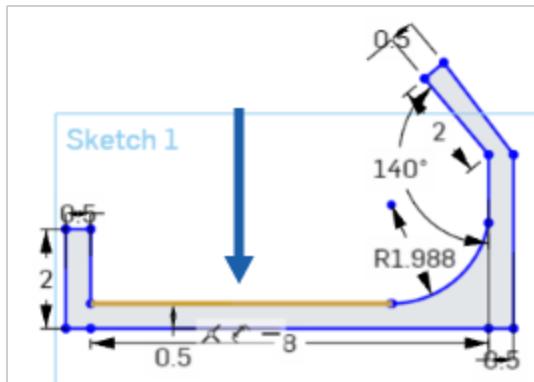


The Use constraint shown above (with the blue background) constrains a vertex in the rectangle's sketch with the center point of the circle in the circle's sketch.

## Video example

With a sketch open, hover 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.

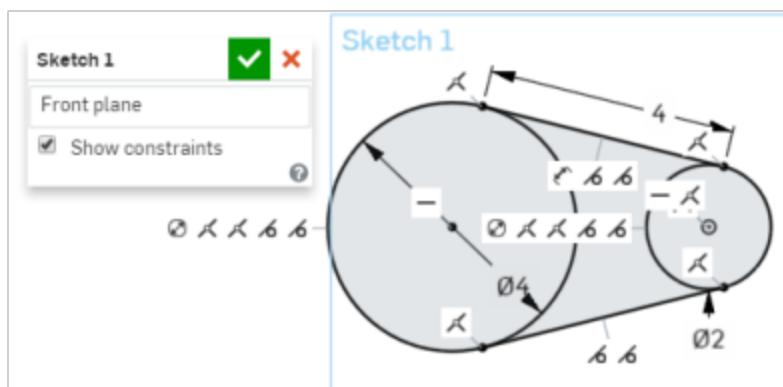
Entities are highlighted in orange upon hover, with the exception of referenced constraints which have a blue background and a darker blue background upon hover.



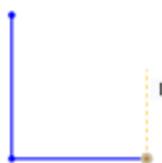
## Tips

You can 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.

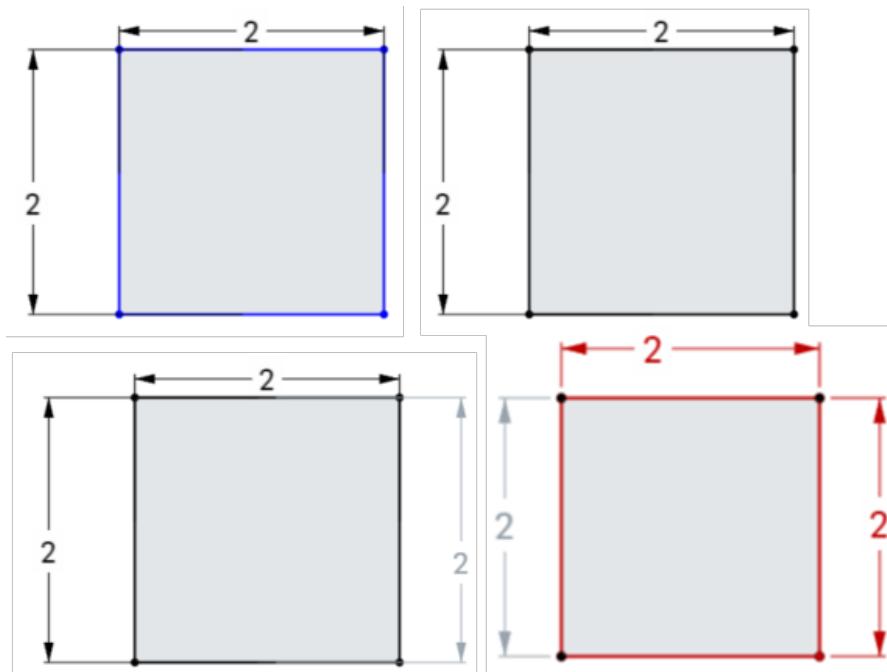


When sketching, constraint indicators appear next to the mouse cursor as the curves snap to inferences.



# Troubleshooting Sketch Geometry

- 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, red on white indicates a problem.
- Adding more dimensions or constraints will further constrain the sketch. Dragging entities can help you understand what constraints or dimensions you may want to add.



See the video titled Sketching Basics and the video titled Dimensions & Constraints for more details.

# Feature Tools

Feature tools create, modify, or manipulate 3-dimensional geometry to create new parts, modify existing ones, or generate construction tools for late use.

## The Feature toolbar

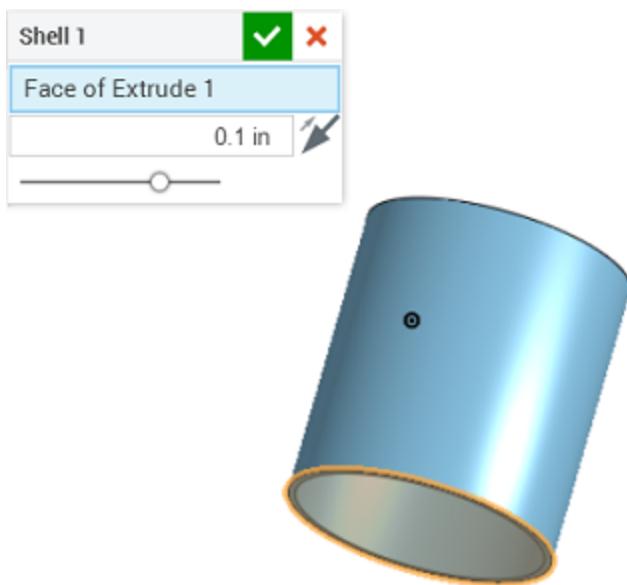


### Get started

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.

### Example

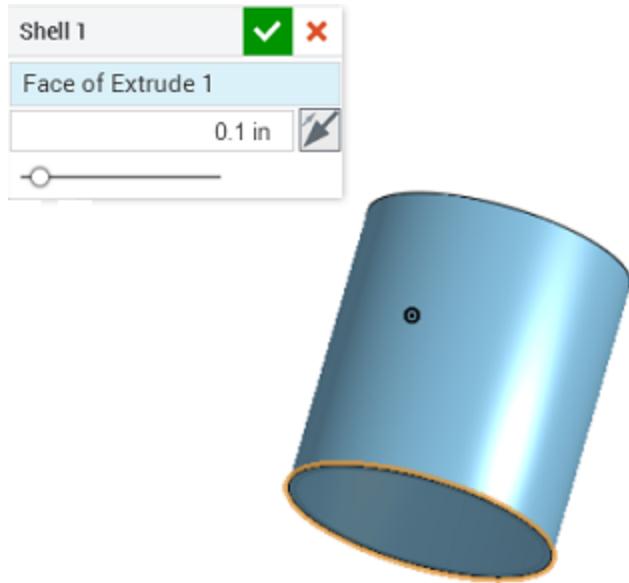
1. With a solid part in the graphics area, click the **Shell** tool .
2. Shell requires that you select one or more part faces to remove in order to hollow out the part:



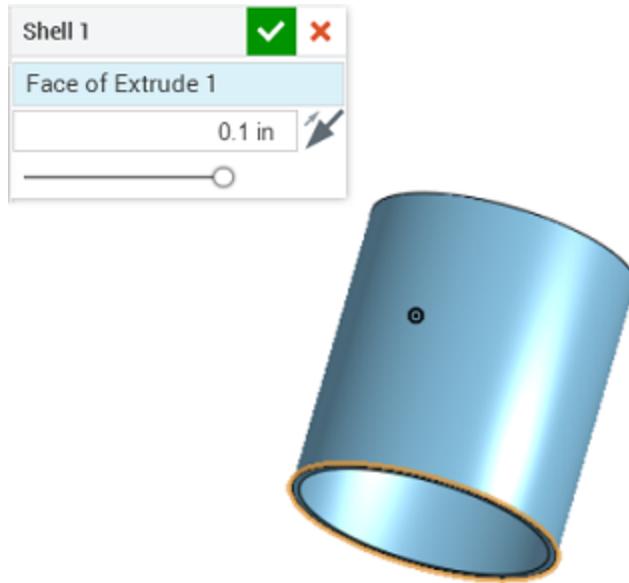
As soon as you click on the face to remove, it is removed.

3. Specify the wall thickness (the system supplies a default). You can type the units of your choice as well: "in" for inches or "mm" for millimeters.

4. Use the directional arrows 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:

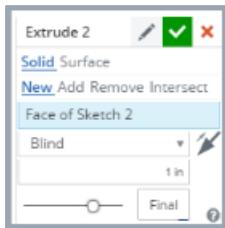


7. Click to accept or click to close without committing any changes.

## 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 can 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.



The arrows in the image above indicate (from left to right) the Preview Slider, the Final button, and the Context-sensitive help button.

- 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.



# Extrude

Shortcut: Shift-e

In the Feature toolbar:



In the **Sketch** toolbar:

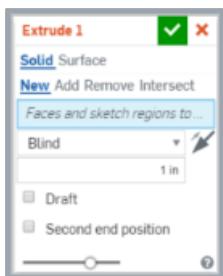


Extrude adds depth to a selected region or planar face along a straight path. Create a new part or modify an existing one by adding or removing material, or intersecting bodies in its path. You can also use Extrude to create solid bodies or surfaces.

## Steps

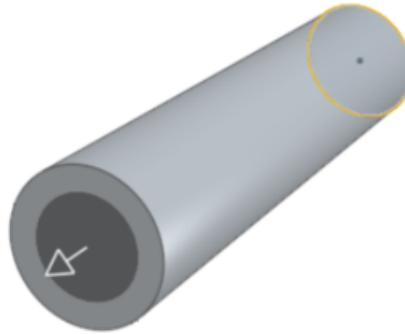
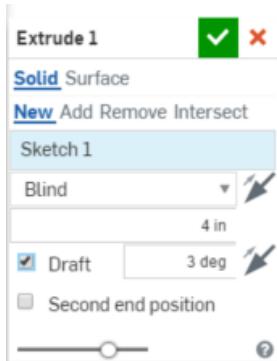
From the **Sketch** or **Feature** toolbar:

1. Click .



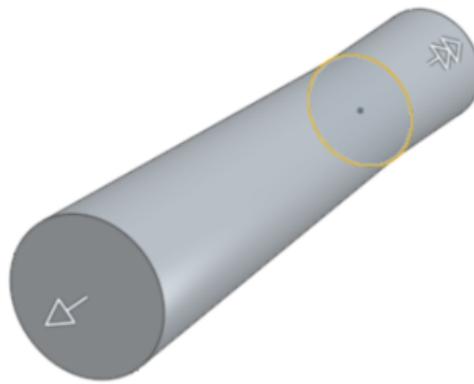
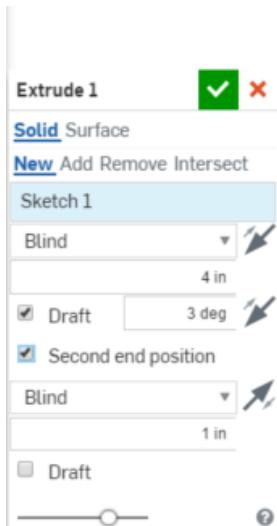
2. Select Creation type:
  - Solid** - Create parts or modify existing parts.
  - Surface** - Create a surface along a sketch curve.
3. Select a Result body operation type:
  - New** - Create new material that results in a new part.
  - Add** - Create material added to the existing material.
  - Remove** - 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.
  - Through all** - Through all selected parts.

6. Specify whether to switch to the opposite direction, optional.
7. Check 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:



The model is shaded to show the original solid (dark area) and that of the drafted solid (lighter area).

8. Optionally, check to extrude in a second end position about the sketch plane:



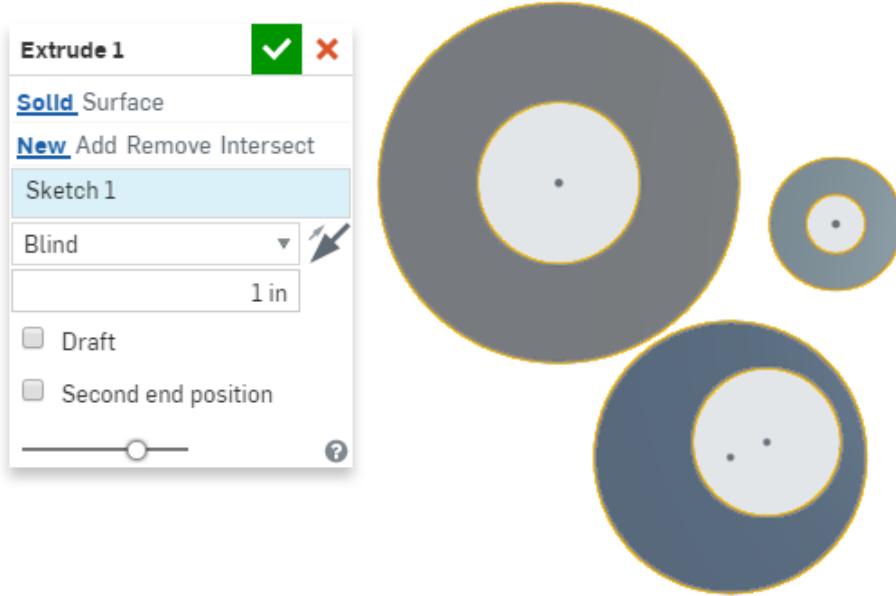
Extruding in a second end position offers all the same end conditions and a separate depth field, as well as the option to create a draft (with the sketch plane as the neutral plane).

9. Enter a depth (for each end position, if necessary).
10. Click .

Remember, you can use the Preview slider to visualize the result before accepting the feature (with the check).

## Extrude nested sketches

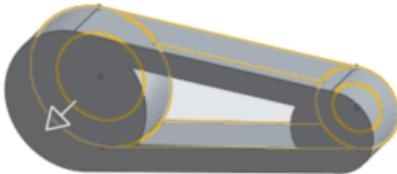
When extruding with a sketch open, Onshape automatically selects all of the regions in the sketch, except when nested sketch entities exist. In this case, Onshape selects only the region between the nested sketch entities, as shown below:



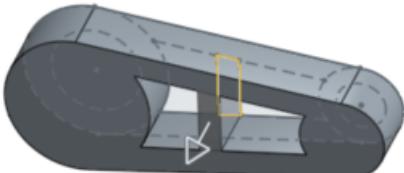
## Extrude New/Add (new material)

Create new material or material that results in a new part.

**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:

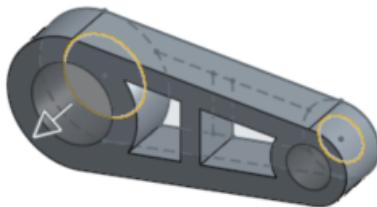
- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching 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 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.

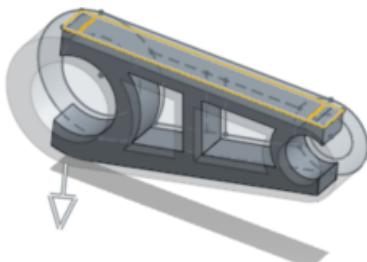
**Remove** - Take material away



## Extrude Intersect

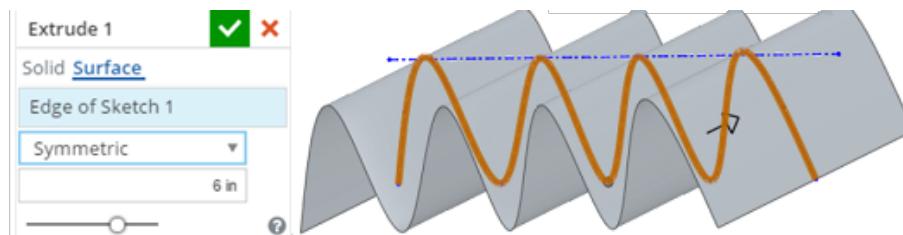
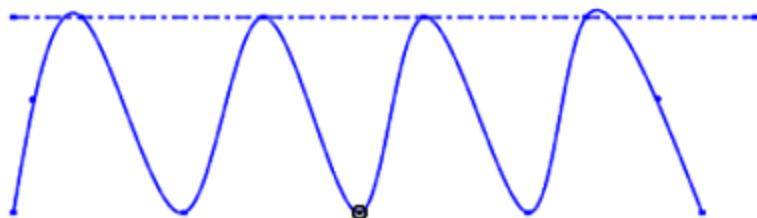
Leave material only where intersections exist.

**Intersect** - Leave material only where intersections exist



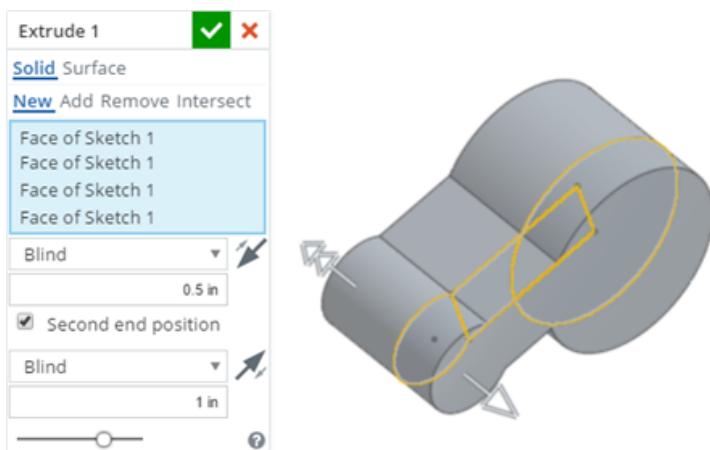
## Extrude Surface

Create a surface along a sketch curve. For example:



## Extrude Second Direction

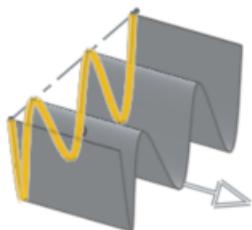
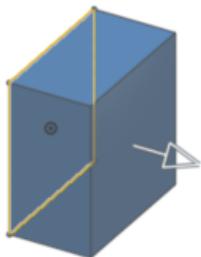
Extrude in two directions differently about the sketch plane. For example:



## End conditions

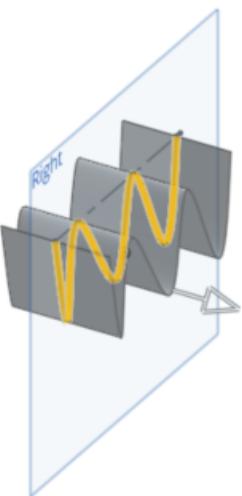
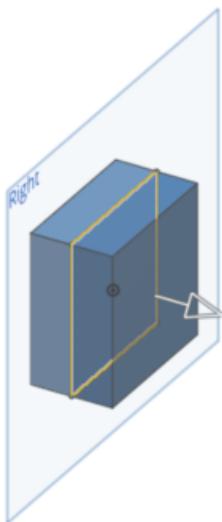
### Blind

To a specified distance in one direction



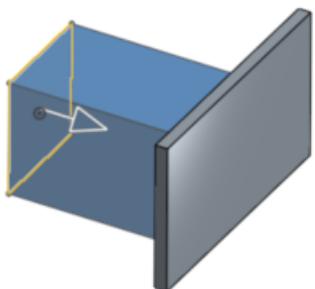
### Symmetric

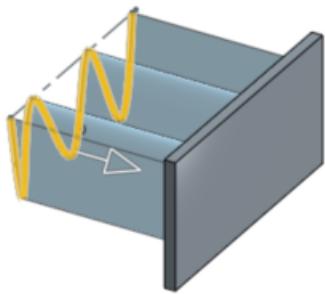
To a specified distance equally in both directions around the sketch plane



## Up to next

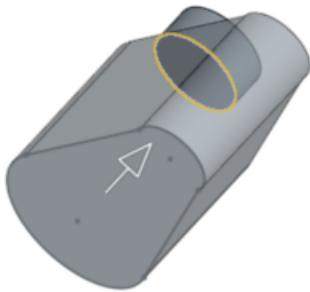
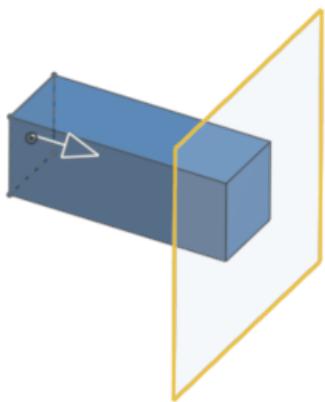
Up to the next geometry encountered in the given direction; if there is no existing geometry encountered, the extrude cut fails; note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed

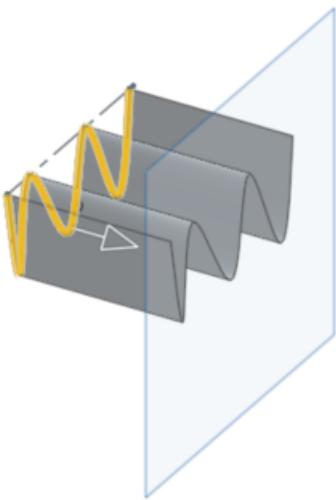




## Up to face

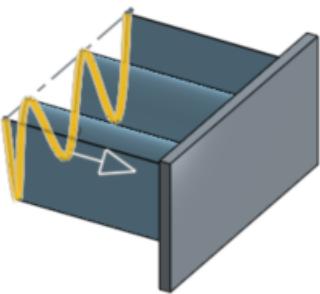
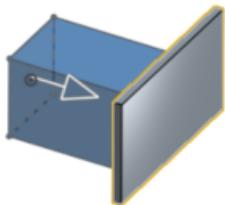
Up to the infinite face underlying the selected face or plane (select a plane or face of a part); note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed





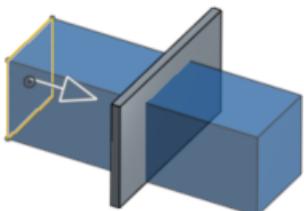
## Up to part

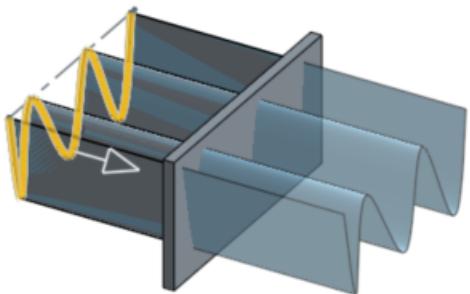
Up to the next part encountered in the given direction; if there is no part encountered, the extrude will fail. Note that the sketch region or entity being extruded must fall entirely within the target entity for the extrude to succeed



## Through all

Through all selected parts

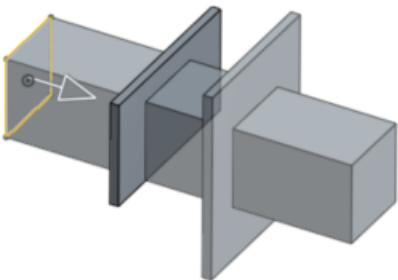




## Merge scope

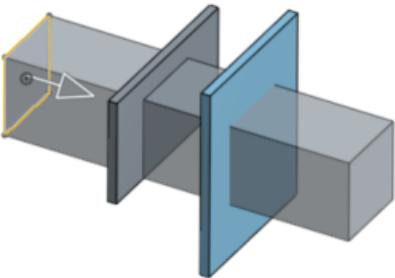
### Merge scope: with all

Merge extrusion with all parts it intersects



### Merge scope: particular part

Select a specific part with which to merge



# Revolve

In the Feature toolbar:



In the **Sketch** toolbar:

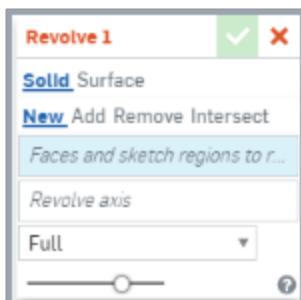


Revolve projects a selected region or planar face about an axis. Create a new part or modify an existing one by adding or removing material, or intersecting bodies in its path. You can also create solid bodies or surfaces.

## Steps

From the **Sketch** or **Feature** toolbar:

1. Click .



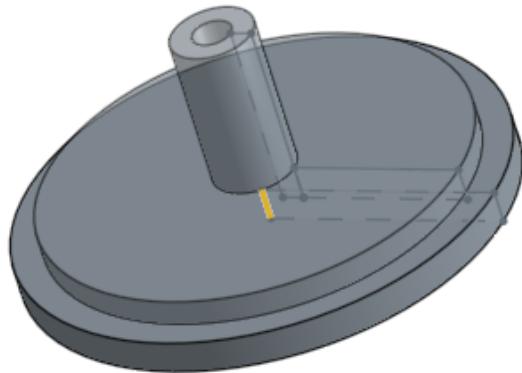
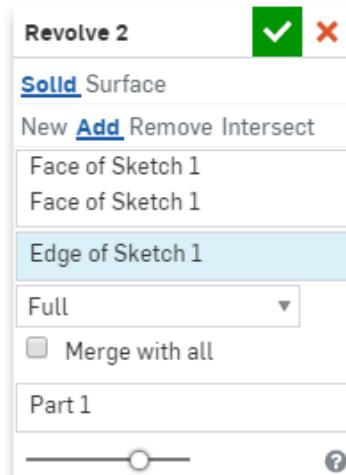
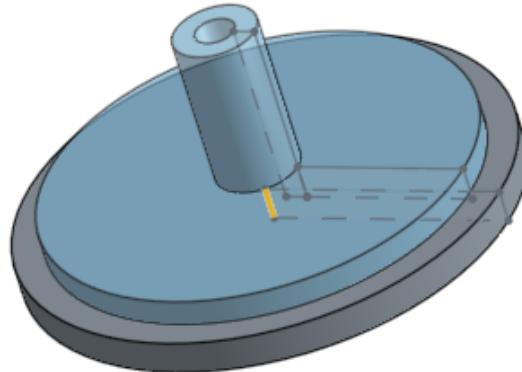
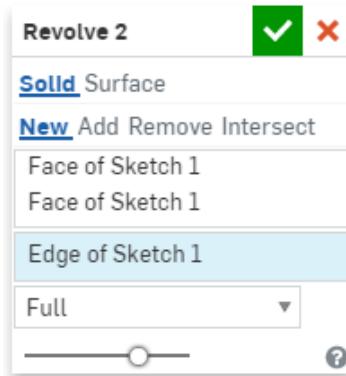
The arrows in the image above indicate (from left to right) the Preview Slider and the Context-sensitive help button.

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 faces, edges, or sketch regions to revolve.
4. Activate the **Revolve axis** field, then click the axis about which to revolve.
5. Choose whether you want to define a solid from faces or regions, or add a surface from edges or sketch curves. If you define a solid, 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
- Two directions - Revolve in both directions for the same or different numbers of degrees

7. Click .

## Revolve new material (New, Add)

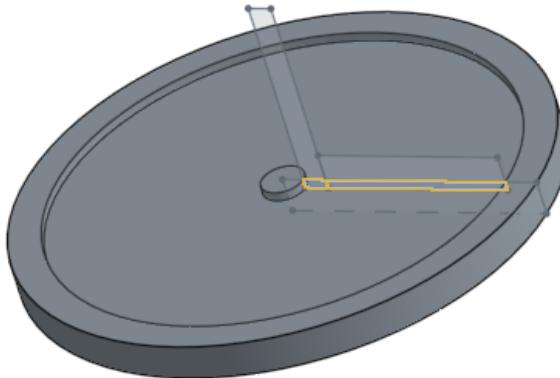
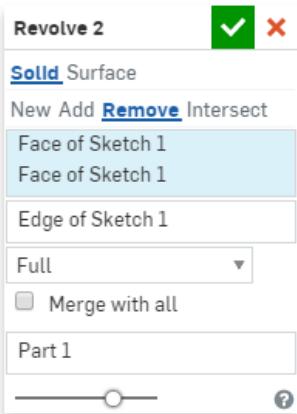


When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

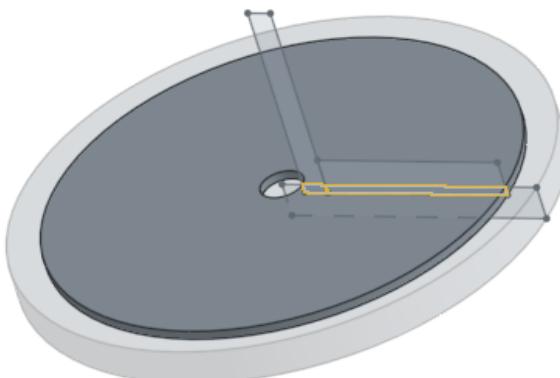
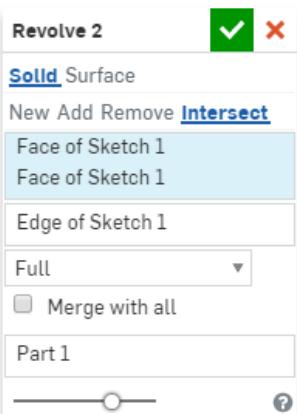
- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching 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 error. For **New**, no merge scope is available since New does not boolean the result.

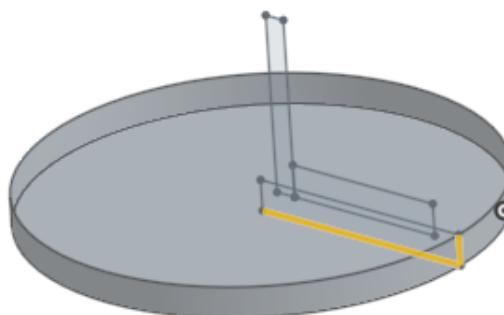
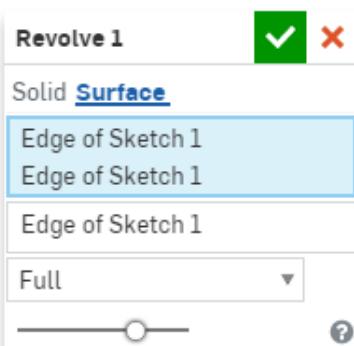
## Revolve Remove



## Revolve Intersect

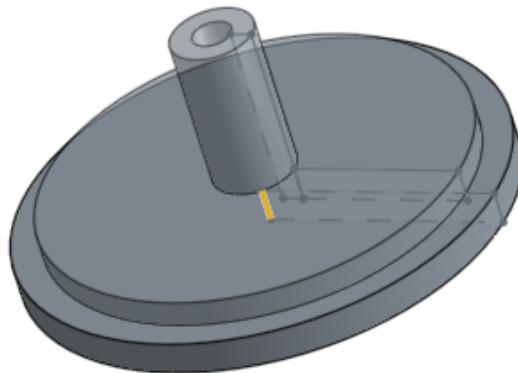
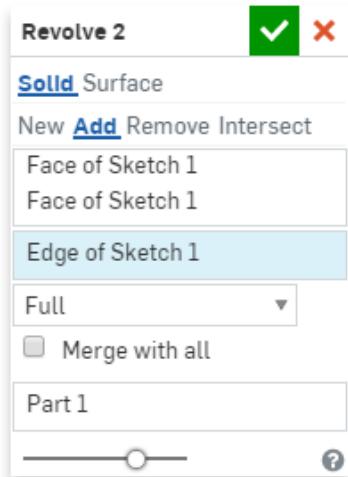


## Revolve Surface

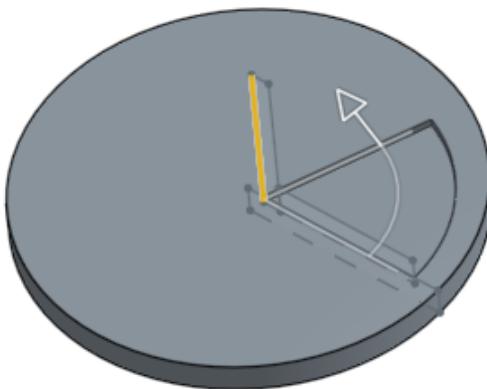
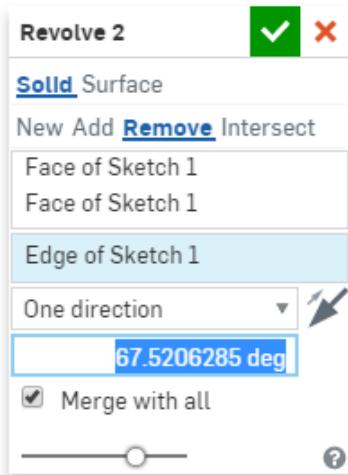


## Revolve type examples

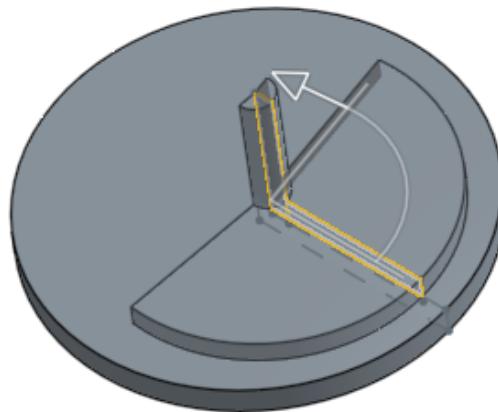
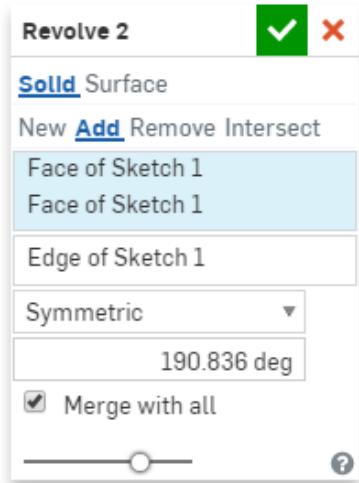
Full



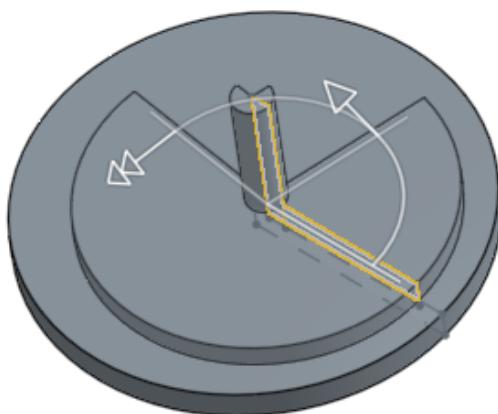
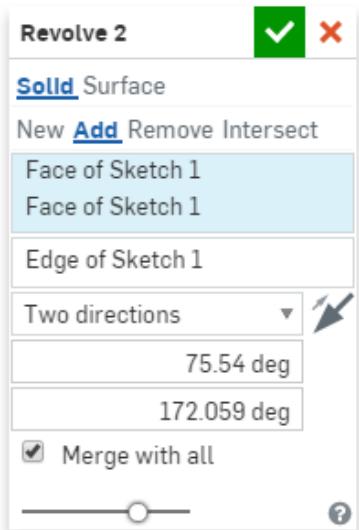
## One direction



## Symmetric

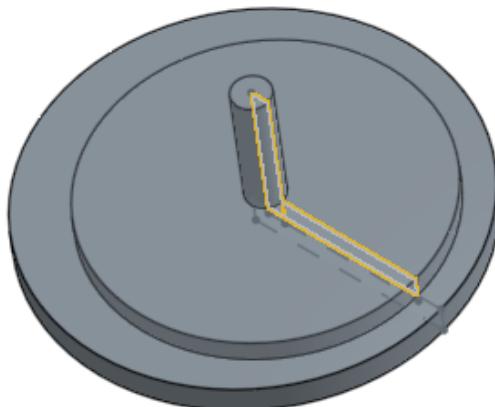
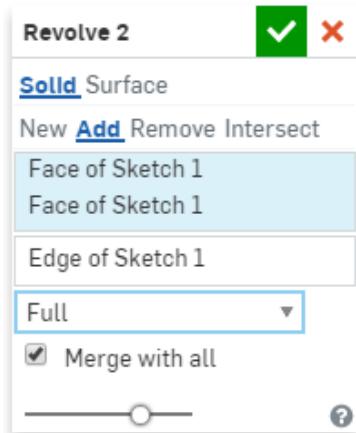


## Two directions

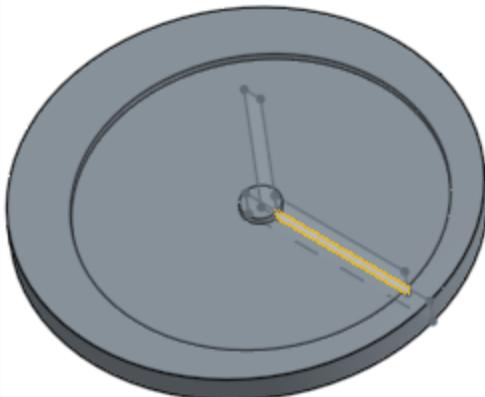
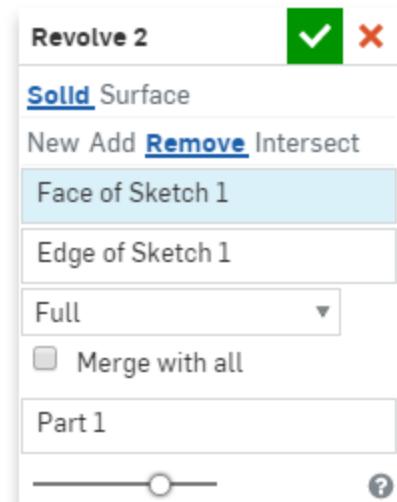


## Merge scope

Merge scope: with all



Merge scope: with specific part



# Sweep



Sweep uses a selected region, curves, or planar face moving along a path to define a shape (either solid or surface body). Create a new part or modify an existing one by adding or removing material, or intersecting bodies in its path.

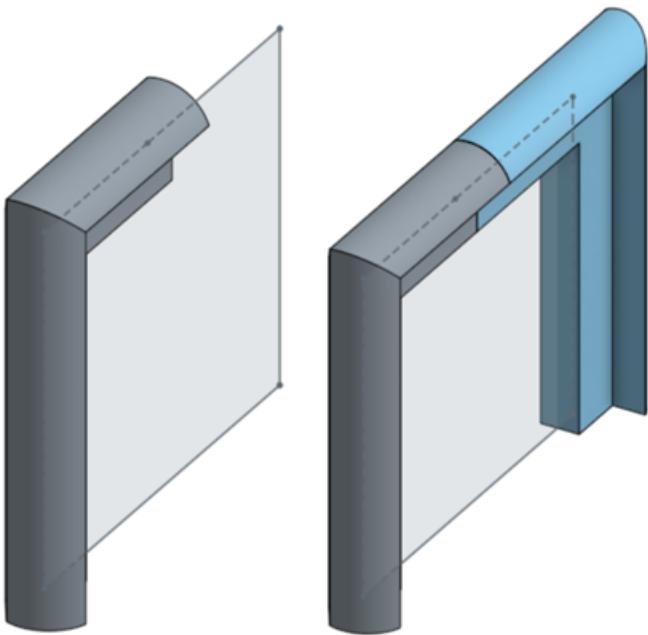
## Steps

1. Click .



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 choose:
  - **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. Click .

## Sweep new material (New, Add)

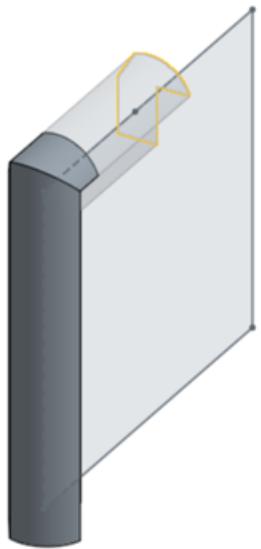


When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

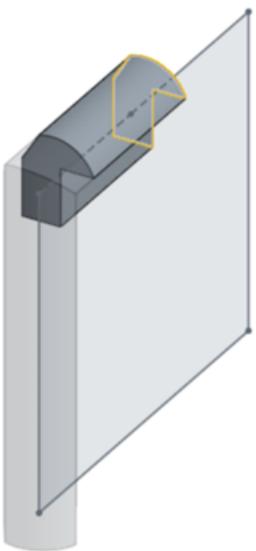
- If the geometry touches or intersects with only one part then that part is automatically added to the merge scope.
- If multiple parts touch or intersect the geometry, then there is ambiguity and you must select which parts to merge with (the merge scope).
- A shortcut to selecting multiple touching 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 error. For **New**, no merge scope is available since New does not boolean the result.

## Sweep Remove



**Sweep Intersect**





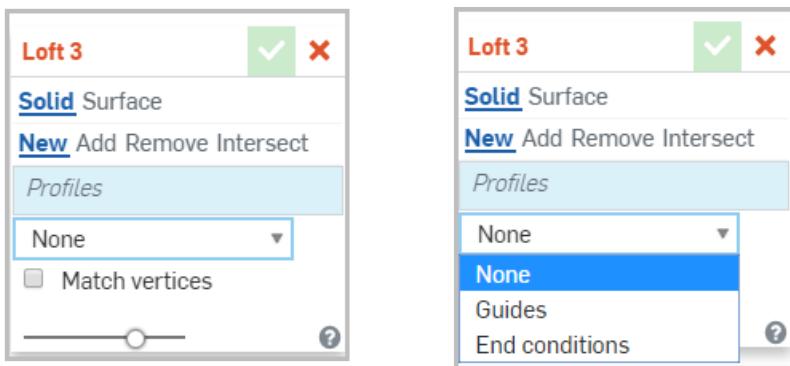
# Loft



Use profiles (sketch regions or sketch curves) and optional guide curves to define shapes that smoothly transition between them. Create solid or surface bodies or modify existing solid bodies.

## Steps

1. Click .



2. Select Creation type:
  - Solid** - Create parts or modify existing parts
  - Surface** - Create a surface
3. Specify:
  - 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 the start profile (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 connected curves as a single chain, select them from the Feature list as a complete sketch.
5. Select from the drop down whether to use optional guides (to help define the loft) or end conditions:
  - Guides** - Select the 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):
    - 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 (you can use the scroll wheel or the keyboard to change these values).

6. Select optional vertices to match (to define corresponding locations on each profile):

- a. Click **Match vertices**.

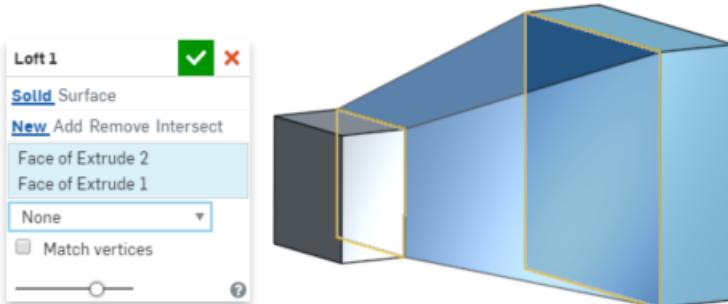
- b. Select one set of vertices (one vertex on each region/face/edge/point).

## Loft solid

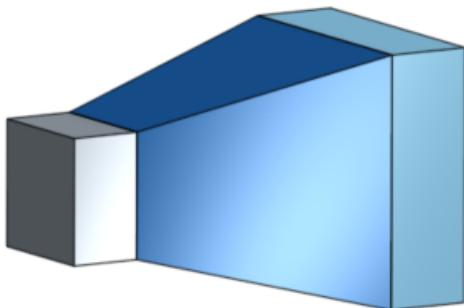
### Before loft



### During loft



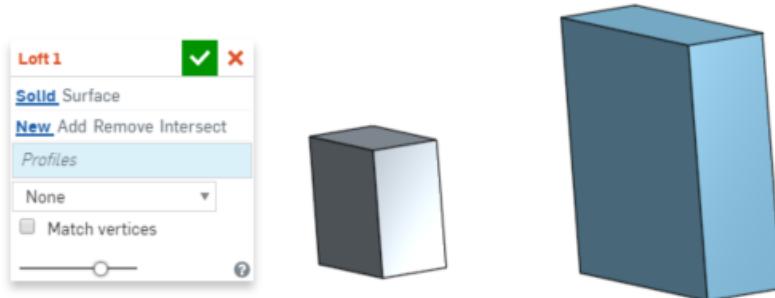
### After loft



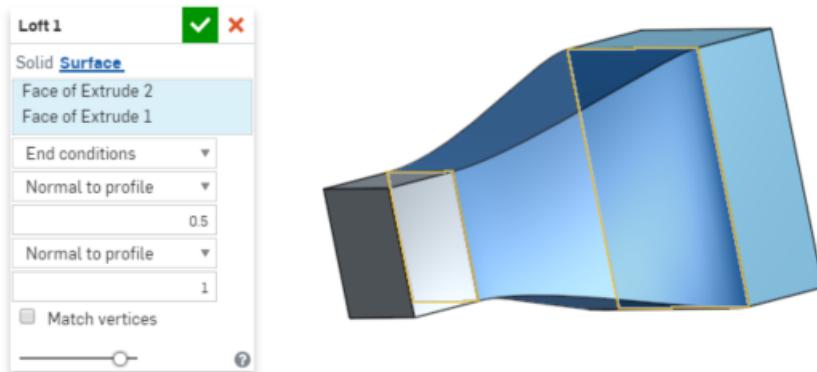
## Loft solid with end conditions

This example presents the multiple end conditions available with loft.

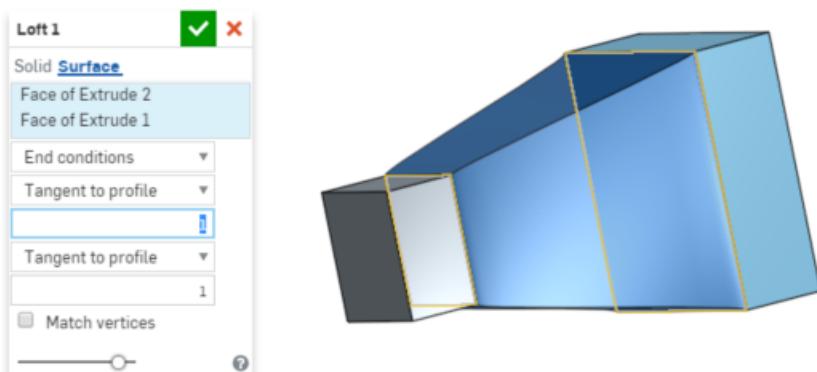
### Starting profiles, no end conditions



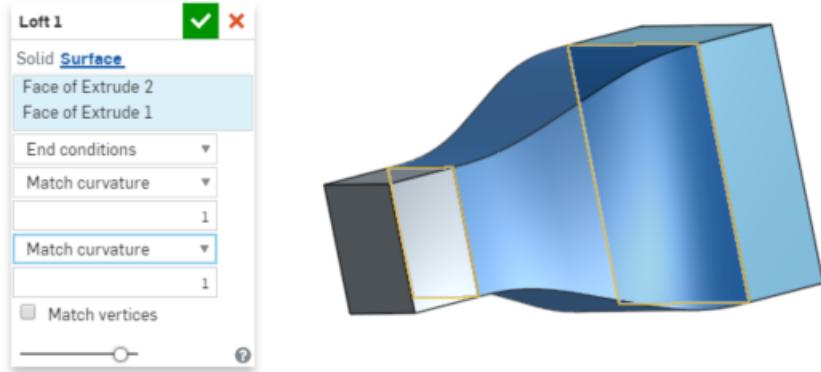
### Normal to profile end condition



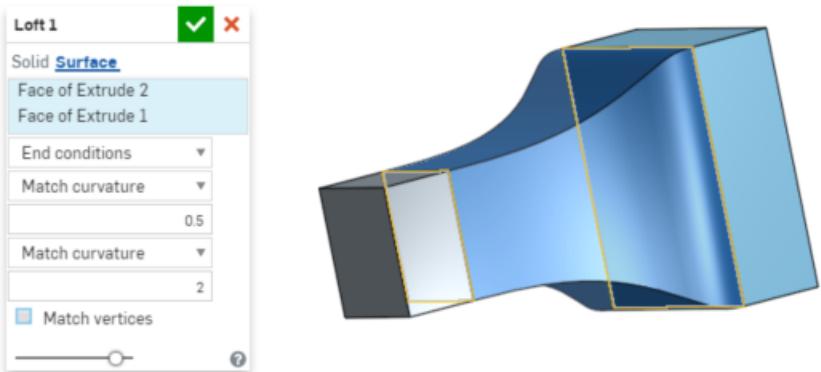
### Tangent to profile end condition



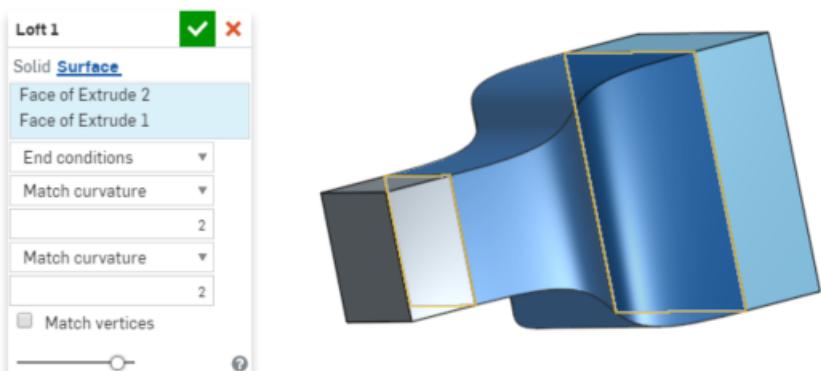
### Match curvature



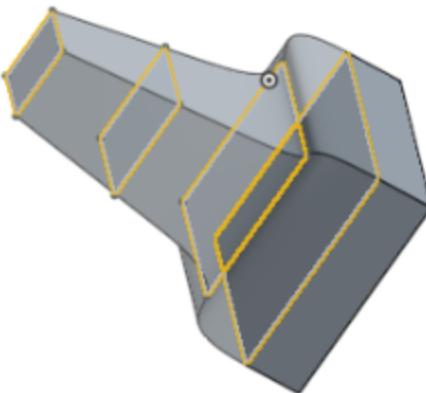
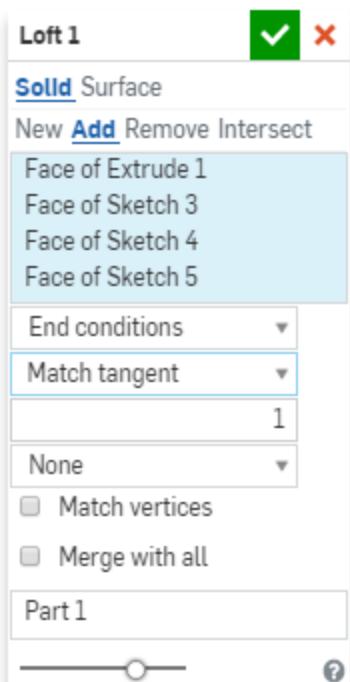
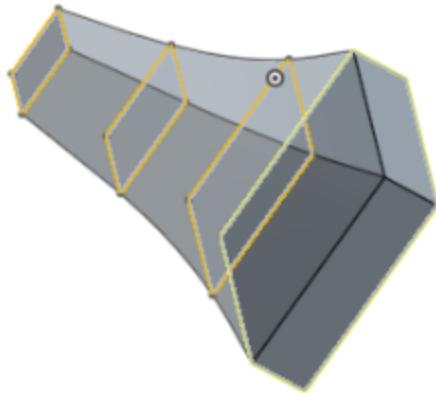
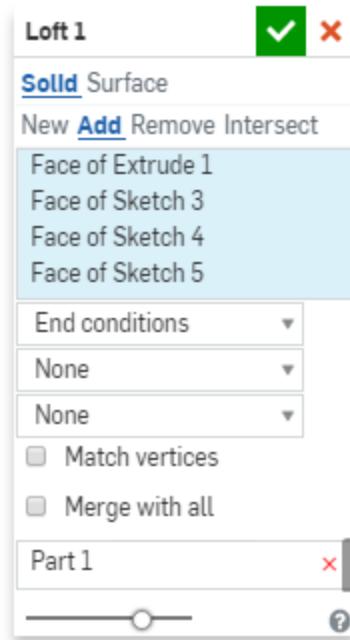
Match curvature with magnitude specification

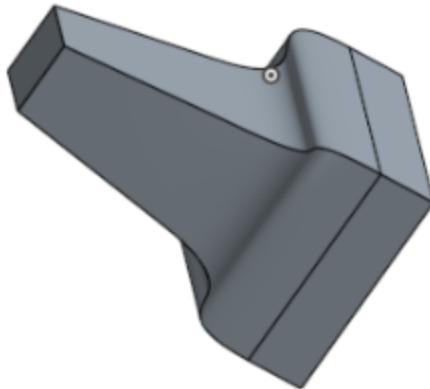


Magnitude example



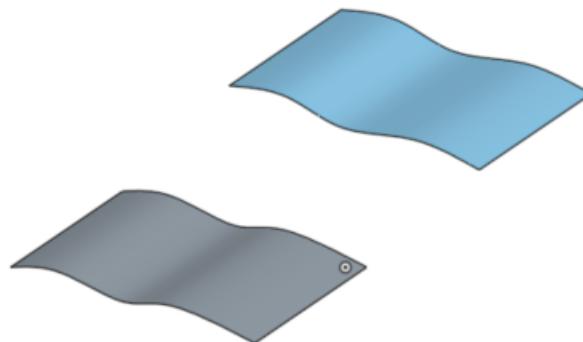
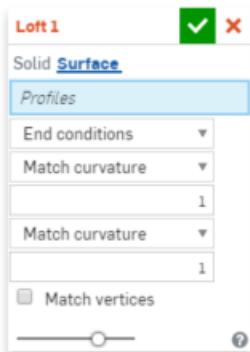
Match tangent example



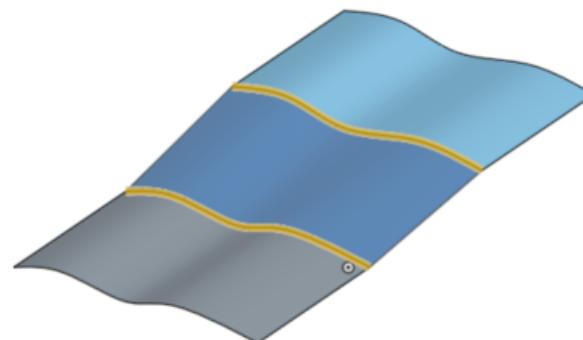
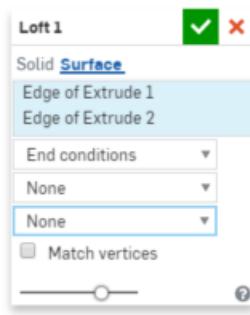


## Loft surface

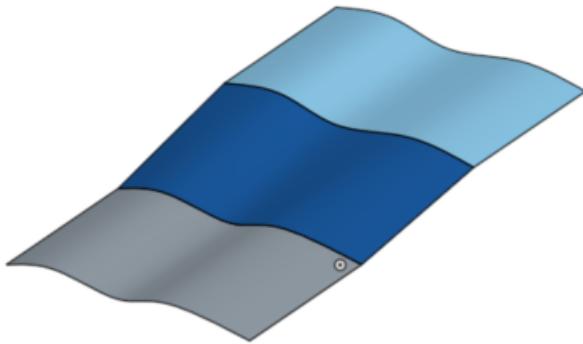
Before loft



During loft

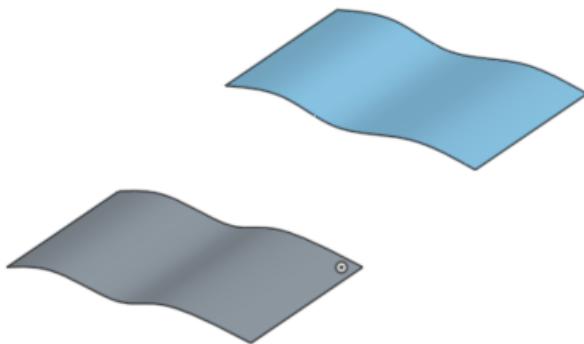


After loft

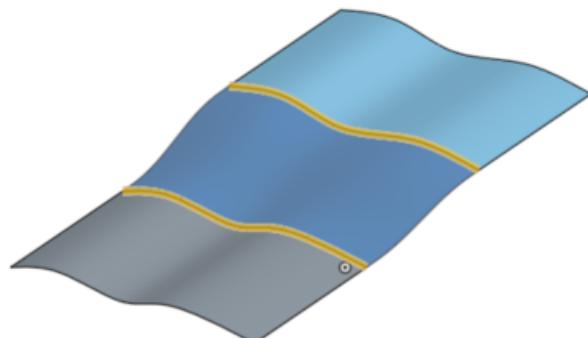
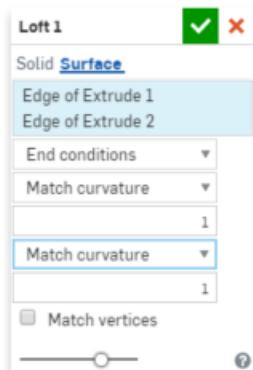


## Loft surface with end conditions

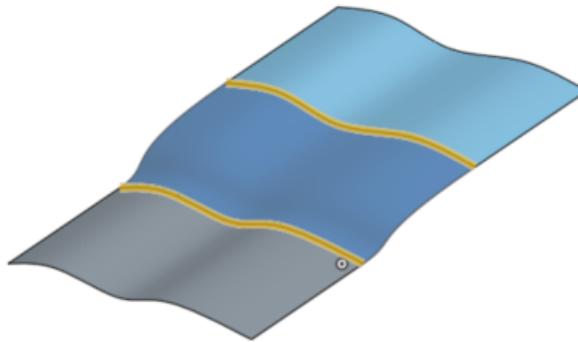
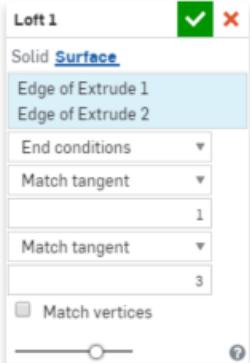
Before loft



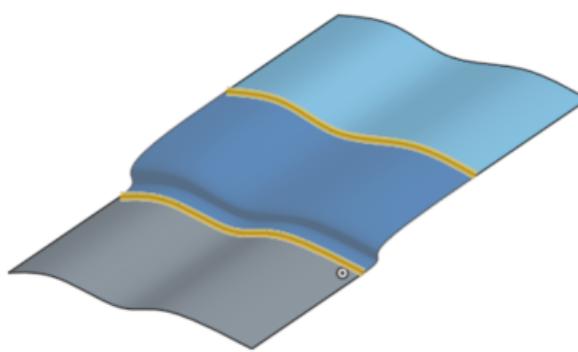
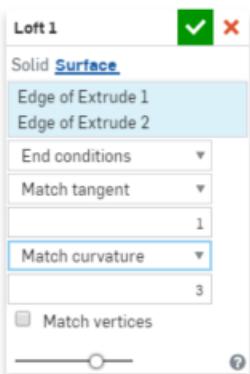
Match curvature



Match tangent, with magnitude



Match tangent, Match curvature, with Magnitude



## 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.
- 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 error. For **New**, no merge scope is available since New does not boolean the result.



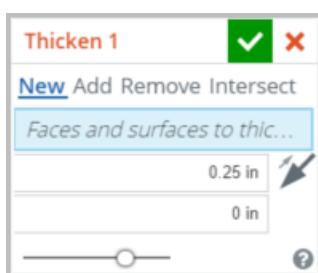
# Thicken



Thicken adds depth to a surface. Create a new part or modify an existing one by giving thickness to a surface body and convert it to a solid, adding or removing material from an existing body, or intersecting bodies in its path.

## Steps

1. Click



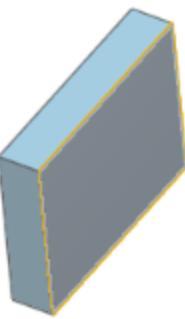
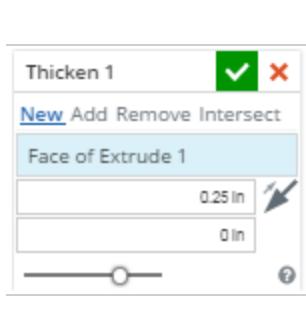
2. Select whether to:
  - New** - Create a new solid
  - Add** - Add to an existing solid
  - Remove** - Subtract material from an existing part
  - Intersect** - Keep only intersecting materials
3. Select the part face (or surface) in the graphics area.
4. Specify the thickness of material to be added 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 Surface

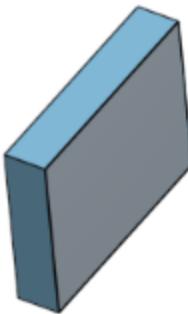
Before Thicken - Surface



Thicken - Surface

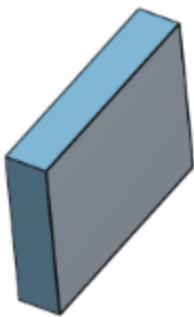


After Thicken - Surface

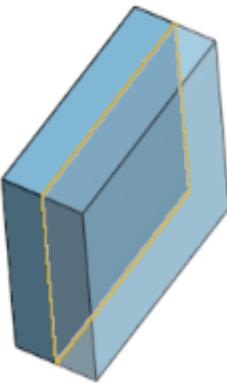
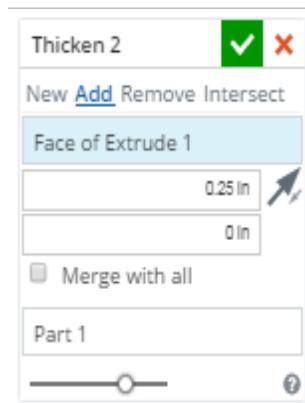


## Thicken Part

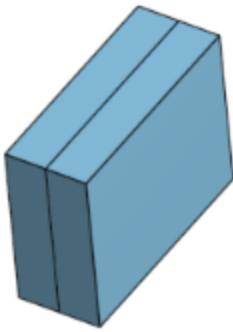
Before Thicken - Part



Thicken - Part



After Thicken - Part



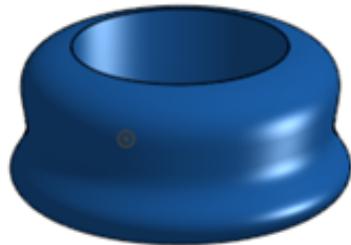
When adding material, you have the option to merge that material with other parts that touch or intersect its geometry:

- 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, you can check **Merge with all** to add all touching or intersecting parts to the merge scope.

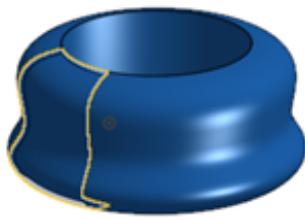
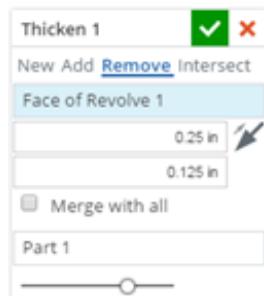
If the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will error. For New, no merge scope is available since New does not boolean the result.

## Thicken Remove

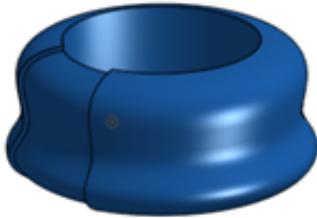
Before Thicken - Remove



Thicken - Remove



After Thicken - Remove



# Fillet



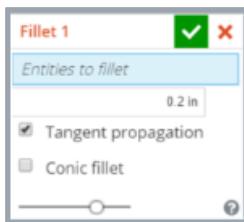
Shortcut: Shift-f



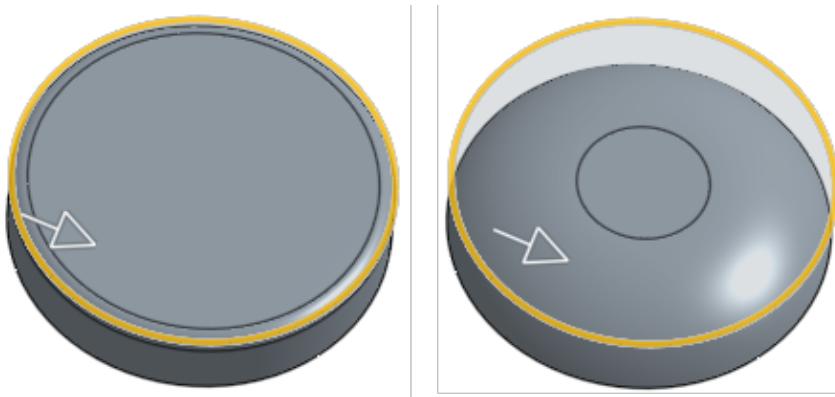
Fillet rounds sharp interior and exterior edges and can be defined as a standard constant radius, or more stylized conic.

## Steps

1. Click .



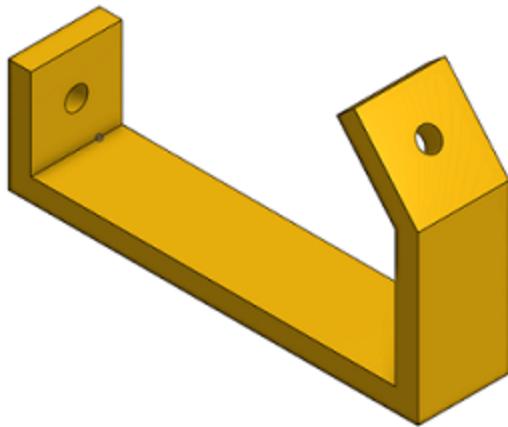
2. Select any edges or faces of the part you want to round or fillet. Onshape automatically applies the correct feature to the edge.
3. Enter a radius value (you can also use the drag manipulator, as indicated in the image below by an arrow, to visualize the fillet and approach an estimate value):



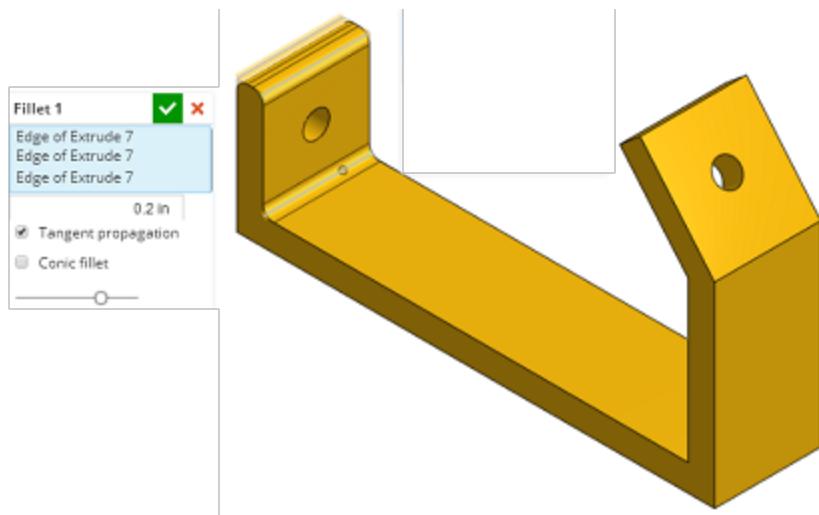
4. Optionally, check **Tangent propagation** to automatically extend the fillet to tangent edges.
5. Optionally, check **Conic fillet** to apply a Rho value for a more stylized fillet:
  - A value of 0.5 results in a parabolic curve
  - A value less than 0.5 results in an elliptical curve
  - A value greater than 0.5 results in a hyperbolic curve
6. Click .

## Fillet

### Before fillet

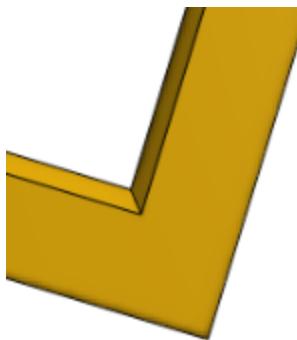


**With fillet**

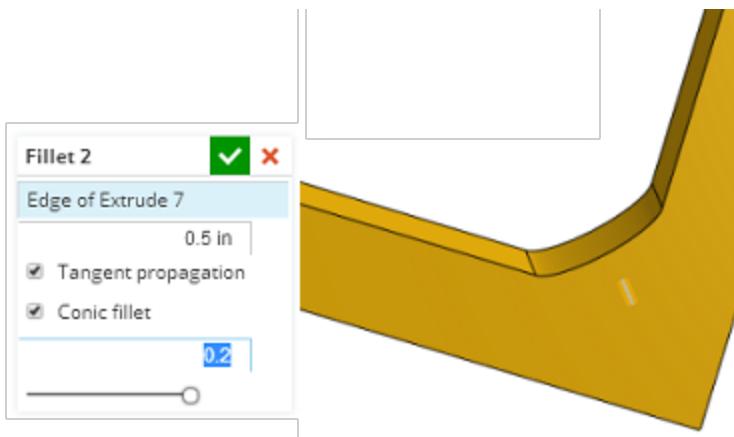


## Conic Fillet

**Before conic fillet**

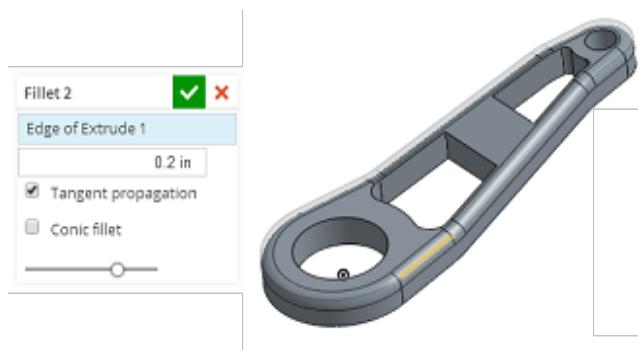
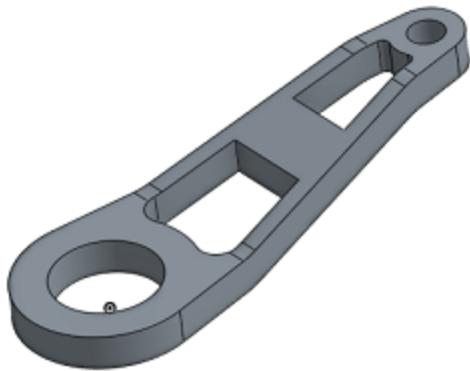


**With conic fillet (elliptical)**



## With Tangent Propagation

With tangent propagation





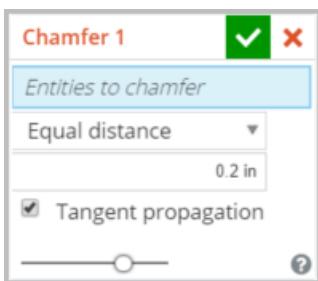
# Chamfer



Chamfer breaks sharp edges with a bevel. Define by the distance to break from the edge and by the angle made with the surface.

## Steps

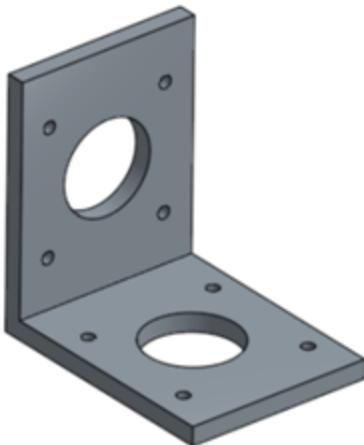
1. Click

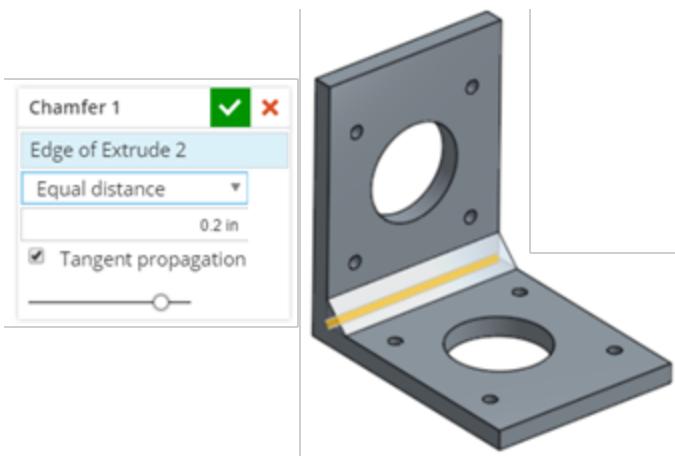


2. Select any edges or faces of the part to which to apply the chamfer.
3. Enter a width for the chamfer; Onshape applies the 45 degree angle by default.
4. Optionally, check **Tangent propagation** to extend the selection along surrounding edges.
5. Click .

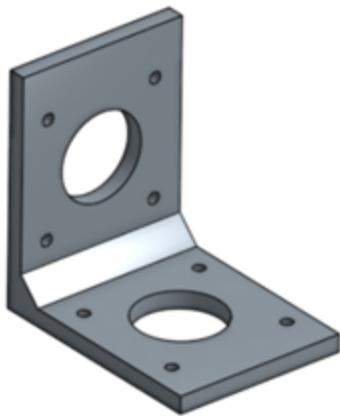
## Equal-distance Chamfer

Before equal-distance chamfer



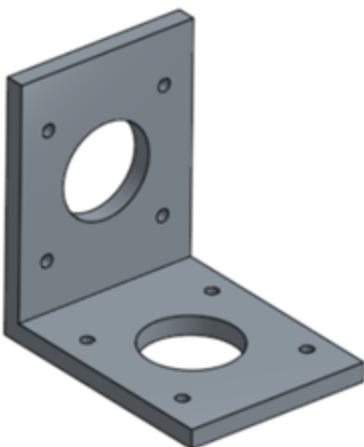


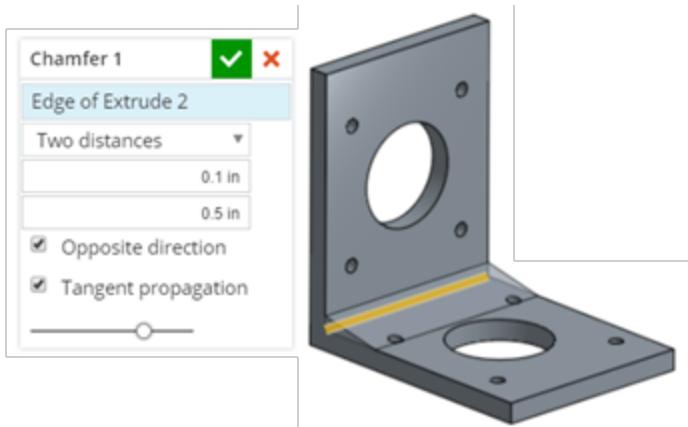
With equal-distance chamfer



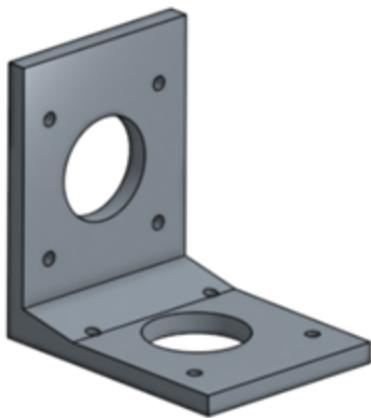
## Two-distance Chamfer

Before two-distance chamfer



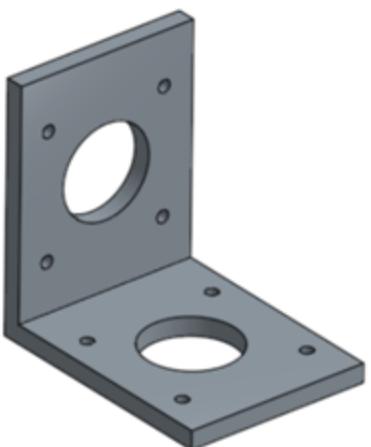


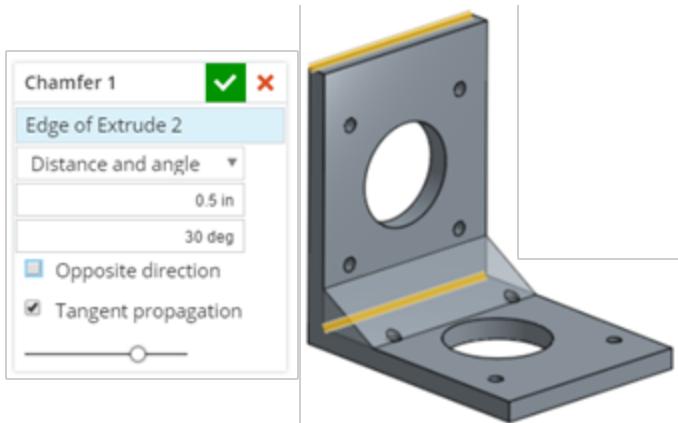
With two-distance chamfer



## Distance-and-angle Chamfer

Before distance-and-angle chamfer





With distance-and-angle chamfer



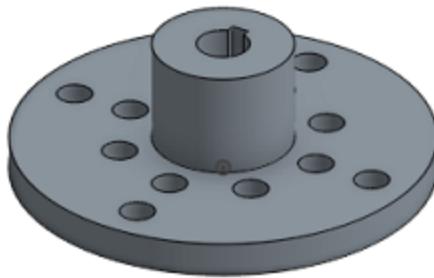
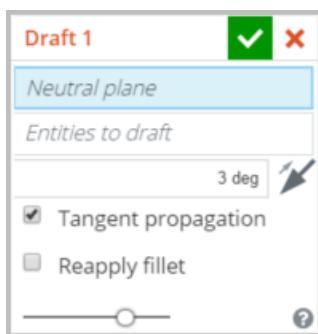
# Draft



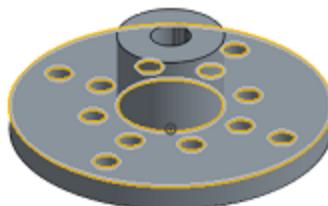
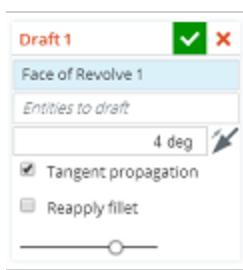
Apply a taper to one or more selected faces in order to facilitate pulling a part from a mold.

## Steps

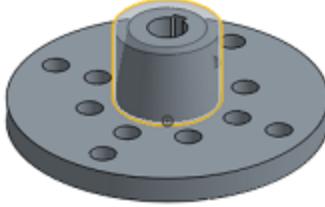
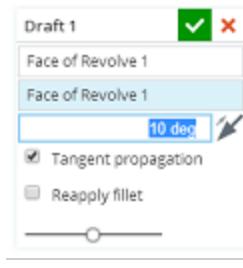
1. Click .



2. With focus on the *Neutral plane* field in the dialog, click on the face of the part to act as a neutral plane.



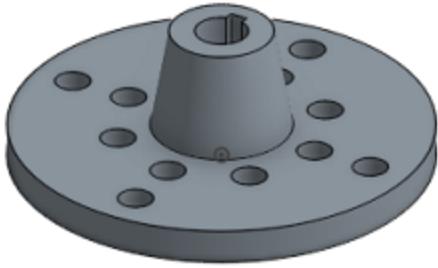
3. Click in the *Entities to draft* field, then select (all of) the faces to which to apply the draft.



4. Specify the degree of draft in the numeric field.
5. Indicate whether to apply the draft along tangent propagation; this allows you to select as shown above and the draft is applied to all tangent faces.

Note that:

- Tangent propagation will select only faces that steeper than the draft angle.
  - With Reapply fillets unchecked: steep fillet faces are treated as draft faces, not fillet faces. This will generate cones and preserve the parting line edges. Frequently, with large draft angles, this produces undesirable geometry.
  - With Reapply fillets checked: steep fillet faces are treated as fillets and reblended. This results in cylindrical faces instead of cones, modified the parting line edges and more often produces a desirable result.
  - In all cases, fillets that are not steep will be reapplied.
6. Optionally, use the slider to visualize the difference between before the draft is applied and after.
7. Click .



## Tips

You can use the direction arrows in the dialog to change the direction of the draft.

# Shell



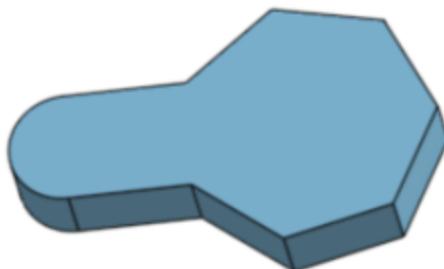
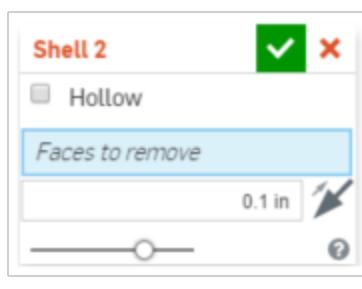
Shell removes material from a body to produce a cavity of constant wall thickness with the option to remove zero faces (hollow) to many faces of the part (shell).

## Steps

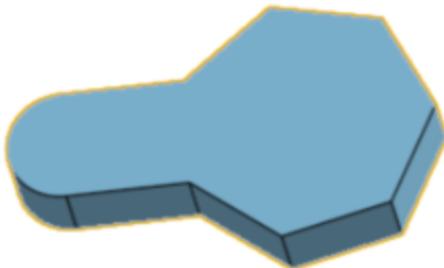
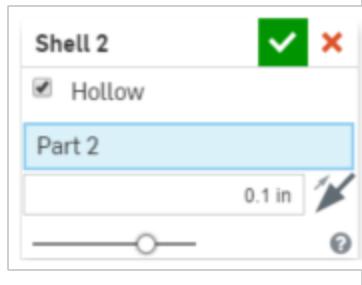
1. Click .



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.)



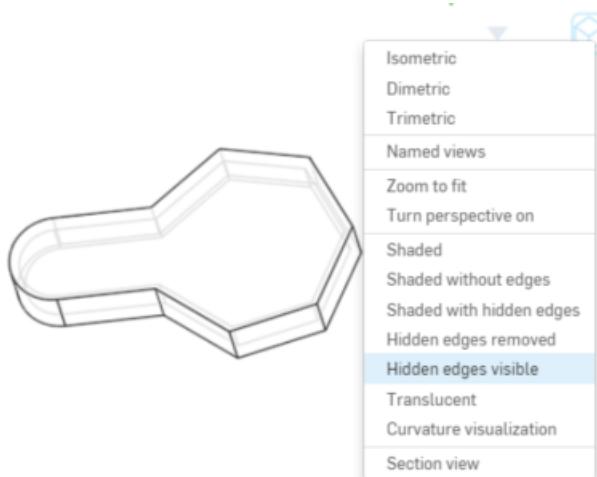
Optionally, check the **Hollow** box to shell (hollow) the part without removing any faces:



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.



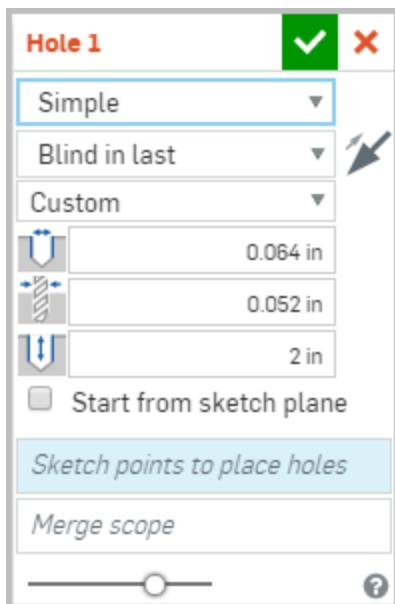
# Hole



Create simple, countersink, and counterbore holes at sketch points, using ANSI/ISO standards or custom specifications.

## Steps

1. Click .



Note that the selections for the last hole created are presented as defaults when you open the Hole dialog.

2. Select a hole style:
  - a. Simple - A uniform-diameter drilled hole
  - b. Counterbore
  - c. Countersink
3. 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
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

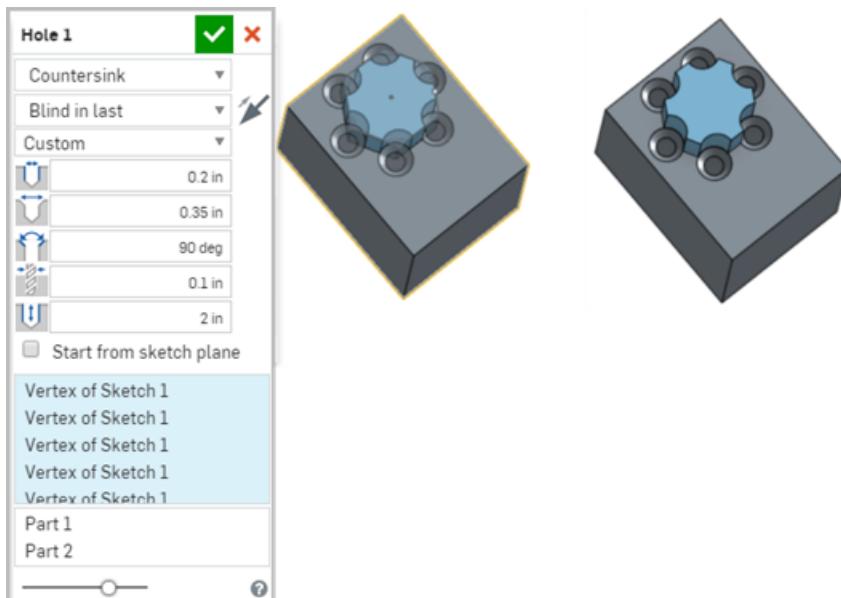
- Fit - Close, Free/Standard
- Drill size - where appropriate
- Pilot drill diameter - where appropriate
- Threads/inch and percent thread engagement - where appropriate

Percent thread engagement refers to how much of the thread is available due to the change in the diameter of the tap hole.

- When choosing Custom, enter:
    - Hole diameter
    - Counterbore/Countersink diameter - where appropriate
    - Counterbore depth, Countersink angle - where appropriate
    - Hole depth
5. Indicate *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. With focus in the *Sketch points to place holes* field, select points in the sketch at which to place holes (box select is available).
  7. With focus in the *Merge scope* field, select the part(s) to contain the holes.
  8. Click .

## 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.

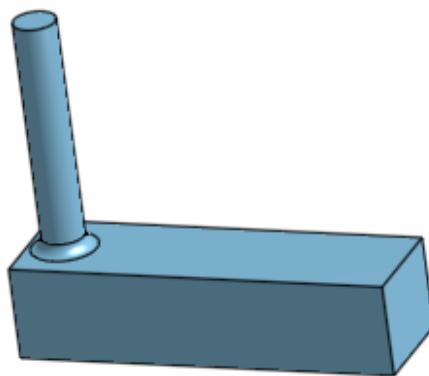
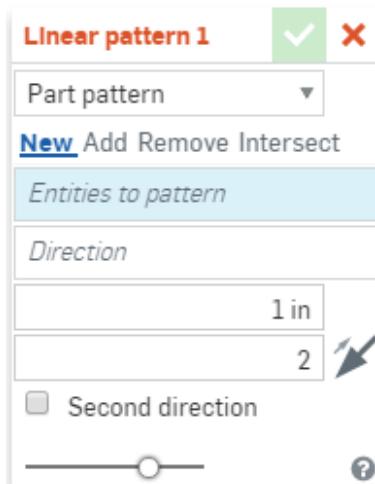
# Linear Pattern



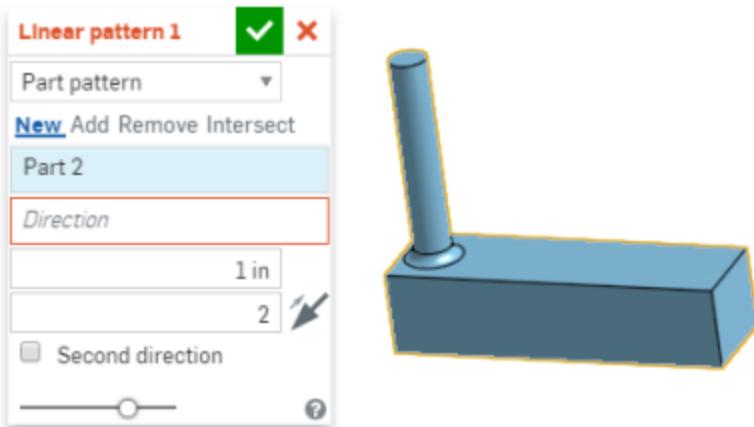
Replicate selected parts, faces, or features and arrange them in a row or grid pattern. Create new parts or modify existing parts by adding or removing material, or intersecting bodies in its path. For information on creating circular patterns, see "Circular Pattern" on page 232.

## Steps to create linear pattern

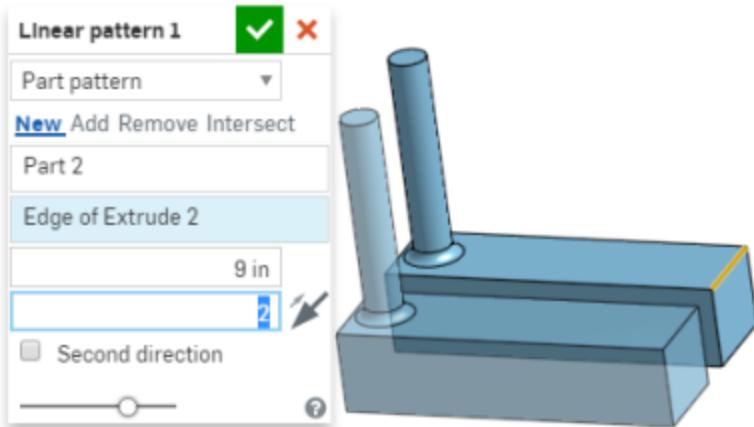
1. Click



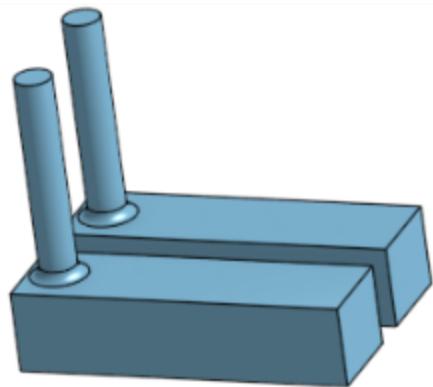
2. Select the pattern type: Part, Feature, or Face:
  - a. Part - To pattern an individual part
  - b. Feature - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc)
  - c. Face - To pattern a specific face on a specific part
3. With focus on the *Entities to pattern* field, select the parts, features, or faces on the part to replicate into a pattern.



4. Set focus in the *Direction* field, and then select an edge or face of the part along which to place the replicated pattern parts.
5. Enter the distance between each pattern part, and then the number of repetitions. Select a direction for the pattern in the workspace (shown below as the highlighted edge):



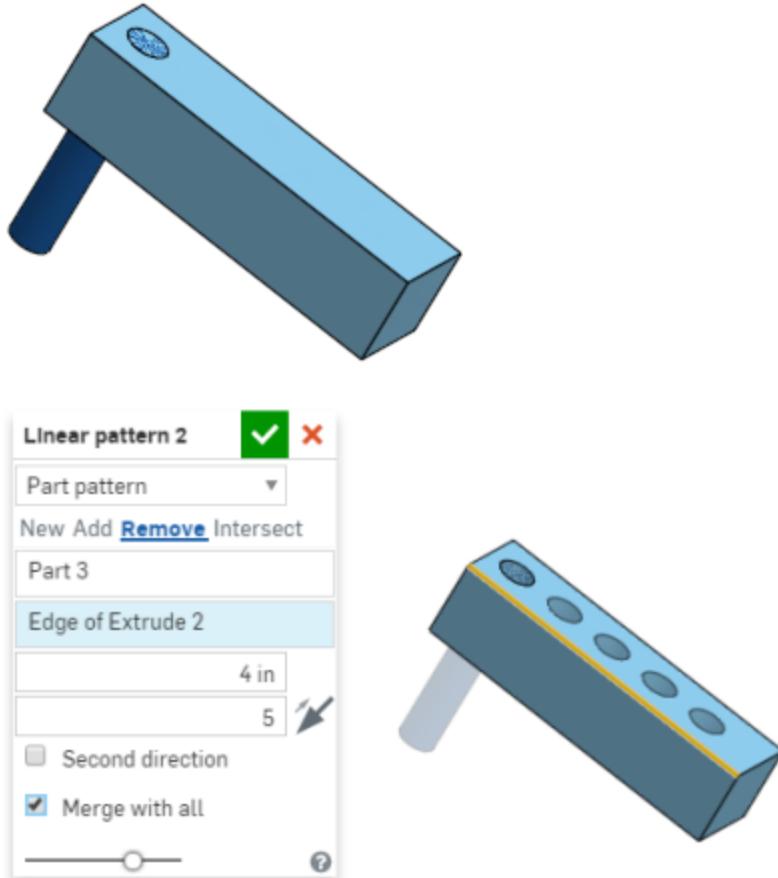
6. Click



You can also remove material or limit the result to only intersecting material.

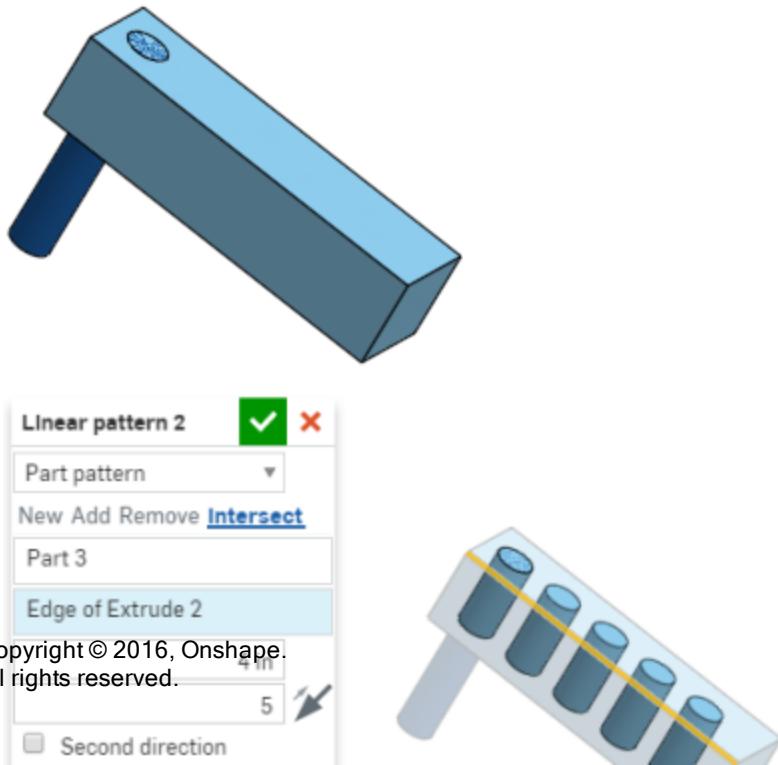
## Removing material example

Select the part to pattern, and then **Remove**:



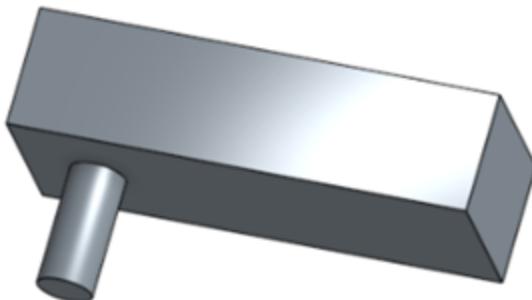
## Intersecting material example

Select the part to pattern, and then **Intersect**:



## Steps to create linear face pattern

Creating a face pattern requires a face on the part to replicate. This example starts with this part:



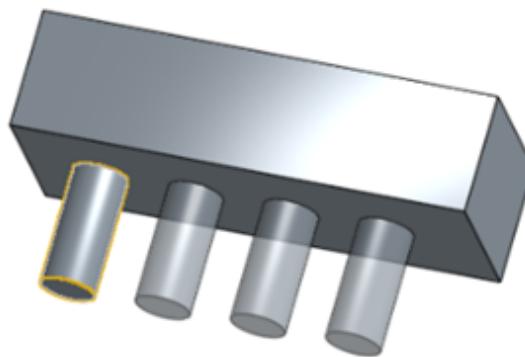
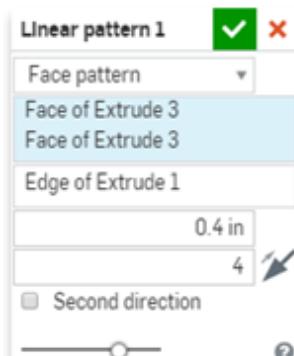
1. Click .

2. Select **Face pattern**.

3. Select the face or faces :

In this case, the cylindrical face and the top face of the cylinder were selected.

4. Set focus in the *Direction* field, and then select an edge or face of the part along which to place the replicated pattern face.
5. Enter the distance between each face, and then the number of repetitions.



## Steps to create linear feature pattern

Create a pattern of one or more features in the Feature list, which will be applied to the selected part, in sequence.

1. Click .

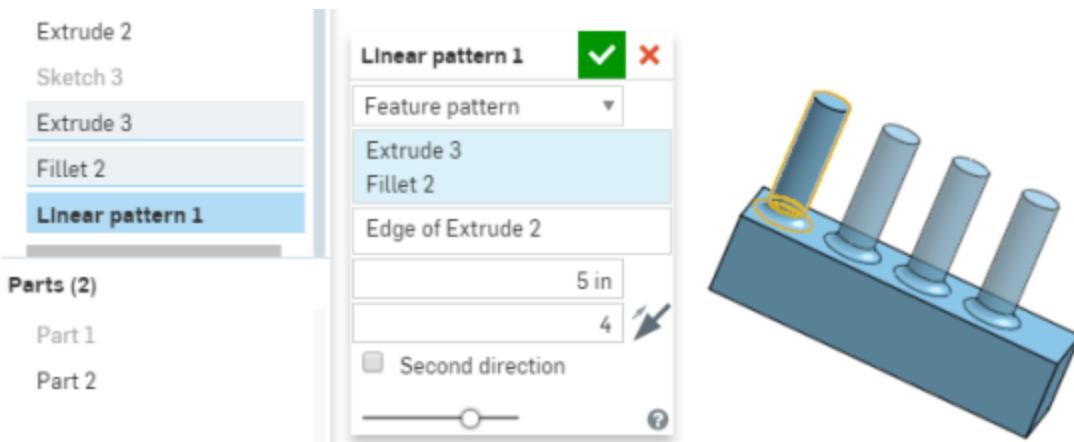
2. Select **Feature pattern**.

3. Select the Features to pattern, either in the graphics area or in the Feature list.

4. Select an edge or face to specify direction.

5. Enter a distance between features.

6. Enter the number of pattern instances desired:



## 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.)

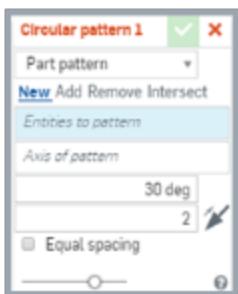
# Circular Pattern



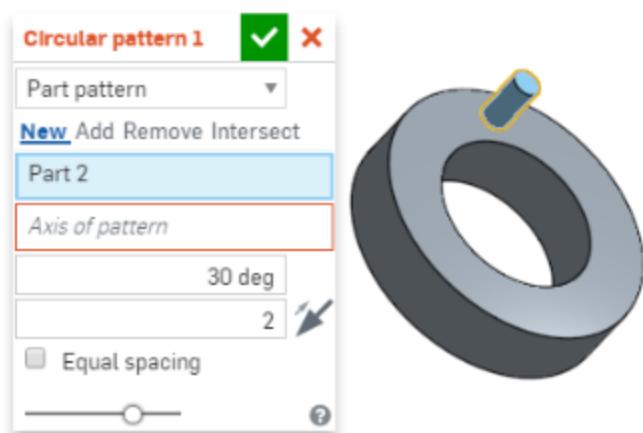
Circular pattern replicates selected parts, faces, or features about an axis. For information on creating linear patterns, see "Linear Pattern" on page 227.

## Steps to create circular pattern

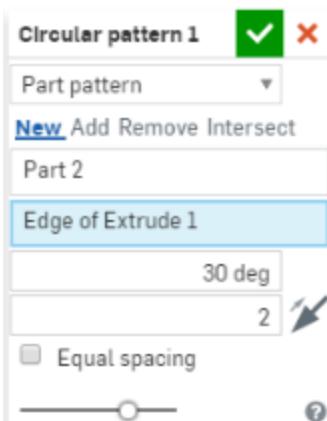
1. Click :



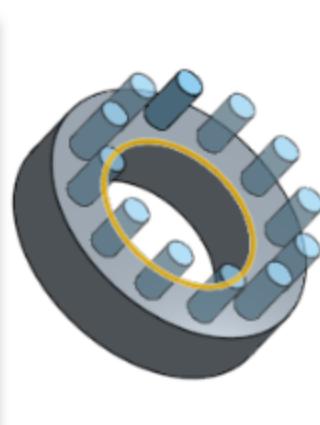
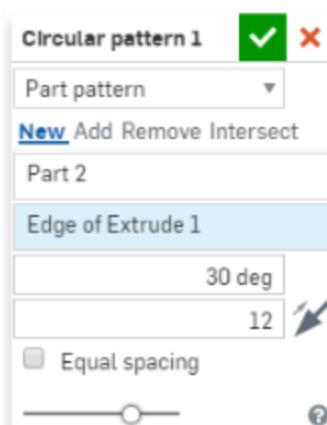
2. Select the pattern type: Part, Feature, or Face:
  - a. Part - To pattern an individual part
  - b. Feature - To pattern a specific feature (or features) listed in the Feature list (an extrude, fillet, sweep, sketch, etc)
  - c. Face - To pattern a specific face on a specific part
3. With focus on the *Entities to pattern* field, select the parts or faces on the part to replicate into a pattern.



4. Set focus in the *Axis of pattern* field, and then select an edge, face, or conic or cylindrical face of the part about which to place the replicated pattern parts (in this case selecting the cylindrical face, Face of Extrude 2, uses the axis defined by that face).

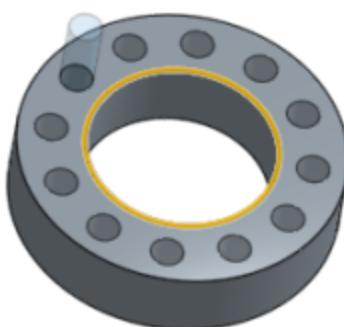
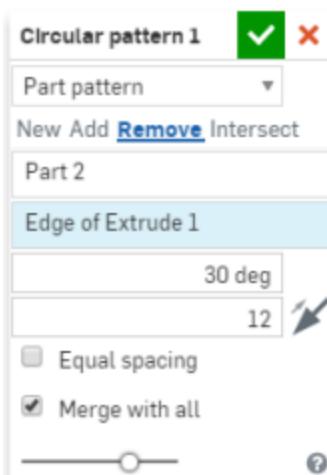


- Enter the distance between each pattern part, and then the number of repetitions **30 deg** and **12**, below).



This places each pattern part 30 degrees apart and creates a total of 12 instances, including the original part.

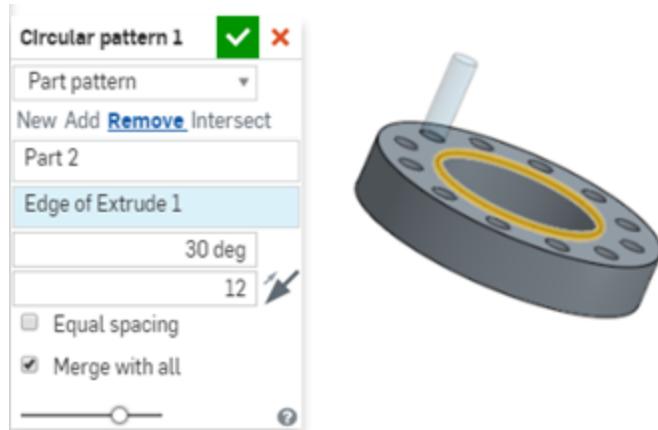
- The **Equal spacing** box allows you to place the pattern parts within the specified degrees.



You can also remove material or limit the result to only intersecting material.

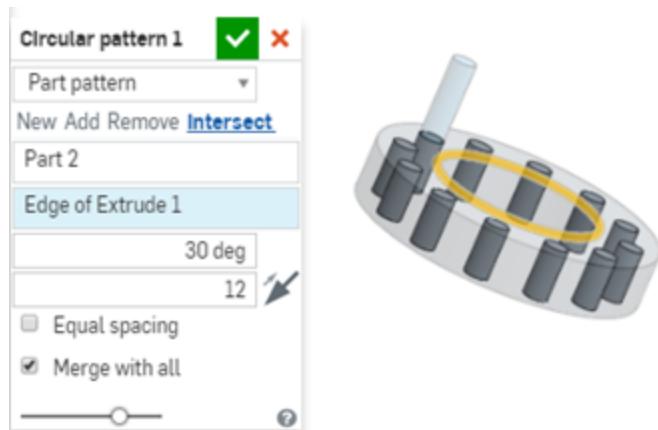
## Removing material example

Select the part to pattern, and then **Remove**:



## Intersecting material example

Select the part to pattern, and then **Intersect**:

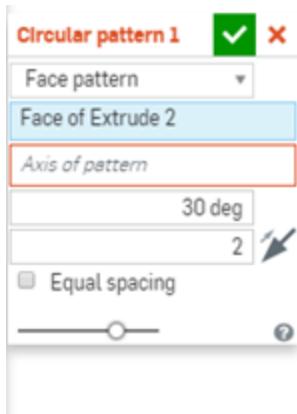


## Steps to create circular face pattern

Creating a face pattern requires a face on the part to replicate. This example starts with this part:



1. Click .
2. Check **Face pattern**.
3. Set focus in *Faces to pattern* and select the face of the small cylinder:

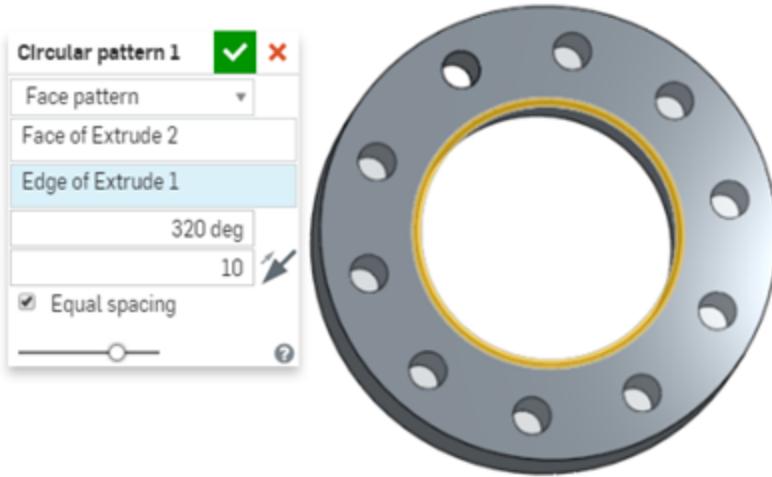


4. Set focus in *Axis of pattern* and select the outside face of the large cylinder:

The axis can be: an edge, face, or a conic or cylindrical face.

5. Enter the distance between each pattern part, and then the number of repetitions.
6. The **Equal spacing** box allows you to place the pattern parts within the specified degrees. Notice how the

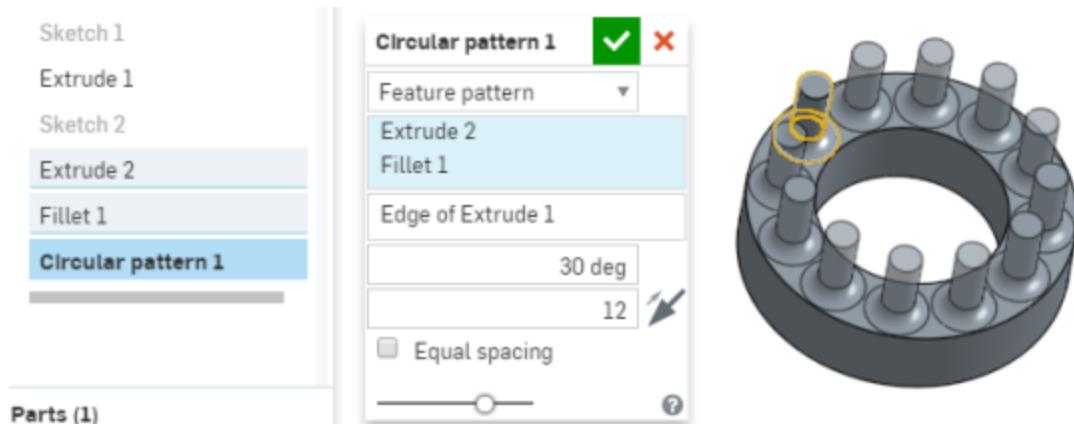
parts change when the Equal spacing box is checked:



## Steps to create circular feature pattern

Create a pattern of one or more features in the Feature list, which will be applied to the selected part, in sequence.

1. Click ..
2. Select Feature pattern.
3. Select the Features to pattern, either in the graphics area or in the Features list.
4. Select an edge or face to specify direction.
5. Enter a distance between the features.
6. Enter the number of pattern instances desired:



## 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.)

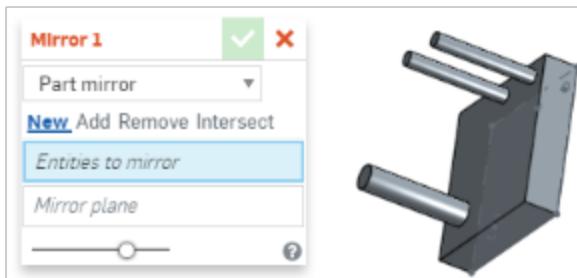
# Mirror



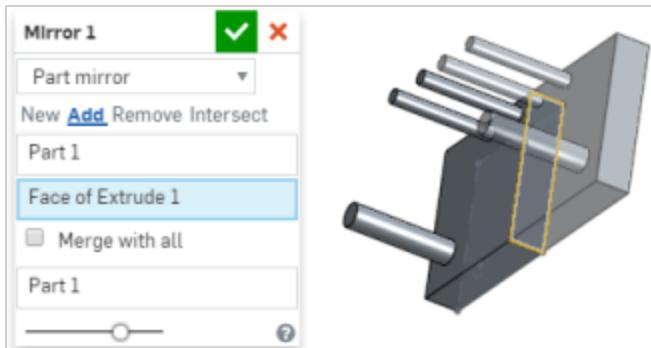
Mirror replicates one or more selected parts about a specified plane or planar face. Create a new part or modify an existing one by adding or removing material, or intersecting bodies in its path.

## Mirroring parts

1. Click



2. 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.
3. With the focus on the *Entities to mirror* field, select the part or parts you want to mirror.
4. Click in the *Mirror plane* field to give it focus, then select the plane or planar faces about which to mirror.



5. Notice that with the slider towards the right, you get an instant preview of the result.
6. Select whether to merge the new part 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.

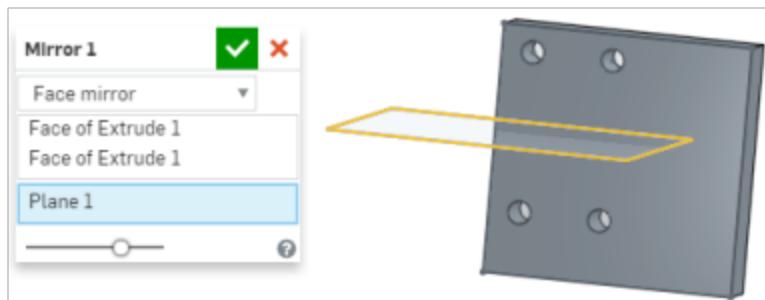
Note that if the Boolean is set to Add, Remove, or Intersect and nothing is set in the merge scope, the feature will error. For **New**, no merge scope is available since New does not boolean the result.

7. Click .

## Mirroring faces

Mirror simple bosses and holes contained entirely within the bounding geometry, about a plane or planar face.

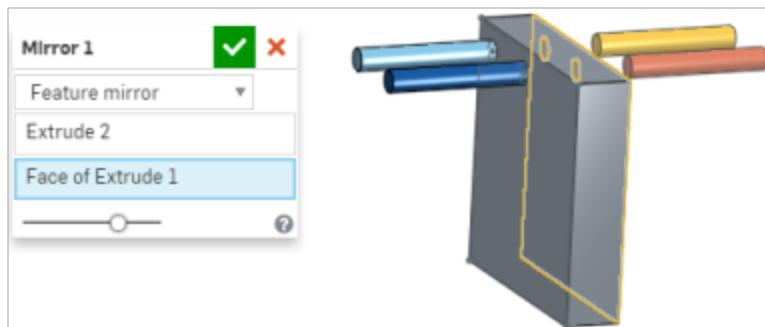
1. Click .
2. Select **Face mirror** from the drop down.
3. Select the faces to mirror.
4. Select the plane or planar face about which to mirror.



## Mirroring features

Mirror features contained entirely within the bounding geometry, about a plane or planar face.

1. Click .
2. Select **Feature mirror**.
3. Select the features to mirror, either on the model or in the Feature list.
4. Select the plane or planar face about which to mirror.



# Boolean



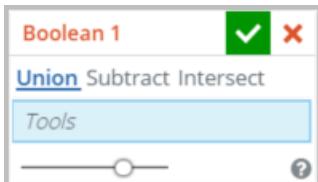
Boolean modifies part by merging bodies together (Union), removing a tool body from a target (Subtract), or calculating the intersection between two or more bodies (Intersect).

## Boolean union (merge parts)

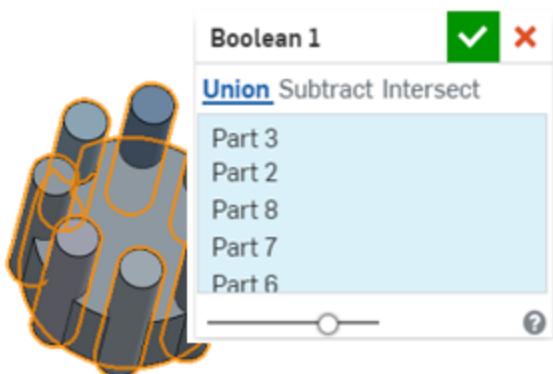
This example starts with these parts in the graphics area (created using a circular pattern):



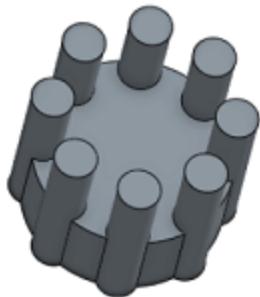
1. Click .



2. Confirm that **Union** is selected.
3. Click to set the focus in the *Tools* field, then click all of the parts (small cylinders and larger cylinder).



4. Click .

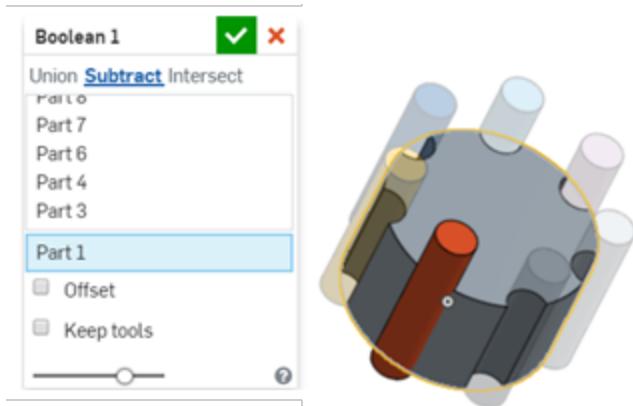


## Boolean subtract (remove parts)

This example starts with these parts in the graphics area (created using a circular pattern):



1. Click .
2. Select **Subtract**.
3. Click to set the focus in the *Tools* field, then click each of the small cylinders.
4. Click to set the focus in the *Target* field, then click the larger cylinder.



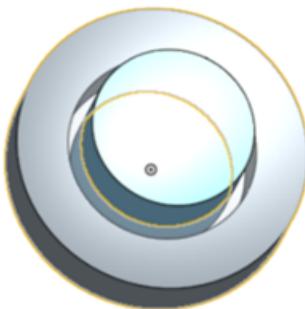
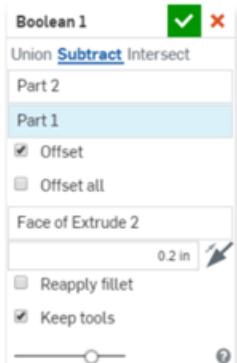
5. Click .



## Boolean subtract, offset

Subtract a part from another, leaving clearance between the two:

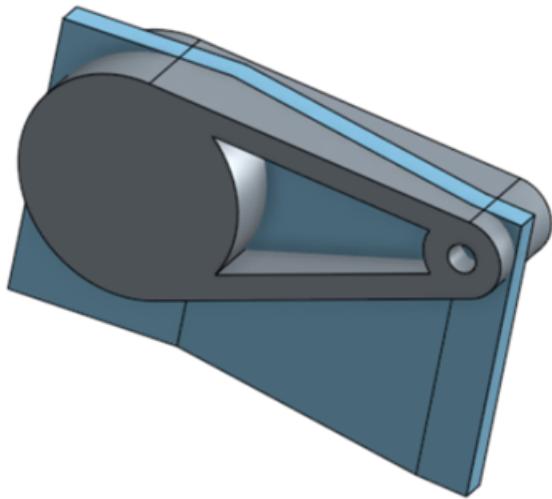
1. Click .
2. Select **Subtract**.
3. Specify whether to Offset one part, or all parts (**Offset** or **Offset all**).
4. If offsetting one part, select the face of the part to offset in Tools field.
5. Select the Target part (to offset from) in the Target field.
6. Specify the distance of the offset.
7. Indicate whether or not to keep the tools.



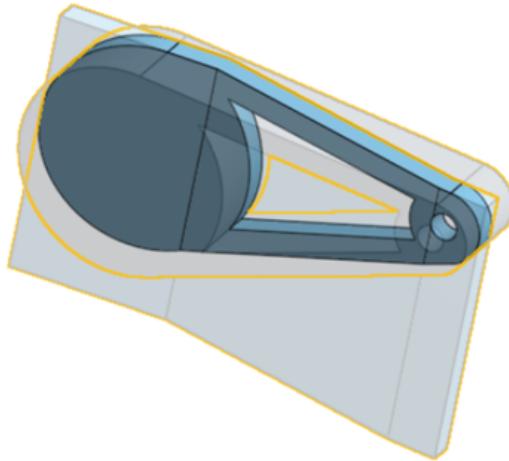
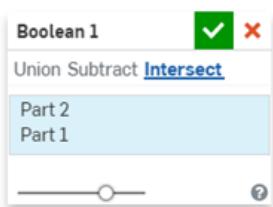
8. Click .

## Boolean intersect

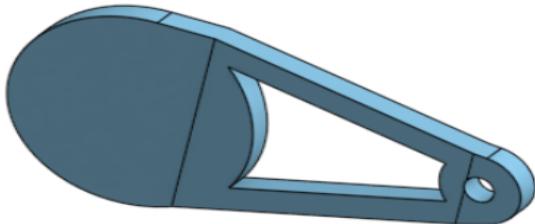
This example starts with these two parts:



1. Click
2. Select **Intersect**.
3. Click to set the focus in the *Tools* field, then click each of the two parts.



4. Click



## Tips

- With the Subtract option, you have the choice to use the **Keep tools** checkbox to keep the parts used to cut

the main part. This is useful when creating fitted parts within the Part Studio.

- Use the **Intersect** option to keep only the material that intersects the selected parts.
- When parts are merged and as a result some parts no longer exist, the attributes of the earlier selected part (such as part name) are retained.

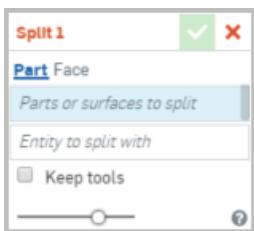
# Split



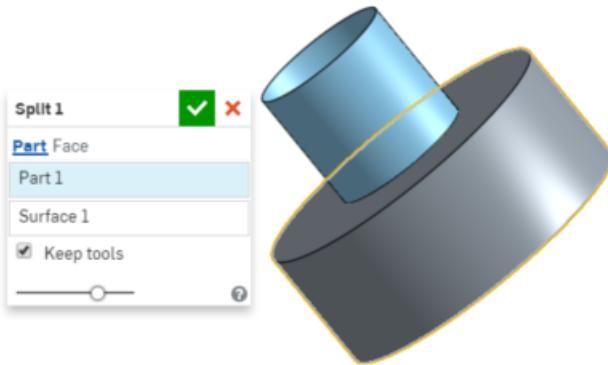
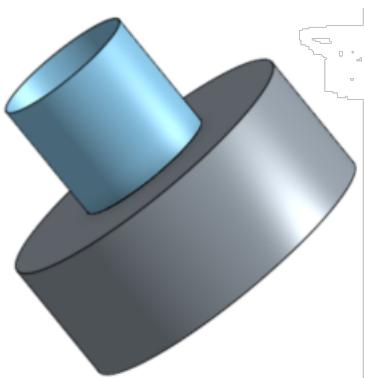
Split uses a plane or surface to separate an existing part or face into multiple new parts or faces.

## Splitting a part

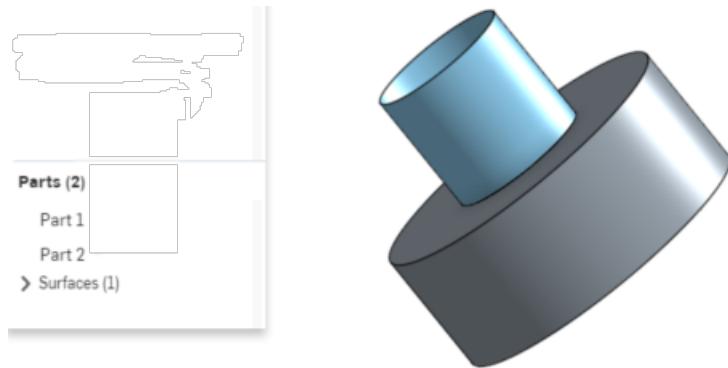
1. Click .



2. Select the part or parts to split (in this example, Part 1). This is an example of an extruded part with an extruded surface.



3. Select the entity with which to split the selected part (in this example, Surface 1).
4. Select **Keep tools** to keep the entity with which you split the part; leave this unchecked to discard the entity with which you split the part (in this case Surface 1).



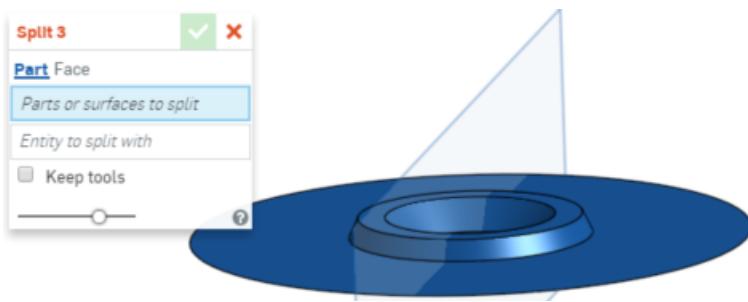
As a result, Part 2 is listed in the Parts list.

5. Click .

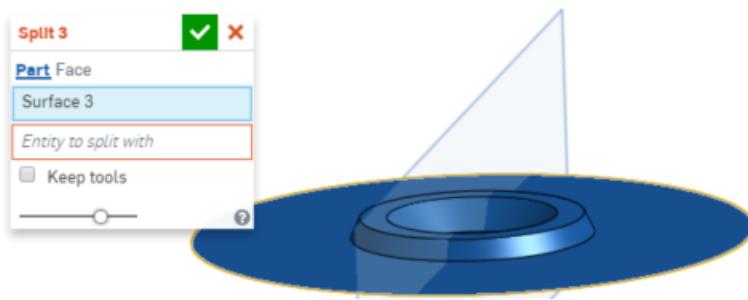
The colors seen in your Part Studio may differ from these. Parts resulting from the Split part tool will be the same color.

## Splitting a surface

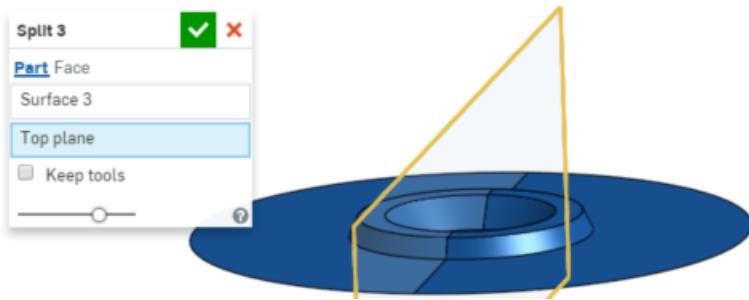
1. Click .



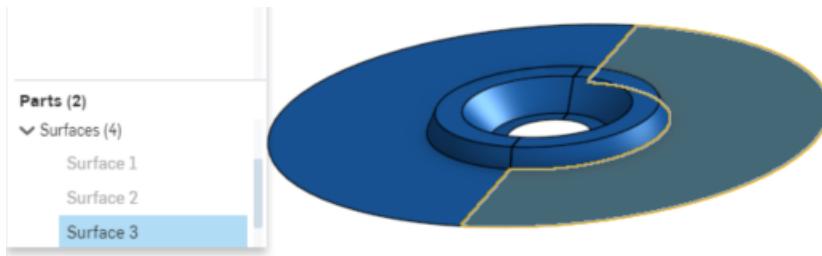
2. Select the surface to split:



3. Select the split tool, Right plane:

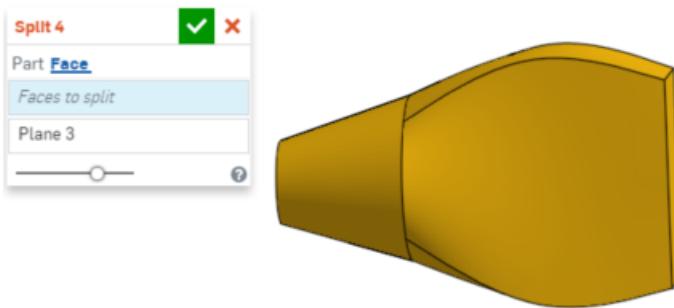


4. Click .  
5. Note that there is an additional surface listed in the Parts list:

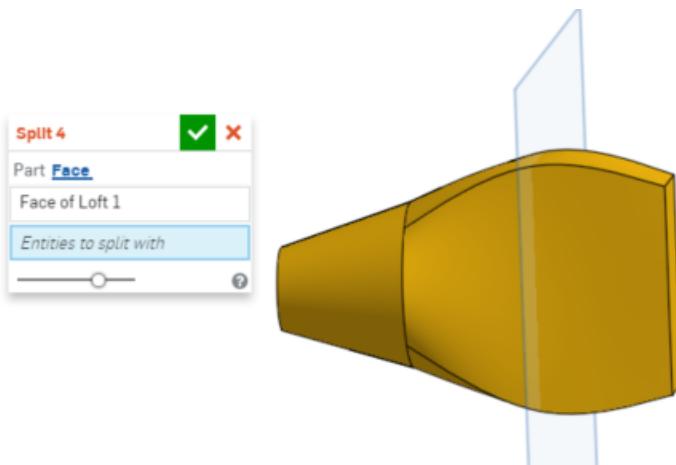


## Splitting a face

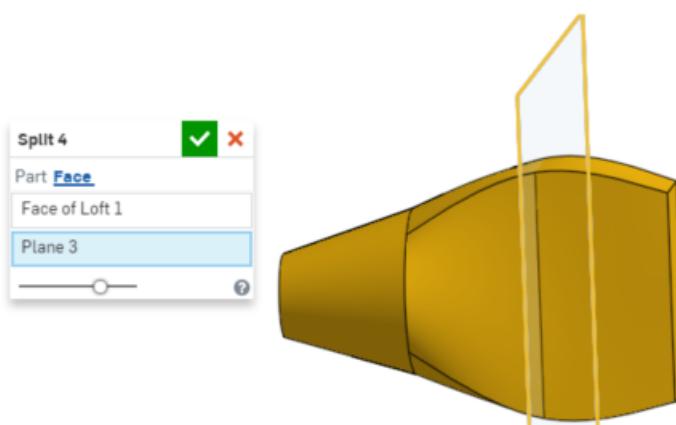
1. Click .



2. Select the face to split:



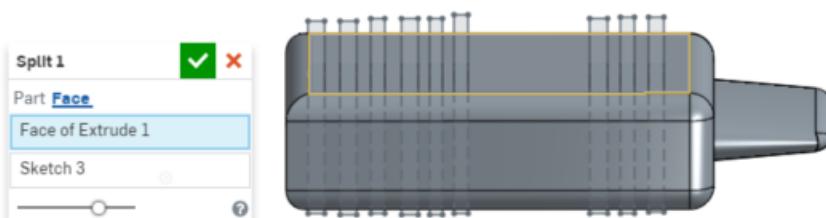
3. Select the split tool, Plane 3:



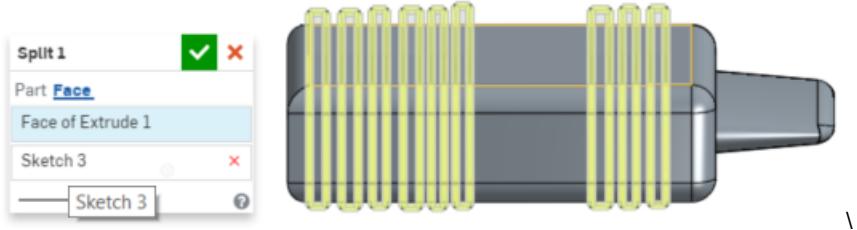
4. Click

## Splitting a face with sketch entities

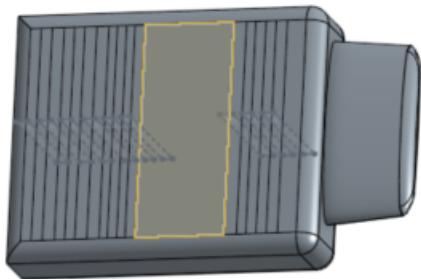
1. Click and select the face to split.



2. Select the sketch entities with which to split the face.



3. Click







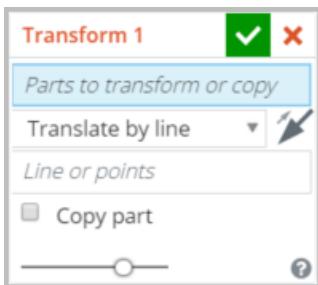
# Transform



Transform adjusts a part's location and orientation in 3D space with the option to copy the part in place.

## Steps

1. Click



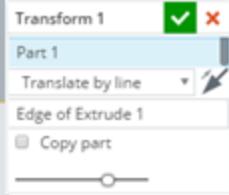
2. Select the method of moving the part (transform type):
  - Translate by line** - Select an entity such as a part edge
  - Translate by distance** - Specify a value and select an entity to indicate direction
  - Translate by XYZ** - Specify axis values to move along or optionally, use the drag manipulator that appears to position the part along axis
  - Transform by mate connectors** - Specify two mate connectors by which to reorient the placement of the part.
  - Rotate** - Move the part about an axis 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 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.

If you need to create multiple copies of a part at once, use the Pattern feature with a distance of 0 (zero).

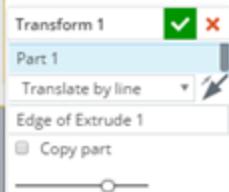
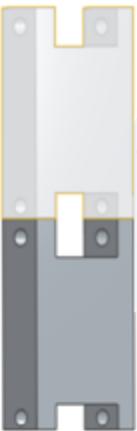
3. Click .

## Translate by line

Before translate by line

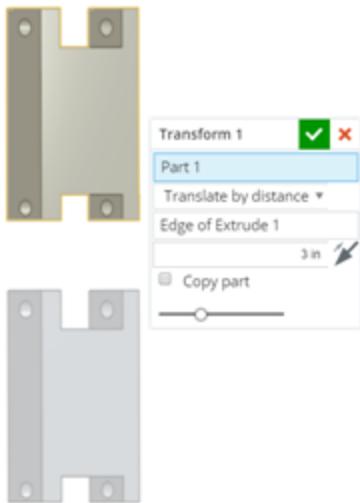


After translate by line

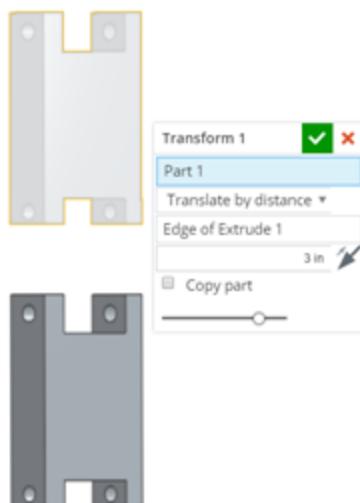


## Translate by distance

Before translate by distance

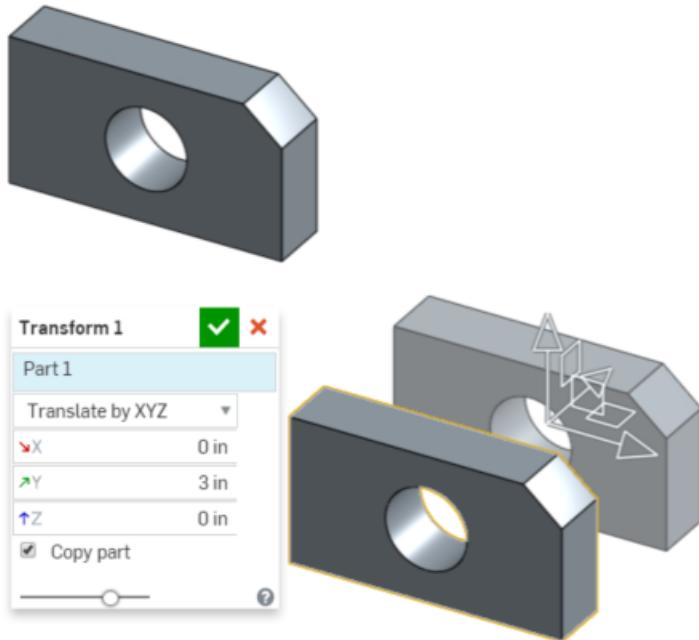


After translate by distance

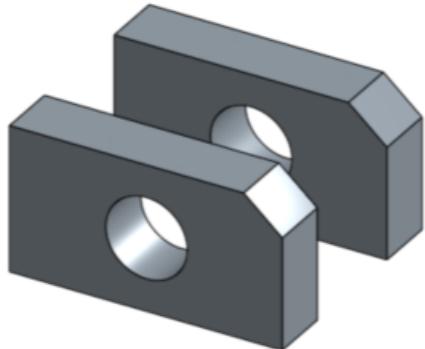


## Translate by XYZ

Before translate by XYZ

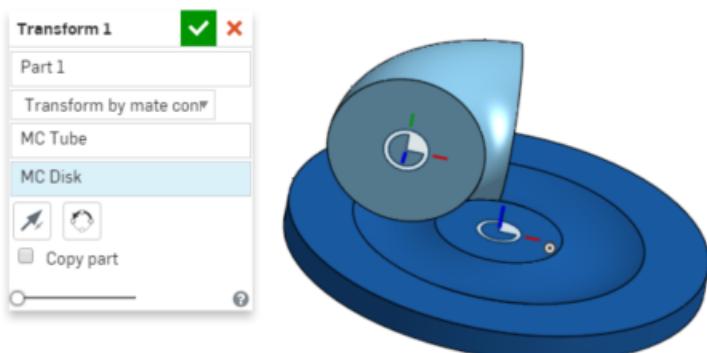


After translate by XYZ with copy part option checked

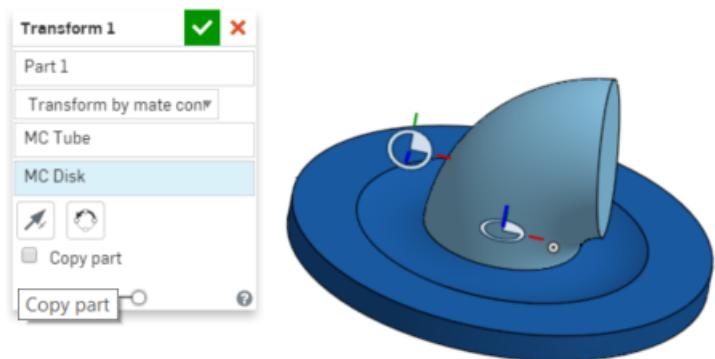


## Transform by mate connectors

Before transform by mate connectors

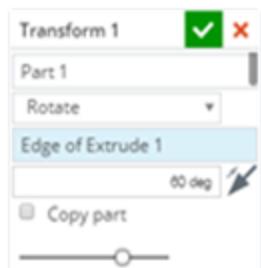
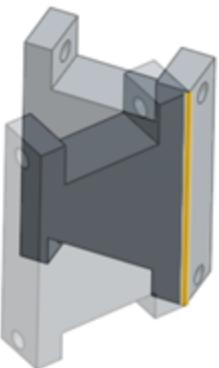


After transform by mate connectors

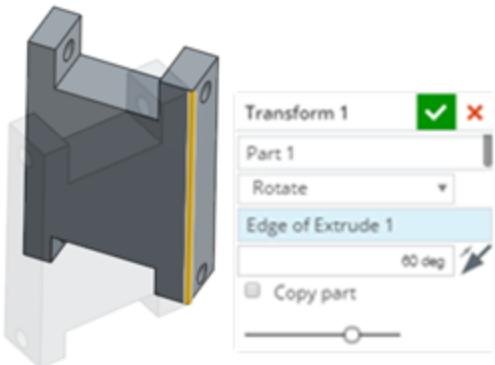


## Rotate

Before rotate

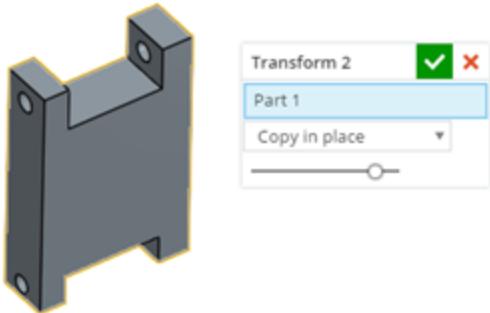


After rotate

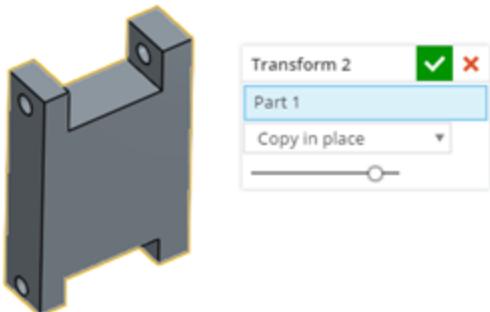


## Copy in place

Before copy in place

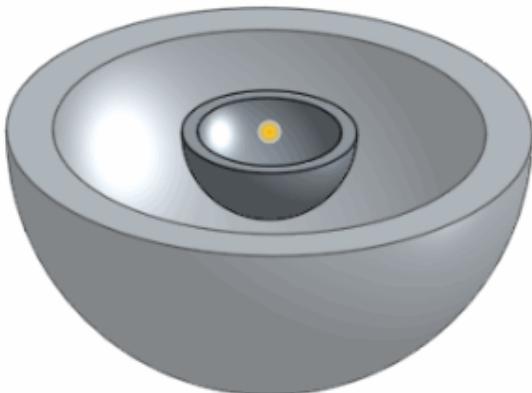
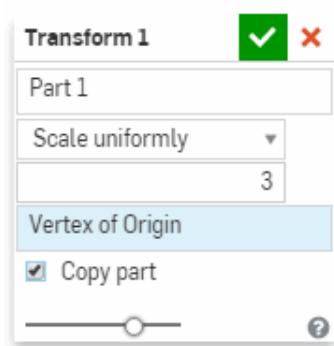
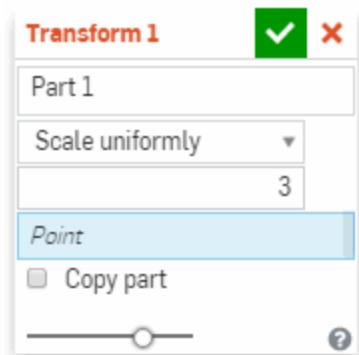


After copy in place

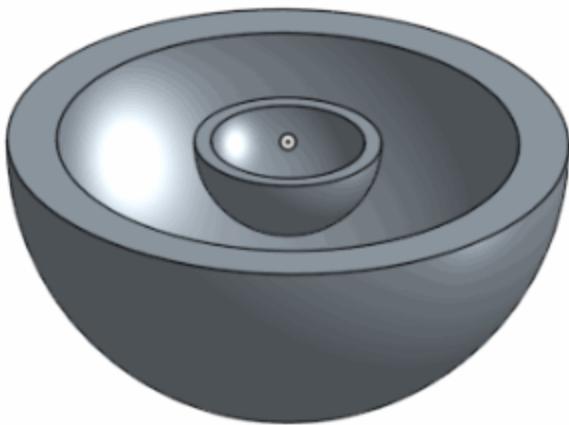


## Scale uniformly

Before scale uniformly



After scale uniformly



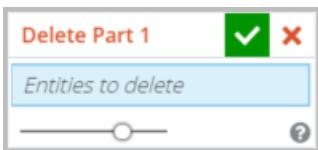
# Delete Part



Delete one or more parts or surfaces; this is a parametric operation that creates a delete-part feature and can be undone.

## Steps

1. Click .



2. Select the part or surface to delete.
3. Click .
4. Notice that the deleted part or surface is no longer listed in the Feature list, and a new Features appears, **Delete Part**.

## Tips

- Delete Part is useful when you want to use a part as a tool body in multiple Boolean features and later discard it.
- You can also select a part in the graphics area and press the Delete button. This action also creates a parametric operation, and can also be undone.
- You can 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 89 for more information.

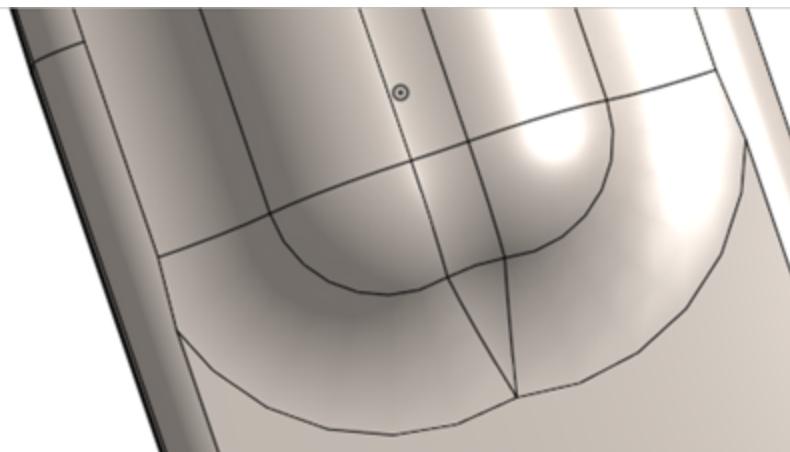
# Modify Fillet



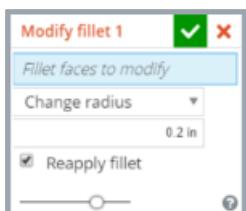
Alter or remove existing fillets or rounds; this Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

## Steps

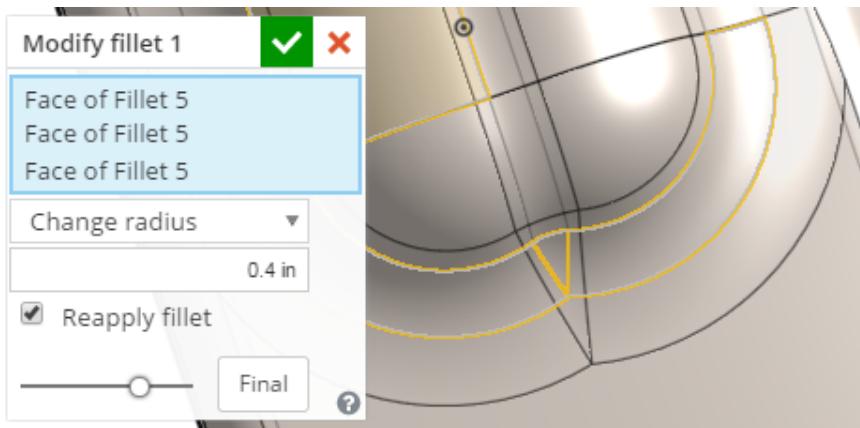
This example uses this area of a part:



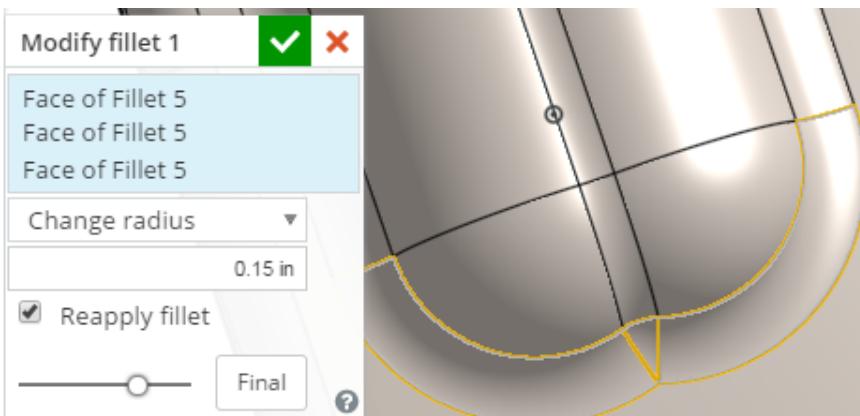
1. Click



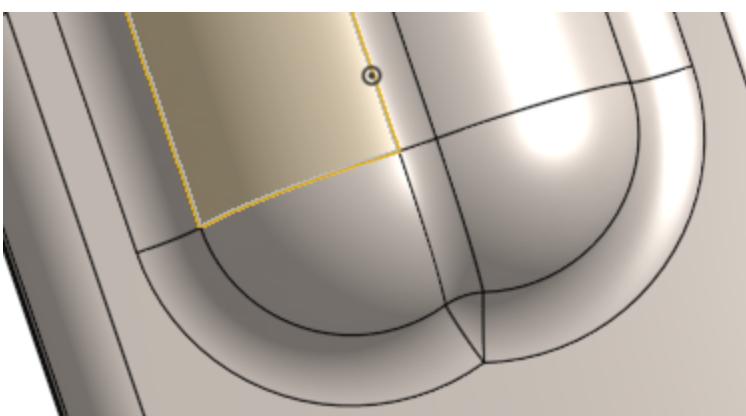
2. Select the fillet faces to change or remove.



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 **✓**.

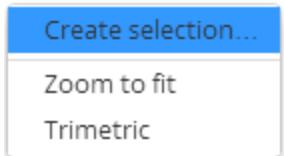


## Tips

- 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 can be difficult to select all necessary faces. You can make it easier by using the "Create Selection" on page 59 option on the context menu:



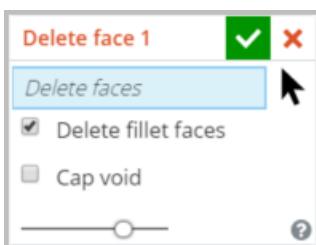
# Delete Face



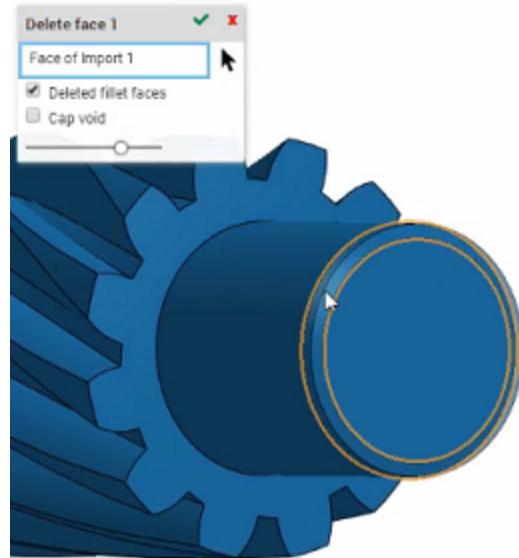
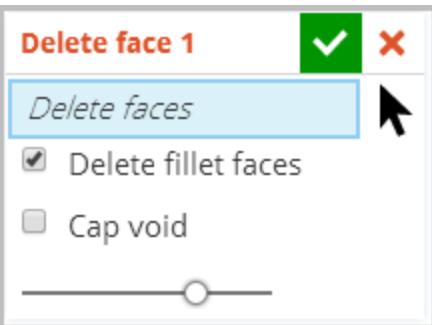
Remove geometry from a part. The surrounding faces will be extended until they intersect in an effort to make the part a valid solid. (If the surrounding faces do not intersect, the action fails.) This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

## Steps

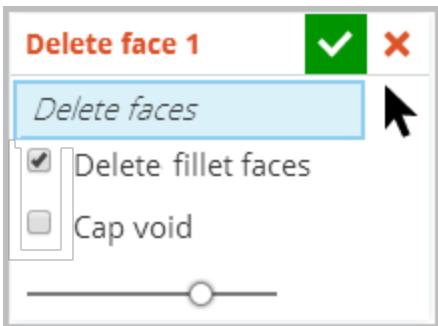
1. Click .



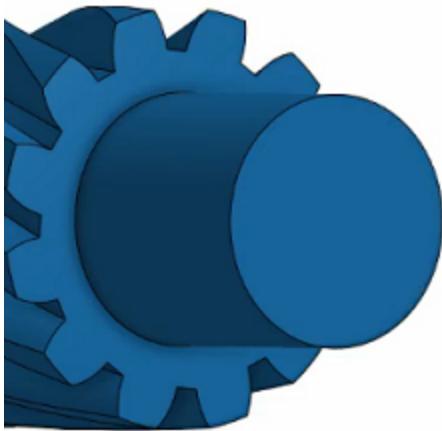
2. On the model, select the part face or faces to delete.



3. Check **Delete fillet faces** to indicate whether or not to delete the adjacent filleted faces as well. Check **Cap void** to put a face across the space; leave it unchecked to extend surrounding faces until they intersect in an effort to create a valid solid.



4. Click .



## Tips

The "Create Selection" on page 59 arrow  (next to the *Faces* field) can be useful to select related faces for Delete face.

# Move Face



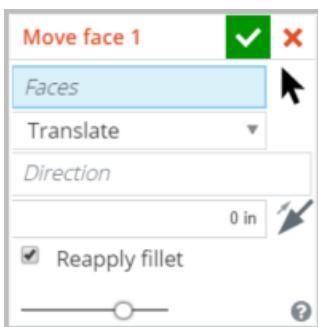
Translate, rotate, or offset one or more selected faces. This Direct Editing tool is especially convenient if you don't have the parametric history of the part, as is often the case with an imported part.

## Steps

The following examples start with this part:



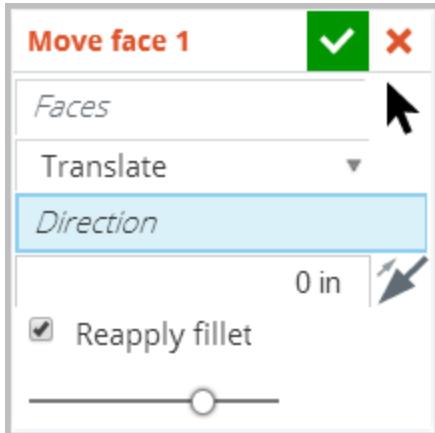
1. Click .



2. Select the type of move:

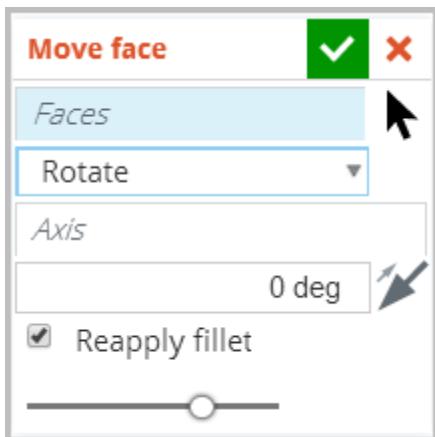
- **Translate** - Move one or more faces in a specified direction for a specified distance.
  - Select any face or combination of faces (Faces field).
  - Select an edge to define a vector (Direction field).
  - Click and drag the arrow graphically on the model, or type in a value (Numeric field).

Use the directions arrows to change the direction of the translate, if necessary.

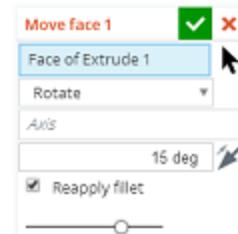


- **Rotate** - Rotate one or more faces a specified number of degrees.
  - Select any face or combination of faces (*Faces* field).
  - Select the axis to rotate from (*Axis* field).
  - Specify the number of degrees to rotate.

Use the direction arrows to change the direction of the rotation, if necessary.



**Rotate Face**



**Rotate Axis (during)**

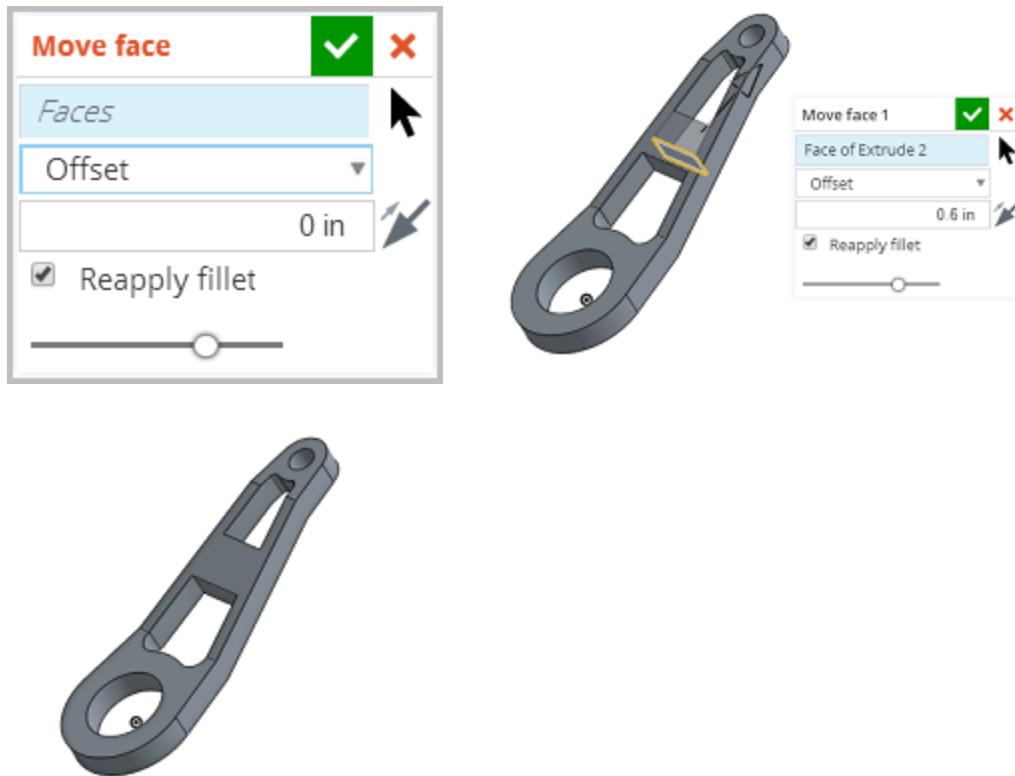


**Rotate Axis (after)**



- **Offset** - Typically used with non-planar faces to increase or decrease a radius, for example.

- Select any face or combination of faces (*Faces* field).
- Specify the value of the offset (*Numeric* field).
- Use the direction arrows to change the direction of the offset, if necessary.



3. Click

## Tips

The "Create Selection" on page 59 (next to the *Faces* field) can be useful to select related faces for Move face.

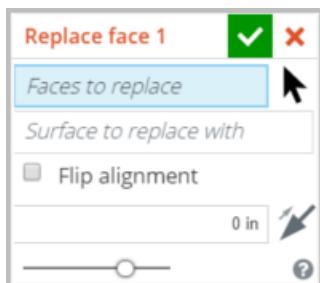
# Replace Face



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.

## Steps

1. Click

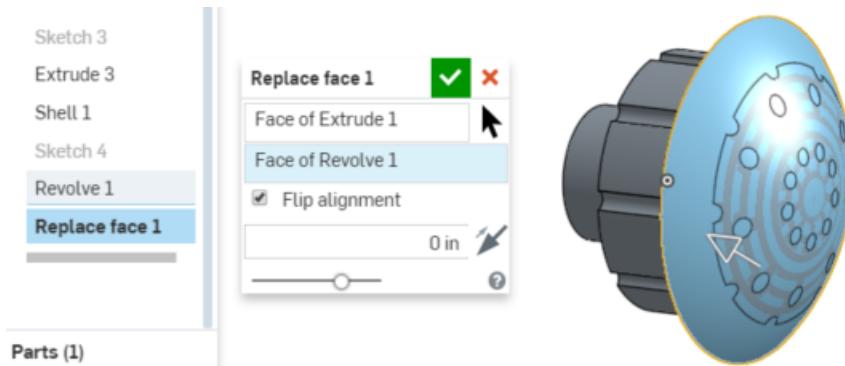


2. On the model, select the face you want to trim or extend.

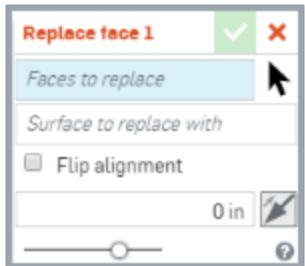


3. Select the surface to use for the replacement.

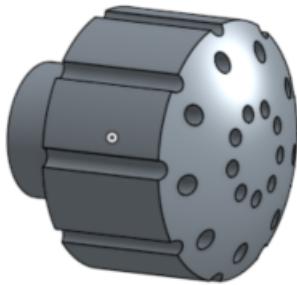
Notice that a surface has been created in this example, to use to extend the face to. You can hide and unhide this part in the Parts list of the Feature list box:



4. Optionally provide an offset distance, or flip the alignment.



5. Click



## Tips

The "Create Selection" on page 59 arrow (next to the *Faces* field) can be useful to select related faces for Replace face.

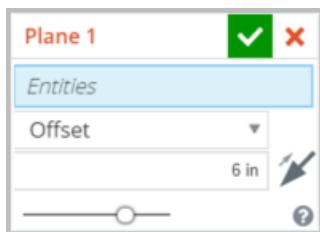
# Plane



Create a new construction plane.

## Steps

1. Click



2. Select an entity on which to base the new plane.
3. Make further specifications where necessary (see below).
4. Click

You can create a plane based on the relative position to another entity, including:

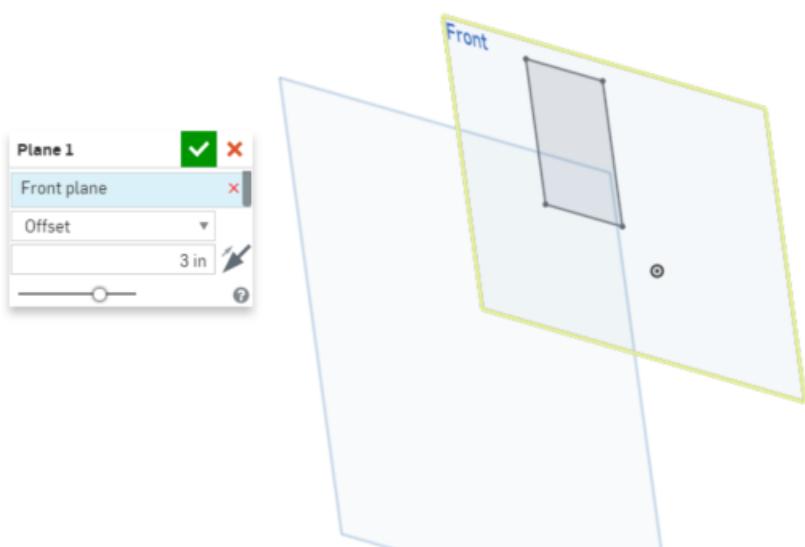
- **Plane** - Select another plane or planar face
- **Point** - Select a vertex, sketch point, or the origin
- **Line** - Select a linear edge, sketch line, or cylindrical face to get its axis

Note that pre-selecting a planar face (solid or plane) and creating a plane defaults to Point normal plane.

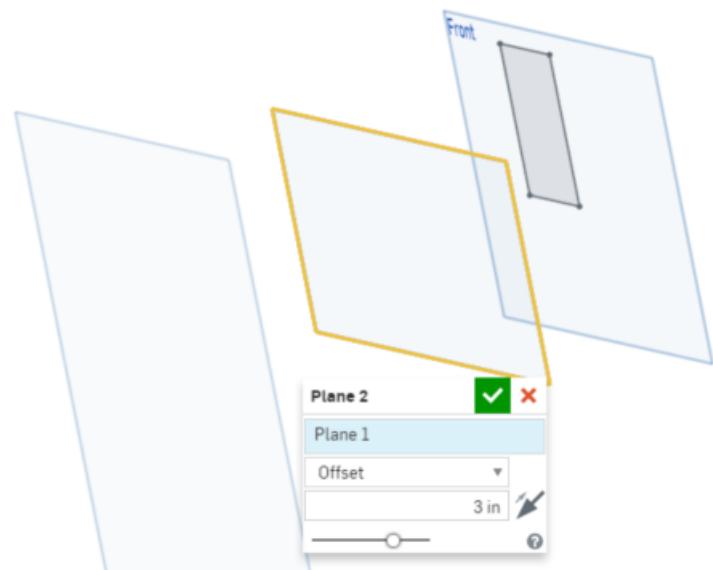
## Create offset plane

Create a plane a specified distance from another plane using a plane and a distance value.

Offset from a planar face:

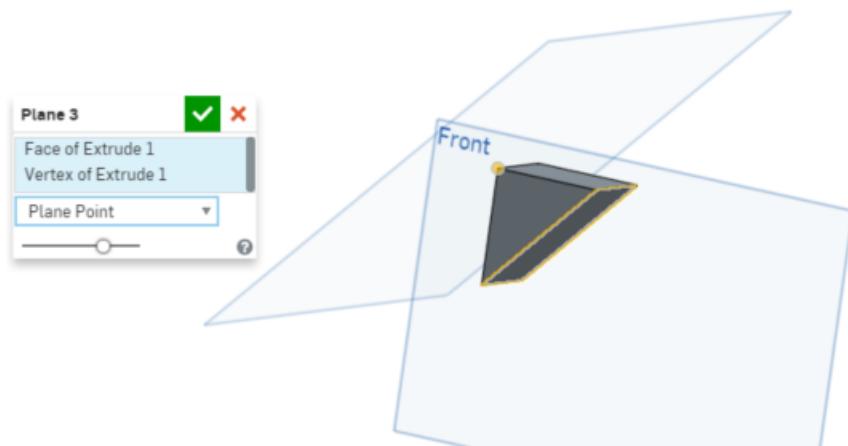


Offset from another plane:



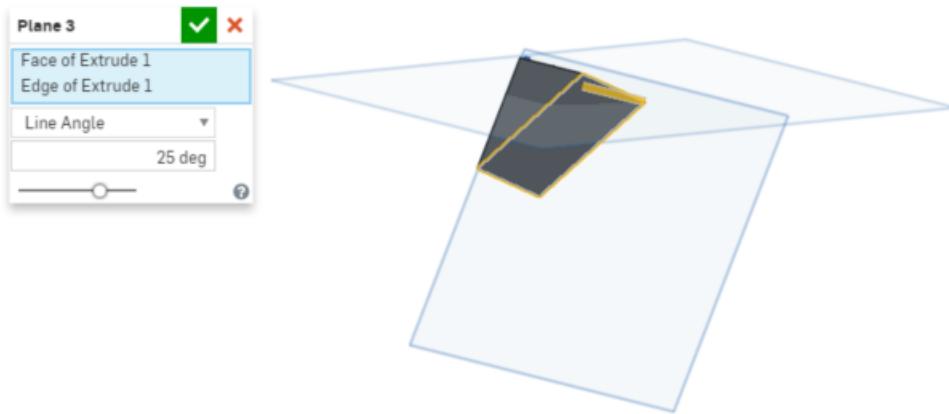
## Create plane point plane

Create a plane that passes through a point, parallel to a plane, using a plane and a point.



## Create line angle plane

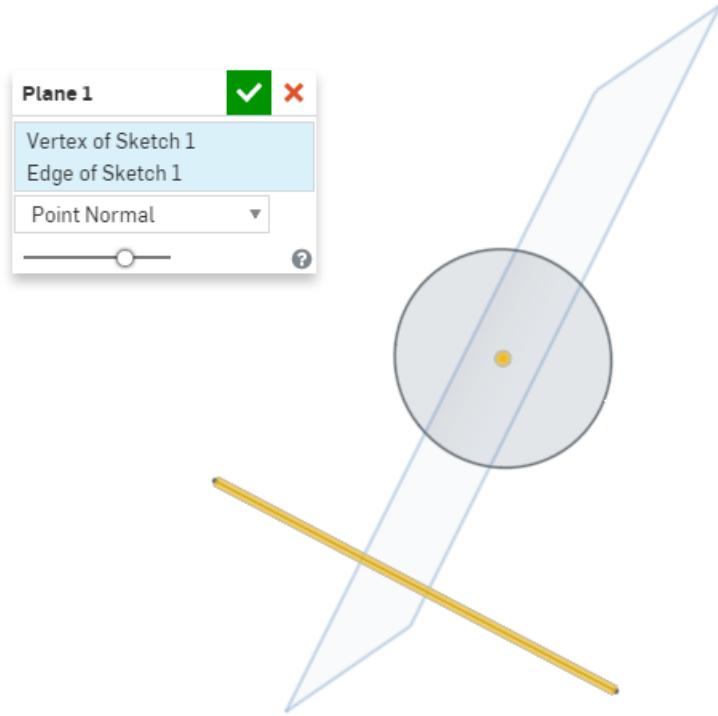
Create a plane that passes through a line at an angle, using a line, reference geometry (such as a plane, point, or axis) and an angle value.



## Create point normal plane

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 (or vertex). The point is always the origin of the plane and the axis or line is always the normal of the plane.

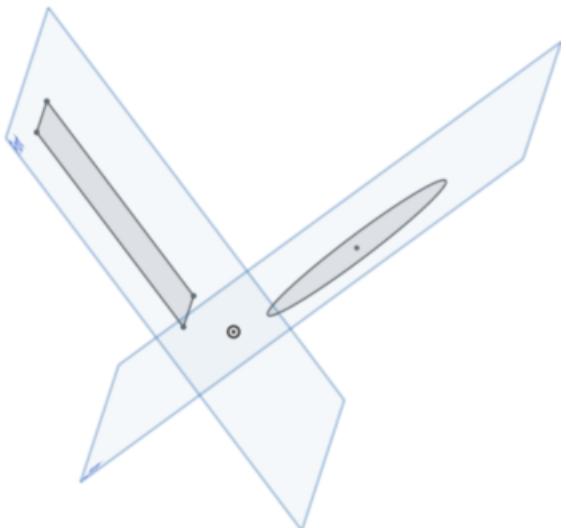
Note that pre-selecting a planar face (solid or plane) and creating a plane defaults to Point normal plane.



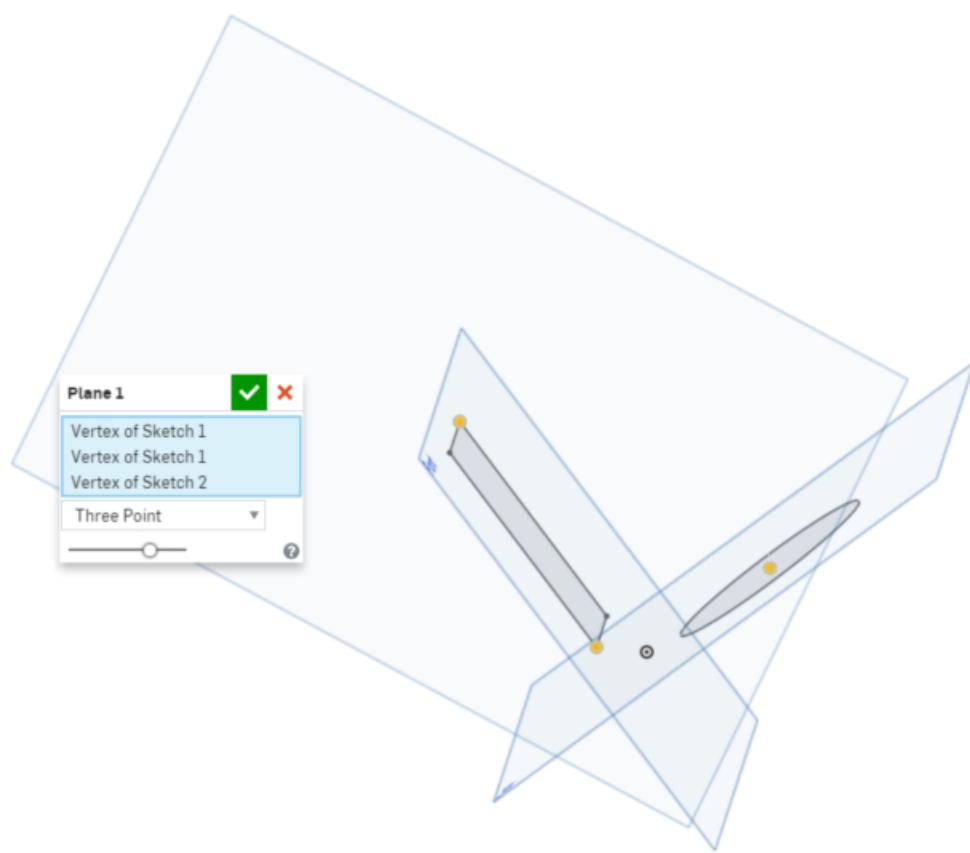
## Create three point plane

Create a plane that passes through three points, using three points.

The starting sketches, on two planes:

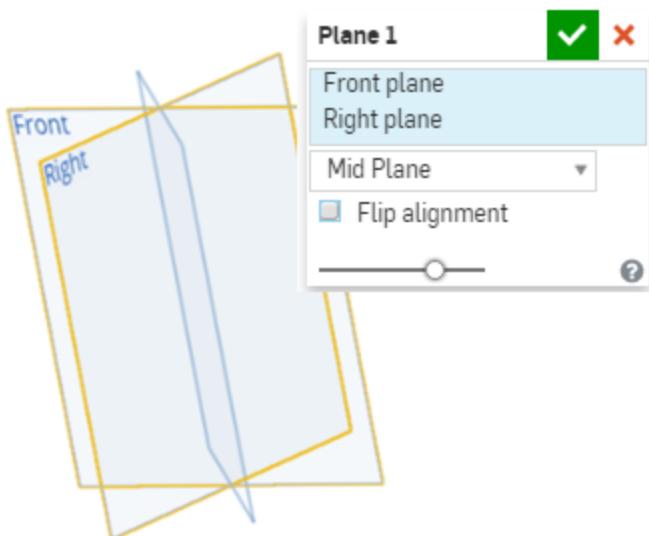


The resulting third plane:

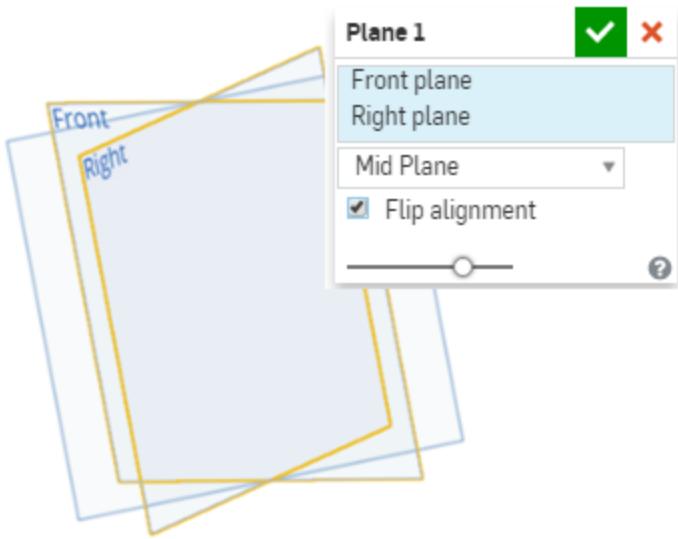


## Create mid plane

Create a plane at the intersection of two other planes

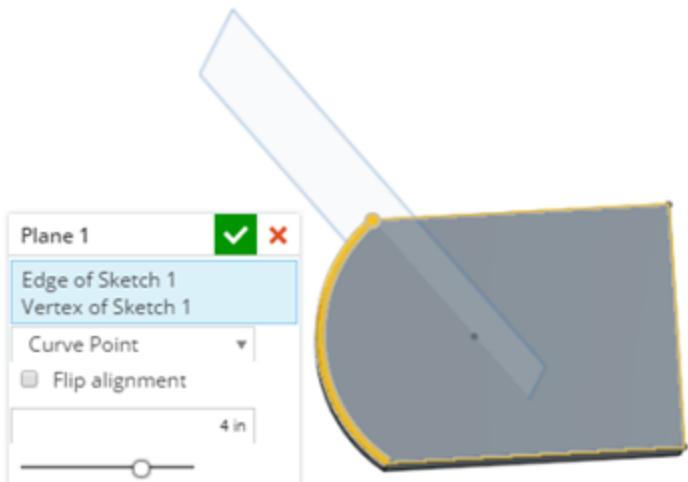


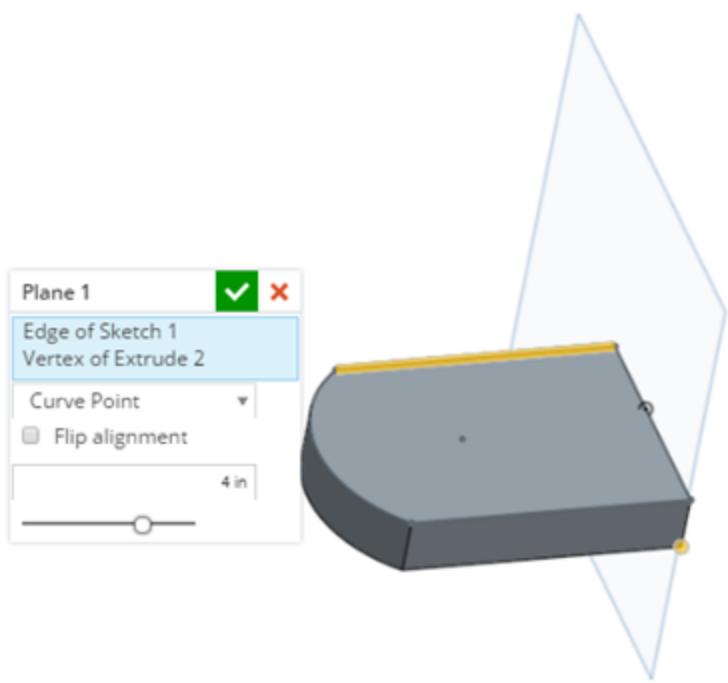
Flip alignment:



## Create a curve point plane

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 (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.

# Helix

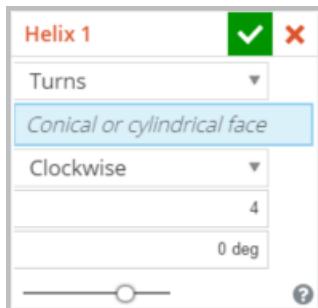


Create a helix using a conical or cylindrical face.

A helix can be used for sweeps (to create a simple spring). A helix doesn't consume the part used to create it.

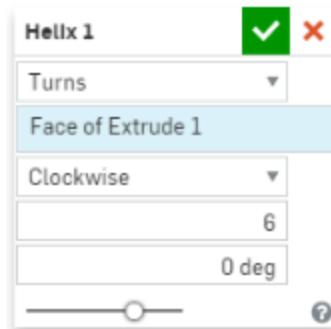
## Steps

1. With a cone or cylinder in the graphics area, click



If you don't see the Helix icon, expand the Plane/Mate connector icon group: or .

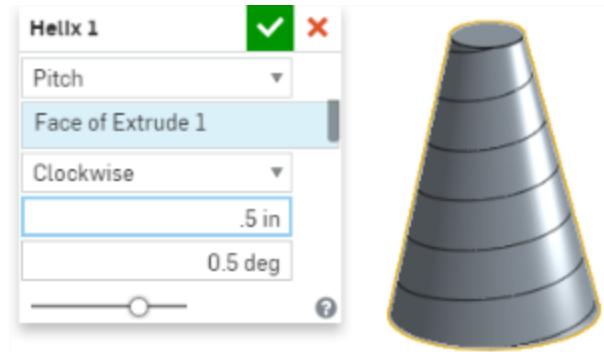
2. In the dialog, choose whether to create the helix based on number of turns (revolutions) or pitch.
3. In the graphics area, select the base entity of the helix: either a conical or a cylindrical face.
4. Select the direction of the revolutions: Clockwise or Counterclockwise.
5. Specify the start angle.



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.

a. For Pitch:

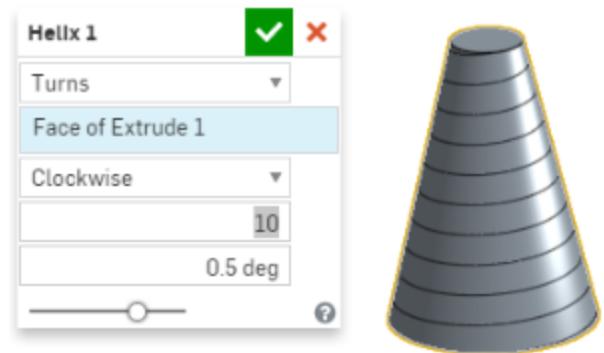
Specify the helical pitch in your document's unit of measure:



Pitch: the distance traveled axially in each revolution

b. For Turns:

Specify the number of revolutions:

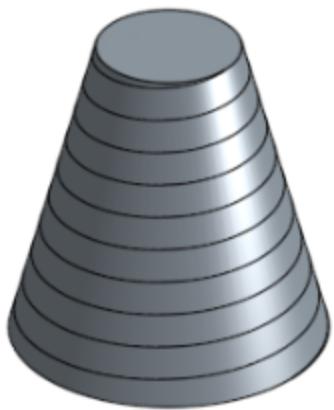


6. Click

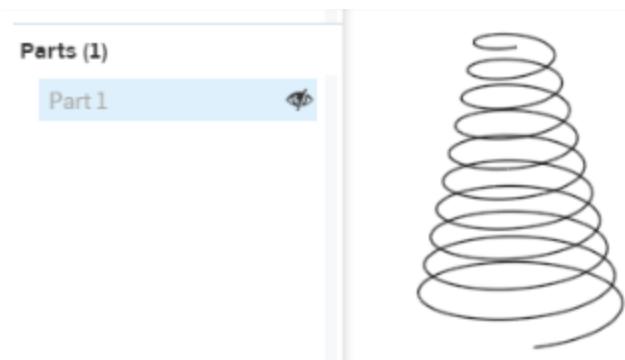
## Examples

### Creating a spring

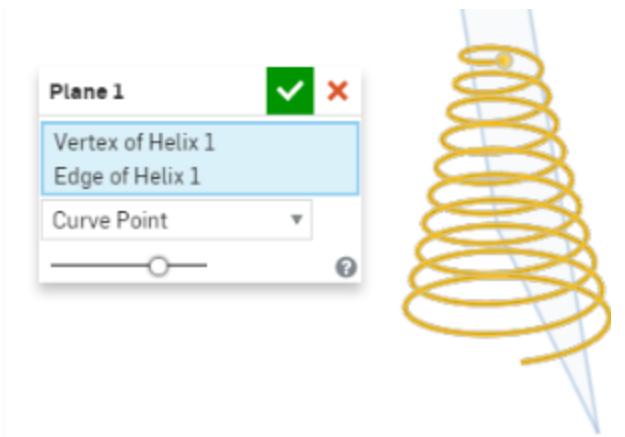
1. Create a helix as described above.



If it helps, you can hide the cone or cylinder (use the in the Parts list).

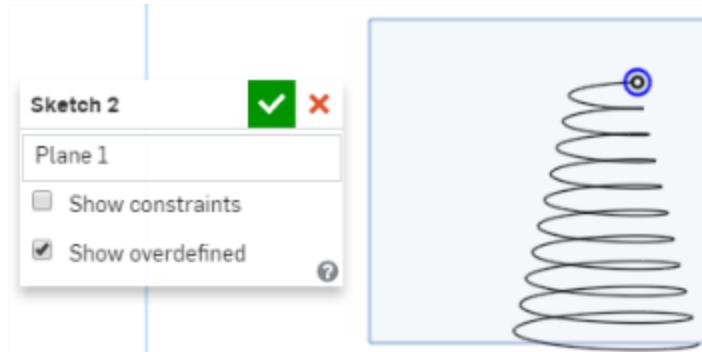


2. Create a curve point plane using the helix and the vertex of the helix:

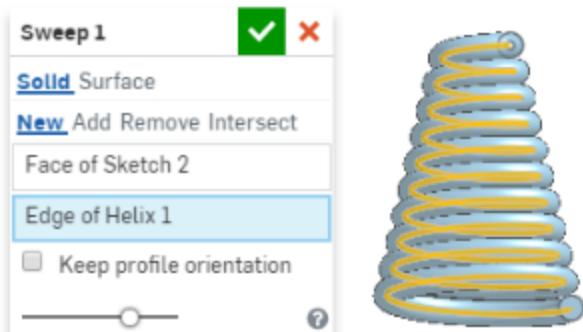


The plane in the image above is a new plane, intersecting the helix vertex and normal to the edge of the helix.

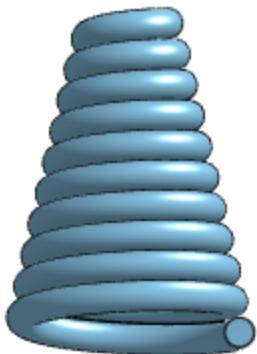
3. Create a circle on the plane, using (  ) the helix vertex for the center of the circle:



4. Sweep the circle along the helix (path):

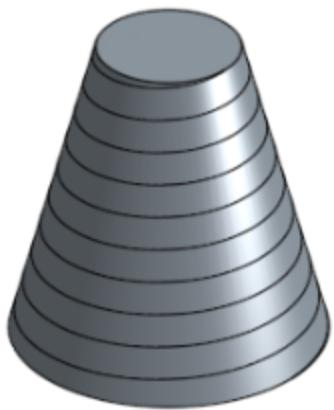


5. Click .

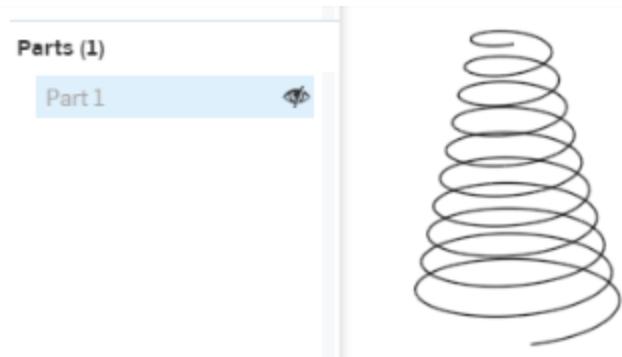


## Creating a plane point plane

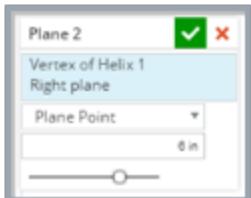
1. Create the helix as described above.



If it helps, you can hide the cone or cylinder (use the in the Parts list).



2. Create a plane point plane using the helix vertex and a plane:



3. Click .

# Mate Connector



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 and used within a mate to locate and orient part instances with respect to each other.

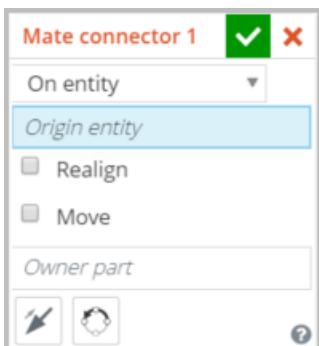
Two part instances are positioned in an assembly by creating a **Mate**. The two instances are positioned by aligning a **Mate connector** defined on one instance with a Mate connector defined on the other instance.

To learn more about Mates, see "Mates" on page 320. To learn about Mates and Mate Connectors watch the video below.

Use the shortcut key **k** to hide/show mate connectors in an assembly.

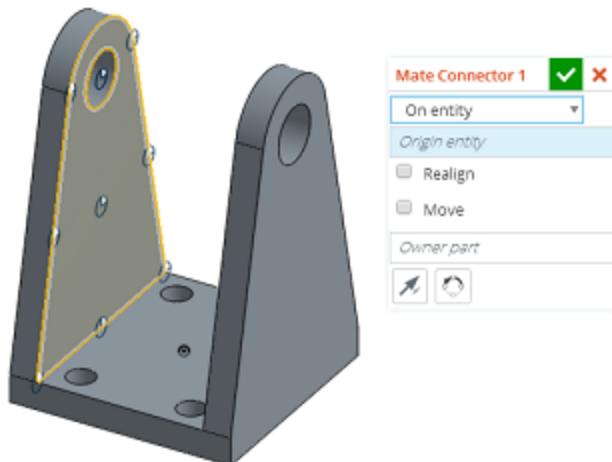
## Steps

1. Click

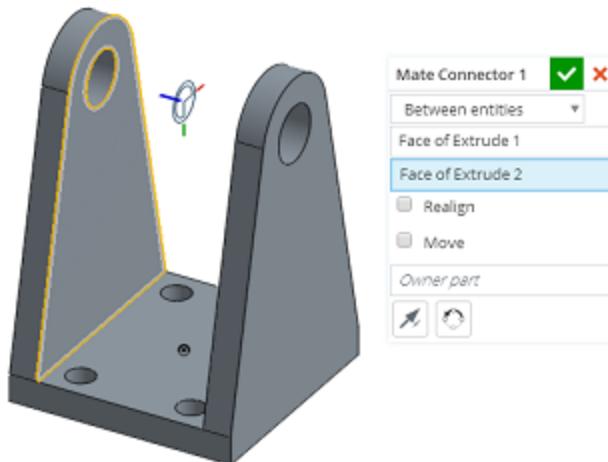


2. Choose between creating a mate connector **on** a part (entity) or **between** parts:

- **On entity** - Create a Mate connector on a part:



- **Between entities** - Create a Mate connector halfway between two entities on the part:



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 .

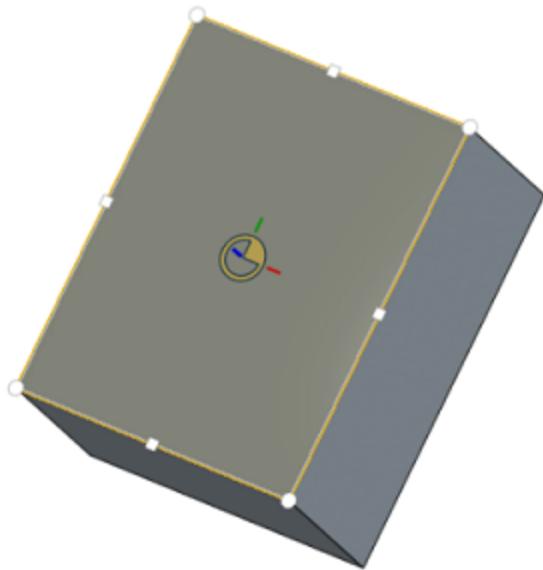
## Visualizing Mate connector points

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.

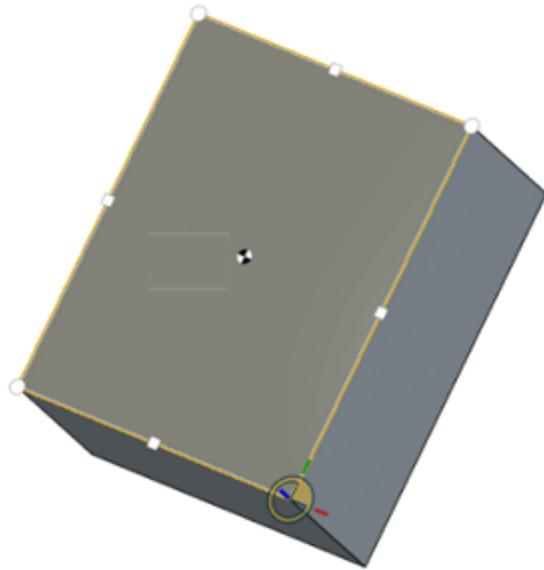
To lock mate inferences when you see the one you want to select, depress the Shift key when mousing.

Each face and edge of a part has default inference points:

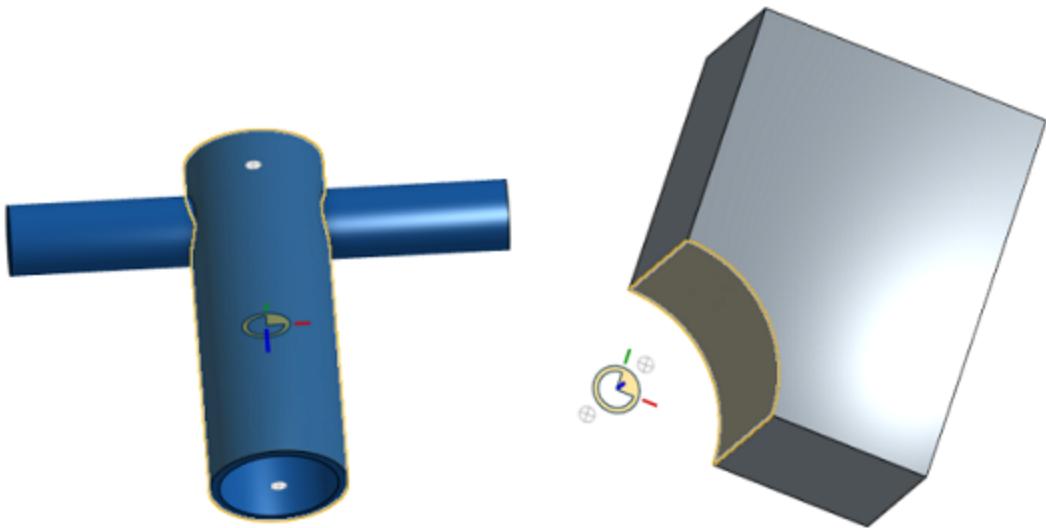
- At the centroid
- At the midpoints
- At the corners



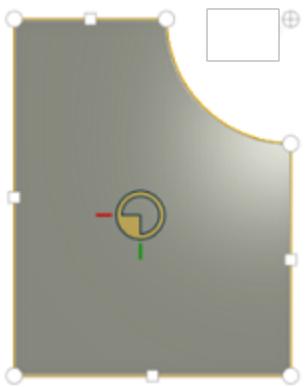
Before the default Mate connector is highlighted at the centroid (seen above), you might see the centroid point icon (seen below):



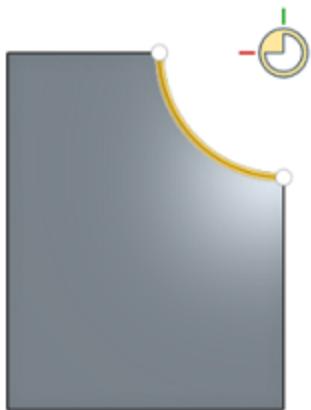
For cylindrical faces, inference points appear on the axis of the **cylindrical** and **partial cylindrical** face:



Select a planar face that has a partial cylindrical edge and the Mate connector inference points include the centroid of the axis:



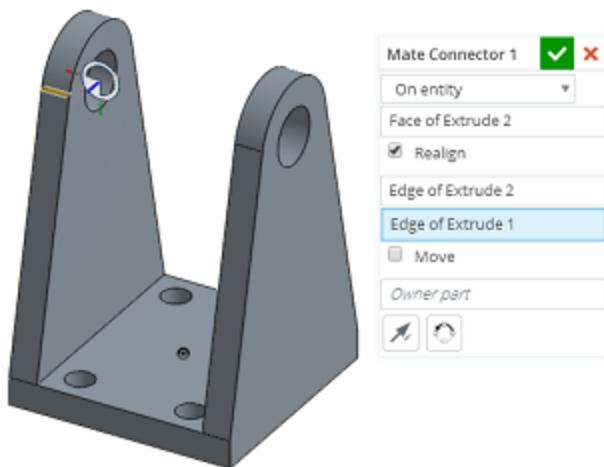
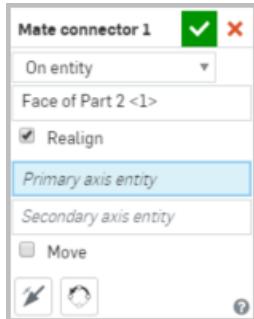
Hover over the edge of the partial cylindrical face and the default Mate connector appears at the centroid of the axis:



To zero in on a specific inferred point or default mate connector without waking up others as you move the cursor, you can use the SHIFT key to prevent other Mate connectors from appearing.

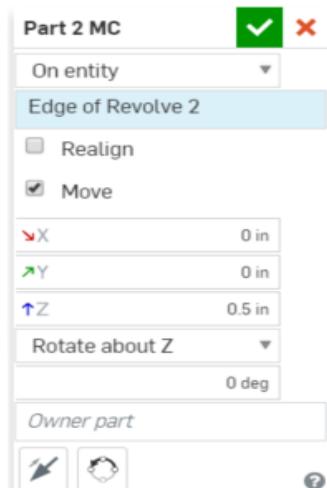
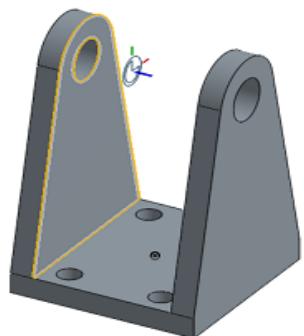
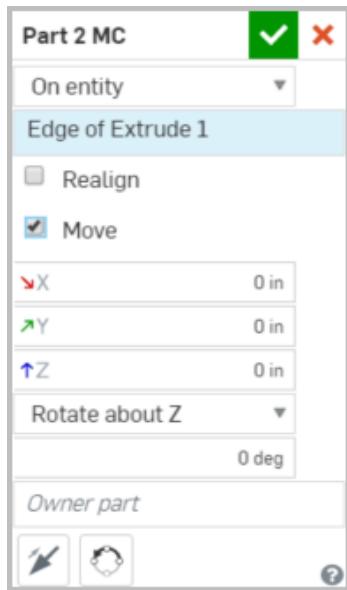
## Realign Mate connectors

Change the orientation of the Mate connector along a primary and (optionally) a secondary axis.



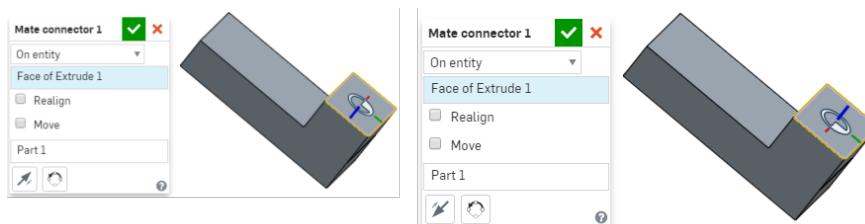
## Move Mate connectors

- **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 can also use the Rotate field to specify a rotation of a specified number of degrees.



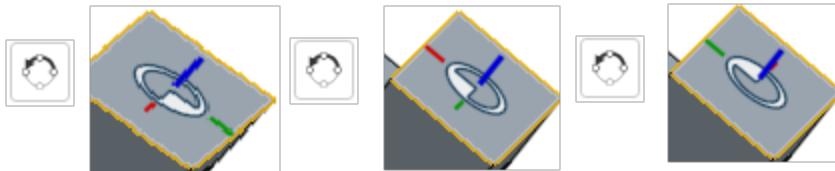
## Flip primary axis of Mate connector

Flip the primary axis 180 degrees.



Reorient secondary axis of Mate connector

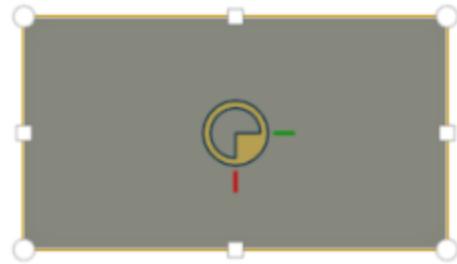
Move the primary axis one quadrant at a time through the X/Y coordinates.



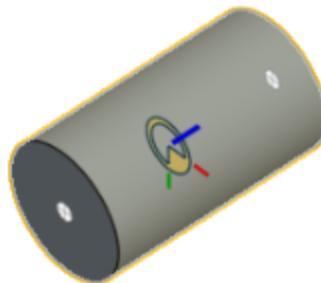
## Inference points and defaults

The inference points for potential Mate connectors available when you select an edge or face are:

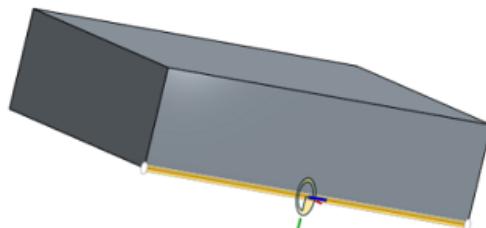
**Planar face** - Parallel to the face at every vertex, arc center, edge midpoint, and the face centroid



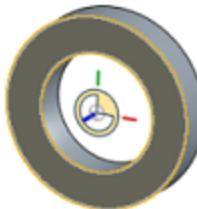
**Cylindrical face** - Perpendicular to the face axis at the middle and ends



**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



## Hiding and showing Mate connectors

Once created, you can 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 can view mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

## Tips

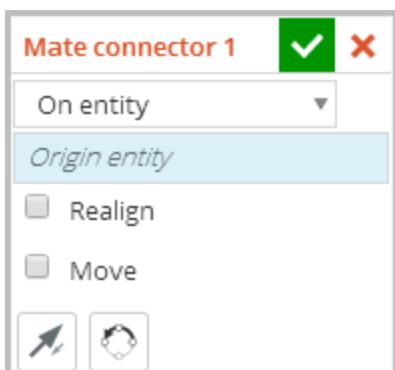
- If the behavior is not what you expected, try flipping the primary and/or secondary axis on the Mate connector.
- Use the SHIFT key to keep the mate connectors you want visible as you move the pointer to select one. This can be useful when the inferred point for potential Mate connector you want is 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 can be created in both the Assembly and the Part Studio. Creating a Mate connector in the Part Studio has two advantages:

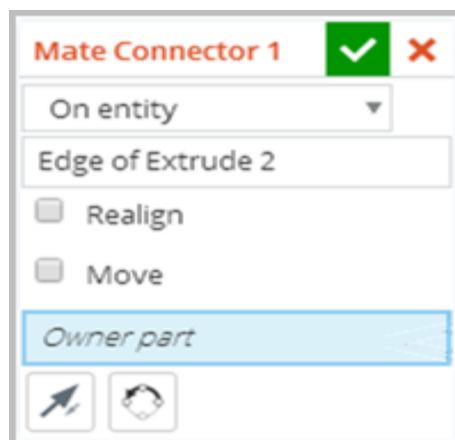
- You can reference sketch entities in the Part Studio. This gives you the ability to define the Mate connector in more positions than are possible in an Assembly.
- A Mate connector defined in a Part Studio is available for reuse on every instance of that part in every assembly in which it is instanced.

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 can be unclear which part owns the Mate connector. Use **Owner Part** to specify which part owns the Mate connector.



# Derived

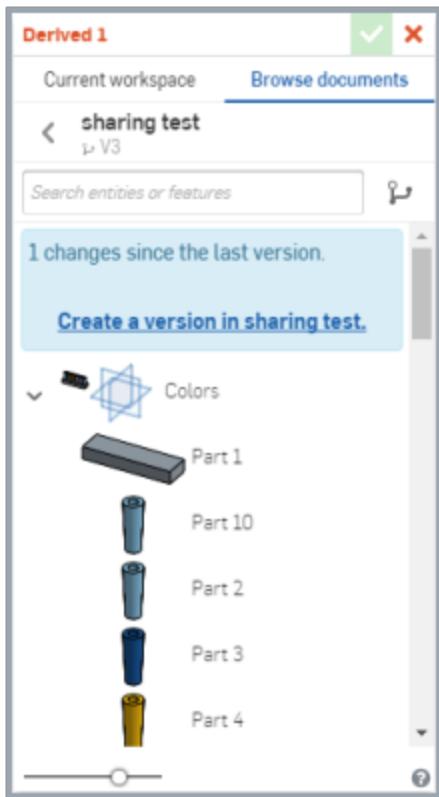


Insert parts, sketches, surfaces, helices, planes, or mate connectors from one Part Studio into another in the same or a different document (thereby [linking the documents](#)), with an associative link.

## 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. Optionally, click 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:

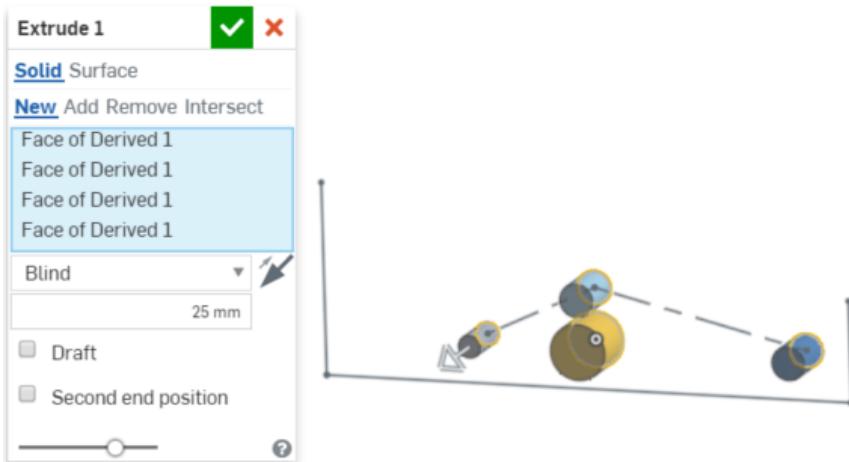


You can select from other documents only if that document has one or more versions. A notice is displayed regarding the state of the document: if no version exists or if a newer version exists. See [Linking Documents](#) for more information.

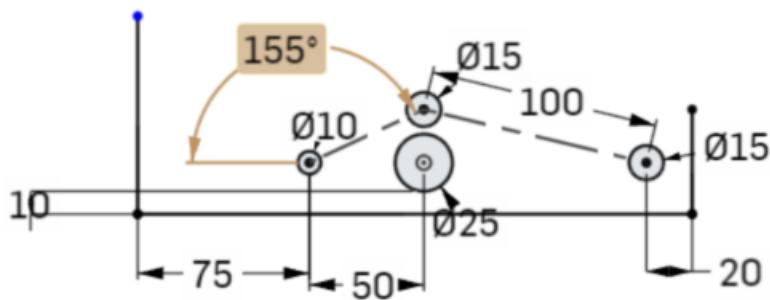
3. Click .

## Example

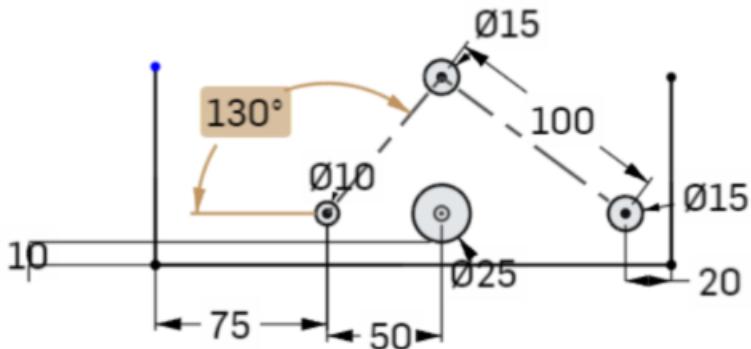
- After selecting a sketch you can use that sketch to perform an extrude in the target Part Studio.



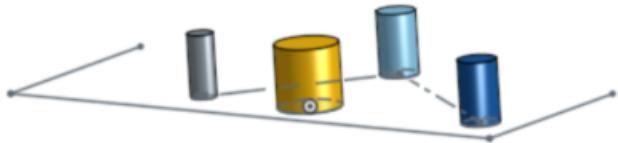
- In the parent Part Studio, when you make a change to the sketch, from this:



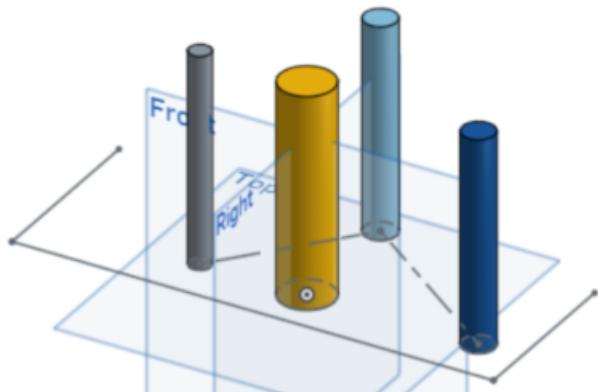
to this:



- The change is reflected in the target Part Studio. Since the circles weren't selected in the original extrude operation, they appear as holes in the part:



4. You can use the sketch in many Part Studios as a derived feature. Then in each Part Studio, continue with varied designs:



## Tips

- You can 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 cannot select a derived feature for insertion more than once in the same operation. You can 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 cannot insert a feature from Part Studio A to Part Studio B and then to Part Studio A again, the operation will fail.

# (x) Variable



Create a variable for use in expressions in a Part Studio, and assign a value. Use the variable in dimensions and expressions.

## Steps

1. While in a Part Studio, click **(x)**.



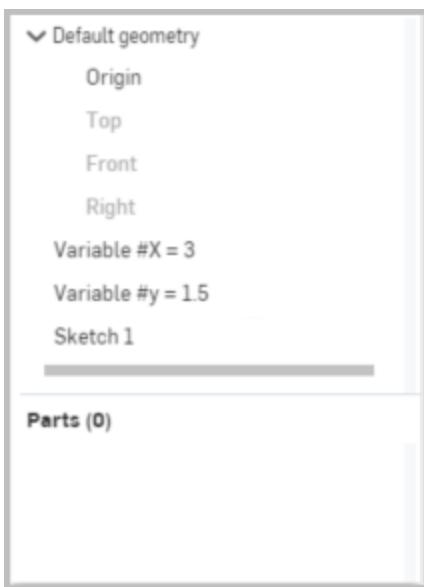
2. In the dialog, select individual features:

Enter a name (usually a single letter), and a numeric value (decimals accepted):



The value entered in the Name field becomes part of the name (in the title of the dialog and in the Feature list).

3. Click :

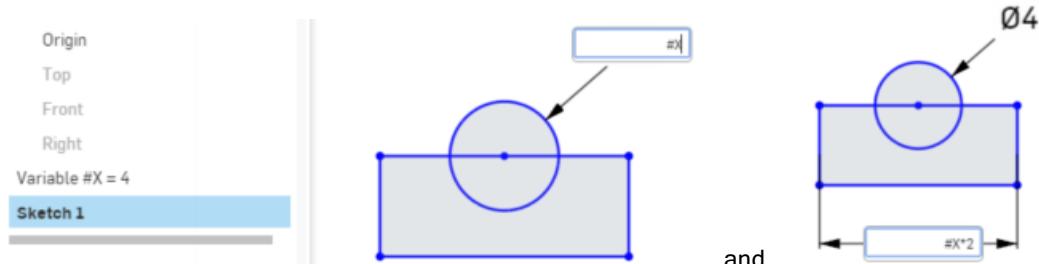


Note that the variable name is case-sensitive.

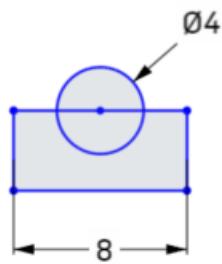
## Examples

### Using the variable in a dimension

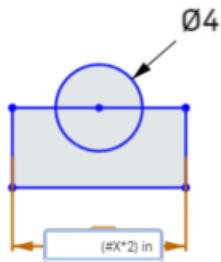
1. Begin creating a dimension as usual.
2. In the dimension field, enter **#** and the **variable name** (and optionally, as part of an [expression](#) (as shown below):



3. When you save the dimension, the variable is replaced with the value and the expression (if applicable) is solved.



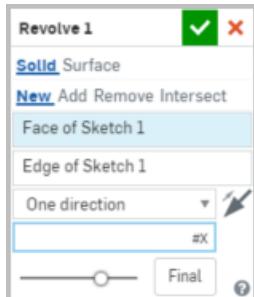
- When you double-click the dimension for editing, the variable (and expression) is displayed:



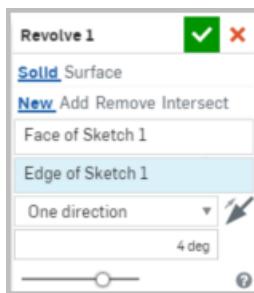
## Using the variable in a solid body (revolved) feature

You can use variables anywhere you can use expressions in a Part Studio. For example, in an extrude or revolve operation:

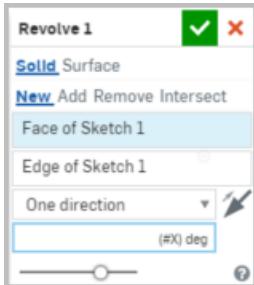
- Start the operation as usual (in this case, Revolve).
- For less than full revolves, in the numeric value field, enter # and the **variable name** (or optionally, as part of an expression):



- Accept the feature.
- When you edit the feature, the solution is displayed in the numeric value field:



5. Click in the field and the variable (and expression, if applicable) is displayed:



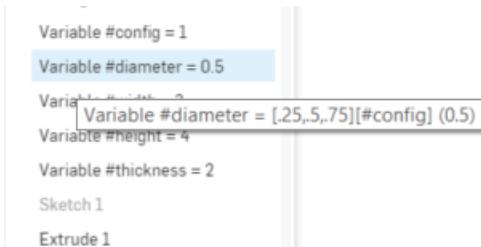
## Using arrays in variables

Variable values can contain expressions but must evaluate to a scalar value. You can 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.

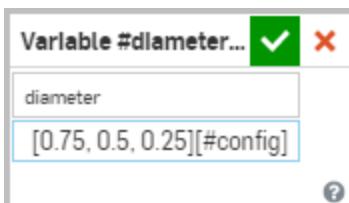
1. Create a variable to hold the array and (zero-based) index:



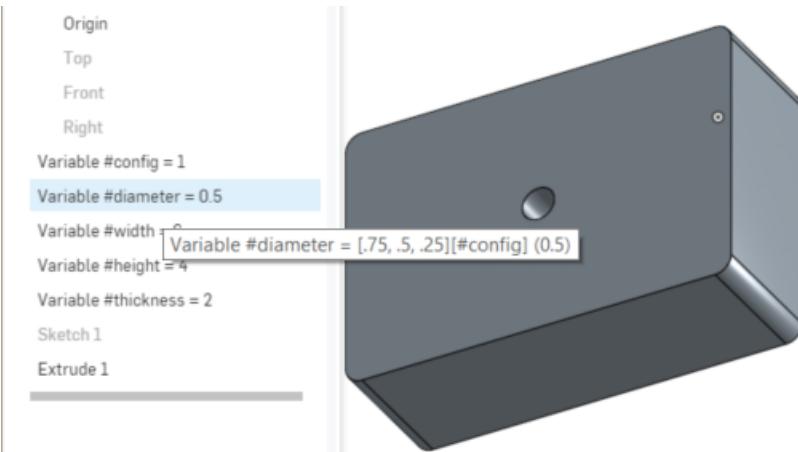
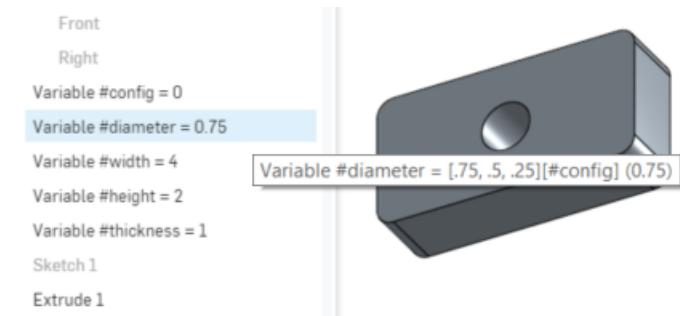
2. Create variables to hold length, width, height, and thickness arrays.



Hover over any variable to see the definition and the current value (in parenthesis). For example, a diameter variable array may be:



3. Change the value of the index variable to change the indices of all array variables:



## 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.

# Custom Feature



Custom features are written in a programming language called FeatureScript and are created in an Onshape tab called a [Feature Studio](#). These custom features can be added to your Feature toolbar for use in documents to which you have write access.

For an overview of how to create custom features, see the video "[How to Create an Onshape Feature in 90 Seconds.](#)"

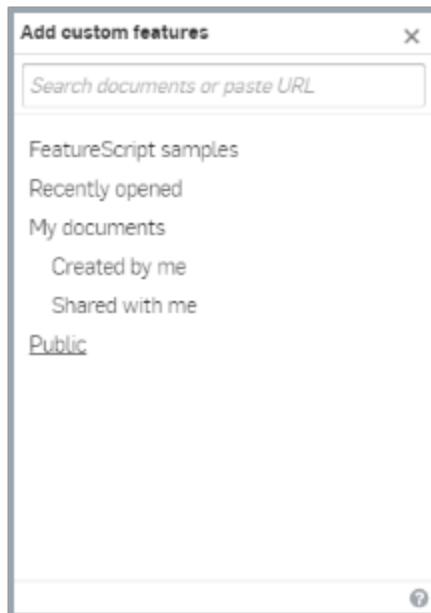
You can:

## Add 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 can add custom features when you are in a Part Studio to which you have write access or when you are viewing a version that contains the Feature Studio that defines the features.

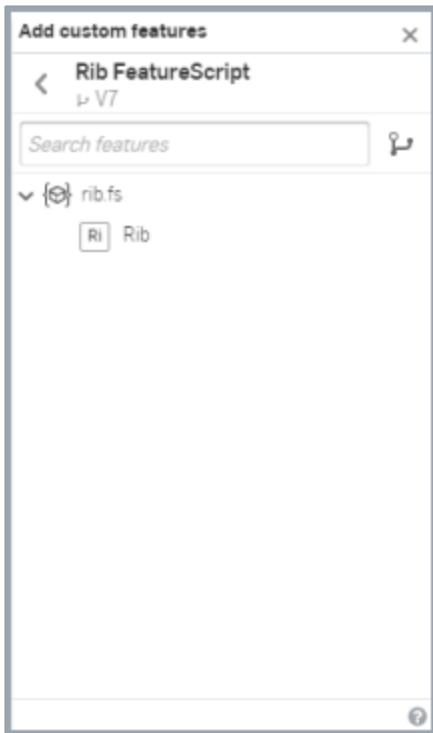
To add a custom feature from another document to your Feature toolbar:

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 can 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 (the Feature Studio, in this case, rib.fs) to insert all custom features inside it (each represented by its own icon on your toolbar), or select one feature (in this case, Rib).

The selected custom feature's icon appears on the Feature toolbar (seen below as the Ri icon to the left of the Custom feature icon):



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. You can 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.

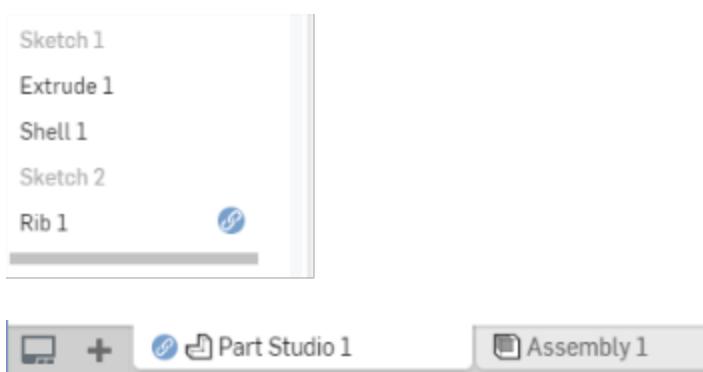
### Use a custom feature

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 indicated in the Feature list by a link icon . Custom features behave like other Onshape features; they can be edited, suppressed, hidden, and deleted.

### Update a custom feature

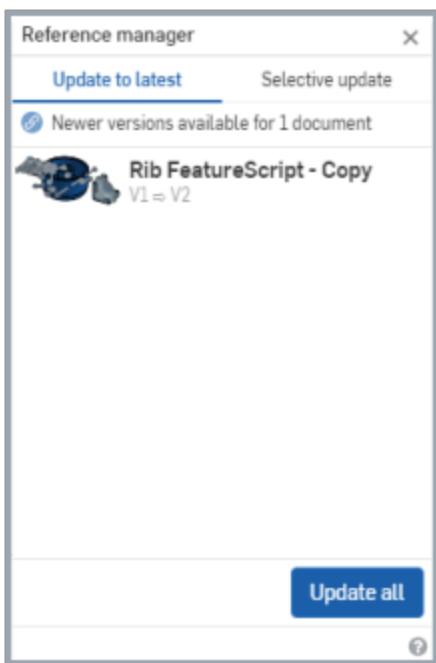
When a newer version of the document from which you inserted the custom feature is created, the link icon in the Feature list highlights in blue, and an identical icon appears on the Part Studio tab.

This is a notification only and no action is required.

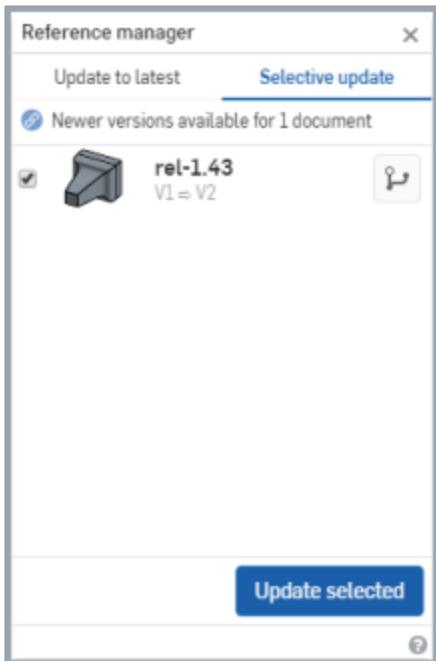


To update the version of the custom feature being used:

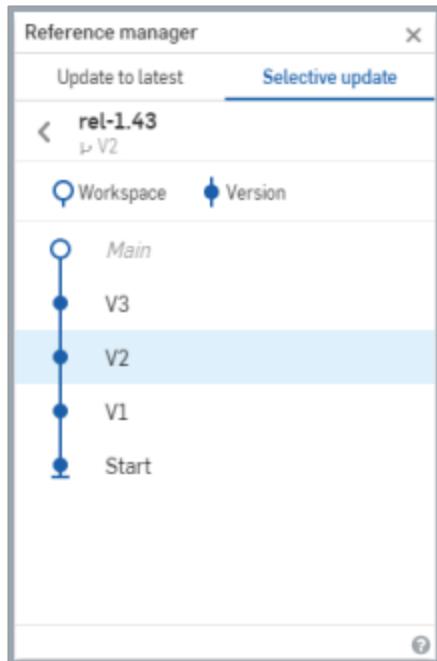
1. Click the update icon (or right-click the feature and select *Update linked document*) to access the Reference manager, in which you can choose update options:



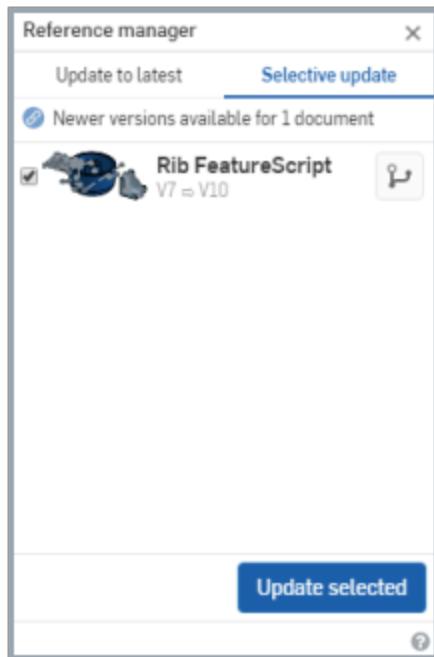
2. To update to the latest version, click *Update all*.
3. To update to a specific version:
  - a. Click *Selective update*:



- b. Select the document (if there is more than one) and click  for that document:



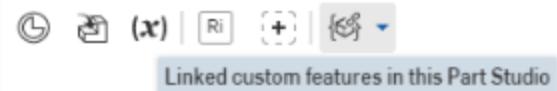
- c. Select the version to update to.  
d. Repeat for any other documents in the list, if necessary.  
e. Click *Update selected*:



## 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.

## 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:



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.

## Tips

- 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 documents.
- Adding a custom feature to your toolbar (or opening a document containing a custom feature) automatically turns on the FeatureScript notices, indicated by 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 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**:

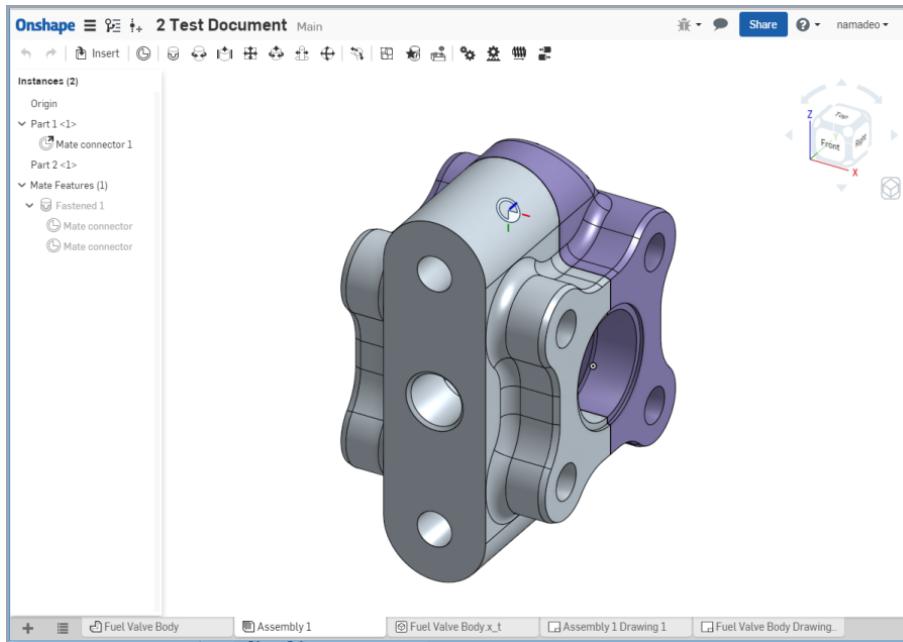
- Take a long time to regenerate, or otherwise consume excessive resources, interfering with your ability to work with the Part Studio until you remove the custom feature.
- Modify variable values or geometry in the Part Studio in an attempt to cause harm.

Publishing malicious FeatureScript is against the Onshape Terms of Use and will not be tolerated.

Please report malicious custom features using the Feedback button in the Help menu.

# Assemblies

An Onshape Assembly tab is where you define a hierarchical structure of part and subassembly 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 subassembly, and/or instance a part directly.



## 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**.

## Basic steps to assembling parts

1. "Insert Parts and Assemblies" on page 306.
2. Create "Mate Connector" on page 281.
3. Create "Mates" on page 320.
4. Create "Relations" on page 375 if desired.

The tools and functionality available for Assemblies include:

- "Insert Parts and Assemblies" on page 306 dialog - For selecting parts and subassemblies to include in an assembly
- "Triad Manipulator" on page 317 - For moving parts and assemblies around the graphics areas and for movement between parts

- "Mates" on page 320 - For defining movement between parts
- "Mate Connector" on page 281 - For defining where parts connect to each other
- "Snap Mode" on page 365 - Drag and drop parts to create Mate connectors and Mates on the fly.
- "Group" on page 381- For defining spatial relationships between parts
- "Assembly Feature Lists" on page 385 - A list of part instances and mate features in an Assembly
- Context Menus in Assemblies - Select from a list of actions
- "Assembly Measure tool" on page 387 - Acquire measurement information about part edges and faces



# Insert Parts and Assemblies

Shortcut: i



The Assembly toolbar is active when an Assembly tab is active.

**Insert Parts or Assemblies** inserts an instance of a part or an assembly into the active Assembly. You can instance a specific part or all parts defined in a Part Studio, an assembly defined in a different Assembly tab, as well as parts and assemblies defined in other documents.

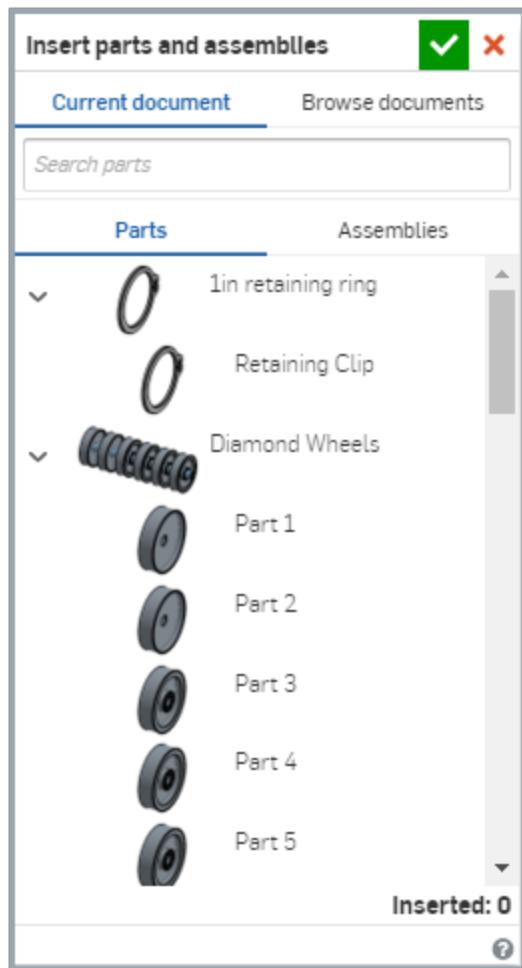
The default positioning is the alignment of the Part Studio origin (of the part or subassembly being inserted) with the Assembly origin (of the assembly being inserted into).

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.

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.

## Steps

1. Click the **Insert parts and assemblies** tool .
2. Select **Parts or Assemblies**.
3. Use the Search box to search for a particular part or assembly, or select from the list.
  - a. Select Current document to select parts/assemblies from the currently active document.
  - b. Select Browse documents to select parts/assemblies from other documents which have versions and to which you have edit permission. This is referred to as [Linking documents](#).



If you click the parts to insert and then close the dialog, the parts are inserted into the Assembly with the Assembly and Part Studio origins aligned. 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.

4. Drag to reposition the parts.

## Assembling immediately

When a component you are inserting has a root-level 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 insert has more than one explicitly-defined root level mate connector, you can 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 can pan and rotate freely as you insert a part instance (or subassembly), even in Snap mode.  
If no explicitly-defined root mate connectors are on the component being inserted, then a normal non-snap free drag is available, even if Snap mode is turned on.

# Linking Documents

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.

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 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.

## 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.

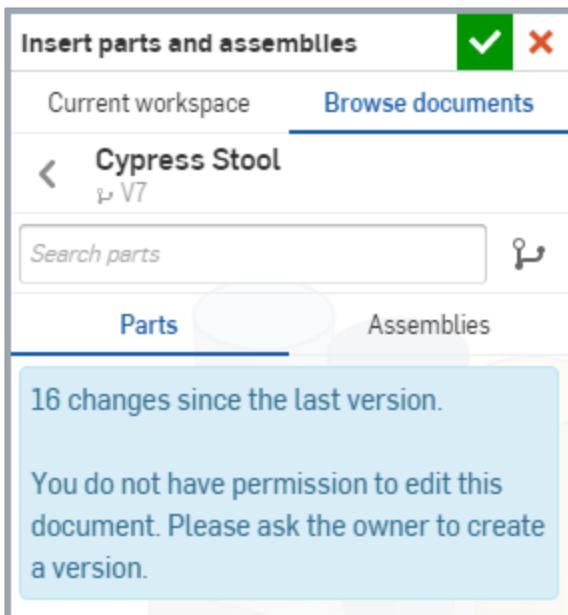
A key aspect of linking documents is that all of Onshape's document permissions work seamlessly. You decide when a document should change from Editable to View only to Reference only on a per user basis - and you can always change permissions whenever you want.

## Steps

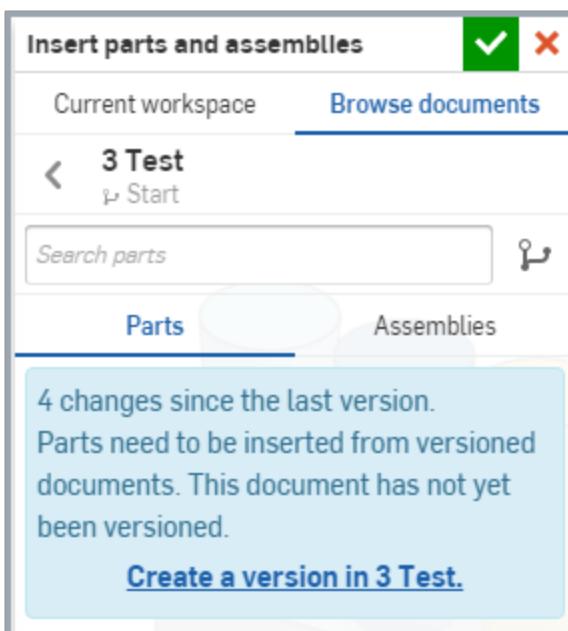
In an Assembly:

1. Click Insert .
2. To insert a part or assembly from another document, select Browse documents.
3. Use the Search box or use the filters and select a document from the list.

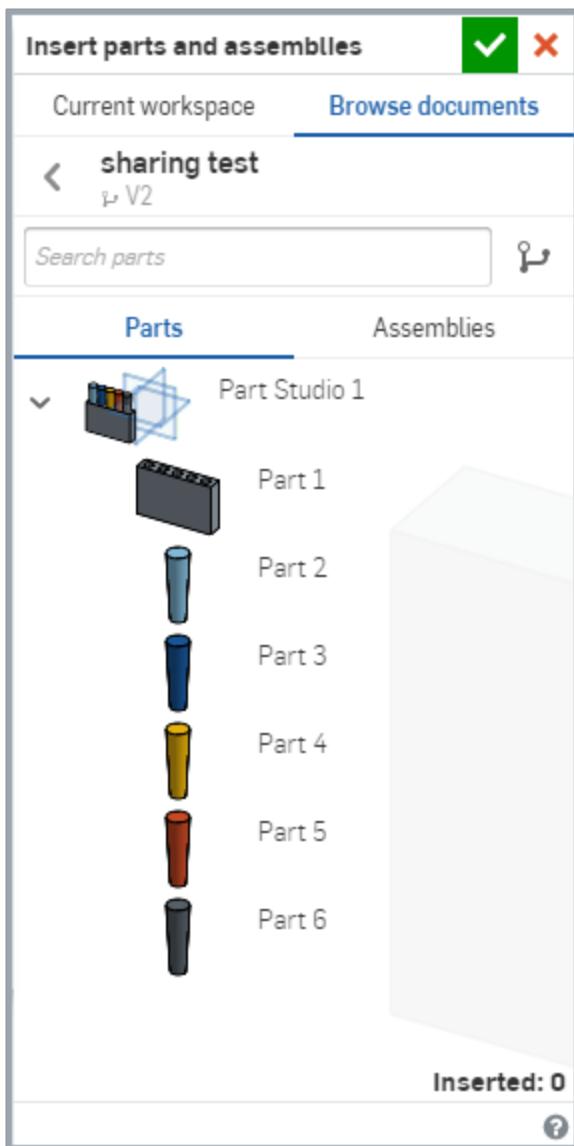
If the selected document doesn't have any versions, or you do not have edit permission, Onshape displays a notification.



If you are the owner of the document, you are prompted to create a version (if none exists). Click the link to open the document in a new tab:



4. Select a part or assembly from the document.



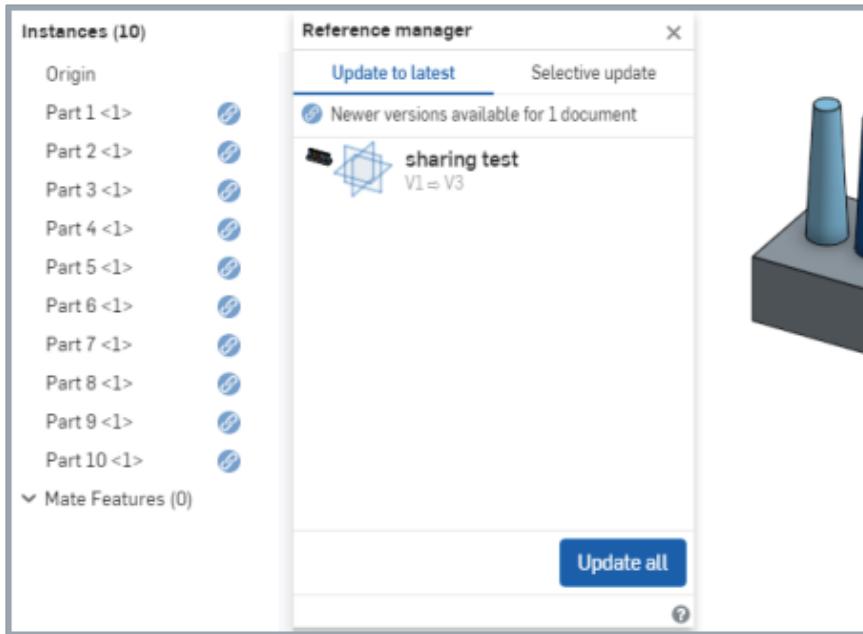
## Updating linked documents

When a referenced document is updated, Onshape adds a special icon next to the part in your document:

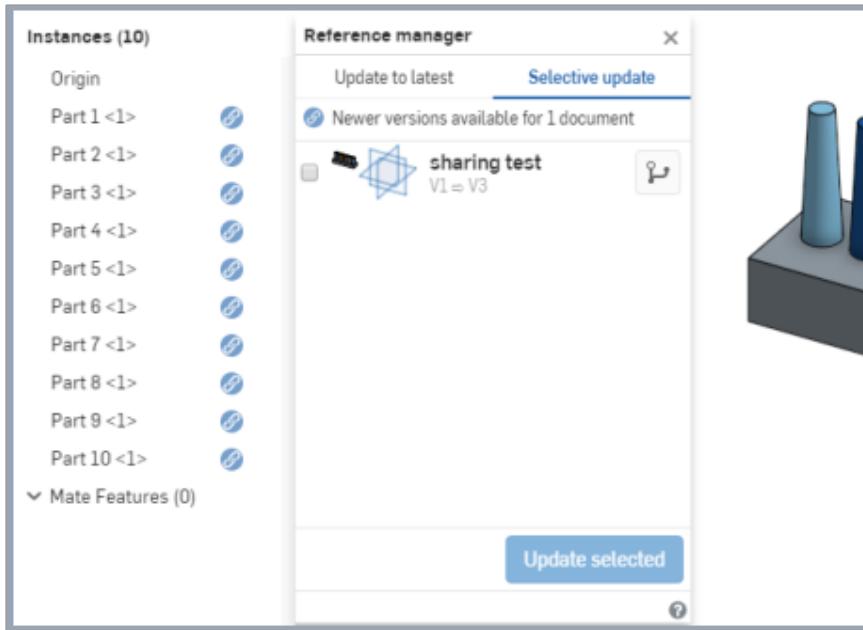


The gray link icon indicates a part referenced (linked) from another document. The white link with the blue background indicates a part referenced from another document, and that the part now has another version. This notification icon is also visible on the Assembly tab.

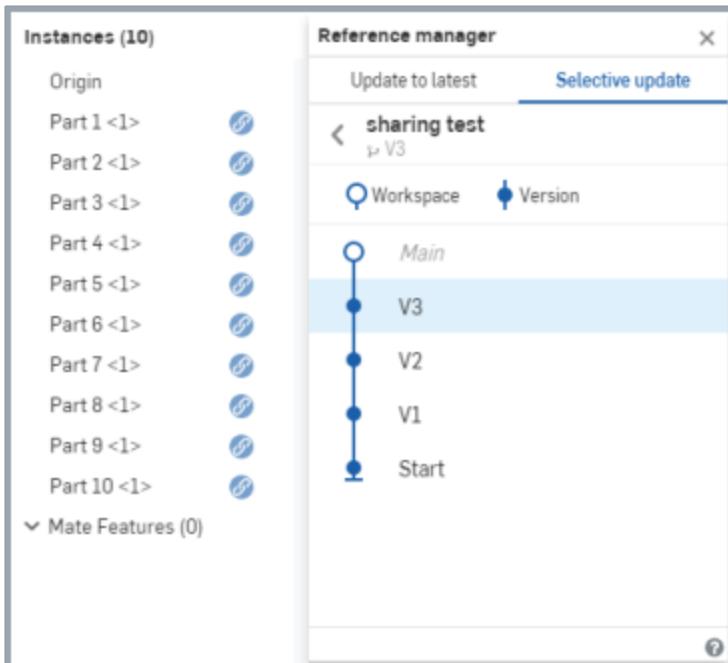
1. Click the icon with the blue background  to begin an update.
2. If more than one part has a newer version, you can click Update all to update all parts to their latest version.



3. To perform an update on specific items, click Selective update to select the parts and versions.



4. Click  to visualize the version graph before making a decision.



5. Select the version.

6. Click Update.

If you want to revert back to the versions you had before the Update, click the Undo button.

## Tips

- While the case described here is for Linked Documents, you can also 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 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.

# Managing Assemblies

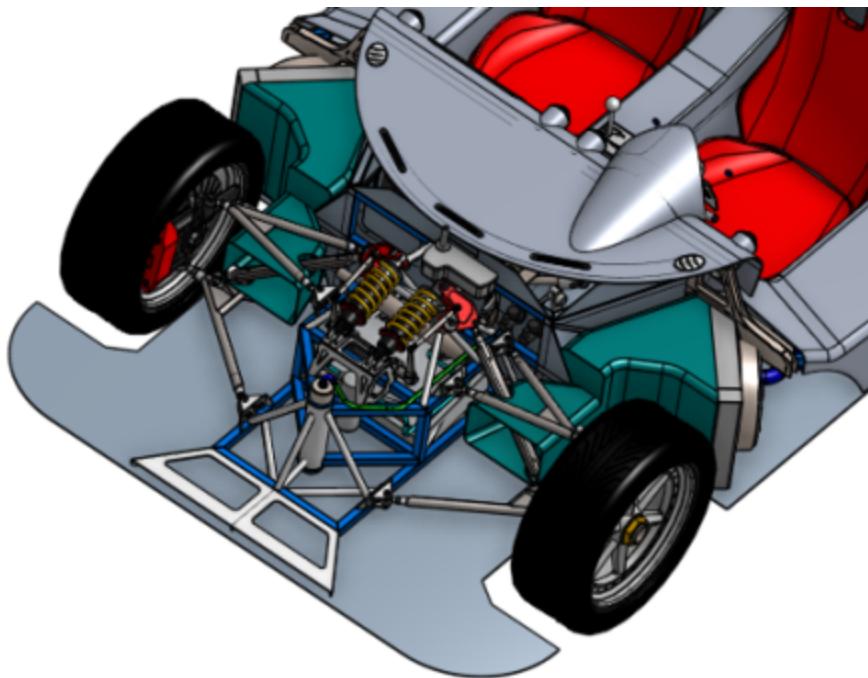
To aid in the process of assembling parts, Onshape provides some convenient tools:

- Hide (selection)
- Hide other parts
- Hide all parts
- Isolate (selection)

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.

Access these commands from the context menu for selected parts in an assembly.

The examples use this model:

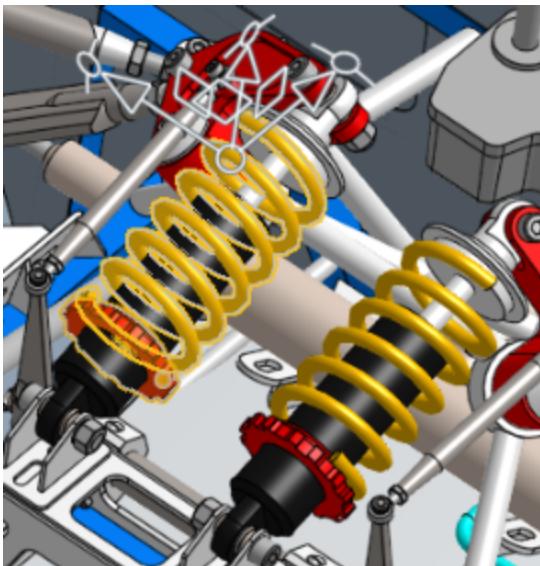


## Hiding parts

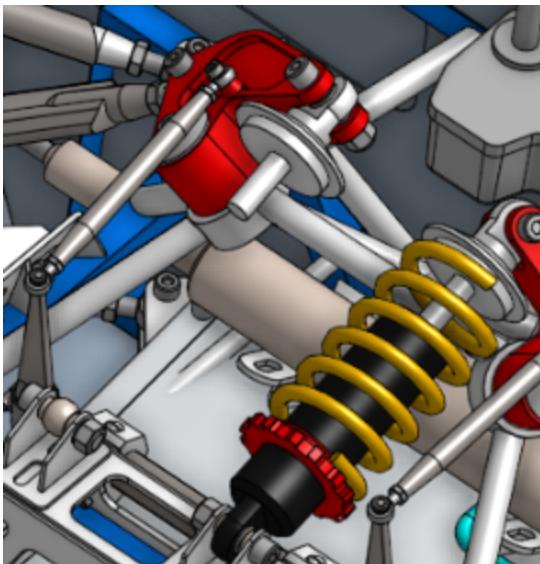
Hide all parts, selected parts, or ‘all other’ parts excluding those selected to aid in visualizing necessary entities for assembling or evaluating movement of an assembly. Select parts in the graphics area or from the Parts list; box select also works for selecting. This command is modal: hide/show.

### [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

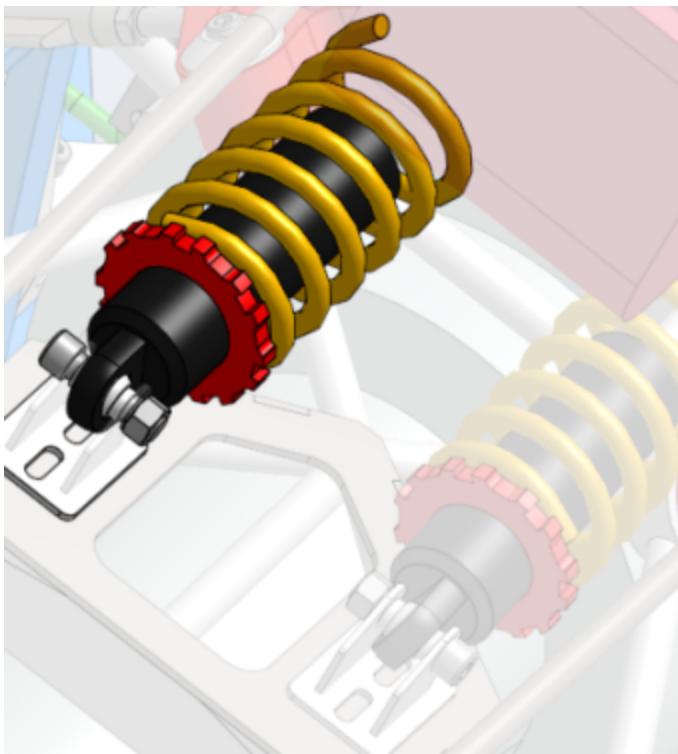
Select the parts you want to visualize and click Hide other parts in the context menu:



## 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 can 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.

Invoking Isolate during the process of defining a mate (with the Mate dialog active), clears the selection list in the dialog.

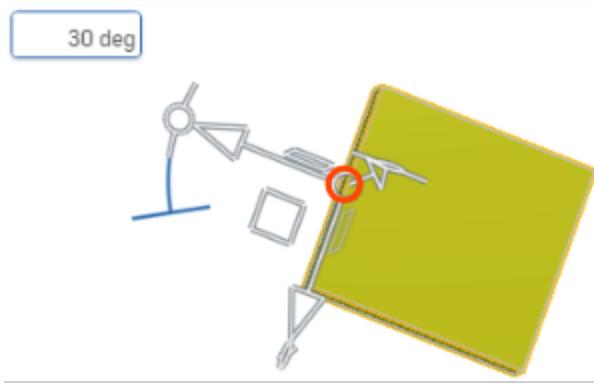
# Triad Manipulator

Once an instance is inserted into an Assembly, you can position it in two ways:

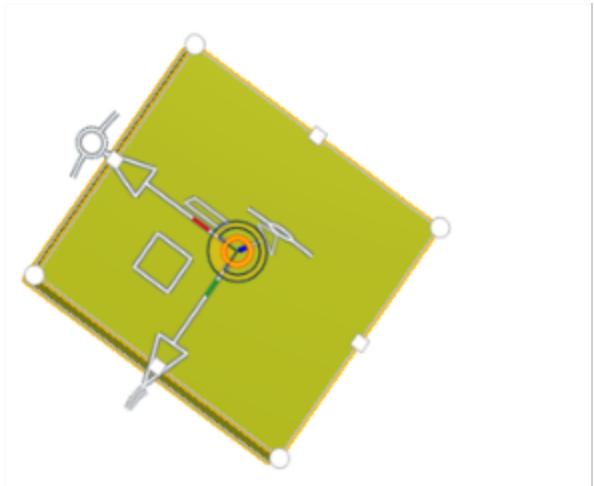
- Use the mouse to click and drag it (referred to as **free drag**).
- Click on it to activate a triad manipulator (referred to as **manipulator drag**).

## Repositioning the manipulator itself

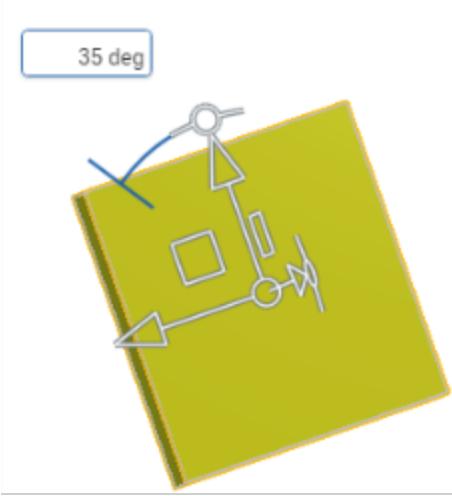
Click a part to visualize the manipulator. Use the center circle (highlighted in orange below) to move the manipulator without moving a part.



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:

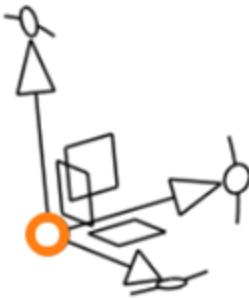


Enter a numeric value in this field to define the position of the part in relation to the mate connector.

You can snap it to other entities in the Assembly to redefine the entity's position and orientation. You can place this center on an inferred Mate connector, 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:

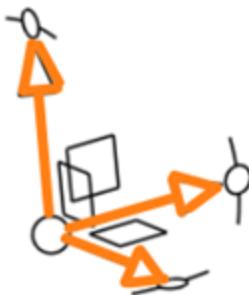
- Move to origin (this simply moves the part, placing its reference point at the origin; it does not mate or fix the part).



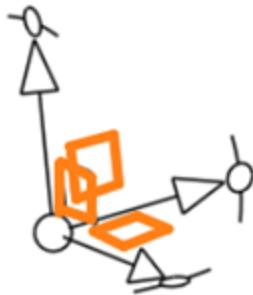
## Move the instance along an axis

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

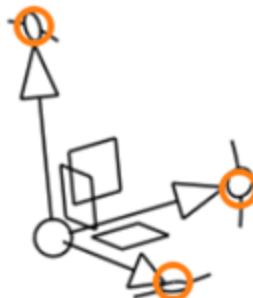


## Rotate the instance around the triad X, Y, or Z axis

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.

# Mates

Shortcut: m



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.

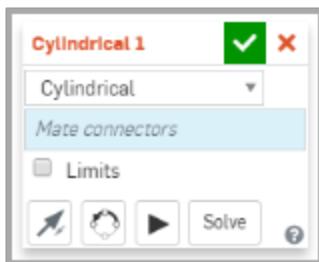
Use the shortcut key **j** to hide/show mates in an assembly.

Note that you can mate a part to the high level Origin in an Assembly. You can also Fix a part in order to test the movement of assigned mates using the context menu.

See the videos titled Getting Started with Assemblies and Assembly Mates to learn more about building assemblies and mating in Onshape.

## 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 can also check the box to apply limits of movement. Other options/action include:



- Flip the primary axis, Z orientation of the instances.



- Index 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.

## Mate context menu

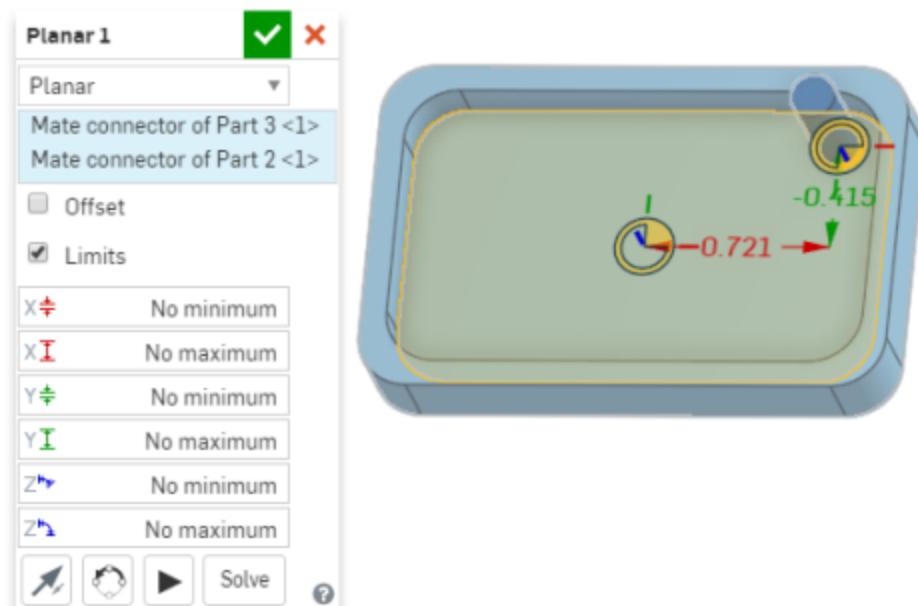
Use the Mate context menu to access the following commands:

- **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** - Show all mate connectors
- **Isolate** - Dim and deactivate 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 314.
- **SUPPRESS** - Visualize the assembly without the mate (and without deleting the mate)
- **Clear selection** - Clear all selections
- **Delete** - Remove the mate from the assembly

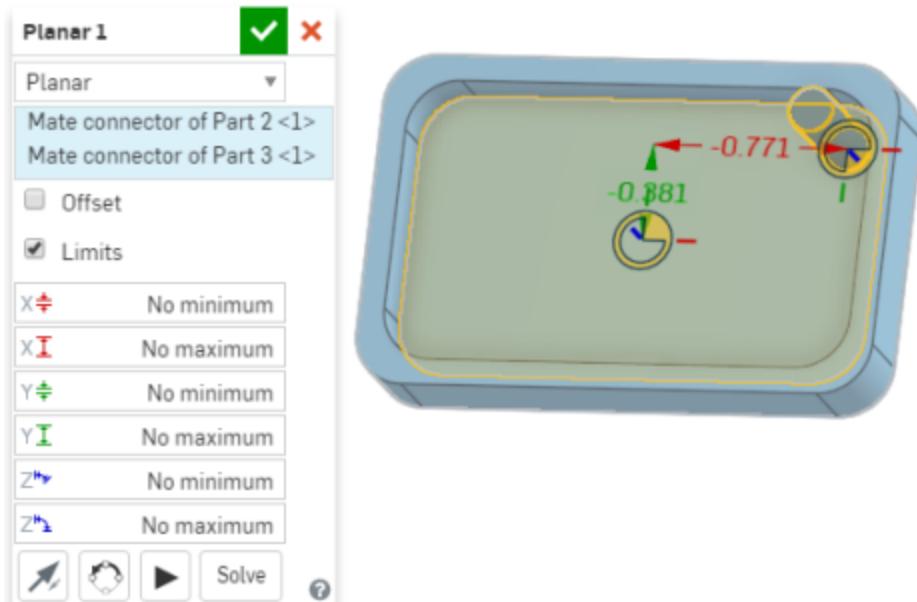
## LIMITING movement

You can specify movement limits of all mates except Ball, Fastened, and Tangent. Onshape provides visual cues for limit distances, providing distance values, in default units, from the second mate connector selected to the first. Notice the measurement animation:



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.

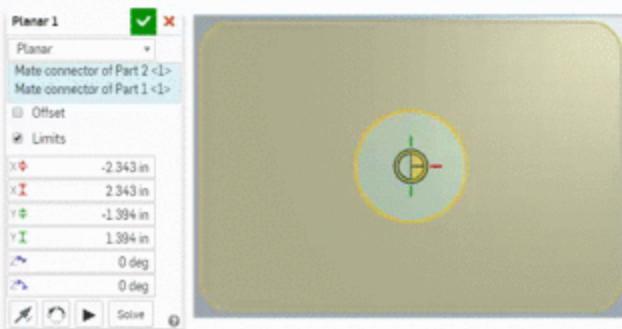
Now, switch the order of Mate connector selection and notice the distance values:



Notice that in this scenario, the Y value is negative and the X value is negative. 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 min and max 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:



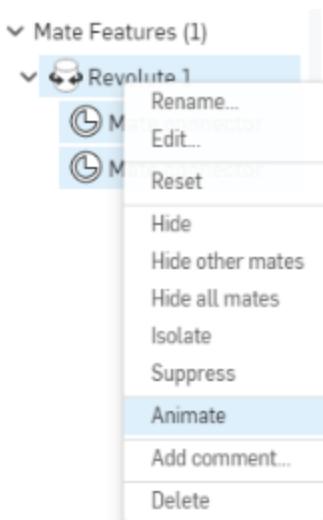
You can use [expressions and trigonometric functions](#) in numeric fields in Assemblies.

## Animating movement within an assembly

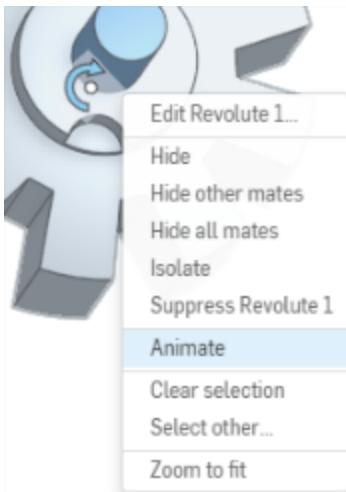
Use the Animate command (found in the context menu for mates and mate indicators) to drive the assembly from a single mate (or single DOF within a mate). Other mates and relations in the assembly are also enforced and honored.

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:

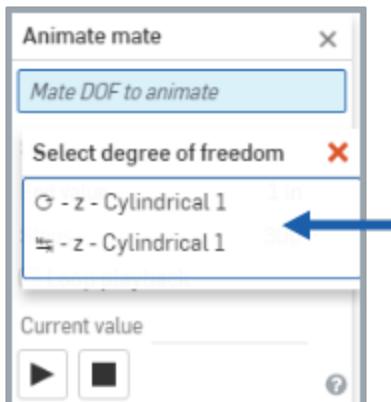


*The Animate command on the Mate context menu*

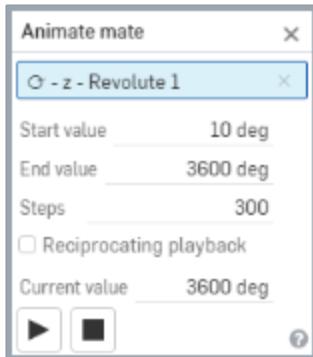


*The Animate command on the Mate indicator context menu*

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 Stop 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 can enter up to 36000 degrees here (100 revolutions), which is 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. Check *Reciprocating playback* to play the animation of the degrees of freedom continuously until you manually stop it.

*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

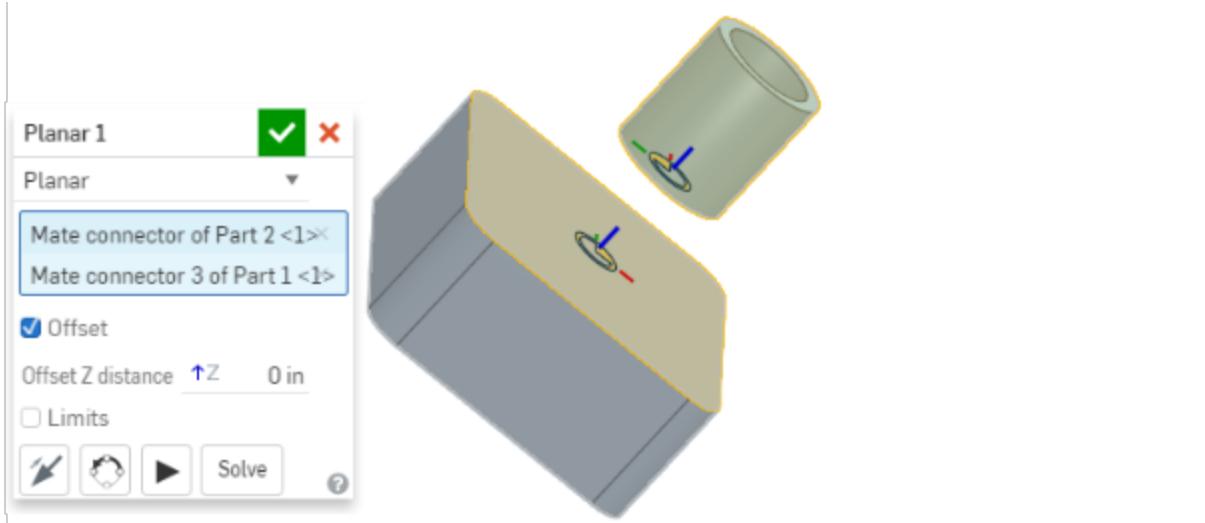
- The Animate command works with various graphics modes, like Isolate, mate indicators and mate connectors.
- Animate helps you explore the relationships between mates, their constraint systems, and gives you a way to show off your design (especially with the playback loop feature).

## Offset parts during assembly

Offsetting parts from one another during assembly is available for the following mate types:

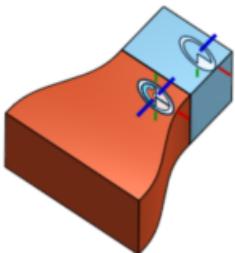
- Planar offset - Along the Z axis
- Slider offset - Along the X and Y axes
- Revolute offset - Along the Z axis
- Pin slot offset - Along the Z axis
- Fastened offset - Along the X, Y, and Z axes

You can also drag the parts and observe the distance values in the graphics area. These can help determine the specific values to enter in the dialog:

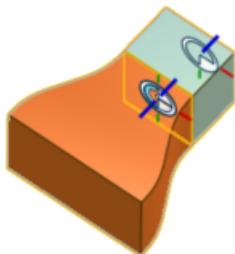


## Copying/Pasting assembled parts

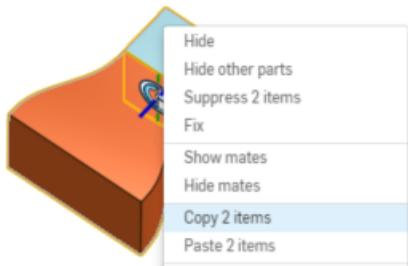
You can copy and paste parts that have been mated in an Assembly:



Select the parts:



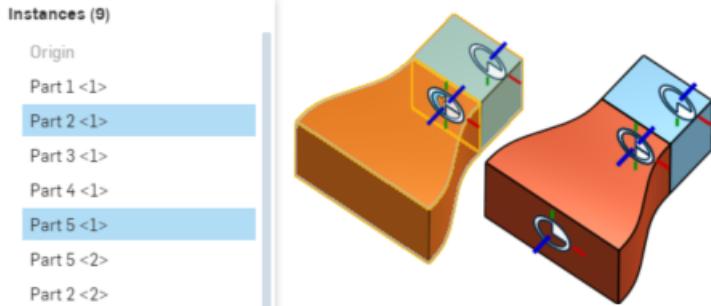
From the context-menu, select Copy parts:



From the context-menu, paste the parts:



The parts are pasted directly over the copied parts. Click the parts drag the copies away:



Notice that the parts, 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 can hide the parts 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, and red indicates a problem:



Fastened



Revolute



Slider



Planar



Cylindrical



Pin slot, with an arrow in the direction of the slot



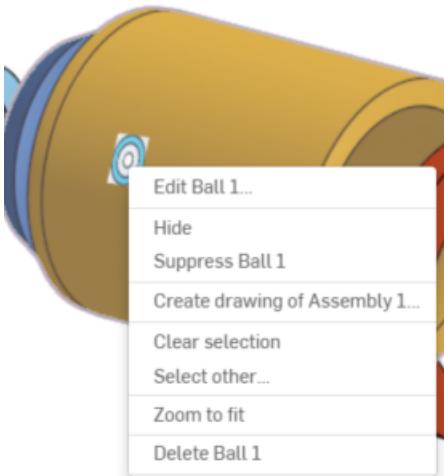
Ball



Tangent

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 can take action on the mate:

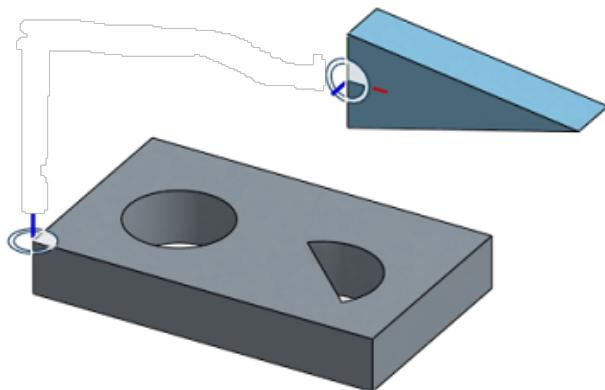


- Select a mate, mate connector, or mate relation in the graphics area and its associated instances and mate feature are highlighted in the list.

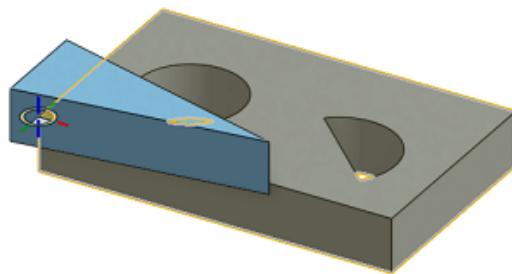
## Concepts

- There is exactly one Mate between any two instances.
- Fixing 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.
- The Mate positions two part instances in relationship to each other, aligning a Mate connector on each instance.

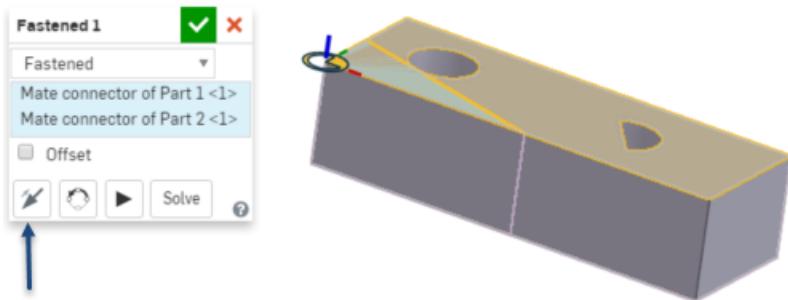
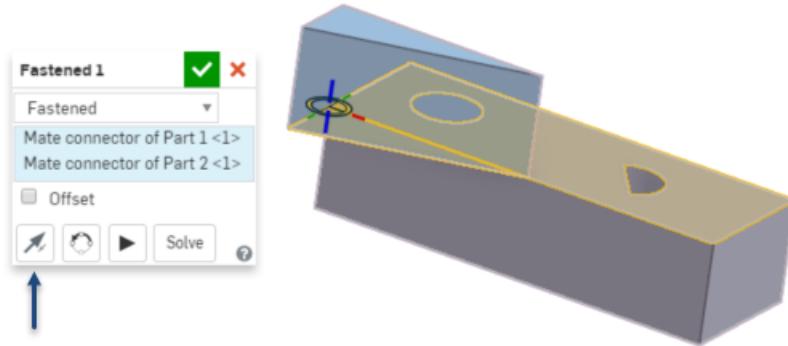
**Before Mate**



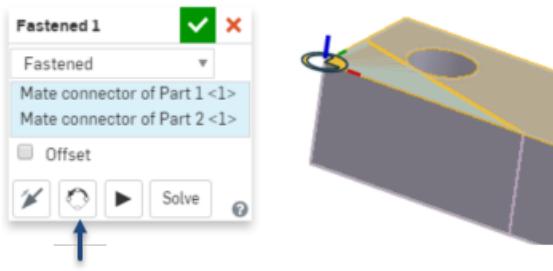
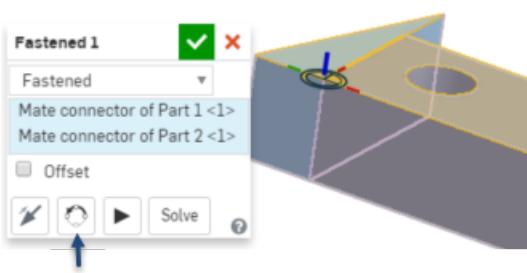
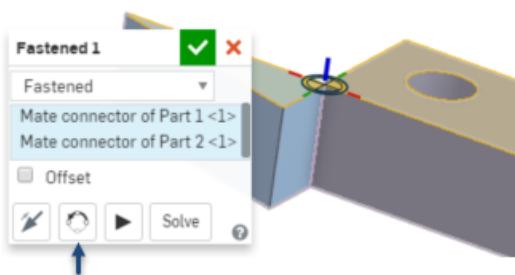
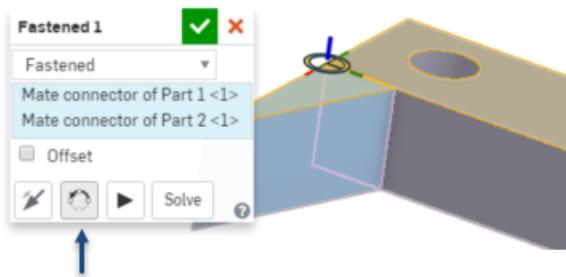
**After Mate**



- The initial position is often a best guess. There are three tools to correct the position:
- The **Flip primary axis** tool flips the major (Z) orientation.



- The **Reorient secondary direction** tool adjusts the orientation in 90 degree increments

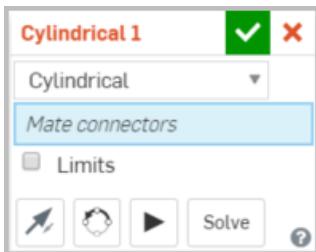


- The play button ► animates the allowed movement between the mate being created.
- The Solve button regenerates the mate in process and the movement of all mates, so you can see how your changes affect the entire assembly.

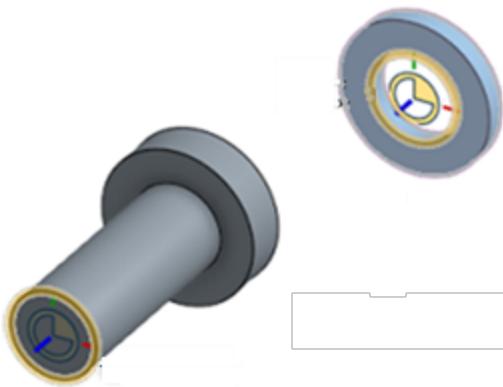
The **Mate type** then specifies the movement behavior.

## Example

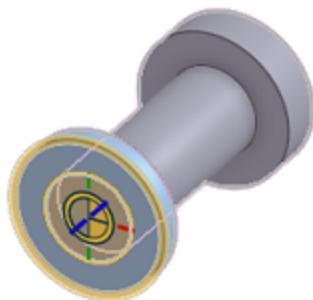
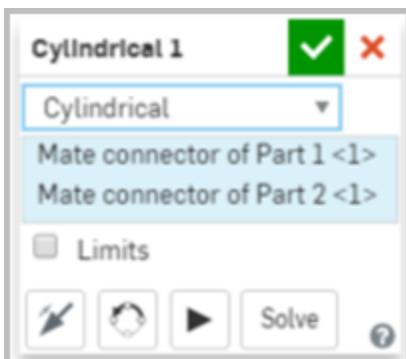
- Select a Mate (for example ) to open the dialog:



2. Select one automatic Mate connector on each part (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 can be useful to create Mate connectors ahead of time. You can create Mate connectors in either the Assembly or in the Part Studio.

# Fastened Mate



Mate two parts and remove all degrees of freedom between them.

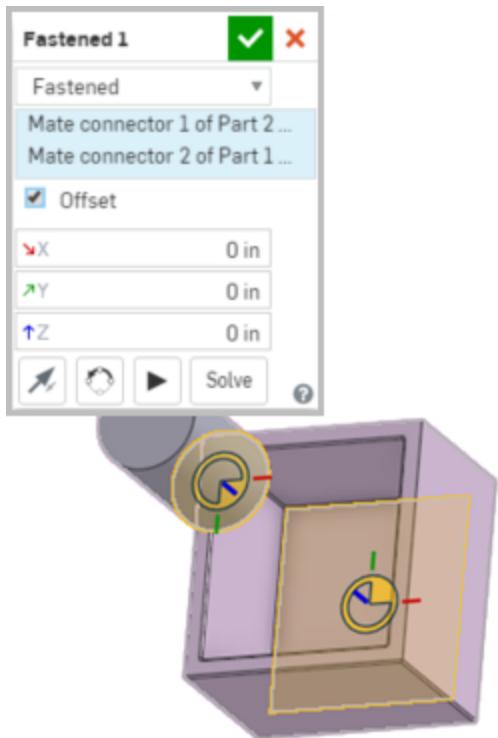
You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

## Steps

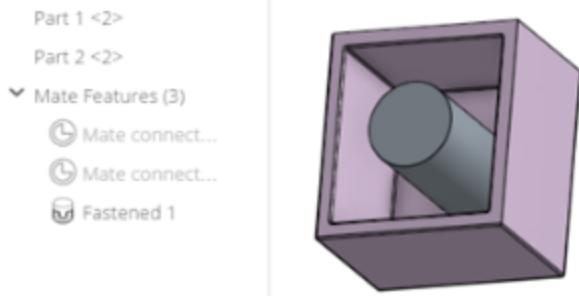
1. Create a **Fastened Mate**  using the two Mate connectors:



If you want to supply an offset distance, check **Offset** and supply a distance. Fastened mates can offset the parts along any combination of the three axis:



2. The final result (without an offset) looks like this:



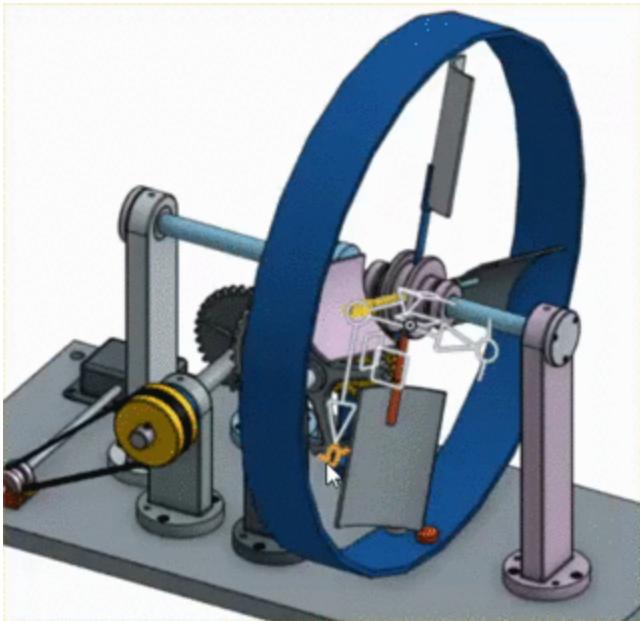
3. Click the part to access the manipulator.
4. Click and drag the various manipulator handles to see which motions are allowed; notice that no motion is allowed. The part has zero degrees of freedom.



## Revolute Mate



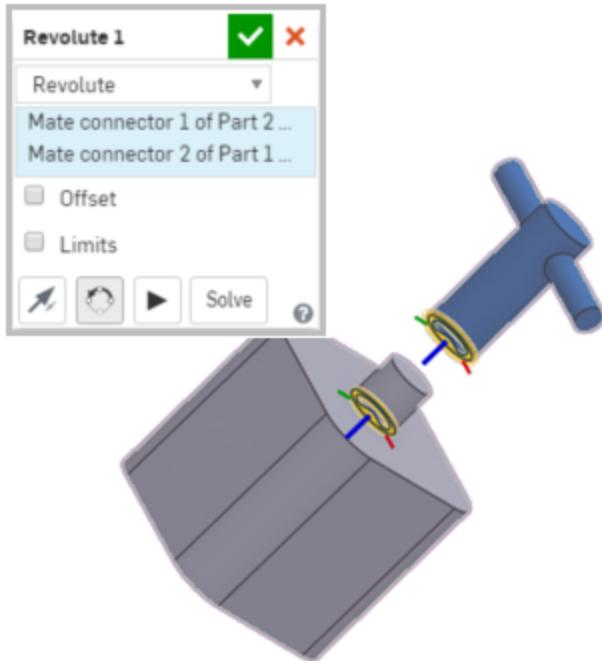
Mate two parts allowing rotational movement about the Z axis. (Rz)



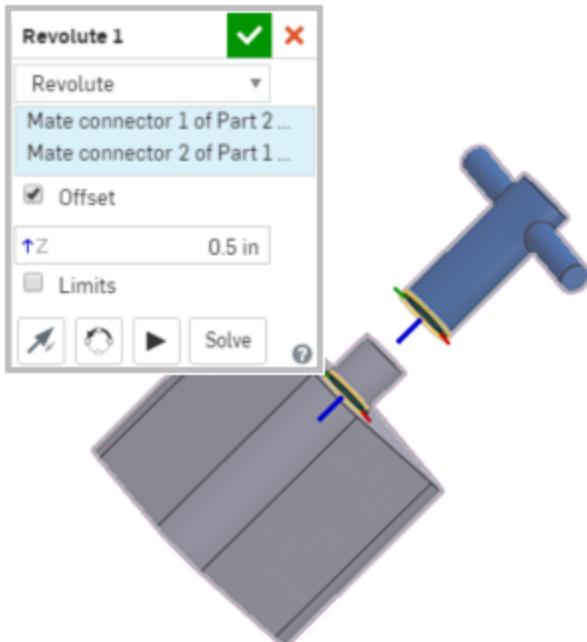
You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

## Steps

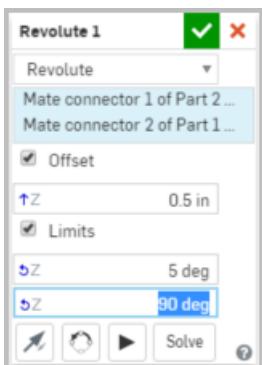
1. Create a **Revolute Mate**  using the two Mate connectors:



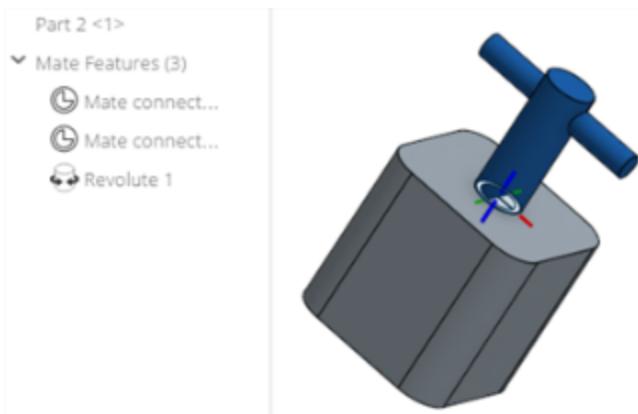
If you want to supply an offset distance, check **Offset** and supply a distance. Revolute mates can offset the parts along the Z axis only: To create an offset between the two parts, click **Offset** and specify a distance:



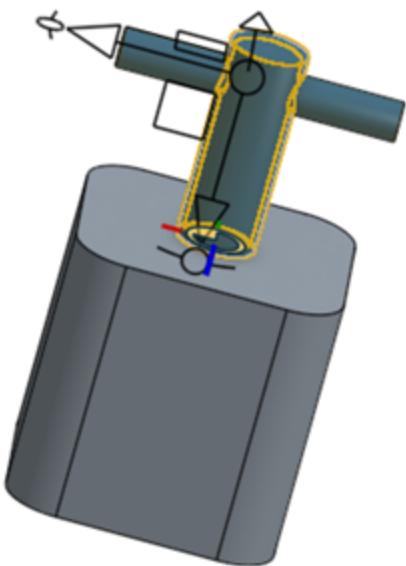
If you want to limit the movement, check **Limits** and supply (optional) minimum and maximum values to control the range of motion of the mate:



2. The final result (without an offset) looks like this:



3. Click a part to access the manipulator:

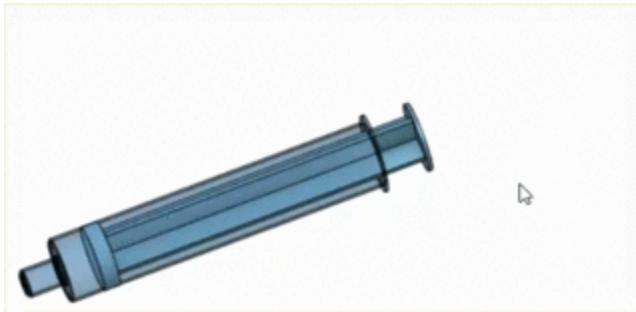


4. Click and drag the various manipulator handles to see which motions are allowed; notice that only rotational movement about the Z axis is allowed ( $R_z$ ).

# Slider Mate



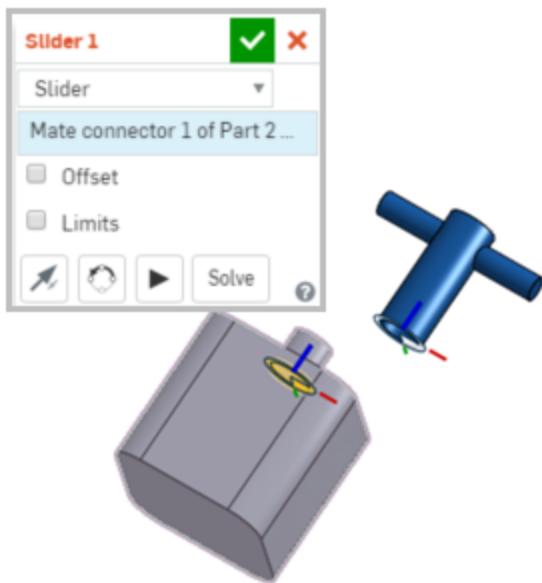
Mate two parts allowing translational movement along the Z axis. (Tz)



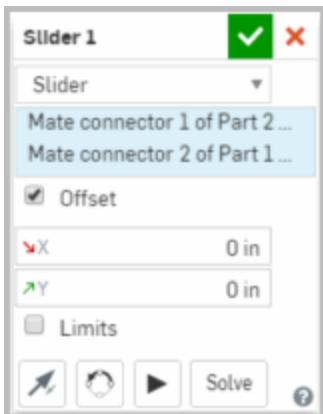
You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

## Steps

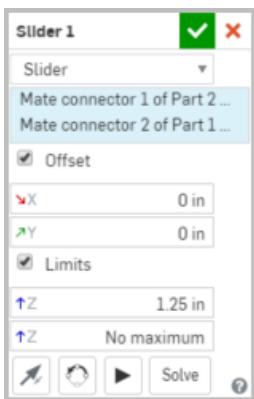
1. Create a **Slider Mate**  using the two Mate connectors:



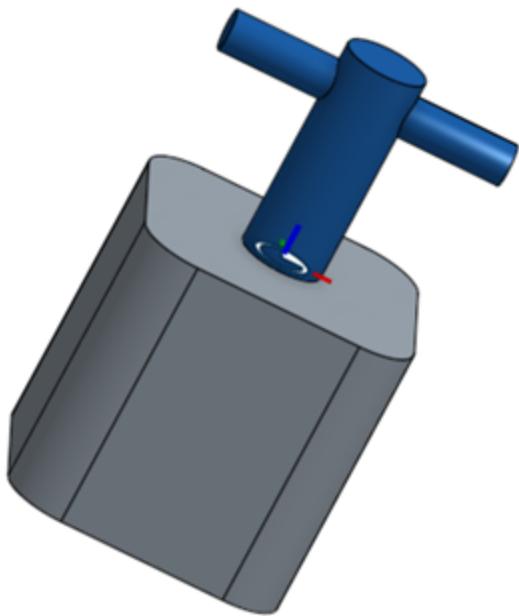
If you want to supply an offset distance, check **Offset** and supply a distance. Slider mates can offset the parts along any combination of the X and Y axis:



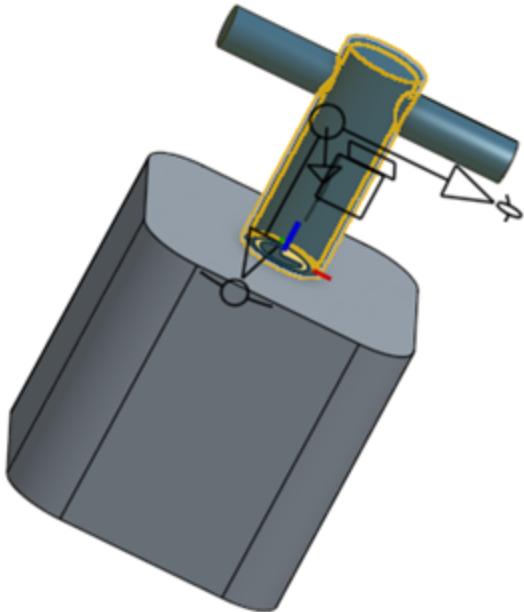
If you want to limit the movement, check **Limits** and supply (optional) minimum and maximum values to control the range of motion of the mate:



2. The final result looks like this:



3. Click the part to access the manipulator:



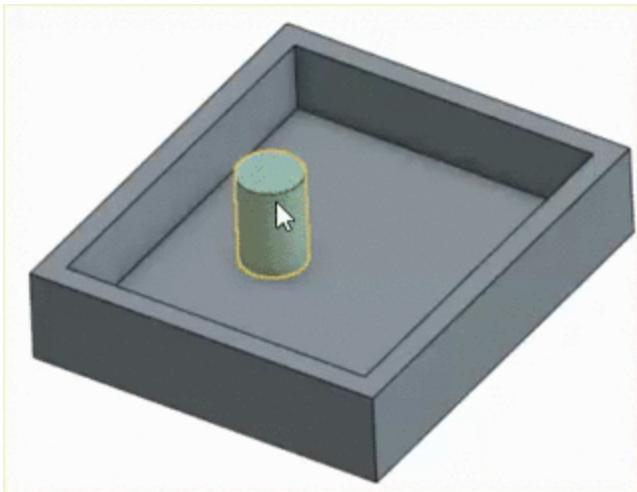
4. Click and drag the various manipulator handles to see which motions are allowed; notice that only translational movement along the Z axis is allowed ( $T_z$ ).



## Planar Mate



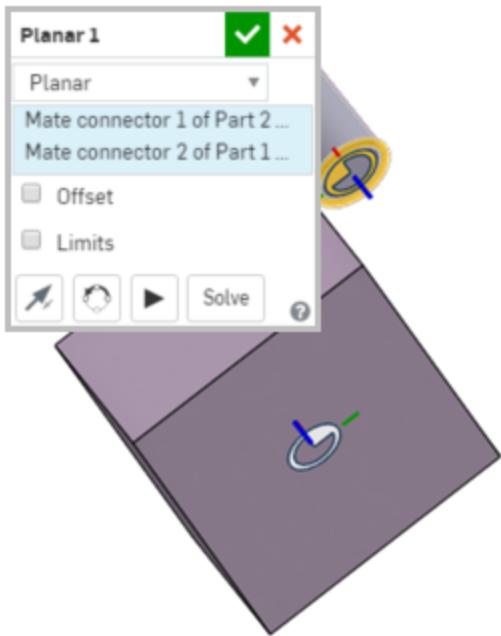
Mate two parts allowing translational movement along the X axis and the Y axis, and rotational movement about the Z axis. (Ty, Tx, Rz)



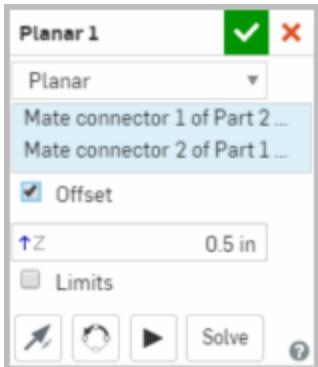
### Steps

You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

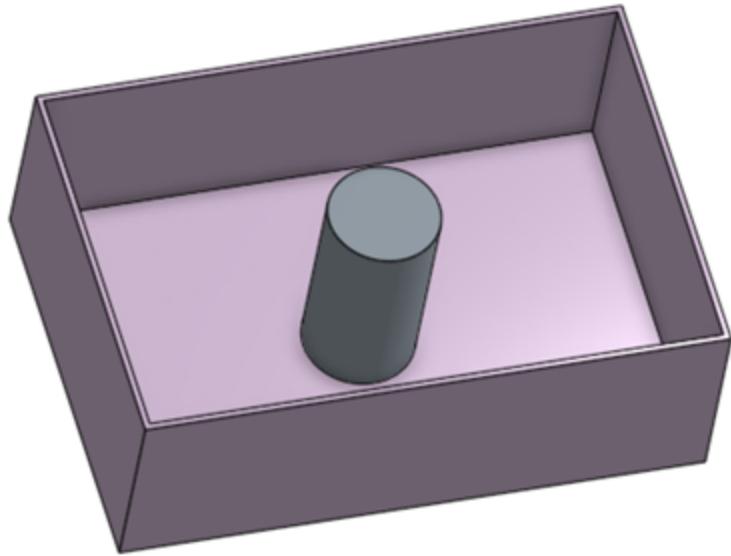
1. Create a **Planar Mate**  using the two Mate connectors:



If you want to supply an offset distance, check **Offset** and supply a distance. Planar mates can offset the parts only along the Z axis:



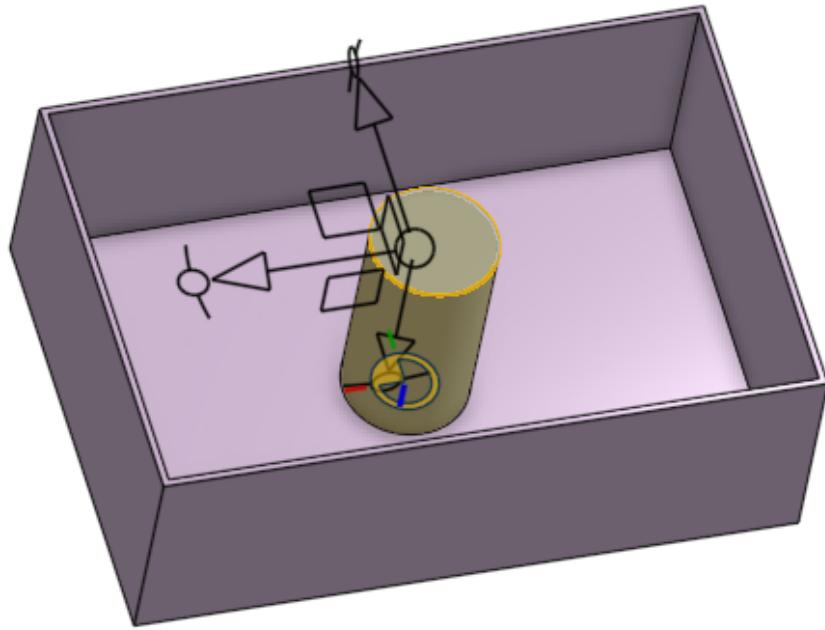
2. The final result looks like this:



3. If you want to limit the movement, check **Limits** and supply (optional) minimum and maximum values to control the range of motion of the mate:



4. Click the part to access the manipulator:



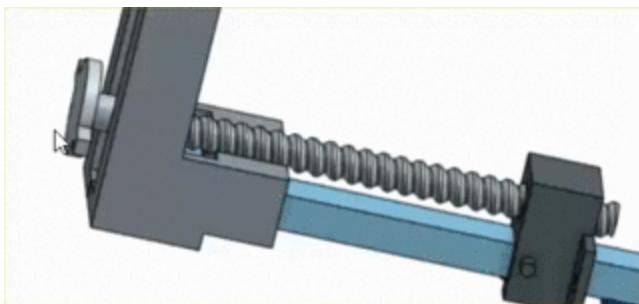
5. Click and drag the various manipulator handles to see which motions are allowed; notice that only translational movement along the X and Y axis, and rotational movement about the Z axis is allowed (Ty, Tx, Rz).



# Cylindrical Mate



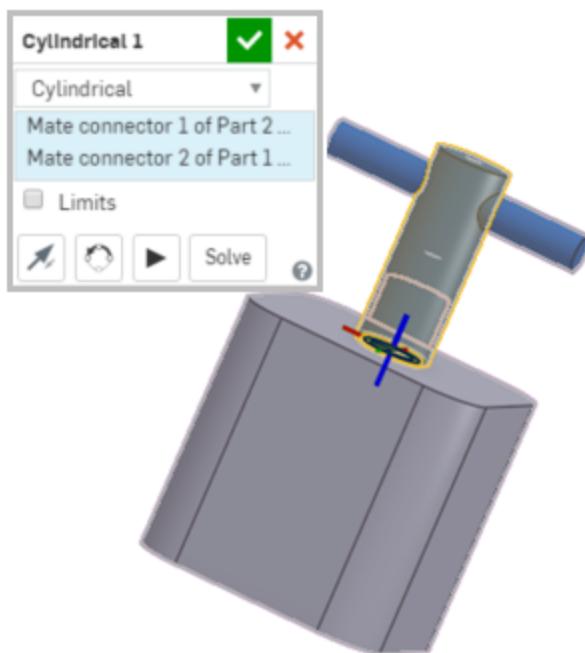
Mate two parts allowing translational movement along the Z axis and rotational movement about the Z axis. (Tz, Rz)



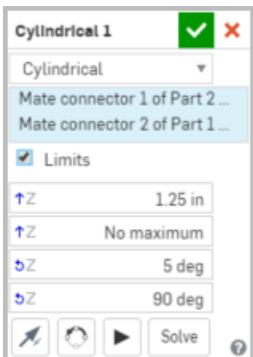
You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

## Steps

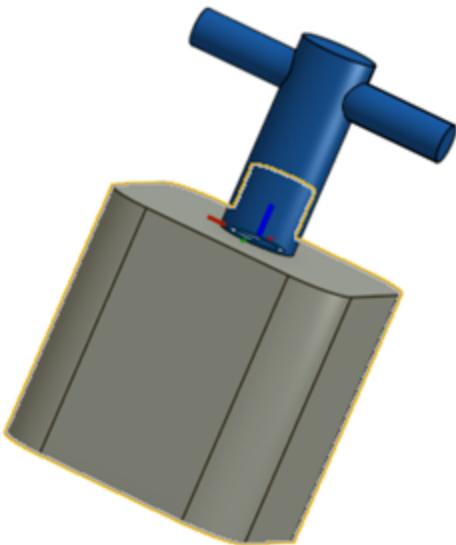
1. Create a Cylindrical Mate using the two Mate connectors:



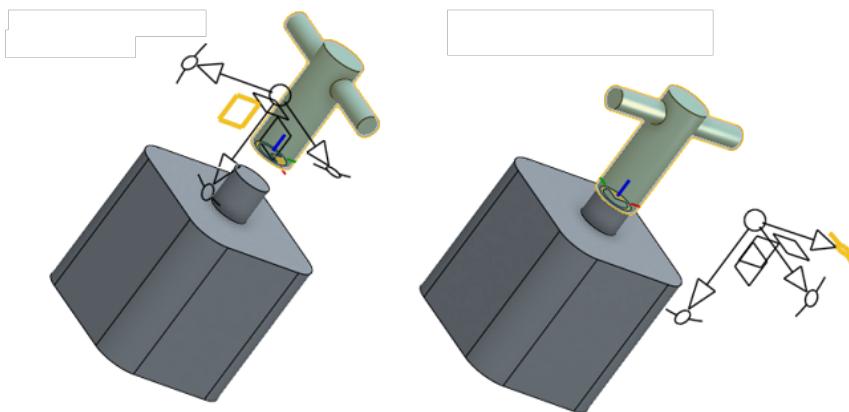
2. To impose limits on the movement of the mate, click Limits and supply minimum distances for the axis of both mate connectors:



3. The final result looks like this:



4. Click the part to access the manipulator:



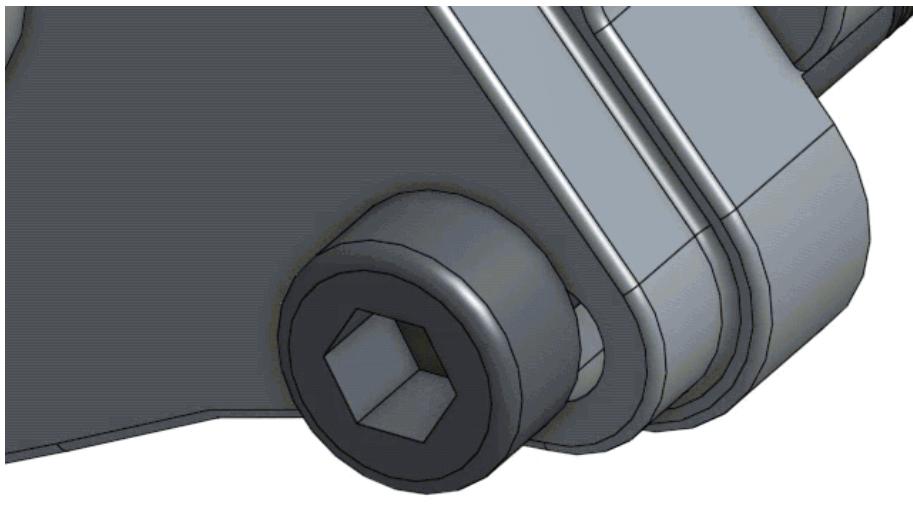
5. Click and drag the various manipulator handles to see which motions are allowed; notice that only translational movement along the Z axis and rotational movement about the Z axis is allowed ( $T_z$ ,  $R_z$ ).



## Pin Slot Mate



Mate two parts allowing rotational movement about the Z axis and translational movement along the X axis. (Rz, Tx)



You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

## Steps

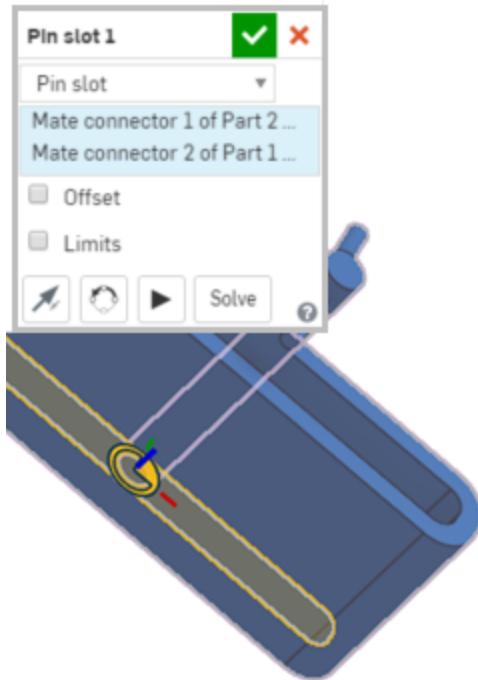
1. Select the **pin** instance first.



2. Select the **slot** instance second:



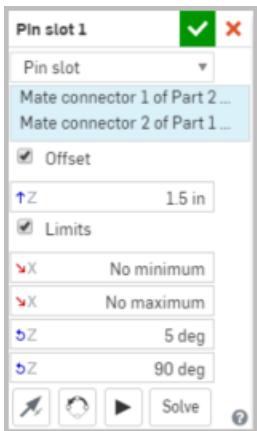
3. Create a **Pin Slot Mate**  using the two Mate connectors:



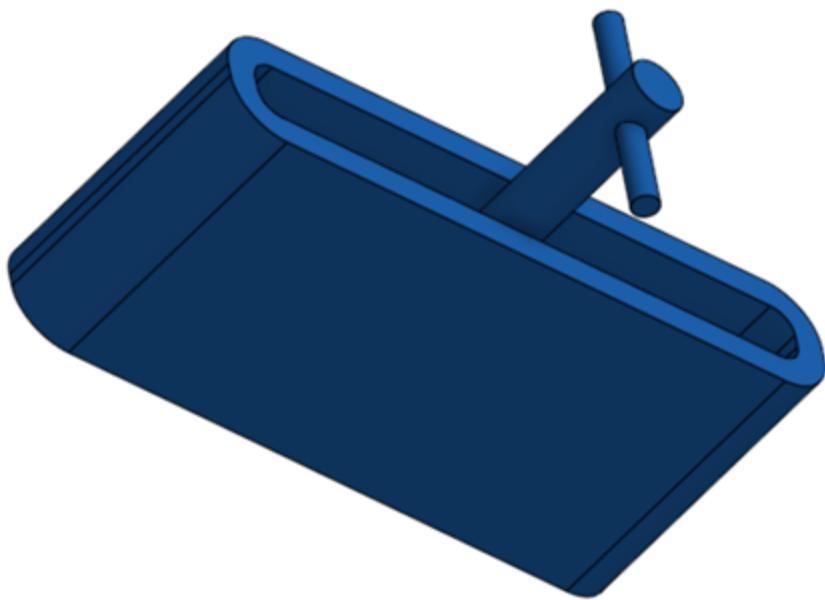
If you want to supply an offset distance, check **Offset** and supply a distance. Pin slot mates can offset the parts only along the Z axis:



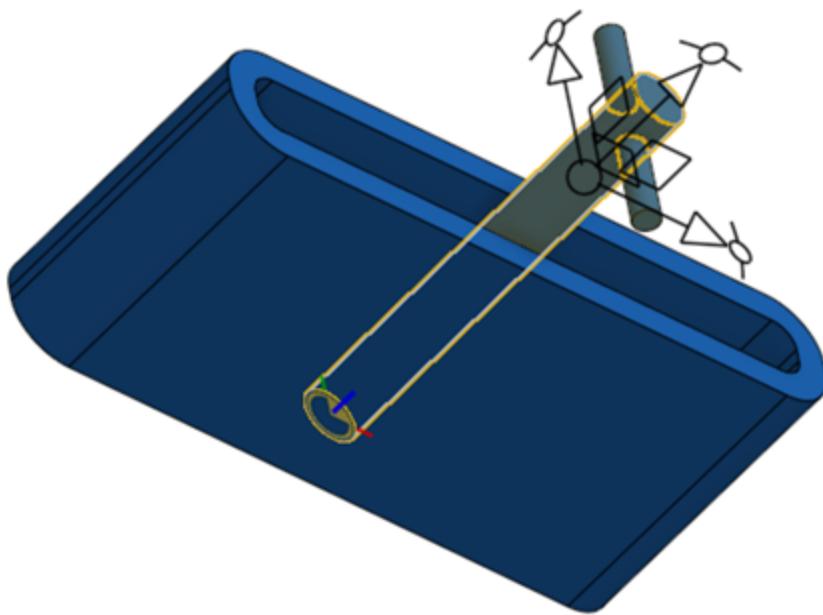
If you want to limit the movement, check **Limits** and supply (optional) minimum and maximum values to control the range of motion of the mate:



4. The final result looks like this:



5. Click the part to access the manipulator:



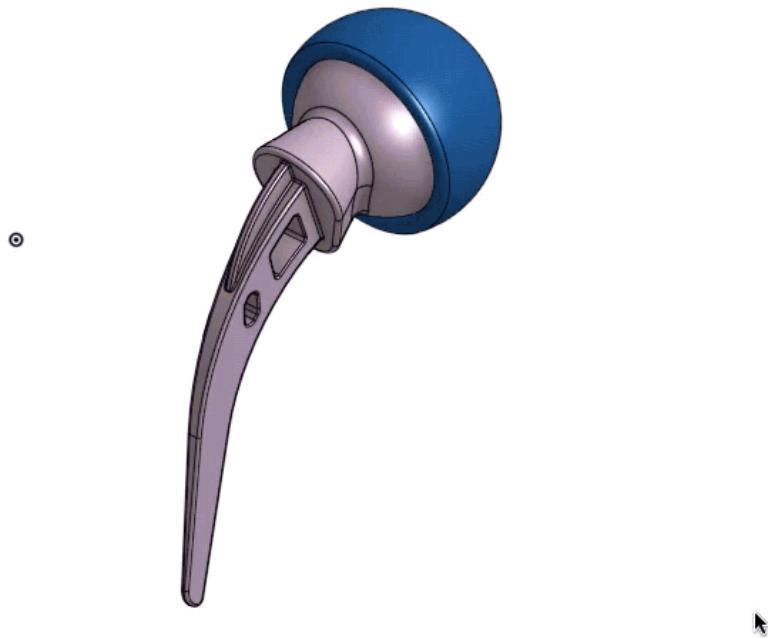
6. Click and drag the various manipulator handles to see which motions are allowed; notice that only rotational movement about the Z axis and translational movement along the X axis is allowed ( $R_z, T_x$ ).



## Ball Mate



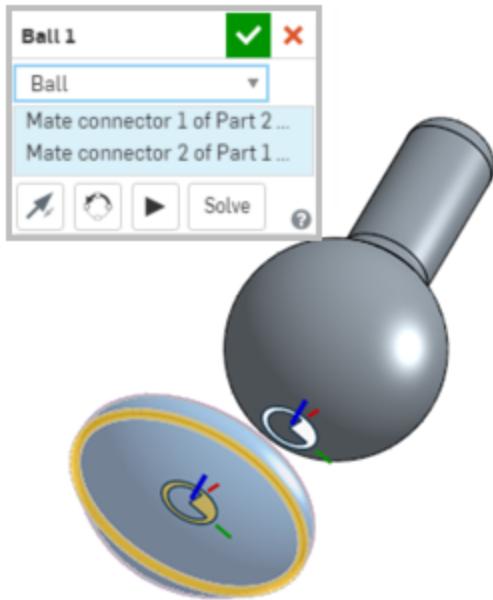
Mate two parts allowing rotational movement about the X, Y and Z axis. (Rx, Ry, Rz)



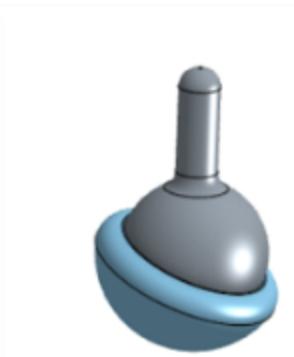
You can begin by creating mate connectors on each part, or use the implicit mate connectors visible upon hover.

## Steps

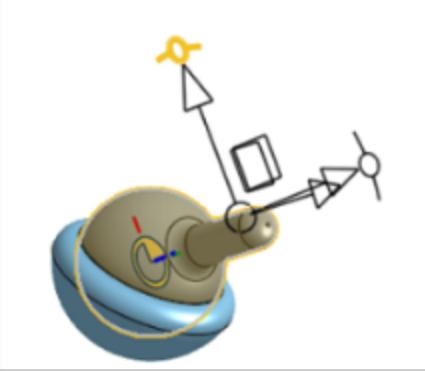
1. Create a **Ball Mate**  using the two Mate connectors:



2. The final result looks like this:



3. Click the part 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).

# Tangent Mate

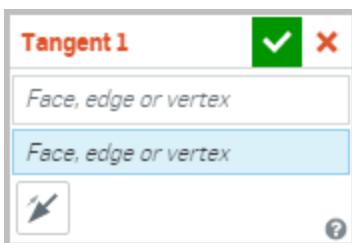


Mate two parts tangent to the selected faces, edges, or vertices.

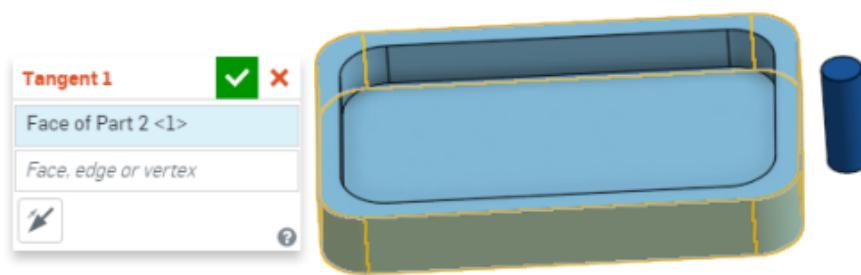
Tangent mates do not require or accept mate connectors.

## Steps

1. Click Tangent Mate 



2. Select a face, edge, or vertex of one part. In this instance, a face of the box is selected. Because of the filleted corners, the face is selected all the way around:



3. With focus in the second field in the Mate dialog, select a face, edge, or vertex of the second part:



In this example, 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.

# Mate Connector



Shortcut: Ctrl-m

In the Feature toolbar:



In the Assembly toolbar:



Mate connectors are local coordinate system entities located on or between parts and used within a mate to locate and orient part instances with respect to each other.

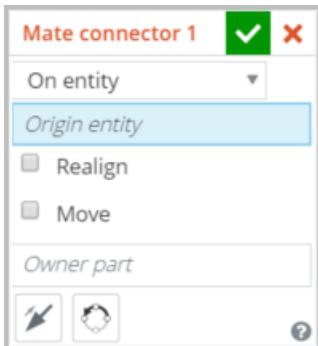
Two part instances are positioned in an assembly by creating a **Mate**. The two instances are positioned by aligning a **Mate connector** defined on one instance with a Mate connector defined on the other instance.

To learn more about Mates, see "Mates" on page 320. To learn about Mates and Mate Connectors watch the video below.

Use the shortcut key **k** to hide/show mate connectors in an assembly.

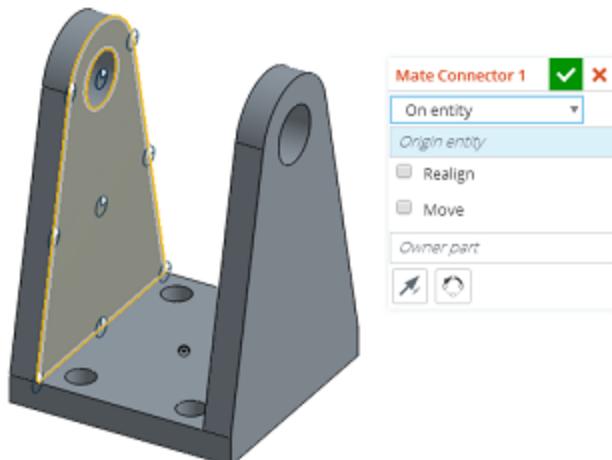
## Steps

1. Click

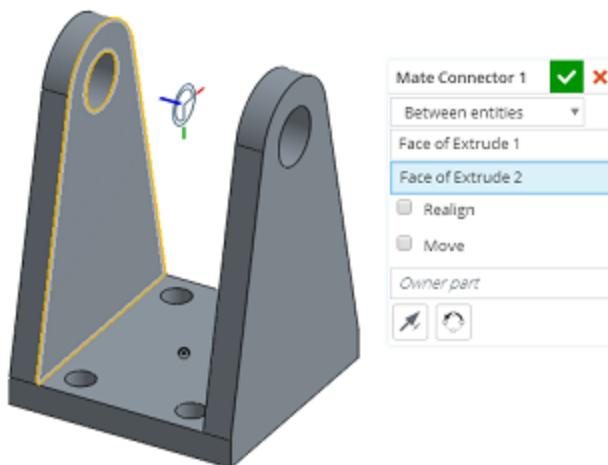


2. Choose between creating a mate connector **on** a part (entity) or **between** parts:

- **On entity** - Create a Mate connector on a part:



- **Between entities** - Create a Mate connector halfway between two entities on the part:



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 .

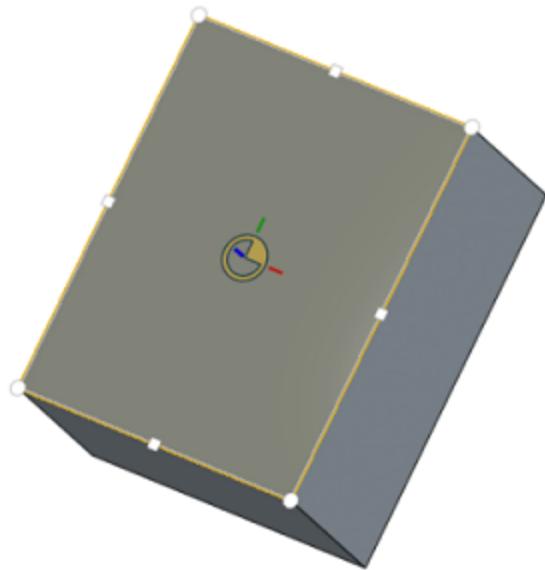
## Visualizing Mate connector points

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.

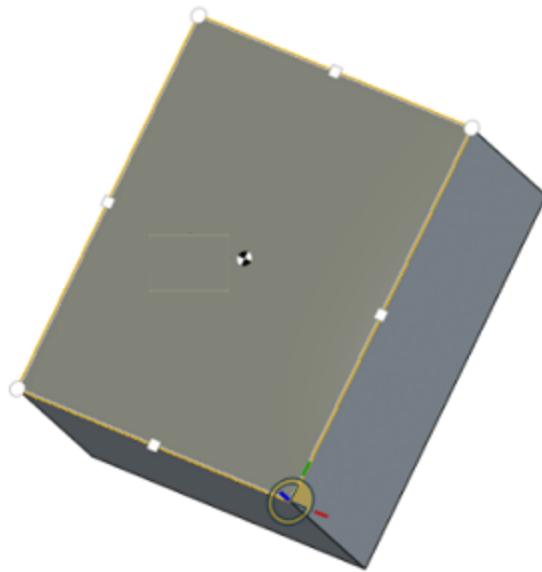
To lock mate inferences when you see the one you want to select, depress the Shift key when mousing.

Each face and edge of a part has default inference points:

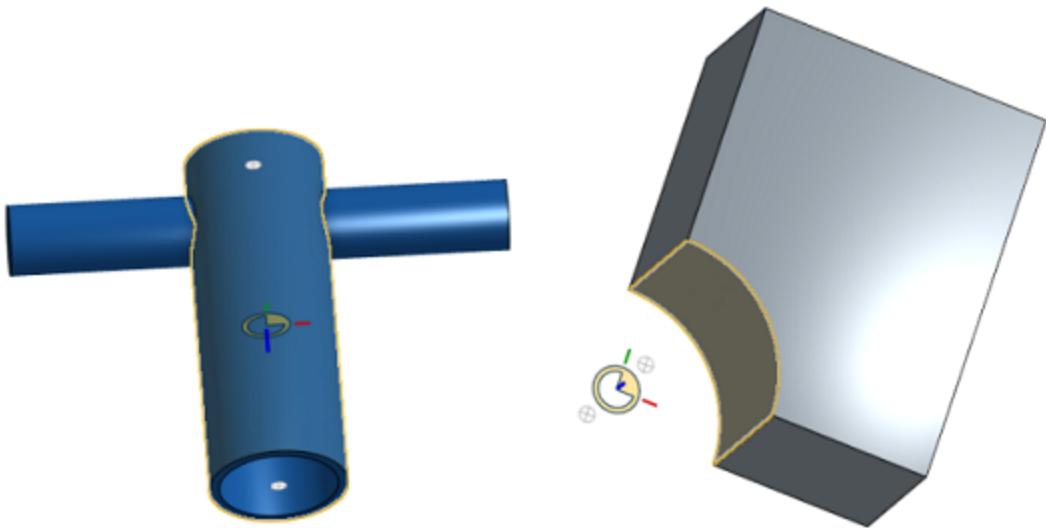
- At the centroid
- At the midpoints
- At the corners



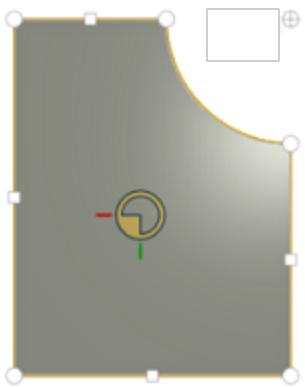
Before the default Mate connector is highlighted at the centroid (seen above), you might see the centroid point icon (seen below):



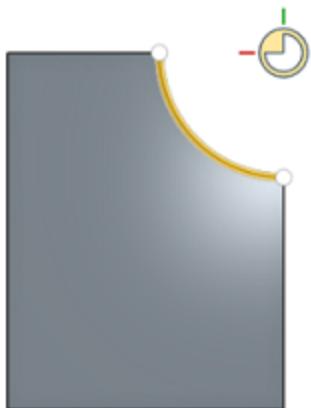
For cylindrical faces, inference points appear on the axis of the **cylindrical** and **partial cylindrical** face:



Select a planar face that has a partial cylindrical edge and the Mate connector inference points include the centroid of the axis:



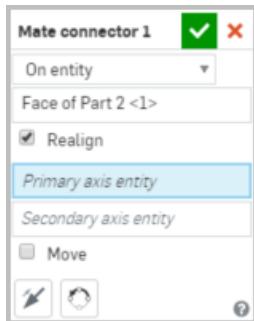
Hover over the edge of the partial cylindrical face and the default Mate connector appears at the centroid of the axis:



To zero in on a specific inferred point or default mate connector without waking up others as you move the cursor, you can use the SHIFT key to prevent other Mate connectors from appearing.

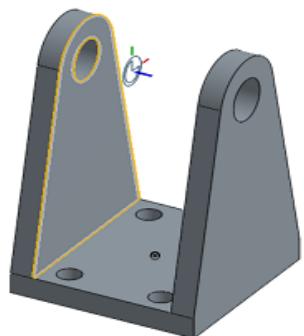
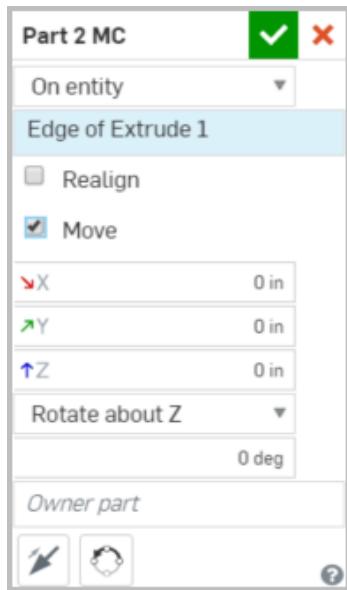
## Realign Mate connectors

Change the orientation of the Mate connector along a primary and (optionally) a secondary axis.



## Move Mate connectors

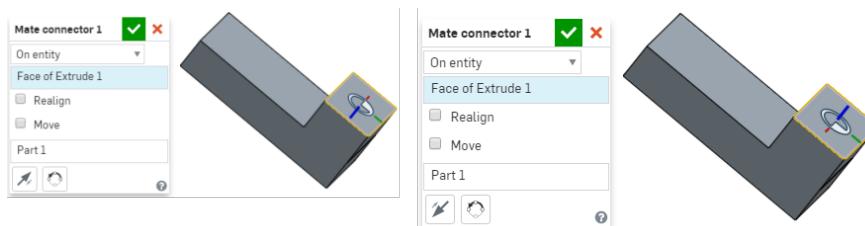
- **Move** - Move the Mate connector a specified distance in a specified direction. The fields are presented in this order:
  - X translation
  - Y translation
  - Z translation
  - You can also use the Rotate field to specify a rotation of a specified number of degrees.



You can use [expressions and trigonometric functions](#) in numeric fields in Assemblies.

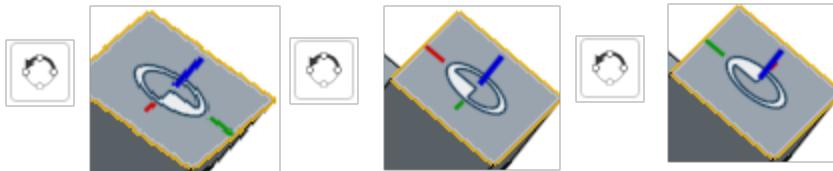
## Flip primary axis of Mate connector

Flip the primary axis 180 degrees.



Reorient secondary axis of Mate connector

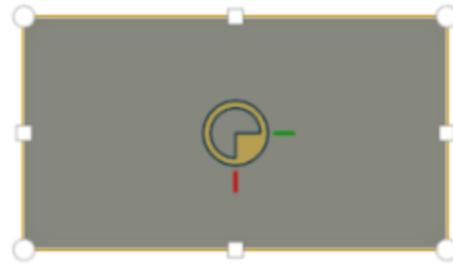
Move the primary axis one quadrant at a time through the X/Y coordinates.



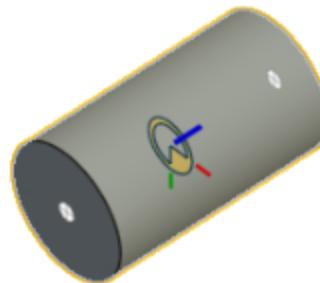
## Inference points and defaults

The inference points for potential Mate connectors available when you select an edge or face are:

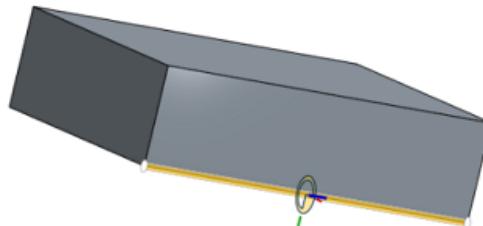
**Planar face** - Parallel to the face at every vertex, arc center, edge midpoint, and the face centroid



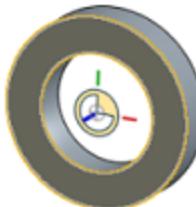
**Cylindrical face** - Perpendicular to the face axis at the middle and ends



**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



## Hiding and showing Mate connectors

Once created, you can 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 can view mate connectors in a Part Studio and keep them hidden in the Assembly, and vice versa.

## Tips

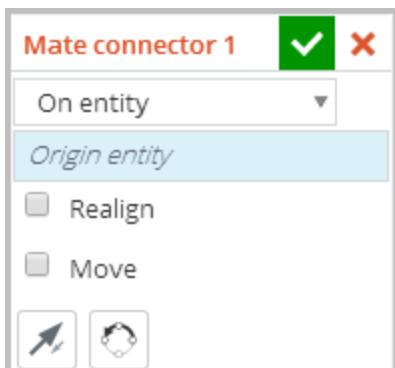
- If the behavior is not what you expected, try flipping the primary and/or secondary axis on the Mate connector.
- Use the SHIFT key to keep the mate connectors you want visible as you move the pointer to select one. This can be useful when the inferred point for potential Mate connector you want is 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 can be created in both the Assembly and the Part Studio. Creating a Mate connector in the Part Studio has two advantages:

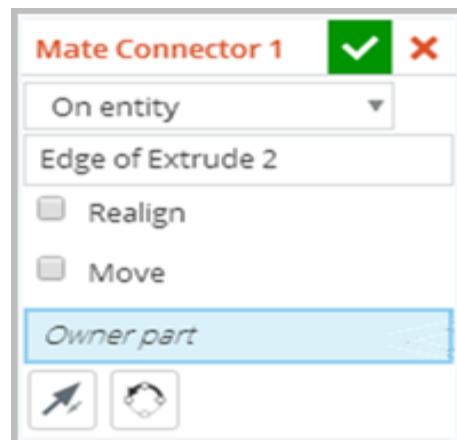
- You can reference sketch entities in the Part Studio. This gives you the ability to define the Mate connector in more positions than are possible in an Assembly.
- A Mate connector defined in a Part Studio is available for reuse on every instance of that part in every assembly in which it is instanced.

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 can be unclear which part owns the Mate connector. Use **Owner Part** to specify which part owns the Mate connector.



# Snap Mode

Shortcut: s



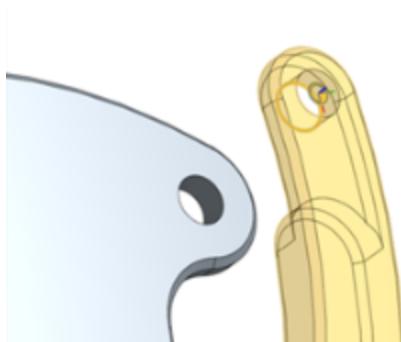
Snap mode can be toggled on or off, and when on allows you to snap one inferred Mate connector (or existing Mate connector) to another (without having to first create separate Mate connectors), opening a Mate dialog box so you can fine tune the placement.

## Steps



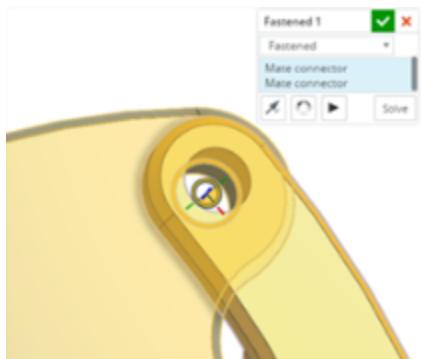
1. Click
2. Hover over a part to activate inferred Mate connectors.
3. Click on the desired Mate connector and drag to desired location.

When you start dragging the Mate connector, the part becomes transparent (to aid you in seeing the Mate connectors of the second part).



4. When you drag to the point of waking up another inferred Mate connector (during the drag) the cursor changes to show that the parts will snap together at those points when released. Upon release , a Mate dialog opens.

You can use the 'A' and 'Q' keys to change alignment during the Snap drag (in place of clicking the Secondary axis icon in the dialog).



5. At this point you can 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 can use the 'A' and 'Q' keys to change alignment during the Snap drag (in place of clicking secondary axis icon in the dialog).
- You can zoom and rotate the graphics area while the part is selected and in the process of dragging.
- When selected, parts become transparent so you can see where you're going with them - when dragging a part near another part, the second part's mate reference points become active/visible.
- As you are inserting a part instance into an assembly, you can pan and rotate as usual, even with Snap mode on.



# Replicate



Replicate takes a seed part or parts 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 part is then replicated and mated to that matching geometry.

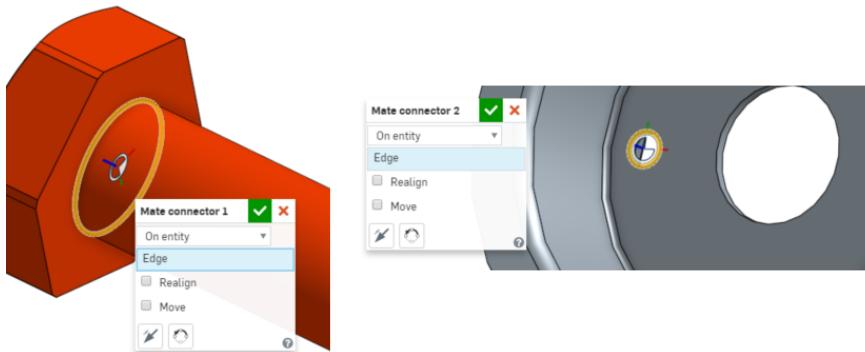
This feature makes completing an assembly and BOM very efficient due to the replication of parts and mates that would otherwise be inserted and assembled manually.

## Steps

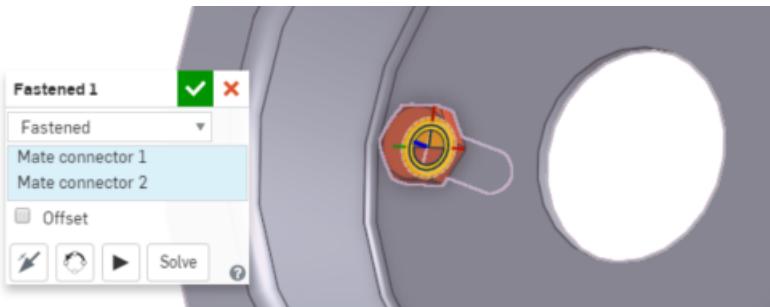
1. Open an assembly with relevant parts inserted.



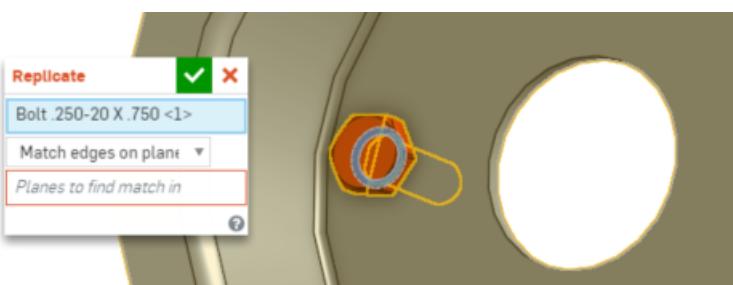
2. Make sure that the part you want to seed is already mated as desired.
3. In this example, first the bolt is mated to the hub, by way of a mate connector on the edge of the bolt and a mate connector on the edge of the hole on the hub:



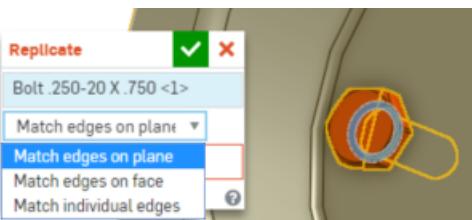
Since the hub will be seeded with replicated bolts, the mate between the bolt and the hub is referred to as an external mate (a seed instance must have only one external mate):



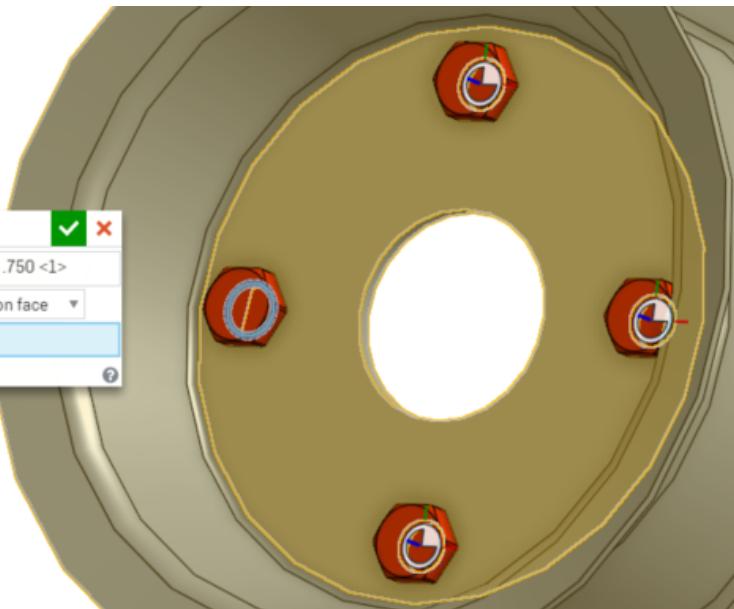
4. To replicate the bolt/mate to the hub, click .
5. For the Seed instances field, select the bolt:



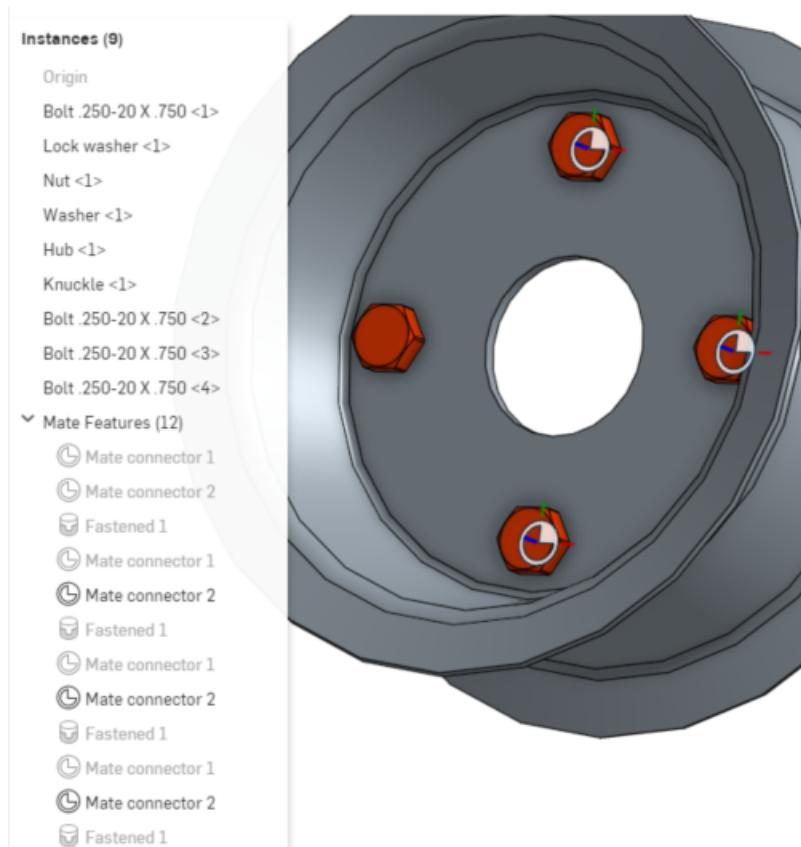
Because the bolt uses an edge for the mate connector, the choices in the Match scope field default to edges:



6. Select your choice (edges on plane, edges on face or individual edges).
7. Select the entity you chose above (plane, face or individual edges). This example shows a Match scope of face:



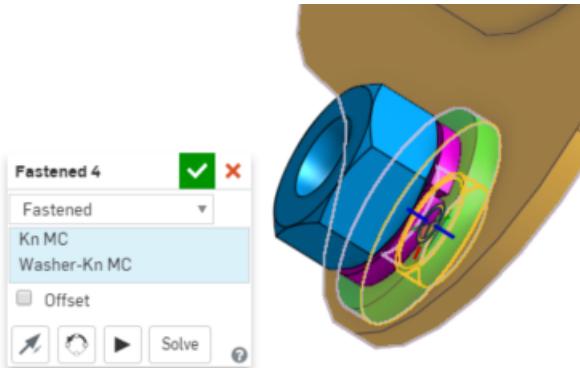
Note that there is no Replicate feature created. If you wish, you can use Undo to remove the actions just taken, or you can edit each feature individually.



This example illustrates seeding with multiple parts (washer/lock/nut):

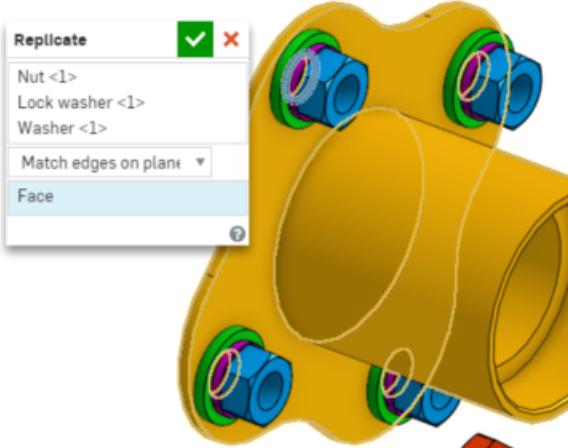
1. To start, the nut is mated to the lock washer; the lock washer is mated to the washer; the lock washer is mated to the knuckle.

There is only one external mate: the one to the part to which to mate the seed parts. In this case, the mate between the washer and the knuckle is the external mate.



2. Click .
3. In the *Seed instances* field, select the part 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):

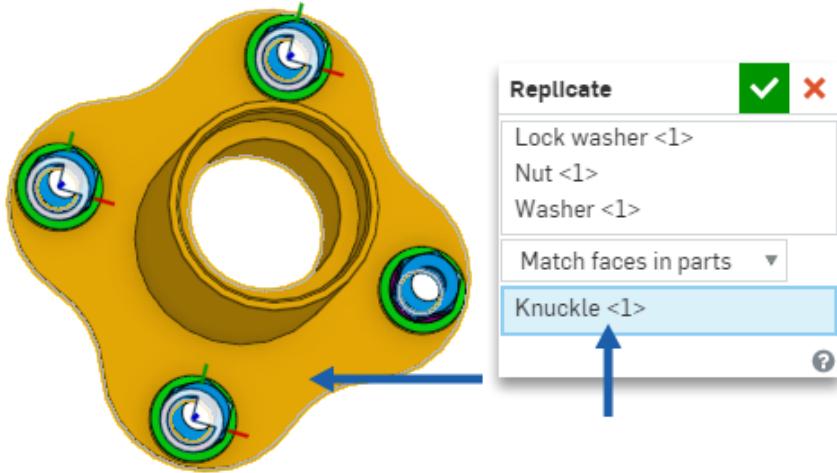


4. Select how to match the mate.

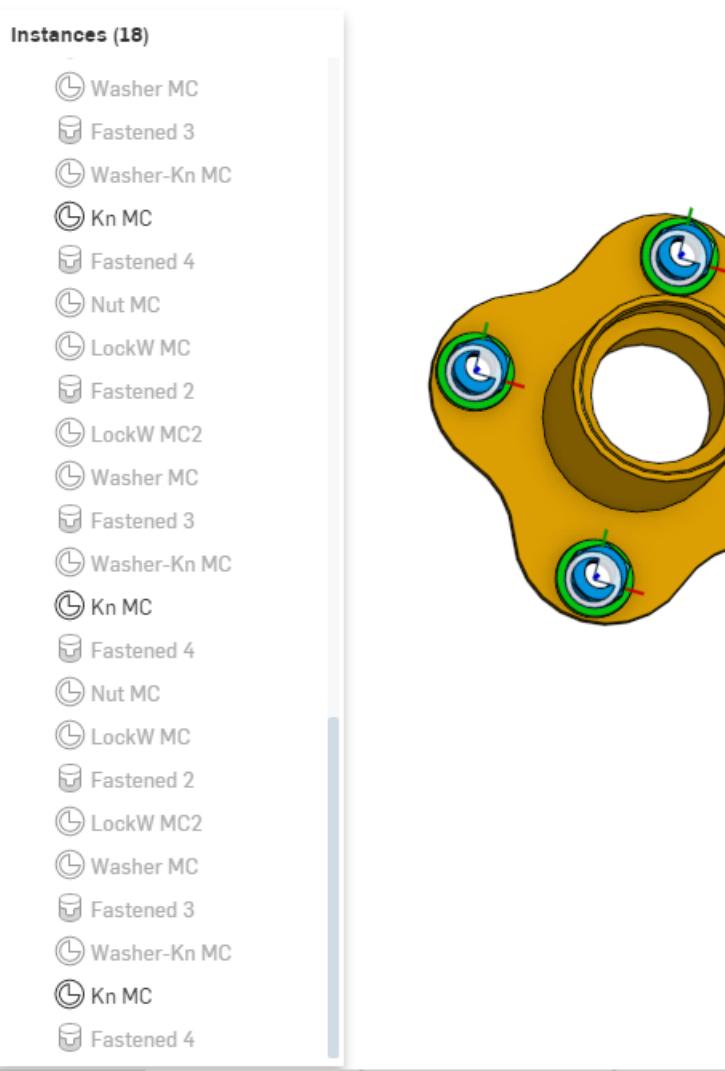
Because this example's seed instance is mated by the cylindrical faces of the holes, the choices are "faces". For more options for matching mates, see "Tips" on page 374 below.

5. Select the respective plane, face, or individual matches to make (depending on your choice above).

In this example, the selection is the planar face of the knuckle:



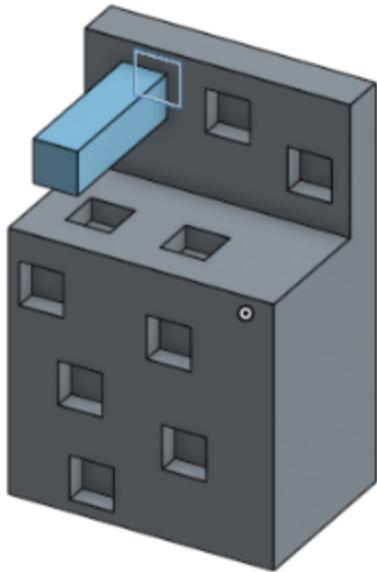
6. Notice that all relevant features are created and listed in the Feature list:



No Replicate feature is created in the Feature list. If you wish to make changes, you can use Undo to remove the actions just taken, or you can edit each feature individually.

## Face match scopes

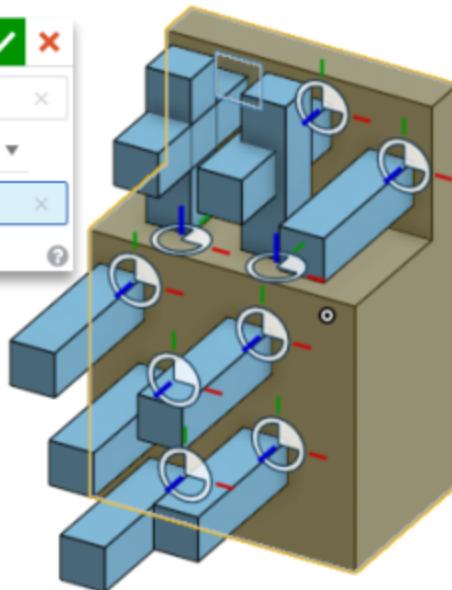
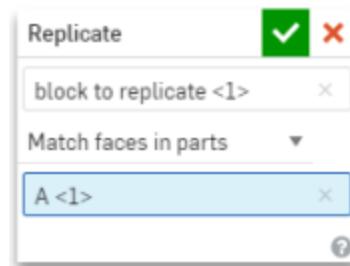
These examples use this model:



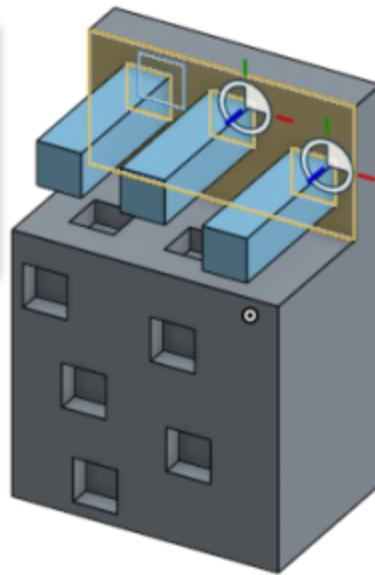
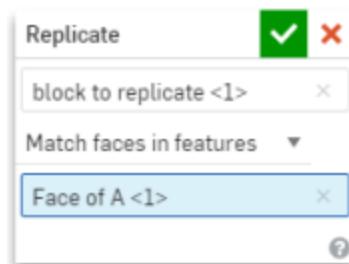
Scope:

Match faces in parts - Replicate the seed instance in the faces of the selected parts

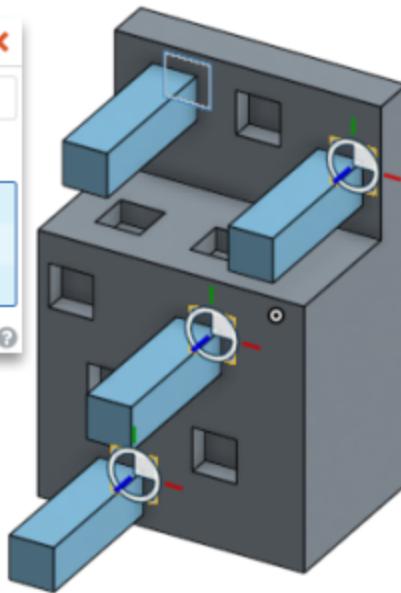
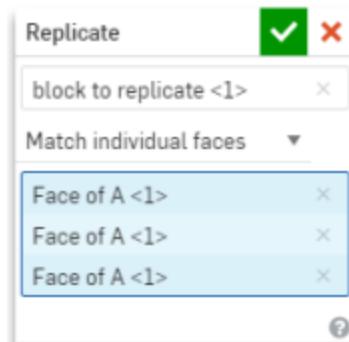
Example:



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



## 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 can be only one mate to the part onto which to replicate the seed instances.

# Relations



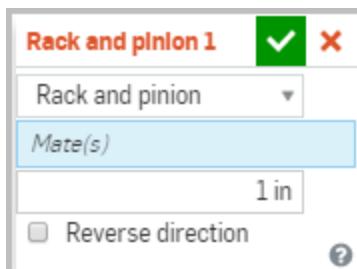
Use relations on mates to constrain degrees of freedom between mates. Onshape currently offers these types of relations:

- "Gear Relation" on page 377
- "Rack and Pinion Relation" on page 378
- "Screw Relation" on page 379
- "Linear Relation" on page 380

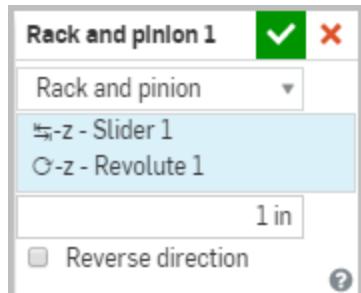
## Steps

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:

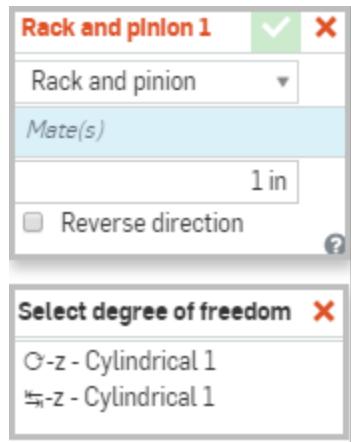


4. Select the required mates either on the model or from the Mate Features list.
5. Specify **degree of freedom for the mate**, if necessary:
  - a. When you select mates with the exact degrees of freedom required by the relation, Onshape displays the degree of freedom in the dialog for each mate:



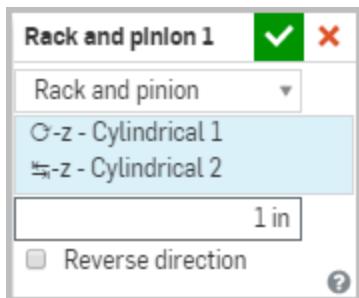
The icons indicate the specific degrees of freedom of each mate: linear movement along the Z axis for Slider, and revolving movement about the Z axis for Revolute.

- 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:



Above is the additional dialog for selecting the specific degree of freedom from the selected mate; in this case Cylindrical. This shows revolution about the Z axis and linear (translation) movement about the Z axis.

- Once the appropriate degrees of freedom are selected for both Mates, the dialog is populated and ready to be accepted:



- Enter any other required information and accept the dialog.

You can use [expressions and trigonometric functions](#) in numeric fields in Assemblies.



# Gear Relation



The Gear relation relates 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 sub-assemblies). 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 cannot 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

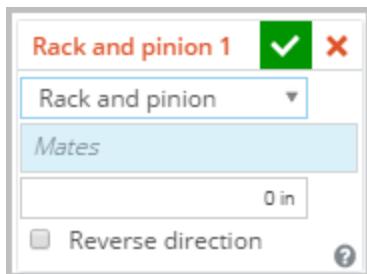


The Rack and Pinion relation relates a mate with a rotational degree of freedom to a mate with a linear degree of freedom.

## Steps

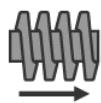
To add the Rack and Pinion relation:

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 sub-assemblies). 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.

Once you select a degree of freedom for the relation, you cannot change it unless you delete the mate from the dialog, change the mate type, or delete the mate and start over.



# Screw Relation

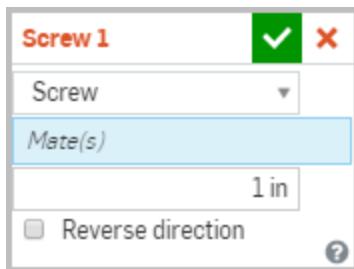


The Screw relation, which you apply to an existing mate, constrains the rotational degree of freedom in one cylindrical mate to the translational degree of freedom in the same cylindrical mate. One part rotates when the other part is translated, and vice versa.

## 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.

# Linear Relation

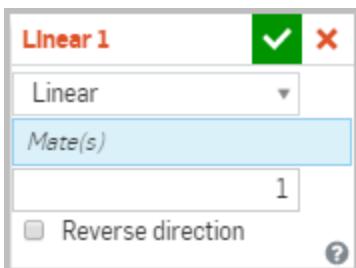


The Linear relation constrains the linear motion between two mates to change at a constant ratio. The first mate will move linearly in one direction as the other mate is moved linearly in one direction.

## 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 sub-assemblies). 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 cannot change it unless you delete the mate from the dialog, change the mate type, or delete the mate and start over.



# Group



Use Group to fix selected instances relative to one another. 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 the **Group tool**.

2. Select the parts to include in the group; pre-select is available.

You can click the Part name in the Feature list, click the part in the graphics area, or click and drag a selection box around parts in the graphics area.

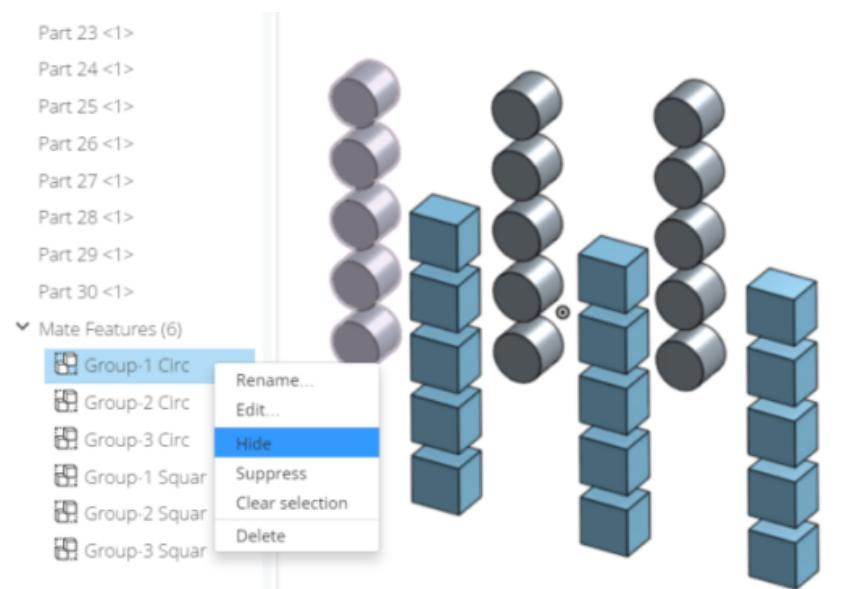
3. Accept ✓.

Notice that a Group feature is added to the Mate Features list in the Feature list box.

## Hiding and showing groups

You can hide (and show) groups from the context menu:

1. Right-click a group in the Feature list.
2. Select **Hide** from the context menu:



Part 24 <1>

Part 25 <1>

Part 26 <1>

Part 27 <1>

Part 28 <1>

Part 29 <1>

Part 30 <1>

▼ Mate Features (6)

Group-1 Circ

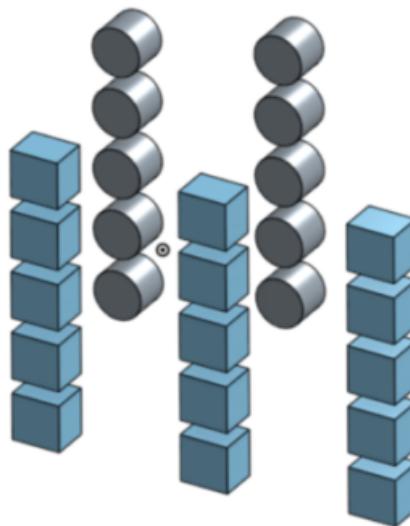
Group-2 Circ

Group-3 Circ

Group-1 Squar

Group-2 Squar

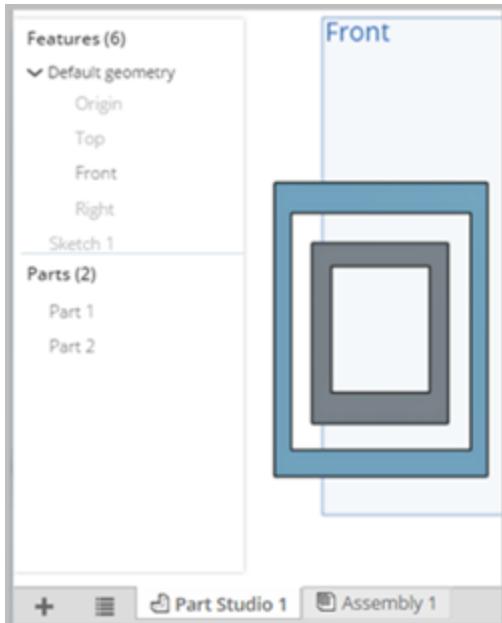
Group-3 Squar



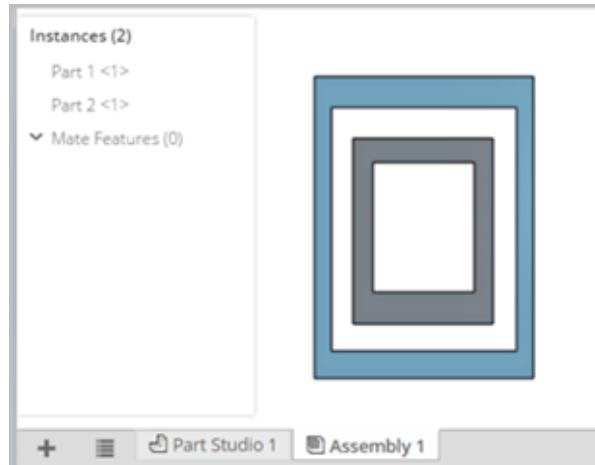
Follow the same procedure to show the group again, selecting **Show** from the context menu.

## Example

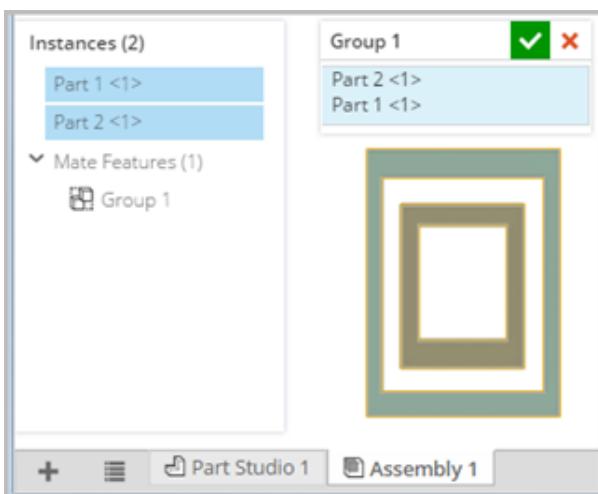
1. Two parts created in a Part Studio:



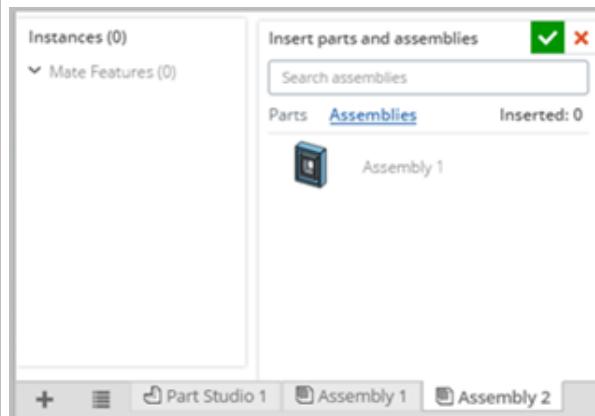
2. Both parts inserted into Assembly 1:



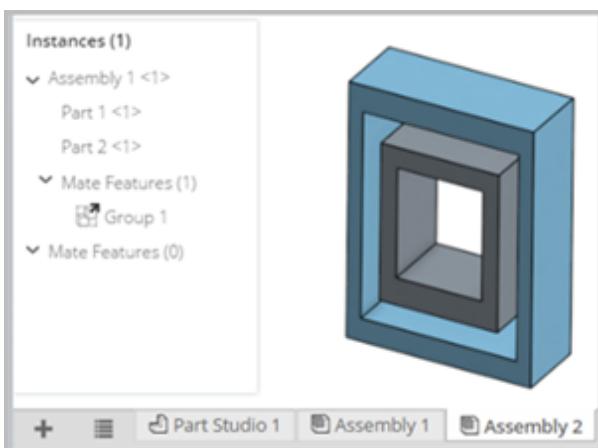
3. Both parts grouped in Assembly 1:



4. The Group being inserted into a new Assembly; notice the group is under the Assemblies heading in "Insert parts and assemblies" dialog:



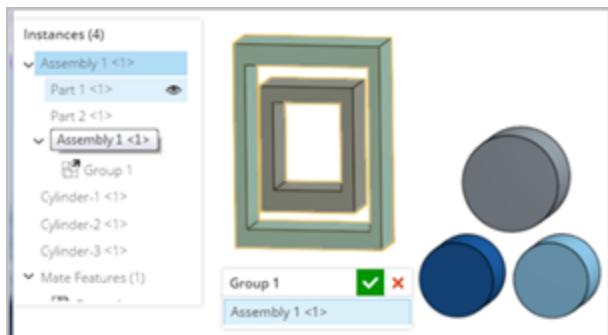
5. The Group in the new Assembly:



## Tips

- Despite the selection of child parts listed in the Feature list, the group moves and behaves as a group. The child parts cannot be acted upon individually.
- You can 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 can 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.

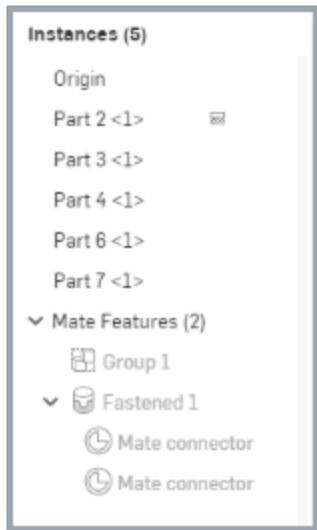


When Part 1 <1> is selected (above), the entire Assembly 1 <1> is selected by default.

- 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.

# Assembly Feature Lists

The Feature 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. (See Context Menus in Assemblies for more information.)



The Assembly Feature list consists of a list of Parts instances and a list of Mate features:

- 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.
- 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 can 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 .
  - **Fix** a part in place, right-click on the instance name and click **Fix**. (To remove the fix, right-click again and click **Unfix**.)
  - 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** a mate, part instance, or subassembly through the Feature list context menu or the context menu available on the feature in the graphics area.
  - **Drag** a part instance name or subassembly name in the Feature list to a new location in the list.

## Tips

- Drag and drop any assembly instance into (or out of) another, or drag it to the top level.
- You can also right-click on an assembly instance for more actions, including restructuring commands:
  - 'Move to new assembly' to create a new Assembly tab and insert this assembly into it automatically
  - 'Insert new subassembly' to reinsert this assembly into this same Assembly again

# Assembly Measure tool

The Onshape measure tool is available in Part Studio, for sketches and parts, and in Assemblies for parts and assemblies; it appears in the bottom right corner of the interface when a selection is made:



Select the part edges, faces, or mate connectors to obtain measure information about, then click the up triangle in the bottom right corner of the window to expand the measurement information (as shown above).

You can use the information displayed to enter values elsewhere in the system, for example, as a dimension.

With the Measurement dialog expanded, click and drag to highlight the value you want to copy:

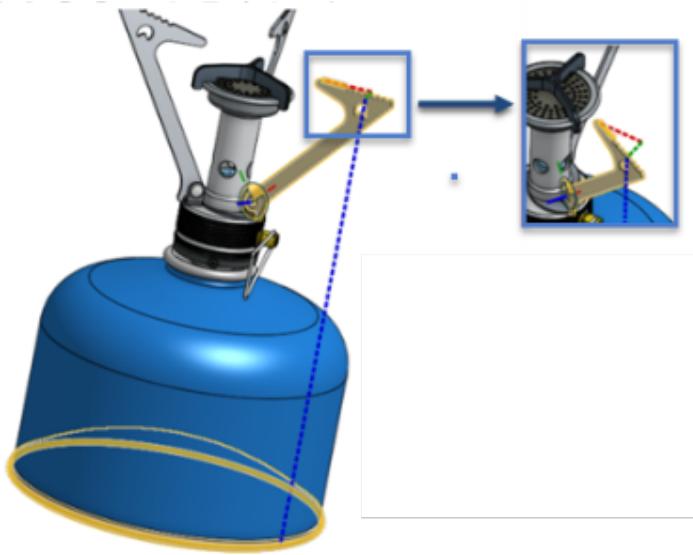
- Before releasing the mouse button when highlighting, use keyboard shortcuts to copy the value
- OR after releasing the mouse button when highlighting, open the Measurement dialog again (the value will still be highlighted) and use keyboard shortcuts to copy the value.

## 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:

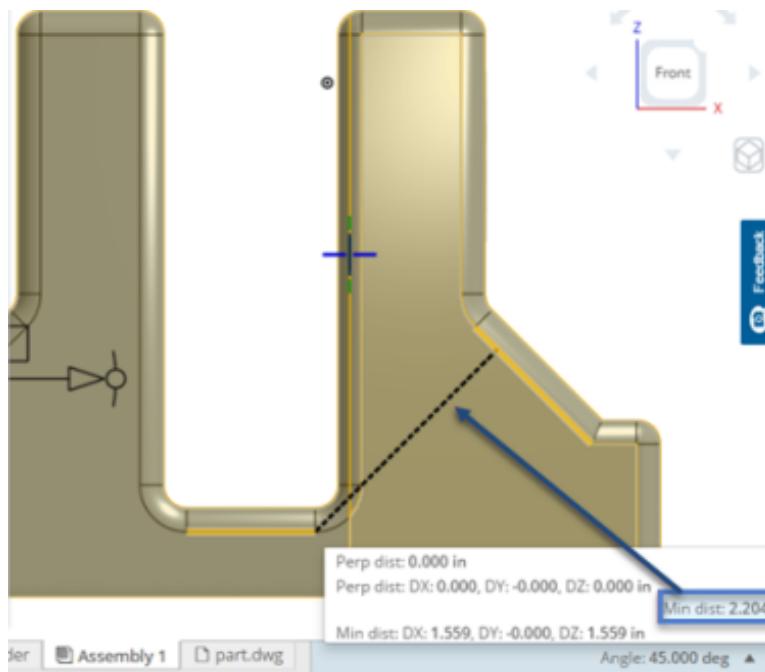
Minimum distances between entities are shown as bold dotted lines:

- 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

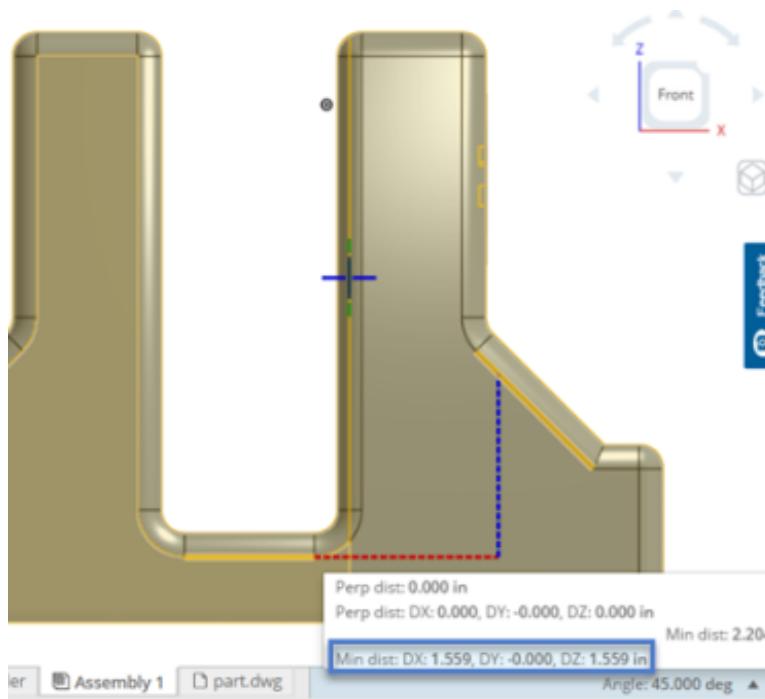


Note that when measuring to the center of a circle, you can select a planar face, edge, and edge of a cylinder

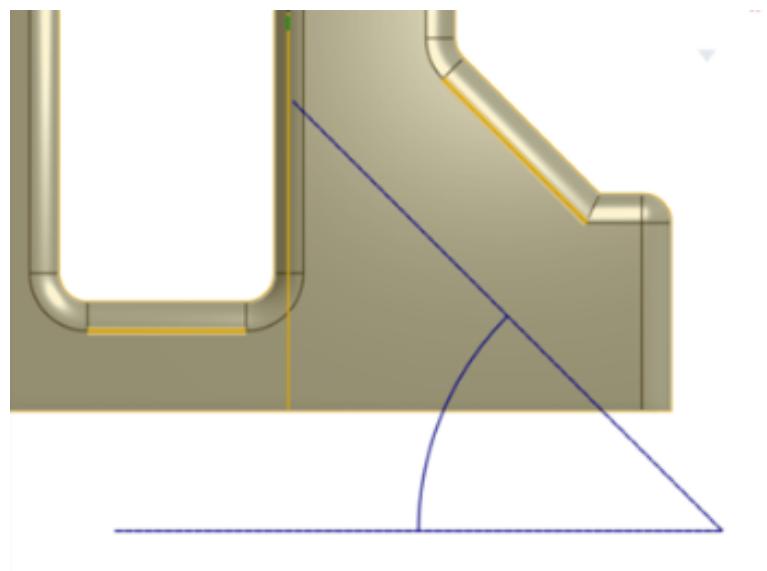
Minimum distance can be shown in two ways, such as:



and



Angles are shown as thin lines:



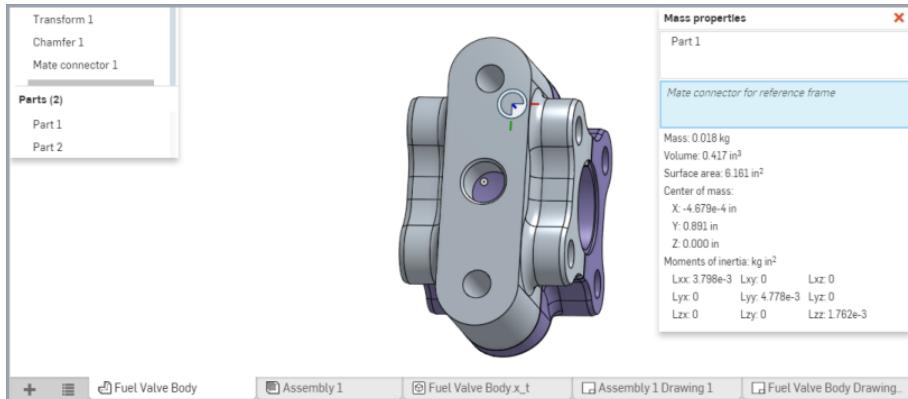
# Mass Properties Tool

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.

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 material assigned, no figure for that part is used in the calculation (and a note is displayed in the flyout to that effect).

Materials can be applied to parts through the context menu on a part in the Parts list (or the graphics area)

To access the Mass Properties dialog, select a part in the Parts list and click the small scale icon that appears in the bottom right corner of the interface:



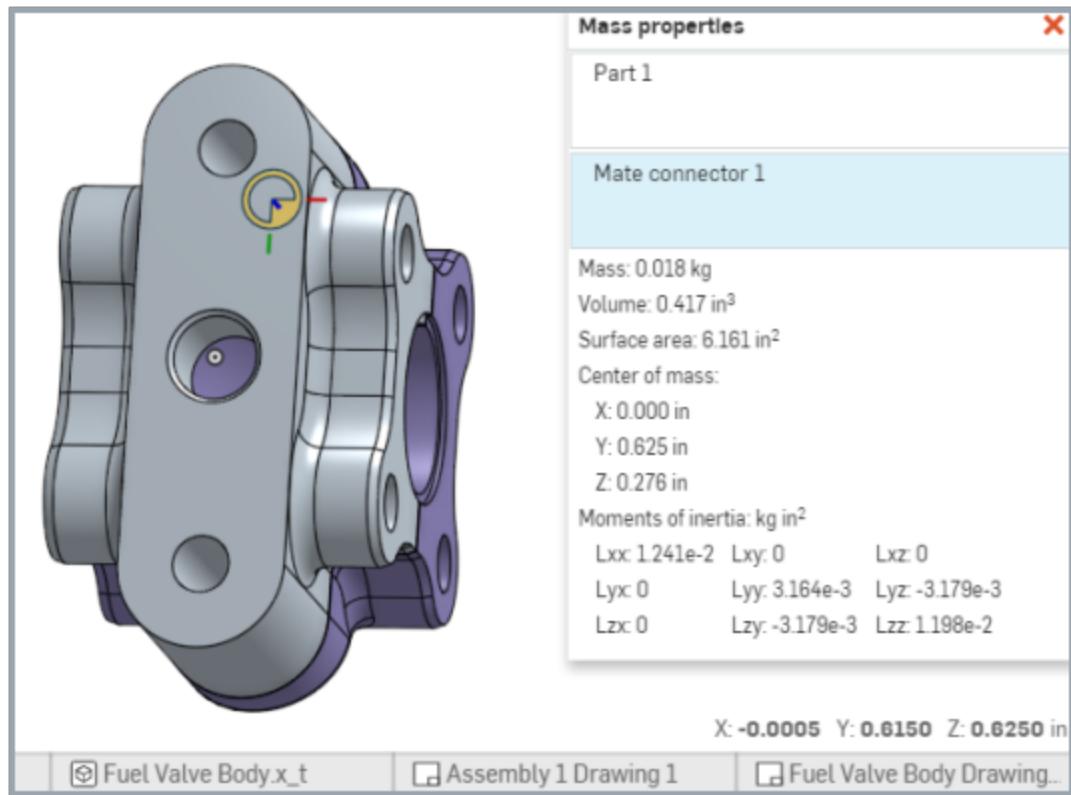
For any intersecting parts, the properties are calculated for each individual whole part and added together.

You can use the information displayed to enter values elsewhere in the system, for example, as a dimension:

With the Mass properties dialog expanded, double-click the value to copy and use shortcut keys to copy to clipboard.

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):



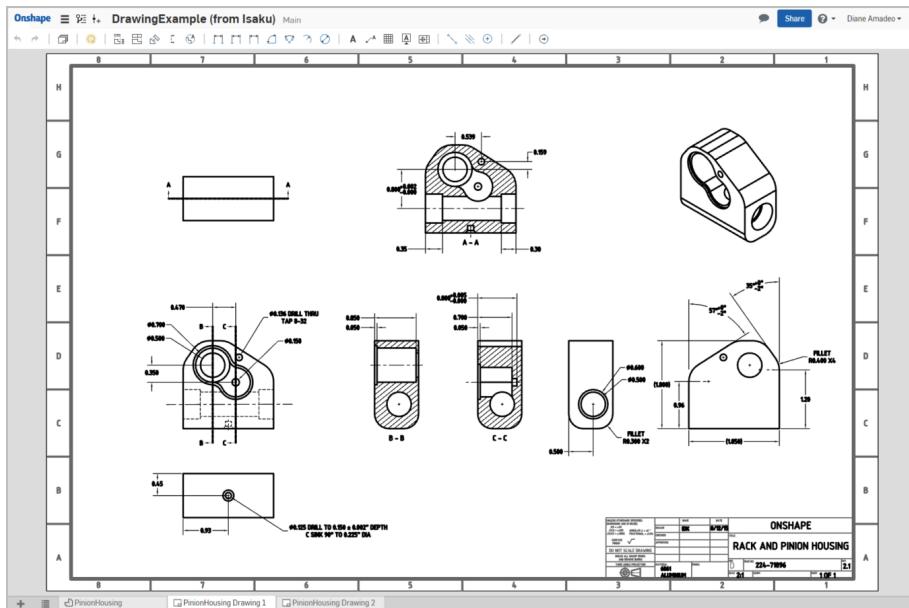
- Mass of all parts that have a material applied
- Volume of all selected parts
- Surface area of all selected parts
- Center of mass of all parts that have a material applied
- Moments of inertia - With respect to the common centroid of the selected parts (not the Part Studio origin) and reported using 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.

## Steps

1. While in a Part Studio (or Assembly), select a part (or many parts) from the Parts list (or Instances lists).
2. Click the Mass properties icon in the lower-right corner of the user interface to access the information in the flyout.

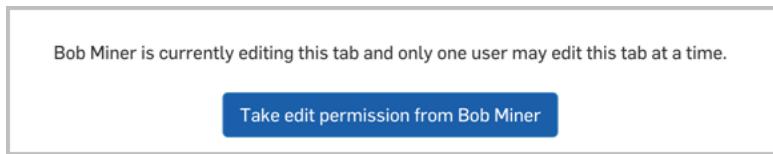
# Drawings

You can 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.



## Important

Keep in mind that for this release, 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 similar to this:



## Keyboard shortcuts

Shortcut	Action
<b>f</b>	Zoom to fit
<b>w</b>	Zoom window
<b>d</b>	Linear dimension
<b>Shift-r</b>	Radial dimension
<b>Shift-d</b>	Diameter dimension
<b>n</b>	Note annotation

Shortcut	Action
<b>Ctrl-q</b>	Update drawing
<b>I</b>	Line
<b>p</b>	Create Projected view
<b>Ctrl-s</b>	Display sheet menu
<b>PgDn</b>	Next sheet
<b>PgUp</b>	Previous sheet
<b>Home</b>	First sheet
<b>End</b>	Last sheet
<b>Delete</b>	Delete selected entity

# Drawing Basics



There are three ways to create a drawing:

- Using a part, Assembly, or Part Studio (presented below)
- As an [empty drawing](#), from scratch
- By "Importing a Drawing" on page 462 a .DWG or .DXF file

## Navigating within drawings

Navigating within drawings is similar to navigating within Part Studios and Assemblies, with a few exceptions:

### Windows

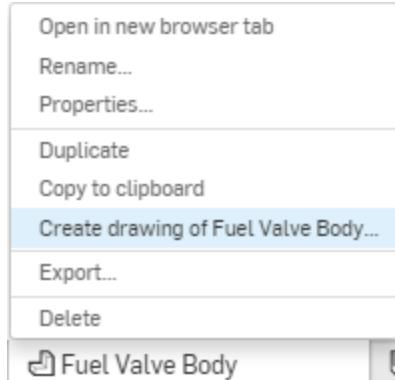
- **Mouse**
  - **Zoom in and out** - Scroll up and scroll down, respectively
  - **2D pan** - Middle-mouse-button click+drag, right-mouse-button click+drag, Ctrl-right-mouse-button click+drag
- **Touchpad**
  - Zoom in and out - Pinch out and pinch in, respectively
  - 2D pan - Ctrl-right-mouse-button+drag

### Apple Mac

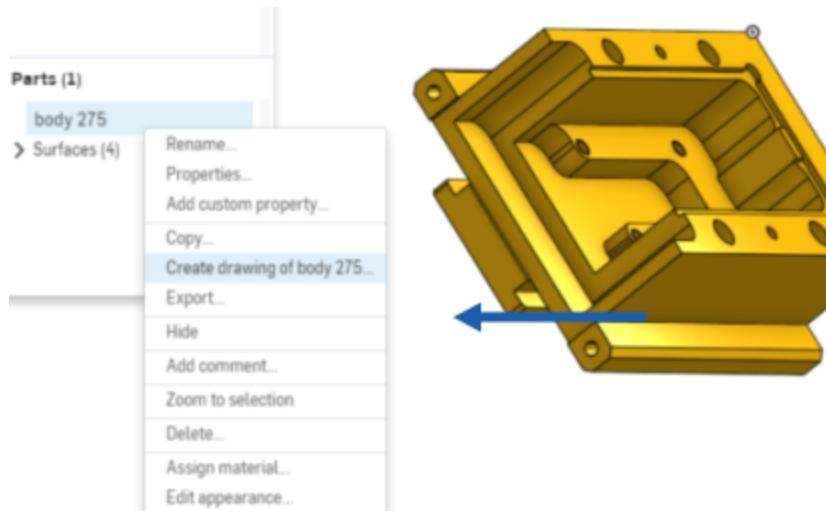
- **Mouse**
  - **Zoom in and out** - Scroll down and scroll up, respectively
  - **2D pan:**
    - Ctrl-right-mouse-button+drag
    - Middle button click+drag

## Basic workflow

1. **Create a drawing** of a part in a Part Studio, an entire Part Studio, or of a subassembly in the Assembly list:
  - a. Right-click on the name of the part in the Part list or assembly in the Assembly list.
    - To create a drawing of all parts in a Part Studio, use the tab context menu:



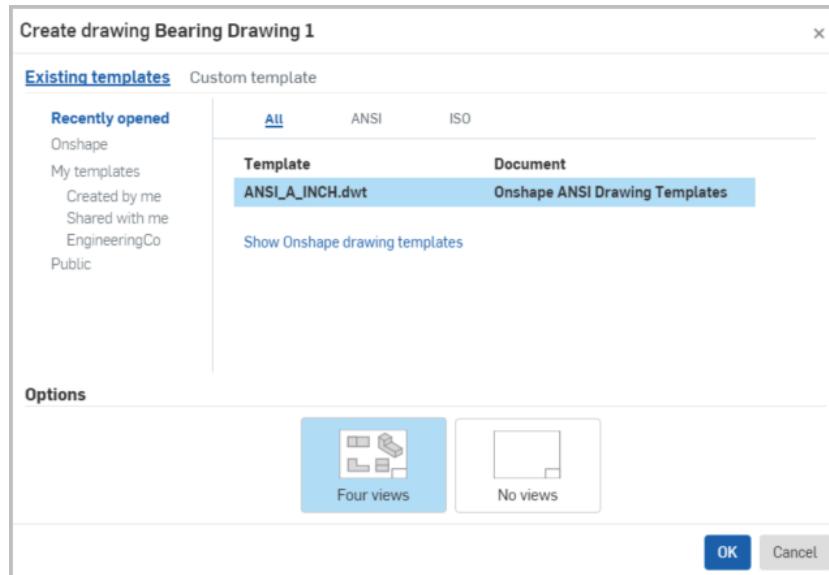
- b. Select **Create drawing of....**



- c. Choose a template.

Notice that you can 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 *My templates*.



You can also create your own "Custom Drawing Templates" on page 402.

- d. Click **OK**.

For more details on creating drawings, see "Creating a Drawing" on page 398.

2. Optionally, **create additional "Views"** on page 412. The drawing is created with default views and you can also create additional views.

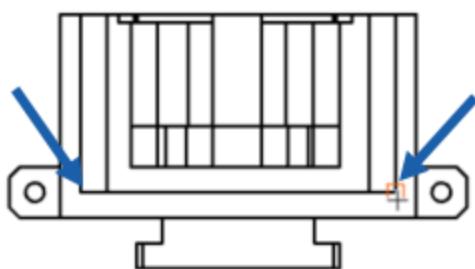
**3. Add dimensions:**

- Select a dimension tool.



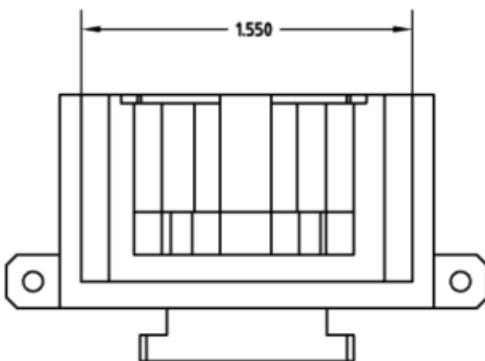
Notice that some of the tool icons have dots; these tools use snap points, the other tools use edges.

- Hover and then click when the appropriate snap points are visible (or select necessary edges).



The dimension text box appears on the click of the second snap point or edge.

- Drag dimension text box to desired location and click to place.



For more details, see "Dimensions" on page 430.

**4. Export to .DXF, .DWG, or PDF:**

- Right-click on the drawing tab.
- Select the preferred format.

Access the downloaded file on your local drive.

**5. Print the file:**

- Open the downloaded file in a compatible application.
- Print the file.

## 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:

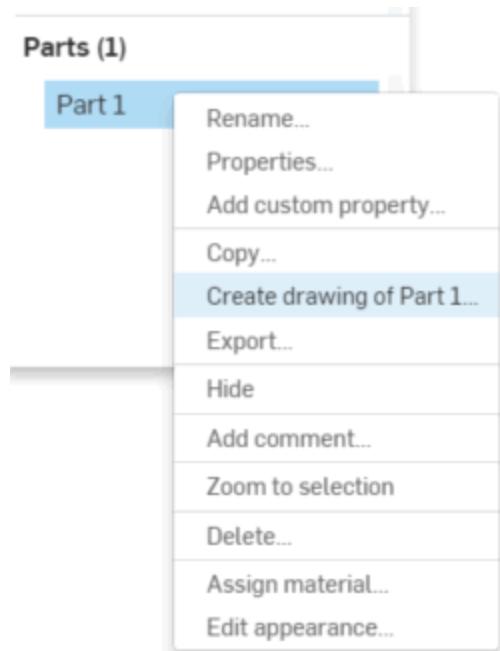
- + - Indicates a requirement to select a position
- ☒ - Indicates a requirement to select an entity

# Creating a Drawing

When you create a drawing of a part in a Part Studio, or in an Assembly, the drawing contains default views. You can also create an empty drawing using the  menu in the lower left corner of the window and select **Create Drawing...**

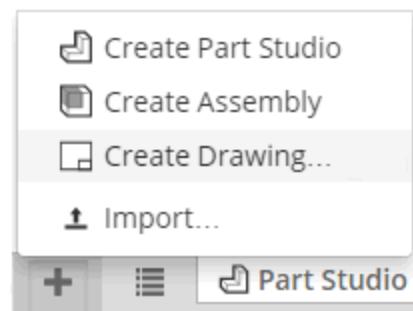
## Create drawing with default views

From the Parts list in a Part Studio or Assembly:



## Create empty drawing

From the Create tab menu:

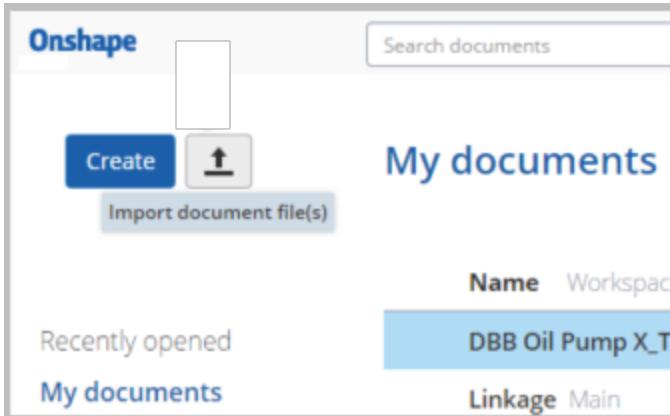


## Create from existing drawings files

Import an existing drawing file in .DWG or .DXF format. You can import from the Documents page and from within a document.

When importing from the Documents page, a new document is created:

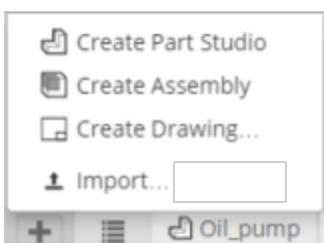
1. Click  (located next to the Create button):



2. Select the .DWG or .DXF file.
3. Select an owner for the document (if available).
4. 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.

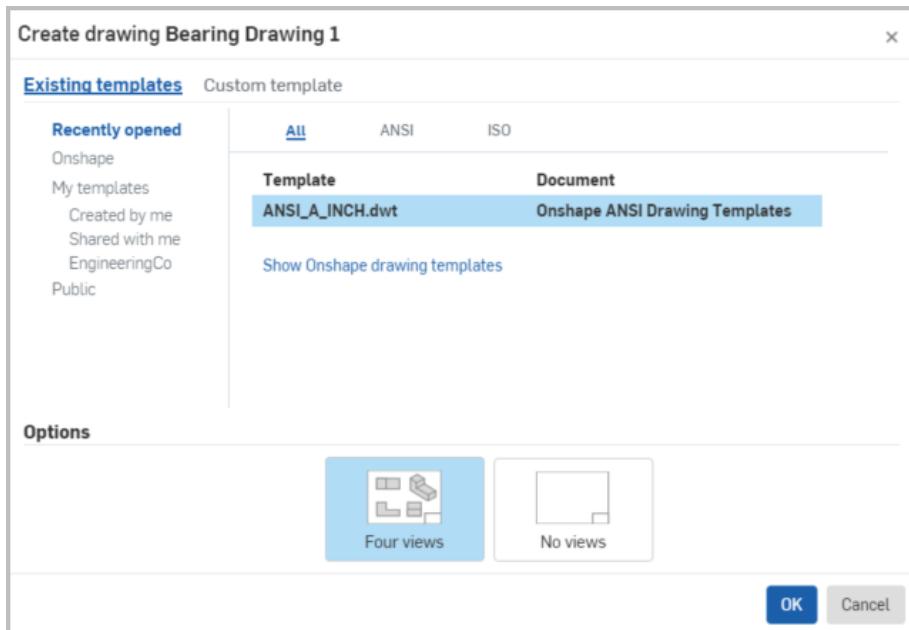


2. Select the .DWG or .DXF file.
3. Note the new tab in your document, with the same name as the file you imported.

## 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 can also create your own [custom templates](#). You can also specify whether to create four standard views or begin with no views.

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 can then:

1. Add more views, see "Views" on page 412.
2. Add dimensions, see "Dimensions" on page 430.
3. Add notes, see "Note" on page 447.
4. If the underlying geometry changes, you might want to update the drawing, see "Updating a Drawing" on page 461.
5. "Exporting a Drawing" on page 463.
6. Add more sheets, see "Sheets" on page 407.

# Custom Drawing Templates

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.

## Customizing a public template

If you need a custom drawing template, perhaps with your Company name on it, follow this procedure:

1. Sign in to your Onshape account.
2. On the Documents page, type **Templates** in the Search box.
3. The search results will include at least 2 documents owned by Onshape and containing drawing templates:  
For example, "Onshape ANSI Drawing Templates" and "Onshape ISO Drawing Templates".
4. Open the document containing the template you want to customize.
5. 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.

6. 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:

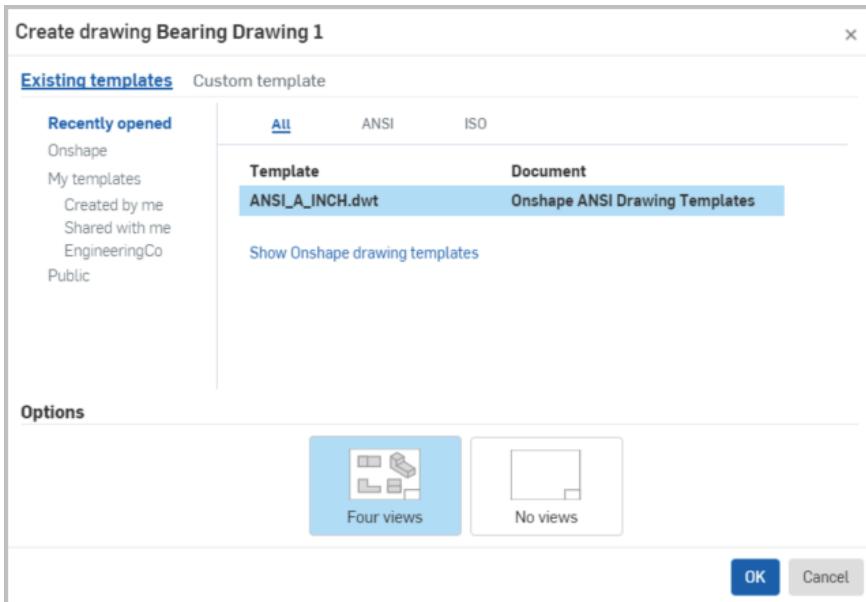
- There are 2 sheets in the DWT file - one for the first sheet of a drawing and a second sheet for continuation sheets in your drawing. You may need to edit both sheets.
  - The template contains many settings that are helpful when creating Onshape drawings. You'll generally see better behavior if you 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.
7. 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.
  8. To access your newly created custom template, create or open a new document in Onshape. This document will contain your custom templates.
  9. Click on the "+" menu in the lower left corner of the Onshape window and choose **Import** to import the DWT file you just saved. 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 can choose it as the template for your new drawing.

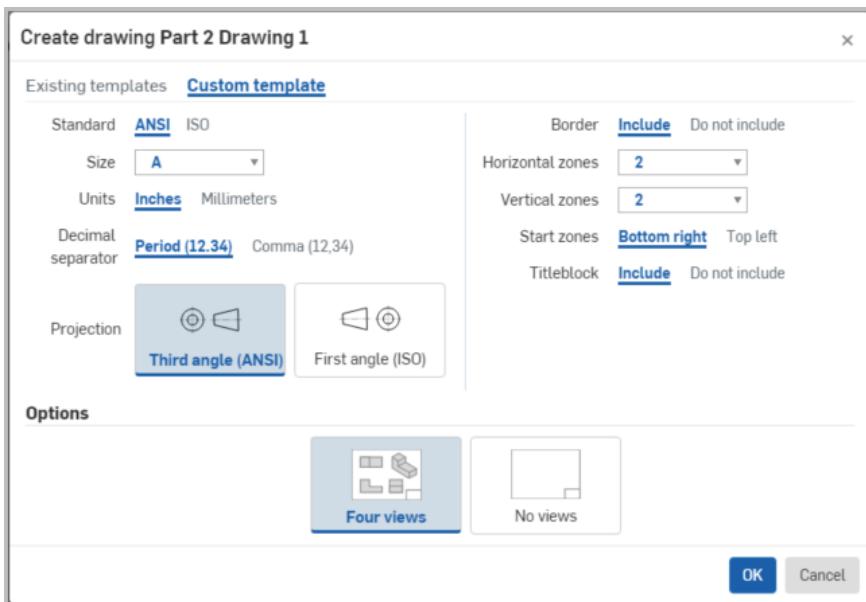
## Creating a custom template

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*:



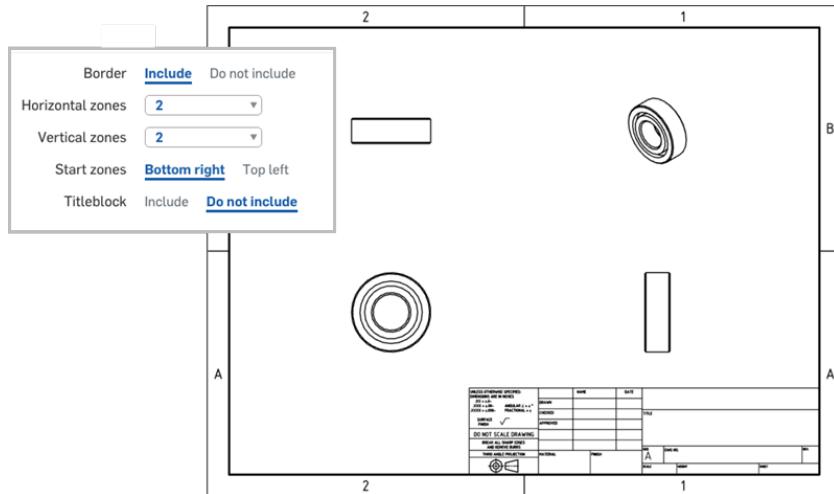
2. At the top of the dialog, select **Custom template** to access the *Custom template dialog*:



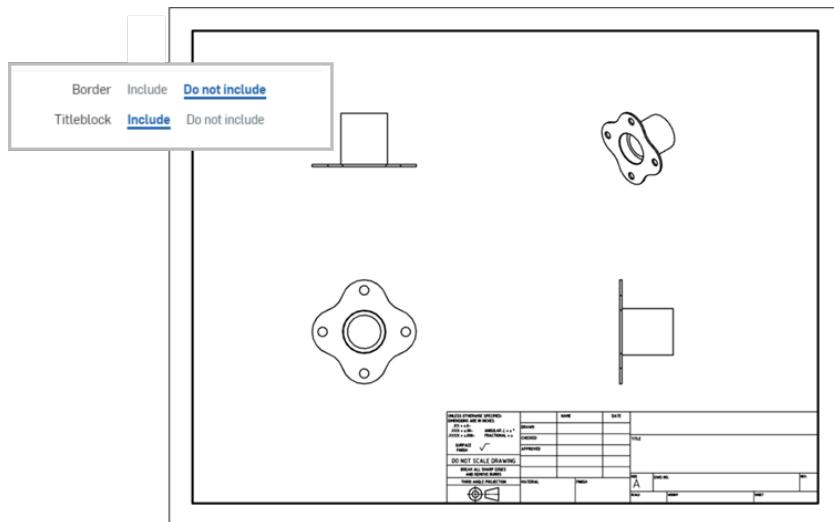
3. Design your template:
  - Standard - ANSI or ISO**
  - Size** - Choices are presented based on the Standard selected
  - Units** - Inches or Millimeters (defaults are by standard, but you can choose whatever you want)
  - Decimal separator** - Period or Comma (defaults are by standard, but you can choose whatever you want)
  - 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:

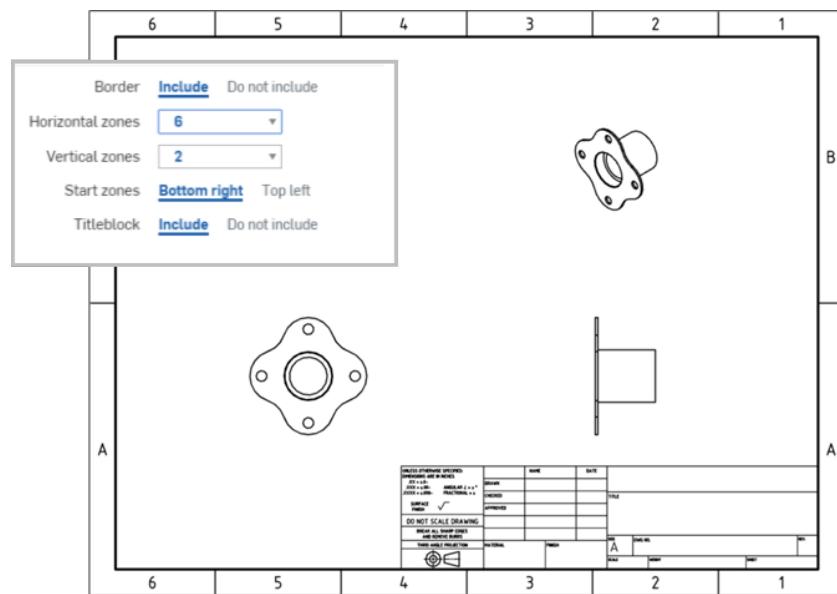
Drawing with a border:



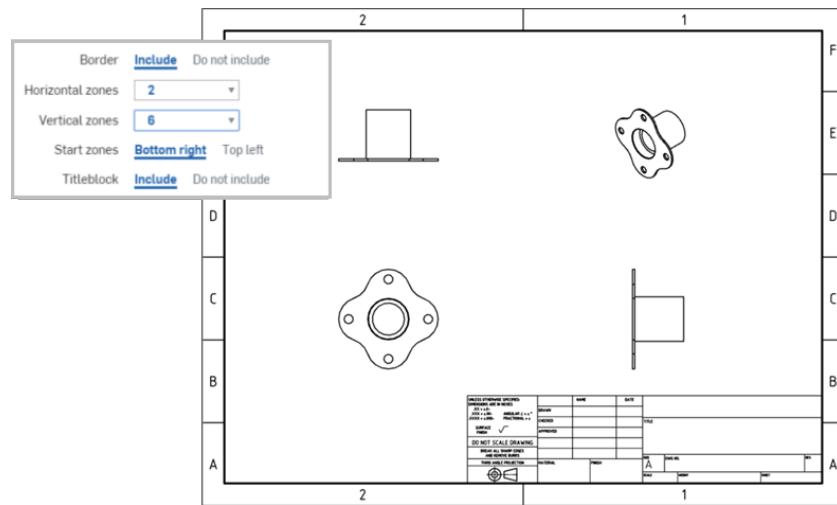
Drawing without a border:



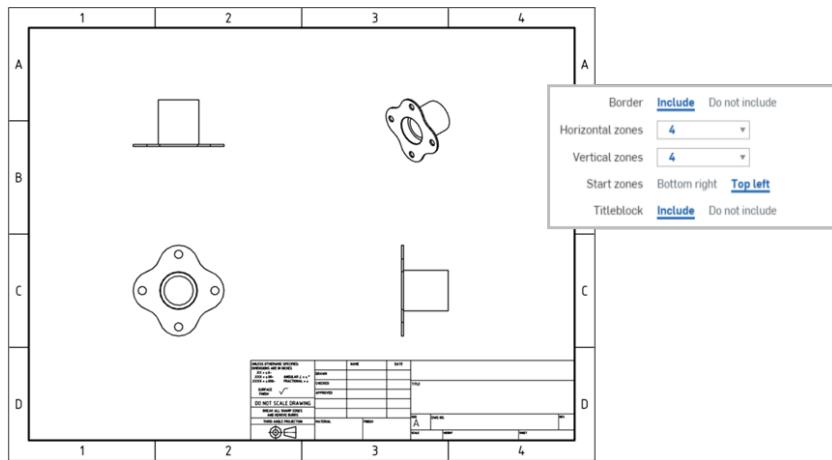
g. **Horizontal zones** - Specify the number of horizontal zones in the border:



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).



# Sheets

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.

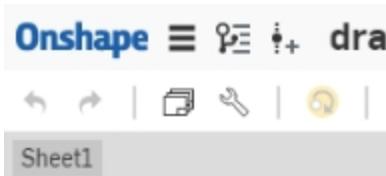
Once the sheet flyout is opened, it remains open with the currently displayed sheet selected in the list. To view another sheet, select it in the flyout, or use "Drawings" on page 393.

## Sheets shortcuts

Shortcut	Action
<b>Ctrl-s</b>	Open Sheet flyout menu
<b>PgDn</b>	Display next sheet
<b>PgUp</b>	Display previous sheet
<b>Home</b>	Display first sheet
<b>End</b>	Display last sheet

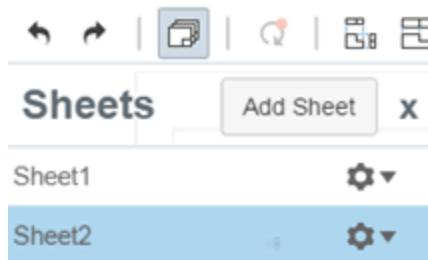
## Viewing and adding sheets

Onshape drawing templates contain multiple sheets: the main sheet and a continuation sheet. 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 .
2. Select the sheet you want to view.



When adding sheets, the additional sheet is added directly after the highlighted sheet in the flyout and is immediately displayed in the drawing area.

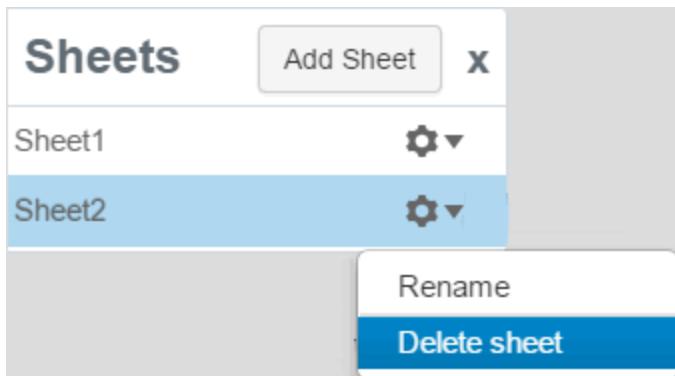
To create sheets:

1. Click .
2. Click **Add Sheet**.

## Deleting sheets

To delete a sheet:

1. Click .
2. Next to the sheet you want to delete, click .
3. Click **Delete**.

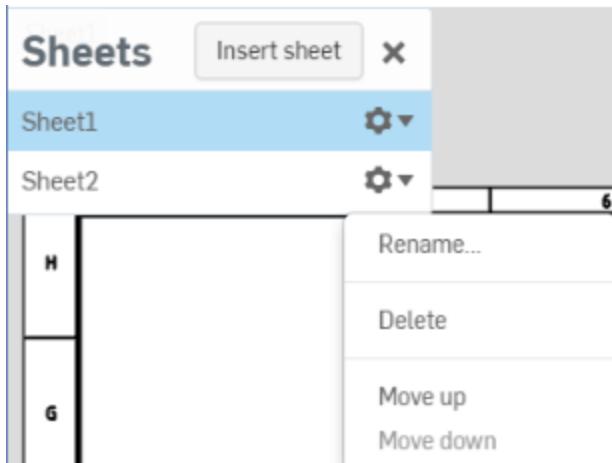


Deleted sheets can be restored by the **Undo**  command or by restoring a workspace at a point in time before the sheet was deleted.

## Reordering sheets

When you have more than one sheet in a drawing, you can reorder them:

1. Open the gear menu next to the sheet:
2. Select **Move up** to reorder the sheet above the sheet preceding it, or select **Move down** to reorder the sheet below the sheet following it:



## Renaming sheets

Renaming a sheet renames the sheet in the sheet flyout only. This does not affect the title of the sheet as specified in the Title block of a sheet.

1. Click
2. Next to the sheet you want to rename, click .
3. Click **Rename**.

## Editing title blocks

All of the Onshape-supplied drawings templates have the following automatic referencing between the drawing's properties and the title block:

UNLESS OTHERWISE SPECIFIED, DIMENSIONS ARE IN INCHES  XX = ±0. XXX = ±.00. XXXX = ±.000.  SURFACE FINISH ✓		NAME	DATE	Part 1 Drawing 1		
DRAWN	Diane	84/01/2016				TITLE
CHECKED						
APPROVED						
DO NOT SCALE DRAWING BREAK ALL SHARP EDGES AND REMOVE BURRS		Drawing description				
THIRD ANGLE PROJECTION	MATERIAL	FINISH	SIZE A	DWG NO. abc123-22	REV. 3	
			SCALE	WEIGHT	SHEET	1 of 1

- Property: Nickname = Title block: Drawn, Name
- Property: Created date = Title block: Drawn, Date
- Property: Part Number = Title block: Part Number
- Property: Description = Title block: Title
- Property: Revision = Title block: Revision
- 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 can edit the fields in the title block as you normally would: drag, copy, and paste. You can also 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.

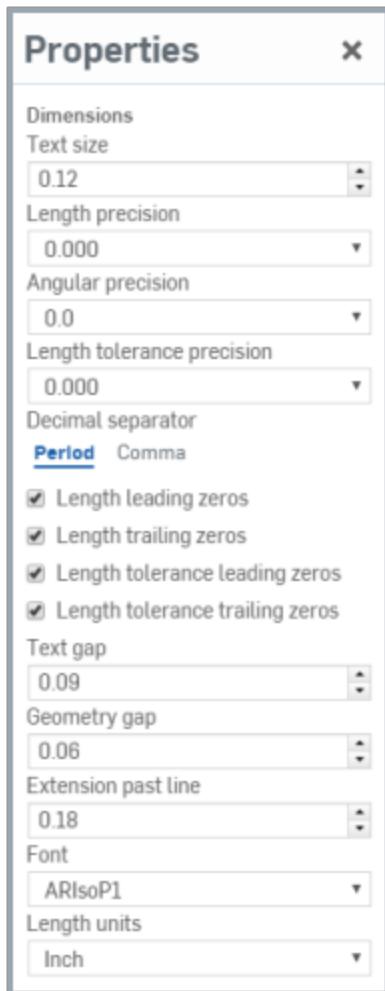


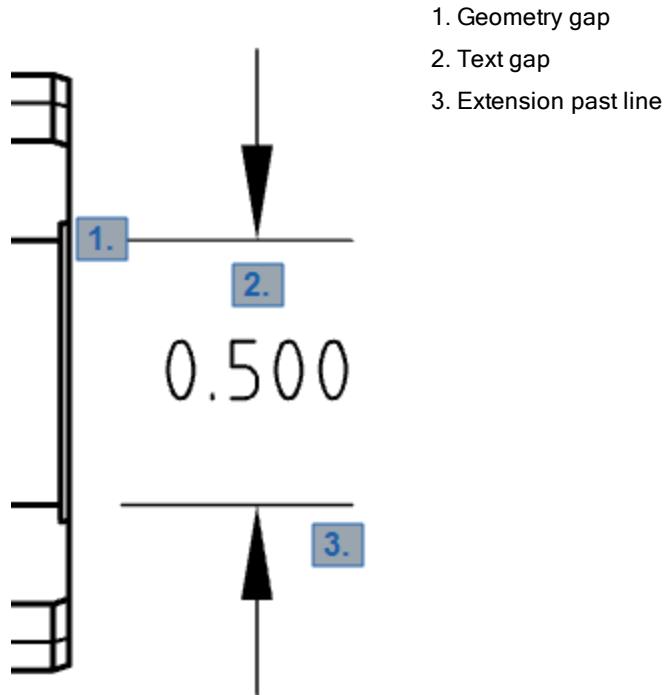
# Properties



Set the specifications for your drawing dimensions in one place in order to simplify formatting. Any modifications made through the Dimension panel (for a specific dimension) prior to or after being made in this flyout, are not overridden by the flyout changes. That is, specifications made in the Dimension panel for a specific dimension always take precedence over any change made through this Properties flyout.

Text settings made here will be reflected in the Notes (and Notes with Leader) commands the next time those panels are opened. Changes made to the specifications in those panels are not reflected in this Properties flyout, however.





## Tips

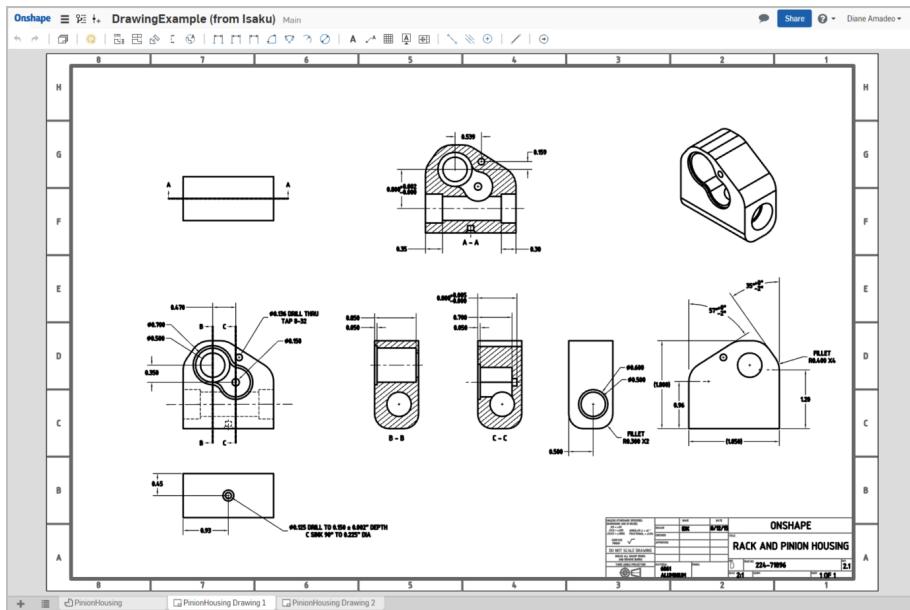
- These settings apply to only dimensions and only in the current drawing. (That is, these settings do not apply to Notes.)
- Making changes in this flyout changes all existing dimensions properties in the drawing, unless you used the Dimension palette to specify settings.

# Views



When you create a drawing from a part or subassembly, you can create it without any views 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, but you can also use a custom template and select the projection.

For example, a standard ANSI drawing may look like this:



All views of a part in a drawing are from the same version of the part. When creating a view (drawing, projected, auxiliary, section) the same part version used is 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:

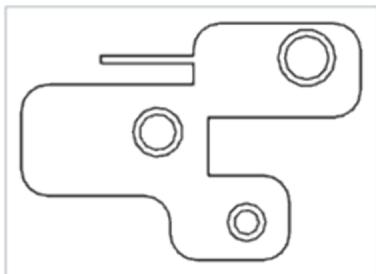
Projected View	
Can be created from:	Can't be created from:
Base views	Auxiliary views
Projected views	Section views
	Detail views
Auxiliary View	
Can be created from:	Can't be created from:

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:	Can't be created from:
Positions in base views	Auxiliary views
Positions in projected orthographic views	Section views
Positions in isometric views	Detail views
	Cut line tangent to cylindrical face
Detail View	
Can be created from:	Can't 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	

## Drawing view

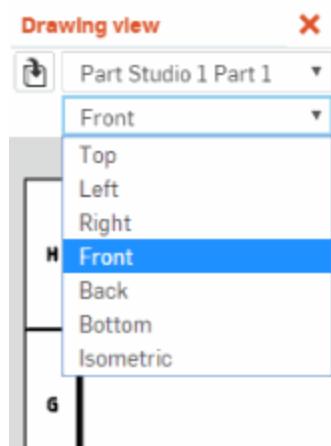


Place a view of the model on the active sheet; use the dialog to select the desired part and orientation. By default, the label and scale are off. To see the scale, double-click the view: the scale dialog opens to the top left of the drawing.



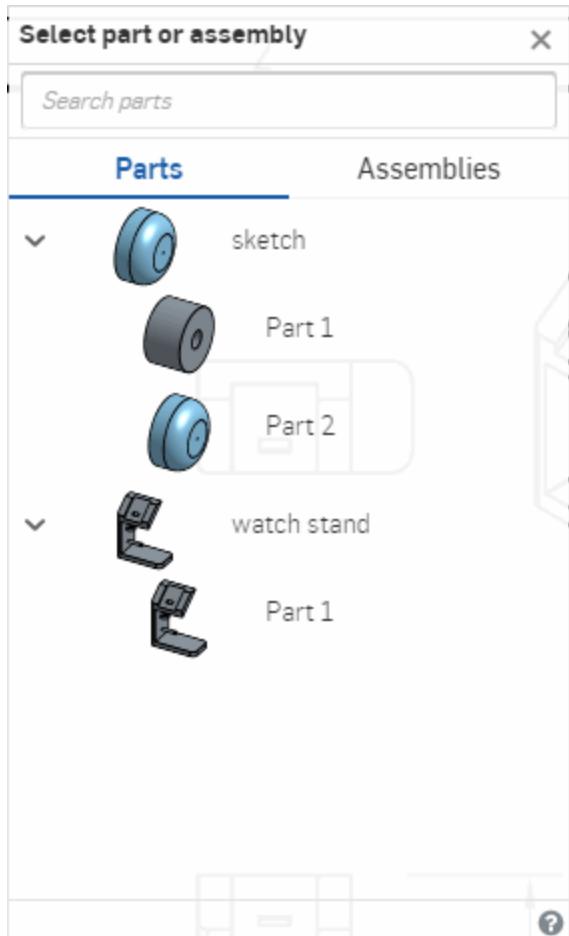


1. Click .
2. In the dialog, either:
  - a. Use the drop down to select a part and then a view:



- b. Use the Insert Part or Assembly icon to search the document (Part Studios and Assemblies) for parts or assemblies:

- i. Click :
- ii. Select from the list:



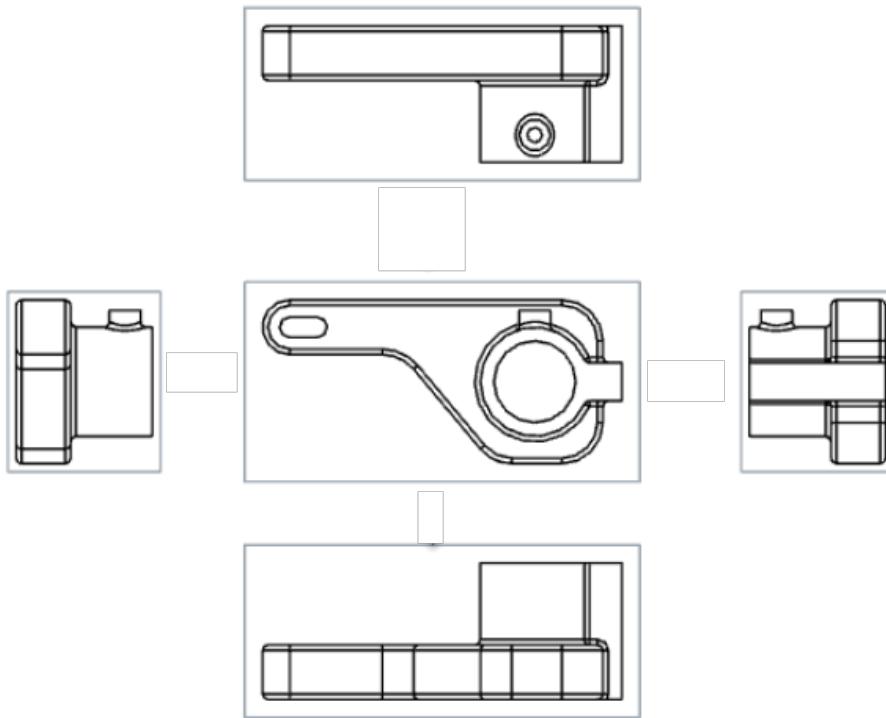
3. Click to place the view.

## Projected view

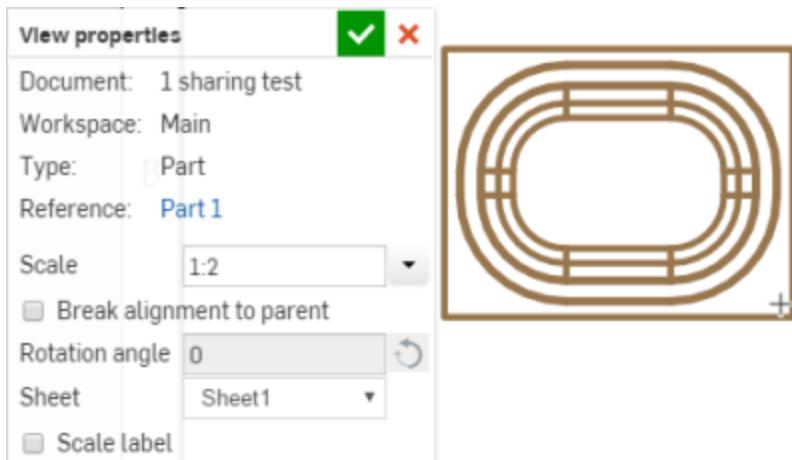


Shortcut: p

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 scale dialog opens to the top left of the drawing.



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. Double-click to edit the view, including setting the Scale and including a Scale label:



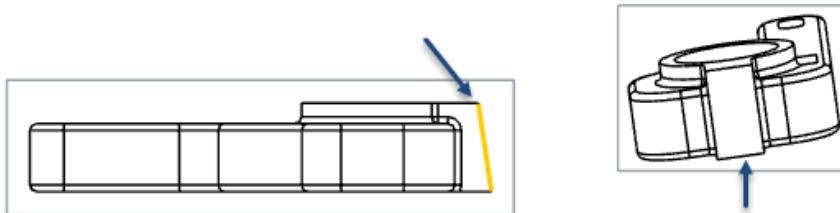
Note that the "View properties" on page 426 dialog formats all input into an N:N or N/N format. For user input values, the second digit or denominator is always set to 1, and you can double-click the Scale label to edit it.

## Auxiliary view

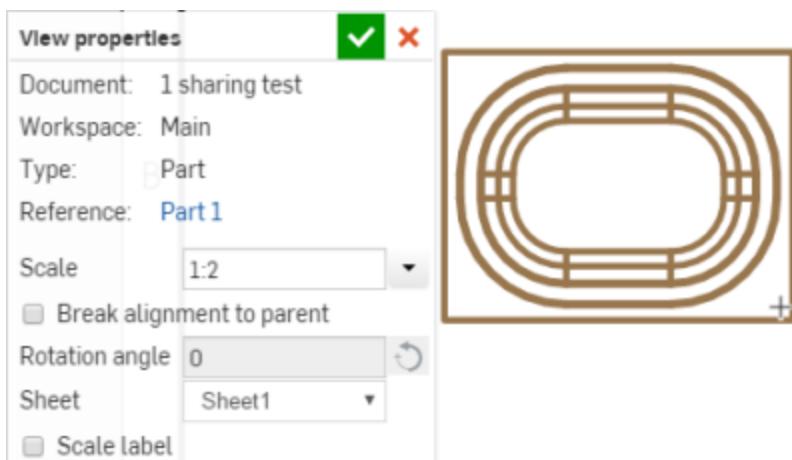


Create an auxiliary view; an orthographic view that is folded out 90 degrees from a selected edge in the parent view (usually from a slanted edge). By default, the label and scale are off. To see the scale, double-click the view: the scale dialog opens to the top left of the drawing.

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 edit the "View properties" on page 426, including setting the Scale and including a Scale label:



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.

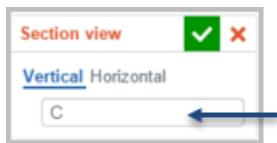
## Section view



Create a section view of an existing view by placing a cutting plane line and specifying a direction and label. Keep in mind that you cannot create section views from: auxiliary, detail, or other section views. By default, the label is on and the scale is off. To see the scale, double-click the view: the scale dialog opens to the top left of the drawing.

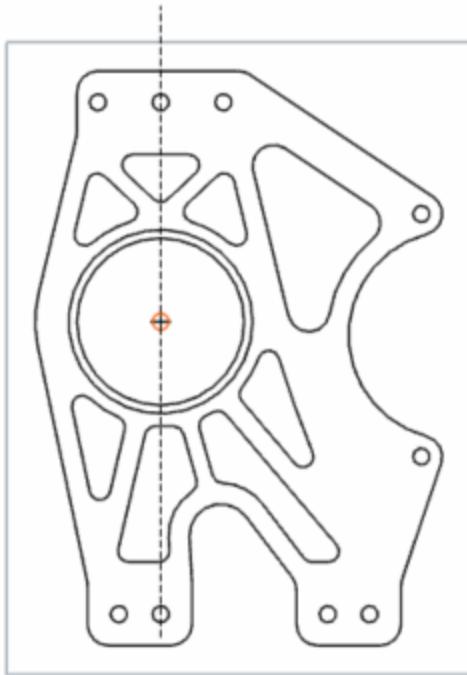
1. Click
2. Select **Vertical** or **Horizontal** in the dialog:

3. Optionally supply a label for the view:



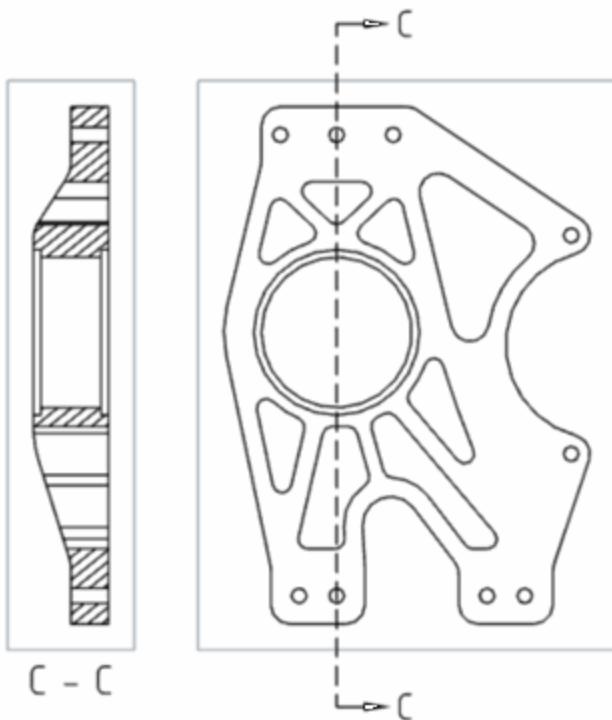
Labels are automatically applied (you can change them) and by default progress from A through Y, omitting I, O, Q, S, X, and Z.

4. Move the cursor over the part for which to create a view. Hover to view snap points.

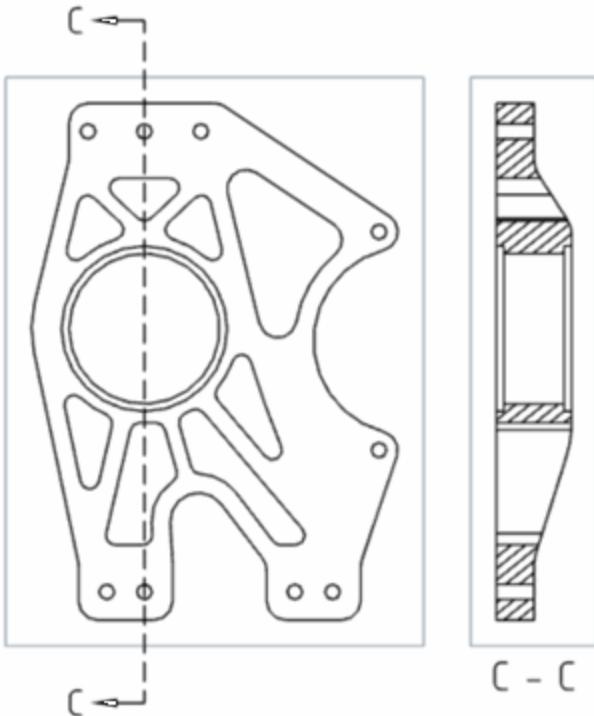


5. When the dotted section cutting line is in the desired place, or the snap point is visible, click once to place the line.

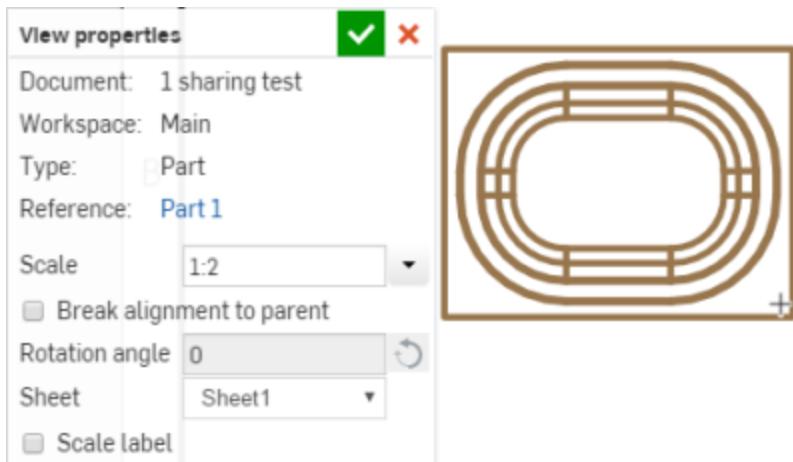
6. Drag the new section view away from the cutting line and click to place it.



Note that dragging the section view to one side or the other before clicking it into place flips the side of the section:



7. Double-click to edit the View properties, including setting the Scale and including a Scale label:

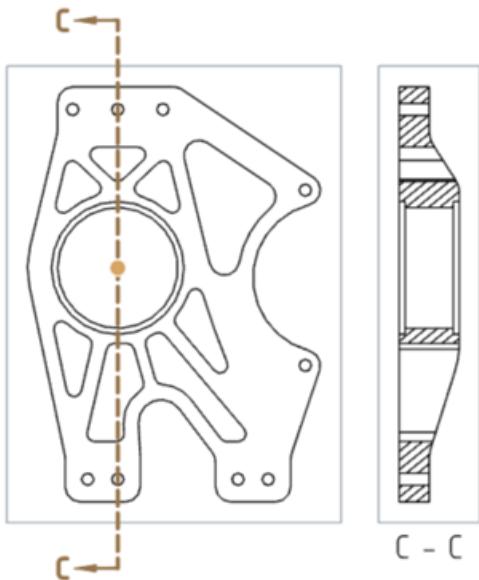


"View properties" on page 426 dialog formats all input into an N:N or N/N format. For user input values, the second digit or denominator is always set to 1, and you can double-click the Scale label to edit it.

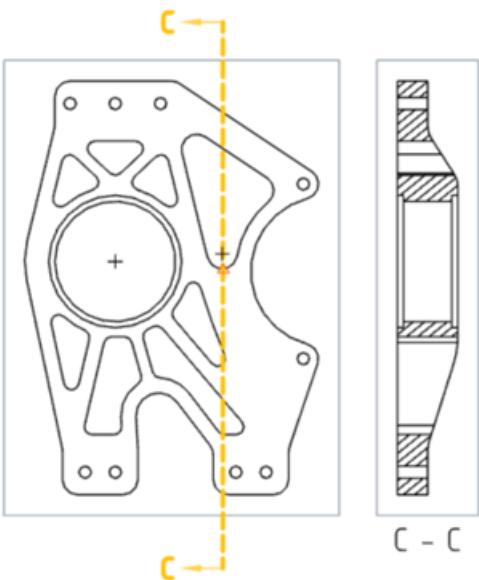
## Moving a section line

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:

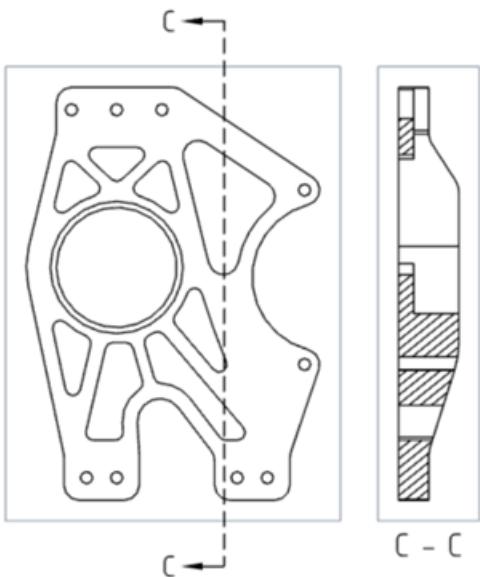


2. Click and drag the snap point to a new location.



3. Click to place the section line.

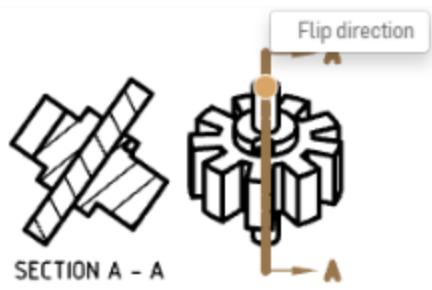
4. Notice that the corresponding view changes.



## 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.

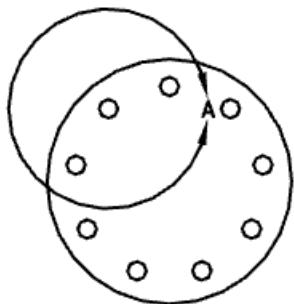


## Detail view

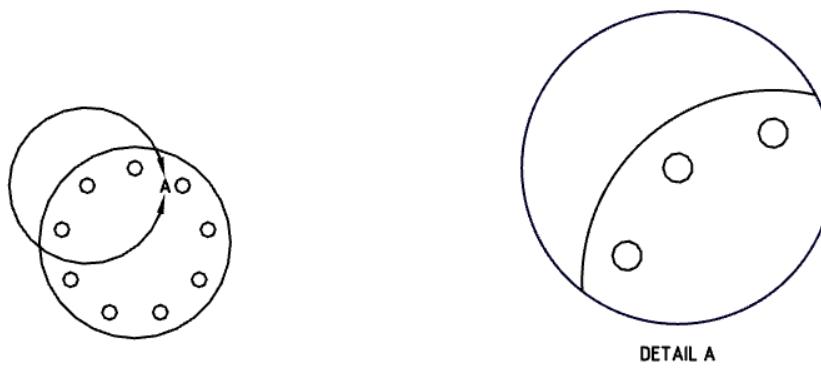
Use Detail view to select an area of an existing view to enlarge for more detail.

1. Click .
2. Click in the approximate center of the area you wish to enlarge (on an existing view).

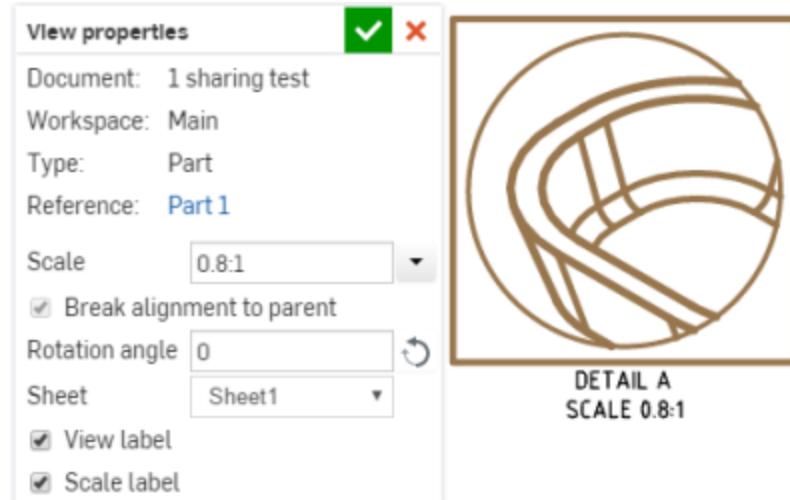
3. Drag and click again to define the circumference of the area.



4. Drag and click again to place the detail view.



Note that you can edit specify scale and edit the labels for detailed views:



## 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**.

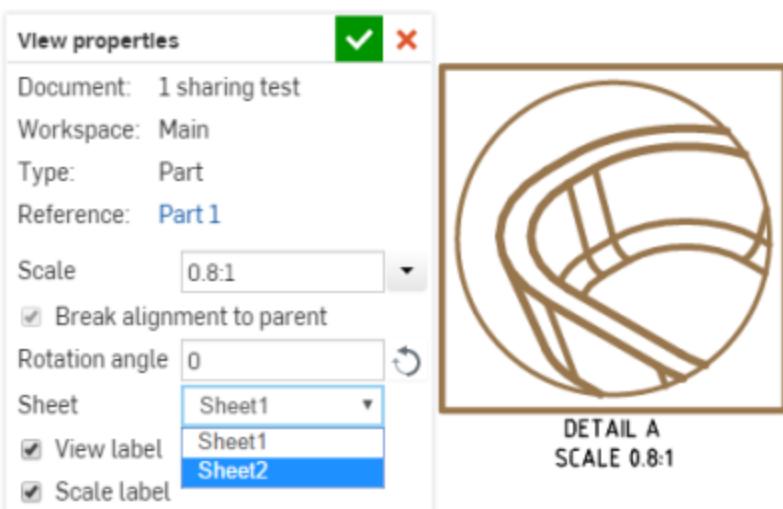
## Moving a view

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:

1. Open the Properties dialog for the view (double-click the view, or right-click and select Properties).
2. In the Sheet drop-down, select the desired sheet.



3. The sheet selected is immediately displayed, with the view on it and the View properties dialog still open for editing.

Note the following:

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.

## Modifying views

### Show/hide lines

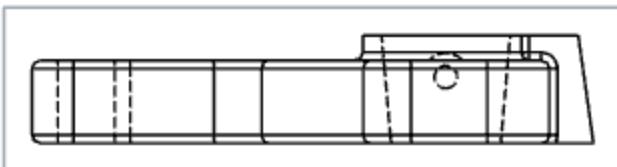
Show or hide the lines of a view that are not visible in current view position.

Select the view and Show hidden lines from the context menu:



- Show hidden lines
- Hide tangent lines
- Break alignment
- Create projected view
- Properties...
- Switch to Introduction Topics**
- Delete

The resulting view:



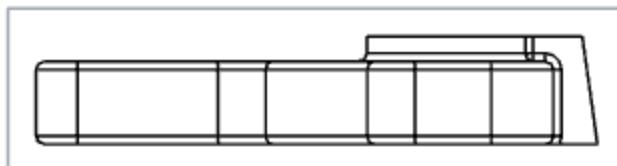
## Show/hide tangent lines

Show or hide tangent lines in a drawing view. Select the view and Show tangent lines from the context menu:



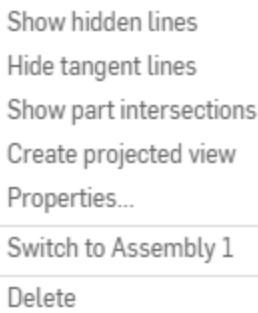
- Show hidden lines
- Hide tangent lines
- Break alignment
- Create projected view
- Properties...
- Switch to Introduction Topics**
- Delete

The resulting view:



## Show/hide part intersections

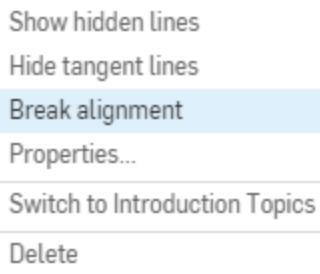
Show or hide the virtual edges (curves drawn at the places where parts intersect) where parts intersect. This setting defaults to Hide for all new views to improve performance. If an assembly view with more than 20 parts does not display correctly because parts interfere with each other or portions of intersecting edges/faces are misidentified as hidden (or visible) in any view, toggle **Show part intersections on**.



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 been completely left out of the view due to having an intersection and being partially obscured from the specific view orientation.

## Break alignment

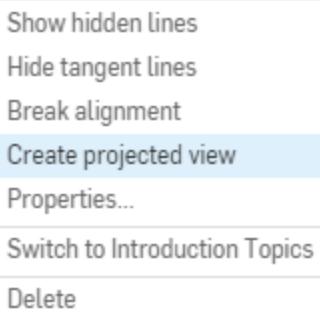
Disconnect the automatic alignment of views derived from other views in order to place them independently on the drawing. When you break alignment, you also break the scale link to the parent.



When breaking 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.

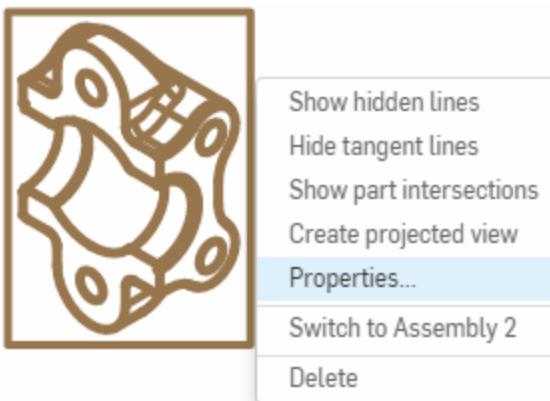
## Create projected view

Create a projected view (see above) from the currently selected view.



## View properties

Select Properties from the context menu, or double-click a view to open the View properties dialog:

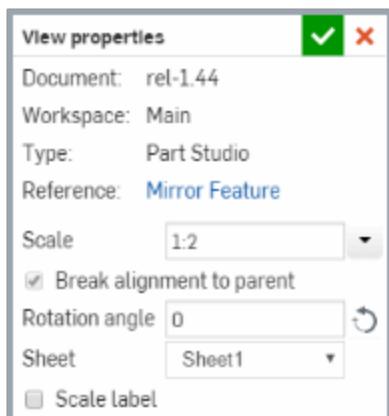


The View properties dialog displays specifics about a view, including:

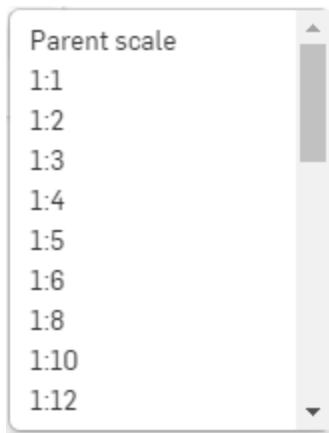
- The parent document of the part or assembly
- The workspace of the part or assembly
- The type of entity: part or assembly
- The name of the Part referenced in the view

You can also open the Part Studio or Assembly that the view is from, and specify a scale, rotation angle, and scale label.

The View properties dialog:



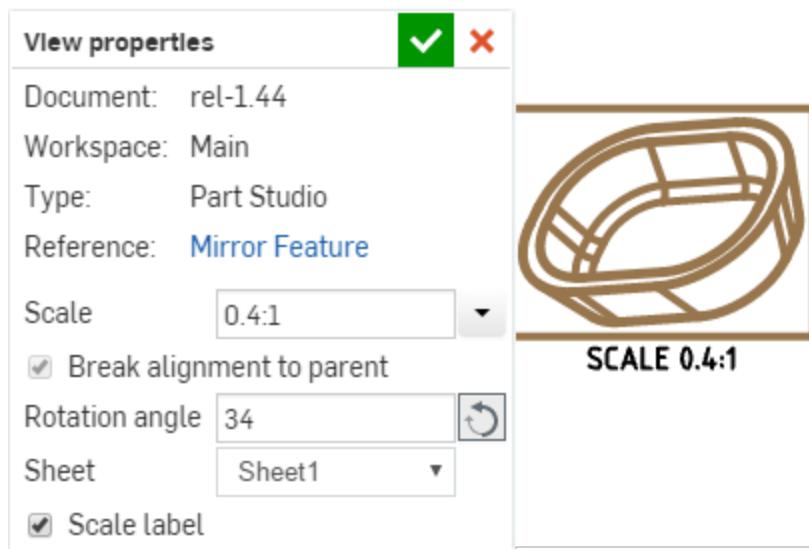
The Scale drop down:

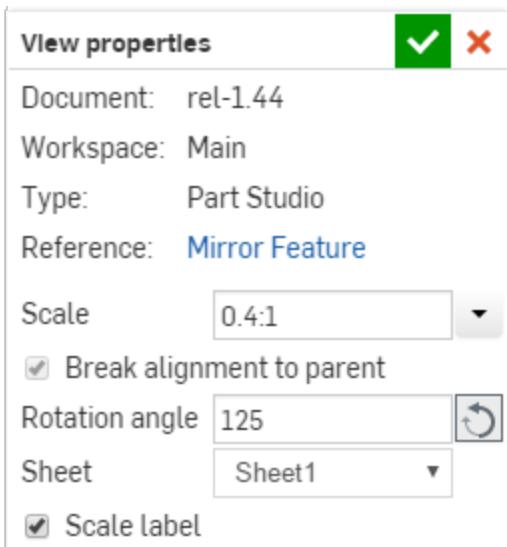


Select **Parent scale** to link the view's scale to its parent.

Rotation 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 broken.

Valid values are between -360 and 360 degrees.





## 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 Break alignment to 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 select break alignment.

# Dimensions



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 the snap point is visible, the point has been snapped to and you can click. There is no need to click directly on the point once it is visible.

Dimension snap points are available only on object lines. Tangent and hidden edges are not dimensionable and therefore have no snap points. In some cases, it may be necessary to cut a section view to provide dimensionable edges.

Editing the value of a dimension causes it to be converted to an Overridden dimension. See "Troubleshooting dimensions" on page 441.

## 2-point linear dimension



Shortcut: d

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.
  - a. Drag to place the dimension box.

## Line-to-line dimension



Create dimensions between parallel lines.

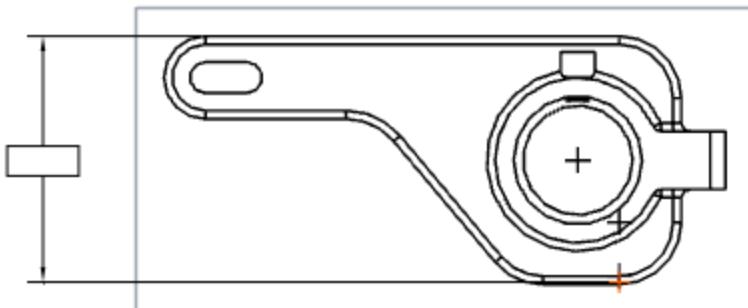
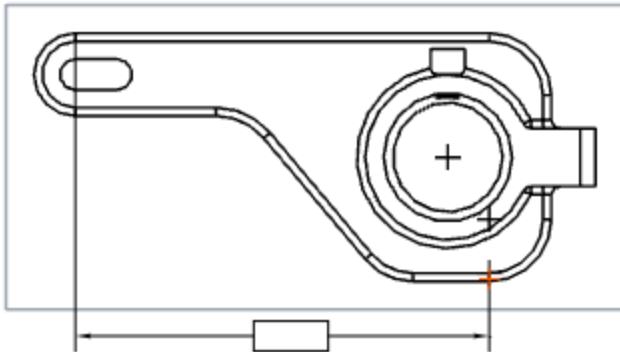


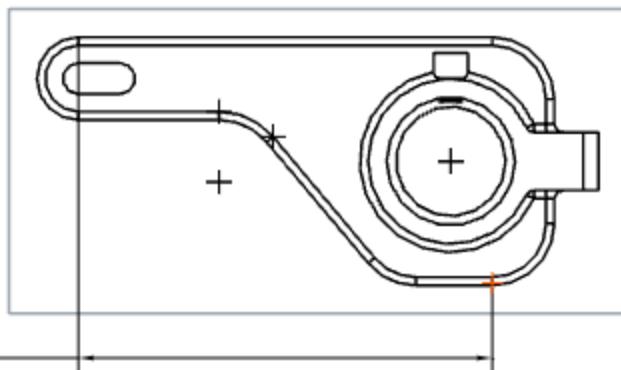
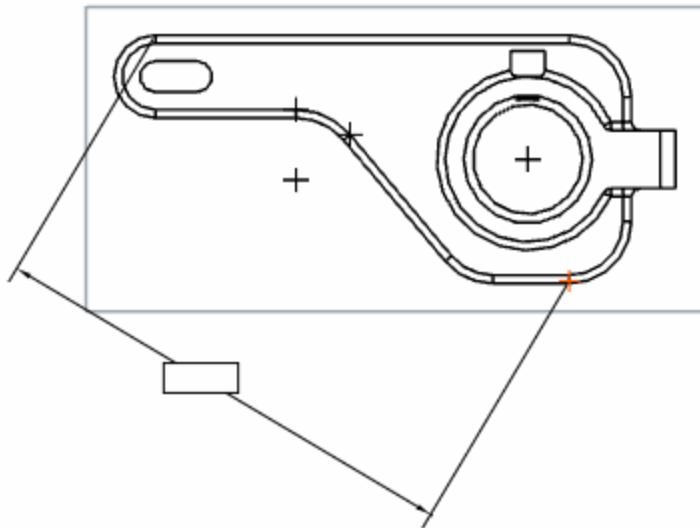
1. Click .
2. Hover over the drawing view to activate the snap points.
3. Click the first line highlight.
4. Click the second line highlight. Note that only parallel lines will highlight for selection.
5. Drag to place the dimension box.

## Placing dimension text

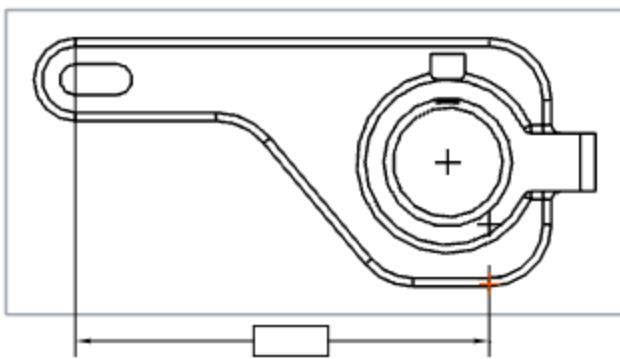
After picking two entities the dimension is drawn in a preview mode to allow final placement:

- Dragging the text around during preview can move the text outside of the extension lines, and also switch between horizontal, vertical, and aligned measurement modes:

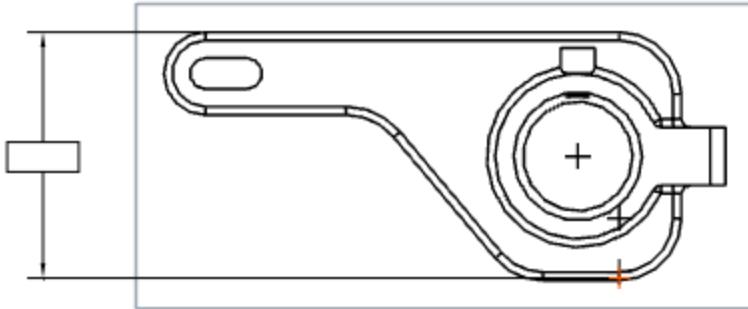




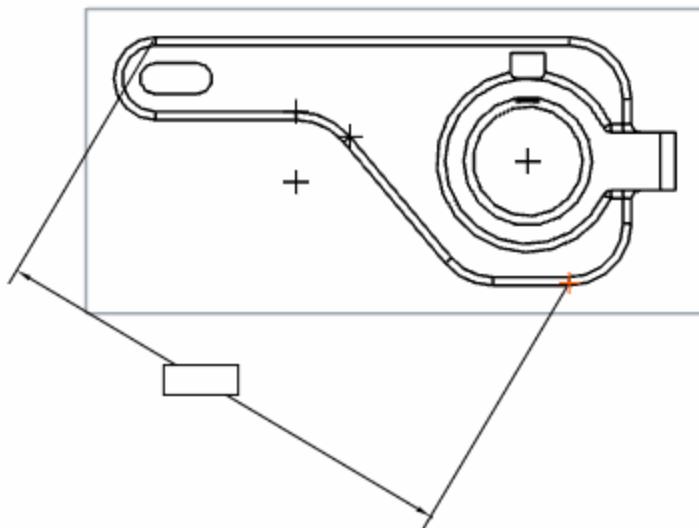
- Dragging the text away from the two chosen snap points up or down the drawing creates a **horizontal dimension line**:



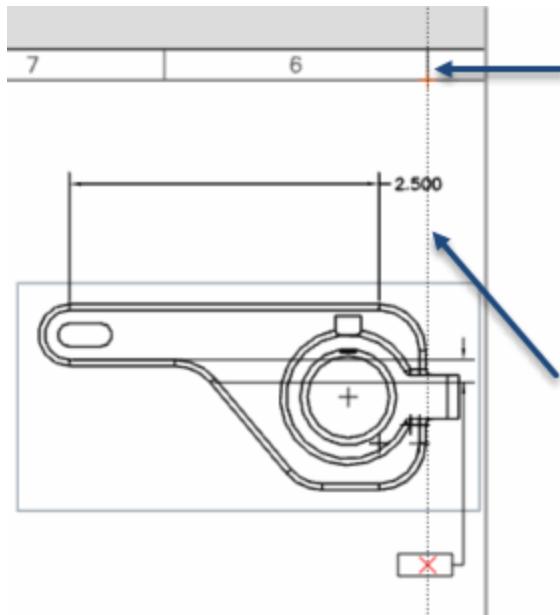
- Dragging the text away from the two chosen snap points toward the side of the drawing creates a **vertical dimension line**:



- 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:



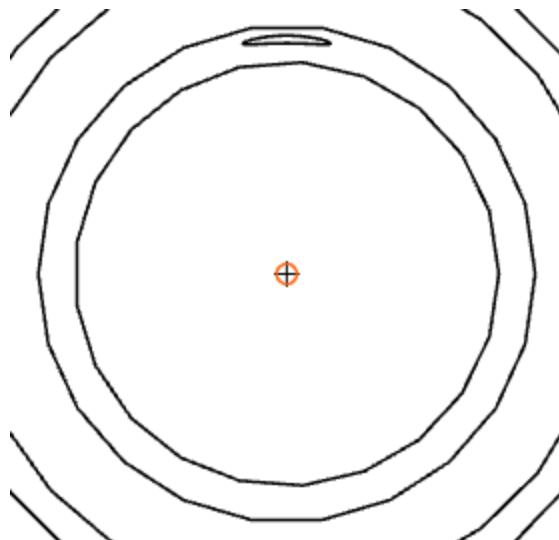
- Horizontal and vertical "projected" snaps are also available during text placement. This allows for lining dimensions up with existing text/dimensions and other locations on the drawing:  
Hover over a marker to wake up alignment. This is available in Preview mode only. Pass over other drawing entities to wake up alignment as well, like other views' entities.



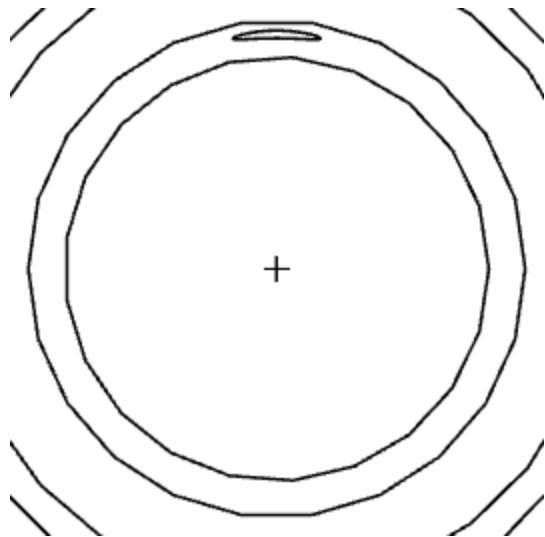
## Center marks on circular edges

When the dimension tool is selected, you can move the cursor over an edge representing a circular edge to 'wake up' the center mark. Once visible, this mark remains visible:

Upon moving cursor over edge, an orange circular snap point appears, with the vertical center marker:



After hover, the orange snap point disappears but marker remains:

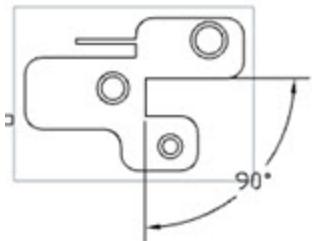


## Line-to-line angular dimension

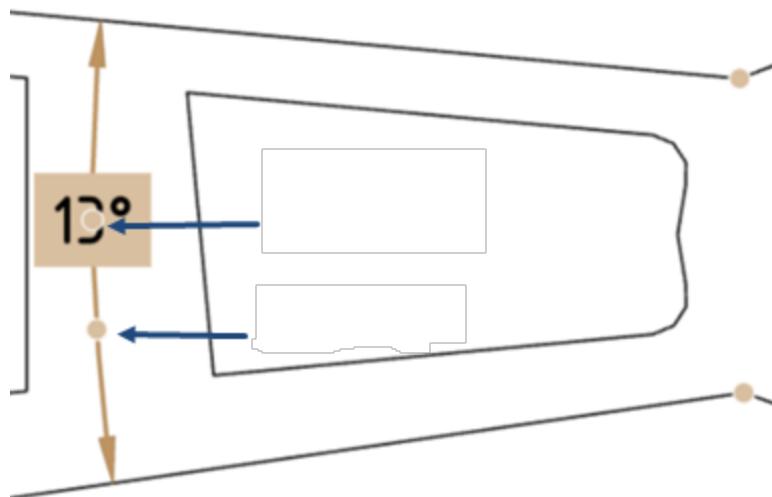


Measure the interior angle between the two legs and the exterior angle formed by two lines.

1. Click two lines.
2. Move the cursor between the lines to preview the inner angle dimension.



Line-to-line angular dimensions have a drag-able grip on the text like other dimensions, plus another drag-able grip on the dimension arc. That second grip is for changing the angle to be measured:



Drag that second grip point across one of the infinite lines through the ends of the selected edges/points to change the measured value.

On line-to-line angular dimensions it changes to the supplementary or vertical angle of the one 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:

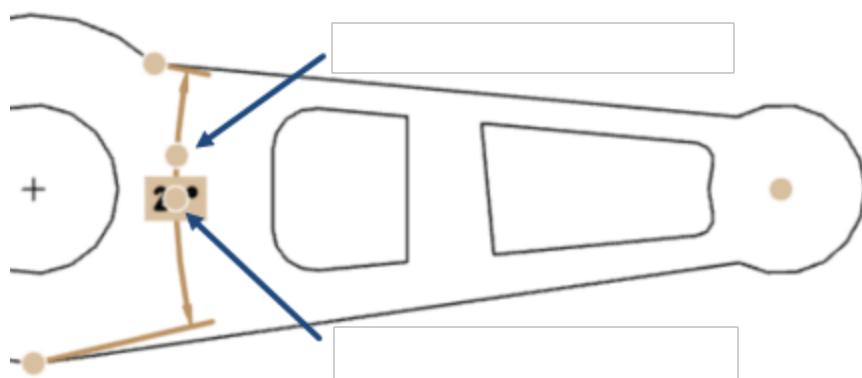
1. Click the vertex.
2. Click a point on each leg on the perimeter of the arc.

Angular dimensions have a drag-able grip on the text like other dimensions, plus another drag-able grip on the dimension arc. That second grip is for changing the angle to be measured:

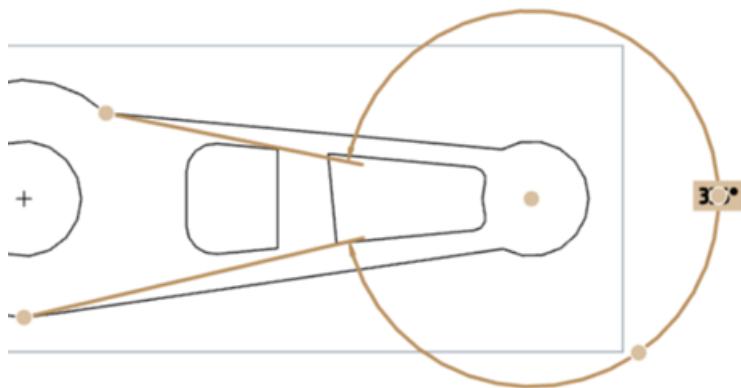
Drag that second grip point across one of the infinite lines through the ends of the selected edges/points to change the measured value.

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):

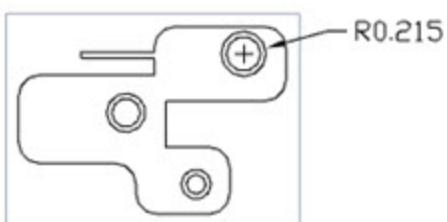


## Radial dimension

Shortcut: Shift-r

Measure the radial dimension of a circle or arc.

1. Select the arc or circle.
2. Move the cursor and click to place the dimension.



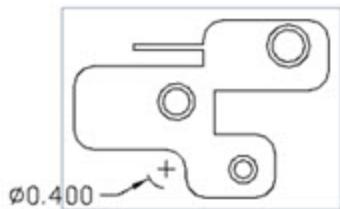
## Diameter dimension



Shortcut: Shift-d

Measure the diameter of a circle or arc.

1. Select the arc or circle.
2. Move the cursor and click to place the dimension.



## Ordinate dimension

Create ordinate dimensions (X, Y pairs) for a feature measured from a datum.



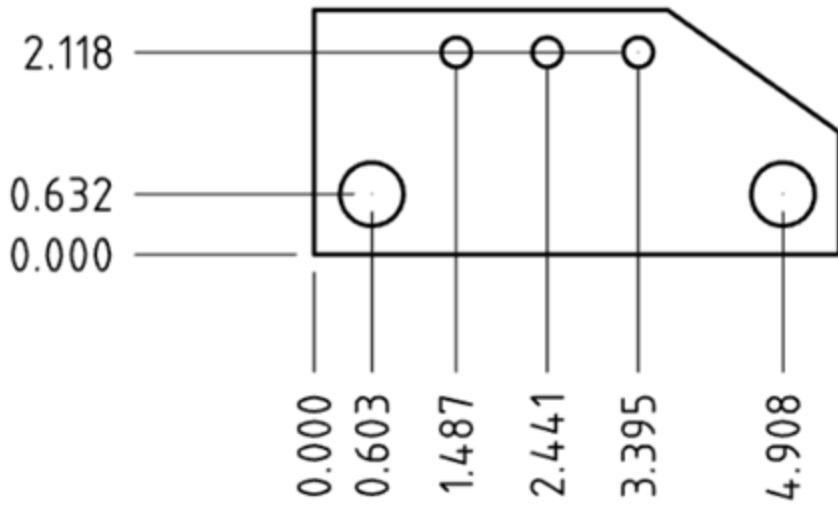
1. Click
2. Click the point to serve as the datum (0, 0).
3. Click each point in one direction (Y, for example) to associate with that datum point.
4. Press Escape to exit the tool.

At this point, one Ordinate dimension group is created.



5. Click
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 can be the same as the first datum chosen.
8. Press Escape to exit the tool.

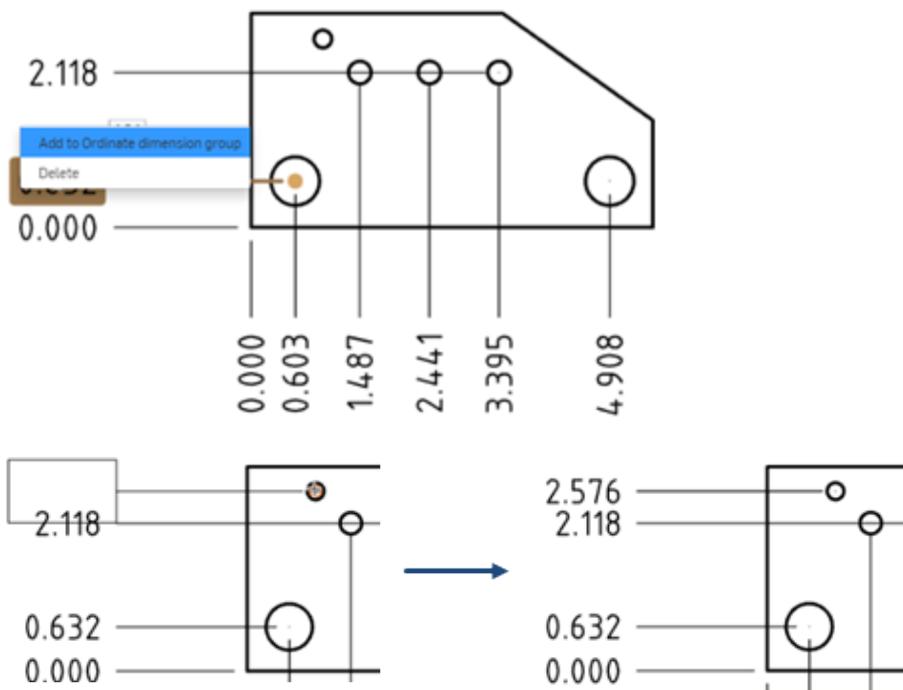
At this point, a second Ordinate dimension group is created.



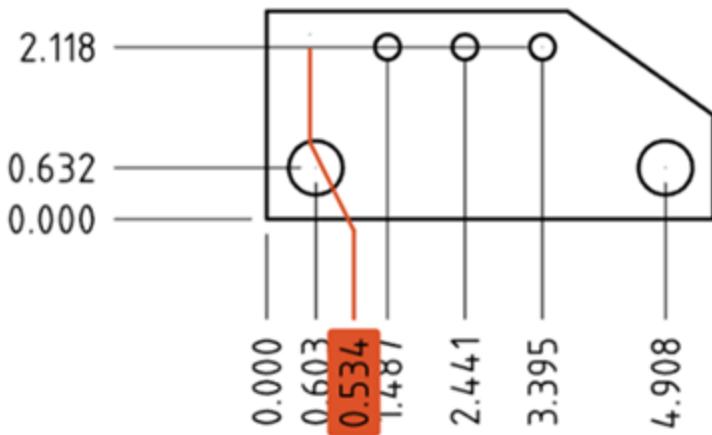
### Tips

- Each direction must have a datum; each time you initiate the command from the toolbar, the first click establishes the datum point (0, 0).

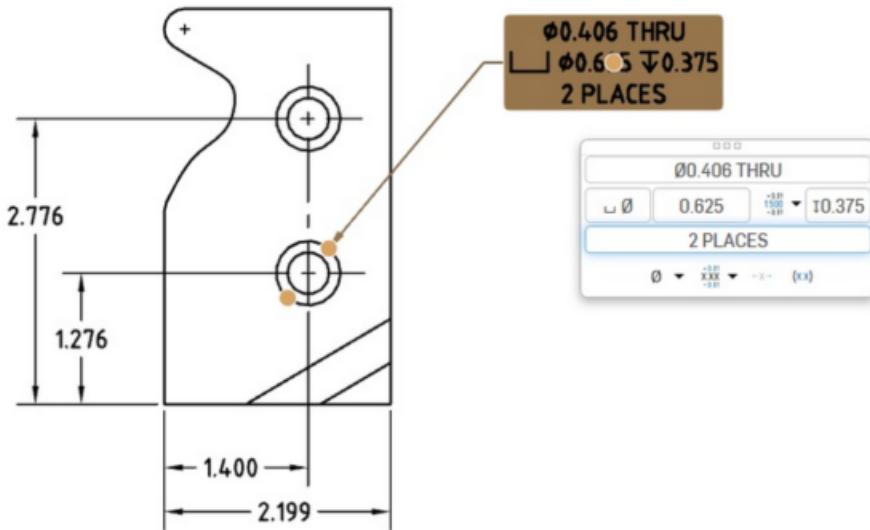
- Each direction of dimensions (Y, for example) consists of a ordinate dimension group with a single datum. To add another value pair to that group, select an existing value in the group, right-click and select Add to ordinate dimension group. This activates the command and the next click will establish the additional dimension 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 can safely delete the dimension (right-click and select Delete, or select and press Delete).



Dimension panel  
±0.01  
↔



You can customize the appearance of dimensions with the Dimension panel. Selecting a dimension causes the dimension panel icon to appear.

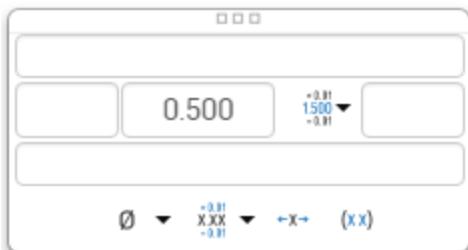
1. With no tool selected, select the dimension.



2. The Dimension panel icon appears:



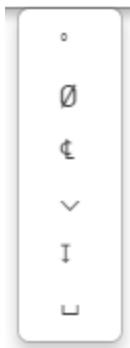
3. Hover over the icon and the panel opens:



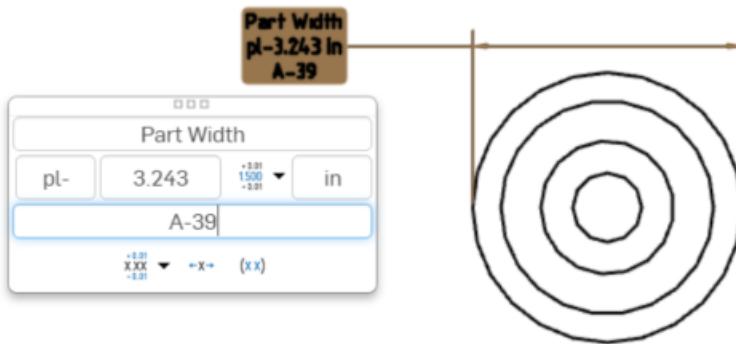
4. What you can enter, moving from top to bottom of the panel:

- **Above text** - Enter the text or symbol to appear above the dimension value.
- **Prefix text** - Enter the text to appear as a prefix to the dimension value.
- **Tolerance display** - Select None, Symmetrical, Deviation, Limits, or Basic
- **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:



- **Precision** - Select the depth of unit precision (zero to 8 decimal places).
- **Reset text position** - Use this toggle to reset the text to the previous location.
- **Add parenthesis** - Use this toggle to add or remove parenthesis around the dimension field.



You can also copy/paste into all text boxes, in dimensions and notes as well.

## Adding symbols

In the text box of the Dimension panel, you can add codes in order to display the symbols of your choice:

### Drafting symbols

- Degree (°), %d
- Plus minus (±), %%p
- Diameter (Ø), %%c

## 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. The dimension may be re-associated to geometry to become associative again.
- **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 can 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.

# A Datum



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:



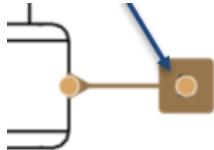
1. Click
2. Enter the necessary label in the dialog.



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.
5. Check the Filled triangle box in the dialog for a filled arrow head, or leave unchecked for an unfilled arrow head.

## Tips

- You can drag a datum to another location after placement: click to select it, then drag.
- To change the label, click to select the datum, then double-click in the highlighted square:

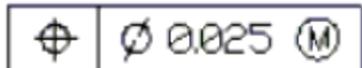


The datum dialog opens and you can change the label and the triangle (filled or unfilled).

# Geometric Tolerance



Often associated with datum, use Geometric tolerance to create and place basic dimension notations in the drawing, like this:



Creating a tolerance:

1. Click

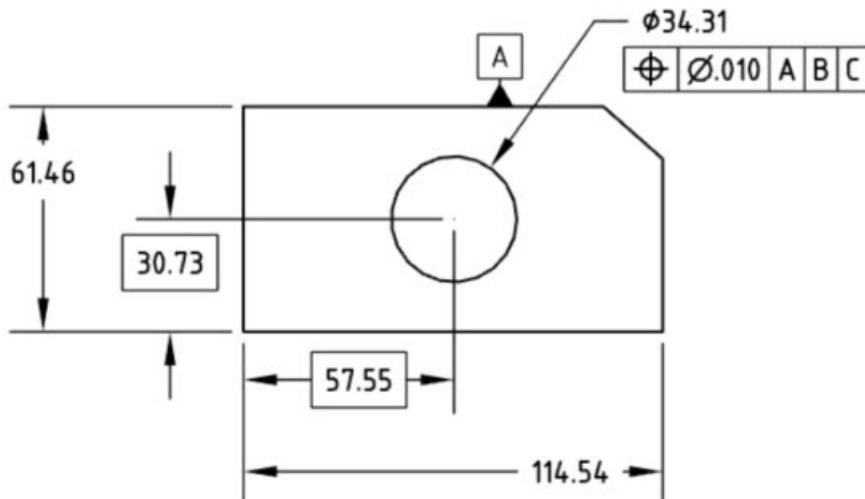
2. In the dialog, from corresponding lists, specify the symbols and associated tolerances for your drawing:



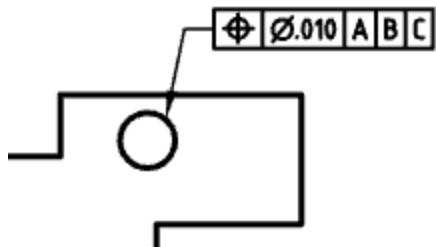
3. Complete the specifications by typing tolerance values in the corresponding boxes:
4. Enter a **Height** for the tolerance.
5. Add the **Projected tolerance zone** symbol if necessary.
6. Click in the graphics area to place the tolerance.

To place tolerance with a leader, hover over drawing view until a snap point appears, click the desired snap point, drag tolerance and click to place.

The tolerance displays in the graphics area.



Tolerance with leader:



Editing a tolerance:

1. Double-click on the tolerance in the graphics area.
2. Make your changes in the dialog that opens.

## Geometric characteristic symbols

Symbol	Characteristics	Type
	Position	Location
	Concentricity or coaxiality	Location
	Symmetry	Location
	Parallelism	Orientation
	Perpendicularity	Orientation
	Angularity	Orientation
	Cylindricity	Form
	Flatness	Form
	Circularity or roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile

Symbol	Characteristics	Type
	Circular runout	Runout
	Total runout	Runout

Material condition symbols

Symbol	Characteristics	Type
	At maximum material condition, a feature contains the maximum amount of material stated in the limits	MMC
	At least material condition, a feature contains the minimum amount of material stated in the limits.	LMC
	Regardless of feature size, indicates that the feature can be any size within the state limits.	RFS

# A Note

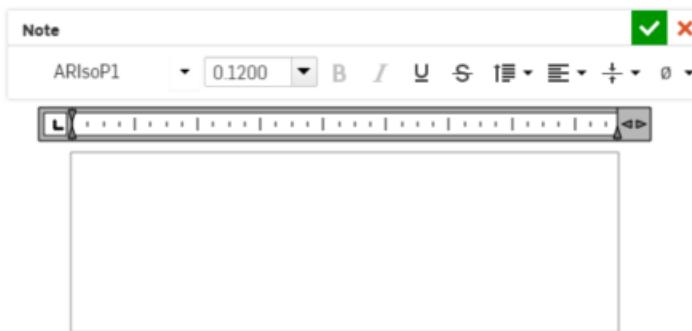
Shortcut: n



Add multi-line text notes to any drawing, wherever you want, and use them to fill in the title blocks as well. You can define the size of the text box as well as format the text itself.

Creating notes:

1. Click A.
2. Click and drag in the drawing space to establish the text box.
3. Click again to set the box and open for editing and formatting.



You can enter unicode characters in notes. For example, use /U+00AE to create the registered trademark symbol ®.

## Formatting notes

You can 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, or triple-click to open the text box with all text selected
- 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 options is disabled in Note with leader command; drafting standards dictate alignment of text in that context.
- **Vertical alignment**  - Indicate the type of paragraph justification in relation to the insertion point of the Note: Top, Middle, or Bottom.

Note that this options 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.

## Formatting ruler



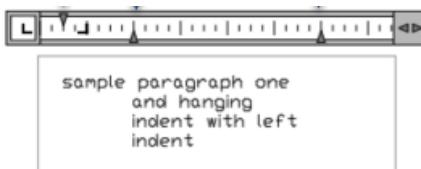
Use the Note Formatting ruler to set paragraph indents and tab stops for Notes.

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:



## 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 rule 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 can 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 show 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 can fill using notes within the boundaries of the title block cells.

For best results:

- Create a text box the size of the cell. You can then experiment with text size, font, etc and see if the text will fit without having to resize the text box.
- You cannot copy and paste text boxes; but you can copy and paste text from one 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 can move the lines of the title blocks, or create your own.

# Note with Leader



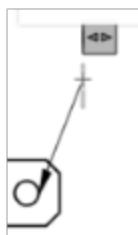
Create notes with leader lines connecting the annotations to a drawing entity. Notes with leaders are useful when the dimension text or annotation does not fit next to the corresponding entity. You can optionally place single or multiple lines of text.

To create Notes with leaders:



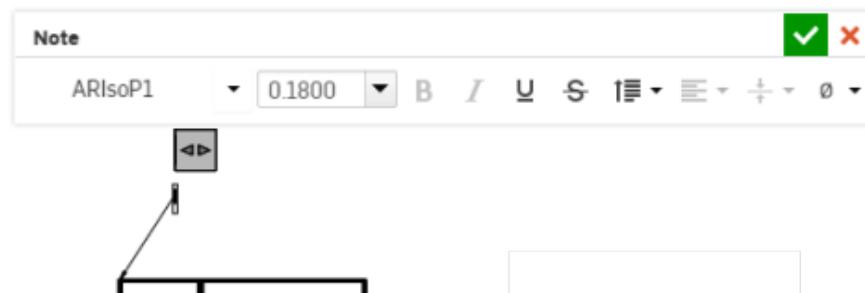
1. Click .
2. Click the start point of the leader. (By default, the start point is the arrow head.)
3. Click the end point of the leader.

A small text box appears directly above the end point of the leader. A Note dialog is above that:



*Small text box above the end point of the leader. Click and draw the small shaded box to expand the text box.*

The Note dialog is above the small text box and is used for formatting text:



4. Enter text in the text box. You can enter multi-line text here.
5. Use the Note dialog to make formatting specifications for the text. For information on using the Note dialog, see "Note" on page 447.
6. When finished, click .

## 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.

## 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 the leader.
2. Drag the leader to its new position.

## Removing leaders and/or text

1. Click to select the leader and/or text.
2. Press the Delete key.

# ① Balloon



Create a simple balloon with a leader line.

To create a balloon:

1. Click .
2. Click the start point of the leader.
3. Click the end point of the leader.
4. You can specify the label in the Balloon dialog.



Onshape keeps track of which number is next in the series. If you delete one, it is reused automatically on the next balloon.

## Removing balloons

1. Click to select the balloon or leader.
2. Press the Delete key.



# Table

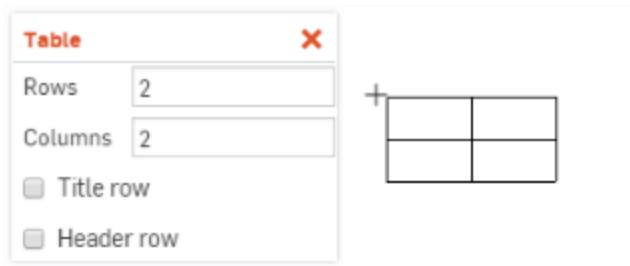


Add fully-customizable tables to any drawing.

Creating a table:

1. Click .

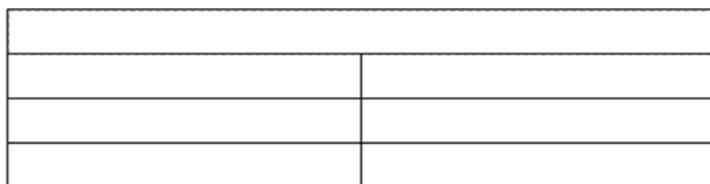
2. The cursor becomes a table icon and the Table dialog opens:



3. Before clicking to place the table, you can enter the number of rows and columns in the dialog.
4. Also in the dialog, 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)



5. To place the table, click on the sheet to set the location (anchored by the upper left corner of the table).
6. Drag down and to the right to size the table; click to set the size.
7. The "Note" on page 447 opens:



8. Click in a cell to enter text; tab from cell to cell, shift-tab for previous cell.

Double-click in any cell to open the Note panel for editing text.

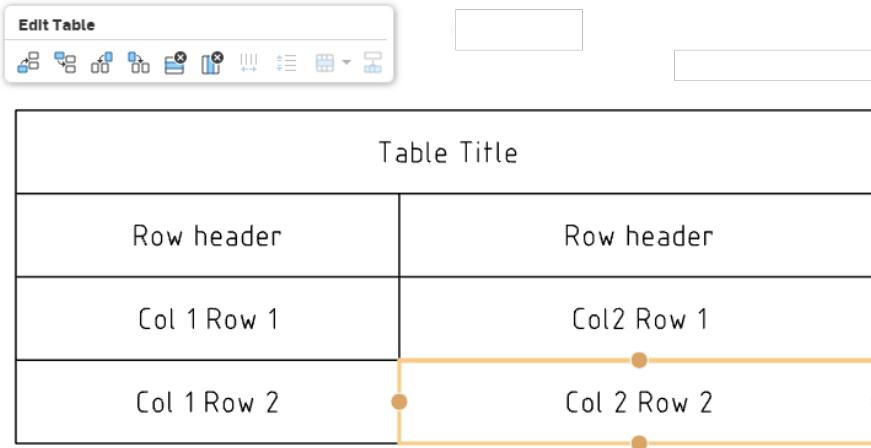
## Formatting tables

After a table is created and the Table dialog is closed, you can hover over a table to activate grab points:



- Drag the top-left grab point to move the table.
- Drag the top-right grab point to resize the width of the table.
- Drag either of the bottom grab points to resize the height of the table.

Single-click in a table cell or row to activate the Edit Table toolbox:



Edit Table toolbox



- Shift-click to select more than one cell
- You can 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:

Table Title		
Column Header	Column Header	Column Header
Col 1 Row 1	Col 2 Row 1	Col 3 Row 1

- **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)

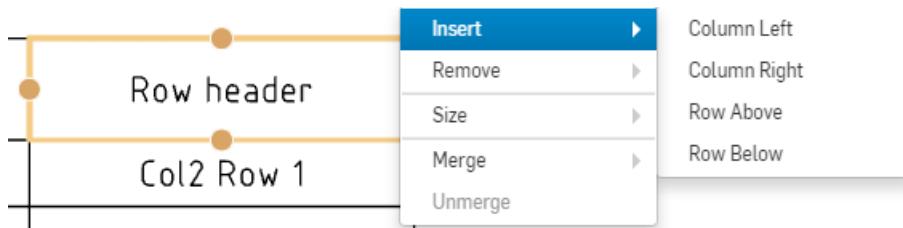
When more than one row is selected, you can also:

- ↑↓ **Size rows equally** - Resize all selected rows to the average height

After merging cells, you can also:

- ☒ **Unmerge cells** - Return last-merged cells to previous unmerged state

Note that you can access these commands from the context menu when at least one cell is selected:



# Drawing Tools



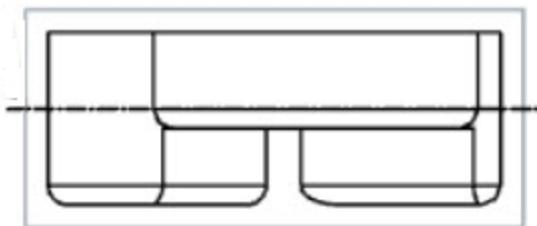
Onshape provides tools for creating sheet geometry: drawing entities like lines and centerlines, created on the sheet outside of a view and meant to represent some part of the 3D model.

## 2 point centerline

Create centerlines using two points on your drawing.

1. Click

2. Select two points to establish a centerline. Note that you can use snap points, but it is not required.



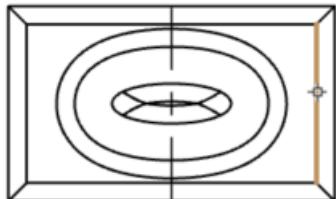
## Line-to-line centerline



Create centerlines using two lines on your drawing.

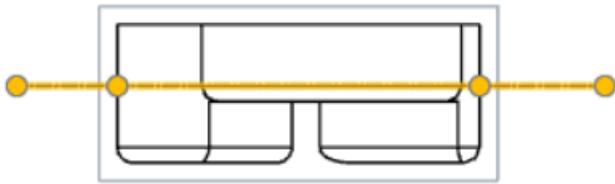
1. Click

2. Select two lines to establish a centerline.



## Removing centerlines

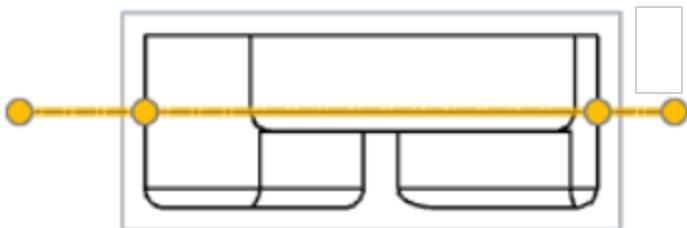
1. With no tool selected, click the centerline (it appears highlighted).



2. Press the Delete key.

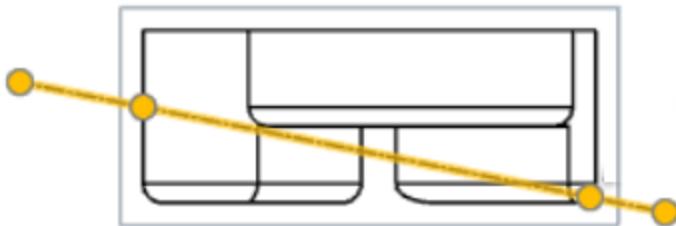
## Modifying centerlines

1. With no tool selected, click the centerline (it appears highlighted).
2. Click and drag an end point to resize the line:



Note that centerlines may be dragged below the distance between the reference points.

3. Click and drag a snap point to move the line:



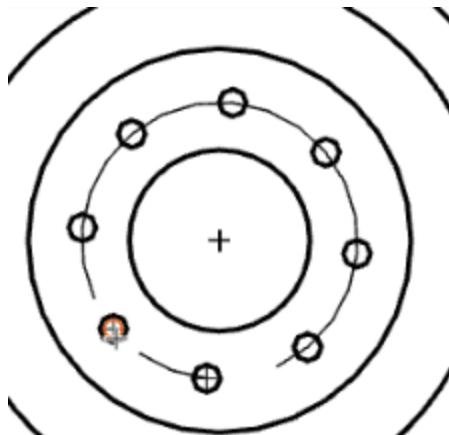
## 3-point circle centerline



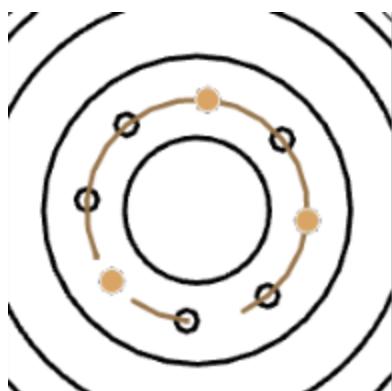
Create a circular centerline for a bolt circle diameter.

1. Click .

2. Click each of 3 points (centers of the holes). The first illustration shows the centerline in process:



The illustration below shows the centerline selected; you can see which holes help define the centerline:



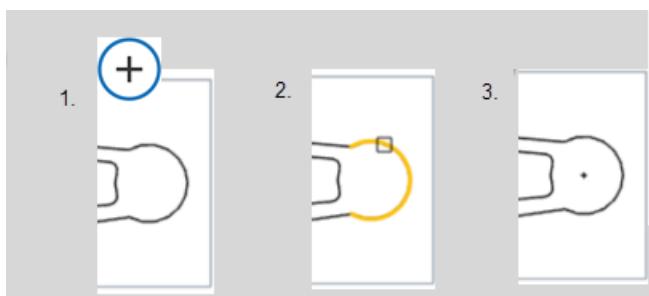
## Centermark



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.

## Line



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:



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.

# Refine Graphics



The geometry in drawings views is displayed in Onshape as short lines. For performance purposes, Onshape uses the minimum number of short lines so that curves look smooth at your current zoom. At times, when you zoom in, the short lines may look coarse.

Use the Refine Graphics tool to recompute the lines, resulting in geometric curves looking smoother. The underlying geometry in your views remains unchanged.

# Updating a Drawing



Shortcut: Ctrl-q



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, as you may expect). 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.

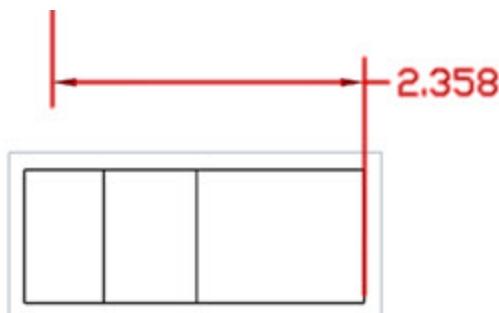
1. Click .

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 broken (or dangling).

You can fix a broken dimension by clicking the grip point and dragging it to where it should be. This is perfectly normal, especially if the change to the part or assembly was significant.



## 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.

# Importing a Drawing



Drawings can 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 .
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.

# Exporting a Drawing

You can export Onshape drawings to the following file types:

- PDF
- DWG
- DXF

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. Select the desired export command:
  - Export to .DWG... in Release 11 format
  - Export to .DXF... in Release 11 format
  - Export to .PDF...
4. If exporting to DXF or DWG, select the version and sheets.
5. Indicate how to show overridden dimensions, if present (by default, overridden dimensions are underlined):
  - a. With underlines
  - b. Hide underlines

Drawings exports are simplified output that is readable by most DWG readers.

# Printing a Drawing

You can 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

A Feature Studio is a tab containing FeatureScript, a programming language that you can use 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](#).

For detailed information on using custom features within your Onshape account, see [Custom Feature](#).

# Importing & Exporting Files

You can 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 can export to another CAD format, as well as simply download non-CAD files. For more information, see the links below.

- [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.

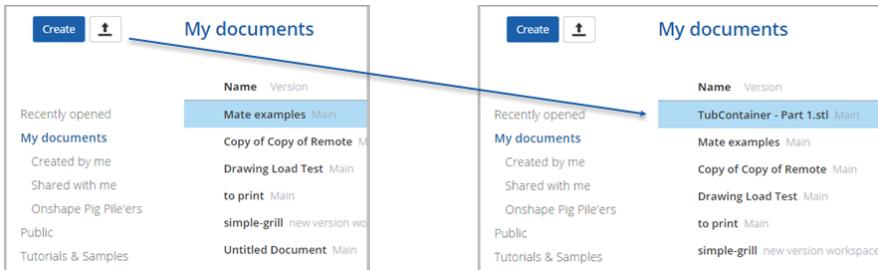
Read more on [supported file formats](#).

# Importing Files

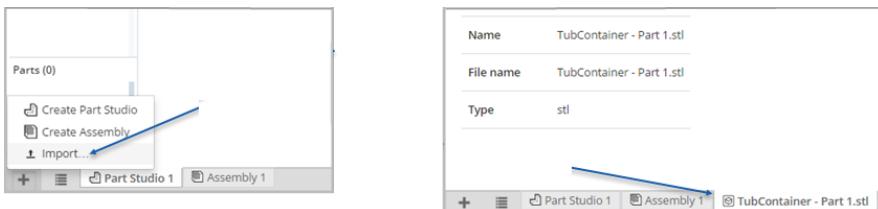
You can import (upload) any type of file into Onshape, either into an existing and open document, or as its own document (from the Documents page). For a list of supported files, see "Supported File Formats" on page 478.

How Onshape handles your import depends upon where you initiate the import:

- 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.



- From **inside a Document** - Creates new Onshape tabs (Part Studio or Assembly) in the active document; the tab names reflect the naming of the file.



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/or 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.

## Processing CAD files

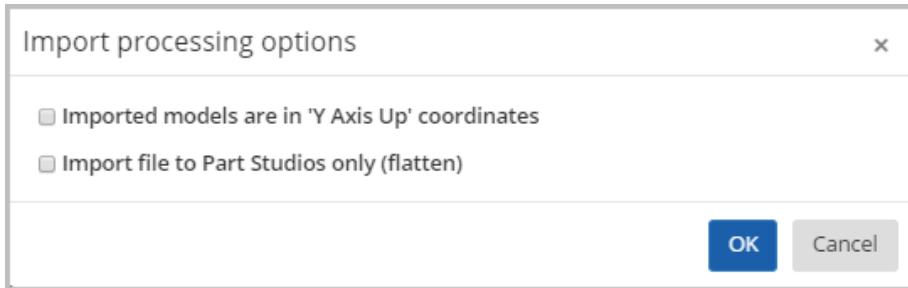
When Onshape recognizes an imported file as a CAD file (based on its file extension), Onshape automatically presents processing options. You can also choose to export to another format from a context menu for an entire Part Studio (including hidden parts), or for a particular part selected from the parts 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 can 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" on page 471.

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. Check this box 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 selecting 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.



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

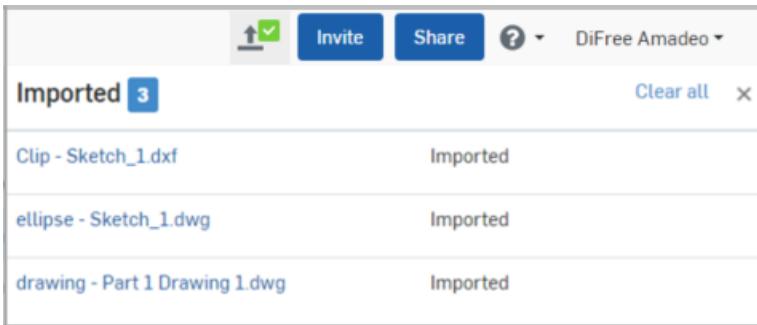
1. Click Import 

The file explorer opens on your local machine.

2. Select a file (or files) to import.

If you belong to an organization, Onshape prompts for the desired owner of the document: select yourself or an organization.

Onshape displays a list of recently imported files.

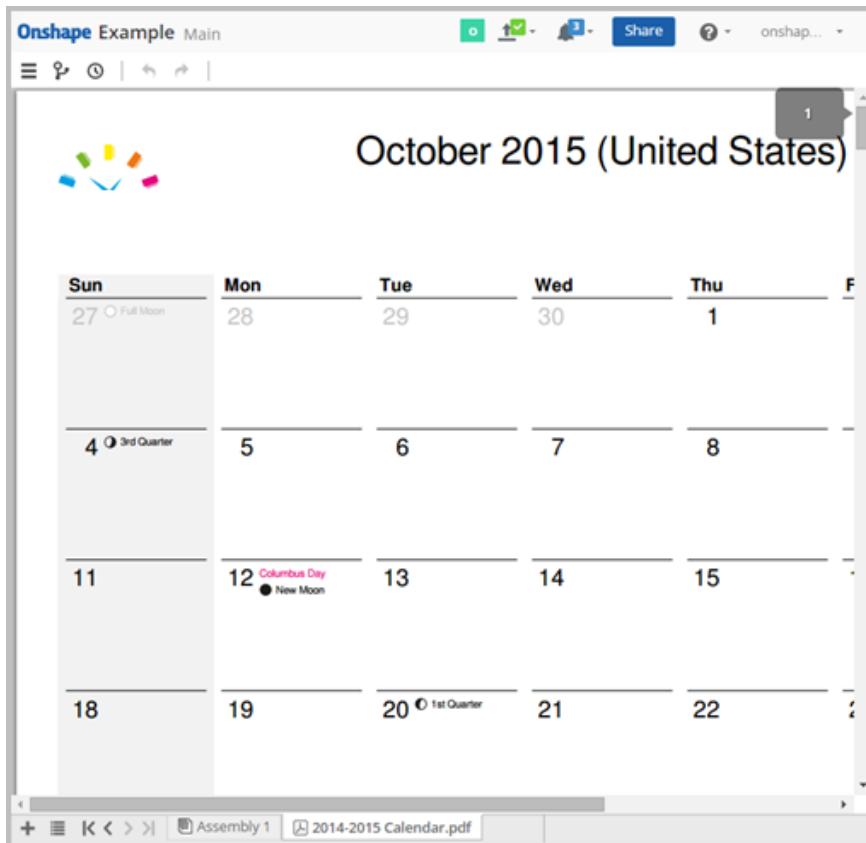


A screenshot of the 'Imported' section of the Onshape interface. It shows three files listed: 'Clip - Sketch\_1.dxf', 'ellipse - Sketch\_1.dwg', and 'drawing - Part 1 Drawing 1.dwg', all marked as 'Imported'. The interface includes standard buttons like 'Invite', 'Share', and a user dropdown.

File Name	Status
Clip - Sketch_1.dxf	Imported
ellipse - Sketch_1.dwg	Imported
drawing - Part 1 Drawing 1.dwg	Imported

3. Now you can:

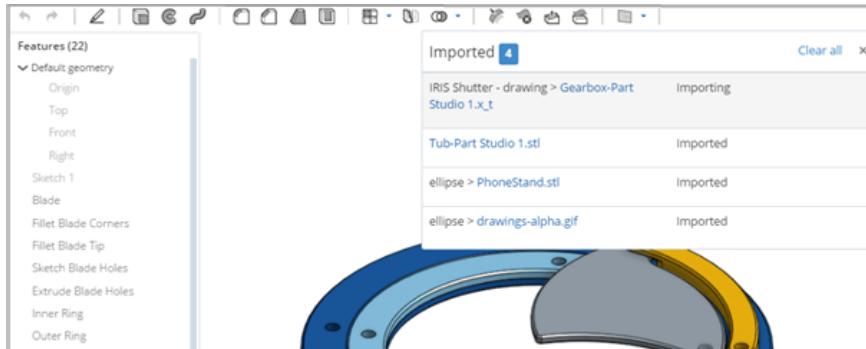
- Click on the file name in the import list to immediately view the file in Onshape, through the document that was automatically created for it, in its own named tab:



- Or click the X in the upper right corner 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

- Once in an open document, at the bottom of the page, click and select **Import**.
  - The file explorer opens on your local machine; select a file to import.
- Onshape displays an Imported dialog list showing the "document name > file/tab name":



- Once the import is finished, click the X in the upper right corner to close the dialog.

You could also click on the blue file name in the Import dialog to open the file immediately in its tab.

4. The imported file is now in the 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 478.

## Importing SolidWorks files

As stated in "Supported File Formats" on page 478, Onshape supports the import of SolidWorks native parts and assemblies. This section describes what you need to know to successfully import from SolidWorks.

### SolidWorks assemblies

Onshape needs all of the parts and subassembly files alongside the top level .sldasm file to successfully import a SolidWorks assembly. Follow these steps to import a SolidWorks assembly:

1. Use the Pack & Go tool in SolidWorks to 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.

Additionally, Onshape and SolidWorks both run on the Parasolid modeling kernel and exporting a SolidWorks assembly as a Parasolid (.x\_t) file and then importing it into Onshape also works.

### SolidWorks parts

There is no special workflow required to import a SolidWorks part file (.sldprt) into Onshape. Simply select the part file when prompted by the import dialog.

That said, exporting a SolidWorks part as a Parasolid (.x\_t) file and then importing it into Onshape also works.

# Exporting Files

Onshape enables you to export parts and surfaces (from Part Studios), entire Part Studios, entire Assemblies, as well as sketches and planar faces for use elsewhere.

- Surfaces can 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 can be exported individually from the Parts list (in Part Studios) or selected in the graphics area (use the context menu, Export option) to:
  - STL
  - Native or standard formats (Parasolid, ACIS, STEP, IGES, Solidworks, and Rhino)
- 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.

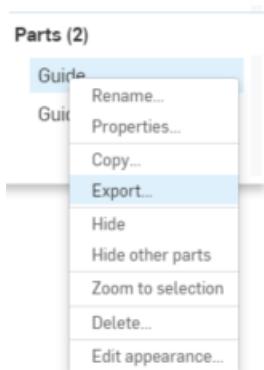
Note that exporting as individual files creates a zip file with multiple files each containing a single part.

- Sketches can be exported to DWG and DXF formats, in Release 11 format
- Planar faces can be exported to DWG and DXF formats, in Release 11 format

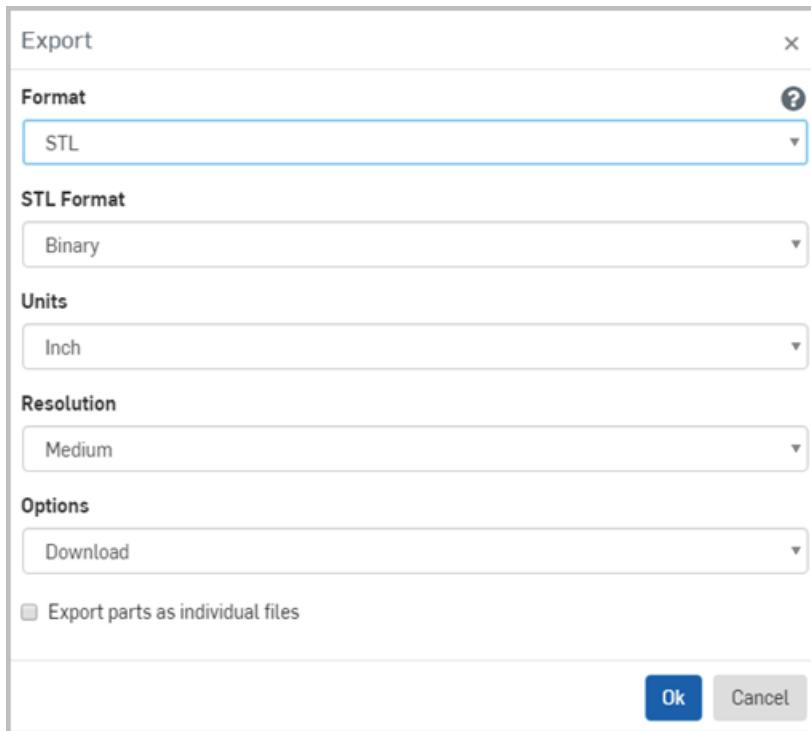
Note that the downloaded data will not contain features or parametric history. See the topics below for more information.

## 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:



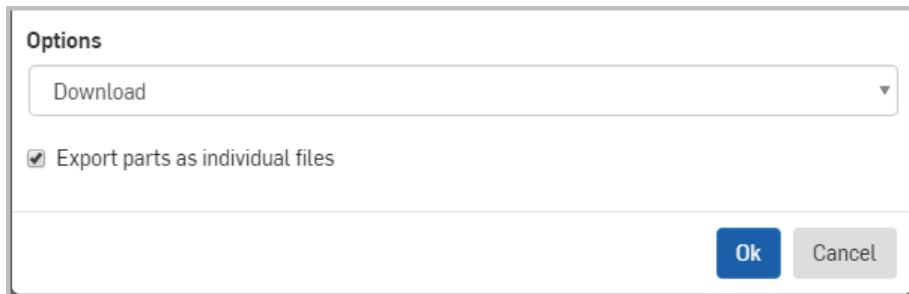
When exporting a part to STL format, you have the following options:

- 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

Setting	Value
Angular deviation (deg)	6.25
Chordal tolerance (in)	0.004724
Minimum facet width (in)	0.01

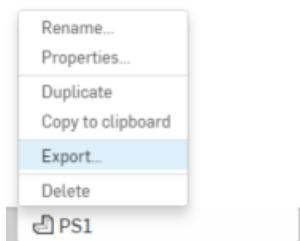
- **Download**, **Download and store file in a new tab**, or skip the download and **Store file in a new tab**.

Note that when multiple parts are selected, you can specify whether to export the parts as one file, or as individual files, zipped together:

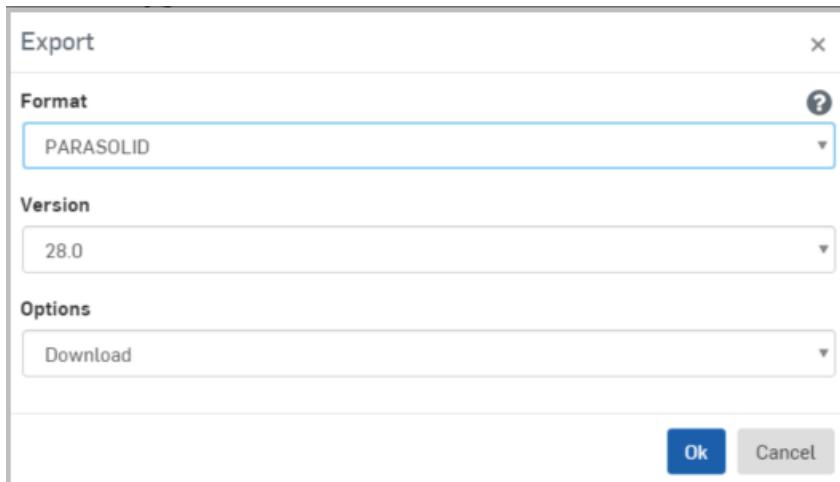


## Exporting Part Studios

To export an entire Part Studio, access the Export command from the context menu on the Part Studio tab:



Specify the parameters to use:



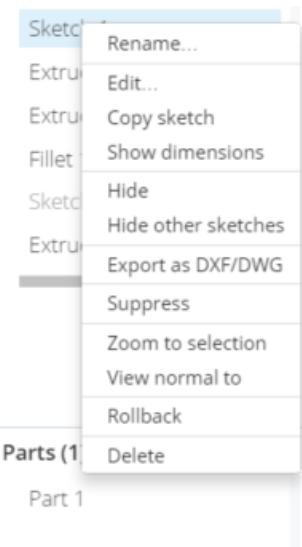
Check your file downloads location for the file upon completion.

## Exporting sketches or planar faces

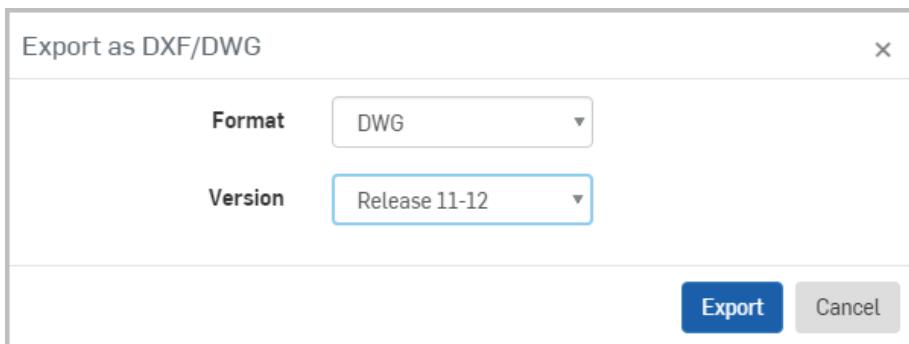
Sketches are exported in the document's default units, and planar faces are exported with outer solid geometry only, no dimensions or interior geometry.

## Export a sketch from the Feature list

When exporting a sketch from the Feature list in a Part Studio, your only option is to export as DXF/DWG in Release 11 format:

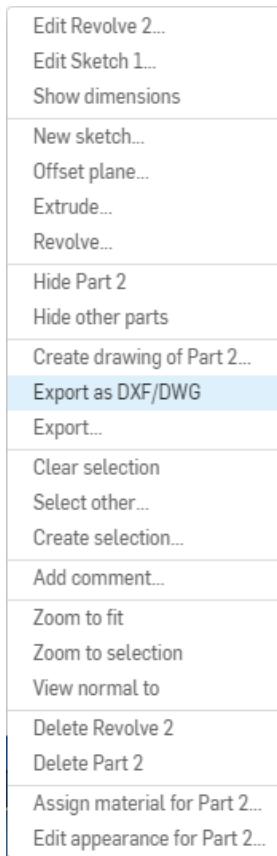


Select the format and click **Export**:



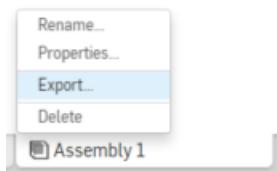
## Export a planar face from the context menu in the graphics area

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.

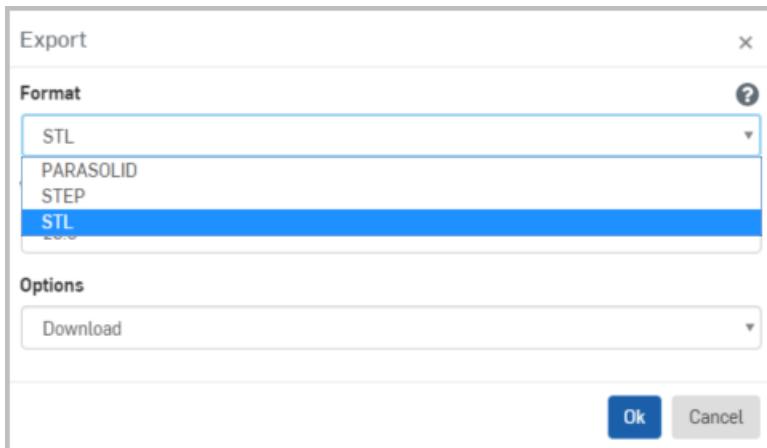


## Exporting from Assemblies

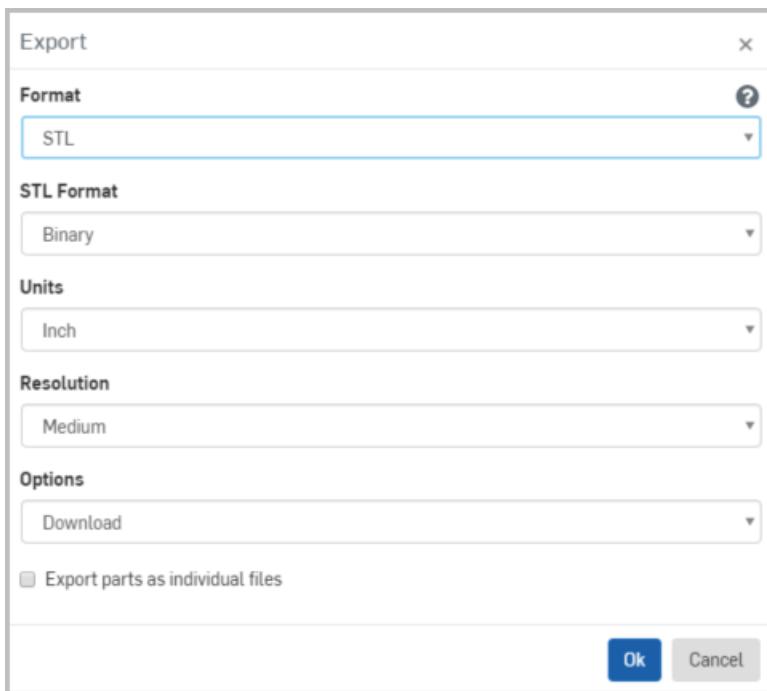
You can export all parts from Assemblies as either one file containing the entire Assembly, or as a zip file of individual files for each part in the Assembly using the tab Export option:



Select a format (note that only STL and Parasolid formats allow export of individual parts files, zipped together):

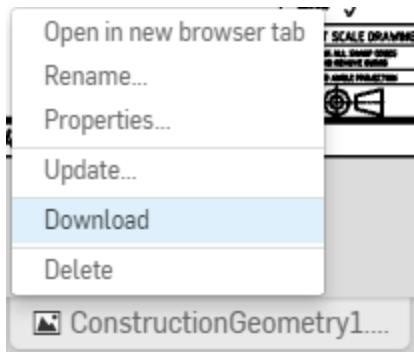


When selecting STL or Parasolid, indicate how to package the files. Check the box to download one file for each part, zipped together, or leave unchecked to export as one zip file containing one file per part:



# Downloading Files

Files that cannot be processed upon export (primarily non-CAD files) can be downloaded through the Onshape tab context menu. You can download any tab that can be represented as a file.



**Download** copies the file in its current format to your local machine, giving it the same name and file type. You can import and download any non-native file type into and out of Onshape.

# Supported File Formats

For any tab that represents a part or assembly, Onshape automatically writes it to a supported format of your choice when you initiate an export. Onshape can write Part Studios, Assemblies, and tabs containing other imported CAD files.

## For Part Studios

### Onshape can write Part Studios to:

- Parasolid v10 to v27
- ACIS 21
- STEP AP203 and AP214 (Geometry only)
- IGES 5.3
- SolidWorks 2004
- CATIA v5 R14 and R19
- STL
- Rhino (.3dm)

### Onshape can read Part Studios from:

- Parasolid v10 to v27
- ACIS up to 21, 2016 1.0
- STEP AP203 and AP214 (Geometry only)
- IGES up to 5.3
- CATIA v4 from 4.15 to 4.24
- CATIA v5 from R7 to R25 (v5-6R2015)
- CATIA v6 R2010x to R2013x, R2015x
- SolidWorks 1999 to 2015, 2016
- Inventor 9 up to 2015
- Pro/ENGINEER, Creo from Pro/E 2000i to Creo Parametric 3.0
- JT formats up to 10
- Rhino (.3dm)
- Collada 1.4.1 (without joints data)

## For Assemblies

### Onshape can write Assemblies to:

- Parasolid v10 to v27
- STEP AP203 and AP214 (Geometry only)
- STL
- Rhino (.3dm)

### Onshape can read Assemblies from:

- Parasolid v10 to v27
- ACIS up to 21
- STEP AP203 and AP214 (Geometry only)
- IGES up to 5.3
- [SolidWorks Pack and Go files](#)
- JT formats up to 10
- Rhino (.3dm)

## For Drawings

### Onshape can write Drawings to:

- AutoCAD Release 9 (from Onshape Part Studio)
- AutoCAD 2013 (from Onshape Drawing tab)

### Onshape can read Drawings from:

- AutoCAD up to 2013

For more on exporting see, "Exporting Files" on page 471.

# Real Time 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.

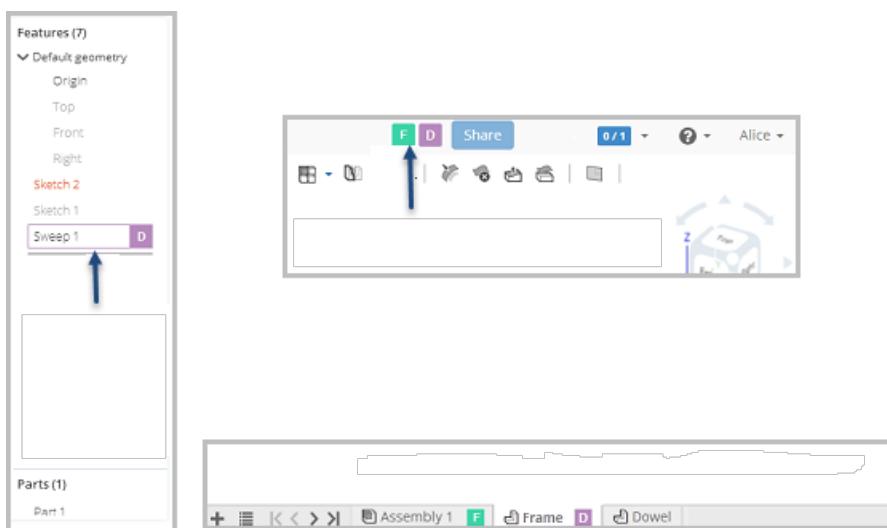
The creator of the document must Share it with the other Onshape users before they can collaborate.

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** below.)

## 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 she can see from the social cues that Diana is also working in that Part Studio. Fred is working in the Assembly (Assembly 1). Each user knows the other users are in the document, and where, based on the social cues:



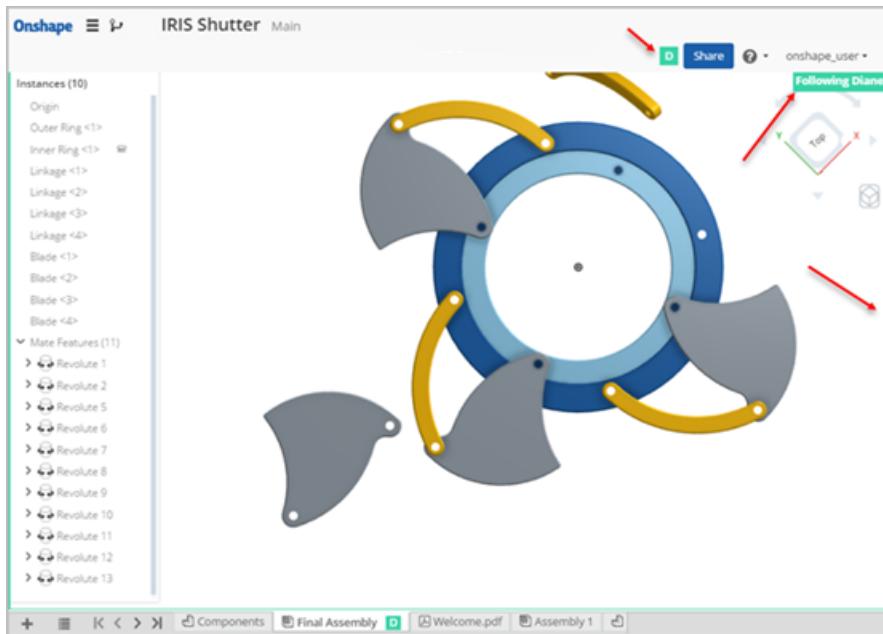
The arrows in the images above indicate the various social cues that indicate who is working in each document, Feature, or tab.

The document owner 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 482.

## 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. Note the banner and outline indicate the collaborator 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 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.

## Tips

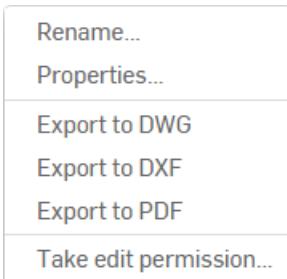
- A single collaborator can have many followers
- A follower may follow only one user at a time
- For tabs that do not support collaboration (drawings, for example, or third party applications), a user who has shared permissions to edit the document can "steal" focus on a non-collaborative tab:
  - While 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:



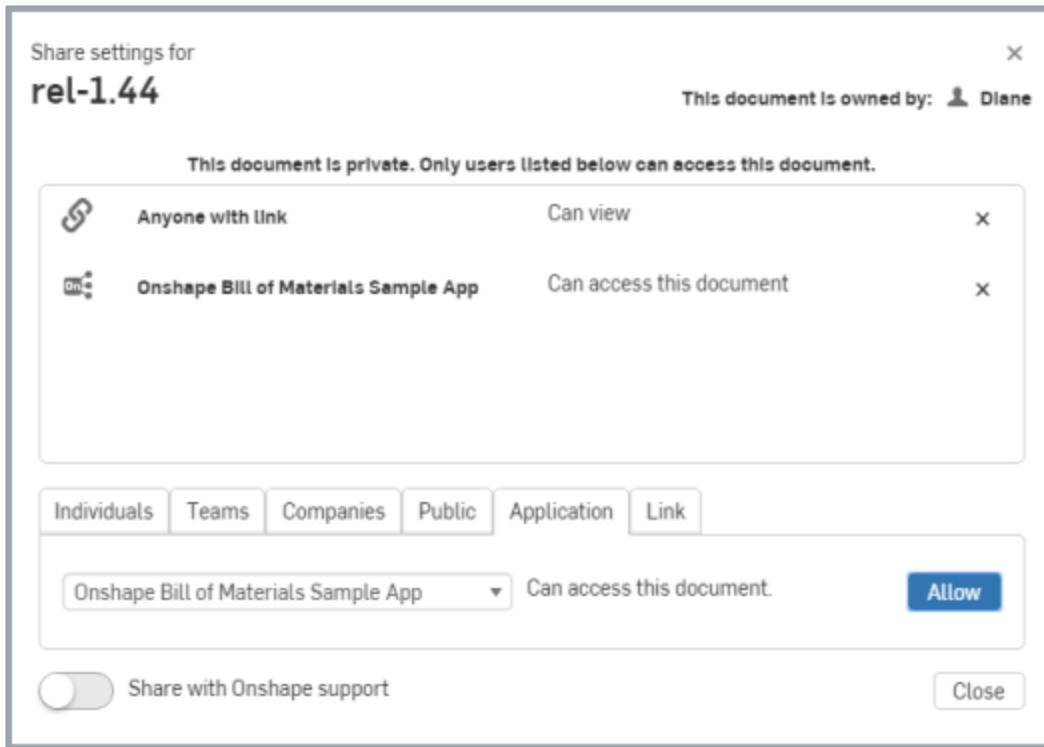
Others users will see the non-collaborative message once you have focus. When you leave the tab, it becomes available to other users again.

[Share](#)

# Share Documents

Share a document to collaborate with other designers, change permission levels, make a document publicly available, or [transfer ownership](#) to another user or company. You can also share the document with Onshape Support, if needed.

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 team or a company.

## Sharing a document

[Share](#)

1. Click [Share](#).
2. Select the appropriate tab:
  - a. Individuals - To enter one or more individual user email addresses. You can also copy and paste a comma-separated list here. Onshape provides type-ahead support and records new email addresses as you enter them.
  - b. Teams - Teams of which you are a member appear in the drop down. Selecting a team sends a share message to all members of that team.
  - c. Companies - 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.
  - 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.
  - 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](#) on in your account manager.

- f. **Link** - Copy a document-specific URL to the clipboard in order to send it 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 will have the option to sign in (with existing account credentials) or create an Onshape account.

When you create a Share document Link from within an active tab in a document, the link directs the recipient to the specific tab that was active when the link was created. If you want to direct individuals to specific tabs within a document, open the Share dialog with the desired tab active.

Your collaborators receive an email with your message, and a link to the document in Onshape. If the recipient is already an Onshape user, then he or she can click the link to access the document. If the recipient is not already an Onshape user, he or she will be directed to create an account before accessing your document.

Unshare a document with a user at any time by clicking the 'x' beside the user name in the Share dialog. Users may also remove themselves from a shared document from the gear menu on the Documents page, or through the Share dialog.

## The owner of the document

This 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 "Managing Companies" on page 555 (for those with a "Professional Subscription" on page 537 account). There may be only one owner of a document. In the case of a company owning a document, the owner permissions are actually assigned to the owner of that company.

Ownership is the highest level of permissions, giving this user the right to [transfer](#) that ownership to another user or to a company.

## Listed users

This area lists all users, companies and "Creating and Managing Teams" on page 549 that the document has been shared with. The current permission is shown in the drop down box to the right and can be changed by the owner of the document. 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.
- **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.

## Permissions

For each share operation, select the document permission level for each user or group of users:

- **Owner** - Full permission to the document including: editing, sharing, commenting, and transferring ownership.
- **Can edit & share** - Permission to edit the document, share it with other users, and comment on it.
- **Can edit** - Permission to edit the document and comment on it.
- **Can view, comment, copy & export** - Permission to view the document, comment on it, copy it (and edit the copy), and export it (but not edit it).
- **Can view, copy & export** - Permission to view the document, copy it (and edit the copy), and export it (but not edit it).
- **Can view** - Permission to view the document only (read-only).

Keep in mind that any permission that includes edit permission also includes comment permission. Any permission with **view** prohibits users from editing.

## Removing permissions

All documents with a company as the owner can be deleted only by 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.)

## 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:

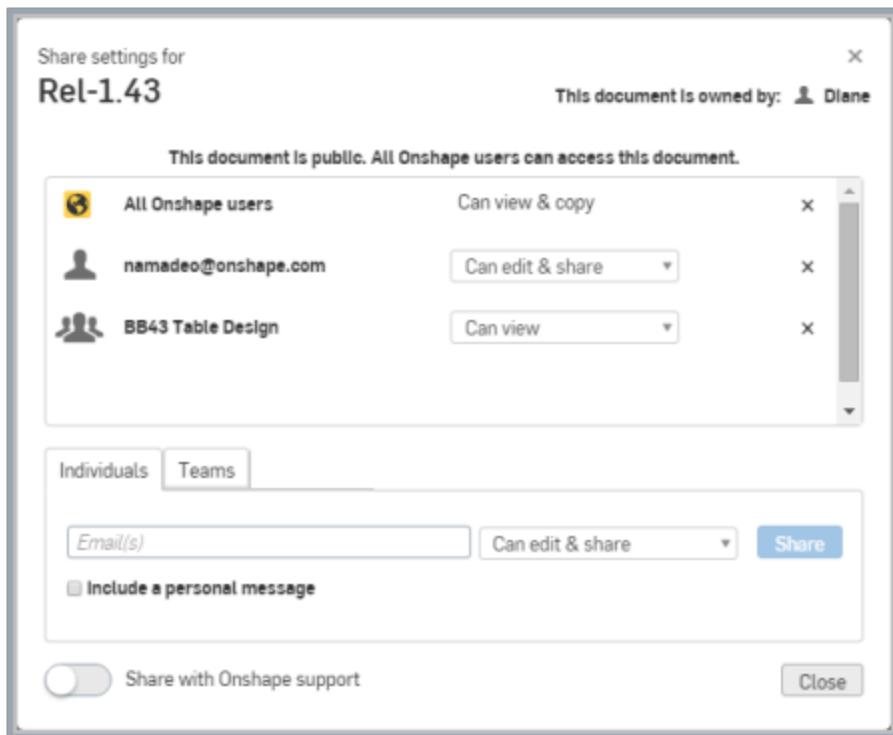
1. Select the document on the *Documents* page and click  (or click  from the open document).
2. Select the appropriate tab: Teams or Companies.
3. Select the team or company name from the dropdown list.
4. Select permissions for the team or company (and all members of that organization), as described 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 **Close** when finished.

Unshare a document at any time by clicking the 'x' beside the name.

## Making a document public

You can make a document available to all Onshape users by selecting the **Public** tab and clicking **Make public**. 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 clicking the 'x' next to **All Onshape users** in the Share dialog.



On the Documents page, public documents appear with a badge next to them:

My documents					
Name	Workspace	Modified	Modified by	Owned by	Size
Untitled document	Main	2:32 PM Today	me	me	3 KB
Free document 8	Main	2:26 PM Today	rachel amadeo	DiFree Amadeo	3 KB
Untitled document	Main	1:53 PM Today	me	Test Company	233 ...

## Sharing a document with Onshape support

If you would like help with a document or you have encountered a bug, you 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 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.

# Comments on Workspaces and Features

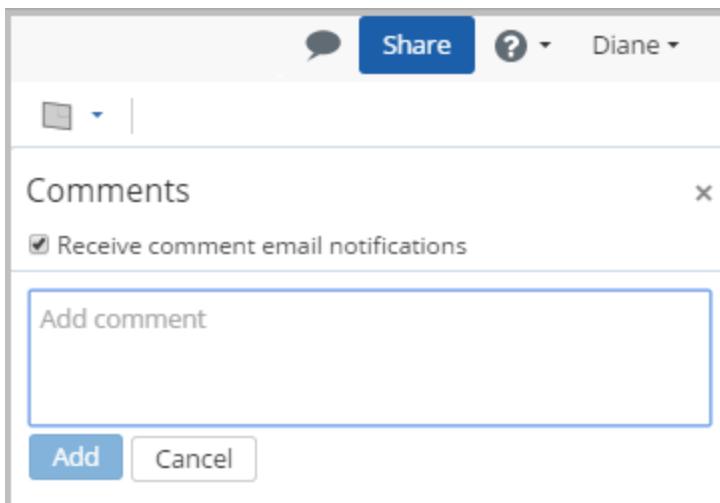
Collaborating users can communicate with each other, in a workspace, with comments. Owners of documents and those the document is shared with directly (and with 'edit' or 'comment' permission) can create comments, see each other's comments, leave replies, and opt to receive email notifications of comments.

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

Click  to open the comment flyout:



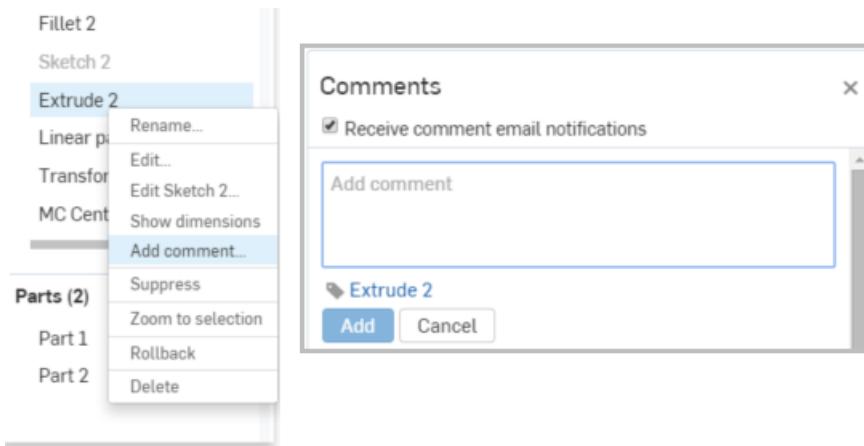
Note that the flyout remains open until you close it. Click the small x in the upper right corner of the flyout to close it.

## Adding general comments

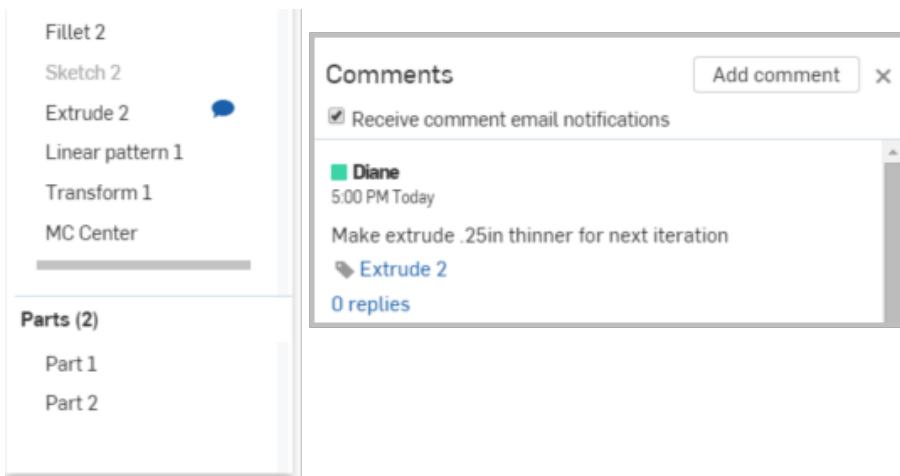
While you are in a specific tab, open the Comment flyout and enter a comment. Any collaborators who have email notification turned on will receive an email notifying them of the comment, with a link to that document. The link brings the collaborator to the first tab of the document.

## 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**:



When you click **Add** in the Comment flyout (above), a comment icon appears next to the feature in the list (below):

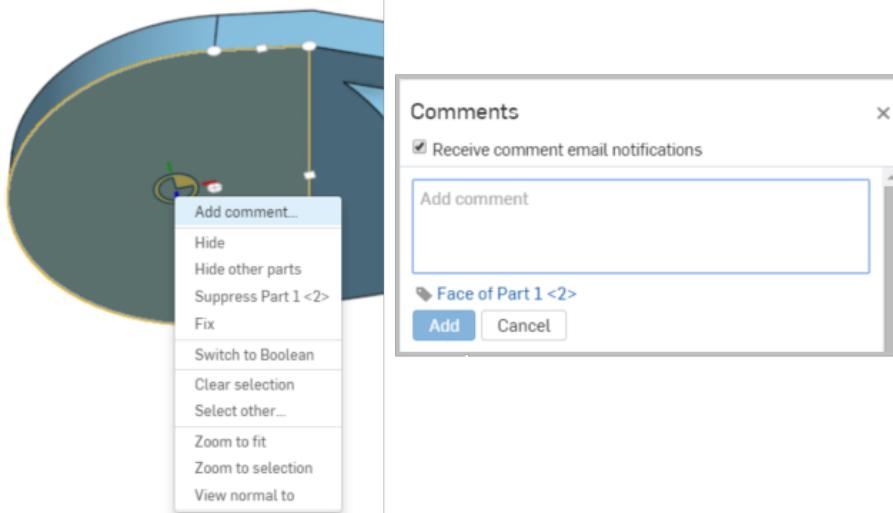


When the Comment flyout is closed, the comment icons in the Feature list disappear. The icons are visible only when the Comment flyout is open.

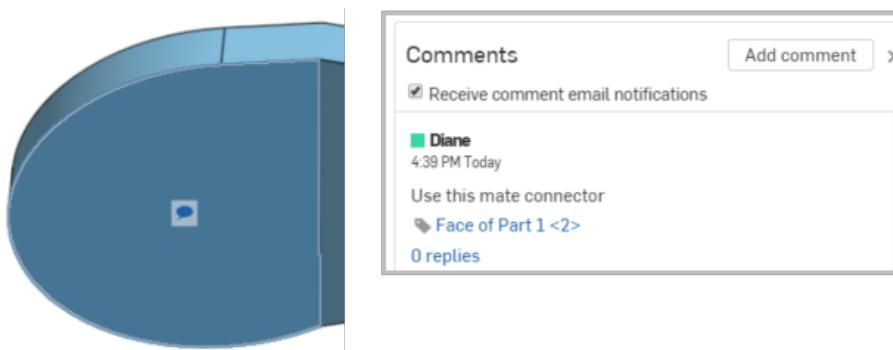
Features in the Feature list will not have comment icons in the graphics area. Their comment icons are displayed in the Feature list.

## Adding comments on implicit mate connectors

You can attach a comment directly to an implicit mate connector. Hover to activate implicit mate connectors, then right-click to access the context menu. Select **Add comment** to open a new comment on the Comment flyout:



When you click **Add** in the comment flyout (above), a comment icon appears on the implicit mate connector (below):

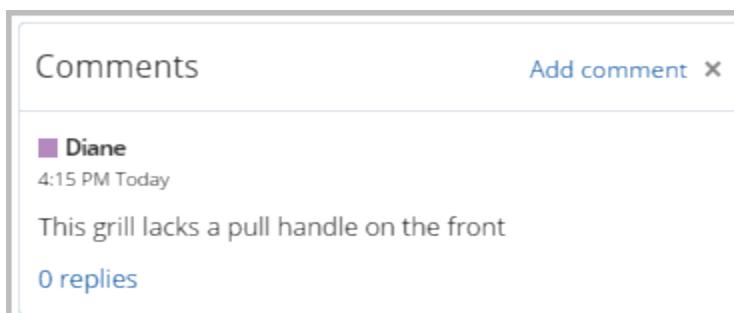


When the Comment flyout is closed, the comment icons in the graphics area disappear. The icons are visible only when the Comment flyout is open.

To delete the comment, click the small 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.



Active user comment (above)

Comments Add comment X

Diane  
4:15 PM Today

This grill lacks a pull handle on the front

0 replies

Inactive user comment (above)

## 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 can grant 'comment' permission to that user (and also revoke it). All of the permissions that allow editing automatically also allow commenting. Only the **Can view** permission does not allow commenting.

Share settings for Copy of Copy of Remote X

Who has access This document is owned by: Diane

This document is private. Only users listed below can access this document.

Share with an individual  Share with an organization  Make public

namadeo@onshape.com

Include a personal message

Can view & comment ▾

Can edit & share  
Can edit  
**Can view & comment**  
Can view

Share

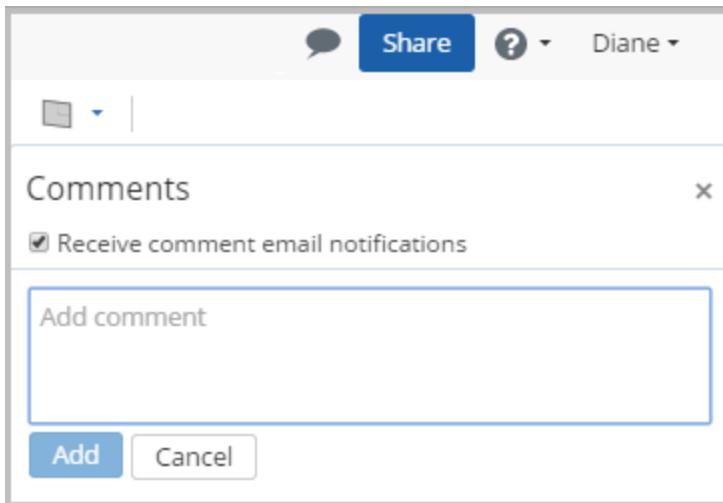
Share with Onshape support

Close

When users have comment permission on a document, they are automatically opted in to receive notification emails when:

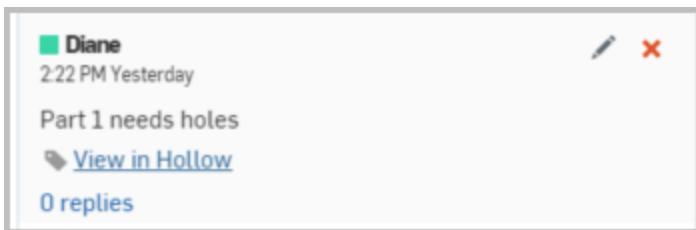
- A new comment is made
- A reply to a comment thread they have participated in is posted

Users who receive access to a document through an organizational share are not automatically opted-in for email notifications. You can, however, elect to receive email notifications by checking the box in the comment fly-out:

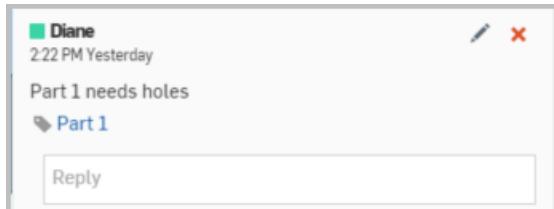


## Tips

- 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:



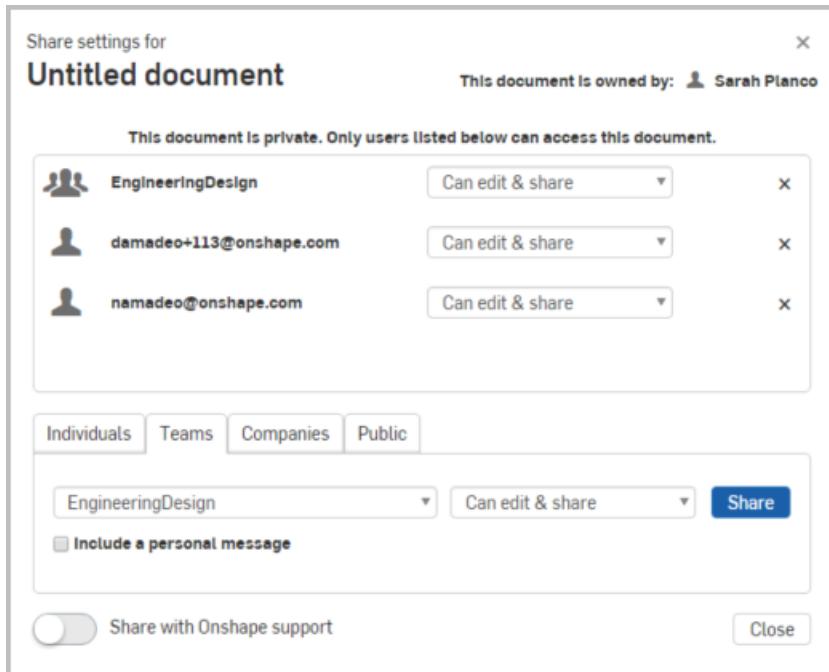
Click the new link (Part 1, above) to highlight that part in the graphics area.

- 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.

# 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 who the owner of the document is: 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](#):

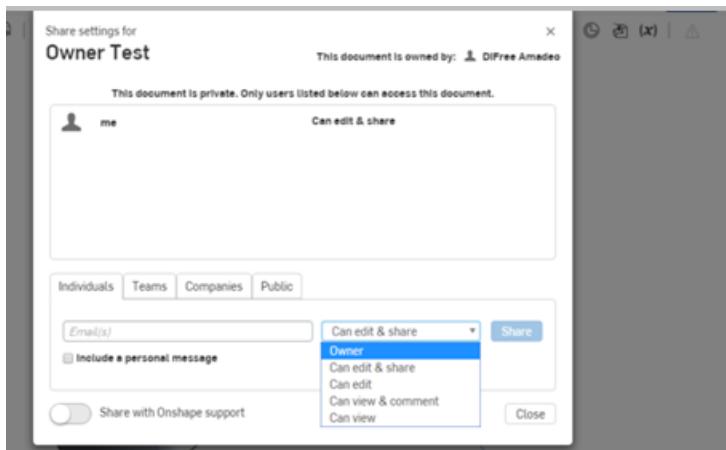


## Transferring ownership

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 498 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...**:



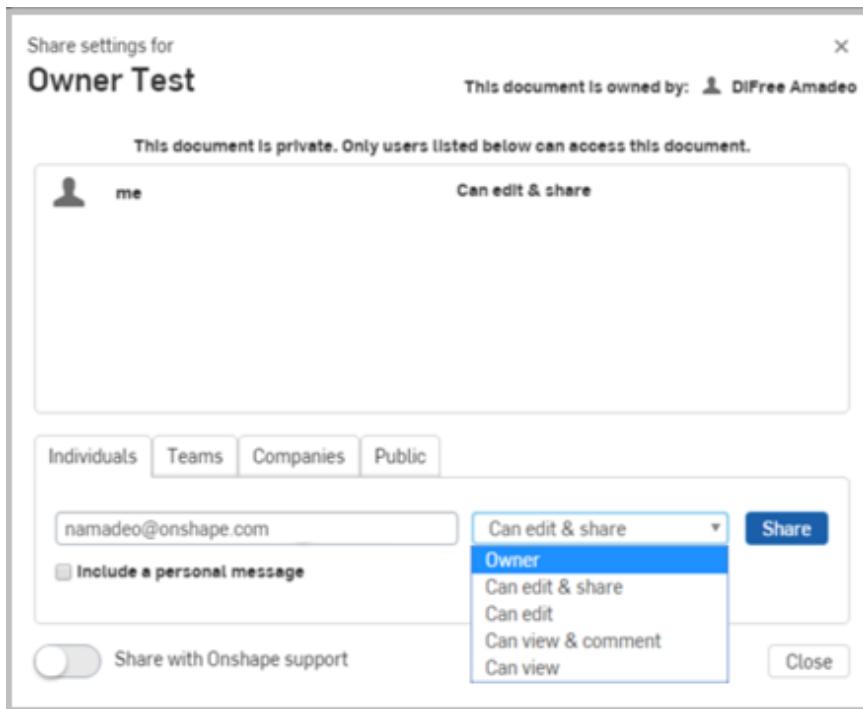
On the Documents page: select a document, expand the gear menu, and select Share:

Recently opened					
Name	Workspace	Modified	Modified by	Owned by	Size
<b>Owner Test</b> Main		10:28 AM Today	me	me	169 KB ⚙️
transferownership test	Main	9:32 AM Oct 30	me	me	Open
Untitled document	Main	9:48 AM Oct 9	Nicholas Amadeo	me	Open in new browser tab History Versions
rel-1.38	Main	11:00 AM Oct 29	me	Eng	Rename document... Copy workspace...
Halloween	Main	1:42 PM Oct 26	me	Eng	Share... Make public...
copy-AutoAssemble	Main	2:32 PM Oct 22	me	me	
Assemble Demo	Main	1:23 PM Oct 20	Abe Feldman	Abi	
center point arc	Main	11:45 AM Oct 16	me	me	

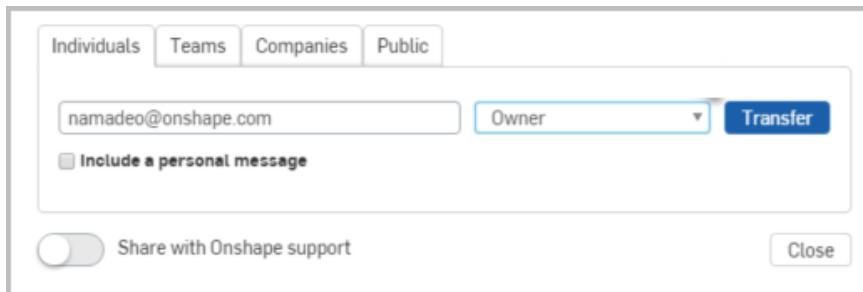
- Once the Share dialog is open, enter the user's email address on the *Individuals* tab.

To transfer ownership to a company, select the *Companies* tab and select the company in the dropdown.

3. Select the permission level (Owner) from the drop down:

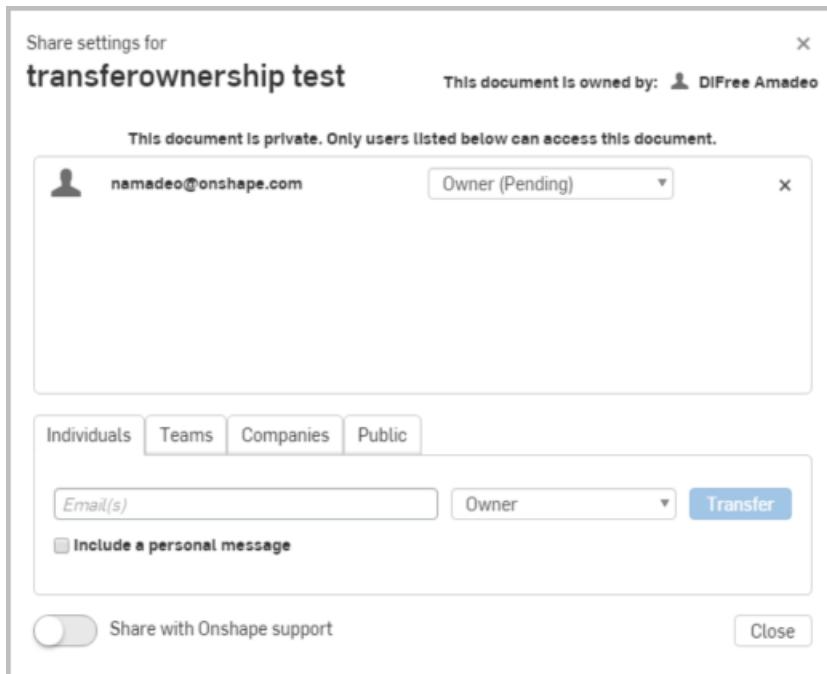


4. Click Transfer:



5. Click Close.

At this point, the transfer of ownership is pending until that user [accepts the transfer request](#).

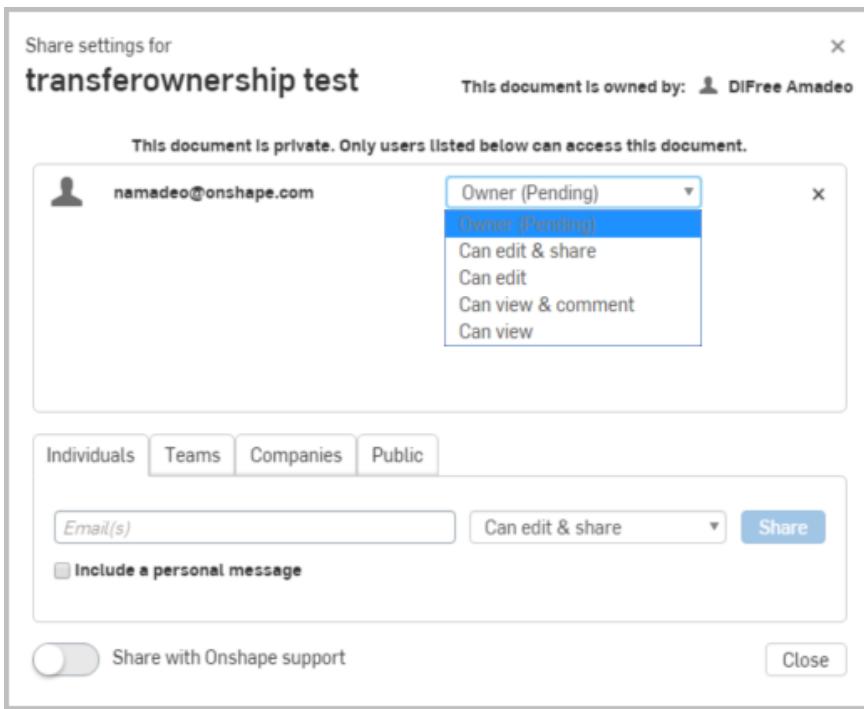


If both users are part of a the same company plan, and when transferring ownership to a company, the transfer is automatic and doesn't require explicit acceptance. This also means that there is no opportunity to revoke the transfer.

## 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.



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:

Owned by	Size	More Options
DiFree Amadeo (Transfer pend...	4 MB	

To complete the transfer, the recipient must accept the transfer:

1. Use the gear menu and select Accept Ownership:

Name	Workspace	Modified	Modified by	Owned by	Size
planes	Main	2:01 PM Nov 9	me	me	366 KB
Owner Test	Main	10:28 AM Nov 2	me	Nicholas Amadeo	174 KB
loftTest2	Main	4:57 AM Nov 2	Radha Krishna Tirrey	Radha Krishna Tirrey	166 KB
transferownership test	Main	9:32 AM Oct 30	me	me	4 MB
Assemble Demo	Main	1:23 PM Oct 20	Abe Feldman	Abe Feldman	29 MB
Free document 8 - Copy	Main	2:28 PM Aug 19	Smith Smythe	Smith Smythe (Transfer...)	3 KB
Public	3-doc	2:27 PM Aug 13	Di-ProCo Amadeo	Di-ProCo Amadeo	
doc 1	Main	3:21 PM Aug 10	DiPro Amadeo	DiPro Amadeo	

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** This document is owned by: Smith Smythe

This document is private. Only users listed below can access this document.

damadeo+113@onshape.com Can edit & share

me Transfer pending **Accept** Decline

Individuals Teams Companies Public

Email(s) Can edit & share Share

Include a personal message

Share with Onshape support Close

## 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:

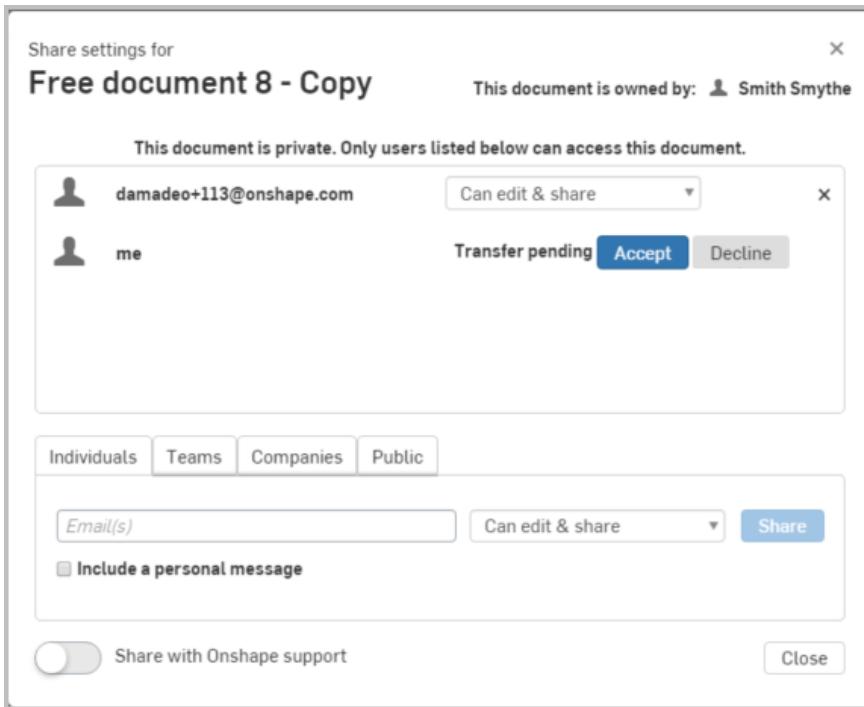
Owned by	Size
DiFree Amadeo (Transfer pend...)	4 MB

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:

Shared with me					
Name	Workspace	Modified ▾	Modified by	Owned by	Size
planes	Main	2:01 PM Nov 9	me	me	366 KB
Owner Test	Main	10:28 AM Nov 2	me	Nicholas Amadeo	174 KB
loftTest2	Main	4:57 AM Nov 2	Radha Krishna Tirrey	Radha Krishna Tirrey	166 KB
transferownership test	Main	9:32 AM Oct 30	me	me	4 MB
Assemble Demo	Main	1:23 PM Oct 20	Abe Feldman	Abe Feldman	29 MB
Free document 8 - Copy	Main	2:28 PM Aug 19	Smith Smythe	Smith Smythe (Transfer ...)	3 KB
3-doc	Main	2:27 PM Aug 13	Di-ProCo Amadeo	Di-ProCo Amadeo	Open
doc 1	Main	3:21 PM Aug 10	DiPro Amadeo	DiPro Amadeo	Open in new browser tab History Versions
					Rename document... Copy workspace... Share... Make public... Accept ownership Decline ownership Remove me from share
					Open and record
					Extract With Foreign Data Extract

2. Or click the Share button (from the Documents page or from within the open 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

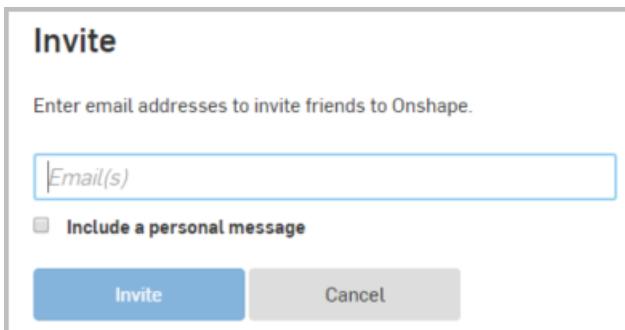
- If you create a document and give ownership to a company, you gain Full access permission to that document, allowing you to edit, share/change permissions, make Public/Private, and delete the document. However, since the company owns the document, the company owner can then remove your access to the 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
- In order for a Free user to accept ownership, they must be below their plan limits
- 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

# Invite Friends

Invite colleagues to join the Onshape community!

1. Click **Invite friends**.



2. Enter a valid and active email address, or comma-separated list of email addresses.
3. Check the box to include a personal message; enter a message in the text box that appears.
4. Click **Invite**.

# Document Management

Onshape captures the state of every tab in a workspace every time an edit is completed (by all users working in that workspace). 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 can always make changes with confidence that if the changes don't work out, you can find and restore any earlier state. In addition, you can always use [version, branching, and merging](#) to explore multiple design variations in parallel, either on your own or with collaborators.

## Terms

- **Version** - A named state of the document. Versions are immutable and separate from workspaces. To capture a workspace at a particular point in time, save it as a version through the **Save version** button in Version Manager.
- **Workspace** - An active modeling/design space.
- **Active branch** - The branch of the document in which the currently open version or workspace is located.

## 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.



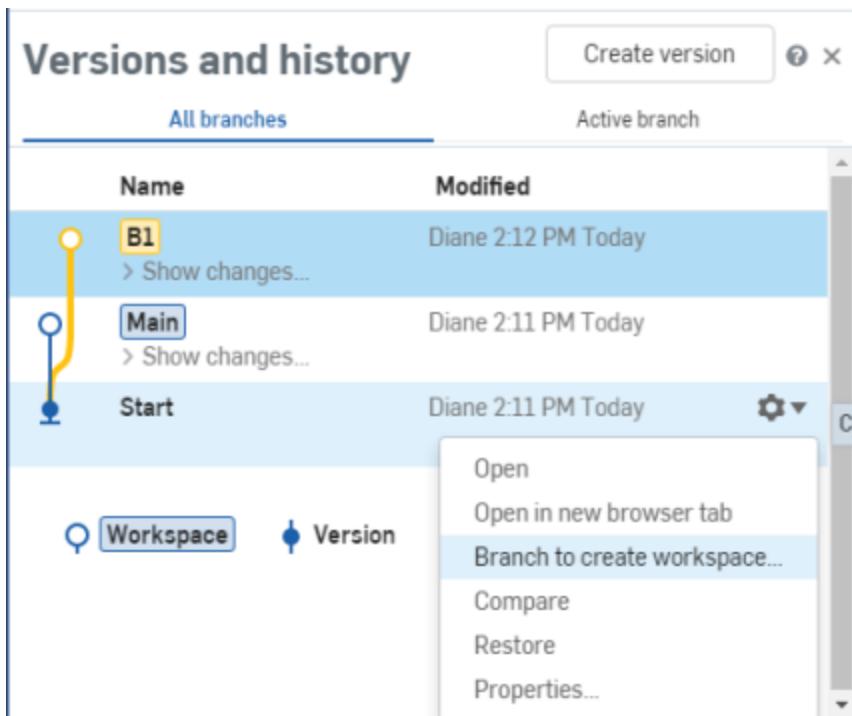
The icons in the image above (from left to right) are: Document menu, Versions and history flyout, and Create a version. The bold name is the Document name and the lighter name is the Workspace name.

A screenshot of the 'Versions and history' flyout. It lists two versions: 'Main' and 'Start'. The 'Main' version is highlighted with a blue background and a blue dot icon. Below it, there is a link 'Show changes...'. The 'Start' version is shown below with its modification date. At the bottom of the flyout, there are three navigation buttons: 'Workspace' (highlighted), 'Version', and 'Change'.

The Main workspace and Start version are automatically created for you. These can be renamed.

## About versions

A **version** is a snapshot of a document at a particular point in time. The geometric data (and the accompanying data like Part names, etc.) of that version is unchangeable. You can, however, change the meta data of a version (more on this later). You can [create many versions](#) of a document. You can also [branch](#) your work at a version:



The features shown in the image above include (from top to bottom):

- Create version button, which allows you to create a version from the currently open workspace.
- The currently open workspace, highlighted in dark blue.
- The currently selected workspace, highlighted in light blue.
- The Branch command, in the drop down menu.

## Create a version

To create a version behind the scenes and still work in the same workspace, use .



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 indicates the incremented number, starting from 1. You can always rename the versions.

**Versions and history**

All branches	Active branch
Name	Modified
B1 Show changes...	Diane 2:14 PM Today
V2 Show changes...	Diane 2:14 PM Today
V1 Show changes...	Diane 2:14 PM Today
Main Show changes...	Diane 2:11 PM Today
Start	Diane 2:11 PM Today

Workspace Version Change

The first arrow in the image above points to V2, the second version saved through .

The second arrow in the image above points to V1, the first version saved through .

## Accessing version and history information

Managing the workflow around versions and workspaces is performed through the **Versions and history flyout**, accessed by clicking .



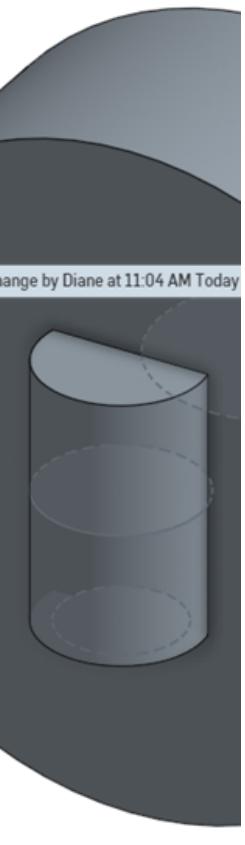
Onshape automatically records the state of each tab (Part Studio, Assemblies, etc) at each persisted change made to all tabs for every workspace by every user. This history of modifications is listed in the flyout. At any time you can click **Restore** to restore the branch/workspace to a particular point in its change history, click **Return to <branch-name>** to return to the currently active branch at its current, and click a history point to visualize the design at this point.

**Onshape**  **Versioning** Branch-1

Viewing Part Studio 1 :: Insert feature : Sketch 3 | [Restore](#) | [Return to Branch-1](#) | 

**Versions and history**

All branches	Active branch
<b>Name</b>	<b>Modified</b>
Branch-1 ▼ 16 changes	Diane 11:06 AM Today
Part Studio 1 :: Hide : Sketch 2	
Part Studio 1 :: Edit : Extrude 4	
Part Studio 1 :: Insert feature : Extrude 5	
Part Studio 1 :: Insert feature : Sketch 4	
Part Studio 1 :: Insert feature : Extrude 4	
Part Studio 1 :: Insert feature : Sketch 3	 Change by Diane at 11:04 AM Today
Part Studio 1 :: Hide : Right	
Part Studio 1 :: Hide : Front	
Part Studio 1 :: Hide : Top	
Part Studio 1 :: Hide : Origin	
Part Studio 1 :: Insert feature : Extrude 3	
Part Studio 1 :: Edit : Sketch 2	
Part Studio 1 :: Show : Sketch 2	
Part Studio 1 :: Insert feature : Extrude 2	
Part Studio 1 :: Insert feature : Sketch 2	
Part Studio 1 :: Update part metadata from Version	
<b>Main</b> 0 changes	Diane 10:53 AM Today
Main version ► 2 changes	Diane 10:53 AM Today
Start	Diane 10:58 AM Yesterday



This graph displays all versions and workspaces 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.

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 can:

- 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 513 the document at two history points.

## Meta data for workspace and versions

You can create and edit meta data for parts, Part Studios, Assemblies, and foreign data in order to support your preferred design processes.

Meta data is defined and edited through the Properties dialogs found on the context menus:

- **Parts** - Through the context menu on the part listed in the Part Studio Feature list in Part Studios, and through the Properties dialog for a version or workspace through the Version Manager.
- **Part Studios and Assemblies** - Through the context menu on the tab and through the Properties dialog for a version or workspace through the Versions and history flyout.
- **Foreign data files**- Through the context menu on the tab and through the Properties dialog for a version or workspace through the Version Manager.

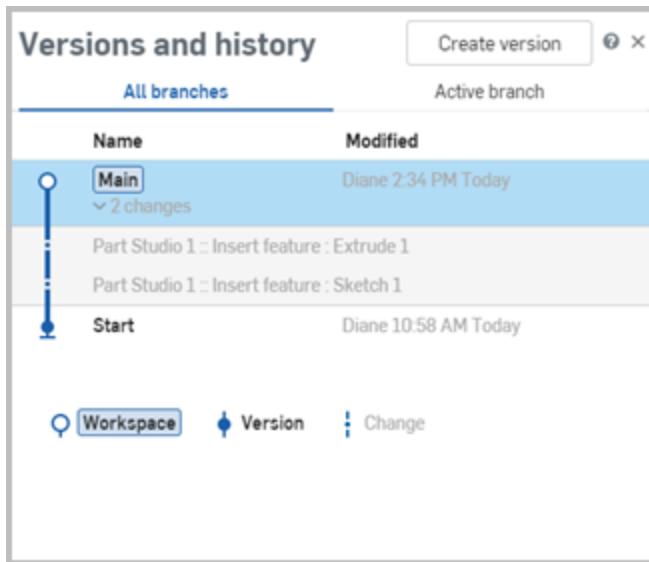
# Versioning and Branching

You can create versions (which are View only) and branch a version to create a new workspace. You can also "Comparing" on page 513 workspaces and versions, any combination of the two.

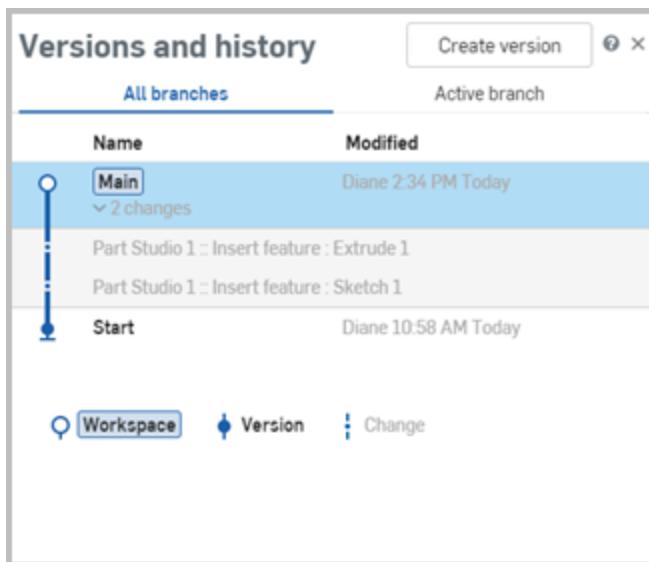
## Creating a version

To create a version of a document:

1. With a document open, click  to open the Versions and history flyout:



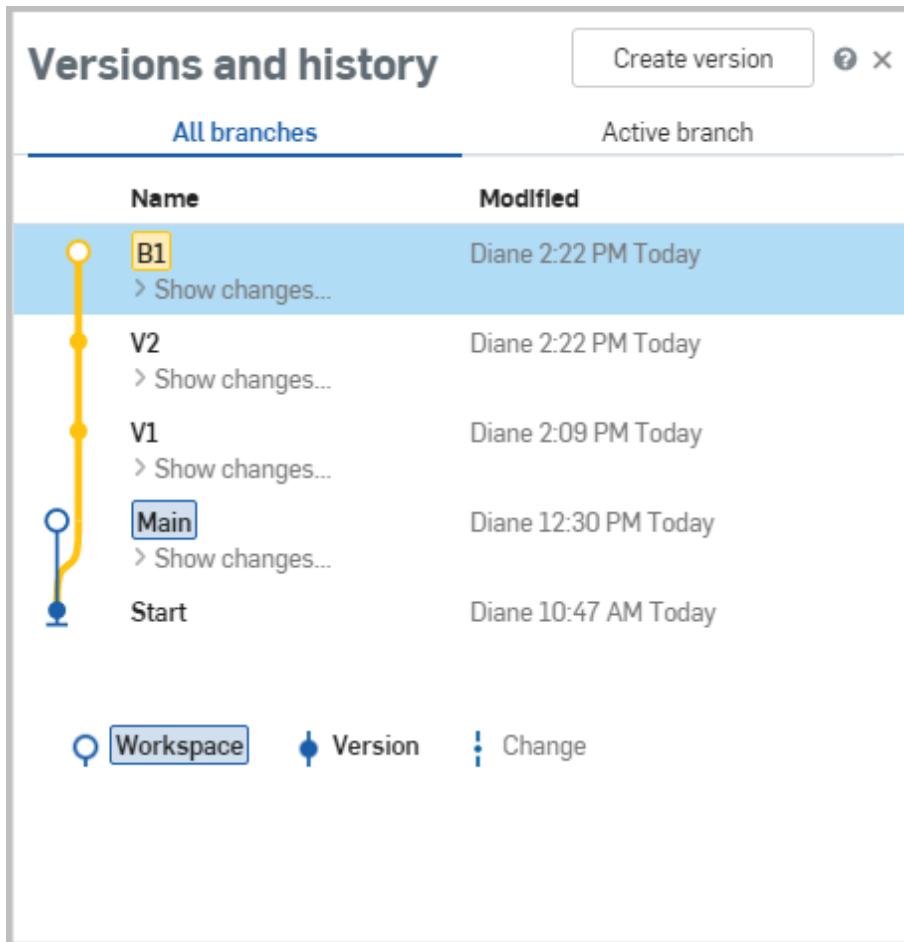
2. Click the workspace from which to create a version; this makes that workspace active.
3. Click Create version:



4. In the dialog that appears, enter a name and description for the new version.

5. Click:
  - a. Save - Save the new version and remain in the currently active workspace
  - b. Save and edit version properties - Save 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.

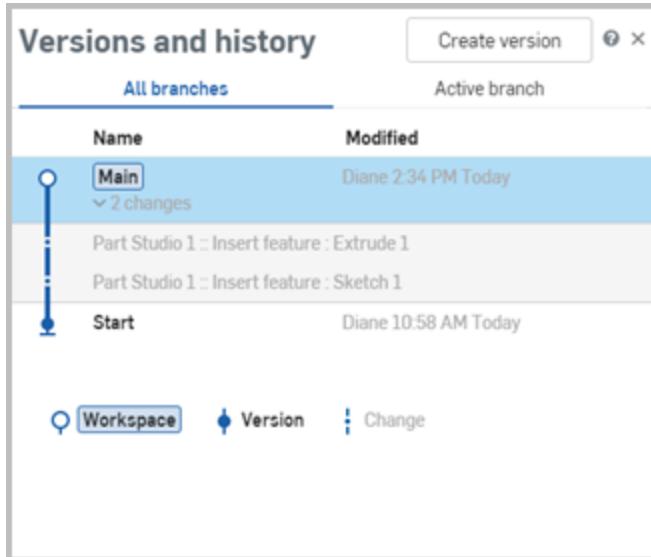
Note that the new version is shown in the Versions and history graph.



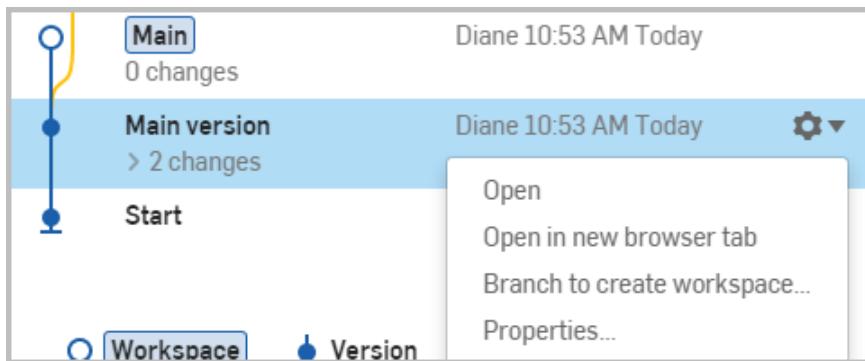
## Creating a branch

To create a branch of a document:

- With a document open, click  to open the Versions and history flyout:

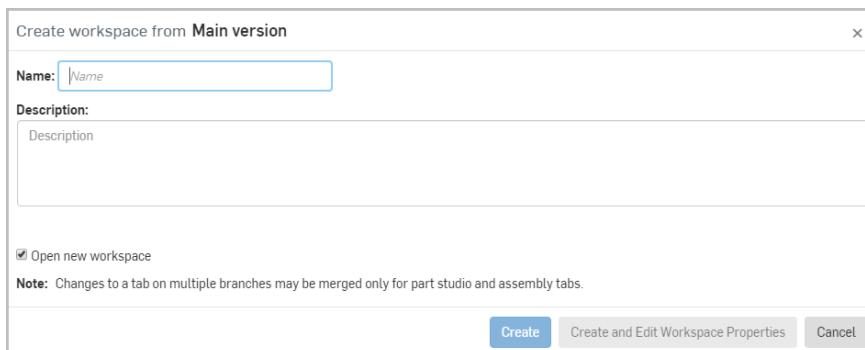


- Hover over the version entry from which to create a branch and click the Gear menu that appears:



- Select **Branch to create workspace**.

- In the Properties dialog that appears, enter the name and description for the branch:



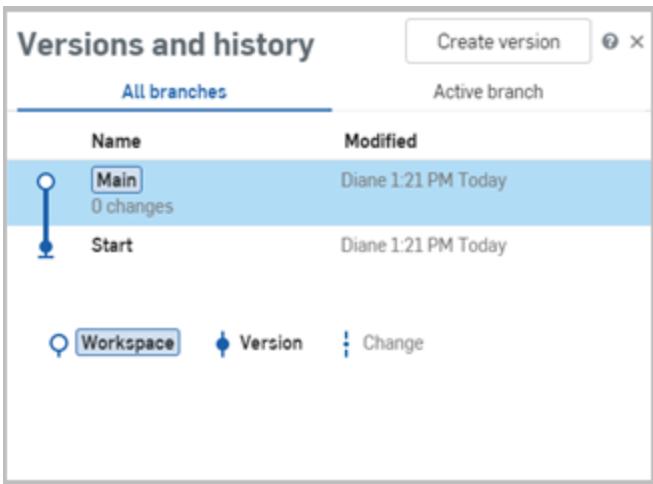
5. Click:
  - a. Create - Create the branch and optionally open the new workspace (check box)
  - b. Create and Edit Workspace Properties - Create the branch (optionally open it) and also open the Properties dialog for the new workspace. This Properties dialog includes names and descriptions for each tab and part of the new document workspace.

You can [compare](#) two workspaces, two versions, or a version and a workspace. You can also [merge](#) one or more branches.

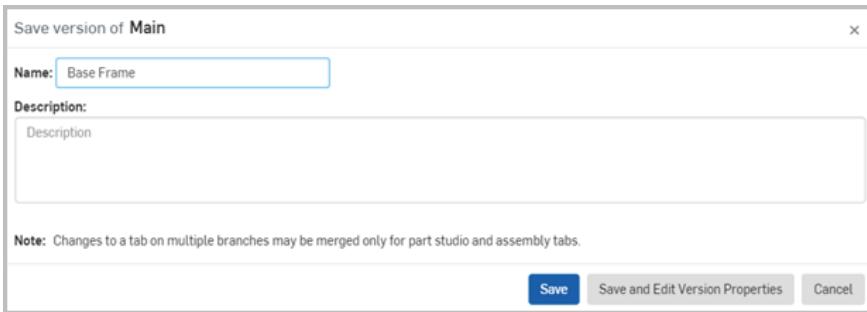
## Branching example

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 **Version tool**  to open the Version Manager:



2. Click .
3. Name the first version *Base Frame*; click **Save**:



Name	Modified
Main	Diane 1:24 PM Today
Base Frame	Diane 1:24 PM Today
Start	Diane 1:21 PM Today

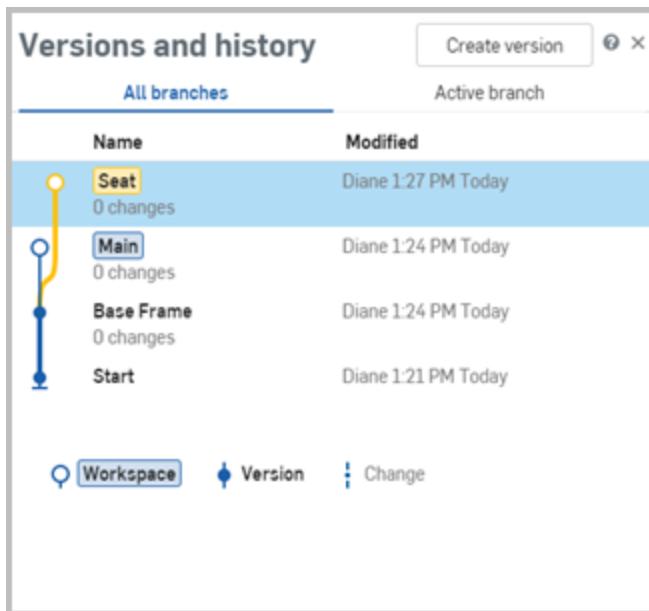
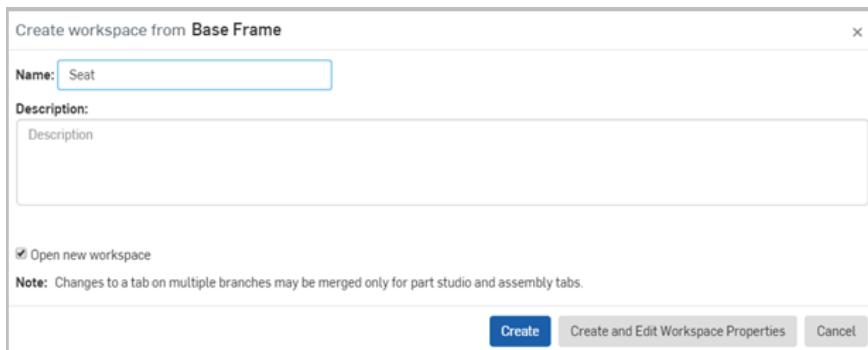
Workspace   Version   Change

Each designer can then create their own workspace from the *Base Frame* version, perhaps labeled *Seat, Brakes, Shocks*.

- From the flyout, on Base Frame version, select **Branch to create workspace...** from the gear menu.

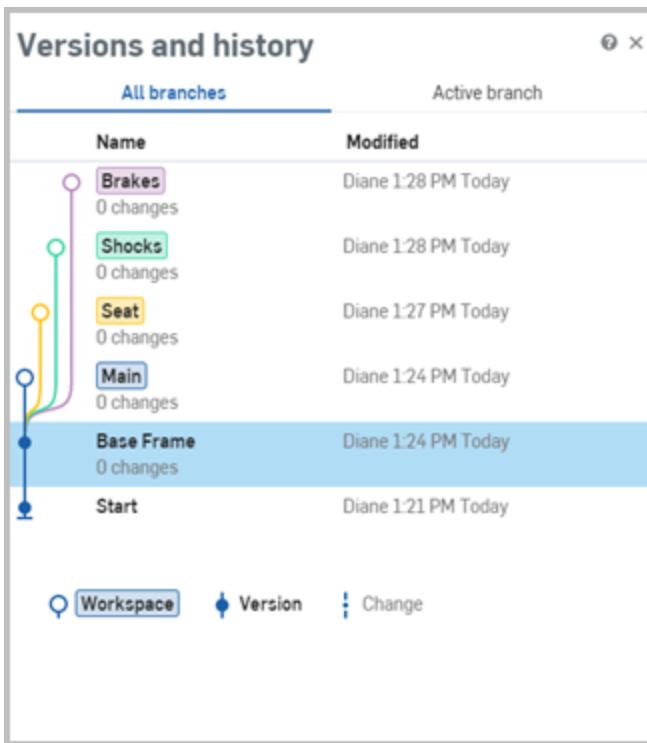
The 'Branch to create workspace...' option is highlighted in the context menu.

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:



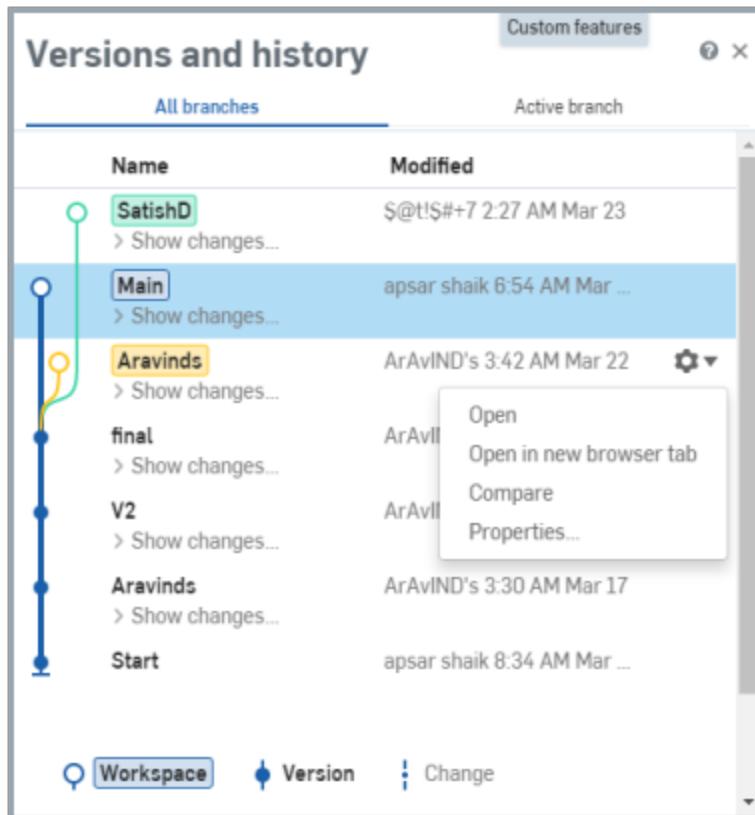
The workspace of each designer is, at this point, identical to the *Base Frame* version. As they continue to design in their own branches, their designs evolve, independently of the Base Frame version and independently of each other.

As they work, they are free to create versions of the workspaces.

# Comparing

Onshape provides a mechanism for graphically and discretely comparing versions and workspaces. You can compare any combination of the two.

Access the Compare action from the drop-down menus in the Versions and history flyout:



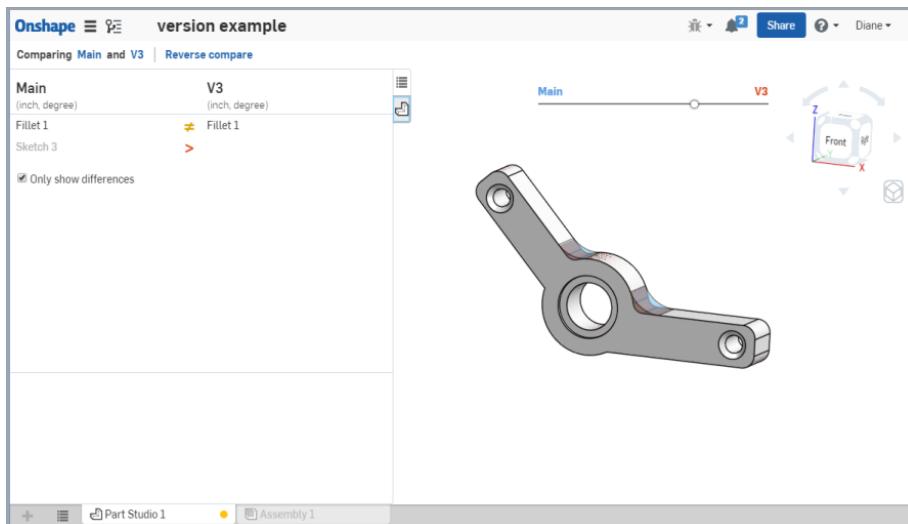
## How it works

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 (the Target), click the menu arrow beside the Target and select **Compare**, as shown above.

## What you see

By default, Compare shows:



## 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 just between the current Part Studio and target Part Studio, see the illustration above.

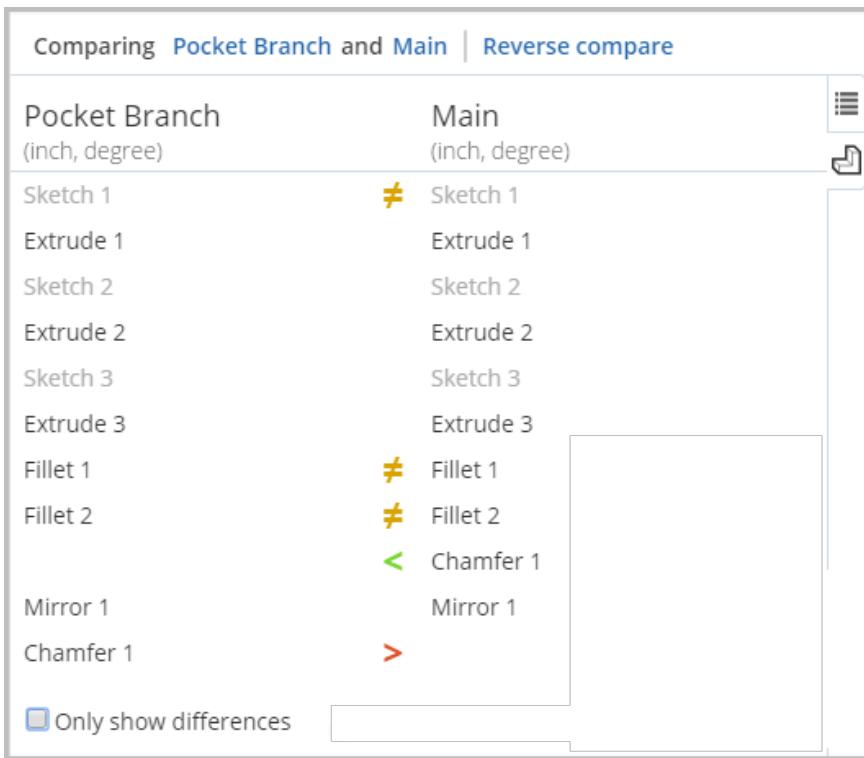
Comparing Assemblies is not yet supported.

3. List all features of both the Base and Target (instead of just the differences):

Comparing **Pocket Branch** and **Main** | Reverse compare

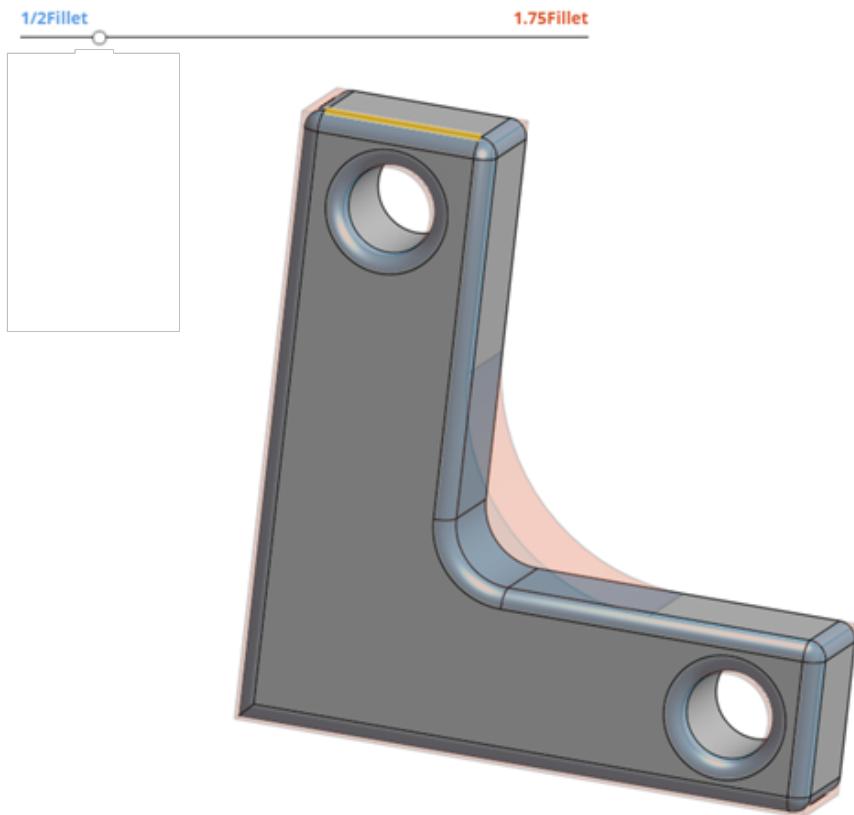
Pocket Branch (inch, degree)	Main (inch, degree)
Sketch 1	≠ Sketch 1
Extrude 1	Extrude 1
Sketch 2	Sketch 2
Extrude 2	Extrude 2
Sketch 3	Sketch 3
Extrude 3	Extrude 3
Fillet 1	≠ Fillet 1
Fillet 2	≠ Fillet 2
	< Chamfer 1
Mirror 1	Mirror 1
Chamfer 1	>

Only show differences

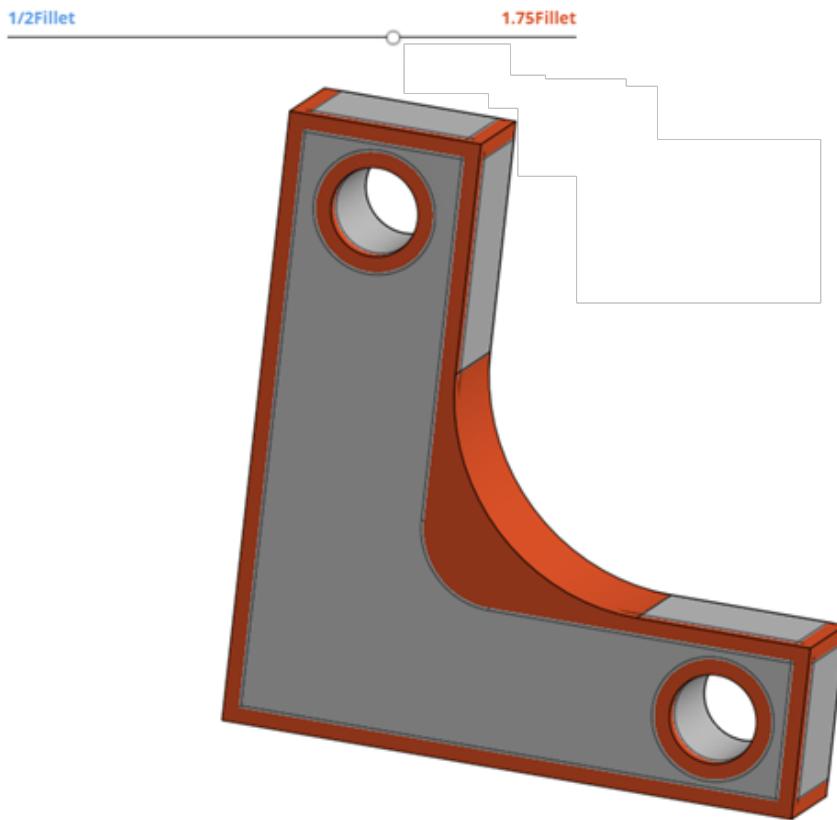


4. Control how much of the Base or Target features are displayed graphically through the slider mechanism:

Visualize more of the Base features by sliding the circle towards the Base label:



Visualize more of the Target features by sliding the circle towards the Target label:



5. Select the tab to see a list of all Part Studios in the document and the differences in more detail:

Main	V3
(inch, degree)	(inch, degree)
Tabs (2)	Tabs (2)
Part Studio 1	Part Studio 1

## Interpreting the lists

The list comparing features or Part Studios between the Base and Target uses the following icons to denote differences:

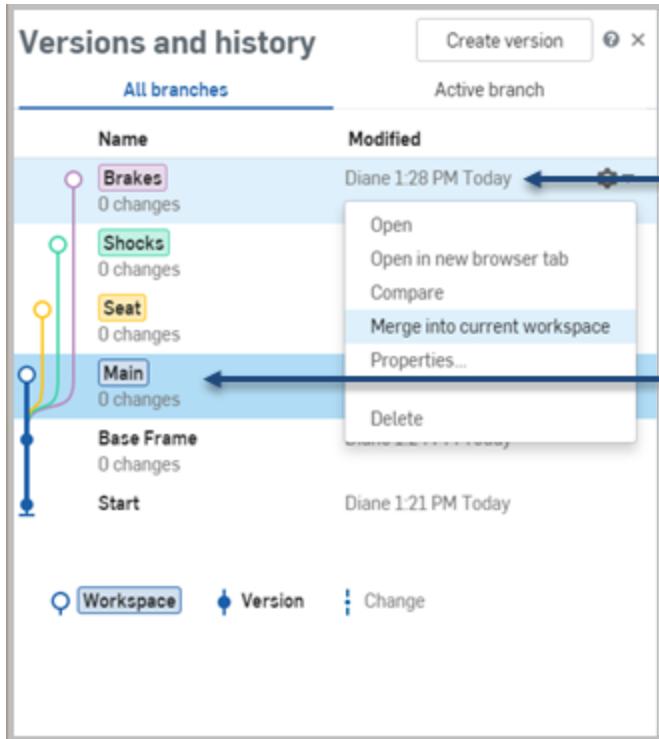
- no icon - The Base and Target feature, sketch, or Part Studio are identical
- - The Base and Target feature, sketch, or Part Studio are not identical
- - The Base has a feature, sketch, or Part Studio that the Target does not have (this feature or Part Studio does not exist in the Target)
- - The Base is missing a feature, sketch, or Part Studio that the Target has (this feature or Part Studio does not exist in the Base)

Note that Compare is only for viewing differences. No action can be taken in this view to revert changes or restore to a previous point in time. For those actions, open the Versions and history flyout .

# Merging

Onshape provides a mechanism for merging from a document version or workspace (referred to as the Source) into your currently active 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.

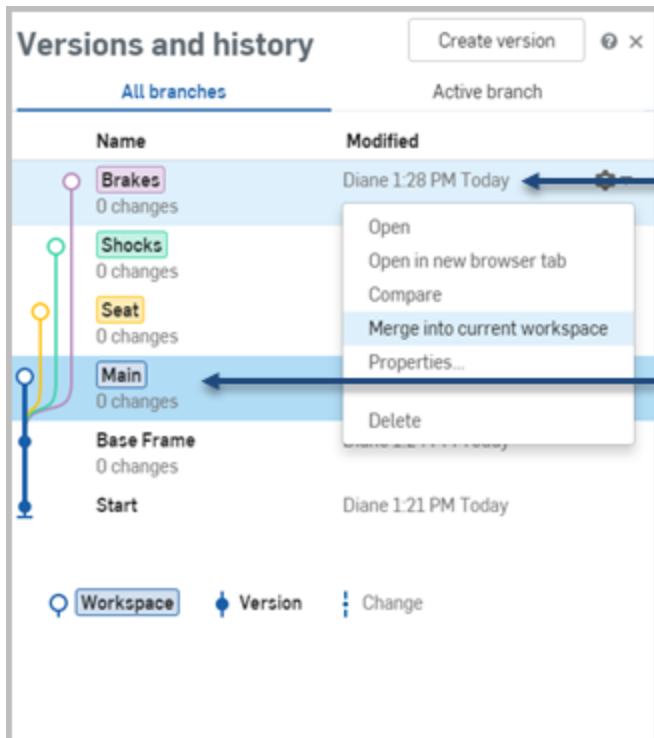
Access the Merge action from the drop-down menu in the Version Manager flyout:



## How it works

When you open the Version Manager 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), click the menu arrow beside the Source and select **Merge into current workspace**.



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.

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.

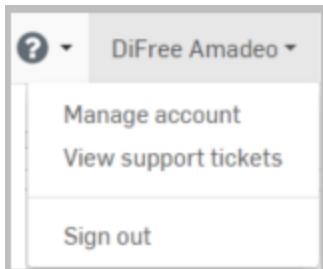
# Managing Your Onshape Account

Your Onshape account includes access to Onshape via:

- Browser, via: Chrome, Firefox, and Safari
- Mobile devices, including: iPad, iPad mini, iPod, iPhone, and Android devices

To learn more about mobile devices and operating systems supported, see "Onshape Mobile Devices" on page 19. For more information about Onshape on a mobile device, watch this video: Mobile Devices Video.

Click your name in the upper right corner to access your Onshape account Information.



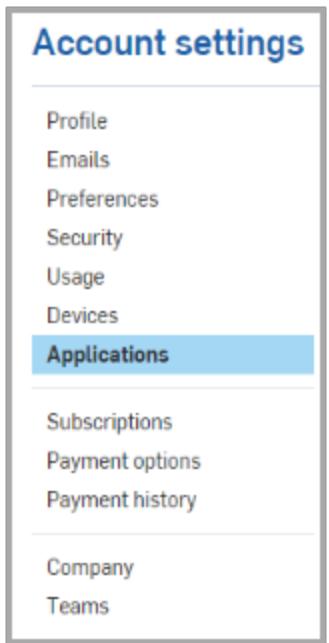
For Free subscription users, the top of the page contains information about metrics. Professional subscription users do not have this banner because there are no limits.



Use **Manage account** to view and manage your profile and user account, including:

- **Profile** - Name, username, nickname, biographical information, and company name
- **Emails** - Email addresses associated with your account
- **Preferences** - Unit preferences for all documents you create, including for length, angle, and mass
- **Security** - Reset your password and enable/disable two-factor authentication
- **Usage** - Number of private documents, private document storage, and total storage
- **Devices** - The mobile devices authorized to use your account
- **Applications** - A list of the third-party applications which you have purchased and allowed to access your Onshape account, including the ability to control application access to Onshape documents manually.
- **Early Visibility** - A list of early functionality that you may opt to use on a test basis
- **Subscriptions** - The details of all Onshape subscriptions for which you are a member, including payment details; you can cancel, upgrade, and change credit card information here
- **Payment options** - Basic credit card information associated with any account for which you are responsible for payment
- **Payment history** - A list of all charges made against your account
- **Company** - This tab appears when you are either the owner of a Company Professional subscription, or have been added to such; also lists the basic company information as well as all users associated with the company
- **Teams** - Teams you are a member of; ability to create teams (if allowed by your subscription type) and view a list of members

The remainder of this topic explains the tabs on this page:



## Profile

Onshape automatically records the first and last names you specify during sign up; here you can also enter an optional company name, and a personal nickname for display in the system (in the upper right-hand corner of the user interface).

## Profile

First name

Last name

Username (Onshape forum name)

diane

Nickname (Your name as seen by other users)

Diane

Bio

Share something about yourself

Company name

**Update profile**

## Emails

You can specify up to three email addresses with which to access your Onshape account. One of the addresses 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.

Note that any email addresses associated with an account (even those not designated as primary) cannot be used to create another Onshape account.

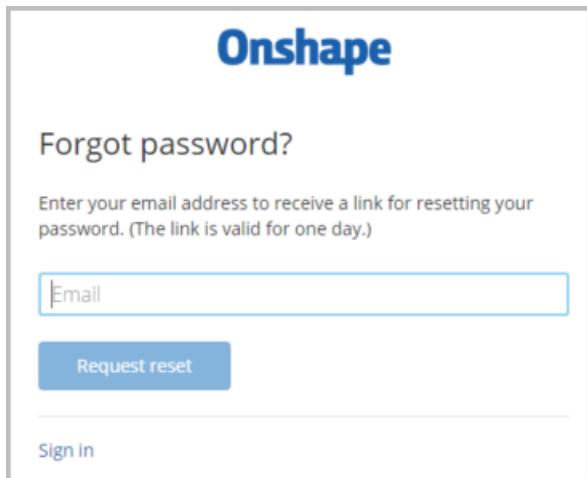
The screenshot shows the 'Email' settings page. It lists three email addresses:

Type	Email Address	Status	Action Buttons
Primary	User@company.com		
Other	Otheremail@gmail.com	Verified	<button>Make primary</button> <button>x</button>
	Otheremail2@home.com	Unverified	<button>Resend verification email</button> <button>x</button>

At the bottom, there are two buttons: 'New email' and 'Add'.

Remove an email from your account by clicking the small "x" next to the email listing (shown above).

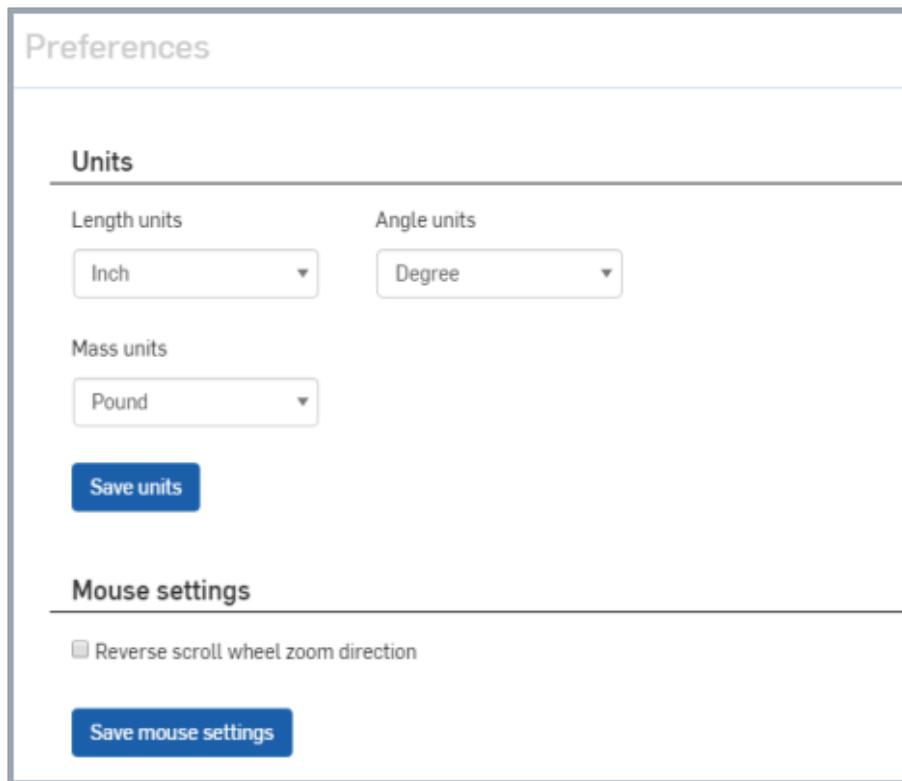
You can use any of the verified email addresses on your account to request a reset for a forgotten password.



The image shows the "Forgot password?" page of the Onshape interface. At the top is the Onshape logo. Below it is the heading "Forgot password?". A text input field labeled "Email" is provided for users to enter their email address. Below the input field is a blue button labeled "Request reset". At the bottom of the form is a link labeled "Sign in".

## Preferences

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



The image shows the "Preferences" page. The first section is titled "Units" and contains three dropdown menus: "Length units" set to "Inch", "Angle units" set to "Degree", and "Mass units" set to "Pound". Below these is a blue "Save units" button. The second section is titled "Mouse settings" and contains a checkbox labeled "Reverse scroll wheel zoom direction" which is unchecked. Below this is a blue "Save mouse settings" button.

In addition to setting default units for all documents you create (through this Settings tab), you can also change and specify default units for a specific workspace in a document through the "Document toolbar" on page 42 in a document.

Despite the 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 can still specify a different unit type (for example "10mm") in a numeric field.

## Mouse settings

Reverse the scroll wheel direction for zoom. By default, scroll down to zoom in and scroll up to zoom out. Check this box to reverse those directions and set:

- Scroll down to zoom out
- Scroll up to zoom in

## Security

Change your Onshape system password, and also enable (or disable) "Two-Factor Authentication" on page 560.

The screenshot shows the 'Security' section of the Onshape settings. It includes fields for 'Password' and '2 Factor Authentication', along with 'Change password' and 'Enable' buttons.

When resetting your password, a list of guidelines appears. Each requirement is checked as your password fulfills the requirement:

### ← Password

The screenshot shows the 'Password' reset page. It features input fields for 'Password' and 'Confirm password', an 'Update password' button, and a sidebar showing password strength requirements: 8 characters minimum, 1 number, 1 lowercase, and 1 uppercase.

## Usage

This image shows the Usage page for a Professional subscription; a Free subscription user sees slightly different figures as well as a button to Upgrade.

Private documents	12 documents
Private storage	20 MB
Total storage	20 MB

View the usage metrics for your account, including:

- Number of private documents currently created (including those in Trash)
- Amount of private storage space used; storage used for private documents only (including those in Trash)
- Amount of total storage space used; storage used for all documents you own (including those shared with you and those in Trash)

## Devices

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.

Devices			
Device type	First used on	Last used on	
Mac OS X (iPhone)	10:33 AM Sep 8	10:33 AM Sep 8	<a href="#">Forget</a>

## Applications

Onshape allows you to use partner third-party applications with your Onshape account. To access the Onshape App Store, navigate to <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 565).

Once signed in to the App store, you can browse the apps available and make purchases.

### Types of apps

Onshape third-party apps are of the following types:

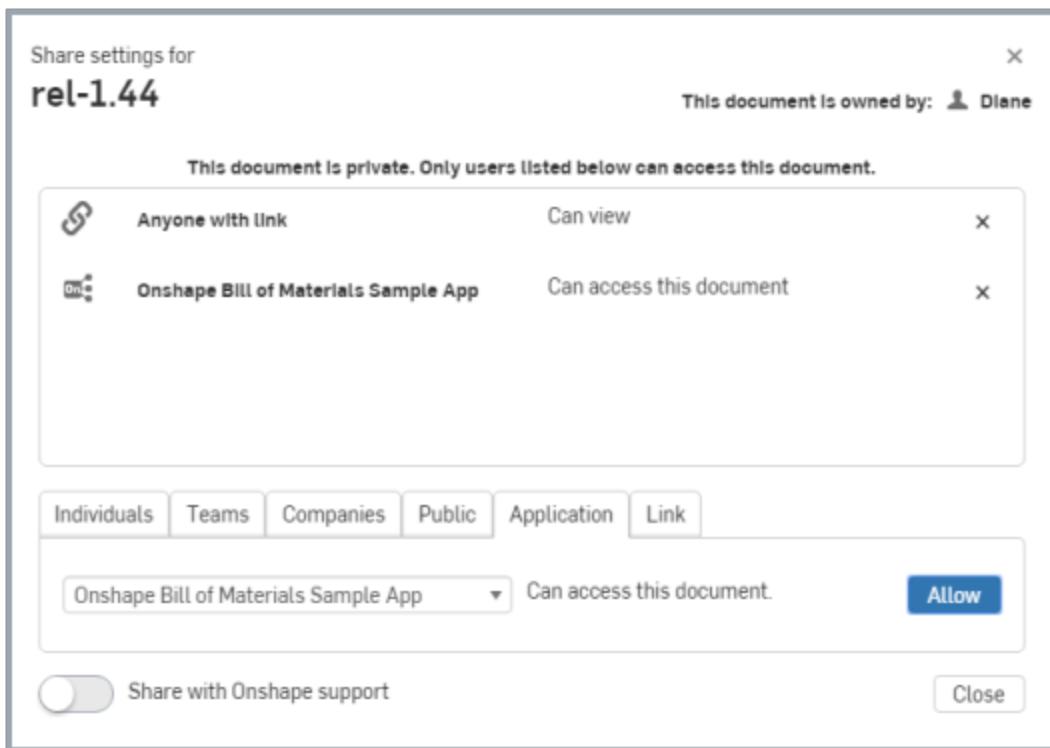
- **Integrated Application** - Accessible from within an Onshape document
- **Connected Desktop Application** - Downloaded from the third-party website and installed on your physical machine
- **Connected Cloud Application** - Accessible from a cloud-based service

### Actions on apps

- **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.

1. Click **Share**



2. On the Application tab, select the application from the drop down and click Allow.
3. To revoke access, click the x next to the application name at the top of the dialog.

Note that 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 (Accounts 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 from the menu at the bottom of your Onshape window.

## Early Visibility

Onshape periodically offers access to not-yet released features through our Early Visibility Program. This tab lists the programs currently in effect and available to approved users.

## Subscriptions

View the list of Onshape subscriptions and app subscriptions you have purchased.

## Free subscription

Free subscription members 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.

## Professional subscription

A user may belong to one or many Professional subscriptions per email address. Note that you have one set of

credentials per email address.

On your account > Subscriptions tab > company page, you can:

- Edit the membership of the company, including "Adding and removing company members" on page 556
- 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

## Education subscription

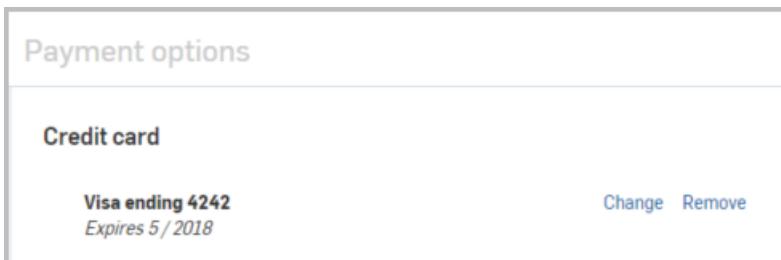
Education subscriptions are for current faculty members, volunteers, or degree- or certificate-seeking students at accredited education institutions. Students must be at least 13 years of age. This plan is to be used solely for classroom instruction, student learning projects, school clubs or organizations, and academic research. This plan is not to be used for government, commercial, or other organizational use.

Education subscriptions allow the same working environment of the Professional 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.

For answers to common questions about Onshape's payment processes and plans, see "Subscriptions and Payment FAQs" on page 539 and "Onshape Subscriptions" on page 533, respectively.

## Payment options

If you are the owner of the account, you can 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:



At no time is any credit card information displayed. You may simply enter new credit card information and that will supersede any previously entered information.

## Payment history

This area lists a chronological history of all payments made for the account. Click **View** to access a print-friendly invoice.

## Company

This area lists all the Companies of which you are either owner or member. Use this page to [manage](#) your companies.

Company

## EngineeringCo [Edit details](#)

Company name: EngineeringCo

Description: *Empty*

Company address:  
*Address*  
*City State*  
*Zip code Country*

Number of users you are paying for: 3 users

<< First < Previous Next >

 DiFree Amadeo	damadeo@onshape.com	Owner
 Nicholas Amadeo	namadeo@onshape.com	Member <a href="#">x</a>
 DiPro Amadeo	damadeo90@onshape.com	Member <a href="#">x</a>

## Teams

All Onshape plans allow you to create Teams of other Onshape users. This is an informal and convenient way to share collectively with a group of Onshape users. There are no document ownership requirements, as with companies, and users can be added or removed at any time by the designated Team Admins. Learn more about "Creating and Managing Teams" on page 549.

Teams

Team	Admin
Team-1	Member
Team-2	Member
Team-Design	Admin

[Create Team](#)

# Upgrading to Professional

Onshape's Professional subscription allows you to create unlimited private documents and take advantage of unlimited storage. You can select an Individual Professional subscription (pay for just yourself) or a Company Professional subscription (pay for 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 in "Creating an Account" on page 23.
- If you already have an Onshape account, click the **Upgrade** button on your account page and follow the instructions below.

To upgrade to a Professional subscription from the Free subscription:

1. On the page that appears, indicate:
  - a. Whether to purchase an Individual or Company subscription
    - If purchasing a Company subscription, enter the company name and the number of users
  - b. The desired payment interval
  - c. Valid credit card information

The screenshot shows the Onshape upgrade interface. At the top, it says "Onshape". Below that, "Subscription type" has "Individual" selected. Under "Professional Subscription details", "Paid monthly" is selected. In the "Credit card" section, there are fields for "Credit card number", "MM/YY", "CVC", "Zip code", and "Country". There are "Add card" and "Cancel" buttons. At the bottom are "Review my purchase" and "Cancel" buttons. To the right, a "Subscription summary" table shows:

Onshape Professional	
Compare subscriptions	
Details	
Subtotal	\$0
Discount	\$0
Taxes	\$0
Total	\$0

2. Click **Review my purchase**.

3. Review the summary.

The screenshot shows the 'Purchase summary' section of the Onshape upgrade process. It displays the following details:

Plan:	Individual Professional
Name:	
Number of users:	1
Paid:	Monthly
\$100 user / month	
Card ending in:	4242
Subtotal	\$100
Discount	\$0
Taxes	\$0
Total	\$100

At the bottom, there are two buttons: 'Confirm purchase' (blue) and 'Cancel' (grey).

4. Click **Confirm purchase**.

Notice that 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.

Onshape displays a final confirmation:

The screenshot shows the 'Thank you for choosing Onshape Professional' message. It includes the following payment confirmation details:

Your payment was successfully processed.  
\$100 has been charged to your card ending with 4242.  
A receipt has been emailed to you.

At the bottom, there is a 'Continue' button.

If you have purchased a Company Professional subscription, the final confirmation message includes a Continue button. When you click this button you are directed to the [Edit details](#) page for your company:

Account Settings Company

Profile  
Emails  
Preferences  
Security  
Usage  
Devices  
Applications  
Early visibility  
  
Subscriptions  
Payment options  
Payment history  
**Company**  
Teams

**EngineeringCo** Edit details

**Company name:** EngineeringCo

**Description:** Empty

**Company address:**  
Address  
City State  
Zip code Country

**Number of users you are paying for:** 3 users << First < Previous Next >

 DiFree Amadeo	damadeo@onshape.com	Owner	
	damadeo+90@onshape.com	Member	

**Add users:**

Email(s) Member Add

# Onshape Subscriptions

All Onshape subscriptions enable you to create documents (public and private) and to share documents with other Onshape users.

Common characteristics of all subscriptions:

- **Public documents** are available with view-only permission to all Onshape users; you can make any of your private documents public. Be aware that users may make private (editable) copies of public documents. Even after you make a document public, you can make it private again.
- **Private documents** can be viewed and edited only by you but can be shared with other users, with the ability to assign specific permissions per user. Just as you can add Share permissions to a document, you can also remove them. Users can remove themselves from the Share list on a document at any time as well.
- **Teams** can be created by grouping individual users under a team name. Documents can be shared with teams. Teams can be deleted at any time (or members can be removed); when a team is deleted, or members removed, any documents shared with those members is also revoked.

## Free subscription

The "Free Subscription" on page 535 is recommended for individuals who wish to try Onshape: students, educators, makers, and anyone wishing to use a professional CAD system for free. Some limitations apply:

- You can create up to 10 private documents using up to 100MB private storage space. If you exceed either limit you are prevented from creating more private documents (until you free-up space), but you can continue to work on your existing private documents.
- You can create as many public documents as you wish, using up to 5GB total storage space (including private and public documents).
- Documents shared with you count toward the 10-private document limit, but not towards storage limits.
- You can [upgrade](#) to a Professional subscription at any time without fear of losing any data at all.

## Professional subscription

The "Professional Subscription" on page 537 is recommended for users who want to create unlimited documents with unlimited storage space available. A Professional subscription can be purchased for \$100/month and paid for at the billing interval of your choice (monthly or yearly). When signing up for the Professional subscription, you have the choice to sign up and pay as an individual or as a company:

### ● For individuals

When choosing an Individual Professional subscription, you choose to pay for only yourself and you are the owner of all documents you create.

Should you discover that you don't need unlimited storage space, you can downgrade at any time to a Free subscription with no loss of data (conditions apply, see "Canceling a Professional Subscription" on page 545 for more information). The downgrade takes place at the end of your billing cycle.

### ● For companies

When choosing a Company Professional subscription, you choose to pay for multiple users with centralized billing and indicate a company owner/user. This purchase option includes the ability for the [company](#) to:

- Own documents
- Add and remove users from the subscription (through the company owner)
- Transfer ownership of documents (through the company owner)

### ● Enterprise

Please call Onshape Sales for more information.

## Education subscription

This subscription is the same as the Professional, and is offered free of charge to students and teachers. The subscription expires after one year, at which point you are automatically downgraded to the Onshape Free subscription. However, you can upgrade to the Education subscription again, provided you still meet criteria.

For more information and to sign up, visit <http://www.onshape.com>.

All documents created in an Education subscription are marked with  forever, even when made public, and even after the Education subscription has been downgraded. See "Canceling an Education Subscription" on page 547 for information on canceling an Education subscription.

# Free Subscription

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 (using up to 5GB of storage). You can create up to 10 private documents (totaling up to 100MB of private storage, which also counts towards the 5GB of total storage). You can share private documents with other users, and you can create [teams](#) for sharing purposes as well.

Documents shared with you count as private documents but do not count towards storage space. If documents are shared with you after you reach the 10 document limit, you can open them in View only mode (non-editable) until you are within the limits again.

Your Free subscription information is shown at the top of the Onshape user interface window:



## Private documents and public documents

In the Free subscription, when you create a document, you become the owner. As the owner of a document, you can:

- **Keep a document completely private** - No one else can see or edit the document
- **Keep a document private and share it** with specific people - Only people you designate can view the document and possibly edit it depending on the permissions you assign, and you can remove a user from the shared list at any time (users can also remove themselves from the share list)
- **Make a document public** - Public documents are available for viewing by all Onshape users. Onshape users can choose to make a private copy of those documents and edit that copy (you can also make your public document private again)

## Working within limits

The **Onshape Free subscription** allows you to create up to 10 private documents (using up to 100MB storage) and unlimited public documents up to a total of 5GB storage space. Note that private documents shared with you count toward your private document limit but are not included in your storage space limit. (Public documents directly shared with another user do not count towards limits.)

If you exceed the 10 document limit (and are still under the storage limit), you will be prevented from creating more private documents, but you may still access and edit the documents you have created.

If you exceed the 100MB storage limit for private documents, you will be prevented from creating more private documents, and also prevented from editing the documents you have already created. Your private documents become View only. Documents shared with you at this point will appear in the list on your Documents page, also in View only mode.

To free-up space so you can edit documents, you can:

- Move a private document to Trash, and then empty it from Trash
- Unshare yourself from a shared document (note that you will no longer have access to it unless it is re-shared with you)
- Make a private document public, thereby giving all Onshape users view-only and copy permissions to it

You can always **download** a document, whether it is View only or editable. Download a document through the Export command.

Once you have made space in this manner, you can create new private documents and edit existing documents.

The "Professional Subscription" on the next page has no limitation on private documents or storage.

# Professional Subscription

Onshape's Professional subscription has no restrictions on the number of documents you can create or how much storage you can use, and is available for \$100/month/user, billable at your desired interval. When signing up for a Professional subscription, you have options to pay as an individual, or pay for multiple users (referred to as a company).

"Company" here means a named, user-visible Onshape entity for centralized billing, ownership, and document sharing for a specified set of Onshape Professional subscription users. These users do not have to be in the same actual company, only paid for by the same user (referred to as the company owner). For example, you can pay for contractors, collaborators, any Onshape user even if they are on another Professional subscription. A Professional subscription user paid for by another company must also be paid for by your company in order to be specified as a member of your company subscription.

## Professional subscription for individuals

Choosing a Professional subscription for an individual (or upgrading to one from a Free subscription), means you are choosing to pay for only your own Onshape use. You can create private and public documents, share documents with other Onshape users, and create [teams](#). All Onshape functionality is available with no limitations on number of documents or amount of storage.

## Professional subscription for a group of individuals

Choosing a Professional subscription for a group of individuals (a [company](#)), means you are choosing to pay for multiple Onshape users and you become the *company owner of the account/company in Onshape*. All users can create private and public documents, share documents with other Onshape users, as well as create documents owned by the company. The company owner can add and remove users. All Onshape functionality is available with no limitations on number of documents or amount of storage.

There is no restriction on how many Professional subscriptions a user may be included in. For instance, a single user may pay for their own Professional individual subscription as well as be a paid member of other companies' Professional subscriptions.

## Documents

In Onshape, when you create a document, you become the owner (this is true in all Onshape subscriptions, for Professional subscriptions for companies, the company can be the designated owner of a document). As the owner of a document, you can:

- **Keep a document completely private** - No one else can see or edit the document
- **Keep a document private and share it** - With specific people, teams, and companies (if they are part of the company) - only people you share with can view the document and possibly edit it depending on the permissions you assign, and you can remove a user from the shared list at any time (users can also remove themselves from the shared list)
- **Make a document public** - Public documents are available for viewing (in read-only mode) by all Onshape users; Onshape users can choose to make a private copy of those documents in order to edit them on their own
- **Company-owned documents** - Documents created within a Professional subscription for companies that have the company specified as the document owner. The company owner is the implicit owner of company-owned documents.

Note that permissions can be applied during the [share](#) process.

## Tips for working with Professional subscriptions

- If you are upgrading from a Free subscription to a Professional subscription, all of your documents are available to you immediately; and your private documents remain private.
- If you create an Onshape account and want to pay for multiple users (i.e. sign up for a Company Professional subscription), your email must not exist in the system yet. When signing up for the Company Professional subscription, you become the company owner with rights to add and remove users from the subscription. The company owner can also transfer company ownership to another user.

If your email does exist in the system (you will see a warning), you must sign in to your Onshape account and use the Upgrade option in your current subscription. Sign in with your current account credentials and then Upgrade your existing subscription via the Manage account page.

- If a Free user upgrades to a Professional subscription and wants to pay for others, that user becomes the company owner. All private documents created in the Free subscription remain private in the Professional subscription.
- People added to a Company Professional subscription become Professional subscription users; any private documents previously owned in a Free subscription remain private - if the user is already a part of another professional subscription (individual or company), there will be a notification during the process, and you can still add them and pay for them.
- Users added to a Company Professional subscription become members of that company and cannot simply downgrade themselves to an Individual Professional subscription or leave the company. The user must request to be removed from the company (by the company owner) first. Once a user is removed from the company, they are downgraded to a Free subscription immediately, not at the end of the payment cycle as is the case with downgrading from Individual Professional to Free. The exception is when a user belongs to more than one Professional subscription; in that case they are removed from the company, but are not downgraded to the Free subscription.
- If a user is removed from a Company Professional subscription and has an Individual Professional subscription (or is a member of another Company Professional subscription), they are not downgraded to a Free subscription.

# Subscriptions and Payment FAQs

## How much do the Onshape subscriptions cost?

Onshape's Professional subscription is \$100/month. Onshape's Free subscription is \$0/month. Call for pricing on the Enterprise subscription.

## What is the difference between the Professional subscription and the Free subscription?

The Professional and Free subscriptions have far more similarities than differences. Most importantly, the Free subscription includes all of the same CAD and data management functions as the Professional subscription.

The key difference is that the Free subscription has limits on the number of private documents you can create, and how much storage you can use. The Professional subscription has no limitations.

With the Free subscription, you can create up to 10 private documents, using less than or equal to 100MB of storage. You can create public documents using up to 5GB total storage (including the 100MB for private documents). You can access any public documents (only those created by you count against your storage limit). You can delete documents in order to create new ones, but these must be emptied from Trash before the limits are refreshed.

## What is the difference between a monthly subscription and an annual subscription?

You are only billed once a year for your annual subscription, and therefore can avoid the administrative hassles of monthly expense approvals or reimbursements. You are also guaranteed the current monthly rate when you purchase an annual subscription.

## Does Onshape store my credit card information?

No, Onshape never stores your credit card information.

## How do I change my credit card information?

You can change your credit card and other payment information through the **Manage 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 can happen for a variety of reasons, and in many cases only your card-issuing bank can 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 can 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.

## Can I cancel my Professional subscription?

You can change your Onshape subscription to a Free subscription at any time, for as long as you like. Your private Onshape data stays private, and stays available in your account.

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 Company Professional 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 Company 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 can change your Onshape subscription to a Free subscription at any time, for as long as you like. Your private Onshape data stays private, and stays available in your account.

Your private documents remain private and your public documents remain public. To adhere to the Free limitations, you choose which documents to remove and which to keep for access. You can also export and download your private documents if you wish.

## Can I centralize payment for several users?

Yes, sign up for a Company 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 **Manage accounts** 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 (with limitations on document creation and storage space). Onshape never deletes your documents and never makes your private documents public.

## If a colleague shares a document with me (a Free subscription user), does it count against my limits?

Documents shared with a Free user count against the private document limit (of 10 private documents), but not to any storage limits. Free users can unshare themselves from documents at any time; but be warned that by doing so, you remove the document from your Documents page list and relinquish access to that document.

## How do I get more storage for my Free subscription?

You can delete documents you are no longer interested in to make room within the free 5 GB of storage, or you can upgrade to the Onshape Professional subscription for unlimited storage. Note that when deleting documents, you must also empty them from Trash before the storage space is refreshed. To reduce the size of a particular document, you can copy the workspace (which removes all history points that use storage space). Be aware that by doing this, you prohibit yourself from restoring that copied workspace back to a specific point in history.

# Setting up Payment

Create your Onshape credentials and enter your credit card information to finish signing up for the Professional subscription:

The screenshot shows the Onshape payment setup interface. At the top, the Onshape logo is displayed. Below it, a section titled "Plan type" contains three radio buttons: "Individual" (selected), "Company", and a question mark icon. Under "Professional Plan details", there are two radio buttons: "Paid monthly" and "Paid yearly". The "Credit card" section includes fields for "Credit card number", "MM/YY", "CVC", "Zip code", and "Country". At the bottom are "Review my purchase" and "Cancel" buttons. To the right, a sidebar titled "Subscription summary" shows the "Onshape Professional" plan with a total of \$0.

Details	
Subtotal	\$0
Discount	\$0
Taxes	\$0
<b>Total</b>	<b>\$0</b>

1. Indicate what type of Professional subscription you want:

- **Individual** - Pay for just one user, yourself
- **Company** - Pay for multiple users, a company

When paying for multiple users, enter the number of users; at the conclusion of this payment process, you can access the [Manage accounts](#) page and specify the details of your company users.

**Subscription type**

Individual  Company [?](#)

**Company Professional Subscription details**

onshape

Number of users paid for by your company:  
1 (including the subscription owner)

Paid monthly  
 Paid yearly

**Credit card**

Add a credit card to make purchases in Onshape

Credit card number  
MM/YY CVC [?](#)  
Zip code Country

[Add card](#) [Cancel](#)

**Subscription summary**

**Onshape Company Professional**

[Compare subscriptions](#)

<b>Details</b>	
Users:	1
Subtotal	\$0
Discount	\$0
Taxes	\$0
<b>Total</b>	<b>\$0</b>

2. Enter account details:
  - Company name
  - Number of users being paid for, including the subscription owner.
  - The payment interval (monthly or yearly)
  - Indicate the interval of payment (monthly or yearly); note that charges are made at the beginning of the payment cycle.
  - Credit card information
3. Click **Review my purchase**.

Review the order details.

4. **Confirm purchase.**

If you have purchased an individual subscription, you now see the "Documents Page" on page 84.

If you have purchased a company subscription, you are directed to add your subscription users:

The screenshot shows the 'Account Settings' section for 'Company'. The left sidebar has a 'Company' tab selected. The main area displays the company details for 'EngineeringCo' with fields for name, description, address, city, state, zip code, and country. It also shows a list of users: 'DiFree Amadeo' (damadeo@onshape.com) is listed as 'Owner'. Another user, 'damadeo+90@onshape.com', is listed as 'Member'. Below this is a 'Add users:' input field with an 'Email(s)' placeholder and a dropdown for 'Role' (set to 'Member') with an 'Add' button.

## Adding users

1. In the Add users text box, enter one or more email addresses (separated by commas).
2. Select the role for the specified users: Member or Admin. Admins can add and remove users from the company.
3. Click Add.
4. Review the list of users and roles. You can change the role of a user here, or remove them from the subscription completely.
5. Click Done.
6. You are directed to the Documents page. Notice the Company name is now listed as a filter on the left:

The screenshot shows the 'Documents' page. At the top, there are 'Create' and upload buttons. Below them is a sidebar with filters: 'Recently opened', 'My documents', 'Created by me', 'Shared with me', 'Test Company' (which is highlighted in blue), 'Public', 'Tutorials & Samples' (which is bolded), and 'Trash'.

# Cancelling a Professional Subscription

To cancel the Professional subscription and move to the Free subscription:

1. Expand the user menu under your user name and select **Manage account**.
2. Select the **Subscriptions** tab.
3. If you have more than one subscription, click your email address to manage your subscriptions:

The screenshot shows the 'Subscriptions' page for the user 'damadeo90@onshape.com'. The subscription type is listed as 'Individual Professional subscription'. Under 'Payment details', it shows a Visa card ending in 4242, expiring 02/19. The renewal is set to 'Yearly' and the next bill is \$1,200.00 on August 10, 2016. The status is 'Active'. There are buttons for 'Change to Company Professional subscription', 'Change credit card', 'Cancel subscription', and a 'View' link. Transaction history at the bottom shows an Aug 10, 2015 charge of \$1,200.00.

4. Click **Cancel subscription**.
5. On the confirmation dialog that appears, click **OK** to cancel.

The confirmation dialog box asks 'Are you sure you want to cancel your Professional subscription?'. It also states 'You will be downgraded to a Free subscription when it expires on: August 10, 2016'. At the bottom are 'OK' and 'Cancel' buttons.

6. In the blue notification that appears, click **Refresh to continue**.

Your Professional subscription has been cancelled. [Refresh to continue.](#) ×

Note that on the date specified that your subscription expires, if your documents exceed the Free subscription limits, all of your documents will be view-only (grayed out). Onshape allows you to choose which documents to keep and which to remove to be in accordance with the limitations of 10 private documents (up to 100MB of storage) and public documents up to 5GB in total storage.

## Complying with Free subscription limits

To comply with the Free subscription limitations, you can:

- Use the gear menu beside a document to make it public and remove it from counting towards limits.
- Delete documents in order to comply with the storage limits (these documents must be removed from Trash before the storage limit registers the decrease).
- Unshare yourself from documents shared with you to remove them from counting towards limits.
- Export/download the document and then move it to trash.

- Note that removing features, Part Studios, Assemblies and/or imported files from an Onshape document will not decrease the document size. (Onshape keeps a history of the document so that you can restore the document to any point in that history, including any deleted entities.)

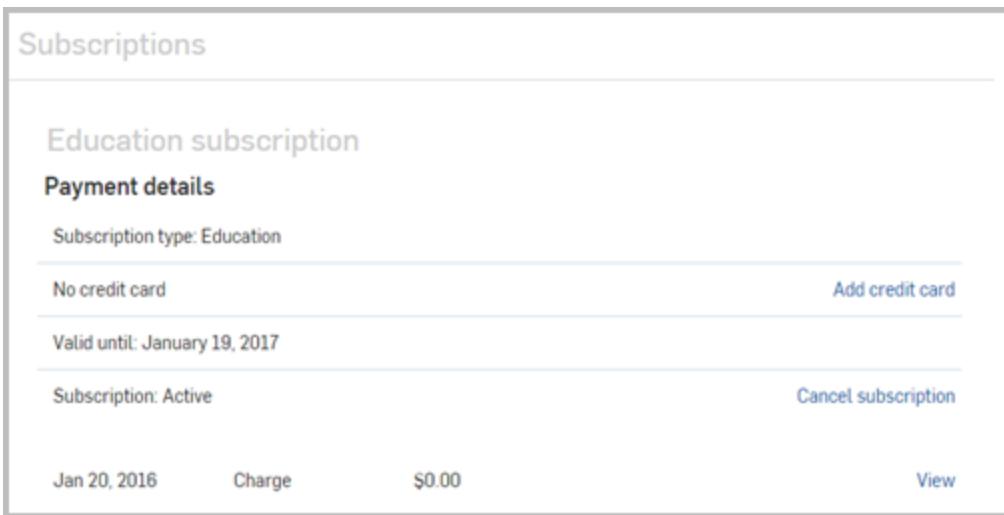
# Cancelling an Education Subscription

Education subscriptions are supplied free-of-charge to students and teachers. All documents created as an Education subscription user are marked with an Edu 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, you can at that point upgrade to the Education subscription again, provided you still meet the criteria.

To cancel the Education subscription and move to the Free subscription:

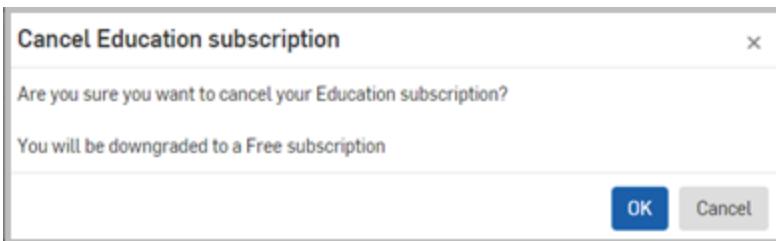
1. Expand the user menu under your user name and select **Manage account**.
2. Select the **Subscriptions** tab.
3. Click Cancel subscription for the Education subscription.



The screenshot shows the 'Subscriptions' section of the Onshape account management interface. It displays the 'Education subscription' information. Key details include:

- Payment details:** Subscription type: Education.
- No credit card:** An option to add a credit card.
- Valid until:** January 19, 2017.
- Subscription status:** Active.
- Cancel subscription:** A blue button to initiate cancellation.
- Charge history:** Shows a single entry for Jan 20, 2016, with \$0.00 charged.
- View:** A link to view more details.

4. Click **Cancel subscription**.
5. On the confirmation dialog that appears, click **OK** to cancel.



The confirmation dialog box has the following content:

Cancel Education subscription ×

Are you sure you want to cancel your Education subscription?

You will be downgraded to a Free subscription

OK Cancel

6. In the blue notification that appears, click **Refresh to continue**.

Your subscription is immediately downgraded to the Onshape Free subscription. If your documents exceed the Free limits, all of your documents will be view-only (grayed out). Onshape allows you to choose which documents to keep and which to remove to be in accordance with the Free limitations of 10 private documents (or up to 100MB of storage) and public documents up to 5GB in total storage.

Note that all documents created through an Education subscription will always have an Edu 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.

## Complying with Free limits

To comply with the Free limitations, you can:

- Use the gear menu beside a document to make it public and remove it from counting towards limits.
- Delete documents in order to comply with the storage limits (these documents must be removed from Trash before the storage limit registers the decrease).
- Unshare yourself from documents shared with you to remove them from counting towards limits.
- Export/download the document and then move it to trash (and empty trash).
- Note that removing features, Part Studios, Assemblies and/or imported files from an Onshape document will not decrease the document size. (Onshape keeps a history of the document so that you can restore the document to any point in that history, including any deleted entities.)

# Creating and Managing Teams

You can create teams in order 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.

It is not required that the members of a team have anything in common; not even an Onshape plan.

One user 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. Members receive notification emails when they are added and removed from a team, and users can 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 can remove any member from the team, thereby removing any Share permissions previously made through the team (but not Shares made on an individual basis).

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 can be assigned during the Share operation:

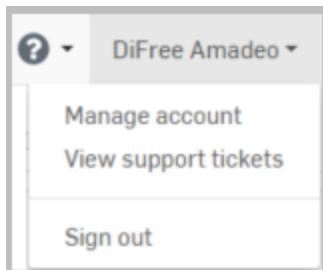
- **View** - open for read only access
- **Edit** - open and make changes
- **Edit and share** - open for making changes and also share with other users
- **View and comment** - open for viewing and inserting comments; no editing allowed

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 **Manage account**:



2. On the page that appears, select Teams from the left panel and click **Create Team**:

Teams	
Profile	Team
Emails	Admin
Preferences	Team-1
Security	Member
Usage	Team-2
Devices	Member
Applications	Team-Design
Early visibility	Admin
Subscriptions	
Payment options	
Payment history	
Company	
<b>Teams</b>	

**Create Team**

3. Enter a name for the team, and a description, or statement of purpose:

← Untitled Team

Team name:

Description (optional):

**Create team** **Cancel**

4. Click **Create team**:

← Team-Design Team

Team name:  
Team-Design

Description:  
The team doing the actual design work.

Team members (1)

Di-ProCo Amadeo	damadeo101@onshape.com	Admin	X
-----------------	------------------------	-------	---

Add team members:  
 Member **Add**

**Delete team**

5. Add members by entering individual email addresses (or copy/paste a comma-separated list of addresses), select a role (Member or Admin).

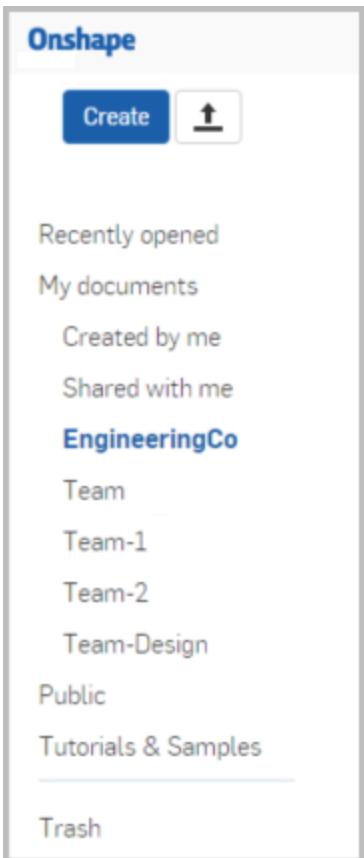
Note that you can return to this page and change a team member's role.

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:

Team	Role
Team	Admin
Team-1	Member
Team-2	Member
Team-Design	Admin

Create Team

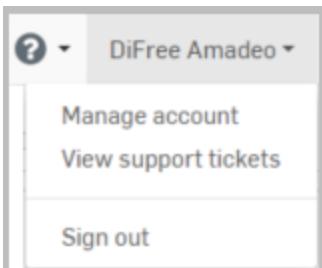
Creating a team also adds a filter for that team in each member's Documents filters on their Documents page. These filters list all documents shared with a particular team. Note the Teams in the list below.



## Removing members and admins

Members can remove themselves from a team, and any member with an Admin role can 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 the user name in the top-right corner of the page and select **Manage account**:



2. Select **Teams** in the left panel to access the list of teams of which you are a member:

The screenshot shows the 'Account Settings' interface. On the left, a sidebar lists various account management options: Profile, Emails, Preferences, Security, Usage, Devices, Applications, Early visibility, Subscriptions, Payment options, Payment history, Company, and Teams. The 'Teams' option is highlighted with a blue bar at the bottom of the sidebar. To the right, under the heading 'Teams', there is a list of teams: 'Team', 'Team-1', 'Team-2', and 'Team-Design'. A prominent blue button labeled 'Create Team' is located below the team list.

3. Select the team in the list from which you wish to remove yourself or another member:

- To remove yourself (as a member): Click **Leave team**.

The screenshot shows the 'Team-1' page. At the top, it says '← Team-1 Team'. Below that, it displays 'Team members (4)'. There is a table listing four members: 'Di-ProCo Amadeo' (email: damadeo101@onshape.com, role: Admin), 'Nicholas Amadeo' (email: namadeo@onshape.com, role: Member), and 'DiFree Amadeo' (email: damadeo@onshape.com, role: Member). At the bottom of the page is a button labeled 'Leave team'.

- 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).

The screenshot shows the 'Team-2 Team' management page. At the top left is a back arrow and the text 'Team-2 Team'. Below this, there are sections for 'Team name:' (Team-2) and 'Description:' (empty). A 'Team members (4)' section lists four users with their emails and roles: Di-ProCo Amadeo (Admin), DiCo Amadeo (Admin), and Nicholas Amadeo (Member). Each user has a delete button ('X') to the right. Below this is an 'Add team members:' section with fields for 'Email(s)', 'Role' (set to 'Member'), and a blue 'Add' button. At the bottom left is a 'Delete team' button.

- 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.

# Managing Companies

The Onshape Company Professional subscription enables you to pay for multiple users, and thereby create a Company within Onshape: a named, user-visible Onshape entity for centralized billing, ownership and sharing for a set of Professional subscription users all on the same billing subscription.

A company is created at the time of the Company Professional signup:

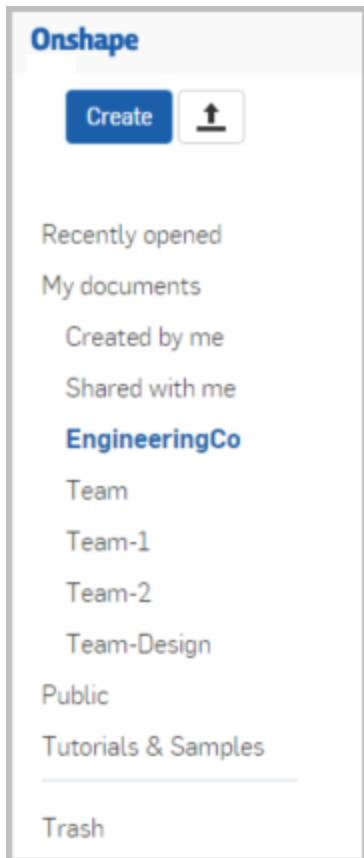
The screenshot shows the Onshape interface for creating a Company Professional subscription. On the left, there's a sidebar with 'Subscription type' options: 'Individual' (radio button) and 'Company' (radio button, selected). Below that is a section for 'Company Professional Subscription details' with a text input field containing 'onshape'. Underneath is a note about the number of users (1 including the owner) and payment frequency (Paid monthly). To the right is a 'Subscription summary' panel for 'Onshape Company Professional' showing a total of \$0 for Subtotal, Discount, and Taxes, with 1 User. At the bottom are 'Add card' and 'Cancel' buttons.

The user who signs up and agrees to pay for the Company Professional subscription becomes the owner of the company. All users listed on the subscription receive notification emails when they are added to the company (and if they are removed from the company).

If an existing Free user is listed as belonging to a Company Professional subscription, that user's plan is automatically upgraded to Professional and the company is charged. Any Onshape user can be paid for and included in a Company Professional subscription, and multiple Professional subscriptions.

## Documents and company ownership

When creating private documents, users who are company members have the choice to select an owner for the document (themselves or a company). When a company is created, a document filter is automatically created and included on the Documents page for all members. Documents owned by the company are listed by the filter:



The creator of a document, the Company owner, and admins are the only users with Full Access to the document, meaning that only they can delete the document. All members of a company can share all company-owned documents that they have access to.

At any point, the Company owner and admins of the Company can remove the user who created the document completely from having any access to the document, and add them back as a collaborator with certain permissions. Permissions can be:

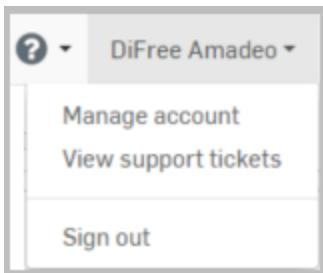
- **View** - open for read only access
- **Edit** - open and make changes
- **Edit and share** - open for making changes and also share with other users
- **View and comment** - open for viewing and inserting comments; no editing allowed

Following are instructions for:

- Adding and removing company members
- Creating company-owned documents
- Removing permissions
- "Share Documents " on page 482

## Adding and removing company members

1. Expand the menu under your user name in the top right corner of the page and select **Manage account**:



2. Select Company in the left pane:

The image shows the "Account Settings" sidebar from the Onshape interface. It contains the following tabs:

- Profile
- Emails
- Preferences
- Security
- Usage
- Devices
- Applications
- Early visibility
- Subscriptions
- Payment options
- Payment history
- Company** (highlighted in blue)
- Teams

- Click the company name link to edit company and membership details:

User	Email	Role	Action
Sarah Plancio	damadeo+111@onshape.com	Admin	
Nicholas Amadeo	namadeo@onshape.com	Member	X
Smith Smythe	damadeo+112@onshape.com	Member	X

- When you have finished, click the left arrow to return to Manage account page.

## Creating company-owned documents

Adding users to a company can only be done by a company Admin or Owner. Users added to a company receive a notification email. All company members can create company-owned documents, but only the company owner and admins can delete company-owned documents.

- Click **Create**.
- Specify a document name.
- In the Owned by drop down, select a company name:

- Click **OK**.
- In the company filter on the Documents page (for every company member), the newly created document is listed.

## Removing companies - Canceling subscriptions

Only the owner of the company can cancel the subscription (thereby transitioning all members without another Professional subscription to Free).

Canceling a company subscription also transitions the owner of the company to a Free subscription (unless this user is also a member of another Professional subscription) and all company-owned documents are kept in the company owner's account. These documents are also removed from the Documents list of all other company members and they no longer have access to them. Each member retains their private documents, however. For more information, see "Canceling a Professional Subscription" on page 545.

# 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 just 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 can be different each time you sign in.

We highly recommend you use 2FA for Onshape and for all websites you use that support it.

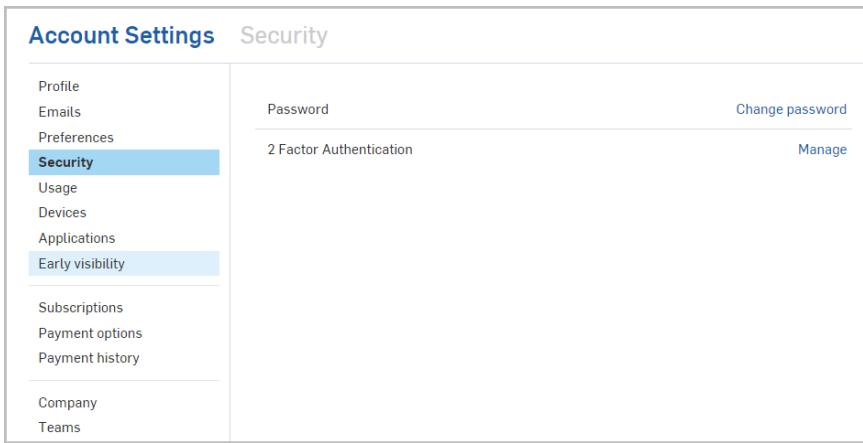
## How it works

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 can recognize. Once you enable 2FA in Onshape, Onshape will prompt you for the 2FA code after you sign in with your password.

You can allow the 2FA mechanism to remember the devices on which you sign in so that once you use 2FA authentication to sign into Onshape from a specific device, you won't need a 2FA code to sign in on that device again.

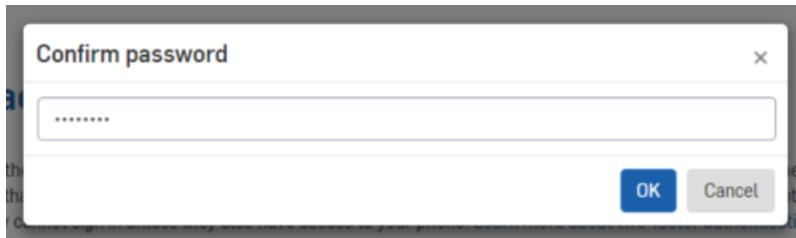
## Enabling and using two-factor authentication

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 **Manage Account**.
4. On the *Security* tab, click **Security**:



5. Click **Enable**.
6. Click **Set up two-factor authentication**.

7. Confirm password:

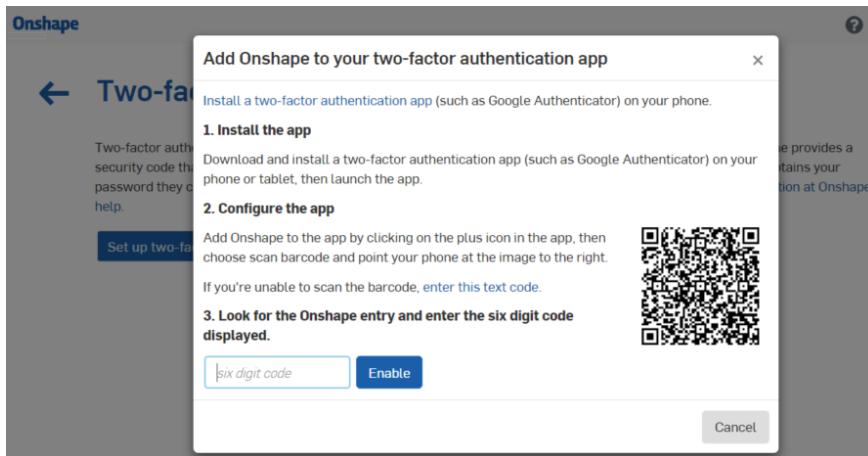


8. Click **OK**.

## Configure the app to work with Onshape

Continuing from the instructions above:

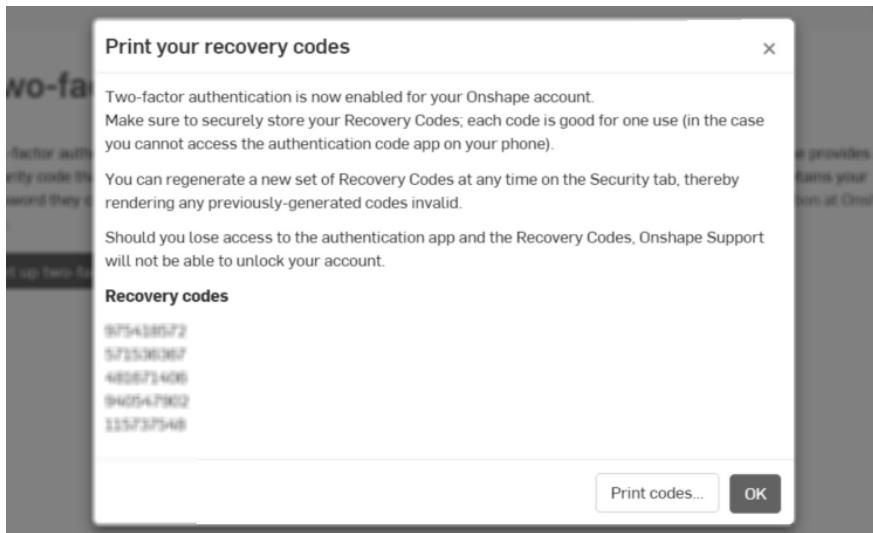
1. Use the Authenticator app on your device to scan the QR code presented in the Onshape user interface:



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 can't 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 can 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.

## Sign 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:

Two-factor authentication

Open the two-factor authentication app on your phone to get your authentication code.

Two-factor authentication code   ?

Remember this computer for the next 30 days

Verify

Don't have your phone? [Enter a two-factor recovery code](#)

2. Open the two-factor authentication app on your device and enter the code generated.

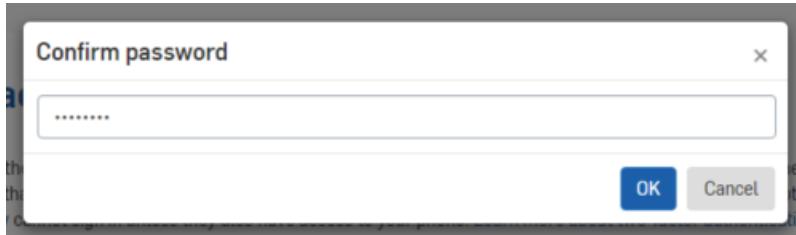
3. Click **Verify**.

In the event that you don't have access to the app, you can click the **Enter a two-factor recovery code** link to enter one of your current recovery codes.

## Disable two-factor authentication in Onshape

You can 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

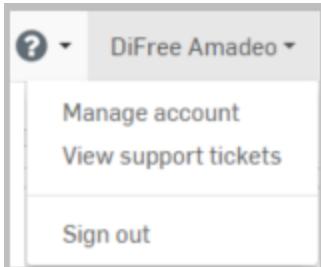
Should you need to replace a device on which you have 2FA enabled for Onshape:

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.

# Reset Password

1. Expand the menu under your user name and select **Manage account**:



2. Select the *Security* tab:

A screenshot of the "Account Settings" page. The left sidebar has several tabs: Profile, Emails, Preferences, **Security** (which is highlighted in blue), Usage, Devices, Applications, Early visibility, Subscriptions, Payment options, Payment history, Company, and Teams. The main content area has two tabs: "Password" and "2 Factor Authentication". The "Password" tab is active. It includes a "Change password" link and a "Manage" link.

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 checked when your password fulfills the requirement:

A screenshot of the "Password" update form. It has three input fields: one for the old password (filled with dots), one for the new password (with a cursor), and one for confirming the new password. Below the fields is a blue "Update password" button. To the right, there is a summary of the password's strength:

- Your password's strength
- 8 characters minimum
- 1 number
- 1 lowercase
- 1 uppercase

4. Click **Update password**.

# App Store FAQs

Some commonly asked questions about the Onshape App Store include:

## 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 feature request feature of the Onshape Forum to request a particular app or type of app.

## 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>.

## How do I view all the apps I've purchased?

Sign in to the Onshape App Store (<https://appstore.onshape.com/>) and select the **My Apps** filter (on the left) to see which apps you've purchased.

You can also sign in to your Onshape account (<https://cad.onshape.com/>), 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 **Manage account > Applications** and view your applications there.

## Can I purchase apps for my Onshape company?

Not at this time. However, apps can 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 can see the tabs related to that app.

## Can I 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 can also immediately revoke Document access from an app.

## How can I instantly revoke Document access from any app?

In your Onshape account, navigate to **Manage account > Applications** and click **Revoke** for the app.

## Can I 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):

- a. Sign in to your Onshape account (<https://cad.onshape.com/>).
- b. Open the document with which you want to use the app.

- c. Click the plus menu  at the bottom of the window and select **Add Application**.
- d. If you have more than one app available, select the one to use with the currently opened document.
- e. 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).

## Can I 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 can't see the app data and suggesting they can purchase it, if desired.

## Can I 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 Feedback 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.

# Beginning Tutorial

This tutorial is a text-based, step-by-step set of instructions that leads you through creating a part in Onshape. If you prefer video tutorials, you can access them on the Documents page after logging in to Onshape: in the Tutorials & Samples filter.

## Available on all devices

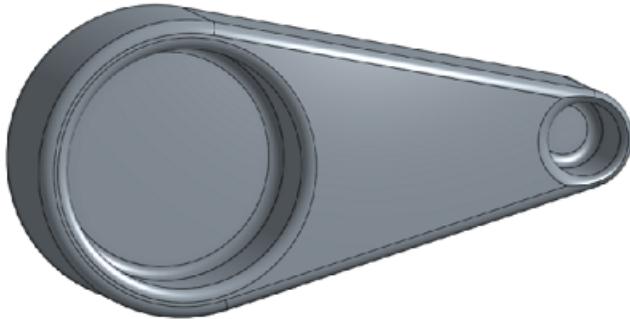
You can follow this tutorial on any device - but be aware that there are subtle user interface differences among devices. For instance, toolbars are similar but not exactly the same. For information regarding the user interface for your device, see *The Onshape User Interface* topic in the Onshape help system on your device.

**Purpose**      Become familiar with Onshape's user interface and comfortable with sketching

**Audience**     Those comfortable with 3D modeling and new to Onshape

**Duration**    15 minutes

**Goal**



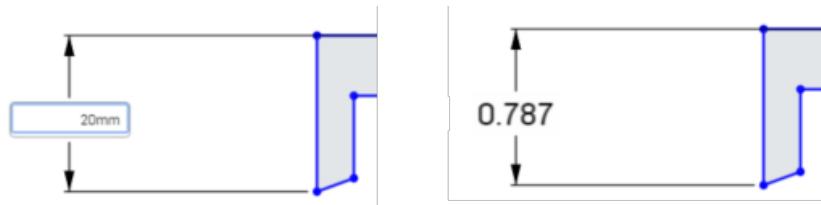
During this tutorial you will learn how to:

1. Create a document and set default units.
2. Sketch using automatic inferencing and constraints.
3. Extrude (boss) to add material.
4. Extrude (cut) to remove material.
5. Apply fillets and rounds, and shell the part.
6. Reorder parametric history.

Next: "Create a Document and Set Default Units" on the next page

# Create a Document and Set Default Units

You can set default units of measure for all documents you create at once, and also set specific units for a particular document. Onshape also allows you to enter specific units into a field on the fly and those units are converted to default units for display. When you click in the field again, the original units entered are shown.



## Learn more about Onshape documents

Onshape documents contain all of your modeling information, represented in tabs: sketch and build parts in a Part Studio, assemble parts in an Assembly, create a drawing of a part in a drawing, import any number of externals files: images, PDFs, etc. Each of these types of information are represented within tabs in a document.

An Onshape document can contain multiple tabs. This tutorial concentrates on a Part Studio tab, which can contain multiple parts and the Feature list (the parametric history) that defines those parts.

All documents are listed on the Documents page, along with the creator's user name and the dates created and last modified. There is also a list of actions you can perform on these documents. Each user sees only the documents each has created, even when a member of a company using a shared subscription to Onshape. Users can also see all documents that have been shared with them or made public by other users. (More on sharing and viewing shared documents later.)

## Create a document

1. On the Documents page, click .

Provide a document name if you want; by default the document title is **Untitled Document**.

2. Click **OK**.

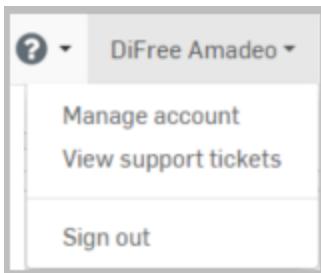
The document opens with a Part Studio active.

Onshape defaults to inch/degree/pound for units of measure.

## Set default units for all documents you create

1. Expand the menu under your name (User menu) and select **Manage account**:

### User menu



2. Select *Preferences* and make the appropriate selections.
3. Click **Save changes**.

## Preferences

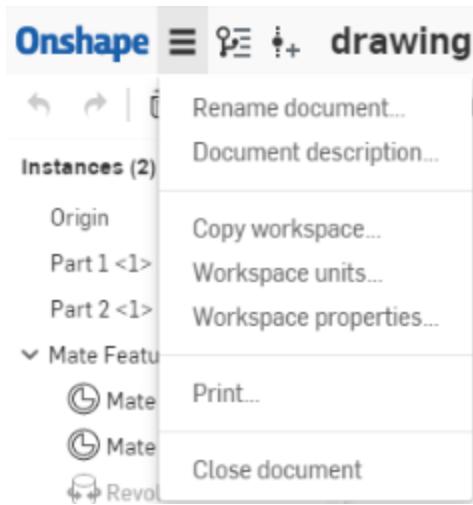
The screenshot shows the 'Account settings' page with the 'Preferences' tab selected. On the left sidebar, other tabs like 'Profile', 'Emails', 'Security', etc., are visible. The main content area is titled 'Units' and contains three dropdown menus: 'Length units' set to 'Inch', 'Angle units' set to 'Degree', and 'Mass units' set to 'Pound'. A blue 'Save units' button is at the bottom.

Click the Onshape logo at the top of the page to return to the *Documents* page.

## Set the default units for a specific document

1. Click a document name in the Documents list to open it.
2. With a document open, click the Documents menu .

## Documents menu



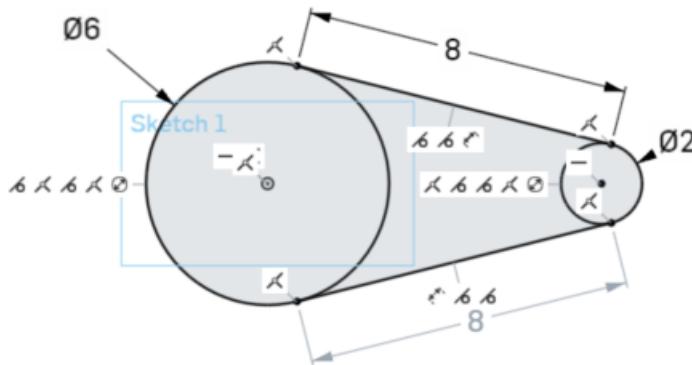
3. Click **Units** and make appropriate selections (as above).
4. Click to save changes.

## See also

- "Onshape Documents" on page 75
  - "Part Studios" on page 89
  - "Assemblies" on page 304
  - "Managing Companies" on page 555
  - "Share Documents" on page 482
- Next: "Sketch with Automatic Inferencing" on the next page

# Sketch with Automatic Inferencing

To help you become familiar with the sketching interface, this exercise walks you through creating this sketch:



2D sketches, made up of sketch entities, are used to create solid models and parts in Onshape. Sketch entities include lines, circles, rectangles, and similar collections of geometry. To help you in relating sketch entities to one another, Onshape provides automatic inferencing: as you create sketch entities, Onshape notices and automatically applies relations, when appropriate, like: horizontal, vertical, parallel relations between sketch entities. This section explains how this works.

## Start a sketch

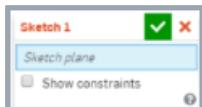
1. Click .

### Sketch tool in toolbar



A sketch dialog opens:

### Sketch dialog



In general: white fields require keyboard input; blue fields require a selection (mouse click in the graphics area); fields with a down arrow require selection from drop down list. Orange indicates an invalid value, error, or unsolvable situation.

2. Select the **Right plane**.

You can select a the plane in the graphics area, or select it in the Feature list. The selected plane is highlighted in orange.

Notice that the Feature toolbar is replaced with the Sketch toolbar.

3. Open the context menu with a right-click in the graphics area, then select **View normal to sketch plane** or use the shortcut key '**n**'.

## View normal to sketch plane

Use **View normal to sketch plane** (shortcut key: **n**) to rotate the sketch plane parallel to the screen. The circled dot in the middle of the sketch plane represents the sketch origin . You can set relationships between the origin (and planes) and sketch entities (called *constraints*). If the sketch planes clutter your view, you can hide them using shortcutkey **p**.

4. Select **Circle tool** .

## Circle tool in toolbar



5. Sketch a circle using the origin in the graphics area as the origin for the circle.

- Select the tool.
- Click to begin, then click again to stop sketching OR click and drag then release to stop.

The center of the circle should be black, indicating it is fully constrained/locked in place on the origin. The outside of the circle should be blue, indicating that it is not yet constrained and can be resized by clicking and dragging (only if there are no tools selected).

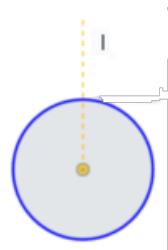
## Onshape color indicators



- Hover the cursor (now a plus sign) over the point of origin in the sketch plane.
- Move the cursor away from the origin vertically. An orange dotted line indicates an inference (in this case vertical).

## Inference example

In this example, you can see the dotted yellow line of displayed automatic inferencing, as well as the Vertical constraint icon:

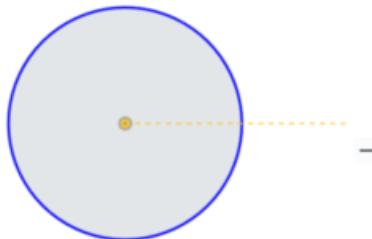


Sketching another entity with this line visible, causes the entity to have a vertical constraint to the origin (and to this circle). The same thing happens if you pull the cursor away from the origin in a horizontal direction.

When you stop drawing, the sketch tool remains selected so you can begin drawing another similar curve immediately.

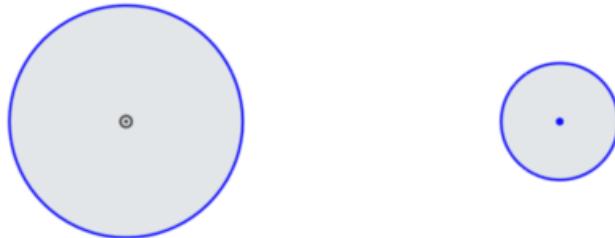
6. Using a horizontal inference, create a smaller circle:
  - a. When your cursor is anywhere along the horizontal axis, a dotted line appears, along with the horizontal constraint icon .

## Horizontal inference



- b. With the horizontal inference line visible, draw the second circle to the right of the first circle.

## Second circle



- c. Click the Circle tool  to toggle it off.

You can also use the **Escape key** or the **context menu > Escape <tool>**.

7. Hover over the center point of the second (right) circle. A small icon appears, hover over the icon and it darkens.

## Constraint example



The center point of both circles should appear highlighted orange indicating that they are related by a horizontal constraint.

The point and curve of the second circle are blue, despite the horizontal constraint. Blue indicates an under-constrained entity, red indicates an over-constrained entity, and black indicates a fully constrained entity.

- Click to save your work and close the dialog. The sketch is now listed in the [Feature list](#).

The Feature list is a history-based list of every sketch and feature, part, and surface created in this Onshape document. As you continue to sketch and create parts, this list can become extensive so it's a good idea to rename features as you go along.

## Rename a feature

- Right-click on **Sketch 1** in the list and select **Rename**.

The Feature list can become extensive so renaming as you go ensures that you can find and edit features more easily later on.

- Specify a name of **Main Sketch**; press **Enter**.

An important aspect of sketching is making sure each sketch captures the design intent necessary to prevent unanticipated changes to the resulting part when modifications are made to the sketch; constraints and dimensions can help.

## Check sketch constraints

- Double-click on **Main Sketch** in the Feature list to open the sketch dialog.  
The sketch entities in the graphics area become active again.
- Specify a 2D view by right-clicking and selecting **View normal to sketch plane** from the context menu (or use the shortcutkey **n**).
- Click and drag the center point of the second circle, and then do the same with the curve of the circle to see how they move.

Because the center point of the circle is horizontally constrained, it can move only along the horizontal plane and because the curve of the circle has no dimension or constraint applied, you can resize it.

Note: If you find you have unintentional constraints applied, you can click on the constraint icon and use the Delete key to remove the constraint.

## Connect the circles

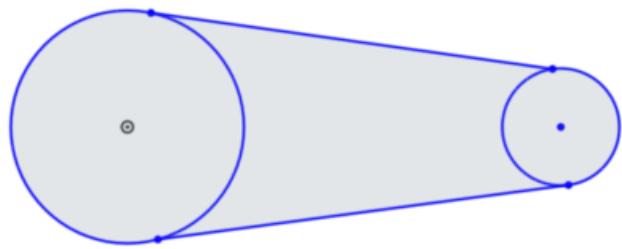
- Select the **Line tool** .

### Line tool in toolbar



- Draw a line from the top edge of the large (left) circle to the top edge of the small (right) circle. Do the same thing from the bottom edges of the circles.

### Example

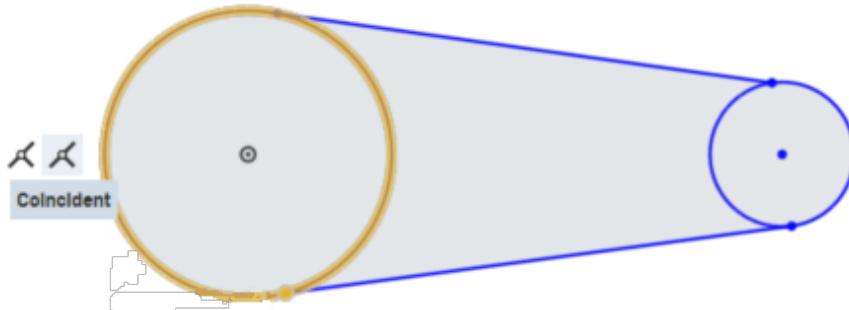


Whenever a cursor is touching a sketch entity, that entity is highlighted in orange. Click when the entity turns orange: this means the entity is selected and there is an automatic constraint applied between the two entities. The shading indicates a closed profile.

3. **Check to see if any automatic constraints were applied:**

- With no tools selected, hover over each of the lines, checking for constraint icons.
- Hover over any constraint icons you come across to see which geometry is highlighted and what the relation is.

## Constraints example

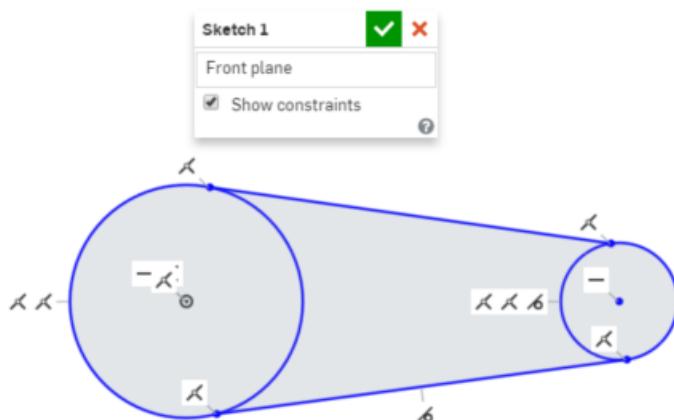


*This shows that the coincident constraint relates to the large highlighted circle and to the end point of the bottom line, also highlighted.*

Hover over the first constraint icon and two pieces of geometry are highlighted.

Note that your results may differ from these.

You can also click the **Show constraints** check box in the Sketch dialog to toggle all constraints in the sketch on and off.



If the display becomes too cluttered with icons, you can click+drag the icons around in order to make viewing easier.

If you find unintentional constraints, click on the constraint icon and use the Delete key to remove the constraint.

## Apply a tangent constraint

1. Click the top line to select it.

Selected entities appear highlighted.

2. Click the edge of the small (right) circle to select it.

Make sure to click the edge of the circle; tangent applies only to sketch entities, not regions.

3. Click the **Tangent constraint tool**  in the toolbar to apply the constraint.

## Tangent constraint tool in toolbar



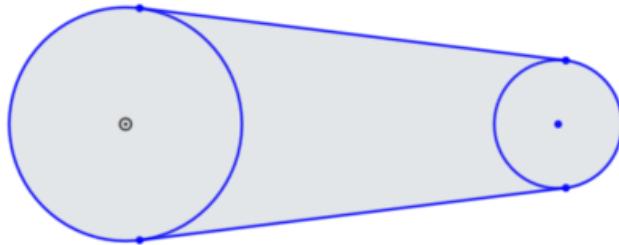
To test the constraint, click and drag the edge of the circle; the top line should move with it.

4. Make sure both lines have tangent constraints applied between each line and each circle. (Perform this operation a total of 4 times, once for each line/circle combination: top line/left circle, top line/right circle, bottom line/left circle, and bottom line/right circle.)
5. Hover on a sketch entity or check the **Show constraints** box in the sketch dialog.

You should see 8 tangent icons.

This forms a closed profile and your sketch forms a third region (indicated by shading) between the two circles (the inside of which are also regions).

## Shaded regions

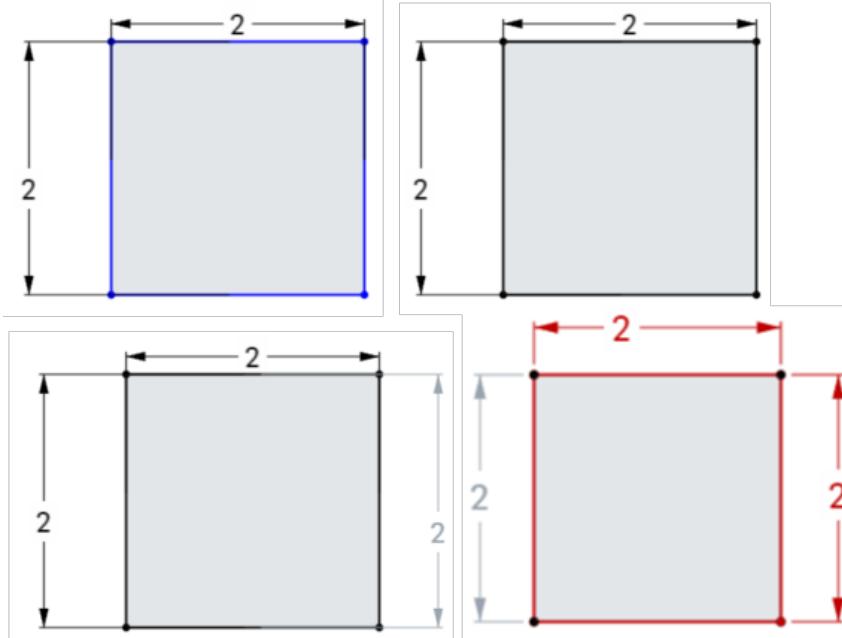


Time to check the definition of the sketch again: click and drag each blue sketch entity geometry to see if and how it moves. Now you can add some dimensions to further define the sketch.

What you did doesn't match what you see here? Try troubleshooting the sketch geometry:

- The color of sketch entities indicate its constrained status:
  - Blue means under-constrained
  - Black means fully constrained
  - Gray indicates a driven dimension
  - Red indicates an over-defined entity

- The color of a constraint icon indicates its constrained status: black on white or gray is well-defined, white on red indicates a problem.
- Adding more dimensions or constraints will further constrain the sketch. Dragging entities can help you understand what constraints or dimensions you may want to add.



See the video titled Sketching Basics and the video titled Dimensions & Constraints for more details.

## Add sketch dimensions

1. Make the top line 8 in. long.

a. Select the Dimension tool

### Dimension tool in toolbar



- a. Click on the top line.  
b. Move the mouse away from the line.  
c. Click again to activate the value field.  
d. Type 8; press Enter.
2. Make the small (right) circle diameter 2 inches. With the Dimension tool selected:
  - a. Click the edge of the small (right) circle.  
b. Move the mouse away from the curve.  
c. Click again to activate the value field.  
d. Type 2; press Enter.

3. Make the large (left) circle diameter 6 inches. With the Dimension tool  selected:
  - a. Click the edge of the large (left) circle.
  - b. Move the mouse away from the curve.
  - c. Click again to activate the value field.
  - d. Type 6; press Enter.

## Experiment with over-constraining

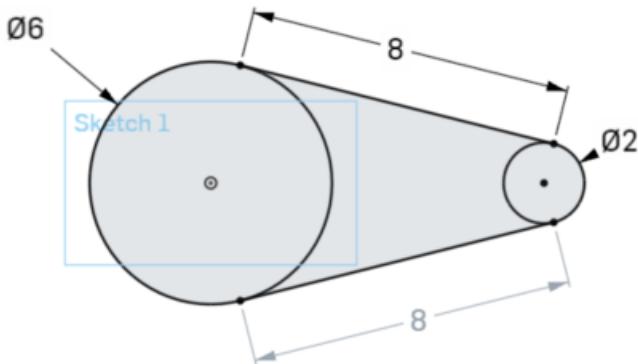
All sketch entities should be black, indicating that all are fully defined. There should be no (blue) entities that you can click and drag.

With the Dimension tool  selected:

1. Click the bottom line and add a dimension of 8.

The dimension turns gray, indicating that it is a driven dimension, dictated by the other dimensions in the sketch.

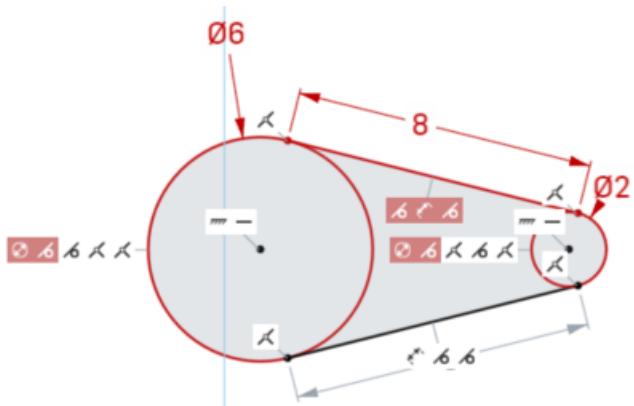
## Driven dimension



2. Escape from the Dimension tool, select the center point of the small circle and click the Fix  icon. Also select the center point of the large circle and click the Fix  icon.

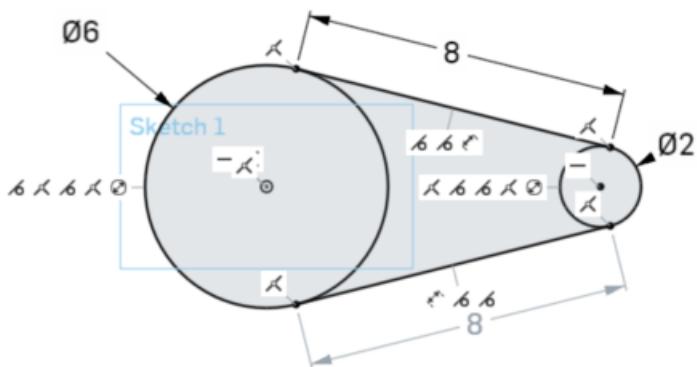
The sketch turns red, indicating that it is over-defined.

## Over-defined sketch



3. Delete the last applied Fix constraint (on the large circle center point). Hover on the center point, click the Fix icon that appears and press the Delete key.
4. Click in the Sketch dialog to accept the sketch.

Your finished sketch should resemble this:



## See Also

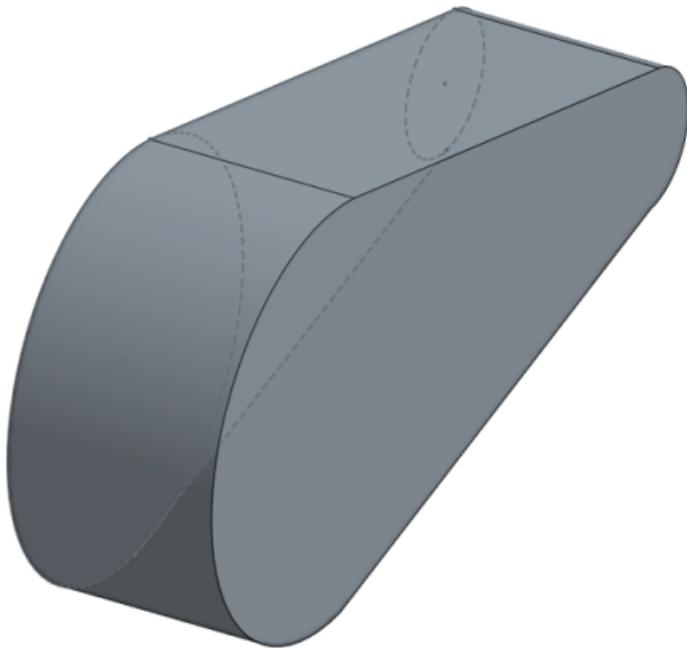
"Automatic Inferencing" on page 175

"Keyboard Shortcuts" on page 69

Next: "Extrude Geometry and Create a Part" on the next page

# Extrude Geometry and Create a Part

Continuing on with the sketch just created, use the **Extrude tool**  to create a part from the sketch. A new feature will be created in the Feature list. Begin with the sketch unopened.



Extrude extends a sketch profile along a path for a specified distance. This creates the solid model.

## Extrude the sketch into a part

With the sketch accepted and the dialog closed, the Feature toolbar reappears.

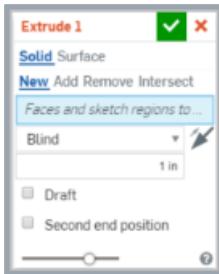
1. Select **Extrude tool**  from the Feature toolbar. (Note that Extrude and Revolve are also available from the Sketch toolbar.)

## Extrude tool in toolbar



The Extrude dialog opens:

## Extrude dialog

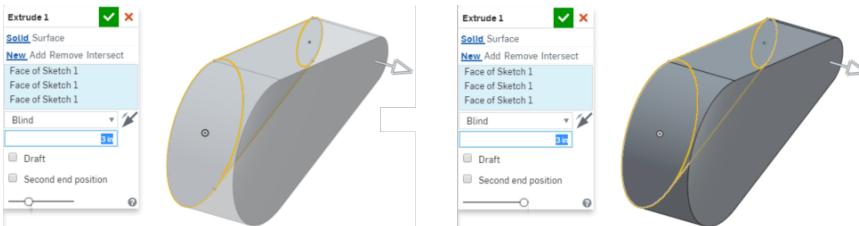


All objects in the graphics area are movable. Click and drag the title bar of the dialog to move it around the graphics area and out of the way, if necessary.

Blue highlighting on a field in a dialog indicates input is through selection in the graphics area.

2. Select regions to extrude by clicking on the shaded region in the graphics area.  
Once selected, a region's shading is darker.
3. Select all three regions.
4. Leave **Blind** to indicate the End type.
5. Click in the value field and change 1 in. to **3 in.**; press Enter.
6. Use the slider bar at the bottom of the dialog to see more or less of the extrude and how it will look when accepted.

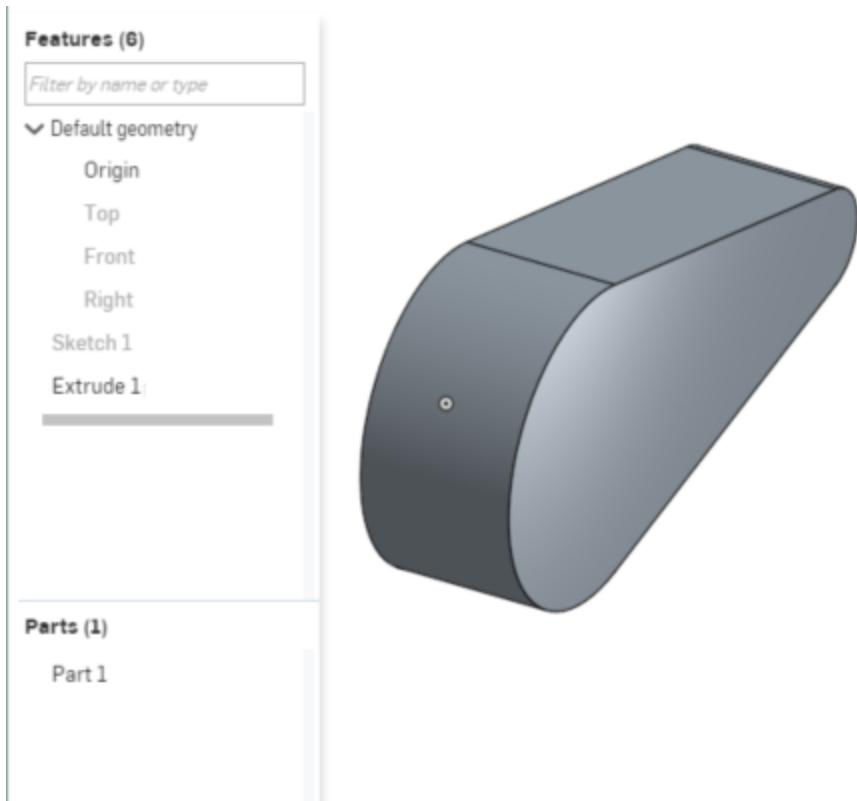
## Slider bar examples



*Move the slider to the right to visualize the feature once all edits are accepted while the dialog is still open for editing.*

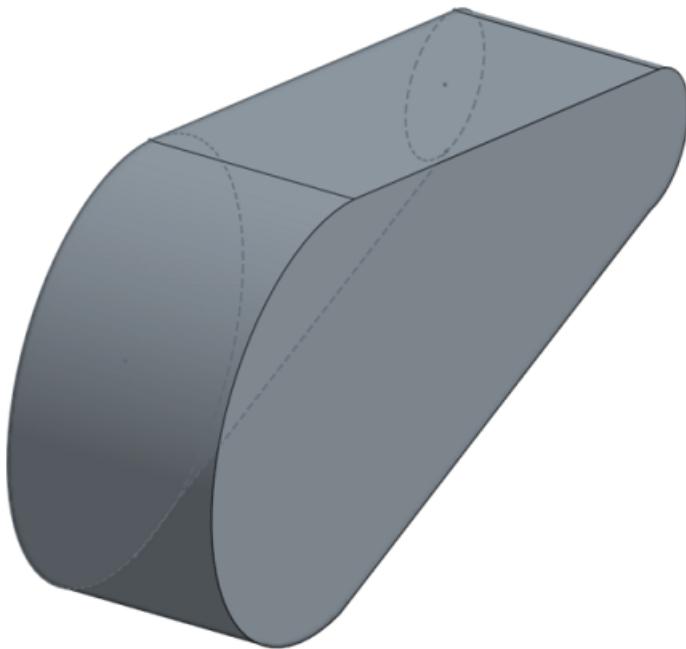
7. Click to accept the feature.
- Extrude 1 is now listed in the Feature list. Note the part also listed in the Parts list.

## Extrude 1 in Feature list



8. In the Feature list, right-click on **Extrude 1**, select **Rename**, and change the name to **Extrude Part**.

Your finished part should resemble this:



## Experiment with Extrude Options

Double-click the **Extrude Part** feature to reopen the dialog. Experiment with Extrude options by using the settings below:

1. **Surface with Blind, 3 in: Select the edges of the circles.**

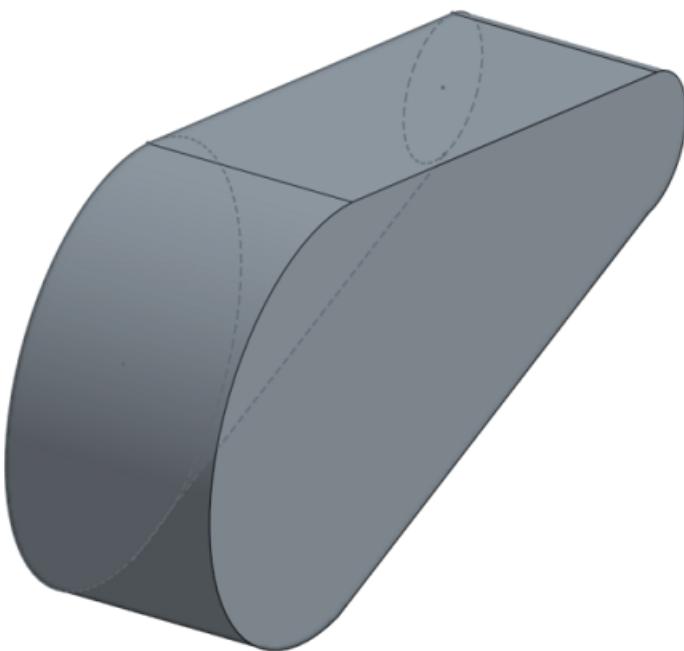
This extrudes only the edges of the circles, forming cylinders.

2. **Surface with Symmetric, 3 in: Select the edges of the circles.**

This extrudes the same way, but in both directions equally about the sketch plane.

3. Close the dialog without saving; click .

Before you continue with the tutorial, make sure your finished part resembles this:



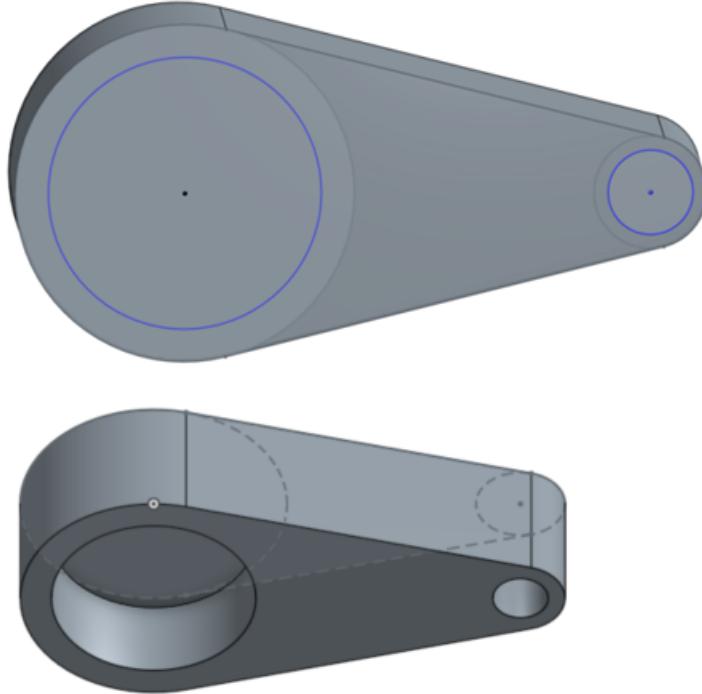
## See Also

"Extrude" on page 182

Next: "Sketch on a Planar Face and Extrude Remove" on the next page

# Sketch on a Planar Face and Extrude Remove

Once you have a solid, 3-dimensional part, you can sketch on the surface of that part. This exercise illustrates sketching on a planar face of a part, and then using Extrude to remove material and create pockets on the part.



## Sketch pockets on an extruded face

1. Click **Sketch** .
2. For the sketch plane, select the planar face of the extrude you just created.

## Face of Extrude

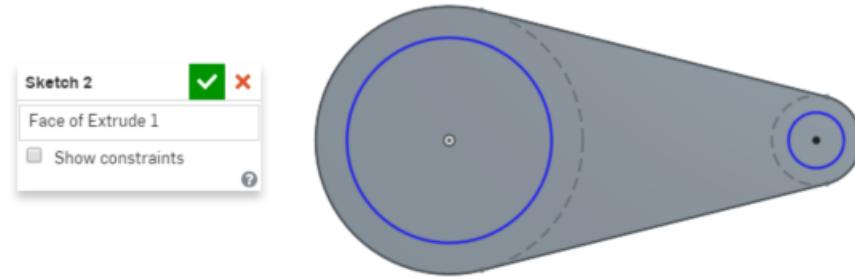


- Add two circles , starting at the center point of the existing circles, only a little smaller. Remember you can use the shortcut key **n** to orient the part normal to plane for easier viewing and sketching.

If you can't see the previous sketch entities to find the center points of the circles, hover next to the sketch name in the Feature list to see the Eye icon . Click on the Eye icon to show the sketch: **Eye icon sketch**



## Two smaller circles

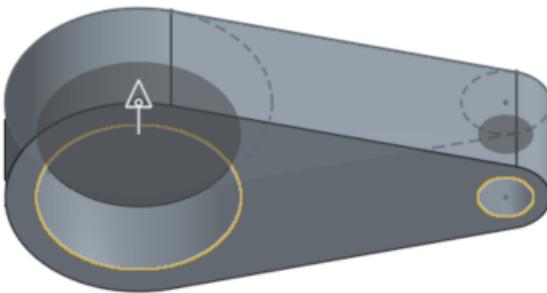
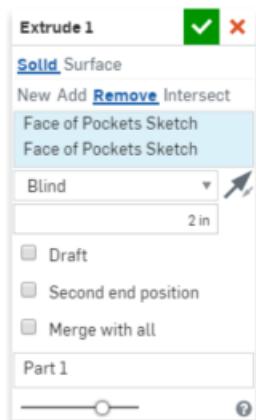


- Dimension the left circle to 5.25in and the right circle to 1.25in.
- Accept the sketch.
- Rename the sketch to **Pockets Sketch**.

## Remove material

- Select the **Extrude tool** .
- With the *Faces and Sketch Regions* field of the dialog active, click on the two new circles you just created. In the field there should be one *Face of Sketch 1* for each circle selected.

## Face of Pockets sketch



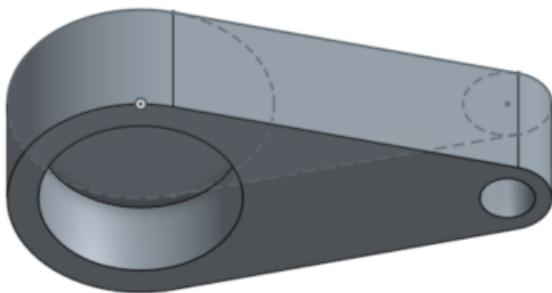
If you mistakenly select other geometry, hover over the name of it in the dialog and then use **X** next to that name to remove it from selection. You can also click on the entity in the graphics area to toggle the selection off.

3. In the dialog, make sure **Solid** and **Remove** are selected as well as the two circles:
  - a. Verify that the drag arrow is pointing into the part in the graphics area. If it is not, click the arrow to reverse the direction or click the Directional arrows in the dialog box .
  - b. Specify **2 in.** and press Enter.
4. Accept the feature; click .

There is a new feature in the Feature list.

5. Right-click **Extrude 2**, select **Rename**.
6. Rename the feature to **Extrude Pockets**.

Your part should resemble this:



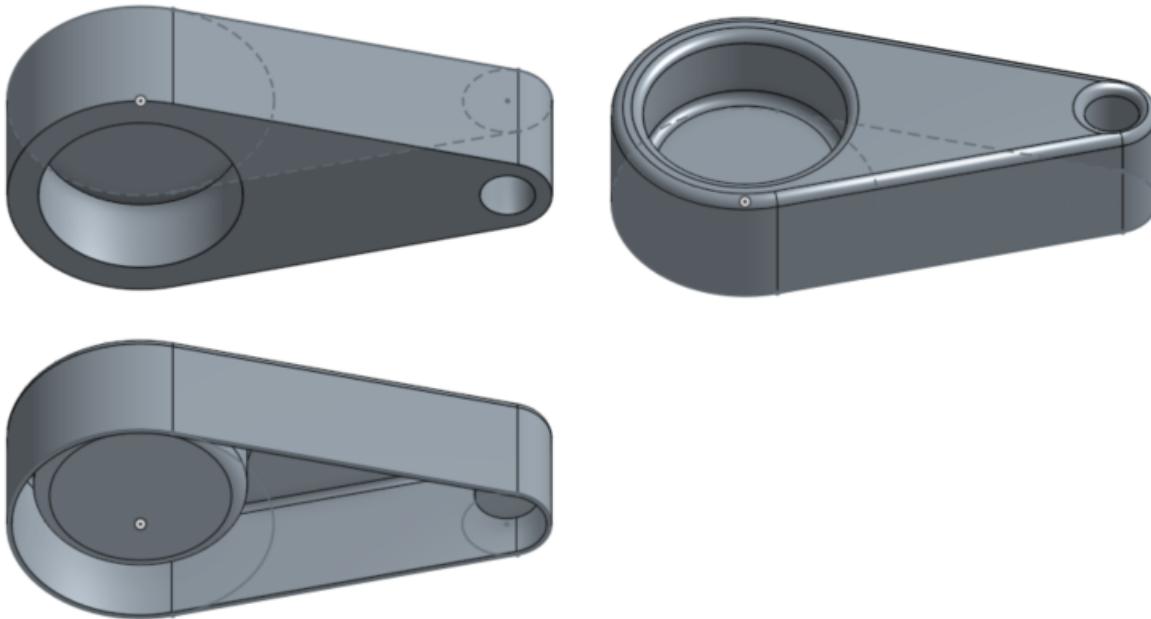
## See Also

"Extrude" on page 182

Next: "Apply Fillets and Shell a Part" on the next page

# Apply Fillets and Shell a Part

This part of the exercise illustrates how to soften hard edges by applying fillets.



## Apply fillets

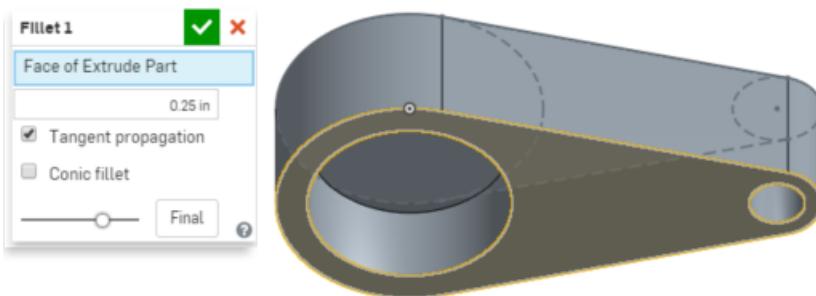
1. Select the **Fillet tool** .

### Fillet tool in toolbar



2. With the *Entities to fillet* field active, select the top face of the part.

### Top face

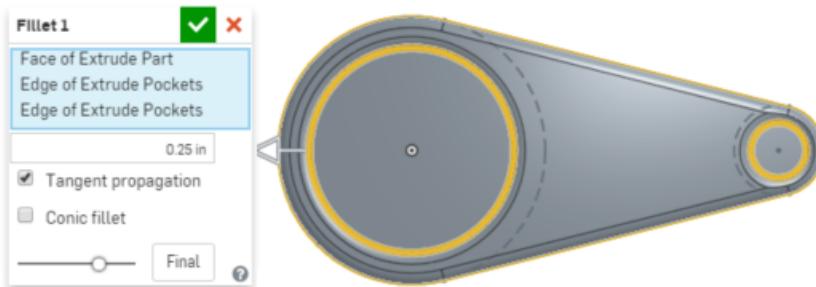


By selecting the face, not only does the top edge of the part have the fillet applied, but also the top edges of both pockets. You could also have selected each edge separately.

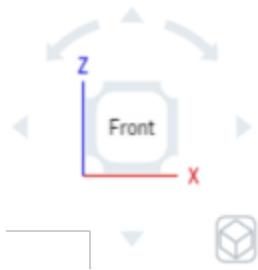
3. Select the inside bottom edges of both pockets.

Because the fillets are being applied to circles and there are no tangents to the edges, it makes no difference if the Tangent propagation box is checked.

## Bottom edges of pockets



Rotate the part if necessary to reach the geometry to click on. Right-click and drag, or use the View Cube and/or the View Cube arrows. View cube



4. Change the fillet dimension to be .25 in. and press Enter to see the result.
5. Click  to accept the feature.
6. Rename the feature to **Fillet Part 1**.

## Apply a Shell feature

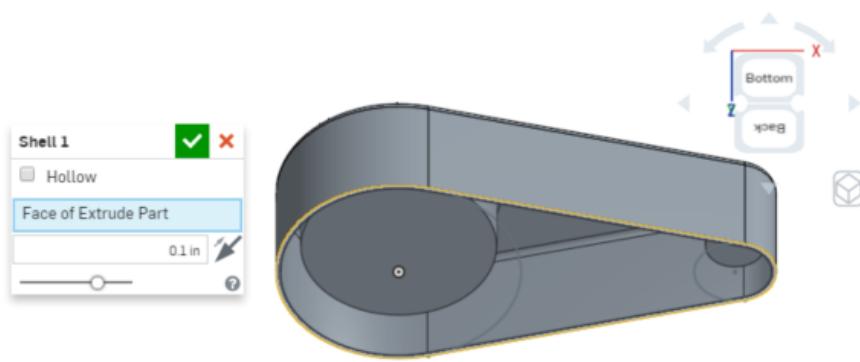
1. Select the **Shell** tool

### Shell tool in toolbar



2. Rotate the part such that you can click on the bottom face. (Depress the right mouse button and drag.)
3. Select the bottom face.

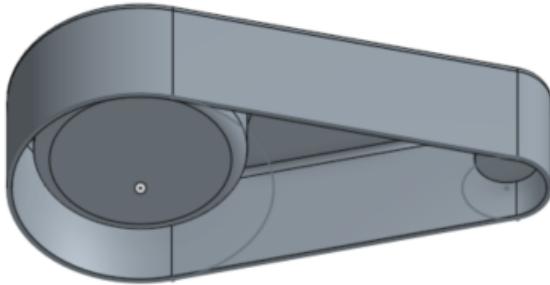
### Shelled part



4. Leave the wall thickness at the default of **0.1 in**.
5. Click to accept the feature.

Notice that the bottom edges of the pockets and the inside edges of the part are also filleted.

6. Rename the feature to **Shell Part 1**.

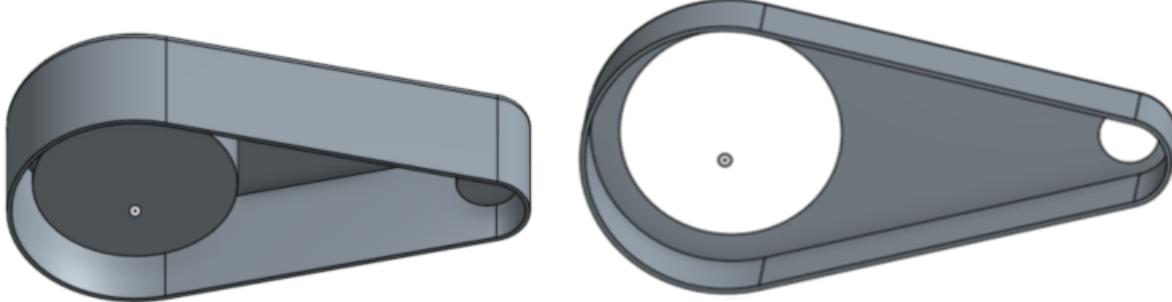


## See Also

- "Fillet" on page 213
- "Shell" on page 223
- "View Navigation and Viewing Parts" on page 35
- Next: "Reorder Parametric History" on the next page

# Reorder Parametric History

The next step of this tutorial demonstrates how changing the order of features in the Feature list can change the resulting part.

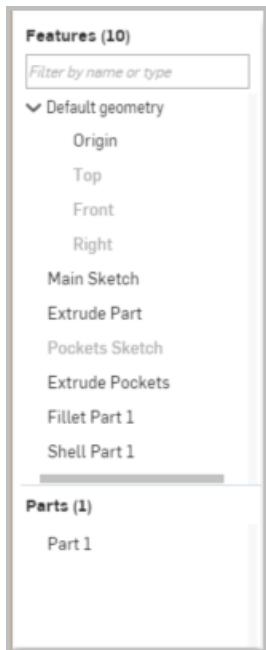


## Reorder Fillet and Shell

1. Examine feature names in Feature list.

Look at the Feature list; because you took time to apply meaningful names, each feature's purpose should be obvious.

## Feature list



Some features in the list may appear grayed-out. These are features that are hidden in the graphics area. Hover over the name to activate the eye icon . Click the Eye icon to hide/show features in the graphics area.

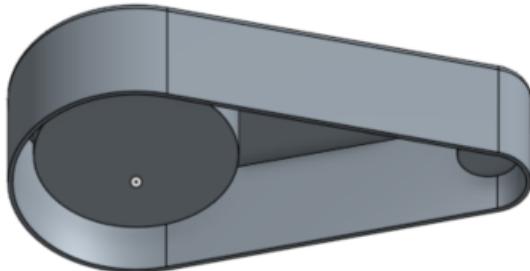
2. Click and drag **Shell Part 1** to just above **Fillet Part 1**.

## Shell above Fillet



Notice how the bottom (inside) fillets change. Since the part is now shelled before the filleting is applied, the inside edges of the part have not been selected for filleting and still have hard edges.

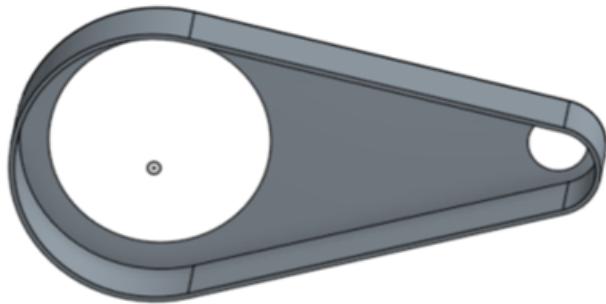
## Bottom fillets changed



## Reorder Shell and Extrude

1. Click and drag **Shell Part 1** to the spot above **Extrude Pockets**.

## Shell above Extrude results in missing pockets



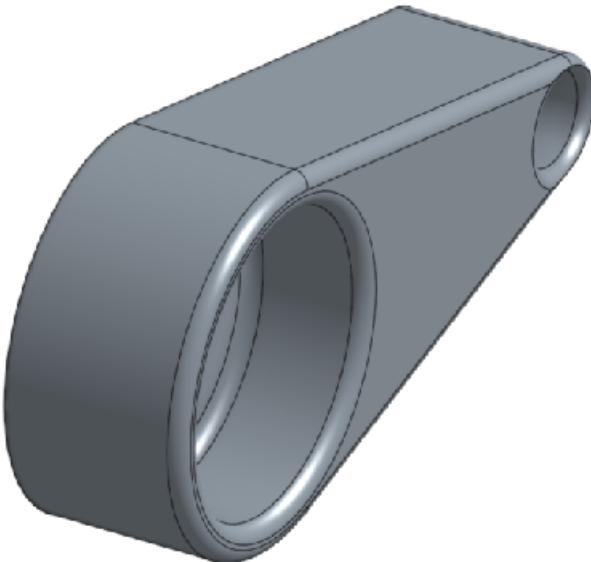
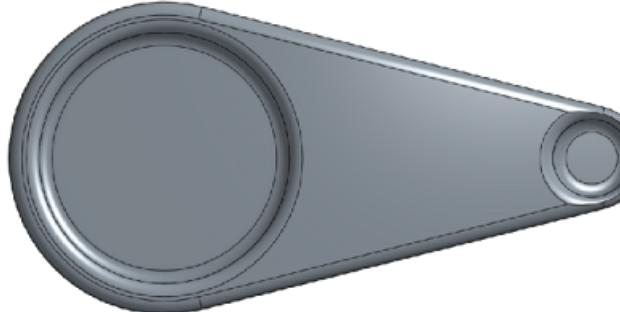
Observe what it would be like to have the part shelled before the pockets are formed: the part now has holes where the pockets used to be because the bottom face was shelled before the circles were extruded into pockets.

2. Click **Undo** ↪ until the part resembles the original *Part 1* before the reordering took place.

Next: "Congratulations!" on the next page

# Congratulations!

Congratulations! You have created your first part using Onshape.



Next: Follow this link for more learning opportunities:

[Onshape Videos](#)

# Help in PDF Format

Click the link below to open this help system in PDF format, in a new tab or window.

[Onshape Help System in PDF format](#)

# Glossary

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**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 or subassemblies that describes both position and movement.

**assembly toolbar**

The series of tools available for creating assemblies.

---

**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 branches of the same document without impacting the 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 who can view or edit a document that other users can also view or edit.

---

**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. Each tab can contain either a Part Studio, Assembly, Drawing, or an imported file like CAD files, PDF files, Word files, etc.

## Documents page

The Onshape page that lists documents and allows the user to open, create, and import documents.

## drag manipulator

A manipulator used to resize features.

## E

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### 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 or surface 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.

## G

---

### graphics area

The large rectangular portion of the user interface in which a Part Studio or Assembly is displayed.

---

I

## **Inference**

An automatic indication during sketching that a constraint may be applied; appears as an orange dotted line.

## **instance**

A part or subassembly used in an Assembly.

---

M

## **mate**

An Onshape feature used to position part 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 move edits to a workspace from one branch into a workspace on another branch of the same document.

---

N

## **navigation bar**

The top bar of the user interface window that contains the Document name and User ID/profile menu.

---

O

## **owner**

The creator of a document.

---

P

## **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 shared with all Onshape users. Public documents are read-only.

# R

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## 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.

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## S

### 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 cannot be brought into an Assembly. In some traditional CAD systems, Onshape surfaces are similar to Sheet Bodies.

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## T

### tab

An entity in Onshape that can contain a Part Studio, Assembly, Drawing, jpg file, PDF file, Word files, 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. There are three toolbars in Onshape: the Sketch toolbar, the Feature toolbar, and the Assembly toolbar.

## traditional CAD

Older desktop CAD systems like SolidWorks, Pro/ENGINEER, CATIA, and Inventor.

## triad manipulator

A manipulator that appears 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

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### user

An individual account that provides access to Onshape; the user name can be seen in Navigation bar, in the right corner.

## V

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### version

A snapshot of a document at a particular time. A version is created using the Save 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, edit properties, delete, merge, and view the History.

### View cube

The cube appearing in the top right corner of the model view 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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