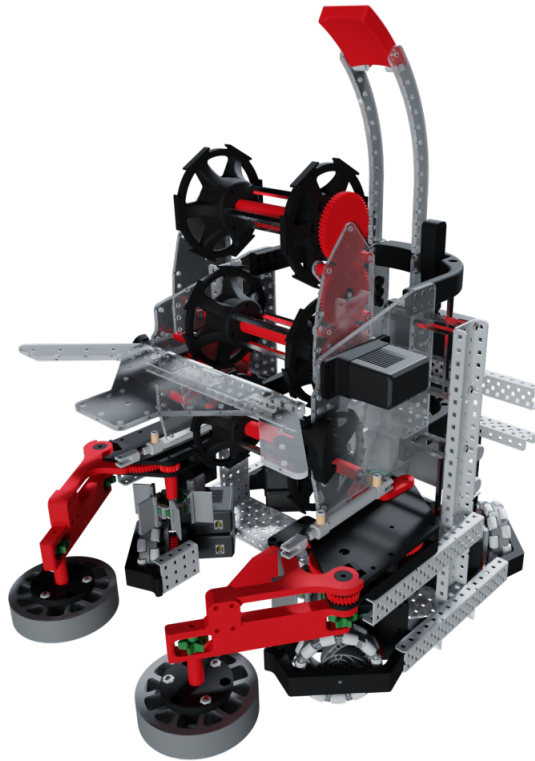

Alex's FeatureScript Documentation

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Welcome to the documentation site for all of my FeatureScripts. FeatureScripts are powerful design tools which can be used to simply or automate many aspects of design in Onshape.

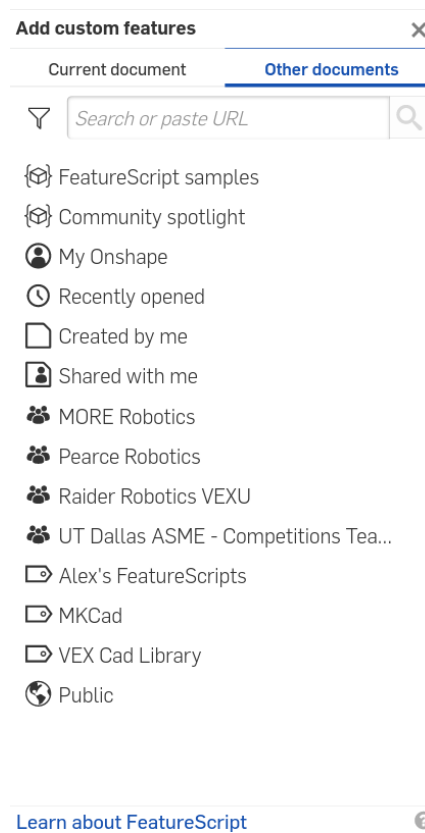
To learn how to install and use FeatureScripts, see [Getting Started](#).

GETTING STARTED

Note: FeatureScripts are only available inside Onshape part studios - not assemblies or drawings.


1.1 Steps for adding FeatureScripts to your toolbar

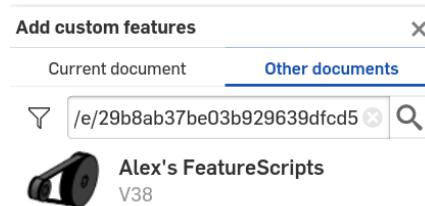
1. Open any Onshape document, then open any part studio you can edit.
2. Open your FeatureScript dropdown by clicking on  in the rightmost part of your toolbar.
3. Choose **Add custom features** to open the **Add custom features** menu.



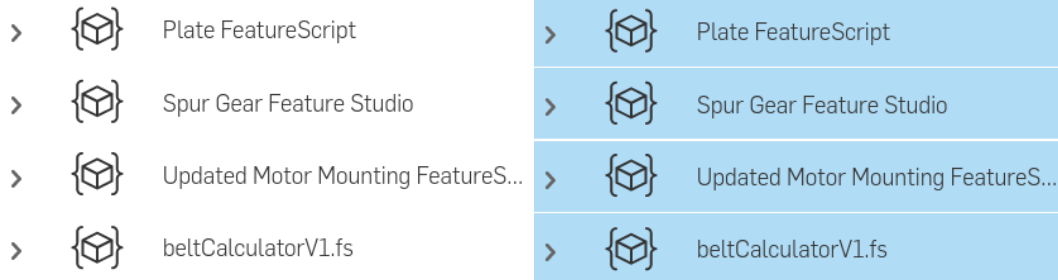
4. Paste a link to an Onshape document containing one or more FeatureScripts into the top search bar of the **Add custom features** menu.



5. Click  or press enter.
6. Click the document which appears.





7. A list of one or more feature studios should appear. Click each desired FeatureScript to add it to your FeatureScript dropdown, which will cause it to turn blue.



8. Click  to close the **Add custom features** menu.

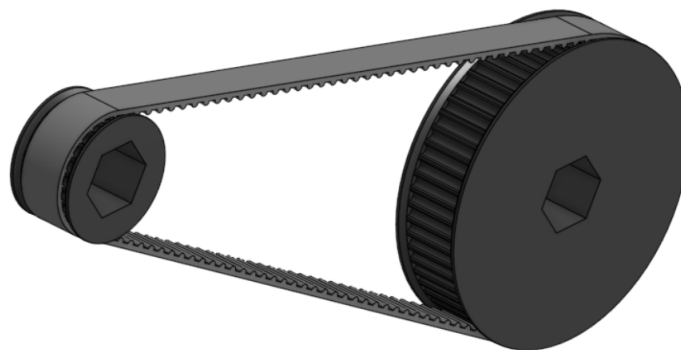
Tip: Once a FeatureScript has been added to your FeatureScript dropdown, you will be able to use it in every part studio you open.

1.2 Steps for using FeatureScripts

1. Open your FeatureScript dropdown by clicking on  in the rightmost part of your toolbar.
2. Choose a FeatureScript you've added from the list of FeatureScripts. An instance of the FeatureScript will be added to your feature tree and allow you to begin making changes.
3. Once you've finished making changes, click .

ROBOTICS FEATURESSCRIPTS


2.1 FRC belt calculator



The FRC belt calculator feature allows users to quickly add belt and pulley assemblies to a part studio. Belts may be either GT2 or HTD, and configurable 3D printable pulleys may also be added.

The FRC belt calculator FeatureScript can be found here: [FRC belt calculator document](#)

2.1.1 Steps for creating belt runs

1. Create an FRC belt calculator feature by selecting it from your *FeatureScript dropdown*.
 2. In the **Position** tab:
 - Specify a **Pulley one position** and a **Pulley two position** to use.
 - If desired, specify an **Offset location** (**Center**, **Pulley one**, or **Pulley two**) and a **Offset** for your belt run, and use  to reverse the direction of the **Offset**.
- ..note ::** If you set the **Offset location** to **Pulley one** or **Pulley two** before adding a corresponding **Custom pulley** in the **Pulleys** tab, the **Offset** will be applied from the middle of the belt run.

FRC Belt Calculator 1
✓ ✗

Position
Belt
Pulleys

Pulley one position
Vertex of Sketch 1 ✕

Pulley two position
Vertex of Sketch 1 ✕

Offset location Pulley one

Offset 0.188 in

3. In the **Belt** tab:

- Specify your desired belt standard (**9mm wide GT2**, **9mm wide HTD**, or **15mm wide HTD**).
- Choose whether you'd like to automatically choose the belt size (**Auto choose belt**) based on the computed center to center distance (as determined by the distance between your desired **Pulley positions**, the teeth of **Pulley one** and **Pulley two**, and the **Center to center adjustment**).
- Specify a supplier inventory you'd like the belt to be chosen from, or choose **Any belt size** to choose from any available tooth count.
- If you've chosen to size the belt manually, enter the desired number of **Belt teeth**.
- Choose whether to **Enable belt teeth** in order to also create an approximate tooth representation.
- If desired, specify a **Center to center adjustment** to modify the standard belt center to center distance.

FRC Belt Calculator 1
✓ ✗

Position
Belt
Pulleys

9mm wide GT2

☐ Auto choose belt

VEXpro belts

VEXpro belts 105T

☒ Enable belt teeth

Center to center adjustment 0 in

4. In the **Pulleys** tab:

- Specify the configurations of **Pulley one** and **Pulley two**:
 - Specify whether you'd like to create a 3D printable **Custom pulley** or **No pulley**.
 - Enter the number of **Pulley teeth**.
 - If you've chosen **Custom pulley**, specify:
 - * The **Pulley width**
 - * The **Bore type** (**Hex**, **Circular**, **VEXpro spline**, or **None**)

- * The **Bore diameter**
 - * Whether to **Enable flanges**
 - * The **Flange width** (if flanges are enabled)
 - * The **Pulley teeth size adjustment**
- Choose whether to create the belt run as a single composite part (**Create composite part**).

Note: Creating a belt run as a single composite part can help simplify the assembly process later on.

FRC belt calculator 1 ✓ ✗

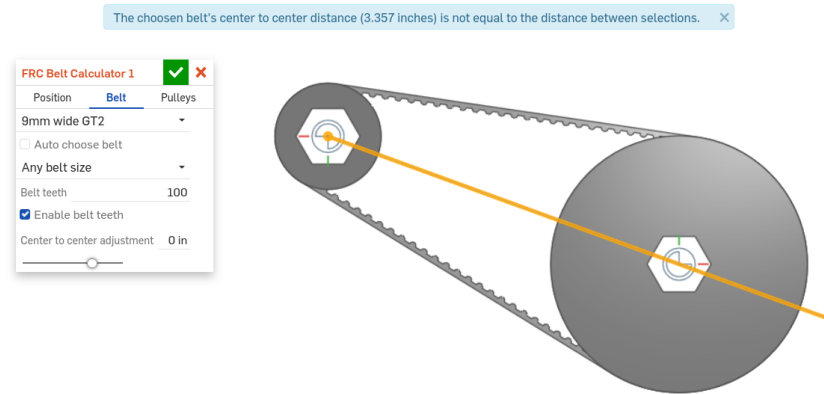
Position	Belt	Pulleys
<div> <div>▼ Pulley one</div> <div> <div>Custom pulley ▼</div> <div>Pulley one teeth 24</div> <div>Pulley width 0.438 in</div> <div>Bore type Hex ▼</div> <div>Bore diameter 0.5 in</div> <div><input checked="" type="checkbox"/> Enable flanges</div> <div>Flange width 0.063 in</div> </div> </div>		
<div> <div>▼ Pulley two</div> <div> <div>No pulley ▼</div> <div>Pulley two teeth 60</div> </div> </div>		
<div> <div>Pulley teeth size adjustment 0 in</div> <div><input checked="" type="checkbox"/> Create composite part</div> <div> <div></div> <div></div> </div> </div>		

5. Click ✓.

2.1.2 Center distance checks

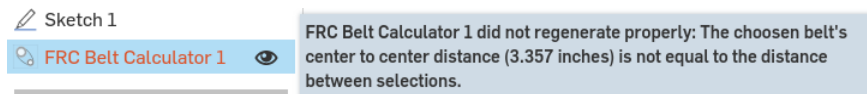
When both pulley positions are defined in the **Position** tab, the FRC belt calculator FeatureScript will automatically measure the distance between the selected positions and report a warning if the distance between the belts does not match the center to center distance of the belt run.

Note: The belt center to center distance is determined by the number of **Belt teeth**, the teeth of **Pulley one** and **Pulley two**, and the **Center to center adjustment**.



To correct a belt center to center distance error, change the distance between your **Pulley one position** and **Pulley two position** selections to match the value displayed in the error tooltip.

Tip: The error tooltip can be seen at the top of the screen while the FRC belt calculator feature is being edited, or by mousing over a broken FRC belt calculator feature in the *feature tree*.



2.2 Plate

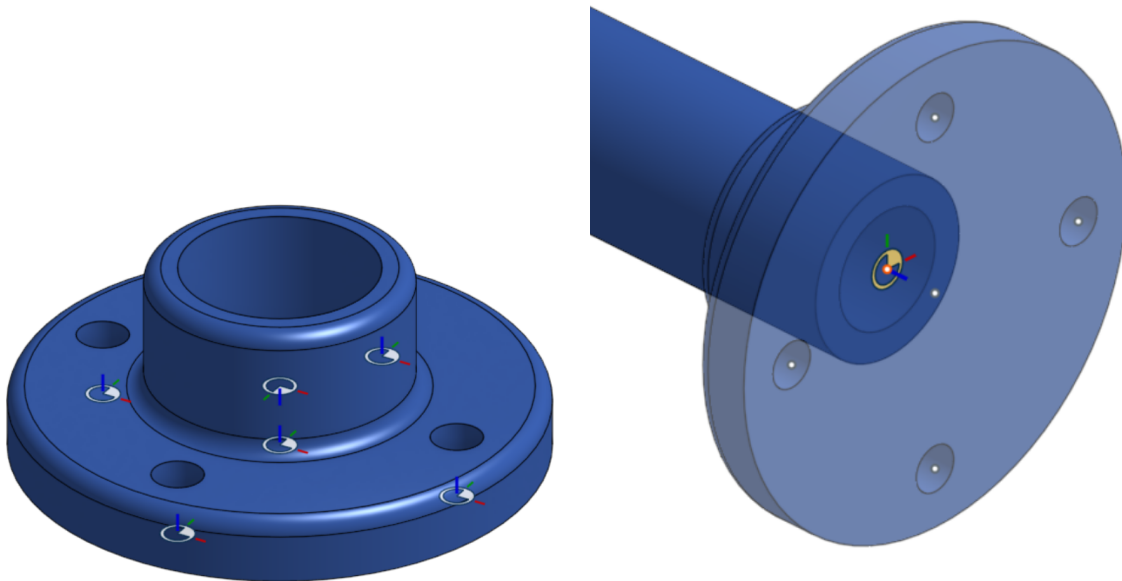
The Plate FeatureScript can be found here: [Plate document](#)

Warning: The latest version of the Plate FeatureScript is still experimental. Thus, there are a few known issues and other bugs associated with it, and it may behave improperly in certain situations.

GENERAL PURPOSE FEATURESRIPTS

These FeatureScripts have functionality which are applicable to a wide purpose audience.

3.1 Point derive



The Point derive feature is a powerful variant of the Onshape Derive feature which allows users to specify locations when deriving parts. Point derive also automatically converts mate connectors attached to parts which are being derived into selectable points, making it easy to derive and use standard components.




Note: Parts don't need to have mate connectors attached to them to be used with Point derive.

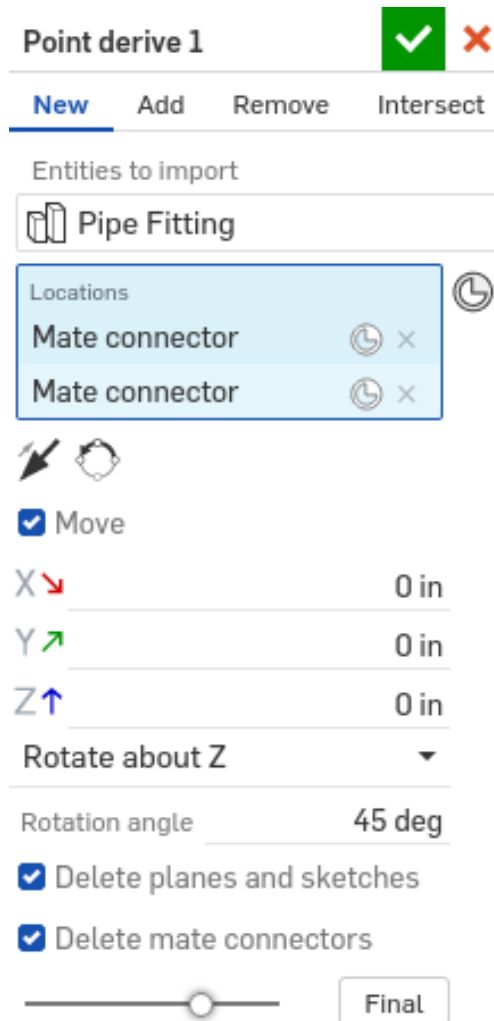
The Point derive FeatureScript can be found here: [Point derive document](#)

3.1.1 Steps for deriving entities

1. Create a Point derive feature by selecting it from your *FeatureScript dropdown*.
2. Specify **Entities to import**.
3. If desired, select one or more **Locations** to place the selected entities at by clicking sketch vertices or mate connectors in the *graphics window*.
4. If the derived part has any mate connectors, they will automatically be instantiated as selectable points in the *graphics window*. Choose the appropriate point to use by clicking on it.

Note: If your selections to derive don't have any attached mate connectors, no selectable points will appear.

5. Use  and  to reorient **Entities to import** as needed.
6. Choose whether to **Move** derived entities relative to their respective **Locations**.
7. Choose whether to **Delete planes and sketches** from **Entities to import**.
8. Choose whether to **Delete mate connectors** from **Entities to import**.
9. If desired, specify a part boolean operation (**Add**, **Remove**, or **Intersect**) and select a **Merge scope** (or choose **Merge with all**) to boolean parts in **Entities to import** to existing parts in the part studio.
10. Click .

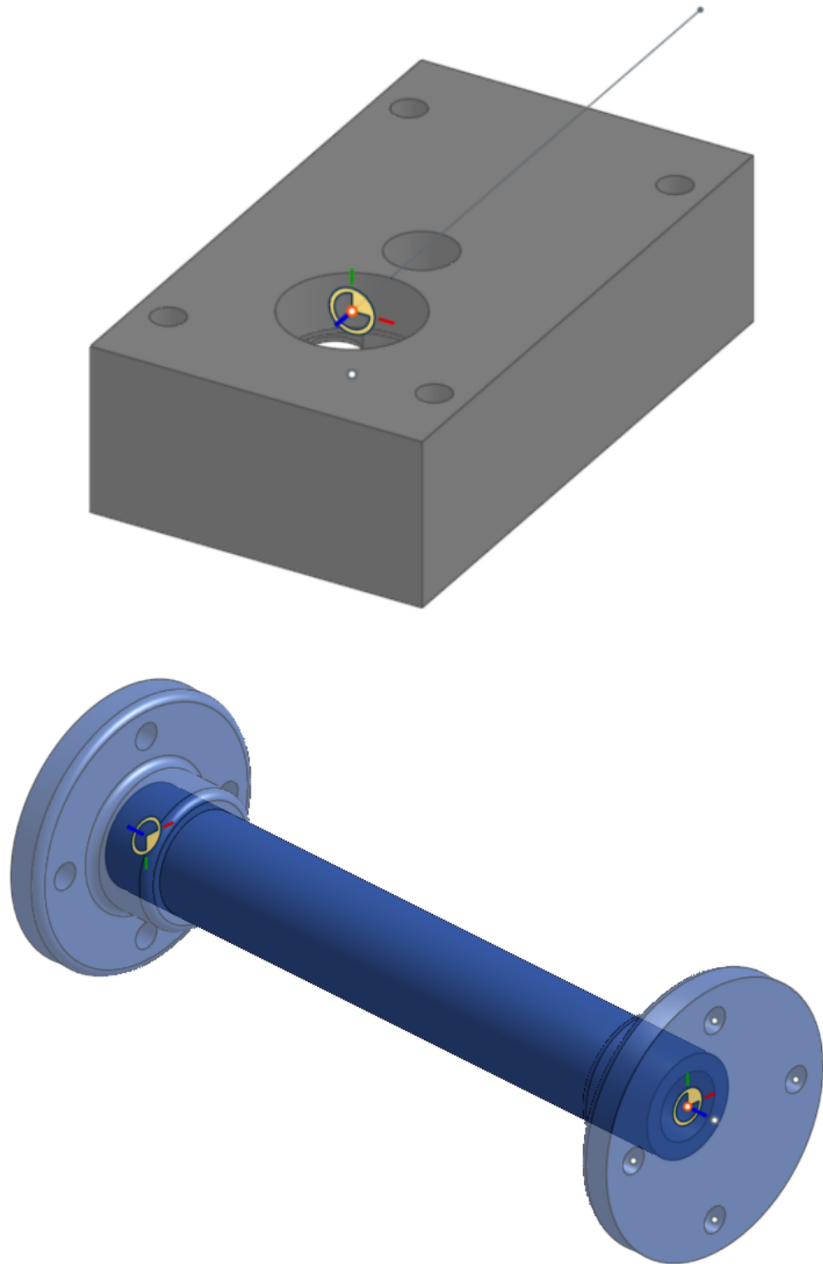


3.1.2 Tips for setting up parts to be derived with point derive

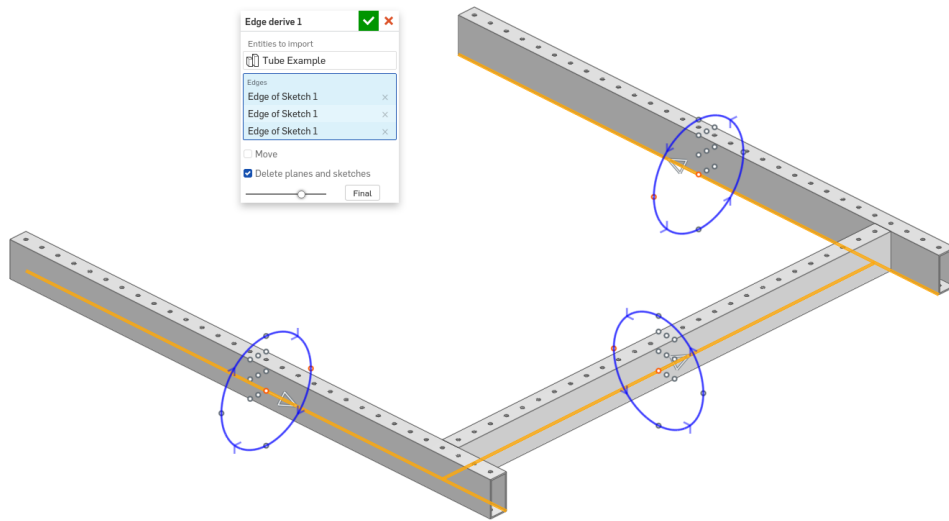
Tip:

- The orientation of mate connectors relative to one another is maintained.
- The first mate connector in the feature list of the derived part studio serves as the default selected point.
- Mate connectors do not have to touch derived entities to be used as selectable points.

3.1.3 Examples



3.2 Edge derive

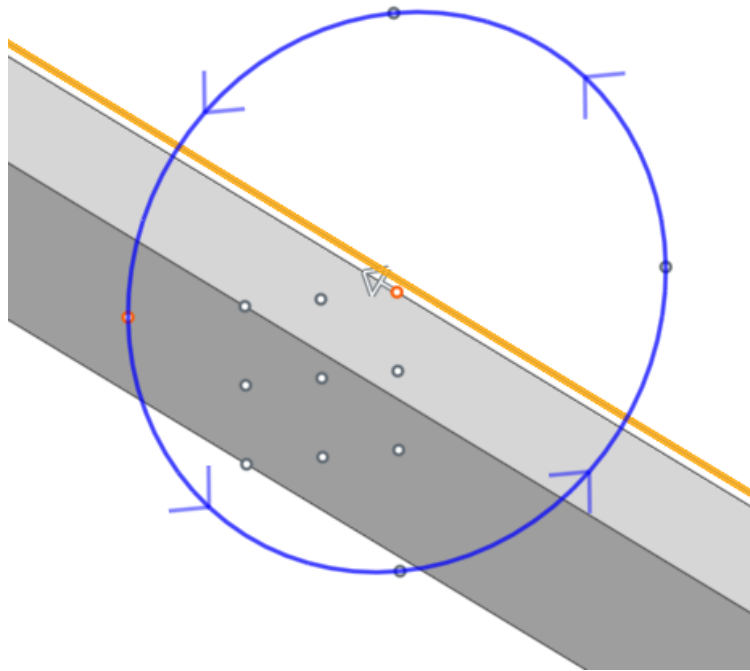



Edge derive makes it possible to quickly derive length-specific parts into a part studio. Mate connectors attached to derived parts are automatically converted into selectable points, and various positioning options are also available by default.



The Edge derive FeatureScript can be found here: [Edge derive document](#)

3.2.1 Steps for deriving entities


1. Create a Edge derive feature by selecting it from your *FeatureScript dropdown*.
2. Specify **Entities to import**.
3. Select one or more edges to derive parts along.
4. For each selected edge, a set of manipulators are automatically created in the graphics window which enable positioning.




1. Choose whether to **Move** parts relative to their reference edges.
2. Choose whether to **Delete planes and sketches** from **Entities to import**.
3. Click .

Edge derive 7




Entities to import



 Tube Example


Edges


Edge of Sketch 4
 

☒ Move


X 

Y 

Z 

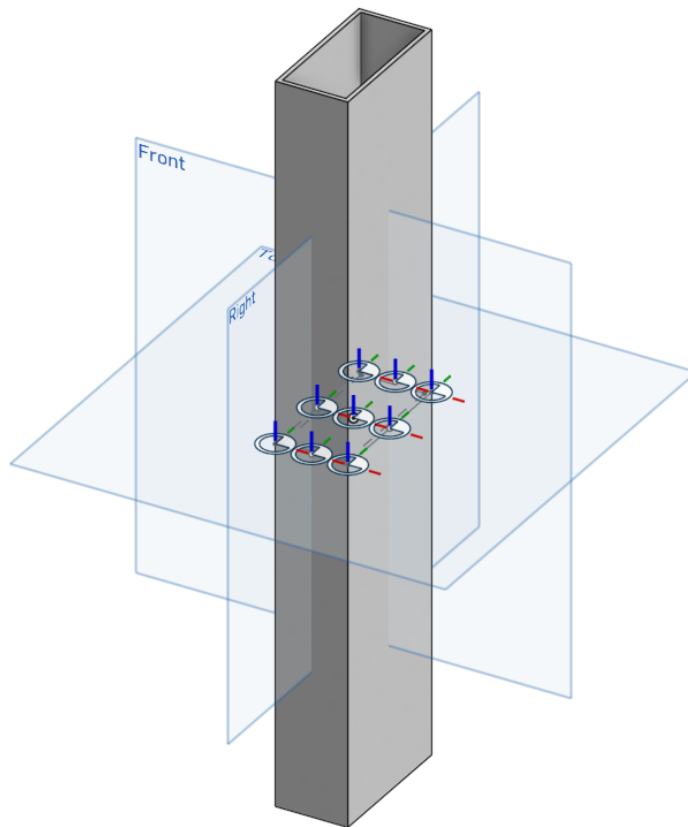
Rotation angle 

☒ Delete planes and sketches



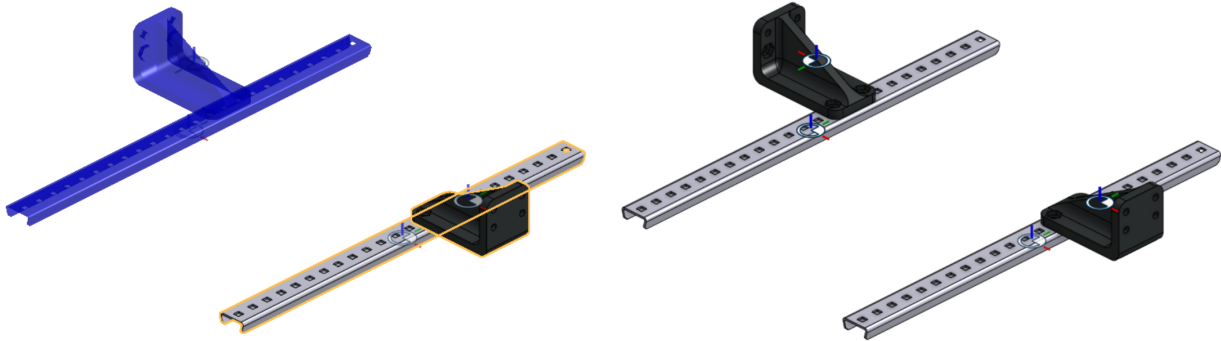
3.2.2 Steps for setting up parts to be derived with edge derive

1. Create a new part studio.
2. In the **Configuration panel** on the right side of the screen, add a **Configuration variable** named “length” (lowercase ‘l’).
3. Use the configuration to drive the length of your part.
4. If desired, add one or more mate connectors with their z-axis facing upwards to the middle of the part. When the part is derived using Edge derive, these mate connectors will be converted into selectable points.
5. Add additional configurations and modifications to the part studio as desired.



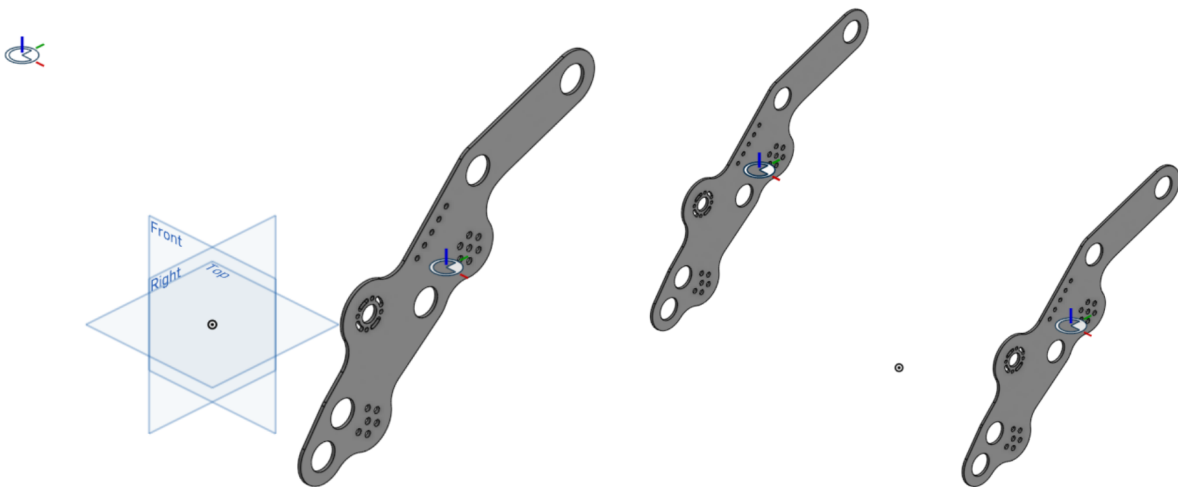
Note: A sample part studio can also be found here: [Edge derive example part studio](#)

3.3 Assembly mirror



Assembly mirror facilitates the assembly of parts by adding mate connectors to parts which allow instances of those parts to be quickly placed in mirrored positions in assemblies.

Important: Since FeatureScripts cannot be used directly in assemblies, the Assembly mirror feature cannot create transformed copies of parts directly inside assemblies. Instead, the Assembly mirror feature works by adding mate connectors to parts inside part studios which allow copies of those parts to be assembled into mirrored positions later on.



The Assembly mirror FeatureScript can be found here: [Assembly mirror document](#)

3.3.1 Steps for creating mirrored parts

1. Create an Assembly mirror feature by selecting it from your *FeatureScript dropdown*.
2. Specify your desired result behavior (**Part center** or **Origin**).

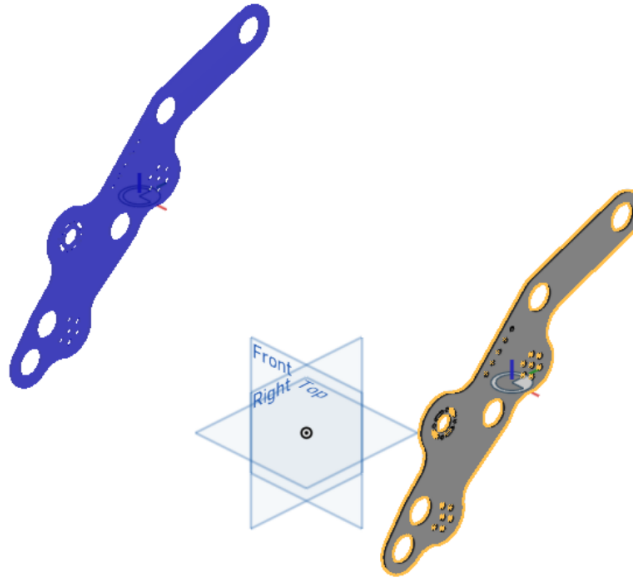
See also:

Result behavior for more information.

1. Select one or more **Entities** (parts or surfaces) to find transforms for.
2. Select a **Mirror plane** to mirror about.

3. The Assembly mirror feature will automatically try each possible transform type (**Linear**, **Z axis**, **Y axis**, **X axis**, and **Mirror**) for each part and select the one that matches a part to its mirror image.


If a valid transform is found, the transform type of the part will be updated with the valid transform, a mirror image of the part will be shown in blue in the *graphics window*, and a mate connector which can be used to assemble the part in the location of its mirror image will be created.



Otherwise, the transform type of the part will be changed to **Mirror**, and a mirrored copy of the part will be created.

Warning: Since Assembly mirror cannot compute surface transform types automatically, the transform type of surfaces will always default to **Mirror** and will need to be changed manually.

Note: The transform type will only be computed automatically when a feature is first selected. If you make changes to the part later on, you may need to reselect the part or update the transform type manually.

1. Choose whether to **Add prefix and suffix** to the mirrored versions of entities.
2. If **Add prefix and suffix** is selected and the name of the part is changed, the name of the mirrored part may not update to match the change. This can be fixed by selecting **Update saved names**, and then **Confirm**, to update the internally saved names of every part
3. Click .

Assembly mirror 2
✓ ✕

Part center
Origin

Entities
 > 2020_INTAKE_SIDE_PLATE ✕

Mirror plane
 Right plane ✕

☒ Add prefix and suffix

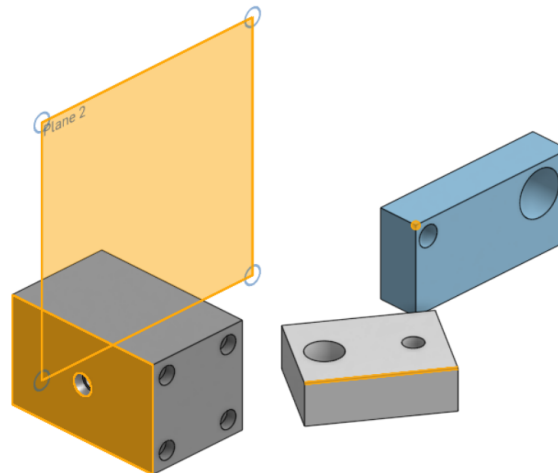
Prefix MIR-

Suffix

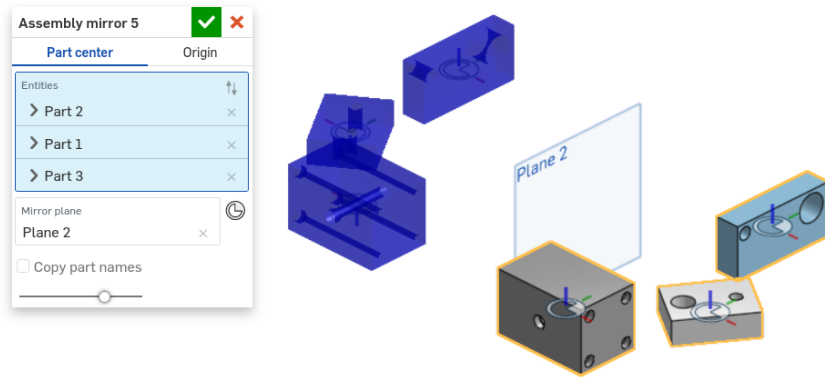
☐ Update saved names


3.3.2 Steps for creating mirrored parts using preselections

1. Before creating an Assembly mirror feature:
 - Pre-select a **Mirror plane** and one or more **Entities** to mirror by either:
 - Selecting parts and planes directly from the *feature tree* or *parts list*
 - Selecting geometry (edges, faces, or vertices) in the *graphics window* which touch the entities you'd like to use



2. Create an Assembly mirror feature by selecting it from your *FeatureScript dropdown*.
3. The **Entities** and **Mirror plane** parameters are filled in automatically according to your preselections.



4. Specify additional parameters as needed.
5. Click .

Tip: Several native Onshape features also support preselections. For example, the Onshape Plane feature will automatically switch to **Line angle** mode if you select a plane and a line prior to creating the feature.

3.3.3 Result behavior

Assembly mirror offers two options for locating mirrored parts; **Part center** and **Origin**.

- **Part center:** Mate connectors are created such that entity mirrors are aligned when their center mate connectors are fastened together.
- **Origin:** Mate connectors are located relative to the world origin, so mirrors are aligned when their mate connectors are fastened to the part studio origin. To assist with this, care should be taken that the assembly origin matches the part studio origin.


Tip: When using **Origin**, make assembling faster and more robust by creating a mate connector at the assembly origin and selecting it via the Mate Features list.

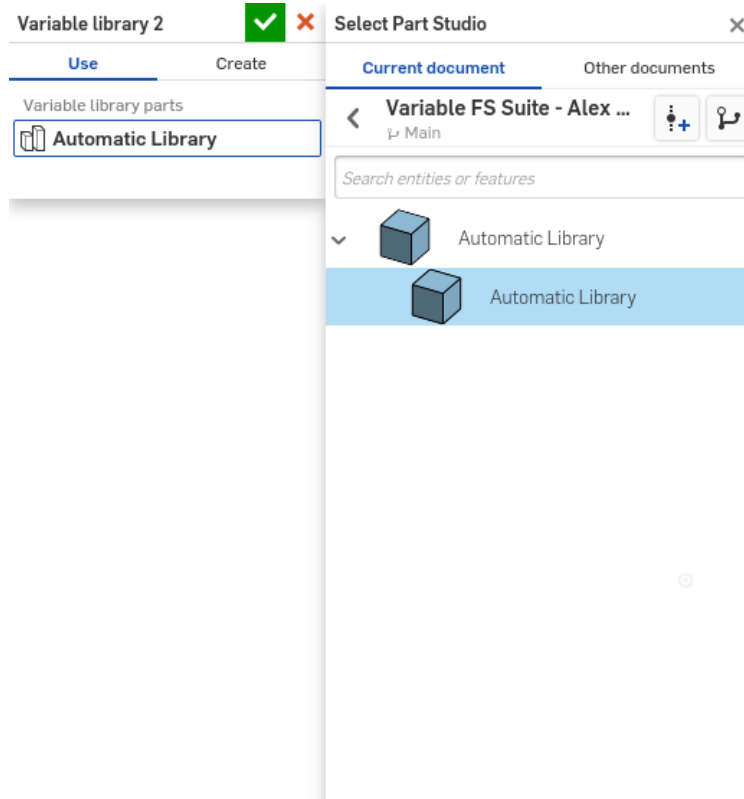
3.4 Variable library

The Variable library feature makes it possible to create and use global variables in Onshape by storing variables on cube parts. Variables may also have associated descriptions which can be viewed via a custom table.

The Variable library FeatureScript can be found here: [Variable library suite document](#)

3.4.1 Steps for using variable libraries

1. Create a Variable library feature in a part studio.
2. Set the Variable library usage behavior to **Use**.
3. Click **Select Part Studio...**, then navigate to another part studio containing one or more library variable cubes and select them.
4. The variables attached to those parts are automatically added to the part studio.
5. Click .



3.4.2 Steps for creating variable libraries manually

1. Create a Variable Library feature in a part studio.
2. Set the Variable library usage behavior to **Create**.
3. Set the library creation type to **Manual**.
4. Add one or more variables. A library cube part is created containing each added variable.
5. If desired, choose **Add descriptions**, then add descriptions to each variable in **Variables**.

See also:

[Variable descriptions](#) for more information.

1. Click .

Variable library 1

✓

✗

Use

Create

Automatic

Manual

Import

Library name

Manual Library

...

☒ Add descriptions

Variables

Variable

Length

Angle

Number

Any

Hole

Variable name

height

Value

3 in

Description

The height of ...

Variable

Length

Angle

Number

Any

Hole

Variable name

width

Value

10 in

Description

The width of s ...

Variable

Length

Angle

Number

Any


Hole

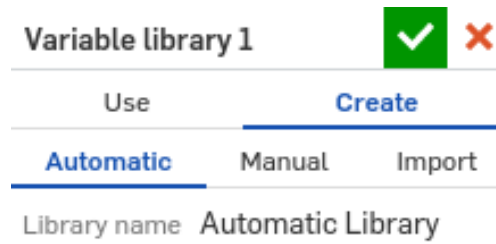
Variable name

depth

3.4.3 Steps for creating variable libraries from existing variables

Note: It isn't currently possible to add custom variable descriptions when creating variable libraries this way.

1. Create a Variable library feature in a part studio with one or more existing variables.
2. Set the Variable library usage behavior to **Create**.
3. Set the library creation type to **Automatic**.
4. Specify a **Library name**.
5. A library cube part is created containing all of the variables currently visible to the variable library feature.
6. Click .



3.4.4 Steps for creating variable libraries from CSV files

1. Open the Variable Library Template, make a copy, and then edit the template as desired.

The Variable Library Template can be found here: [Variable Library Template](#)

A	B	C	D	E	F
	Name	Value	Units	Description	
	Variable Library 1				
	variable_1	1	in	A length variable.	
	variable_2	15	mm	Another length variable.	
	variable_3	30	deg	An angle variable.	
	Variable Library 2				
	variable_4	2	in	Description.	
	variable_5	4	in	Description.	
	variable_6	8	in	Description.	
	variable_7	16	in	Description.	

Note: Keep in mind the following while adding variables: * An arbitrary amount of libraries may be created. * Variable names must start with a single letter, followed by letters, numbers, and underscores (_). Spaces and other special characters are not allowed. * Variable units must parse to a valid Onshape unit abbreviation like in, inch, deg, or degree. Punctuation and capitalization is ignored.

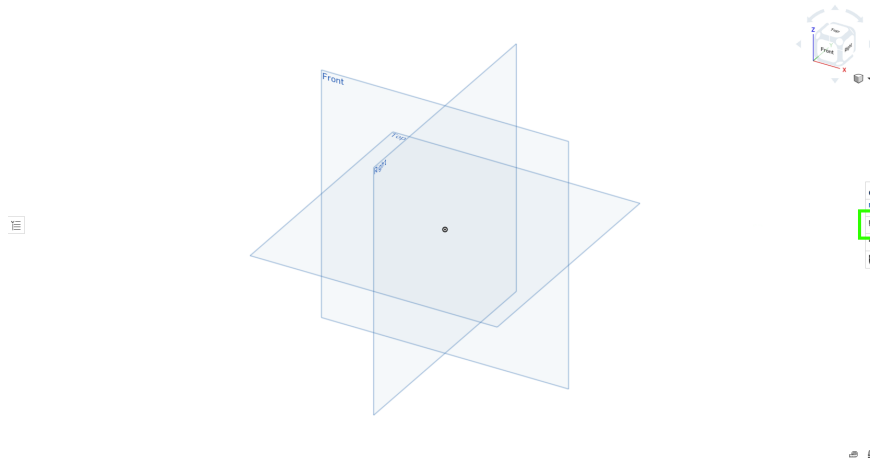
1. After you've finished editing, download the template file as a .csv (Comma Separated Values) file, then upload it into Onshape.
1. Create a Variable library feature in a part studio in the same document as your uploaded CSV file.
- Set the Variable library usage behavior to **Create**.
- Set the library creation type to **Import**.
- Select the **CSV file** you uploaded to Onshape.
- If your CSV file has a top-level header row, select **Skip first row**.
- The variable library feature should add one or more variable library cube parts to the part studio.
- Click

3.4.5 Variable descriptions

Descriptions can be added to manually created variables and to variables imported from CSV files. These descriptions can then be seen later on using a custom table.

To add and use the variable library custom table, you will need to first add it to your custom table tab.

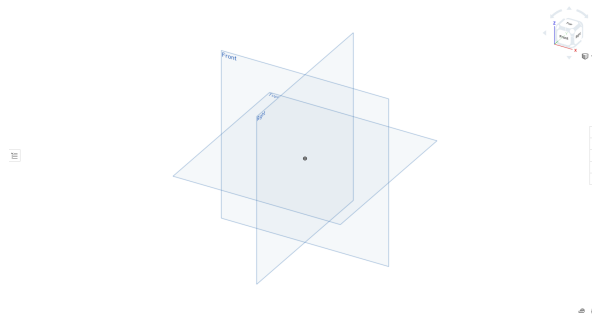
1. Open the **Custom tables** panel by clicking on the corresponding button on the far right side of your screen.



1. Choose **Add custom tables**, then copy and paste the following link into the search bar:
<https://cad.onshape.com/documents/d33530b06909083111cf1770/w/5be0429e68b3332e158a3559/e/2ac1b130ef1d31461f8fca54>
2. Open the resulting document and click **Library variables** to add it to your custom tables.
3. Variables in your part studio which have been created by the library variable feature are displayed in a table format (along with their description, if added).

GLOSSARY

Graphics window The graphics window refers to the model window, where parts, sketches, and other entities can be seen and interacted with.



Editing logic Editing logic is a feature of some FeatureScripts which enables certain parameters to update dynamically as a user is creating or making changes to a feature. For example, creating an extrude which hits another part will result in the boolean operation of the extrude feature being set to **Add**, and the other part being added to the **Merge scope**.

Feature A feature is a modeling tool which exists in part studios. Features are listed in the Feature list on the left side of your screen, and can represent an instance of a custom FeatureScript feature or a native Onshape feature (like the Onshape extrude feature).

Feature dialog The term feature dialog refers to the menu that is shown when creating or editing a feature.

Feature tree The list of features in a part studio which can be found on the left side of your screen.

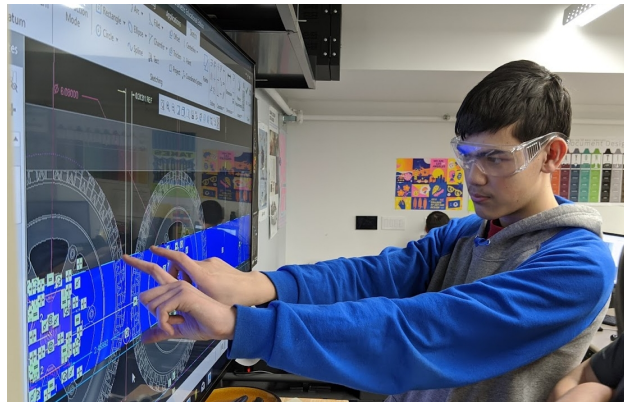
Parts list The list of all parts in a part studio. The parts list exists in the bottom left of your screen, underneath the feature tree.

FeatureScript dropdown Your FeatureScript dropdown is a section of your part studio toolbar which is reserved for custom features you've added there.

Manipulators Manipulators are a feature of some FeatureScripts which enable users to modify the behavior of features via selectable arrows and points in the graphics window.

Preselections Preselections are a feature of some FeatureScripts which allow certain entities to be selected before a feature is created. For example, a plane and a line can be selected before creating a Construction plane feature to change the plane creation mode to **Line angle** automatically.

ABOUT ME



Hello! My name is Alex Kempen. I'm a sophomore college student at the University of Texas at Dallas majoring in Computer Science.

I'm also a mentor of FRC Team 1745. If you ever have a competition with us, feel free to stop by and say hi.

If you have any questions, comments, concerns, or feedback about any of my FeatureScripts or other work, feel free to reach out to me through one of the following methods.

Email: Alex.Kempen@utdallas.edu

Onshape Forums: Alex_Kempen

Chief Delphi: VenomousFire

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