AutoDesk Fusion 360

Table of Contents

1.1: Introduction to Fusion 360

Fusion 360 is a cloud-based CAD/CAM tool for collaborative product development. The tools in Fusion enable exploration and iteration on product ideas and collaboration within a product development team.

Fusion 360 enables fast and easy exploration of design ideas with an integrated concept to production toolset. Fusion lets you focus on the form, function, and fabrication of your products. Use the sculpting tools to explore form and modeling tools to create finishing features. These tools let you quickly iterate on design ideas. Once you have settled on a design, you can create assemblies to validate fit and motion in your design or create photo-realistic renderings to verify the appearance. Finally, you need to fabricate your design. Use the 3D print workflows to create a rapid prototype or the CAM workspace to create tool-paths to machine your components.

Fusion 360 also helps bring design teams together for collaborative product development. All your designs are stored in the cloud, which means you and your team always access the latest data. Fusion also tracks versions of your design as you work. You can use Autodesk A360 to view each version in your web browser and promote an old version to the current version. Finally, use Fusion and A360 to share your designs and track design activity. You can even provide controlled access to your designs without requiring an Autodesk ID.

Fusion 360 uses a hybrid environment that harnesses the power of the cloud when necessary and uses local resources when it makes sense. For example, your design data is stored on the cloud and renders amazing images every time you save a new version of your design. This happens in parallel while you are creating and editing designs locally on your machine. This allows you to harness the power of your computer and the power of the cloud at the same time.

Throughout this course, you explore these areas of Fusion 360. This course gets you started designing with Fusion and helps you understand how it can improve your design processes.

1.2: Preferences

Preferences control default settings in Fusion. The Preferences dialog contains many pages of settings. Any changes you make to the preferences are saved with your Autodesk ID and are loaded when you log into another machine.
Some important preferences to review:

- General: general settings such as versioning (saving), pan, zoom, and orbit.
- General > Design: settings for the design workspaces: model, sculpt, patch.
- General > Drawing: settings for the creation of drawings.
- Material: controls the default physical material and appearance.
- Unit and Value Display: sets the precision and display of units.
- Default Units > Design and CAM: sets the default unit type.

Lesson 1: Setting Your Preferences

Learning Objectives

1. Access preferences
2. Modify preferences settings

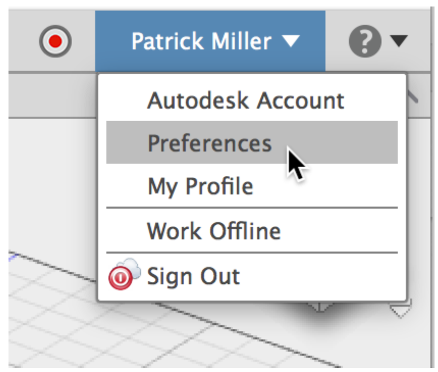
Datasets Required

No dataset are required. You start with a new empty design.

Step-by-step Guides

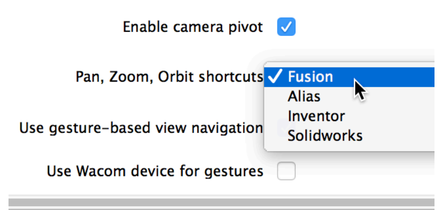
Step 1: Access the preferences dialog box.

1. Click on your name in the upper right corner then select Preferences.



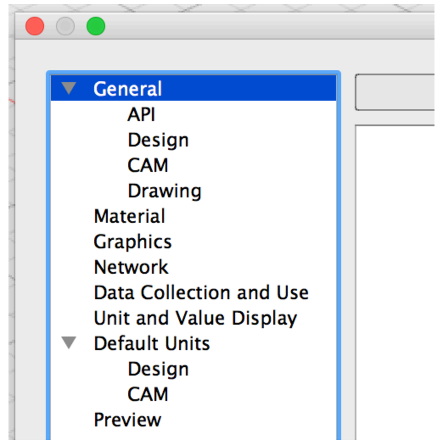
Step 2: Change the General settings.

1. Click General in the preferences list.
2. Scroll to Pan, Zoom, Orbit shortcuts.
3. Select the CAD application you are most comfortable with. This changes the mouse behavior for pan, zoom, and orbit.
4. Scroll through the other options in the General page.



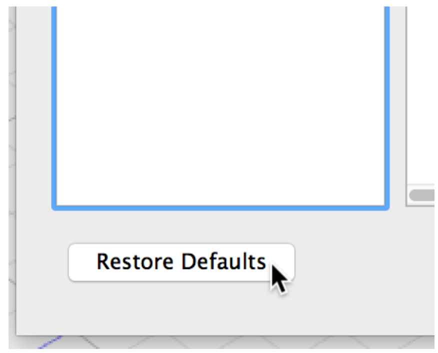
Step 3: – Review the other pages.

1. The Preferences dialog contains multiple pages. Click through the other pages to review the available settings.



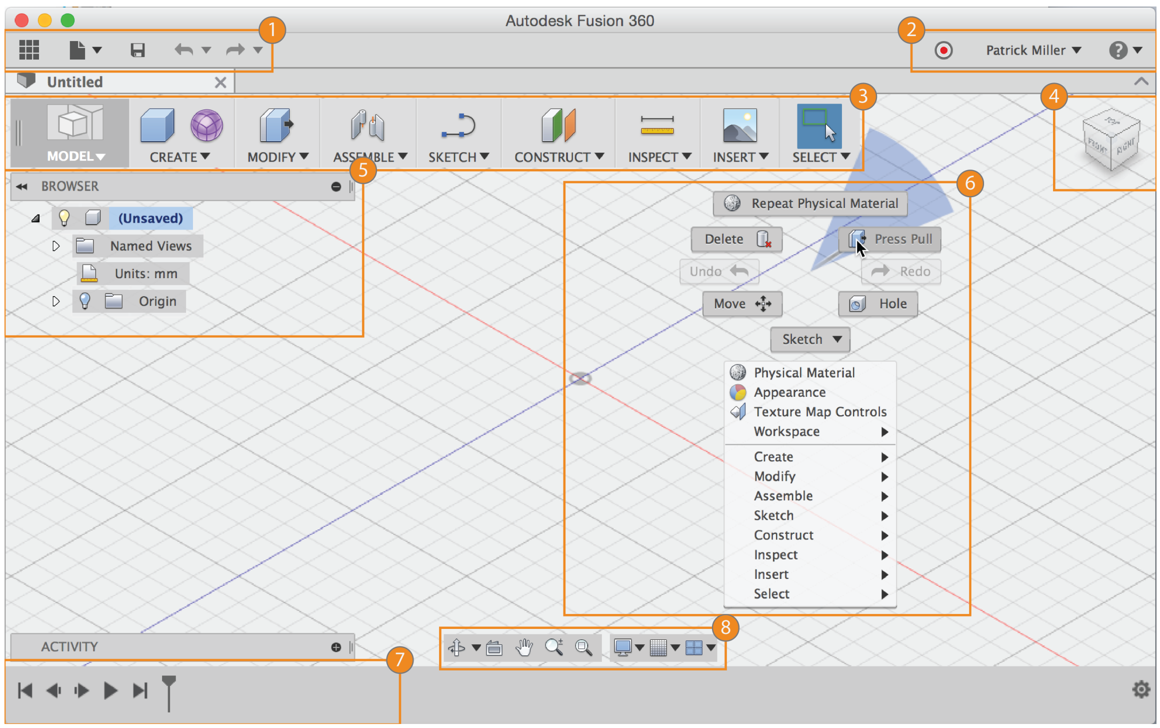
Step 4: – Reset the preferences.

1. If desired, click Restore Defaults in the lower left of the Preferences dialog. This resets all settings to the installed settings.



1.3: User Interface

The UI can be broken up into 8 areas. To help you become familiar with the product we will describe each of these areas and go into more details with how to use it in the following lessons.



1 Application bar

The Application bar is where you'll find and use the following:

	Data Panel – Used for data management and collaboration.
	File – Create a New Design, Save, Export, and 3D Print.
	Save – Save an untitled design or save the changes to a design as a new version.
	Undo/redo – Undo/redo operations.

2 Profile and help

In profile you can control your profile and account settings, or use the help menu to continue your learning or get help in troubleshooting.

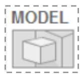
	Profile – In your profile you can access your own personal settings.
	Help – In the help menu you can access online learning content, help, forums, step-by-step tutorials, or link to community content.

3 Toolbar

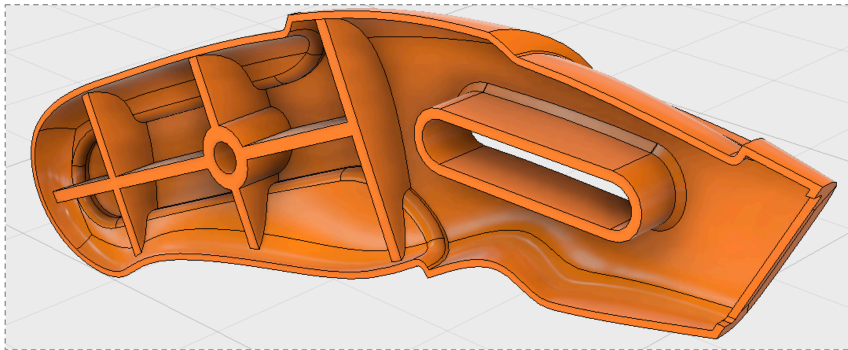
Use the Tool bar to select the workspace you want to work in, and the tool you want to use in the workspace selected.

Your Workspaces

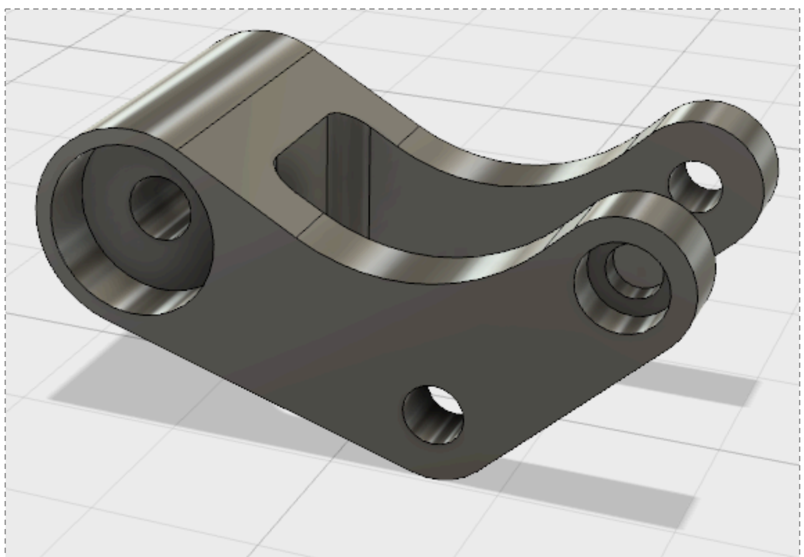
Fusion 360 uses these workspaces to control the commands that are available and the type of data that is created.

	The Sculpt workspace is used to create organic shapes by manipulating faces, edges, and vertices.
	The Model workspace is used to create solids with hard edges and flat faces.
	The Patch workspace is used to create open surfaces to stitch into solid bodies.
	The Render workspace is used to set up the environment and create photo-realistic renderings.
	The CAM workspace is used to create and simulate tool-paths then generate g code for subtractive manufacturing.
	The Animation workspace is used for to create exploded views of an assembly and control over unique animations of parts and assemblies.
	The Drawing workspace is used to generate 2D manufacturing drawings.

> *Note:*
>
> *Very frequently, your designs will require that you work in **both sculpt and model workspaces**, back and forth. You might even throw **patch in there to stitch surfaces** together into a solid. Create the organic shape in sculpt then use model for manufacturing features afterwards.*



Note:
*Use **model** to create designs with hard edges and flat faces. Model creates bodies requiring exact sizes and edges. Entering exact values is not required but is typical.*



4 ViewCube

Use the ViewCube to orbit your design or view the design from standard view positions.

5 Browser

The browser lists objects in your design. Use the browser to make changes to objects

	and control visibility of objects.
6	**Canvas and marking menu** Left click to select objects in the canvas. Right-click to access the marking menu. The marking menu contains frequently used commands in the wheel and all commands in the overflow menu.
7	**Timeline** The timeline lists operations performed on your design. Right-click operations in the timeline to make changes. Drag operations to change the order they are calculated.
8	**Navigation bar and display settings** The navigation bar contains commands used to zoom, pan, and orbit your design. The display settings control the appearance of the interface and how designs are displayed in canvas.

Lesson 1: User Interface Overview

Learning Objectives

1. Familiarize yourself with the UI by creating a simple design
2. Use the toolbar
3. Use the marking menu
4. Control objects in the browser
5. Control operations in the timeline
6. Change workspaces

Datasets Required

No dataset required. You start with a new empty design.

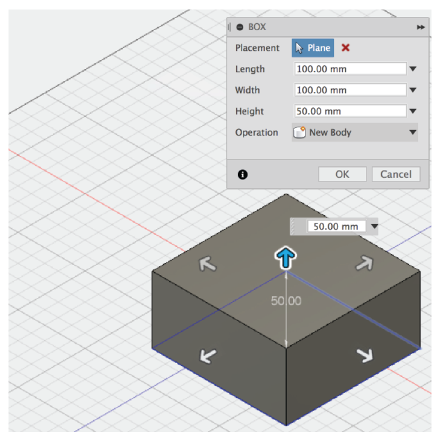
Step-by-step Guides

Step 1: Create a new design

1. Click  then select **New Design**.

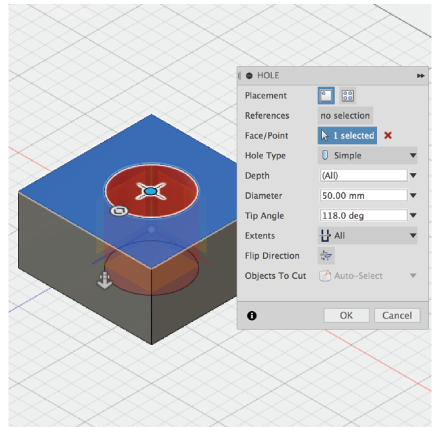
Step 2: Create a box

1. Click Model > Create > Box to start the box command.
2. Select the XZ Plane along the bottom of the canvas.
3. Pick two points to define the length and width of the box.
4. In the Box dialog, use these values:
   Length: **100 mm**
   Width: **100 mm**
   Height: **50 mm**
5. Click **OK**.



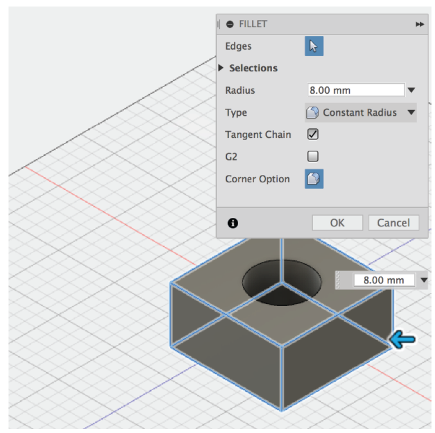
Step 3: Add a hole to the box

1. Click Model > Create > Hole.
2. Select the top face of the box.
3. Drag the center of the hole to the center of the box.
4. Set Diameter to a value of **50 mm**.
5. Change Extents to **All**.
6. Click **OK**.



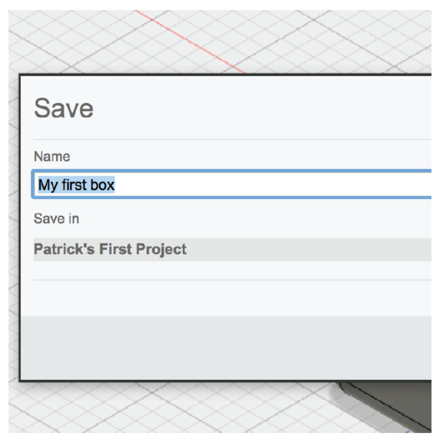
Step 4: Round the edges of the box

1. Right-click an empty area in the canvas then select **Press Pull**.
2. Hold the left mouse button then drag to window select the entire box.
3. Click the top and bottom edges of the hole to deselect the edges.
4. Enter **8 mm** for the Radius.
5. Right-click then select **OK**.



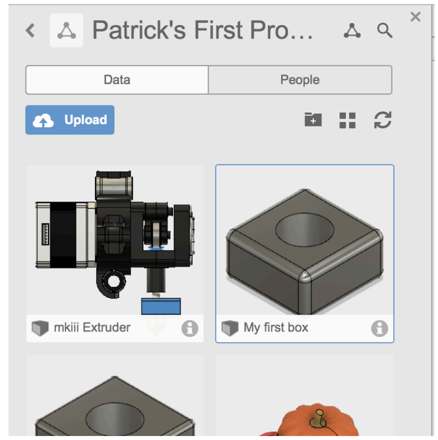
Step 5: Save your design

1. Click  to save the design.
2. Enter **My first box** in the Name field.
3. Set Save in to **<your name>'s First Project**.
4. Click **Save**.



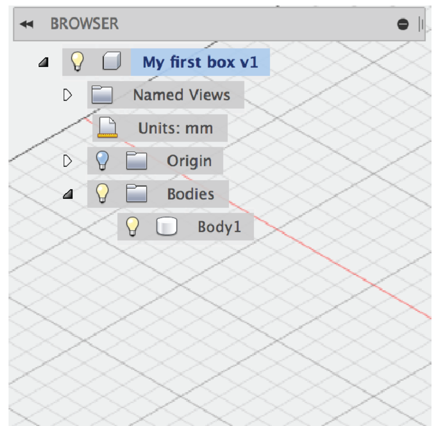
Step 6: Display the Data Panel

1. Click  in the upper left corner to display the Data Panel.
2. The active project name is displayed at the top. Thumbnails of all the designs in the project are listed. All data is stored in A360 in the cloud.
3. Click  again to hide the Data Panel.



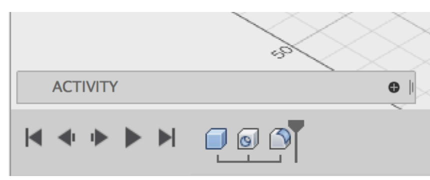
Step 7: Use the browser

4. Click  next to Origin to display the origin planes.
5. Click  the light bulb again to turn the origin planes off.
6. Click  next to Bodies in the browser to expand the folder. There is one body in this design.



Step 8: Use the timeline

1. Click  to replay the operations in the design.
2. Right-click the  fillet operation in the timeline.
3. Change the Radius to **5 mm** and click **OK**.

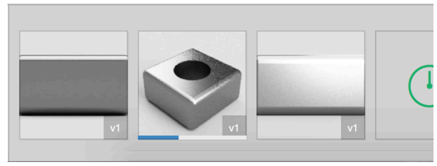


Step 9: Change workspaces

1. Select 

then select 

to switch to the render workspace. Notice the canvas appearance changes and the timeline is replaced with the Rendering Gallery. This workspace is used to render images of your design.



2. Notice the Rendering Gallery at the bottom of the interface. This gallery displays a thumbnail of your cloud renderings and shows the progress of renderings that are in process.
3. If the renderings have been processed, click on of the thumbnails to display the image.
4. Close the Cloud Rendering dialog.
5. Select  then select  to return to the model workspace.
6. Keep the design open. You will use it in the next lesson.

1.4: Navigation

There are three ways to manipulate the view of your design:

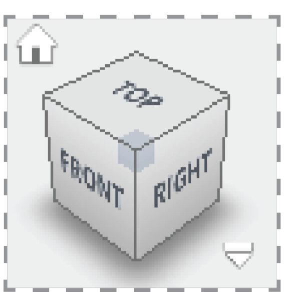
- Navigation bar
- ViewCube
- Wheel button on a mouse

Navigation bar

Use the commands in the Navigation bar to pan, zoom, and orbit the canvas. The menus on the right end control Display Settings and Layout Grid options.

ViewCube

Use the ViewCube to orbit the design in the canvas. Drag the ViewCube to perform a free orbit. Click faces and corners of the cube to access standard orthographic and isometric views.

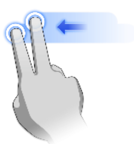
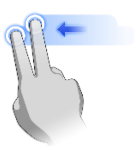


Mouse

Use mouse shortcuts to zoom in/out, pan the view, and orbit the view.

	**Scroll middle mouse button** to **zoom in or zoom out**.
	**Click and hold middle mouse button** to **pan the view**.
SHIFT + 	**Shift Key + middle mouse button** to **orbit the view**.

Mac Trackpad

	Use the **2 finger pinch** to **zoom out**.
	Use the **2 finger spread** to **zoom in**.
	Use the **2 finger swipe** to **pan the view**
	**Hold Shift + the 2 finger swipe** to **orbit the view**

Lesson 1: Navigate the canvas

Learning Objectives

1. Use the commands in the Navigation Bar
2. Use the mouse to zoom and orbit the design

3. Use the ViewCube to navigate the design

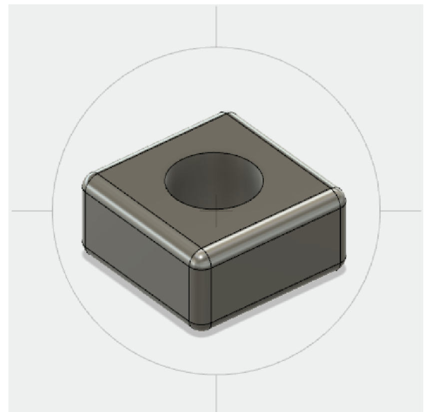
Datasets Required

Use the design from the previous lesson.

Step-by-step Guides

Step 1: Use the Navigation Bar

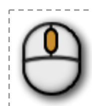
1. Click  Orbit then drag within the circle.
2. Click  Pan then drag in the canvas to pan.
3. Click  Zoom then drag up and down in the canvas to zoom in and zoom out.



Step 2: Use the mouse

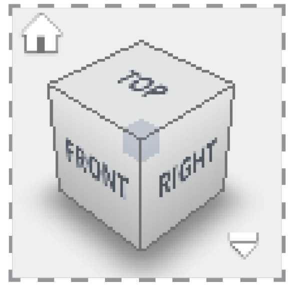
1. Roll the wheel forward and backward to zoom in and zoom out.
2. Click and hold then drag to pan.
3. Hold the Shift key then drag with the middle mouse button to orbit the design.
4. Double-click the middle mouse button to zoom extents.

Note: if you changed the Pan, Zoom, Orbit shortcuts preference then your mouse wheel will behave differently.



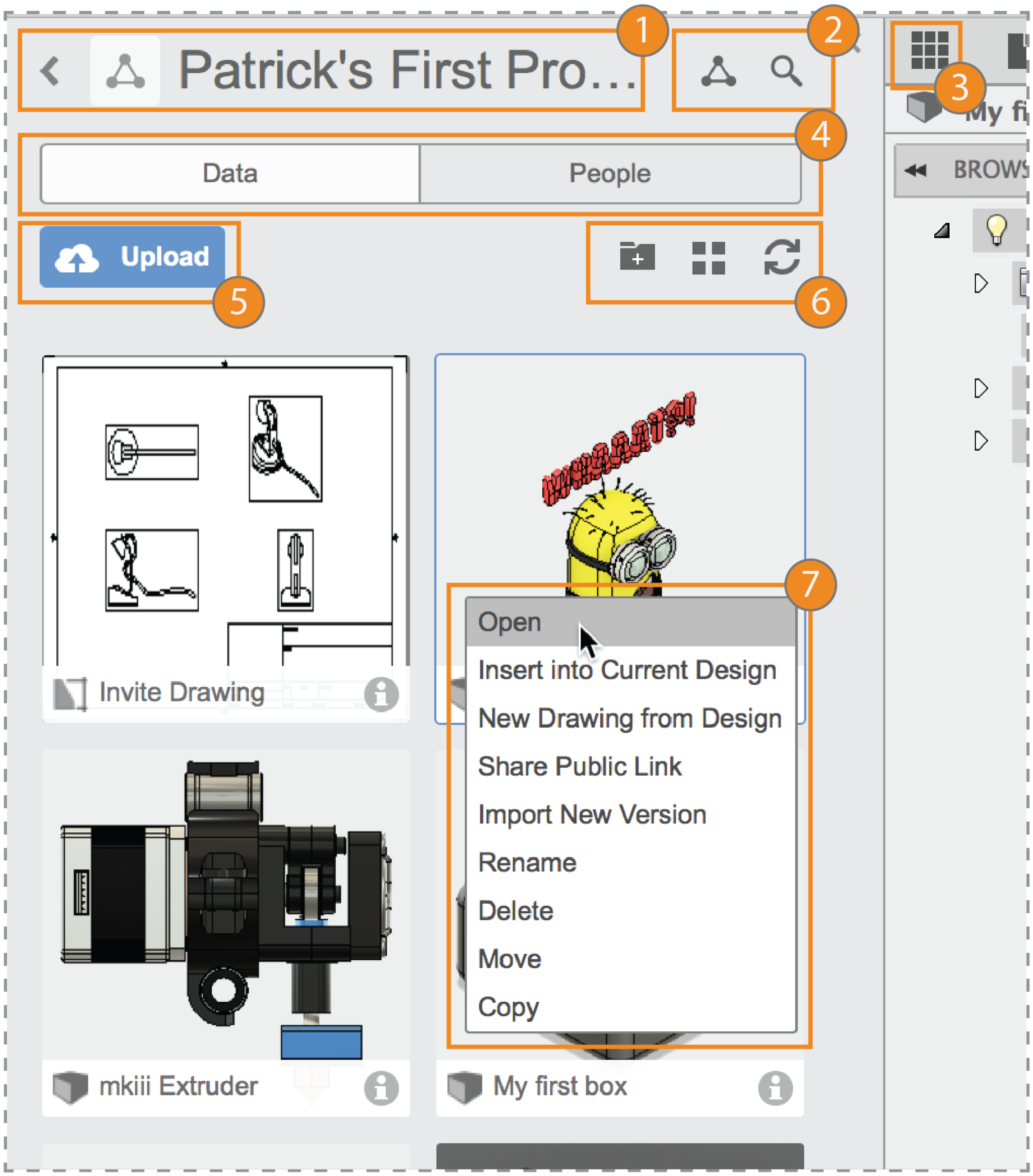
Step 3: Use the ViewCube

1. Left click and drag the ViewCube to orbit the design.
2. Click one of the corners of the cube to go to an isometric view.
3. Click on the FRONT face to go to the front orthographic view.
4. Click  Home to return to the home view.



1.5: Data Panel Interface

Use the Data Panel on the left of the application to access your designs and manage projects.



1 **Project selector**

Displays the name of the active project. Click the back arrow to display a project list.

2 **Project tools**

Displays project data in Autodesk A360 or searches the active project.

Project Details – Opens the active project in Autodesk A360 in your default internet browser.

Search – Search the active project or all projects you have access to.

3 Data Panel show/hide

Click to show or hide the Data Panel.

4 Data or People

Controls the display of data or users in the Data Panel.

5 Upload

Click to upload data.

6 View tools

Create a new folder or change the display of items in the Data Panel.

New Folder – Create a new folder in the active project.

View Options – Choose how to sort and list data.

Refresh – Refresh the data in the Data Panel.

7 Thumbnails

Right-click a thumbnail to access commands for that specific design.

Lesson 1: Manage your design

Learning Objectives

1. Open a design
2. Change the active project

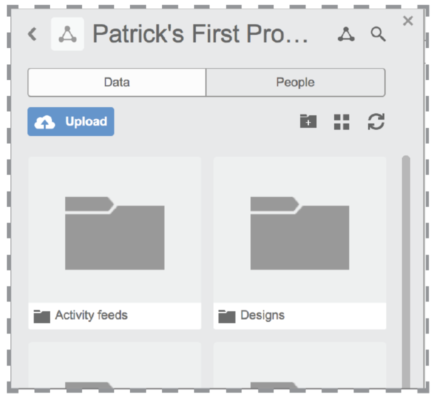
Datasets Required

Use the design from the previous lesson.

Step-by-step Guides

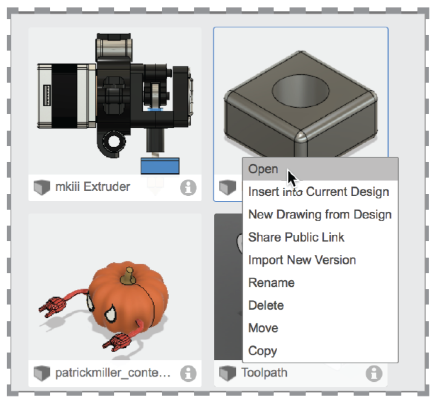
Step 1: Set the active project

1. Click the back button next to the active project in the Data Panel.
2. Double-click **<your name>'s First Project** to make that the active project.



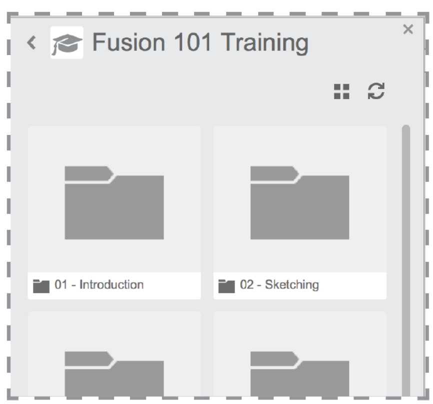
Step 2: Open a design

1. Click the **X** on the tab for My first box. The design is closed and you are presented with an empty design.
2. In the Data Panel, right-click on My first box then select **Open**.



Step 3: Access the Fusion 101 Training project

1. Click the back button next to the active project in the Data Panel.
2. Scroll to the **SAMPLES** category.
3. Double-click Fusion 101 Training to set it as the active project. This project contains the data you will use for the other lessons in this course.



Keyboard Shortcuts

Command	**Windows**	**Mac**
Undo	Ctrl + Z	Command + Z
Redo	Ctrl + Y	Command + Y
Copy	Ctrl + C	Command + C
Paste	Ctrl + V	Command + V
Cut	Ctrl + X	Command + X

Sculpt Workspace Selection	**Windows**	**Mac**
Grow selection	Shift + Up arrow	Shift + Up arrow
Shrink selection	Shift + Down arrow	Shift + Down arrow
Loop selection	Alt + P	Control + P
Loop grow selection	Alt + O	Control + O
Ring selection	Alt + L	Control + L
Ring grow selection	Alt + K	Control + K
Ring shrink selection	Alt + J	Control + J
Previous U	Alt + Left arrow	Control + Command + Left arrow
Next U	Alt + Right arrow	Control + Command + Right arrow
Previous V	Alt + Down arrow	Control + Command + Down arrow
Next V	Alt + Up arrow	Control + Command + Up arrow
Range selection	Alt + M	Command + M
Invert selection	Alt + N	Command + N
Toggle box mode	Ctrl + 1	Ctrl + 1
Toggle control frame mode	Ctrl + 2	Ctrl + 2
Toggle smooth mode	Ctrl + 3	Ctrl + 3
Select edge ring	Double-click an edge	Double-click an edge
Select face ring	Select two faces then double-click a third face	Select two faces then double-click a third face

Edit Form Command	**Windows**	**Mac**
Add geometry	Alt + Drag	Option + Drag
Add geometry and keep creases	Alt + Ctrl + Drag	Option + Command + Drag

2.1: Sketching

Many features that you create in Fusion 360 start with a 2D sketch. In order to create intelligent and predictable designs, a good understanding of how to create sketches and how to apply dimensions and geometric constraints. Fusion does support 3D sketches although, in this module we will cover basic sketching tools to create and edit a 2D sketch. In this lesson we will be building the housing for a hypocycloidal gearbox. ~~–~~Visit robotarm.org to learn more about it.

Lesson 1: Creating a sketch

Learning Objectives

1. Create a 2D sketch
2. Create geometry in a sketch
3. Use constraints to position geometry
4. Use dimensions to set the size of geometry

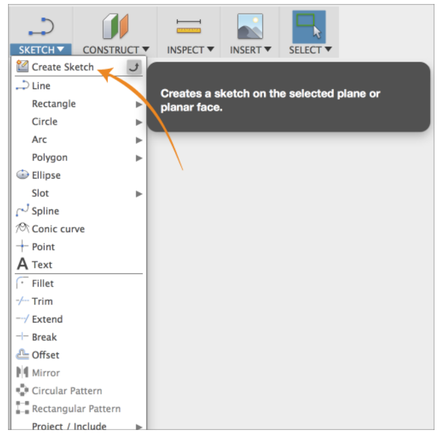
Datasets Required

In Samples section of your Data Panel, browse to:

Fusion 101 Training > 02 – Sketching > 02_Sketching

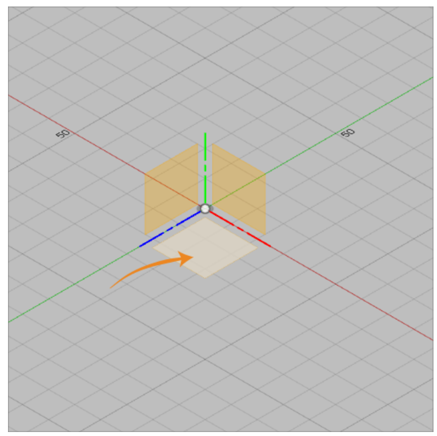
Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides



Step 1: – Start the **Sketch** command

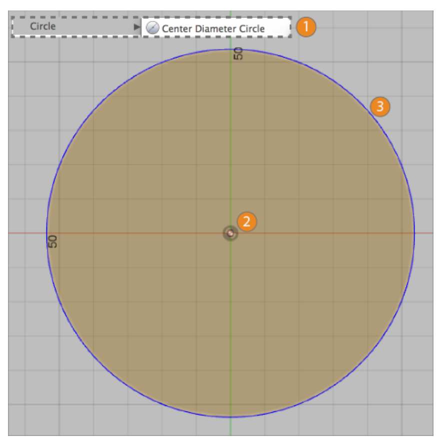
1. Select **Sketch** > **Create Sketch**.



Step 2: – Select the sketch plane

1. You are now prompted to select a "plane" to sketch on.
2. Select the "Front" (XY) plane.

Note: Aside from the origin planes, you can create sketches on one of the 3 default planes, on a custom construction plane or on an existing model face, more on this later.



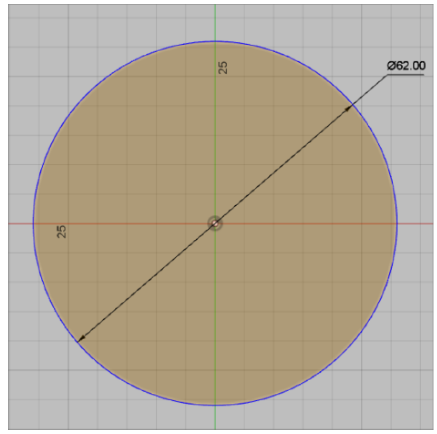
Step 3: – Create a circle

1. Select **Sketch > Circle > Center Diameter Circle**
2. Now hover over the **origin** (center) of the sketch. You should see the cursor "snap" to this location.
3. Click once to begin placing the circle.
4. Move the mouse away from the center to define the size.
5. Click again to place the circle.

Note: In Fusion 360 it is important to "snap" entities to the origin when possible. This accurately grounds the objects and ensures they will behave as expected.

Step 4: – Dimension the circle

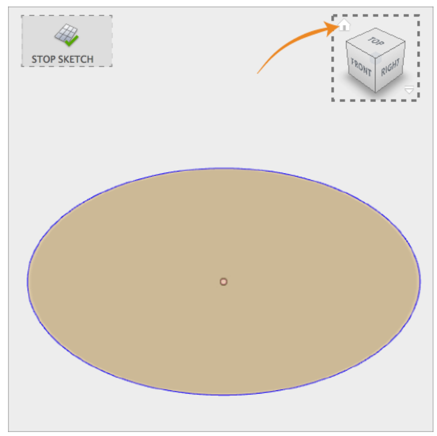
1. Select **Sketch > Sketch Dimension.**
2. Select the circle sketch.
3. Click again to place the dimension.
4. Type in a value of 62 mm.
5. Press **Enter** to accept the value.



Step 5: – Finish the **Sketch** command

1. **Finish Sketch** by clicking on Stop Sketch.
2. Select the home view icon to the let of the view cube.

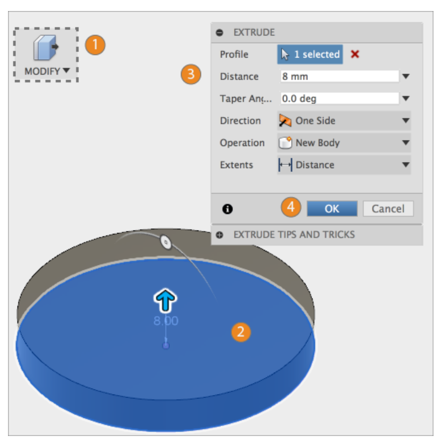
Note: You must "stop" a sketch before you can continue building geometry since Sketch is a mode that you enter and exit within workspaces.



Step 6: – Extrude the Circle

1. Select **Modify > Press Pull.**
2. Select the Profile.
3. Drag the arrow up or type in a value of **8 mm** to set the depth.
4. Press **OK to finish.**

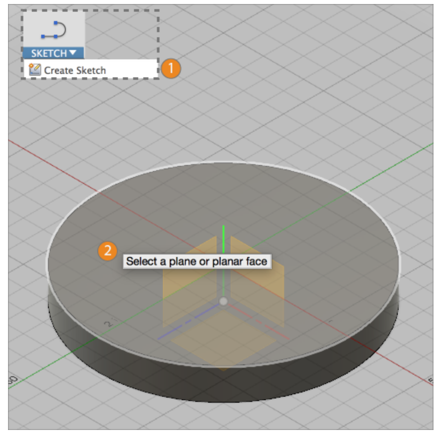
Note: In Fusion 360 you need to select the shaded area in the middle of the circle, NOT the edge of the circle to define profile for the extrude. Later we will look more closely at sketch profiles but for now simply select the shaded area.



Step 7: – Create another sketch

1. Select **Sketch > Create Sketch.**
2. Select the top of the cylinder.

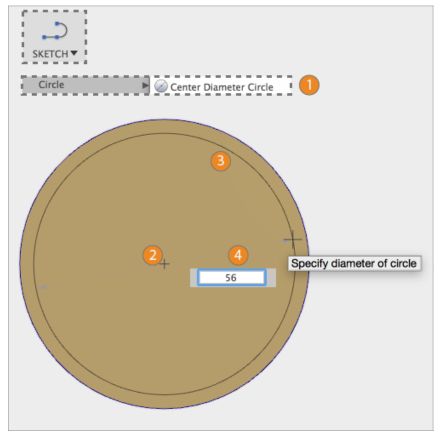
Note: This starts a new sketch on the top face of the cylinder. You will see the outer edge of the cylinder is already captured in the sketch.



Step 8: – Create an inner circle.

1. Select **Sketch > Circle > Center Diameter Circle.**
2. Select the center point of the top face.
3. Start dragging the circle.
4. Type a value of **56 mm.**
5. Press **Enter** twice.
6. Select **Stop Sketch.**

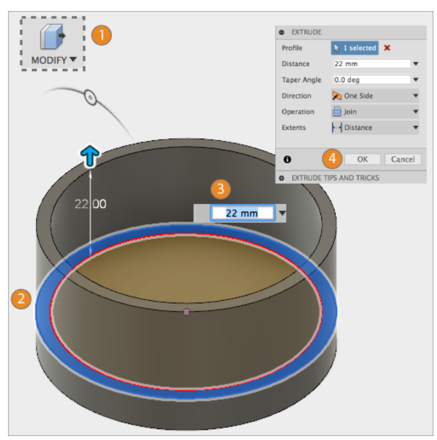
Note: We can also create the inner circle with the Sketch > Offset command. Click on the outer circle and offset it to 56 mm. If you type in a value when creating sketch geometry the dimension will be automatically added. Otherwise you can apply a dimension like in the previous step.



Step 9: – Extrude the inner circle

1. Select **Modify > Press Pull.**
2. Select the inner circle profile.
3. Drag the arrow up or type in a value of **22 mm** to set the depth.
4. Press **OK.**

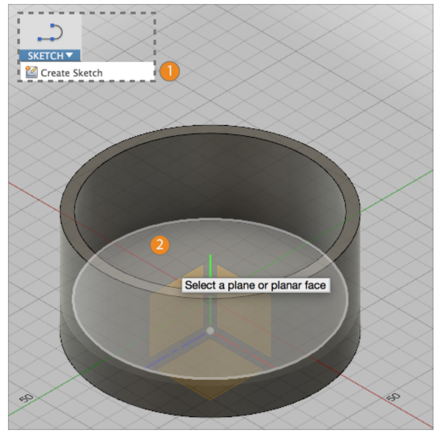
Note: One sketch may contain multiple profiles. In this case there is a profile inside the circle you drew and a profile between the outer edge and the inner circle. You can select multiple profiles for an extrude feature. In this case you will only select the outer profile.



Step 10: – Create another sketch

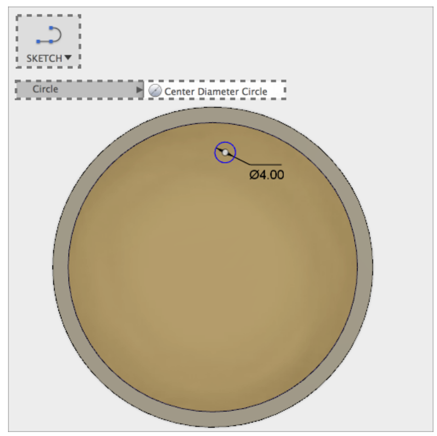
We're now going to start creating smaller circle profiles for the holes so we can pattern them later.

1. Select **Sketch > Create Sketch.**
2. Select the top of the cylinder.



Step 11: – Create a circle

1. Select **Sketch > Circle > Center Diameter Circle.**
2. Select a point in the top part of the inner circle.
3. Start dragging the circle.
4. Type a value of **4 mm.**
5. Press **Enter** twice to accept the value.



Step 12: – Constrain the Circle

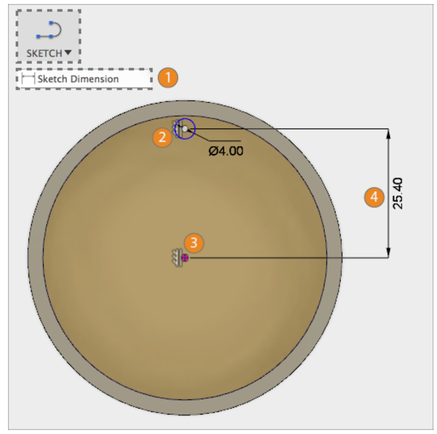
1. Select **Horizontal/Vertical** constraint from the Sketch Palette.
2. Select the center point of the new circle and the center point of face.
3. Press **esc** to exit the command.

Note: Constraints create relationships in your sketch. By saying that these two points are "vertical" determines how the will be aligned in the sketch. These relationships are persistent, meaning that if the center of the face moves, the sketch will move also.

Step 13: – Dimension the circle

1. Select **Sketch > Sketch Dimension.**
2. Select the center of the circle.
3. Select the center of the face.
4. Click to place the dimension.
5. Type in a value of 25.4 mm.
6. Press **Enter** to accept the value.

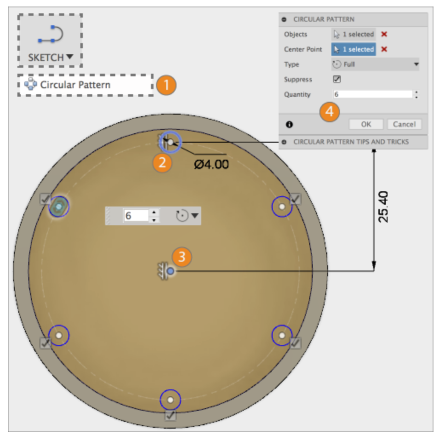
Note: This now has "fully" constrained the circle. This means that the geometry is fully locked down.



Step 14: – Pattern the Circle

1. Select **Sketch > Circular Pattern.**
2. In **Objects,** select the circle.
3. In **Center Point**, select the face center.
4. Change the **Quantity** to **6.**
5. Press **OK.**
6. Select **Stop Sketch** to end the sketch mode and return back into the modeling environment**.**

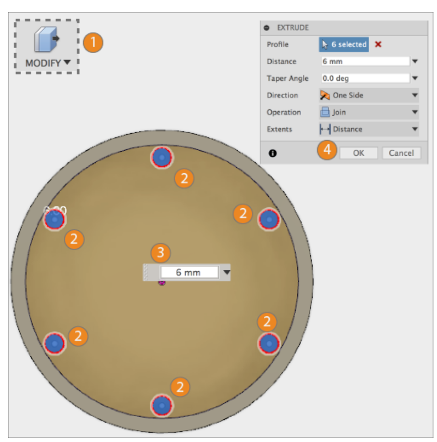
Note: Make sure you initiate what you want to select by click on the "no selection" box before trying to pick the center point for the pattern. Each box should then say "1 Selected." You can press the red "X" next to the box if you want to clear the current selections.



Step 15: – Extrude the Circles.

5. Select **Modify > Press Pull.**
6. Select the 6 Profiles.
7. Drag the arrow up or type in a value of **6 mm** to set the depth.
8. Press **OK.**

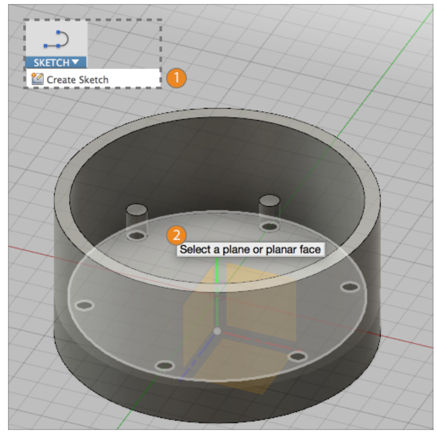
Note: It may help to zoom in a little bit to make sure you are grabbing the correct profiles. Make sure you have six profiles selected. If you need to add or remove profiles later hold ctrl (for Windows) or command (for Mac) and select the profile.



Step 16: – Create another sketch

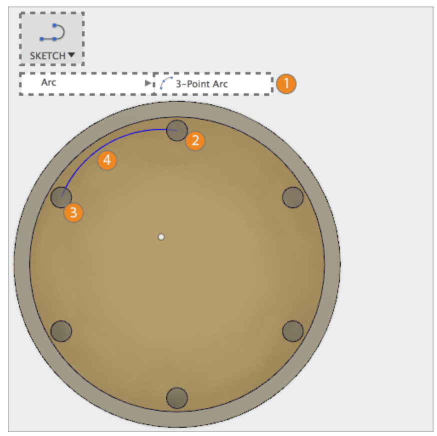
We're almost done with our design. We need to create the pockets for the round gear teeth.

3. Select **Sketch > Create Sketch.**
1. Select the top of the cylinder again.



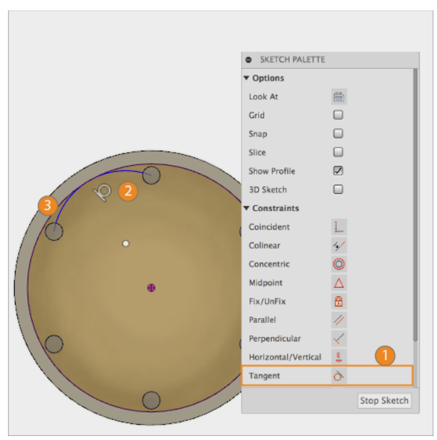
Step 17: – Create an arc

1. Select **Sketch > Arc > 3 Point Arc.**
2. Select the center of one circle.
3. Select an adjacent circle center point.
4. Click the third point near the edge.
5. Press **esc** to exit the command.



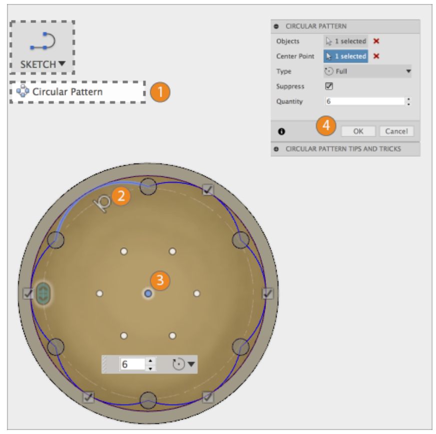
Step 18: – Constrain the Arc

1. Select the **Tangent** constraint from the Sketch Palette.
2. Select the new Arc and the inner edge so that they are constrained.
3. Press **esc** to exit the command.
4. Select **Stop Sketch** to return back to the modeling environment.



Step 19: – Pattern the Arc

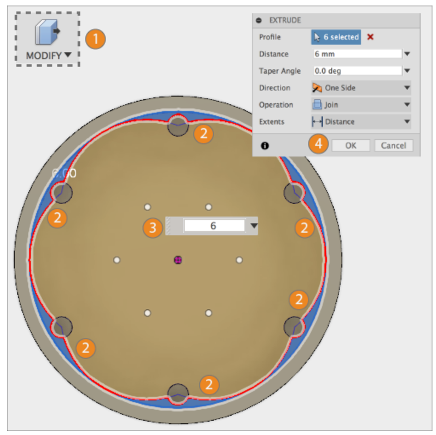
1. Select **Sketch > Circular Pattern.**
2. In **Objects,** select the arc.
3. In **Center Point**, select the face center.
4. Change the **Quantity** to **6.**
5. Press **OK.**
6. Select **Stop Sketch.**



Step 20: – Extrude the Arcs

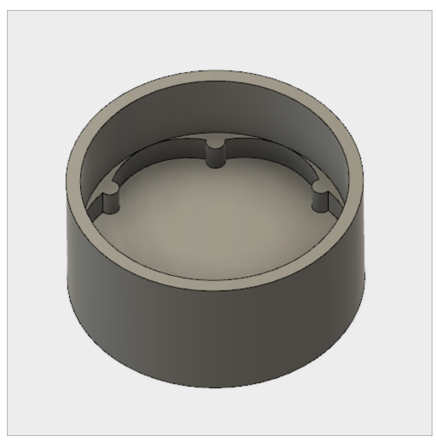
1. Select **Modify > Press Pull.**
2. Select the 6 Profiles.
3. Drag the arrow up or type in a value of **6 mm** to set the depth.
4. Press **OK.**

Note: Notice that by simply creating the arcs, the profiles are automatically created. The geometry we are extruding is the space between the existing geometry and the new arcs.



Step 21: – All Done

1. Congratulations you have finished this lesson and have learned the basics of how to create sketch geometry, dimension sketches, constrain sketches, as well as creating solid models based on sketches. Next we will look at more ways to define geometry in a sketch.



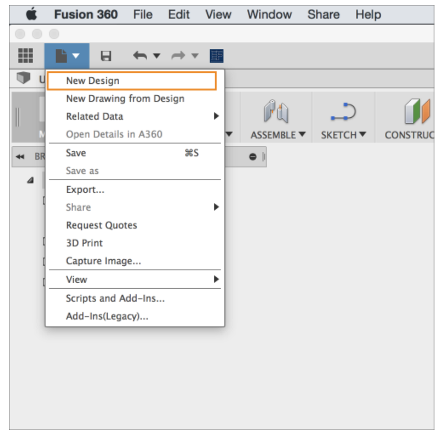
Lesson 2: Creating a sketch

Learning Objectives

1. Create a 2D sketch
2. Use construction geometry
3. More advanced use of constraints

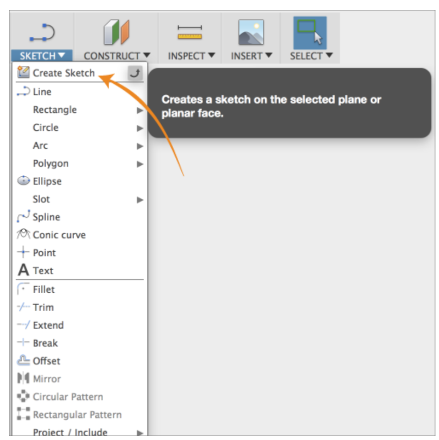
Step 1: Create new Design - Let’s start with creating a new design. We’re going to use this to create a new part.

1. Launch Fusion 360.
2. Start a new design.



Step 2: – Start the **Sketch** command

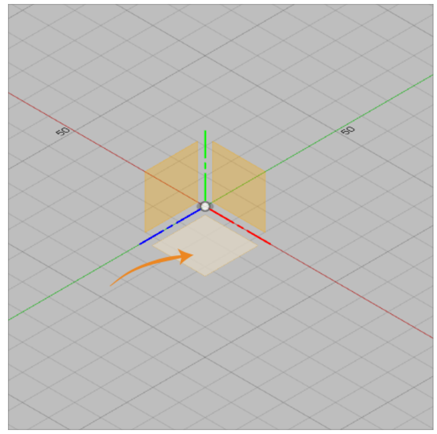
1. Select **Sketch** > **Create Sketch**.



Step 3: – Select the sketch plane

1. You are now prompted to select a "plane" to sketch on.
2. Select the "Front" (XY) plane.

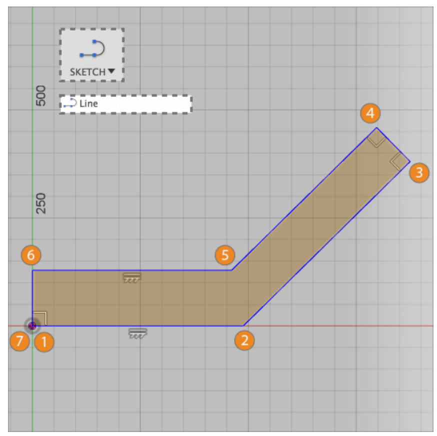
Note: Aside from the origin planes, you can create sketches on one of the 3 default planes, on a custom construction plane or on an existing model face, more on this later.



Step 4: – Create Lines

1. Select **Sketch > Line.**
2. Select the sketch origin.
3. Click to end the line
4. Continue sketching lines as follows
5. Press **esc** to exit the command

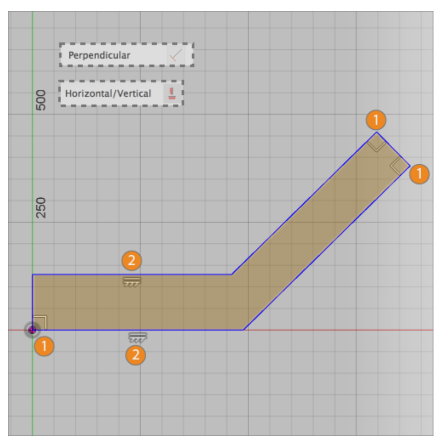
Note: Note as you place the lines some constraints are "automatically" created. If you do not get exactly the same ones don't worry. Also try to make sure your first line is roughly 500mm it is good practice to sketch shapes "close" to the correct size.



Step 5: – Create Constraints

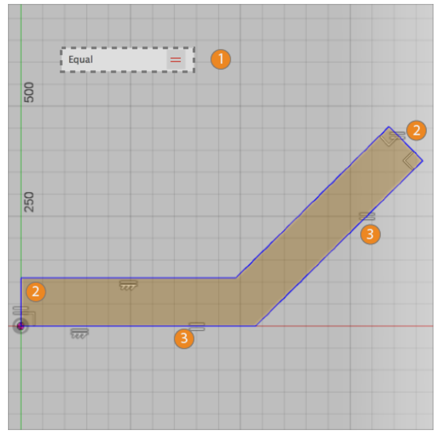
1. Select the **Perpendicular** constraint
2. Select two lines
3. Repeat 3 times (1)
4. Select **Horizontal/Vertical**
5. Select the two lower lines (2)
6. Press **esc** to exit the command

Note: The constraint commands are in the Sketch Palette on the right side of the screen. Some of these constraints may already be created. If they are don't bother recreating them.



Step 6: – Create Equal Constraint

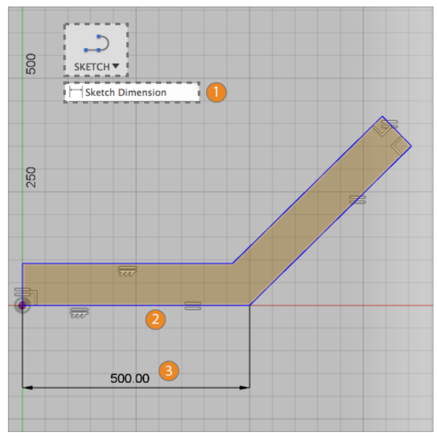
1. Select **Equal constraint**
2. Select the two pairs of lines shown
3. Press **esc** to exit the command



Step 7: – Create Dimension

1. Select **Sketch > Sketch Dimension.**
2. Select the line
3. Click again to place the dimension.
4. Type in a value of 500 mm.
5. Press **Enter** to accept the value.

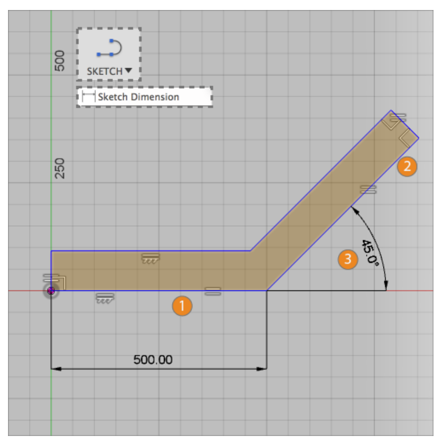
Note: The dimension command is still active and you can go right into placing the next dimensions.



Step 8: – Create Angle Dimension

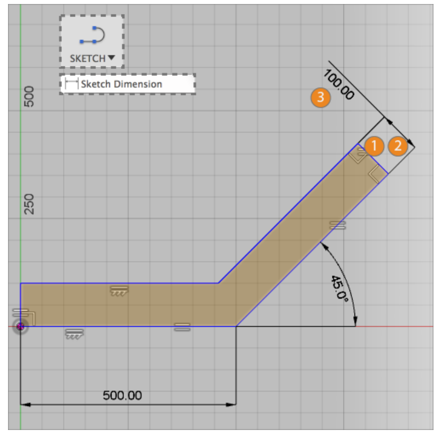
1. Select **Sketch > Sketch Dimension.**
2. Select the bottom line
3. Select the angled line
4. Place the dimension
5. Type 45
6. Press **Enter** to accept the value

Note: The dimension command is still active and you can go right into placing the next dimensions.



Step 9: – Aligned Dimension

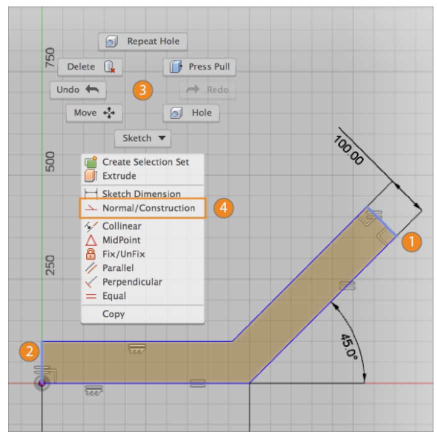
1. Select **Sketch > Sketch Dimension.**
2. Select the upper right line
3. Move just away from the line
4. Notice a small icon on the cursor
5. Select in space again to begin an aligned dimension
6. Type 100
7. Press **Enter** to accept the value
8. Press **esc** to exit the command



Step 10: – Construction Geometry

1. Select Upper line
2. Hold **Shift,** select second line
3. Press **Right Mouse Button**
4. Select **Normal/Construction**

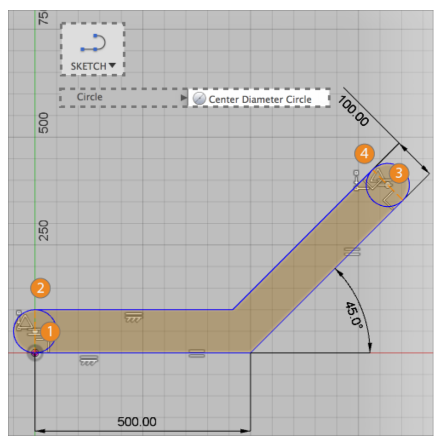
Note: Construction geometry is not considered when looking for profiles. Use construction geometry for reference when creating sketches. It will show as dashed lines to indicate it is construction geometry.



Step 11: – Create 2 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the bottom line midpoint
3. Select the intersection shown
4. Repeat at the top edge
5. Press **esc** to exit the command

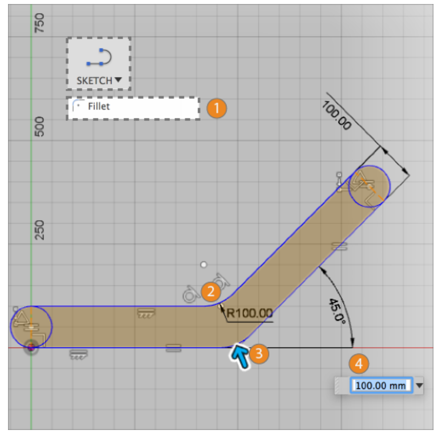
Note: Make sure you "snap" the geometry in place. You should see a midpoint (triangle) constraint created. Try dragging the circles, if they move you missed a snap.



Step 12: – Sketch Fillet

1. Select **Sketch > Fillet**
2. Select the intersection point
3. Select the other intersection point
4. Enter a value of 100 mm
5. Press **Enter** to confirm

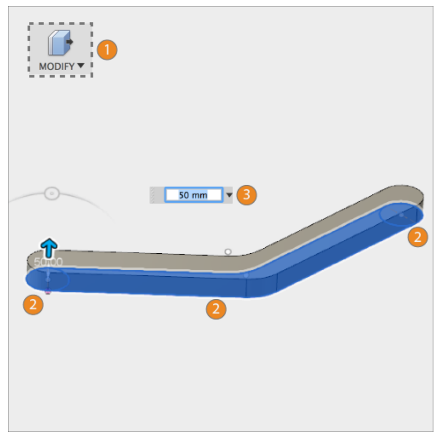
Note: All fillets created at the same time will have an equal radius. Create them separately to have different radius values. Note in many cases creating a fillet in a sketch isn't the best choice. As you see here it deletes the 500 mm dimension.



Step 13: – Extrude the profile

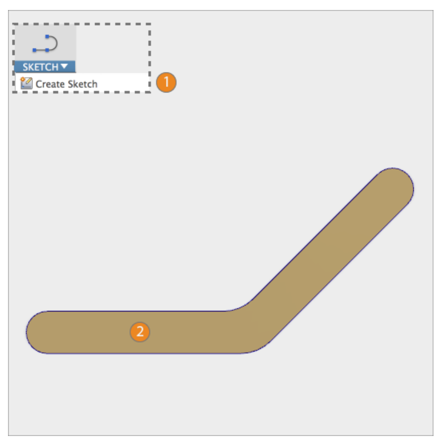
1. Select **Modify > Press Pull.**
2. Select the 3 profiles
3. Drag the arrow up or type in a value of **50 mm** to set the depth.
4. Press **OK.**

Note: One sketch may contain multiple profiles. Here we limited the number of profiles by using construction geometry.



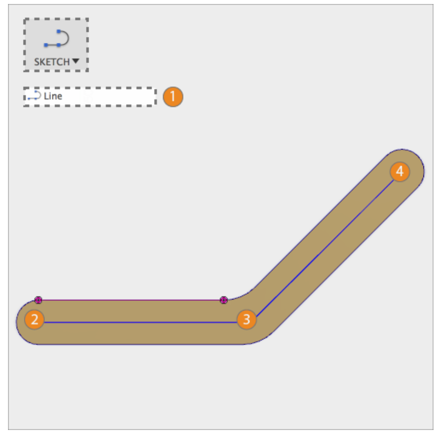
Step 14: – Create another sketch

1. Select **Sketch > Create Sketch.**
2. Select the top of the bracket.



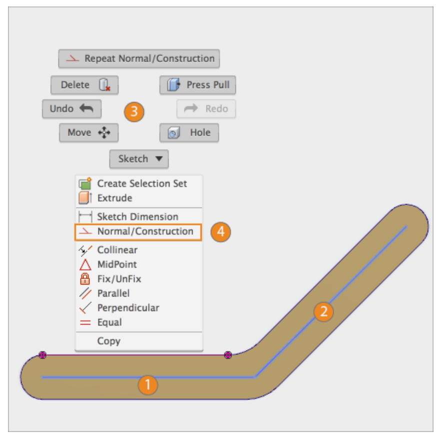
Step 15: – Sketch Lines

6. Select **Sketch > Line**
7. Select the left center point
8. Select near the middle
9. Select the upper center point
10. Press **esc** to exit the command



Step 16: – Construction Geometry

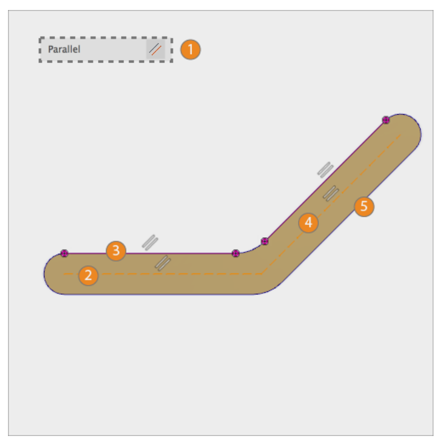
1. Select one line
2. Hold **Shift,** select second line
3. Press **Right Mouse Button**
4. Select **Normal/Construction**



Step 17: – Create Parallel Constraints

1. Select **Parallel**
2. Select lower line
3. Select lower edge
4. Select angled line
5. Select Angled edge
6. Press **esc** to exit the command

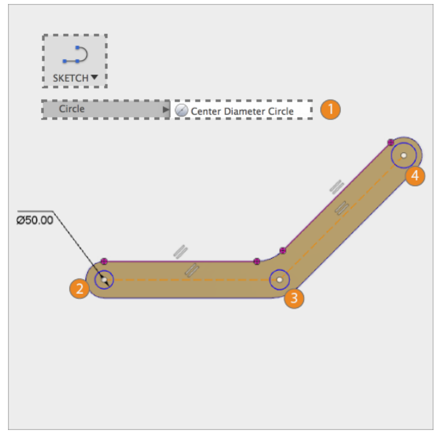
Note: This actually fully constrains these lines. It is worthwhile to learn the different strategies you can take to fully define the sketch lines.



Step 18: – Create 3 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the lower left center point
3. Type **50mm**
4. Press **Enter**
5. Draw 2 more circles

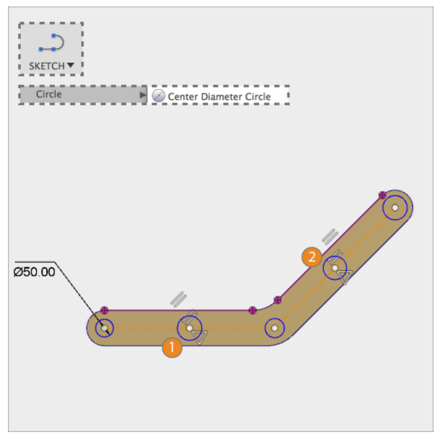
Note: Make sure to "snap" to the end points of the lines when placing the circle center points.



Step 19: – Create 2 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the lower line midpoint
3. Click to place the circle
4. Draw 1 more circle at the midpoint of the angled line

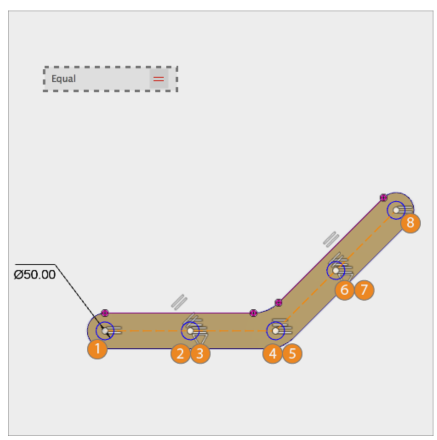
Note: Make sure to "snap" to the midpoints of the construction lines. You should see a small triangle appear to indicate your circle is locked to the midpoint.



Step 20: – Create Equal Constraints

1. Select **Equal**
2. Select 2 circles
3. Repeat until you have all circles equal
4. Press **esc** to exit the command

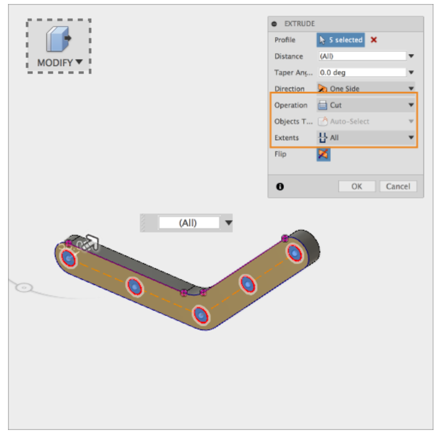
Note: You have to select the circles in pairs. One equal constraint is always applied to two circles. So for each constraint you need to select two circles.



Step 21: – Extrude Circles

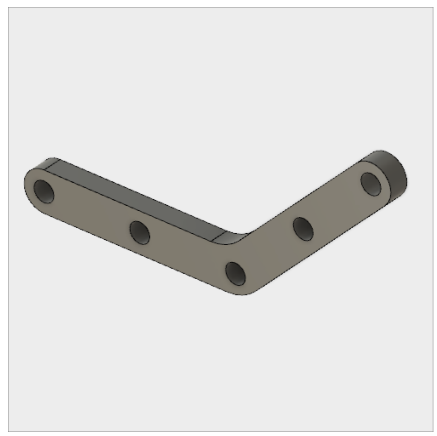
1. Select **Modify > Press Pull.**
2. Select the 3 profiles
3. Drag the arrow into the part.
4. Make sure **Cut** is selected
5. Select **All** from the menu
6. Press **OK.**

Note: One sketch may contain multiple profiles. Make sure to select the 5 circles. You can add or remove profiles from a selection later by using the **CMD** key on mac and **CTRL** key on windows.



Step 22: – Lesson complete!

Congratulations you have finished this lesson and have learned more ways to create relationships in a sketch using constraints and construction geometry. In the next lesson, you'll go through the fundamentals of how to further develop 3D models using various modeling tools.



3.1: Sculpting

Sculpting in Fusion 360 allows for the intuitive freeform creation of organic solid bodies and surfaces by leveraging the T-Splines technology. In the Sculpt Workspace, you can rapidly explore forms by simply pressing and pulling on subdivided surfaces. This “hands-on” approach to 3D modeling allows for fast iteration and early stage conceptualization within Fusion 360. Sculpted forms are easily converted to solid bodies, and can be used in conjunction with Fusion 360’s solid modeling commands. Modeling with T-Splines is unlike any other subdivision-modeling tool. One of the main advantages of T-Splines is the ability to add detail only where necessary - a single T-Spline surface can be incredibly smooth, while still having areas of high detail and remaining easy to manipulate.

Lesson 1: Creating T-Spline Forms

Learning Objectives

1. Create a T-Spline Primitive Form
2. Create a T-Spline Revolve Form
3. Create a T-Spline Sweep Form
4. Create a T-Spline Loft Form

Datasets Required

In Samples section of your Data Panel, browse to:

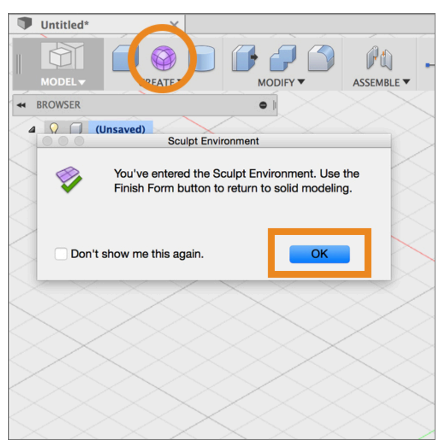
Fusion 101 Training > 03 – Sculpt > **03_Sculpting_Introduction.f3d**

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

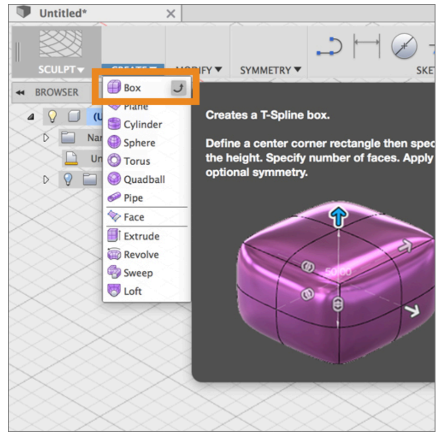
Step 1: Go to the Sculpt workspace – Let’s go to the Sculpt workspace to access the Sculpt tools.

1. In the Model workspace select **Create Form** to enter the Sculpt workspace
2. A dialog box appears, telling you to click **Finish Form** to return to the model workspace when you are finished sculpting.
3. Select **OK**



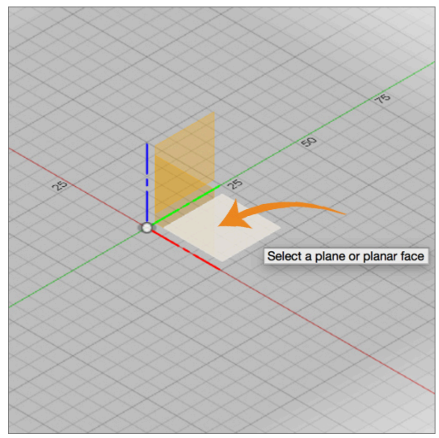
Step 2: – Select the T-Spline Box primitive

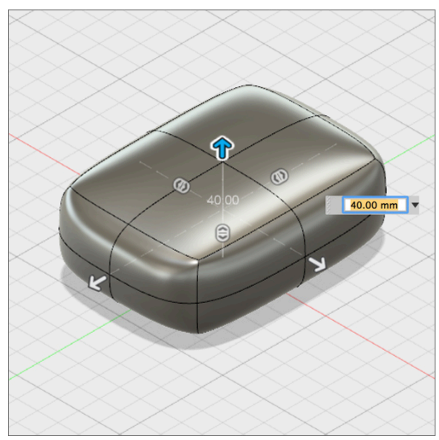
1. Select the drop-down arrow under **Create** to expand the list of creation commands.
2. Select **Box** to create a T-Spline box.



Step 3: Position the Box – When you create a new primitive you first need to indicate which plane you want to build on and then enter the dimensions of the primitive.

1. Select the bottom plane
2. Select the origin to specify the center point of the box 2D profile
3. Drag the mouse and click on the plane again to specify the initial size of the rectangle.

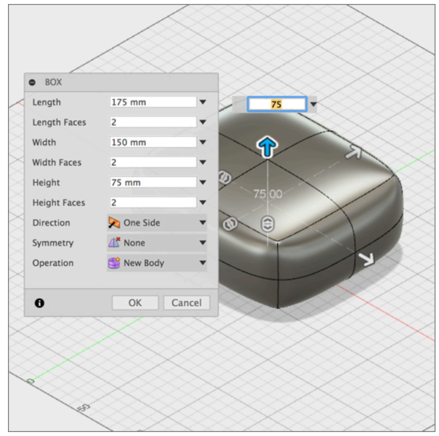




Step 4: Set the dimensions for the box

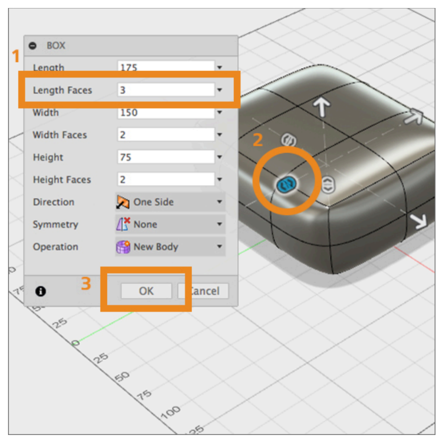
1. In the Box dialog window set the length to **175mm**, and the width to **150mm** respectively by entering these numbers.
2. Drag the arrow pointing up from the box, and set the height equal to **75mm**.

Note: When dragging a manipulator in Fusion 360 the increments for the move are tide to how close or far away the camera is to the manipulator. The closer you are the smaller the move increments, the farther you are the larger the move increments. If you find that the move is changing at too large of increments, simply zoom in to reduce the size of each step.



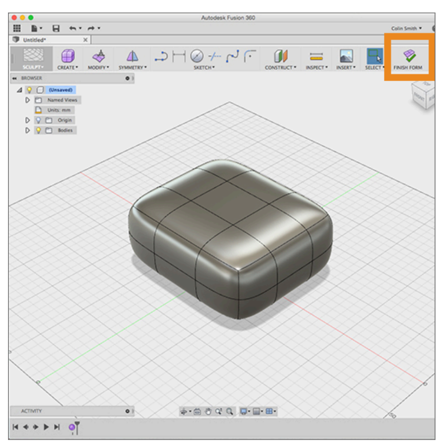
Step 5: Increase the number of faces to the box in Length and Width.

1. In the Box dialog window set the **Length Faces** to **3**
2. To set the width faces, drag the **double-headed arrow manipulator** on the box primitive up to increase the number of width faces until it equals **3**.
3. Select OK to complete the **Box** primitive setup.



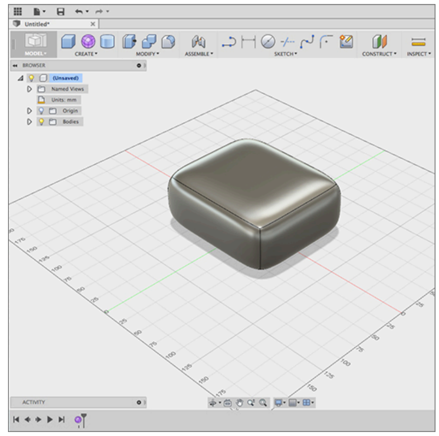
Step 6: Finish the form. You have just created your first T-Spline body. To include this form as part of your solid model you need to indicate that you are done creating T-Spline forms for the moment and that you want to go back to the Model workspace.

1. Select **Finish Form** at the end of the Sculpt workspace toolbar.
2. The T-Spline form is automatically converted to a solid body and you are brought back to the **Model**



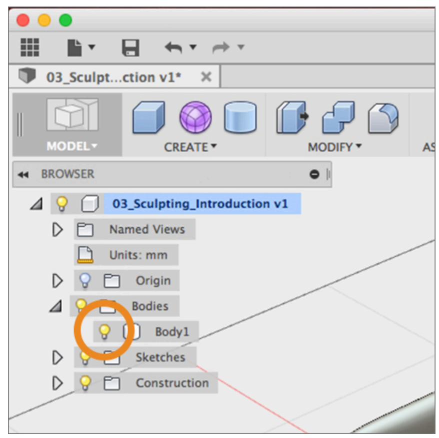
workspace.

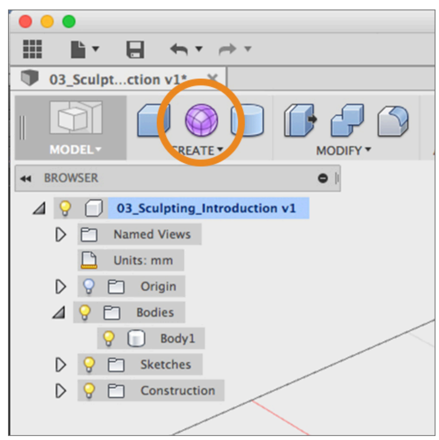
Note: If you create a closed T-Spline form it will be converted to a solid body when you select **Finish Form**. If you create an open T-Spline form, for example a T-Spline Plane, it will be converted to a surface body when you select **Finish Form**.



Step 7: Create a T-Spline Revolve Form. Let’s create a T-Spline form by revolving a sketch curve. The Revolve command creates a form by rotation 2D geometry about a fixed axis

1. In the **Browser**, select the drop-down arrow next to your **Bodies** folder
2. Click the light bulb next to **Body1** to turn off the visibility.
3. Select **Create > Create Form**.

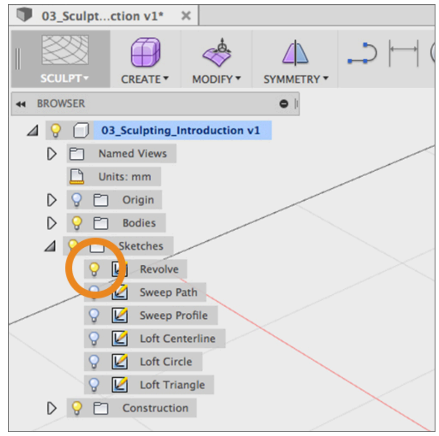




Step 8: – Turn on the visibility of the sketch curve that you will be revolving.

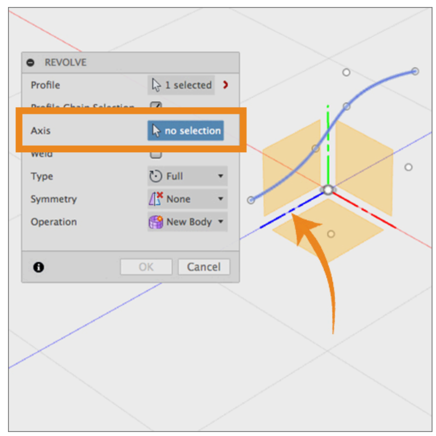
1. In the **Browser**, select the drop-down arrow next to your **Sketches** folder.

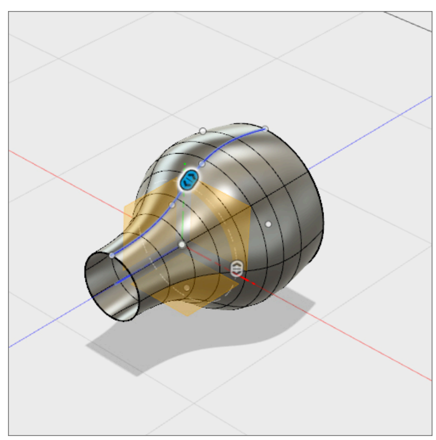
Select the **light bulb icon** next to **Revolve** to turn on the visibility of the **Revolve** sketch.



Step 9: Revolve a T-Spline using the sketch curve as input.

1. Select **Create > Revolve.**
2. Select on the **sketch curve** to identify it as the Profile curve to be revolved.
3. In the dialog window select **no selection** next to **Axis** to show the axis selector in the workspace
4. Select the **Blue** axis. A T-Spline form is revolved 360 degrees around the Blue axis.

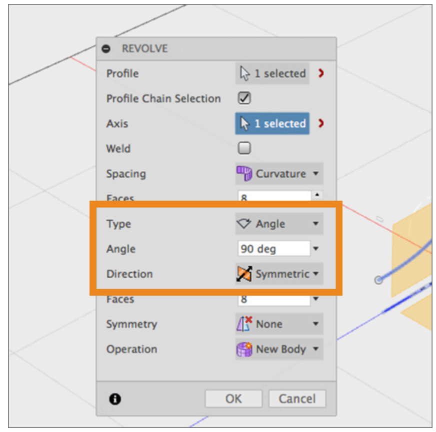


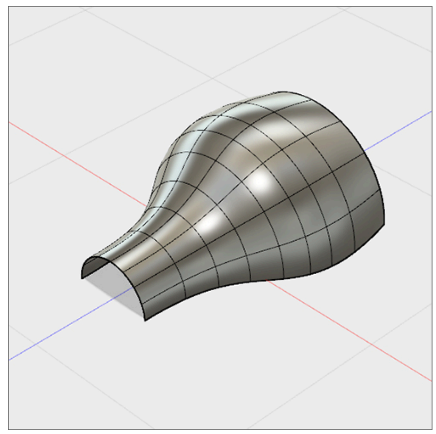


Step 10: Change the Revolve settings. Let's change the settings in the Revolve settings to change the **Angle** the direction that the curve is revolved.

1. In the **Revolve** dialog window, change the **Type** from **Full** to **Angle**.
2. Enter **90 degrees** for the **Angle** field.
3. Change the Direction from **One Side** to **Symmetric**.
4. Select **OK**.
5. Select **Finish Form** to return to the **Model** workspace.

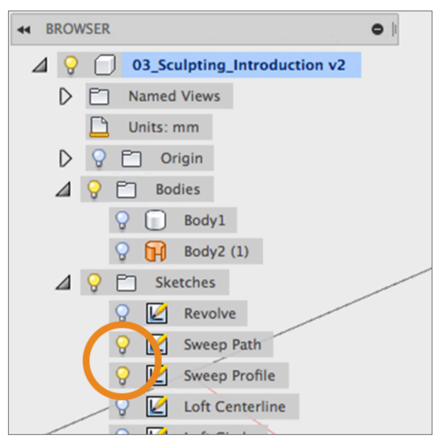
Note: The **Revolve** manipulator can also be used to adjust the number of faces in longitude and latitude of the revolved shape as well as the degree of the **Angle**.





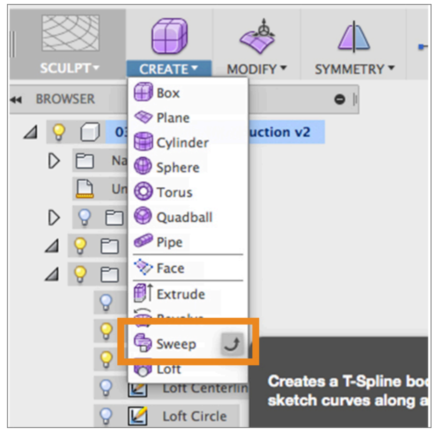
Step 11: Create a T-Spline Form using **Sweep**. The **Sweep** command uses two sketch curves to define a shape. One curve is selected as the **Profile**, which is swept along a **Path** curve to create the shape.

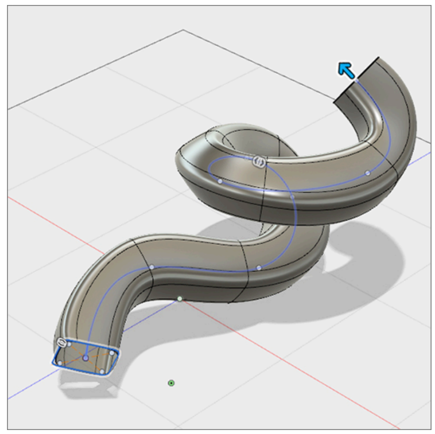
1. In the **Model** workspace hide the previously created **Body 2 (1)**
2. In the Browser in the **Sketches** section select the light bulb icon next to Revolve Sketch to turn off the sketch visibility.
3. Turn on the visibility for the **Sweep Path** and **Sweep Profile** sketches.



Step 12: Start the Sweep command.

1. Select **Create Form** in the **Model workspace** to change to the **Sculpt Workspace.**
2. Select **Create > Sweep** to launch the Sweep dialog window.
3. Select the **Sweep Profile** sketch to identify this sketch as the **Profile**
4. Select **no selection** next to **Path**.
5. Select the **Sweep Path** sketch to identify this sketch as the **Path**. The swept T-Spline Form is generated.

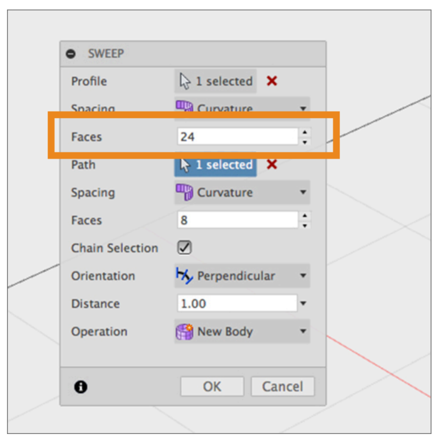




Step 13: Add faces to the swept surface. In order to match our rounded-square profile, we need to increase the number of faces for the profile.

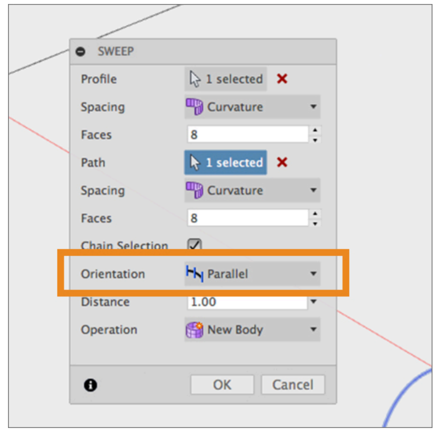
1. In the dialog window, set the number of faces for the **Profile** equal to **24**.

Note: The greater number of faces, the closer the body matches the path.



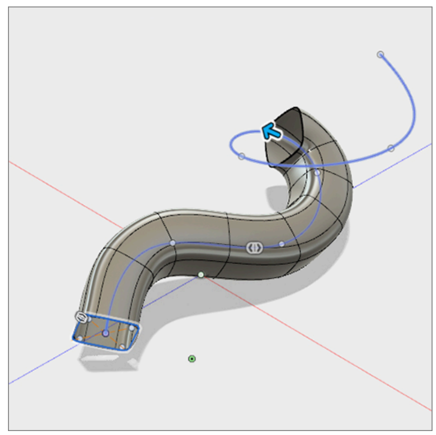
Step 14: Change the sweep orientation. There may be instances where the orientation of the profile as is moves along the path will give you a more desirable result. Use the **Orientation** option to change the sweep behavior.

1. In the dialog window, change the **Orientation** from **Perpendicular** to **Parallel**. Observe how the sweep behavior drastically changes.
2. Change the Orientation **back** to **Perpendicular**.



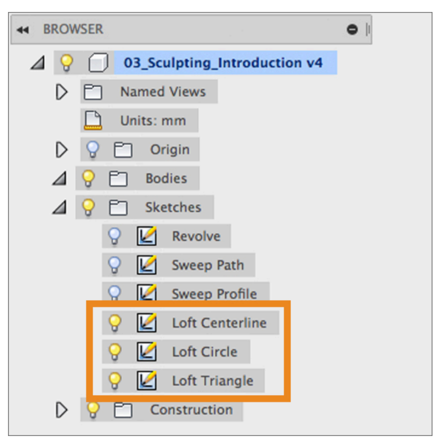
Step 15: Change the sweep distance. The Sweep command allows you to alter the amount of the path curve that is used to create the T-Spline form.

1. Drag the arrow at the end of the **Path** to alter the sweep distance. In addition to using this arrow, we can set the distance in the dialog window.
2. In the dialog window, set the distance equal to **0.5**.
3. The sweep now travels half the length of our Path curve.
6. Select **OK**.
7. Select **Finish Form**.



Step 16: Create a T-Spline Form with **Loft** using 3 sketch profiles as input to create a lofted shape.

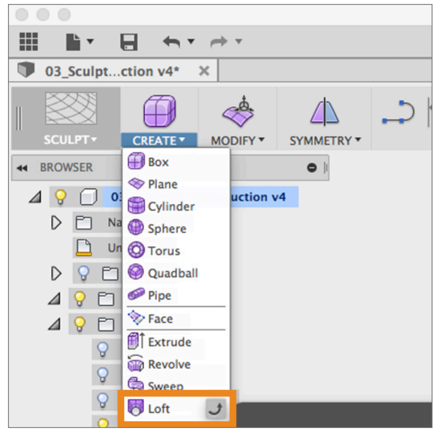
1. Turn off visibility of **Body 3 (1)**
2. Turn off visibility of the two **Sweep** sketches.
3. Turn on visibility for sketches labeled **Loft Centerline**, **Circle**, and **Triangle**.
4. Select **Create > Create Form** to change to the Sculpt workspace

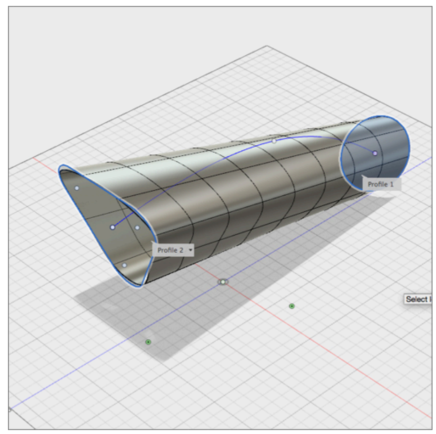


Step 17: Select the profiles to create the loft form.

1. Select Create **> Loft.**
2. Click the **Triangular** profile in the canvas.
3. Click the **Circular** profile in the canvas. This creates a straight lofted shape transitioning between the triangle and the circle.

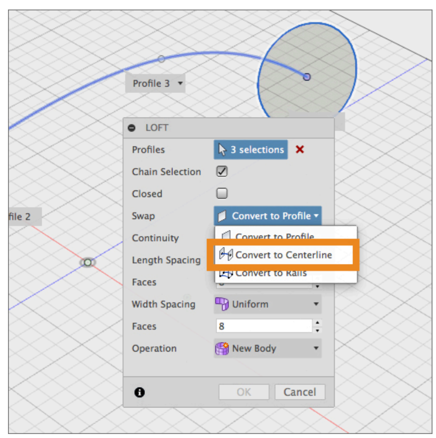
Note: You can have multiple profile shapes to loft between.

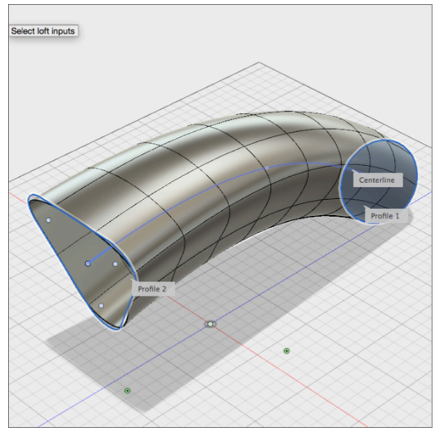




Step 18: Change the Loft shape by defining a centerline. By default the loft will always create straight transitional surfaces between profiles. To control the direction of the surface between the profiles you can add a sketch curve a centerline.

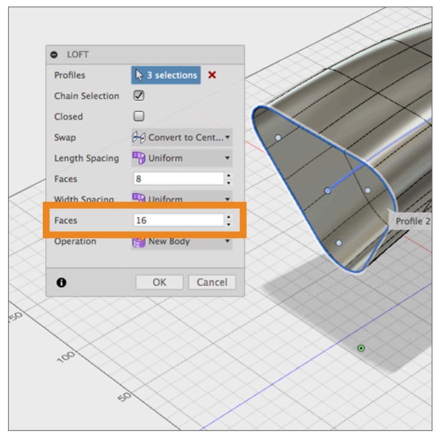
1. Click the **centerline** curve
2. You receive an error because initially the software assumes that you are adding another profile for the loft.
3. To specify this curve as a centerline select **Convert to Centerline** under **Swap** in the dialog window.





Step 19: The T-Spline surface is not matching well to the triangle profile. Let's increase the number of faces to improve the match.

1. In the dialog window, set the number of faces for the **Width** equal to **16**.
2. Click **OK**.
3. Select **Finish Form.**



Lesson 2: Modify a T-Spline Form

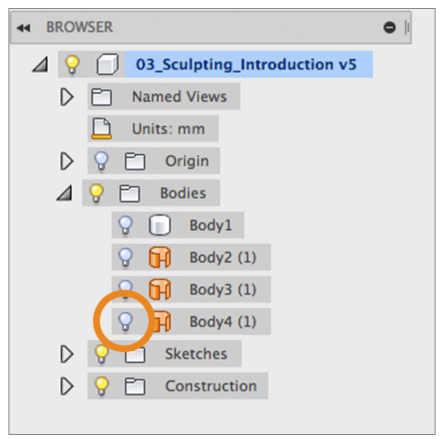
The real power of T-Splines is that it allows for freeform shape manipulation of shapes by moving, rotating and scaling vertices, edges and faces of a T-Spline surface. The most common command you will use to do this is Edit Form. You can also add and remove faces in your forms to get detail where you need it without making the entire form overly complex.

Learning Objectives

1. Move, Rotate and Scale T-Spline geometry with Edit Form
2. Add geometry to a T-Spline body with Edit Form
3. Change the display mode
4. Insert Edges

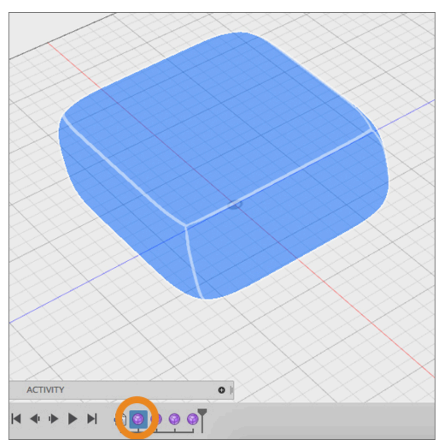
Step 1: Turn off the visibility of the Loft body

1. Turn off visibility of the previously created **Body 4 (1)** from the last lesson.
2. Turn off visibility for any visible sketches.

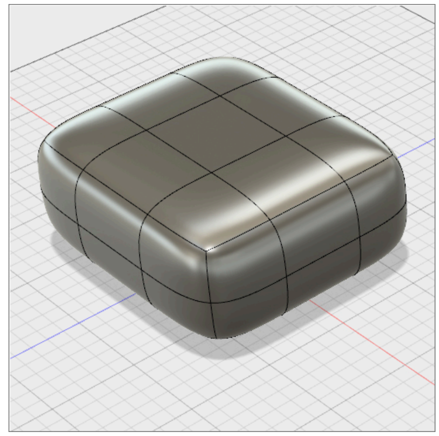


Step 2: Edit the T-Spline shape in the Timeline

1. In the **parametric timeline** at the bottom of the Model window, located the 4 T-Spline form icons.
2. Hover your pointer over the first form icon and you will notice that the primitive box is highlighted on the screen.
3. Right-click on the form icon in the timeline and select **Edit**.



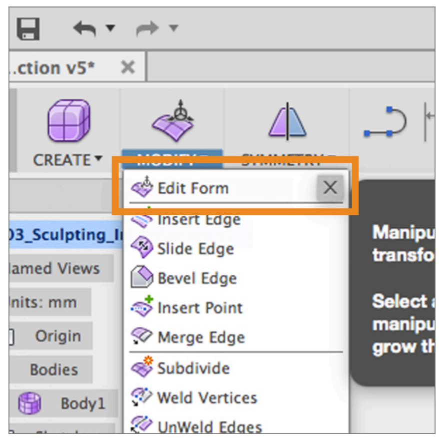
Note: When using Fusion 360 with history turned on, T-Spline operations do not have history in the same way that the operations in the Model and Patch workspaces do. However you can go back in history and make adjustments to your T-Spline shapes and those changes will be recomputed in the history of your model when you exit the Sculpt workspace.

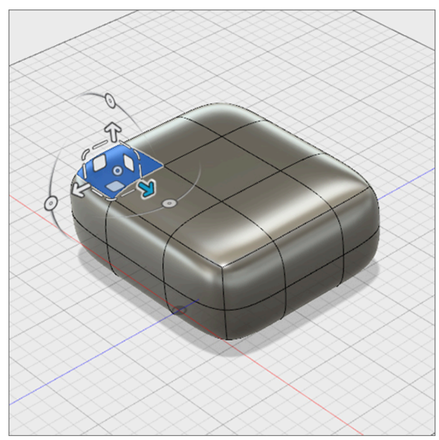


Step 3: Start the Edit Form command

1. Click **Modify > Edit Form**. The Edit Form command can be used to directly manipulate **Faces**, **Edges**, and **Vertices**.
2. Click on a top face located at one of the corners of the box.
3. The manipulator displays on the face with an assortment of tools.

Note: Directly manipulating a face creates the greatest changes in your form, while manipulating a single vertex will result in smaller changes.

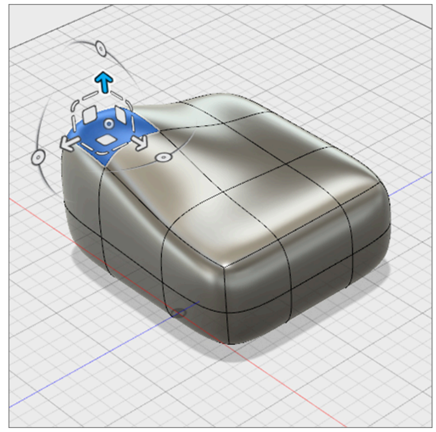




Step 4: Move a face in a single axis direction.

1. Click and drag on the arrow pointing up to translate (move) the selected face up by 30 mm.

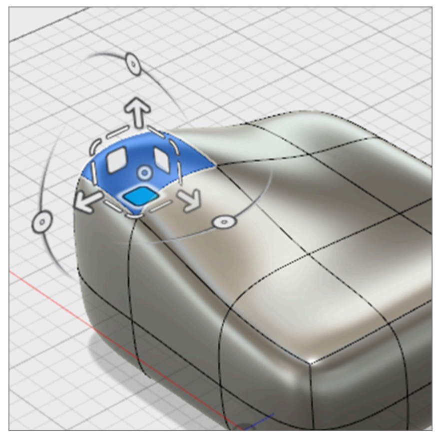
Note: When you move a face on a surface the surrounding faces move to maintain continuity.



Step 5: Move a face in a planar direction.

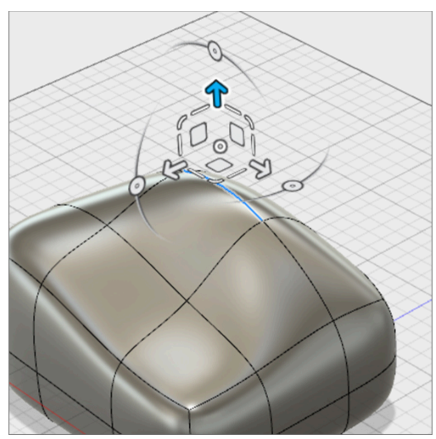
1. Click and drag on one of the **white squares** to translate the selected face on a plane parallel to a given plane in world space.

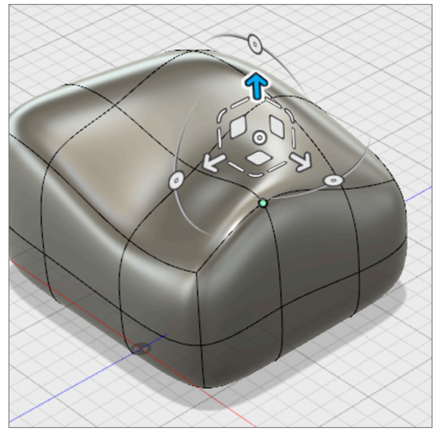
Note: By default the **Edit Form** manipulator uses the World Space coordinates, you can also use View space (based on the camera view) and Local space (based on the normal direction of the surface). These can be changed in the **Coordinate** options in **Edit Form**.



Step 6: Move a T-Spline edge and vertex.

1. Select a single **edge**.
2. Use any of the translate manipulators to compare the effect.
3. Select a single **vertex**.
4. Translate this vertex to see how this creates more subtle changes

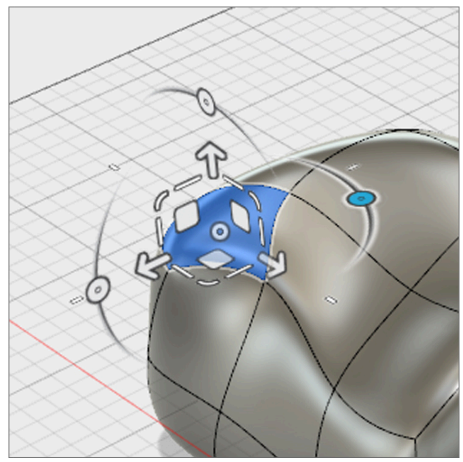




Step 6: Rotation with Edit Form

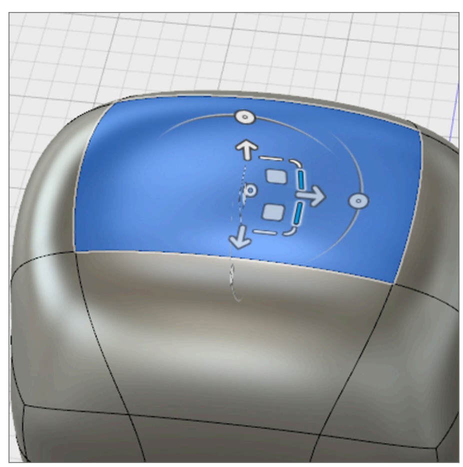
1. Make sure that at least one **face** is selected.
2. Click and drag one of the **circular arcs** to rotate the selected geometry about a single axis.

Note: Be careful not to rotate geometry too far, as self-intersecting faces, or geometry that twists through itself will result in not being able to convert it in to a solid body.



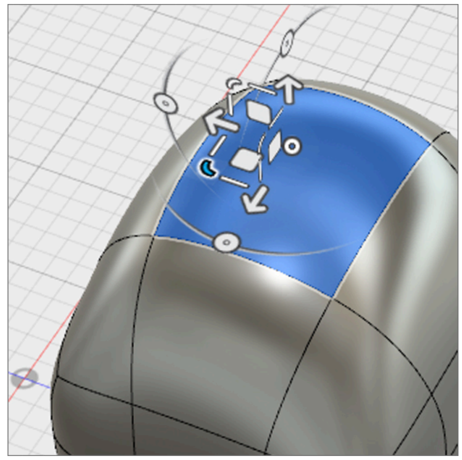
Step 7: Single direction scaling

1. Make sure that at least one **face** is selected.
2. Click and drag one of the **straight lines** to scale the selected face in one direction.



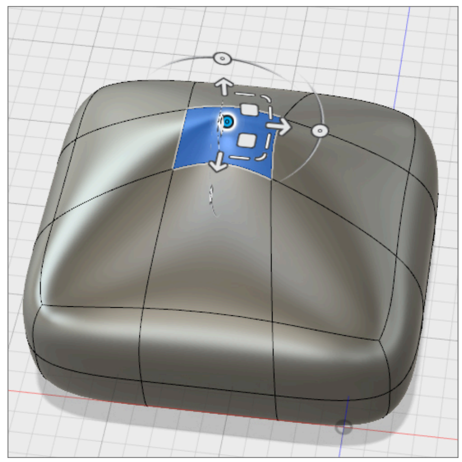
Step 8: Planar scaling

1. Make sure that at least one **face** is selected.
2. Click and drag one of the small **corner manipulators** to scale the selected face along a plane – or in two directions.



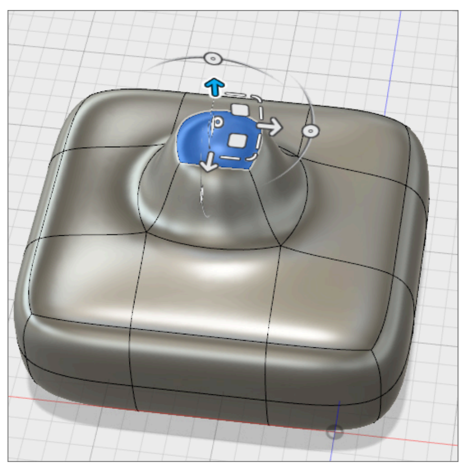
Step 9: Universal scaling

1. Make sure that at least one **face** is selected.
2. Click and drag the **circular manipulator** at the center of the manipulator.
3. Dragging **left** or **right** will scale the geometry in all directions **up** or **down**.



Step 10: Extrude a face to add geometry

1. Select a **single face** of your T-Spline form.
2. With the Edit Form command still active, hold the **alt-option** on Mac or **alt** key on Windows.
3. Click and drag the **single arrow** to add geometry outward from the selected face.
4. Let go of the **alt-option/alt** key as well as the left-mouse button.
5. Select OK to close **Edit Form**

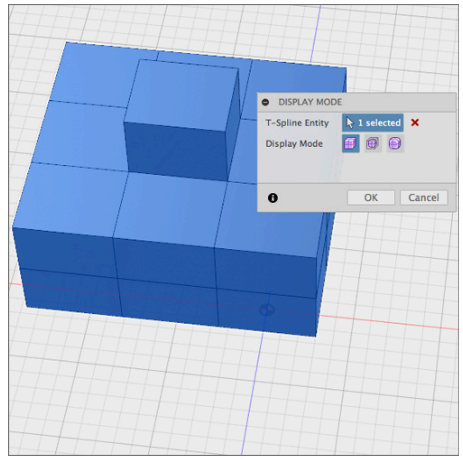


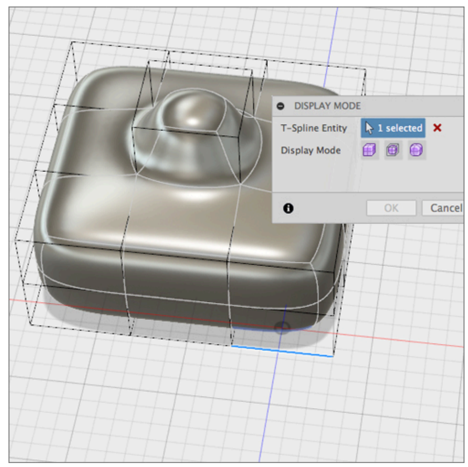
Note: You can use this hotkey function to extrude any edge(s) or face(s) that are selected on a T-Spline form.

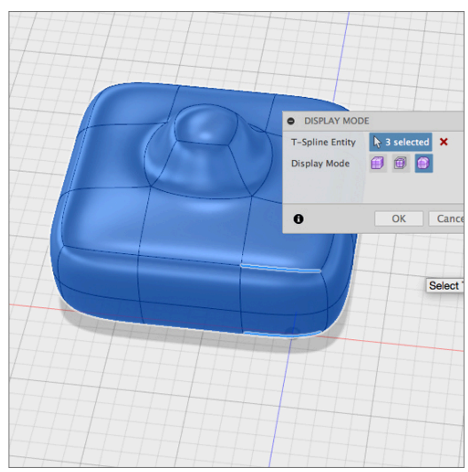
Step 11: Change display mode. When modeling with T-Splines you have 3 different display modes to choose from. Changing the display mode can help you find problem areas on your model and increase performance by not having to constantly smooth the T-Spline shape. For this example we will switch the display to Box Mode – a polygon version of the T-Spline form.

1. Select **Modify > Performance > Display** Mode to bring up the Display Mode window
2. In the Display Mode window select **Display Mode > Box Display** to show a unsmoothed version of the T-Spline form.
3. Select **Display Mode > Control Frame Display** to see a combination of the unsmoothed and smoothed versions of the T-Spline.
4. Select Display Mode > Smooth Display to go back to the smoothed version.
5. Select **Display Mode > Box Display**
6. Select **OK** to close the window

Note: Display mode can also be switch by using the combination of **Control + 1 for Box**, **Control +2 for Control Frame** and **Control + 3 for Smooth** on a Mac. Or **Alt +1**, **Alt +2**, **Alt +3** respectively on a PC.   You can also find these controls in the **Selection Options** section of the **Edit Form** window.

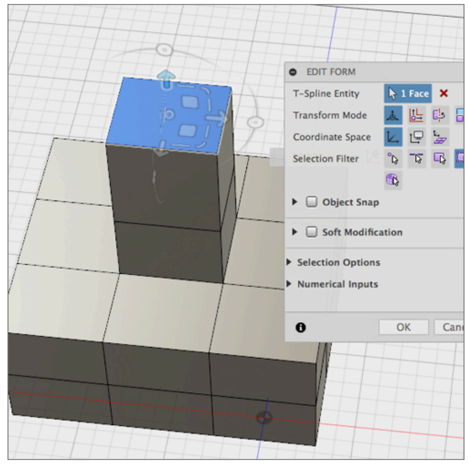






Step 12: Working in Box Display Mode.

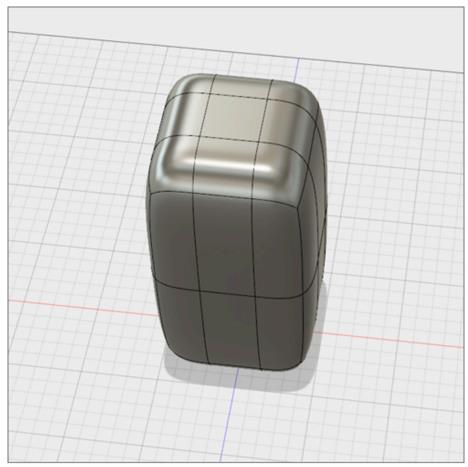
1. Hold down the **right mouse button** to bring up the radial menu.
2. Select **Edit Form** in the radial menu
3. Select a face and hold **alt-option/alt** while dragging the blue arrow up to extrude the face. You will notice that the performance is faster in Box mode that in Smooth mode.
4. Hold **control/alt +3** to return to smooth mode.
5. Hold down the **right mouse button** and select **OK** from the radial menu to close the **Edit Form** window.
6. Select **Finish Form** from the menu bar to go back to the Model workspace.

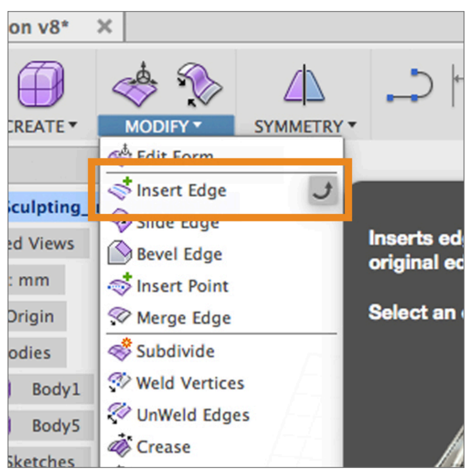


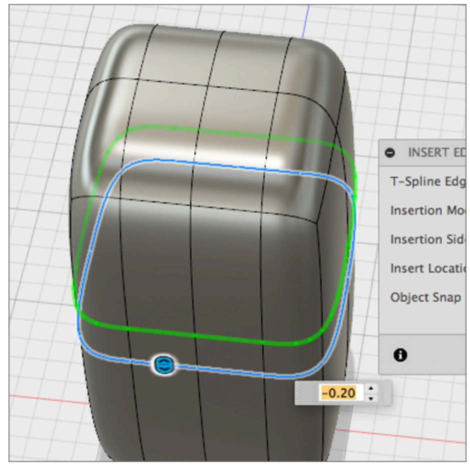
Step 13: Add edges to a T-Spline form

7. Enter the Sculpt workspace by selecting **Create > Create Form**.
8. Create another **Box** primitive whose length, width, and height are **100mm, 100mm**, and **200mm.**
9. Set the number of length, width, and height faces equal to **4, 2,** and **2.**
10. Click **OK**.
11. Click **Modify** > **Insert Edge**.
12. **Double-click** on one of the middle edges to select the entire middle loop.

Note: Double clicking on an edge will select the complete edge loop. Selecting a single edge inserts an edge to a face on either side of the selected edge. Selecting an edge loop adds a second loop.



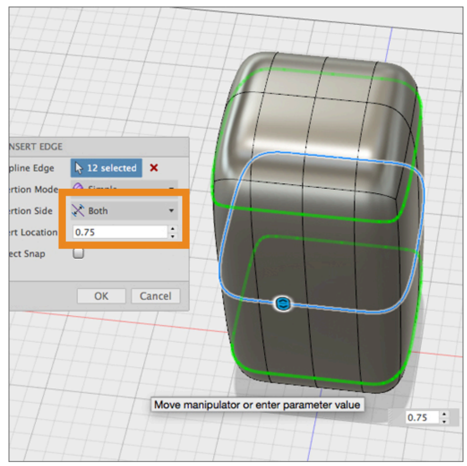


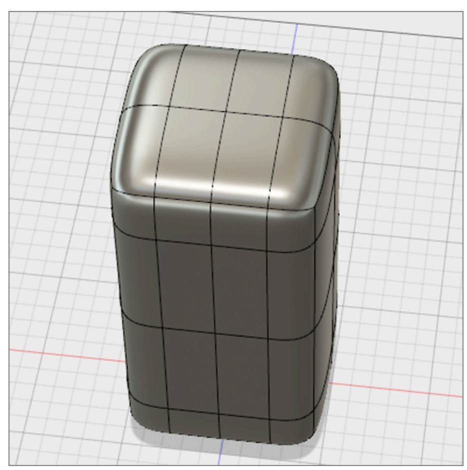


Step 14: Adjust the insert location

1. Click and drag the double arrow to adjust the position of the inserted edge.
2. By hand, or with the text field, set the **Insert Location** equal to **0.75**.
3. Change the **Insert Side** from Single to **Both**.
4. Click **OK**.

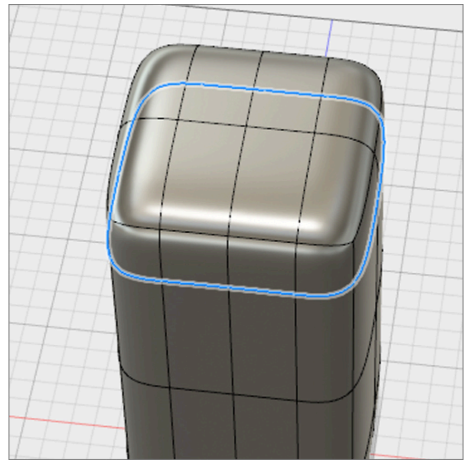
Note: You probably noticed that the shape of the T-Spline form changed after you inserted additional edges. The top and the bottom of the box became sharper by adding more edges near the existing top and bottom edges. If you want to insert an edge(s) and not change the shape of the T-Spline form, change the **Insertion Mode** to **Exact**.

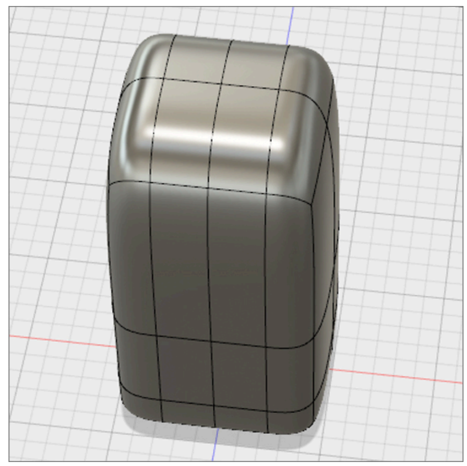




Step 15: Delete and edge

1. Select the recently added the **upper edge loop**.
2. Select **Modify > Delete** or press the **Delete** key on your keyboard.
3. Select **Finish Form**.





Lesson 3: Create a T-Spline Form using a reference image

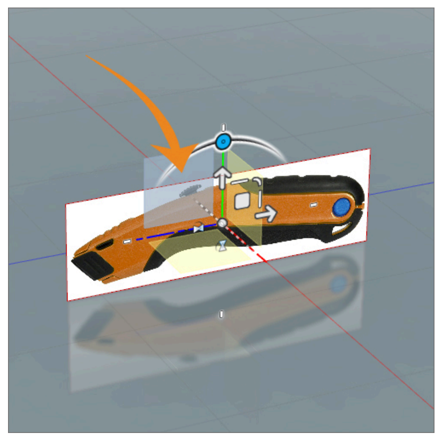
When sculpting a shape with T-Splines it is helpful to have a reference image to guide you. Reference images can be plan view sketches or photographs that are set in the background of the workspace to model from. In Fusion 360 you can attach an image to a work plane and then calibrate the image so that you are modeling in the correct scale. In this lesson we will use a side view photograph of a utility knife as reference to sculpt a T-Spline body that will be the outer shape body of the knife.

Learning Objectives

1. Insert and image in to the workspace using **Attach Canvas**
2. Use **Calibrate** to set the proper scale for the reference image
3. Invoke **symmetry** when modeling a T-Spline box
4. Use **Insert Point** to draw edges on a T-Spline face(s)

Step 1: Attach a canvas

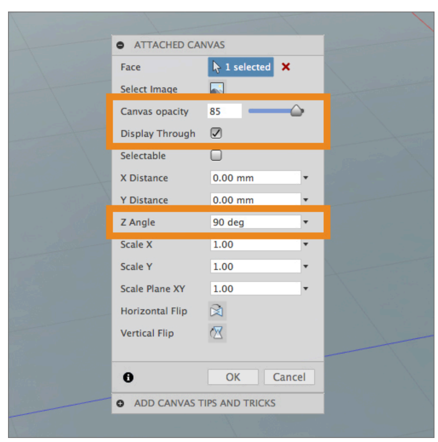
1. Select **Insert > Attached Canvas**.
2. Select the **YZ Plane** (between the green and blue axis) to indicate which plane the image should be attached to.
3. In the dialog window, click the **Select Image** button and navigate to the **03_UtilityKnife.jpg** file.



Step 2: Adjust canvas settings

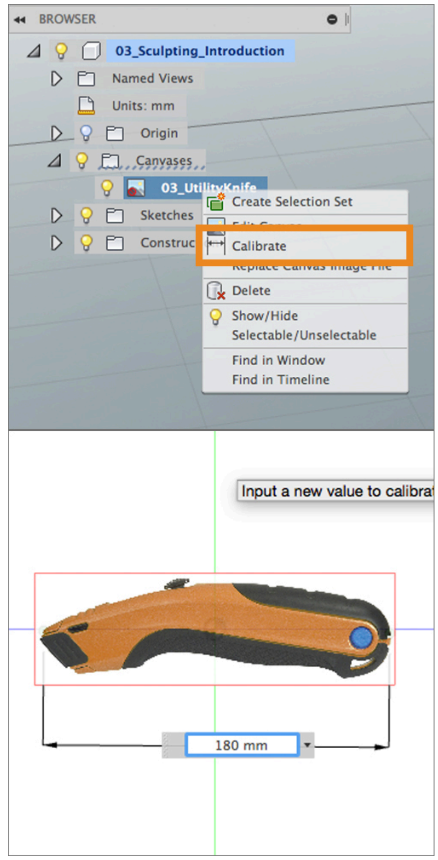
1. If necessary, **rotate** the canvas **90 degrees** to orient it properly
2. Lower the opacity to **85**.
3. Check the box for **Display Through** to ensure that the canvas can be seen through your T-Spline form.
4. Click **OK**.

Note: You don't need to be concerned about the size and scale of the image at this point. We will adjust the scale using the **Calibrate** tool. Calibrating the image ensures that you are modeling in the correct scale in the workspace.



Step 3: Calibrate the image.

1. In the Browser, click the drop-down arrow next to the **Canvases** folder.
2. Right-click on **UtilityKnife** and select **Calibrate**.
3. Click **Right** on the ViewCube to view the utility knife from the side.
4. Click once at the **front** of the utility knife.
5. Click once at the **back** of the utility knife.
6. Enter **180 mm** in the length field and hit enter.
7. The canvas will scale up accordingly.

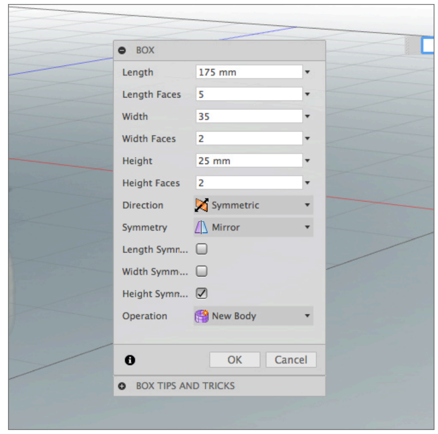


Step 3: Create a box primitive

1. Click the **Create > Create Form** icon to enter the Sculpt workspace.
2. Click **Create > Box**.
3. Select the same side plane (YZ) as the canvas to specify the plane that the Box is placed on.
4. Click once at the **origin** to specify the Box's center point.
5. Move the mouse and **click** again at the outer edge of the reference image to draw its 2D profile.

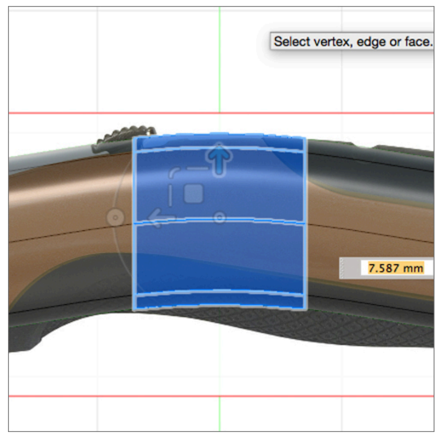
Step 4: Set box dimensions and add symmetry

1. Set the Box's **Length, Width,** and **Height** equal to **175, 35,** and **25 mm**, respectively.
2. Set the number of Length Face equal to **5**, and the width and height faces equal to **2**.
3. In the dialog window, change the **Symmetry** from None to **Mirror**.
4. Check the box for **Height Symmetry**.
5. Select **OK**.



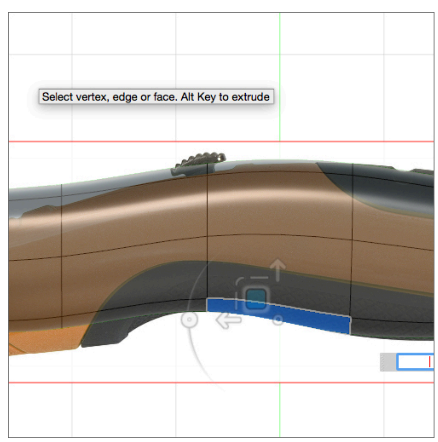
Step 5: move faces to the top of the knife image.

1. If you are not in the right side view still, click on the **Right** side of the **View Cube.**
2. Select **Modify > Edit Form**.
3. Select the middle set of faces by holding the **left mouse button** and dragging to the lower right over top of the faces you want to select.
4. Using the **Planar Translation** manipulator, move the selected faces to align the top of the T-spline body with the top of the utility knife image.



Step 6: move faces to the top of the knife image.

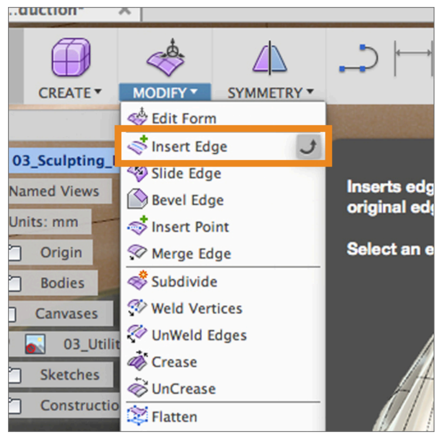
1. To align the bottom in this section, select the **bottom face** and use the **planar translation** and **rotate manipulators**.
5. Repeat the previous 2 steps for the rest of the T-spline form. It will also be helpful to use the **single-direction scale** manipulator in some cases.
6. For more controlled editing, try modifying individual edges.

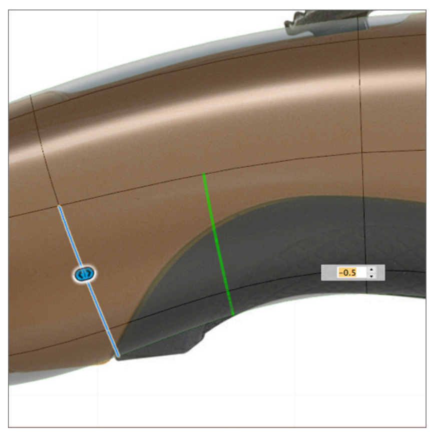


Step 6: Insert edges to get closer to the knife shape. Our T-Spline form is starting to resemble our reference image but there aren't enough edges in the T-Spline to capture all the detail of the Knife.

1. Hold **Shift** then select the edges shown at the front of the knife.
2. Click **Modify > Insert Edge**.
3. Drag the direct manipulator to the right to position the new edges at an **Insert Location** around **-0.5**.
4. Click **OK**.

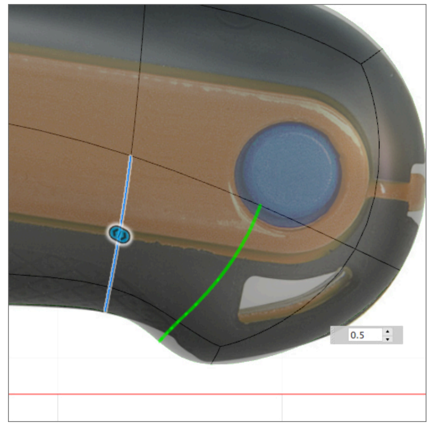
Note: With Symmetry enabled you only need to select the edges one side of the symmetry plane, the matching ones on the other side will be automatically selected also and will be displayed in yellow.





Step 7: Insert additional edges with **Insert Edge**.

1. Holding **Shift** then select the edges shown at the back of the knife.
2. Select **Modify > Insert Edge**.
3. Drag the direct manipulator to the right to position the new edges at an **Insert Location** around **-0.5**.
4. Click **OK**.



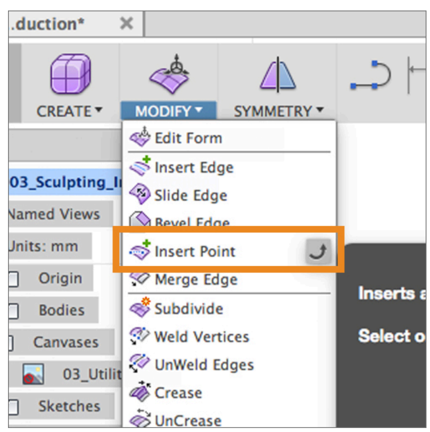
Step 8: Move the inserted edges with **Edit Form**.

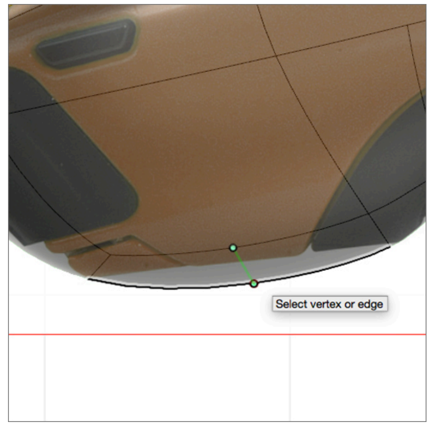
1. Use the **Edit Form** command to manipulate the recently inserted edges to achieve the result shown in the image.

Note: The planar translation manipulator will be extremely useful.

Step 9: Use **Insert Point** to draw edges on a face. To insert the final two edges we need, we'll use the **Insert Point** command. Slightly different from Insert Edge, the **Insert Point** command will easily insert an edge by connecting two points together.

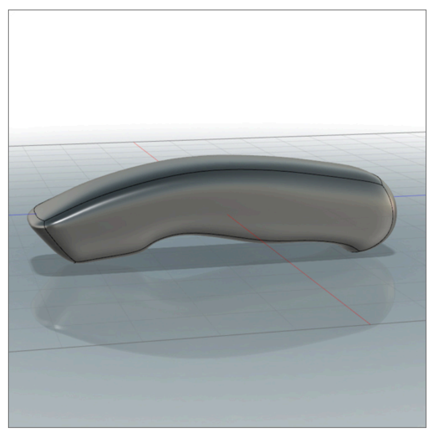
1. Click **Modify > Insert Point**.
2. Hover over the middle of the top edge shown until a **red circle** appears – this indicates the midpoint
3. **Click** and repeat for the edge directly beneath, located along the line of symmetry.
4. Ensure the **Insert Mode** set to **Simple**.
5. Click **OK**.





Step 10: Use **Edit Form** to move faces and edges until you have matched the T-Spline body to the profile of the Knife image.

1. Use the **Edit Form** command to manipulate the recently inserted edges (as well as the surrounding geometry) to get the T-Spline primitive to match as closely as possible.
2. When you are satisfied with the shape of the T-Spline body select Finish **Form** from the Sculpt Menu bar.

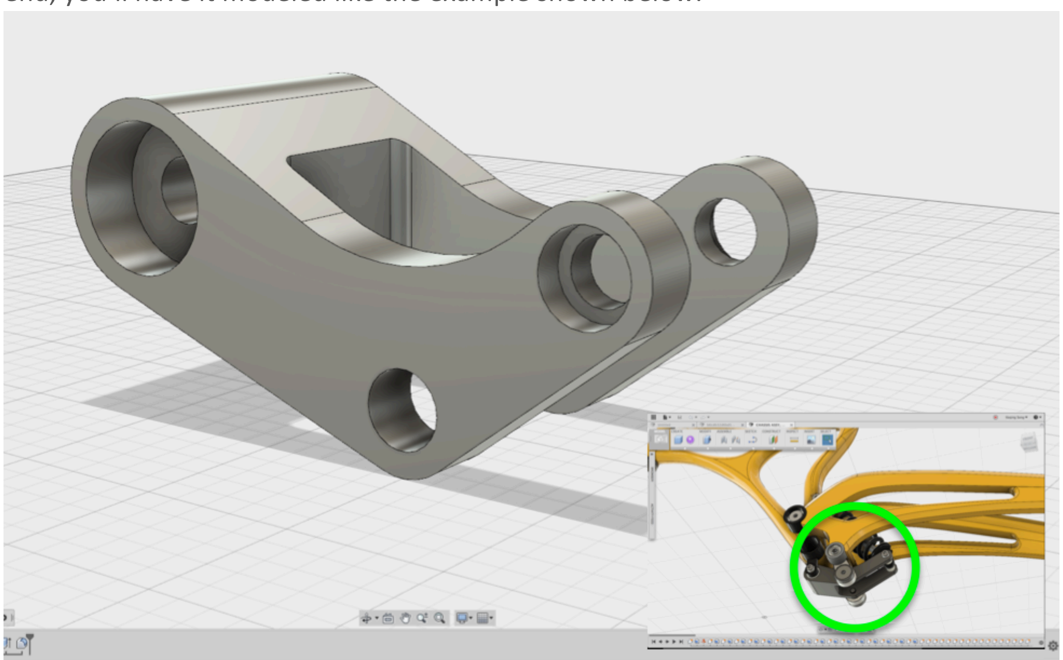


4.1: Modeling

3D Modeling is a key process of getting your ideas from a concept to a read-for-manufacture state, making it core foundation of the product development process. In Fusion 360, there are a couple different ways you can start a design. Chapter 02 and 03 showed you how to start with sketches and with sculpted bodies. This lesson will continue where the previous chapters left off and walk you through the fundamental modeling techniques based sketches, based on a sculpted body, explore these different design approaches, and learn tips and tricks along the way.

Lesson 1: Modeling based on Sketches

We'll be using a sketch of a mountain bike rocker arm to go through this lesson. At the end, you'll have it modeled like the example shown below.



Learning Objectives

1. Creating geometry based on sketches
2. Using sketch lines as reference
3. Using sketches to drive changes in geometry

Datasets Required

In Samples section of your Data Panel, browse to:

Fusion 101 Training > 04 – Modeling > **04_Model_from_sketch**

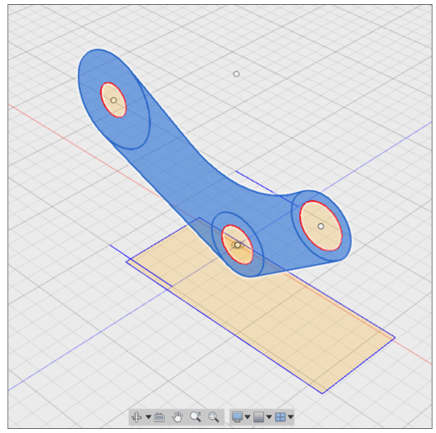
Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

Step 1: Select profiles - Let's start with this sketch of the rocker arm. We're going to use this to create a solid body.

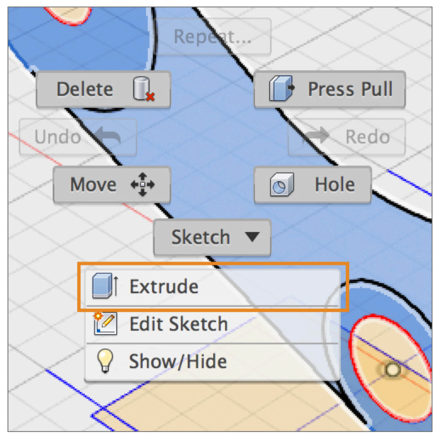
1. Hold down **Shift** and select the profiles shown in the image. Make sure that the 3 center holes are the only profiles not selected.

Note: If you are having trouble selecting certain profiles, use your mouse wheel and zoom in closer; this should make it easier to select.



Step 2: – Start the **Extrude** command

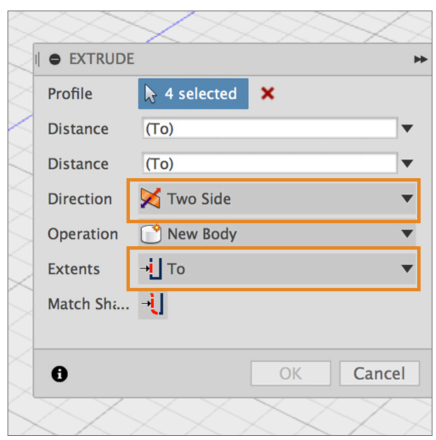
1. Right click on a selected area of the sketch and select **Extrude**. We're going to extrude the selected profiles.



Step 3: – Define the extrude options in the Extrude dialog box

1. Set Direction to **Two Side**.
2. Set Extents to **To**.

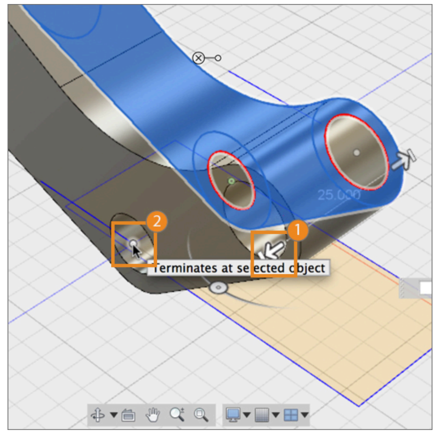
You should now see 2 arrows appear on your selected sketch profiles. We're going to use these arrows to define where we want the extrusion to go. This is especially useful when you have set geometry you can use as reference, much like our sketch here.



Step 4: – Set the distance for the left side

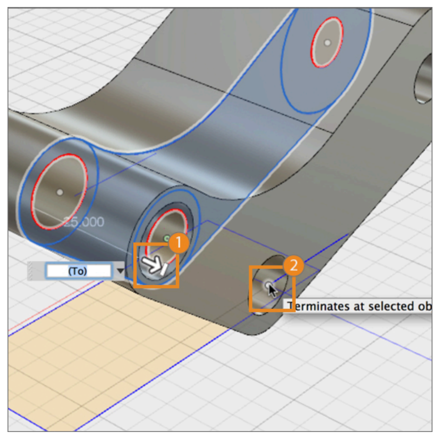
1. Click once on the **left arrow manipulator.**
2. Now hover over the **line sketch** on the left side and click on the **end point** as the extent you want to extrude to.

Note: When Extrude extent is set to "To", make sure you select the line sketch and not the rectangle sketch. When you've done this, the extrusion will automatically terminate at that point. This is selection really useful when you have reference geometry you want to use to create new geometry.



Step 5: – Set the distance for the right side

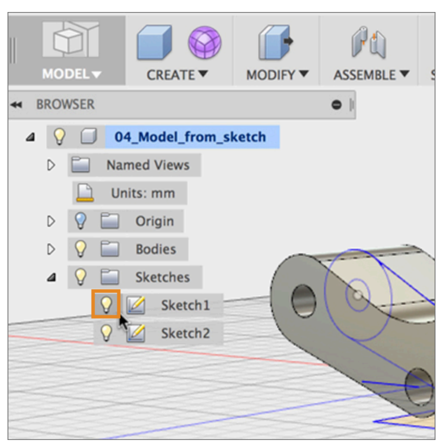
1. Repeat the same thing on the right side. Click once on the **Right arrow manipulator.**
2. Now hover over the **line sketch** on the left side and click on the **end point** as the extent you want to extrude to.
3. Click **OK** (or hit **ENTER**) to finish the command.



Step 6: – Turn sketch visibility back on

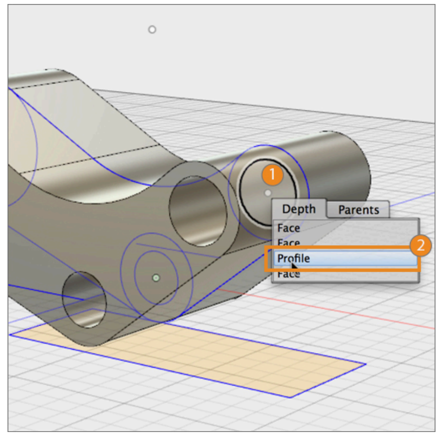
1. Let's go to the browser and within the Sketches folder, click the light bulb icon next to **Sketch1** to turn the visibility of that sketch back on.

The Visibility of a sketch is automatically turned off after a modeling action has been committed based on that specific sketch.



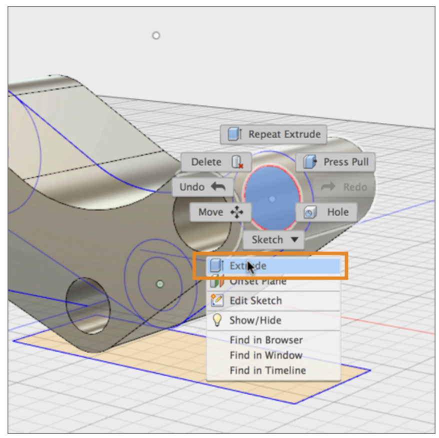
Step 7: – Select a sketch profile behind an obstruction

1. Now we need to select the circle profile to make an extrusion. If you find yourself in this situation where it is hard to select a specific geometry because it is being obstructed, then **hover** over the profile, **click** and **hold**. After a few seconds, you'll see a dialog menu show up, letting you choose what exact entity you'd like to select.
2. Select **Profile**. You should now see the circle profile selected.



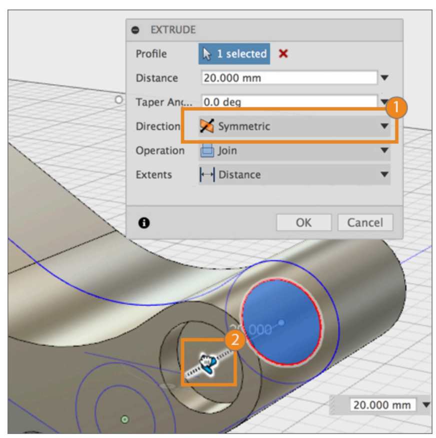
Step 8: – Extrude the circle profile

1. Right click on the selected circle profile and pick **Extrude**. We're going to create this command again to create new geometry.



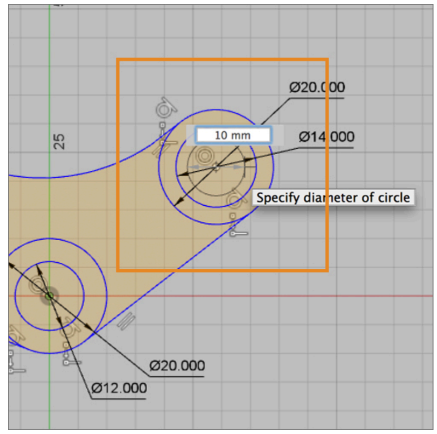
Step 9: – Join the new extruded body

1. Set the Direction to **Symmetric.** Leave Operation as Join.
2. Drag the arrow to **20 mm.** Click OK to finish.



Step 10: – Sketch a new circle profile

1. Right click on **Sketch1** in your browser and select **Edit Sketch.** This will take you back into the first sketch and create more geometry. Notice that the timeline reflects us going back to this sketch item as well.
2. Go to the Sketch drop-down menu and select **Circle** > **Center Diameter Circle**
3. Create a circle snapped the center with a diameter of **10 mm.** Click **Enter** twice to commit. Click **Stop Sketch** to exit out of sketch mode.



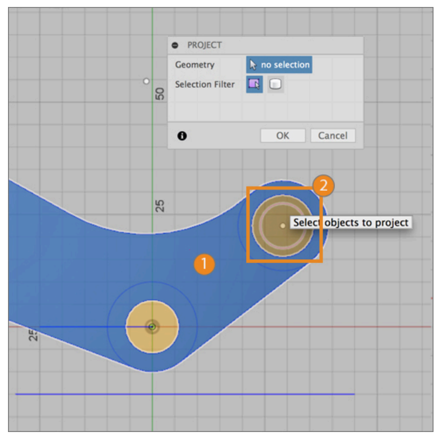
Step 11: – Project the circle onto a new face

Go to the Sketch drop-down menu and select **Project / Include > Project**.

1. First select the outer face,
2. Then select the new circle sketch we just created.

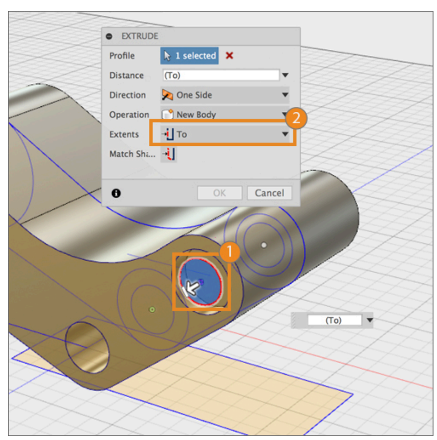
   Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

You should now see that the circle is now project onto the outer face of the model.



Step 12: – Extrude the circle as a cut

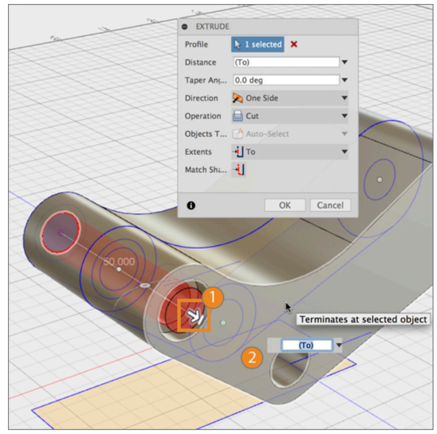
1. Let's select the project circle profile, right-click and choose **Extrude**.
3. In the command dialog, change the Extents to **To**.



Step 13: – Make the cut

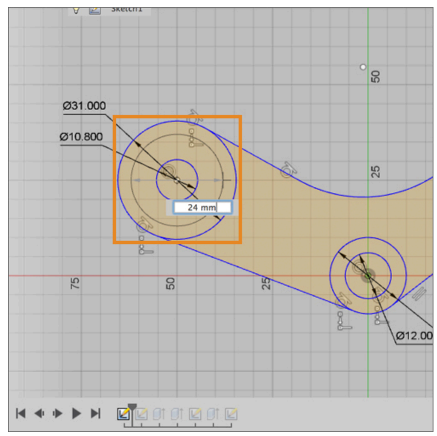
1. Click on the **Arrow Manipulator** to activate the extrusion.
2. Rotate the model to the other side so that we see the other face we want to extrude to. Click on that face and click OK to finish.

You should now see a cut made through the entire width of the model. This cut is now tied to the original circle sketch, thus allowing us to easily make dimension changes moving forward.



Step 14: – Sketch a new circle profile

1. We’re going to move to the other side of the rocker model. Right click on **Sketch1** in your browser and select **Edit Sketch.**
2. Go to the Sketch drop-down menu and select **Circle** > **Center Diameter Circle**
3. Create a circle snapped the center with a diameter of **24 mm.** Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

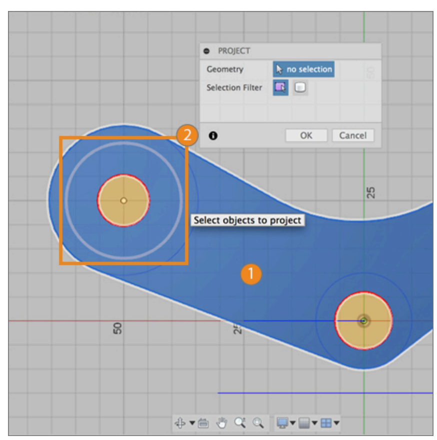


Step 15: – Repeat Project sketch workflow

Go to the Sketch drop-down menu and select **Project / Include > Project**.

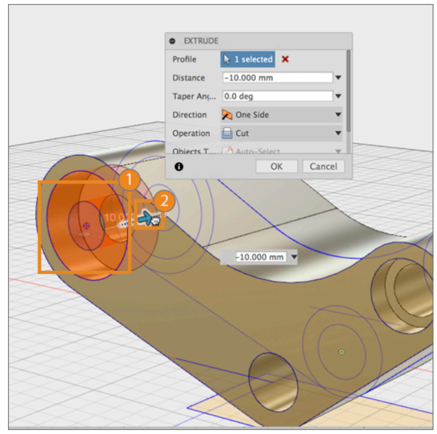
1. First select the outer face,
2. Then select the new circle sketch we just created.
   Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

You should now see that the circle is now project onto the outer face of the model.



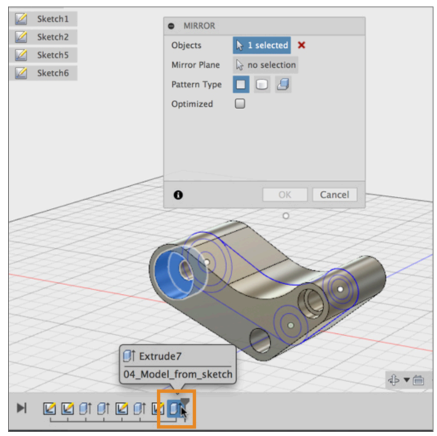
Step 16: – Extrude the circle as a cut

1. Select the area between the project circle and the smaller circle, right-click and select **Extrude**.
2. Drag the **Arrow Manipulator** to **-10 mm**. Click OK to finish.



Step 17: – Mirror the cut on the other side

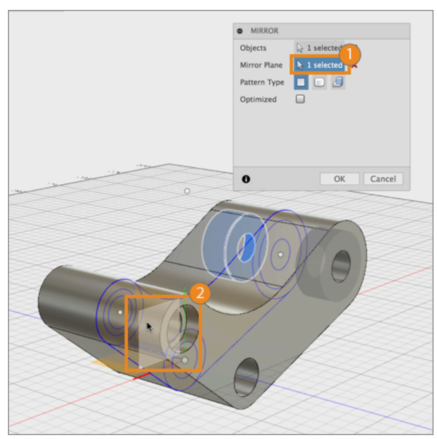
1. Now that we've made this cut, let's mirror it on the other side. Go to the Create drop-down menu and select **Mirror**.
2. Go to timeline and select the last extrusion as the object we want to mirror.



Step 18: – Mirror the cut on the other side

1. Click the **Mirror Plane** option to activate which mirror plane to use.
2. Select the origin plane that is in the middle of the model. Click OK to finish.

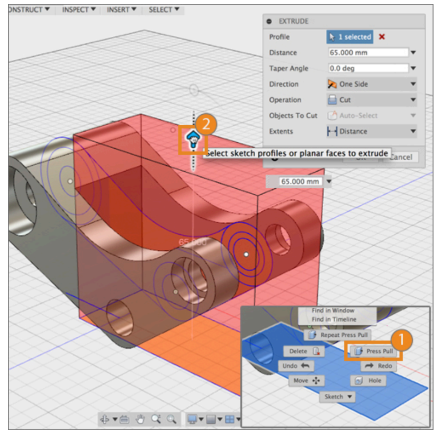
Note: If you have having trouble selecting the origin plane, remember to zoom out or click and hold to get the option to choose what you'd like to select.



Step 19: – Use Press-Pull to cut

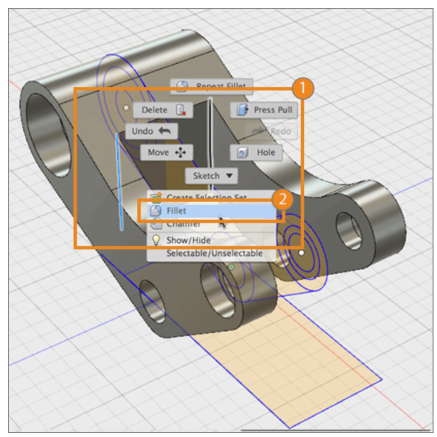
1. Now let's select the rectangle sketch at the bottom, right-click and use **Press-Pull** (on the right of your cursor)**.**
2. Drag the **Arrow Manipulator** through the model so the cut goes all the way through.

Notice that Press-Pull automatically turned into an Extrude command. This is the nature of Press-pull – it adapts to what the action is gives you a predictable outcome. If you had selected an edge and decided to use Press-Pull, it'll automatically turn into a Fillet.



Step 20: – Add a couple fillets

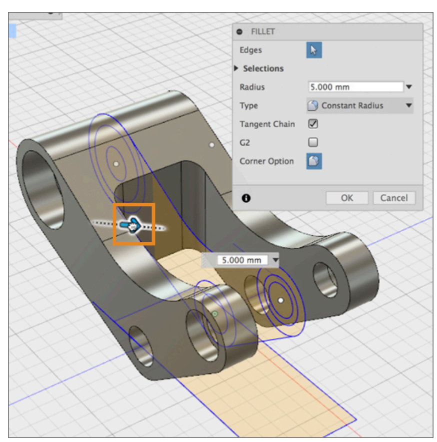
1. Let's finish the model by adding a couple of fillets on the inside edges. Select them by holding **Shift**.
2. Right-click and select **Fillet**.



Step 21: – Add a couple fillets

1. Drag the **Arrow Manipulator** to **5 mm**

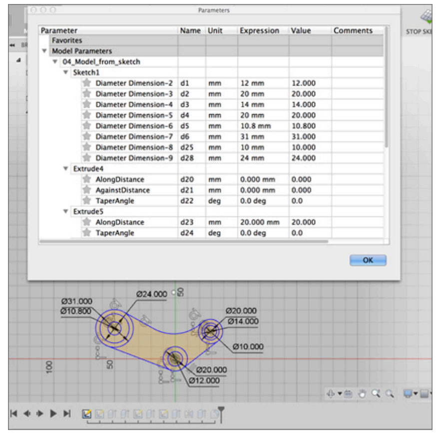
   Click OK to finish.



Step 22: – Making changes to your model

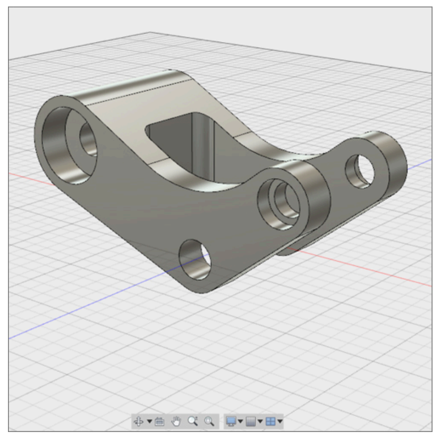
Since all the extrusions, mirror, and fillets are based on the original sketches, we can go back to **Sketch1** and **Sketch2** and any time and make dimension changes without needing to change each downstream feature or worry about any of them failing.

You can also go to the Modify drop-down menu and select **Change Parameters**. This will allow you to change any dimension in a chart form, assign custom names, set values or functions, and see the changes update instantly.



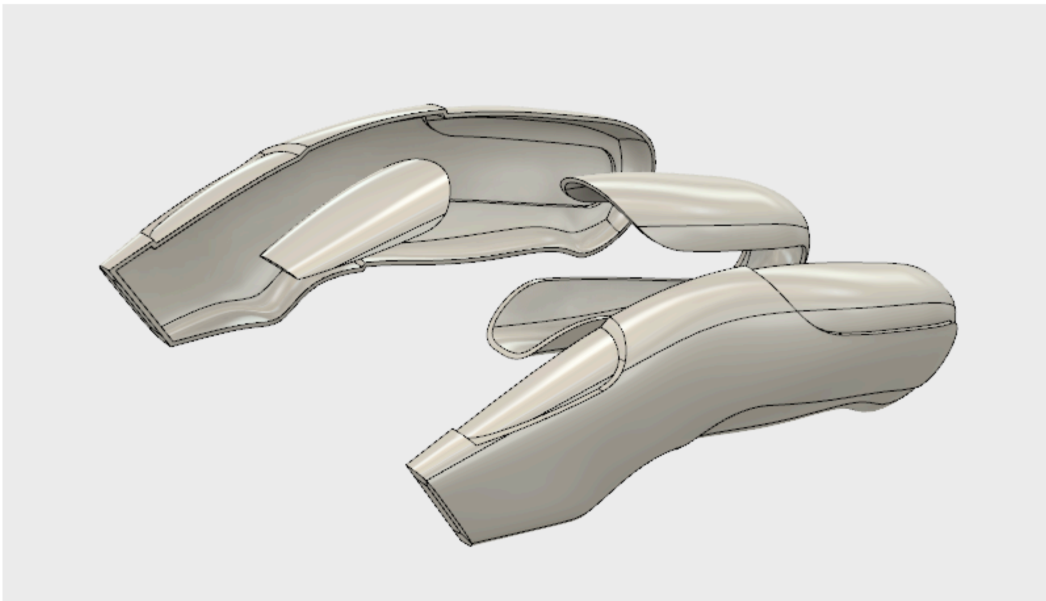
Step 23: – Model complete!

Congratulations, you have completed this lesson on how to model based on sketches! You're ready to move on to the next lesson.



Lesson 2: Modeling based a sculpted body

Now that you've seen how to use model based on sketches, we'll take it one step further and go through how to take advantage of sketches and model geometry based on an existing sculpted body. We'll be using a sculpted utility knife model. At the end of this lesson, you'll have gone from a single model to 4 separate pieces like the example shown below.



Learning Objectives

1. Using sketches and planes to split bodies
2. Using bodies to join and cut other bodies

Datasets Required

In Samples section of your Data Panel, browse to:

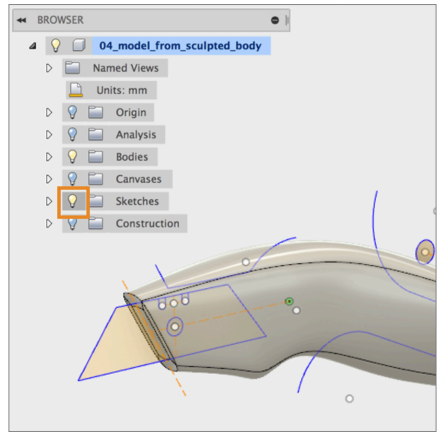
Fusion 101 Training > 04 – Modeling > **04_Model_from_sculpted_body**

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

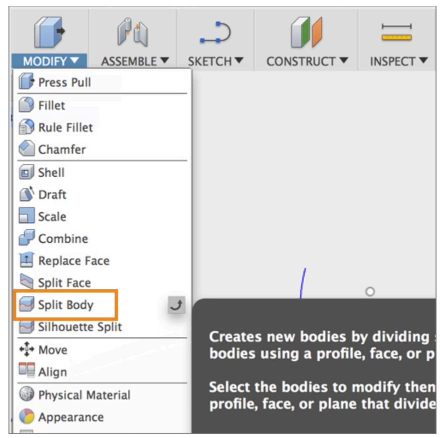
Step 1: – Turn on Sketch visibility

1. Let's start by going to the browser and locating the **Sketch** folder. Click the light bulb to turn on sketch visibility. You should now see a number of sketch lines and profiles appear on your model.



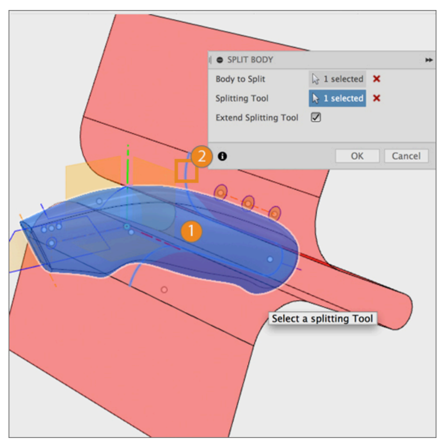
Step 2: – Split the model into 2 bodies

1. In order to create the handle grips, we're going to use a couple of the sketch lines to split the model into 3 separate bodies. Go to the Modify drop-down menu and select **Split Body**.



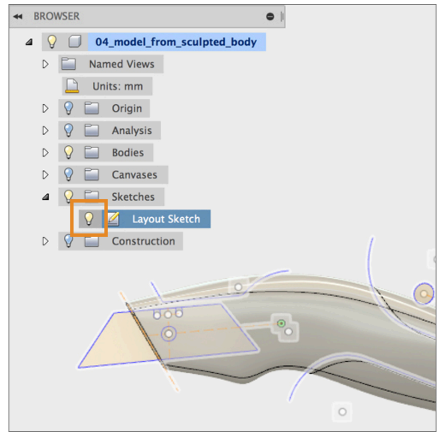
Step 3: – Make the split

1. Select the **body** as the **Body to Split.**
2. Click on **Splitting Tool** to activate the selection. Select the **long grip line sketch** as the **Splitting Tool**. Click OK to finish the split.



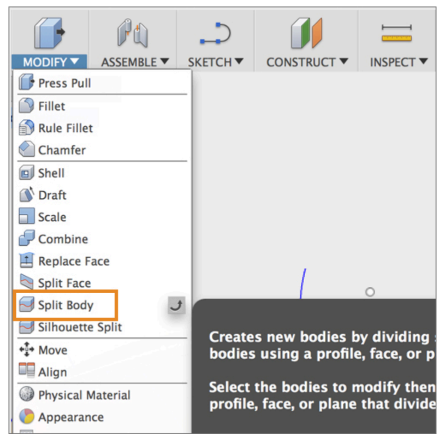
Step 4: – Repeat previous step

2. Let's repeat the previous step and make another split – this time it'll be for the grip at the top of the knife model. Go back to the Sketch folder and turn on **Layout Sketch** visibility.



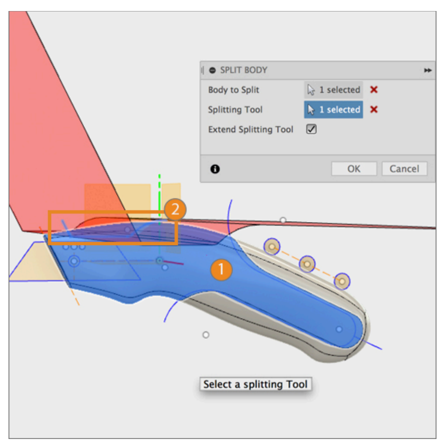
Step 5: – Split the model into 3 bodies

2. Go to the Modify drop-down menu and select **Split Body**.



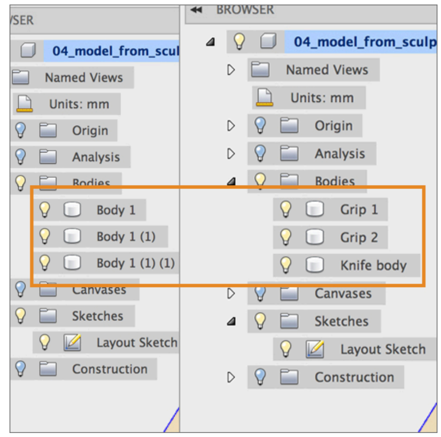
Step 6: – Make the split

3. Select the **body** as the **Body to Split.**
4. Click on **Splitting Tool** to activate the selection. Select the **short grip line sketch** as the **Splitting Tool**. Click OK to finish the split.



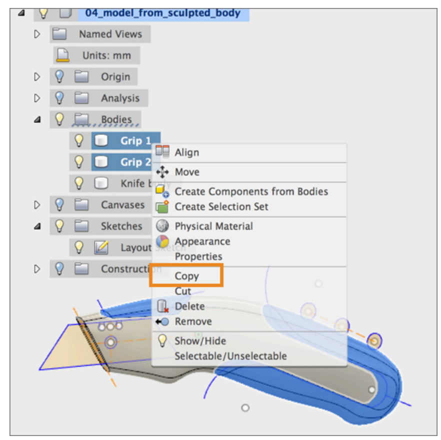
Step 7: – Rename bodies

3. In the Bodies folder, you'll notice that there are now 3 bodies. Let's rename them. **Double click** on the body and rename:
   Body 1 to **Grip 1**
   Body 1 (1) to **Grip 2**
   Body 1 (1) (1) to **Knife body**



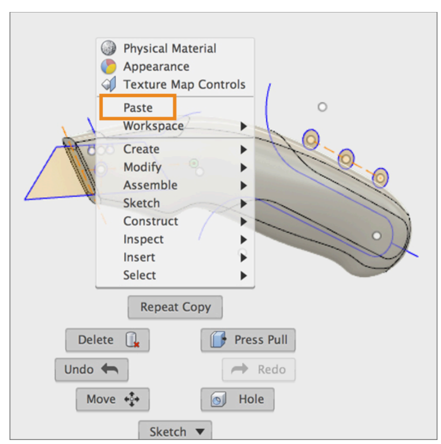
Step 9: – Copy and paste bodies

1. We're now going to perform a modeling technique. Select **Grip 1** and **Grip 2**, right-click and select **Copy.**



Step 10: – Copy and paste bodies

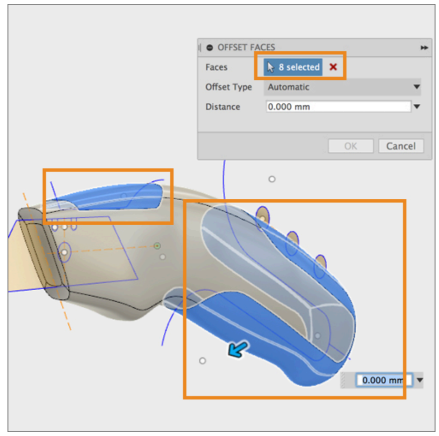
1. Click somewhere on the canvas, right-click to activate the marking menu, then select **Paste.**
2. You should now see 2 more bodies appear in your Bodies folder called **Grip 1 (1)** and **Grip 2 (1)**. Click OK to finish the paste action.



Step 11: – Offset bodies with Press-Pull

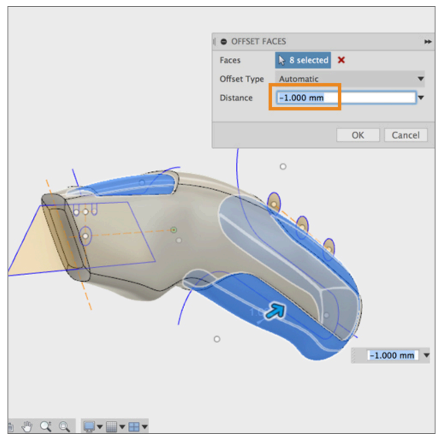
1. Turn off visibility of **Grip 1** and **Grip 2.** We're going to work on the 2 new grip bodies.

2. Right-click somewhere in canvas and select **Press-Pull**. Select all the faces of **Grip 1 (1)** and **Grip 2 (1)**. Make sure to rotate around and get all the faces. Hold **Shift** to add onto the selection. You should have a total of 8 faces selected.



Step 12: – Offset bodies with Press-Pull

1. Enter an Offset Distance of **-1 mm.** Click OK to finish. You should see that the faces have been successfully offset.

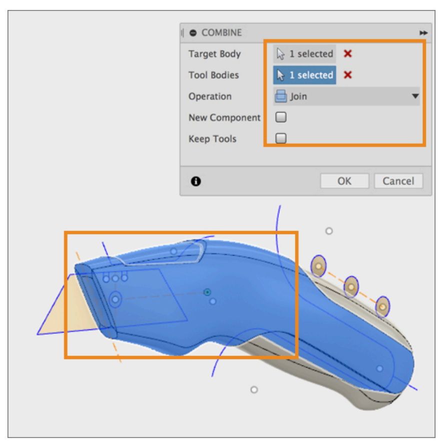


Step 13: – Join bodies with Combine tool

1. Let's now join these offset bodies to the knife body so that they are part of a whole body. Go to the Modify drop-down menu and select **Combine.**

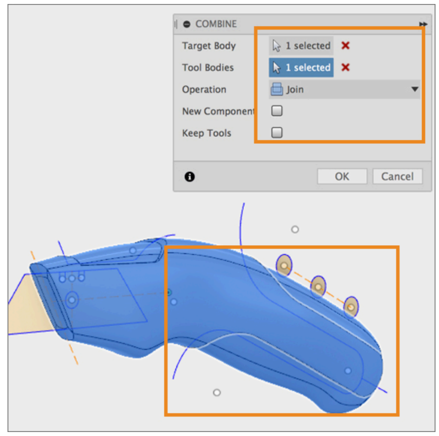
2. In the command dialog, set:
   Knife Body as the **Target Body**,
   Grip 2 (1) as the **Tool Body,**
   Join as the **Operation,**
   Uncheck Keep Tools,

3. Click OK to finish the operation.



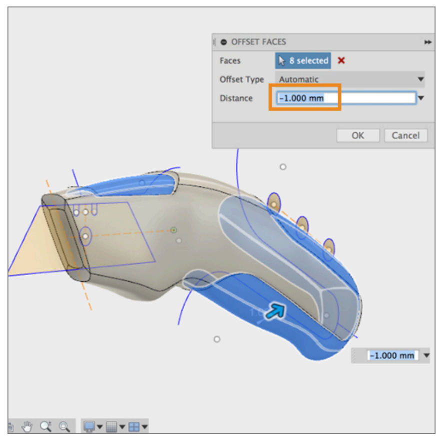
Step 14: – Repeat Combine Join

1. Repeat the previous step, but this time; join **Grip 1 (1)** to **Knife Body**. Click OK to finish the operation.
2. You should now only see **Grip 1, Grip 2,** and **Knife Body** in your Bodies Folder in the browser.



Step 15: – Cut Bodies with Combine tool

1. Now let's use the new knife body to cut the original grips so that they fit exactly right. Turn on visibility of **Grip 1** and **Grip 2.**
2. We're going to focus on **Grip 1** first.

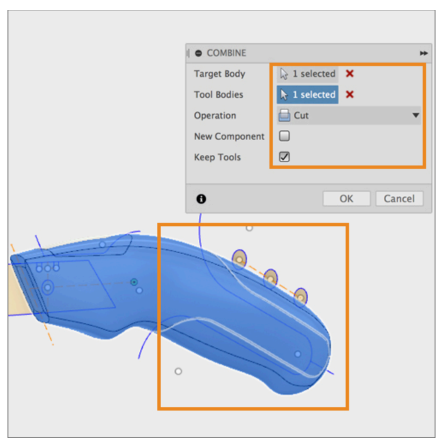


Step 16: – Cut Grip 1 with Combine tool

1. Go to the Modify drop-down menu and select **Combine.**

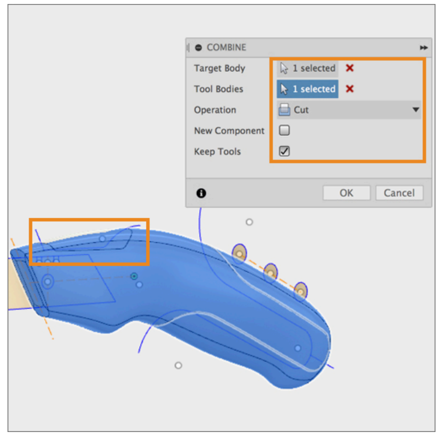
2. In the command dialog, set:
   Grip 1 as the **Target Body**,
   Knife Body as the **Tool Body,**
   Cut as the **Operation,**
   Check Keep Tools,

3. Click OK to finish the operation.



Step 17: – Repeat Combine Cut for Grip 2

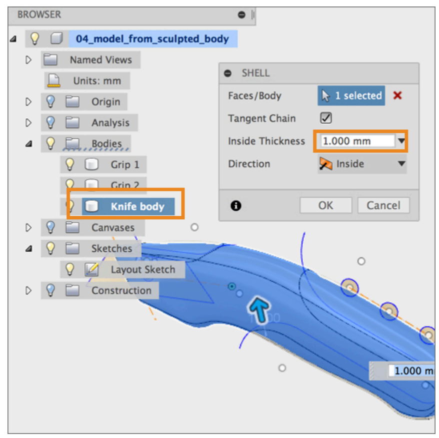
1. Repeat the previous step, but this time, cut **Grip 2** using **Knife Body**. Click OK to finish the operation.



Step 18: – Shell the knife body

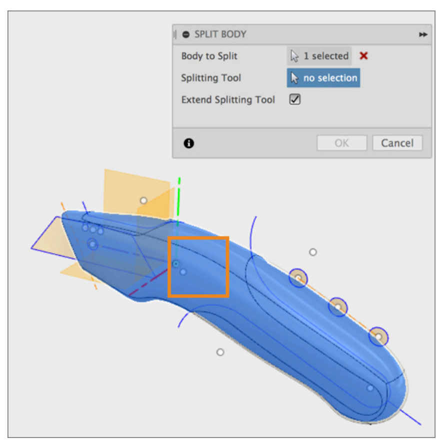
1. Now that we have our grips modeled, let's shell the inside of the knife body. Go to the Modify drop-down menu and select **Shell.**
2. Select **Knife Body** from the browser. Make sure that the shell thickness is **1 mm.** Click OK to finish.

Note: In the Shell command, selecting bodies from the browser will only shell the inside of those bodies. Selecting the face of a body will shell remove that face and shell the inside.



Step 19: – Split Knife Body into 2 pieces

1. Now that we have our knife body shelled, let's split it into 2 pieces. Go to the Modify drop-down menu select **Split Body.**
2. Select **Knife body** as the Body to Split.

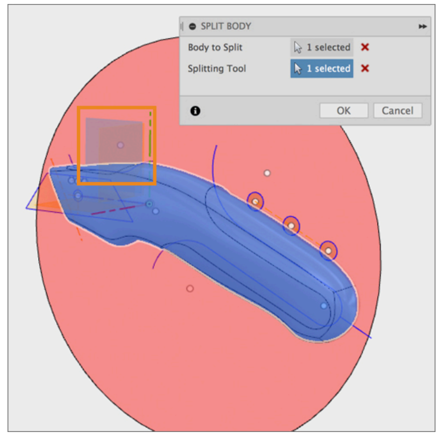


Step 20: – Split Knife Body into 2 pieces

1. Click on **Splitting Tool** to activate the selection

2. Select the plane that cuts down the middle of the **Knife body.** If you have trouble selecting the plane, zoom out until you can select it.

3. Click OK to finish the operation.



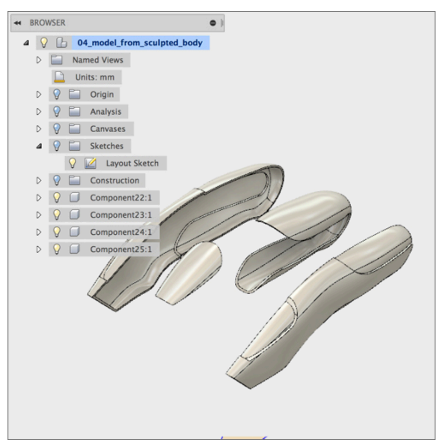
Step 21: – Convert Bodies to Components

1. You should now see 2 knife bodies as well as the 2 grips in your Bodies folder. As the last step, let's convert these into components.
2. Select **all 4 bodies**, right-click and select **Create Components from Bodies.**

Step 22: – Lesson complete!

1. You have completed the lesson!

2. You can now drag the components apart and see all the work we did around the grips and the knife body.



Overview

Fusion 360 organizes and manages data using a centralized, cloud-based, collaboration platform. This enables designers and engineers to work more easily and efficiently together. Use this powerful and secure set of tools to dramatically improve the way you design, visualize, simulate, and share your work, on demand.

Learning Objectives

In this section you will learn how to:

- Create Fusion 360 designs and save versions.
- Create and manage Fusion 360 Group Projects.
- Add and remove users from Fusion 360 Group Projects.
- Find, view, and manage files within Fusion 360's collaborative web browser environment and the in-application dashboard.
- Access Fusion 360 data from a mobile device.
- Import and export files from Fusion 360.
- Publicly share data with external stakeholders.

If you like to watch the video to this tutorial, click here: **Launch Video**

Tips for this exercise:

- To complete the mobile section of the tutorial, install the Autodesk 360 App to your mobile device.
- Partner up with a friend who also has Fusion 360. There is an exercise where you have the option to grant access to your project.
- We recommend installing Google Chrome to best utilize the collaborative capabilities of Fusion 360 (the in-browser 3D viewer is not yet supported for IE, Firefox, and Safari).

Create and Manage Fusion 360 Group Projects: In this section you use data from an existing group project to create, setup, and manage a new group project in Fusion 360. You control who has access to this project. You create a new design, create multiple versions and determine where your new design will be stored. Once finished you then share a specific version publicly so that those not part of your project can access, inspect and review.

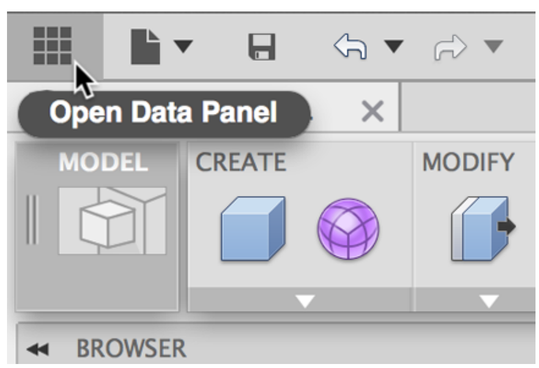
Fusion 360 allows you to securely access data from anywhere. You also control who else has access. The main way Fusion 360 controls access is by using "group projects." Group projects are the control mechanism for how you define who has access to specific sets of information. Group projects ensure that only the correct collaborators have access to your data.

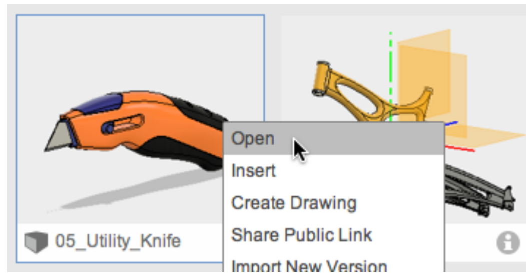
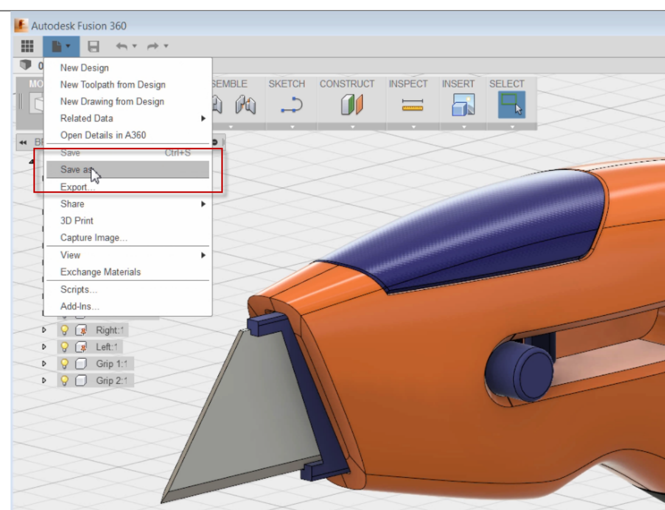
Group projects are like folders in that they allow you to organize partitions of data, but they have the unique ability within Fusion 360 to explicitly define who may access data within it. Projects are unique locations where teams keep all related information in one shared place. People can share and access design data, discuss challenges and successes, and stay current with project activities. Each project has its own data, people, calendar, and wiki.

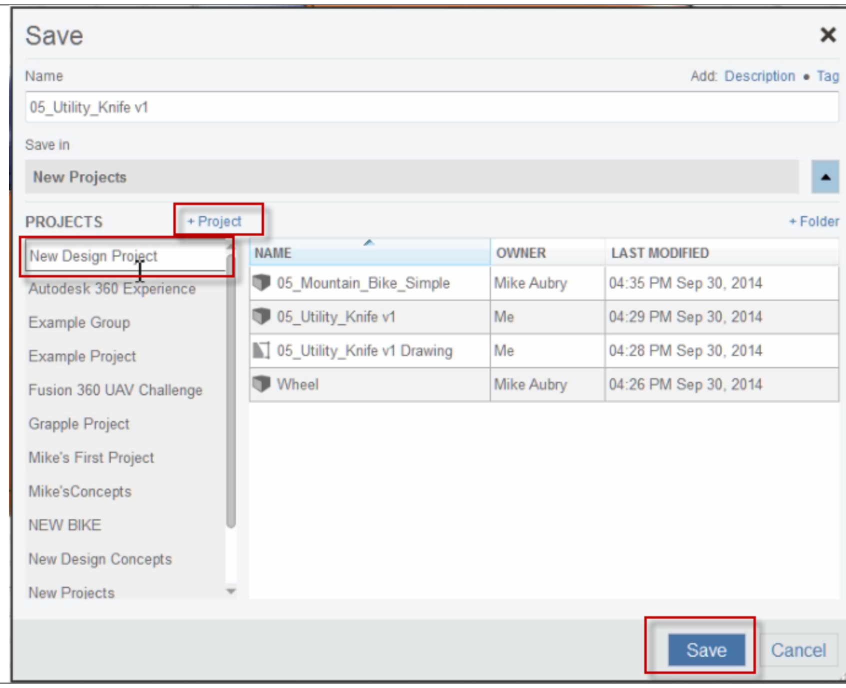
Common Configurations:

Some companies like to use group projects to separate between different jobs or work orders. Others prefer to use group projects to separate between different customers. Students commonly segment by specific assignments or by class. Whatever your configuration Fusion 360 offers flexibility to adjust and adapt as you evolve in your requirements.

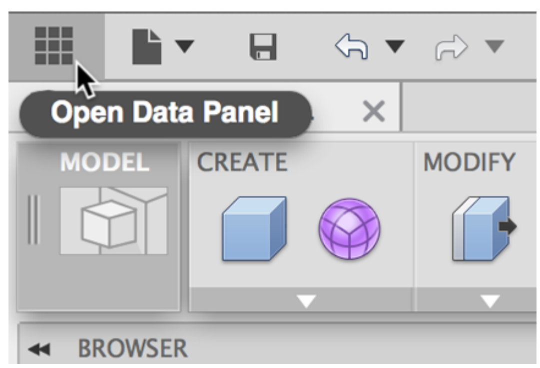
Open Fusion 360 design file: In this section you will open the introductory design file.

Step 1 – Open the Data Panel 1. Open the Data Panel by clicking on the icon located at the top left of the menu bar. 2. The Data Panel will slide open.	

Step 2 – Open the design *In this module we will be using the* ***05_Utility_Knife.f3d*** *file to complete the exercise. If you haven't set up a new project and uploaded the necessary designs, please follow the steps in the Introduction module.* 1. At the top left of the Data Panel, select the project where you uploaded the **05_Utility_Knife.f3d** file. 2. Navigate to this design and either **double-click** or **right-click** and select **open.** 3. When the design has opened in your modeling window, click on the icon to close the Data Panel.	
Step 3 – Start the Save As command 1. Select **File > Save As.**	

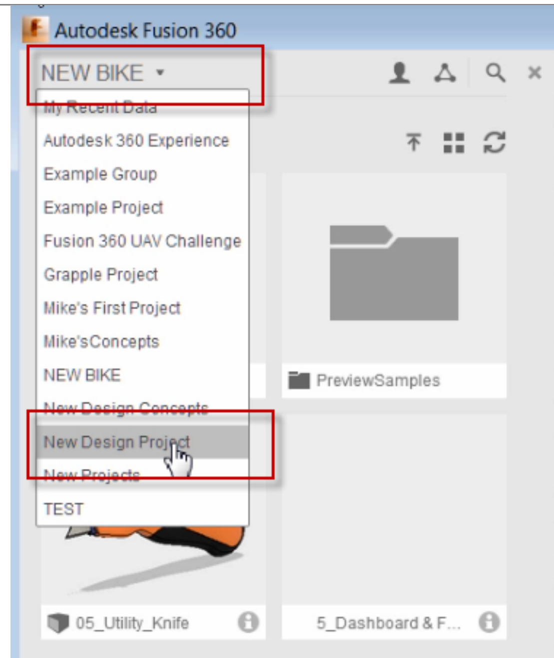
Step 4 – Create a new project 1. Click **+Project** to create a new project. 2. Name the project "**New Design Project**". 3. Select **Save**.	

Upload data to a project: In this section, you upload data from your local drive to the project you just created.

Step 1 – Open the Data Panel 1. Open the Data Panel by clicking on the icon located at the top left of the menu bar. 2. The Data Panel will slide open.	

Step 2 – Set the active project

1. Click on the name of the active project.
2. Select **New Design Project** from the list. This is the project you created in the previous step.

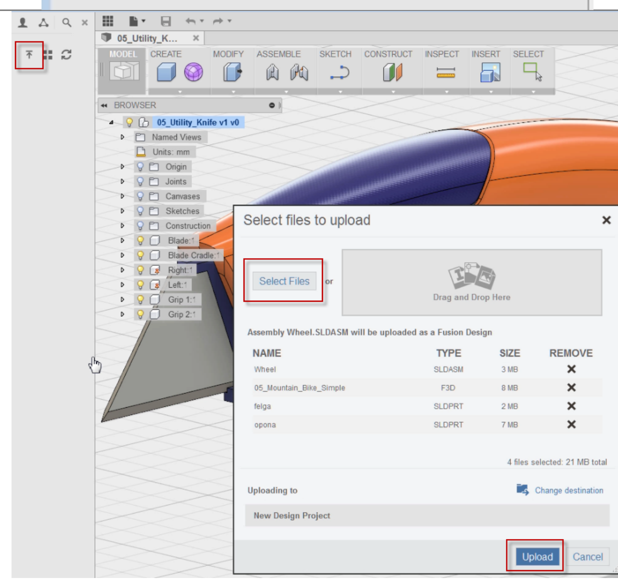


Step 3 – Upload data

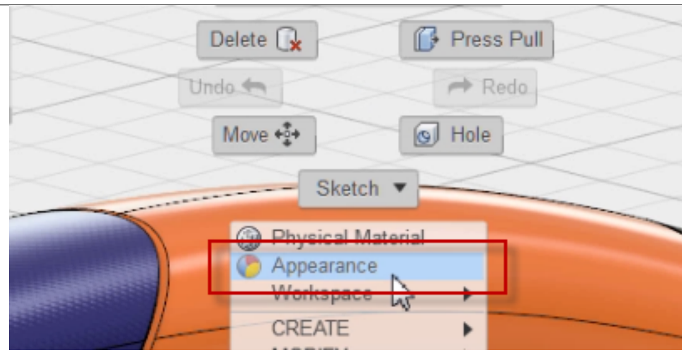
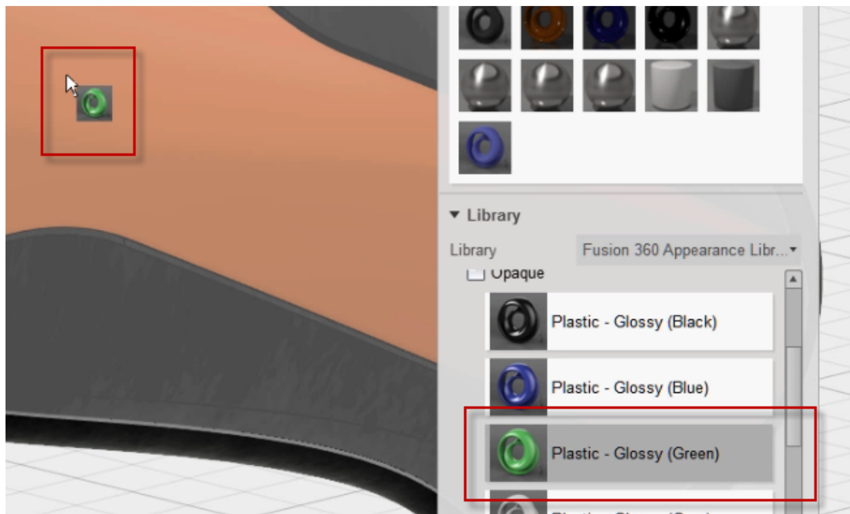
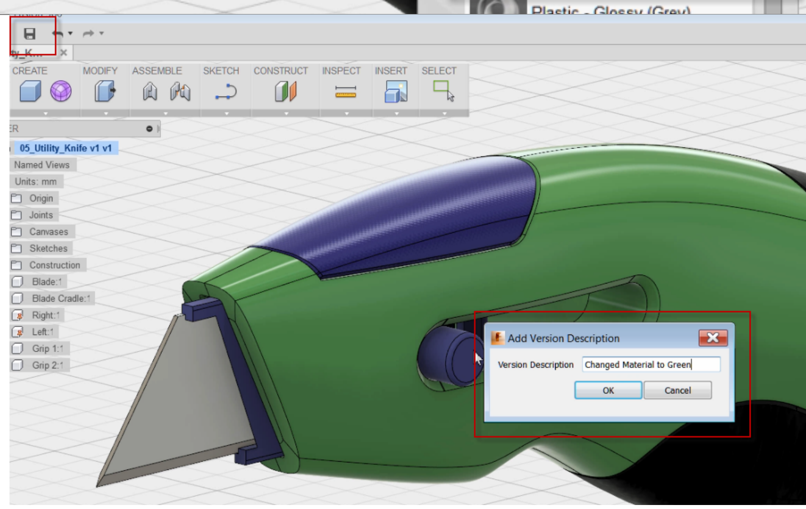
Fusion 360 translates data from many sources including Autodesk Inventor, Dassault SolidWorks, and PTC Creo.

1. Click **Upload**.
2. Click **Select Files** and navigate to the Chapter 5 dataset and upload:
   - 05_Mountain_Bike_Simple.f3d
   - Wheel.SLDASM
   - Flega.SLDPRT
   - Opona.SLDPRT
3. Click **Upload**.

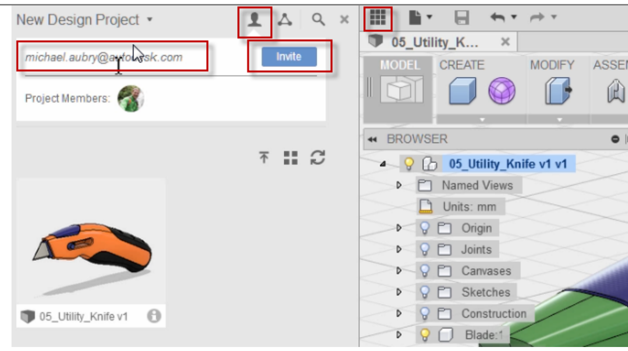
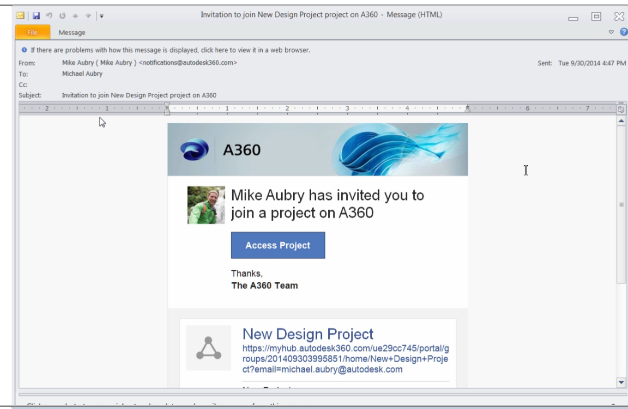
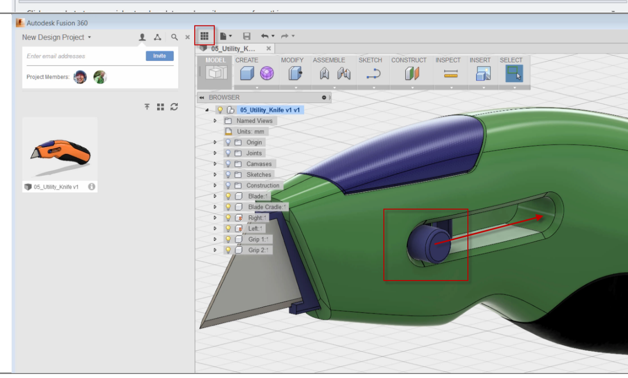
Uploading will continue in the background while you work.



Create new versions: In this section, you make a change to a design then save it to create a new version.

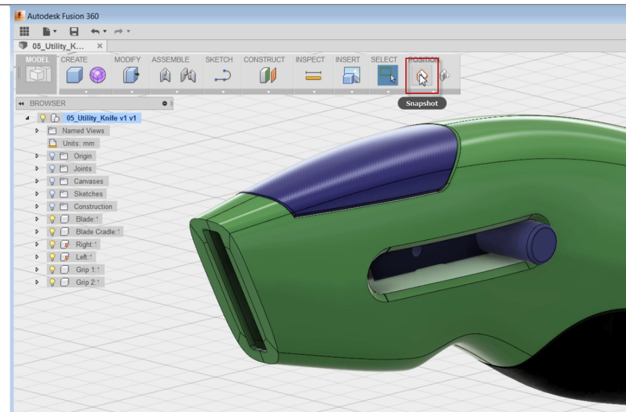
Step 1 – Start the Appearance command 1. Right-click and select **Appearance**.	
Step 2 – Change the appearance 1. Select **Plastic > Opaque > Plastic – Glossy (Green)**. 2. Drag and drop the appearance over the orange cover. 3. Click **Close** to dismiss the appearance command.	
Step 3 – Create a version Fusion 360 manages and stores each version you create. Make a change to an existing model and save it as a new version. 1. Click **Save**. 2. Enter the comment: “**Switched material from orange to green.**” 3. Click **OK**. We will explore where this new version is stored shortly.	

Add a user to your project: In this section you will open the introductory design file.

Step 1 – Add a project member Fusion 360 allows you to control who has access to design and other data within projects. Invite another collaborator to access your project. 1. In the Data Panel, click the **Project Members** button (the head shaped icon). 2. Enter an email address of another Fusion user who you would like to have access. If you don't have one feel free to use michael.aubry@autodesk.com. 3. Click **Invite**.	
Step 2 – Confirmation email 1. Once added, the user you will receive an email notifying him or her of their addition.	
Step 3 – Make a design change 1. Drag the slider to the right to retract the blade.	

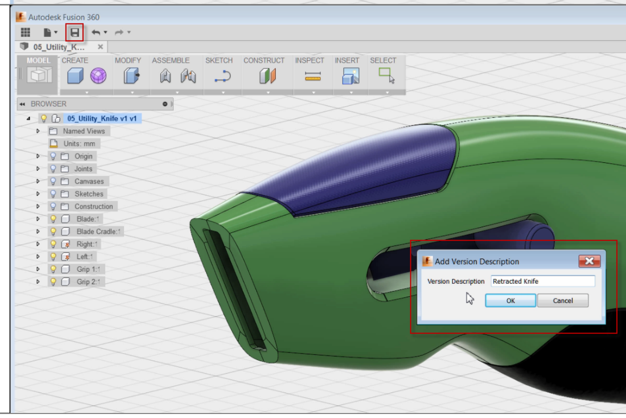
Step 4 – Create a snapshot

1. Click **Position > Snapshot** to create a snapshot to maintain the position.



Step 5 – Save the design

1. Click **Save** to save a new version.
2. Add a comment: “**Retracted knife**”.
3. Click **OK**.
4. **Close** the utility knife design.

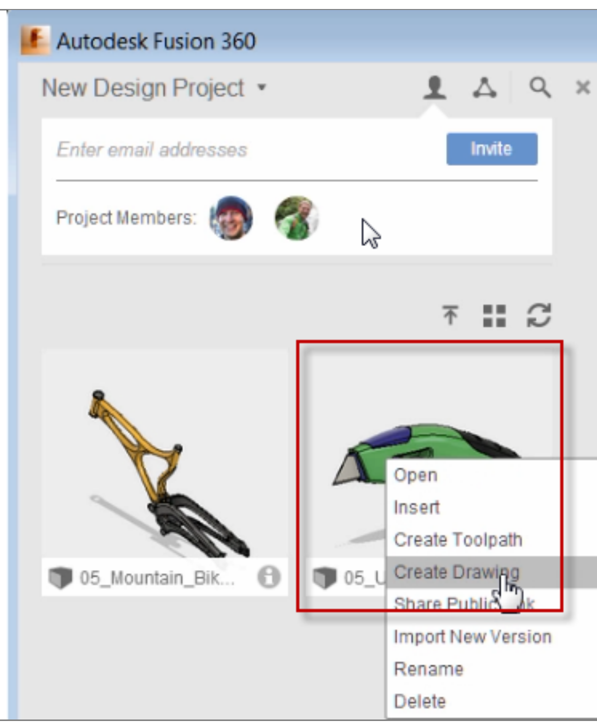


Create a referenced document: In this section, you create a drawing from the utility knife. Fusion maintains the relationship between the design and the drawing.

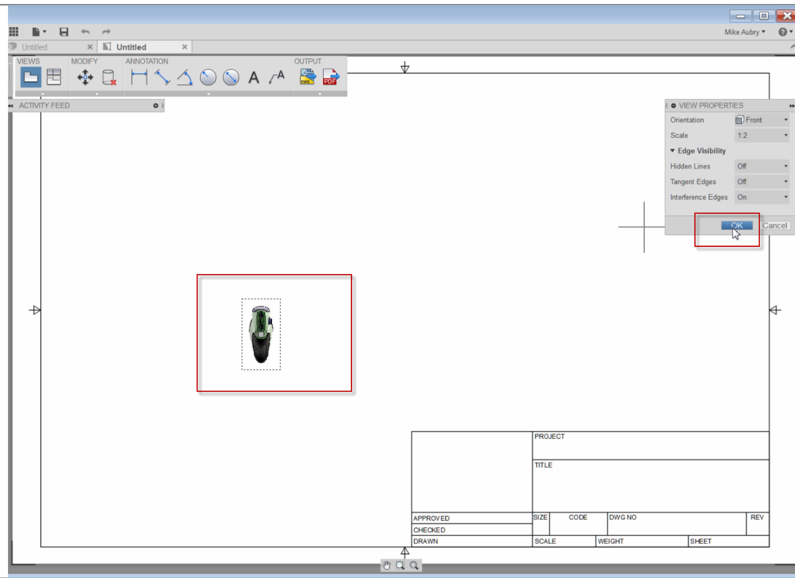
Step 1 – Create a drawing

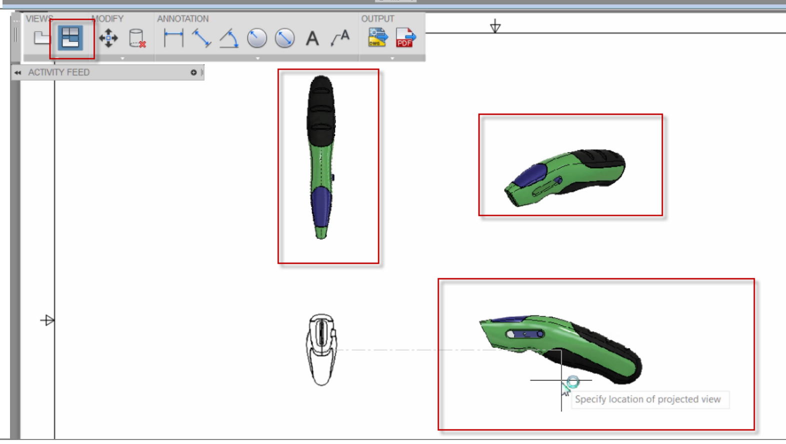
Fusion 360 maintains associative relationships between drawing and design documents. What this means is if you change the design the downstream drawing will be updated as well.

1. In the Data Panel, right-click on the **05_Utility_Knife** design.
2. Click **Create Drawing.**



<table>
<tr>
<td>Step 2 – Place the base view

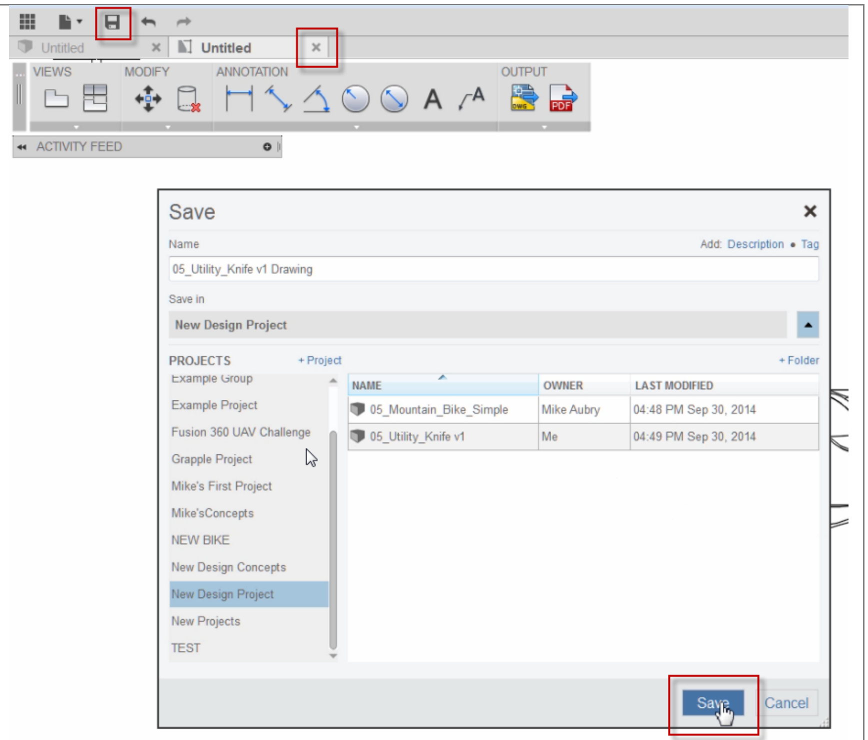
1. Left-click on the drawing sheet to place the base view.
2. Click OK to create the view.</td>
<td>
</td>
</tr>
<tr>
<td>Step 3 – Add projected views

1. Click View > Projected View.
2. Click on the base view.
3. Click in the three areas shown to create top, right, and isometric views.
4. Right-click and click OK to create the views.</td>
<td>
</td>
</tr>
</table>

Step 4 – Save and close

1. Click the **Save** icon.
2. Click **Save** to accept the default name and project.
3. Close the drawing.

You will see the association in a later step.



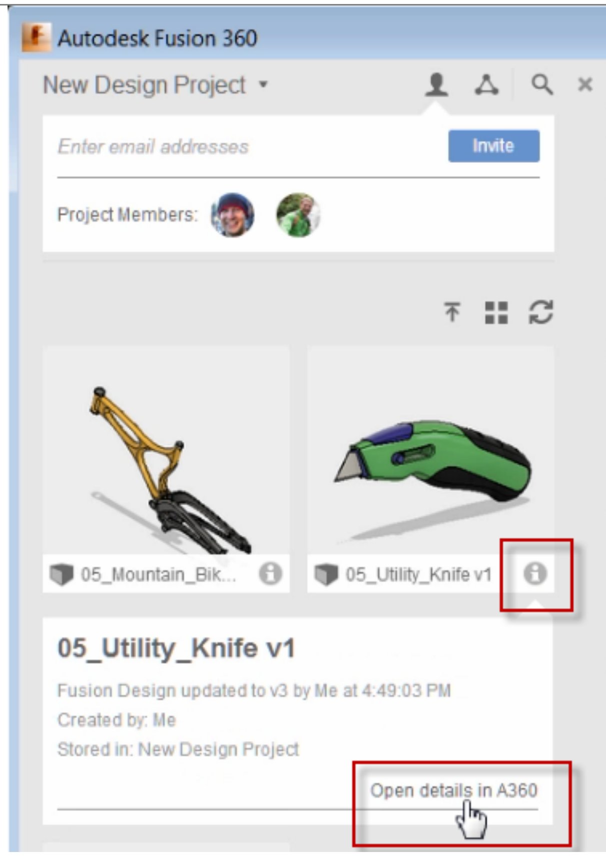
Access data in a web browser: Use Autodesk A360 to access your data in a web browser.

Step 1 – Access Autodesk A360

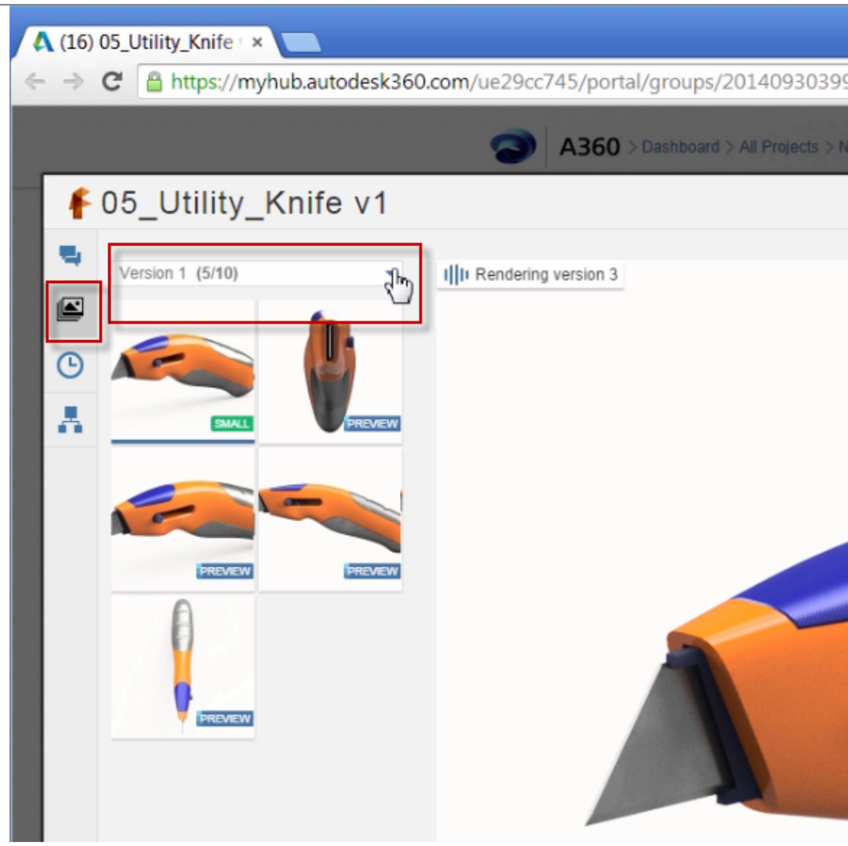
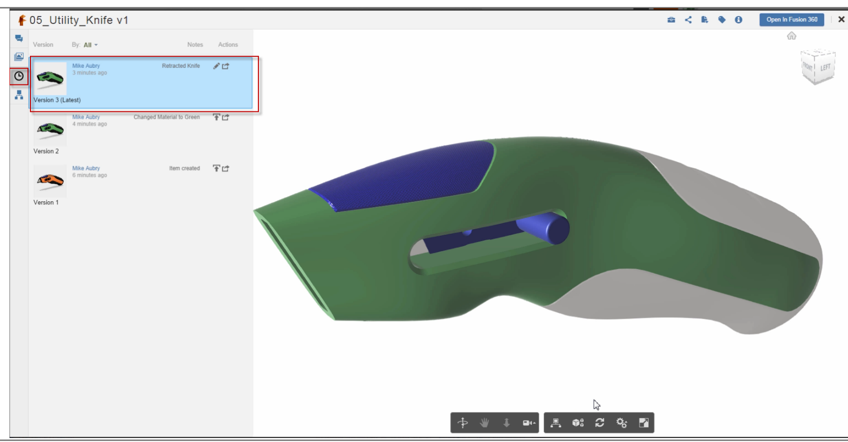
Fusion 360 allows you to manage data centrally through your web browser. Open your design in a web browser.

1. In the Data Panel, click the "**i**" icon on the 05_Utility_Knife to display details about the design.
2. Click **Open Details in A360**.

Suggestion: Consider using Google Chrome for this exercise as your default browser. Some features may not yet be supported for Safari, Firefox and Internet Explorer.

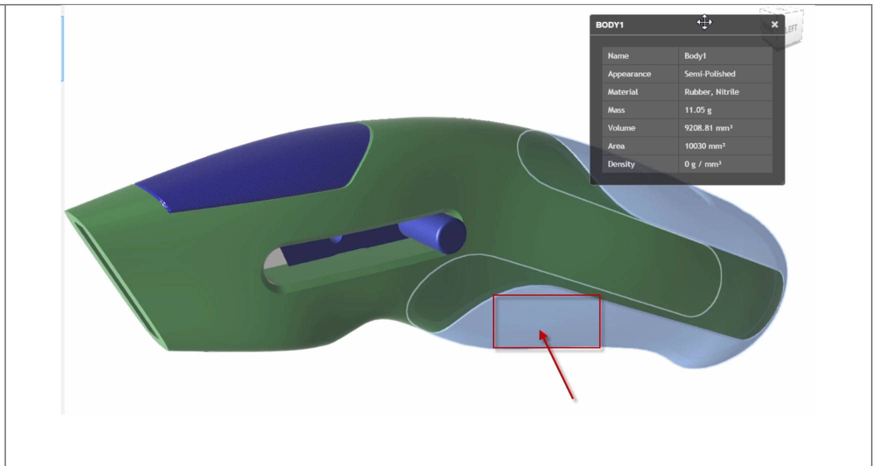


Display versions and use the viewer: In this section, you use the viewer to display the different versions of the utility knife. You also use the viewer to investigate the design.

Step 1 –View renderings Each version that is saved is archived and stored by Fusion 360. View all three versions that were created for the Utility Knife. 1. Click the **Rendering** icon. 2. Select each version of the utility knife to display the renderings of previous versions. The images were created automatically when you saved a new version in previous steps. Cloud based rendering is discussed further in the Rendering chapter.	
Step 2 – View versions 1. Click the **Versions** icon (the watch shaped icon). 2. Click the different versions to view 3D previews. Notice the comments that were entered in previous steps are also available to view.	

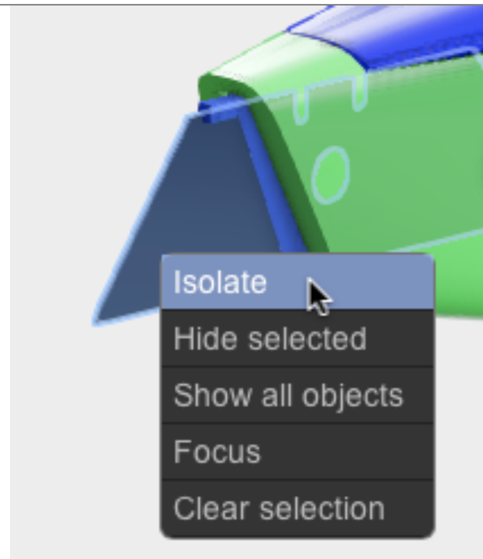
Step 3 – View the 3D design

1. Hold the left mouse button to rotate the 3D model in the viewer.
2. Left-click on an item to select and highlight specific bodies. Notice that information like the material and weight of the selected item is displayed.



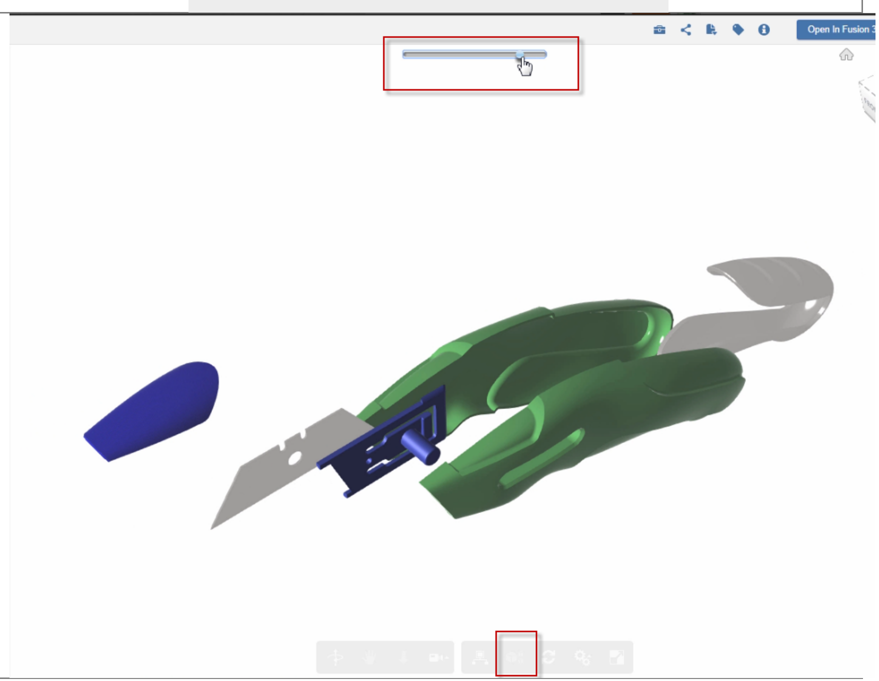
Step 4 – Isolate an item

1. Select an item.
2. Right-click and select **Isolate** to display only display the selected item.
3. Right-click and select **Show all objects** to display the entire design.



Step 5 – Explode the design

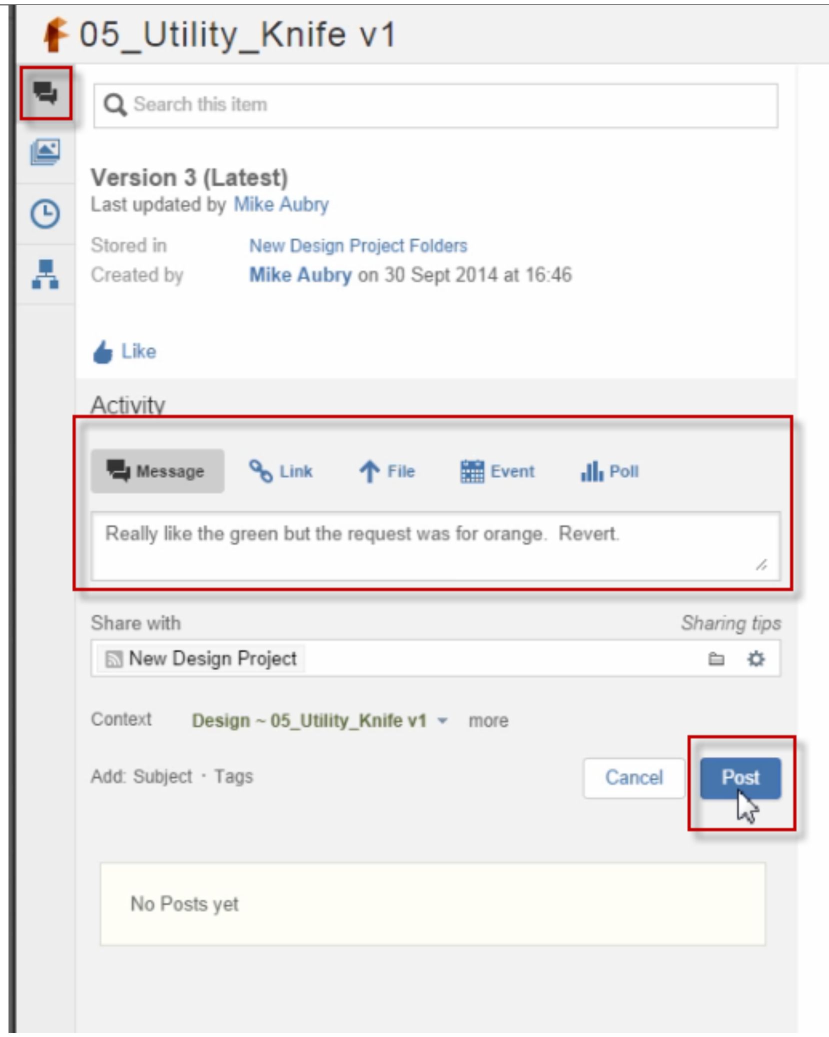
1. Click the **Explode** button at the bottom of the screen.
2. Use the slider to expand and contract the assembly.



Document design decisions: In this section, you add a message to the design in A360. Messages can be seen by anyone with access to the project.

<table>
<tr>
<td>
Step 1 – Document activity

Fusion 360 allows you to collaborate and articulate design decisions within your design. Create a message to describe to your team a design change you will make.

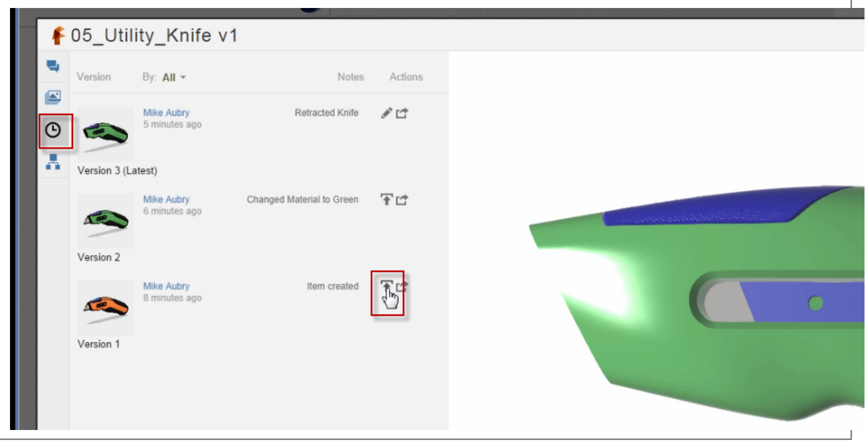
1. Click the Preview icon.
2. Click in the message field and enter: “Really like the green but the request was for orange. Revert.”
3. Click Post to post the message.
</td>
<td>


</td>
</tr>
</table>

Promote a previous version: In this section, you promote a previous version to be the current version. This let's you "rollback" your design.

<table>
<tr>
<td>Step 1 – Promote previous versions

Fusion 360 stores and allows you to promote previous versions. Any archived version can be promoted back to the current version.

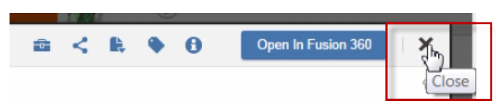
1. Click the Versions icon (the watch shaped icon).
2. Click the Promote icon.</td>
<td>
</td>
</tr>
</table>

View associated data: In this section, you view a list of data associated to the utility knife design then close the view of the design.

<table>
<tr>
<td>Step 1 – View associated documents

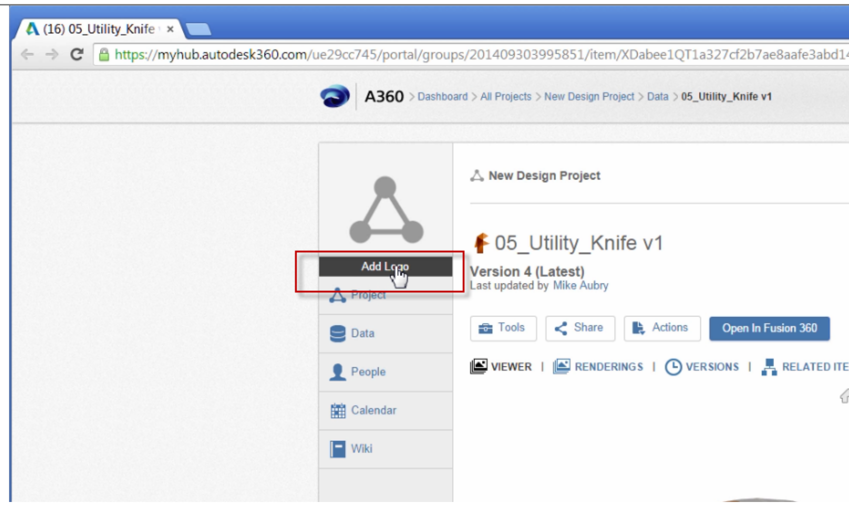
Fusion 360 manages associativity between designs and drawings. View the drawing that you created in a previous step.

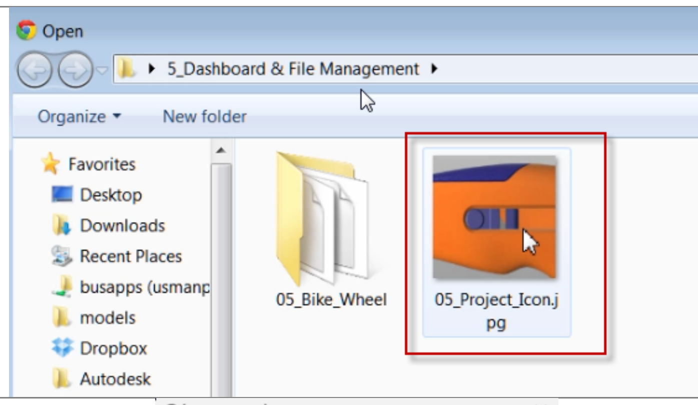
1. Select the Related Items icon.
2. The drawing is listed as an associated item.</td>
<td>
</td>
</tr>
<tr>
<td>Step 2 – Close the viewer

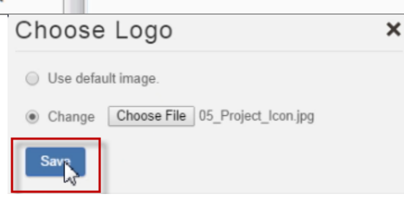
1. Click the X to close the viewer.</td>
<td>
</td>
</tr>
</table>

Customize your project: You can add a custom logo to your project. This makes it easier to differentiate between projects.

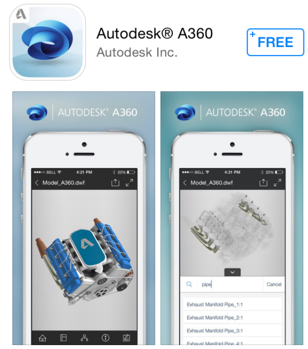
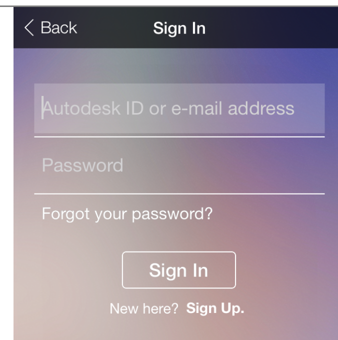
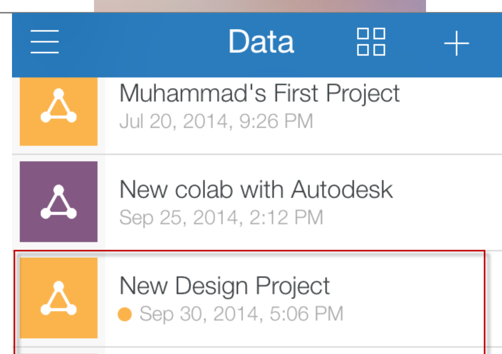
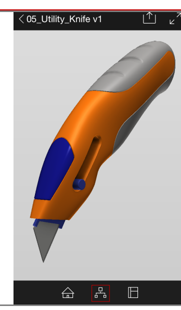
<table>
<tr>
<td>Step 12 –Add a logo to your project

1. Select Add Logo in the A360 web browser.
2. Select Choose File.</td>
<td>
</td>
</tr>
<tr>
<td>Step 2 – Select the image

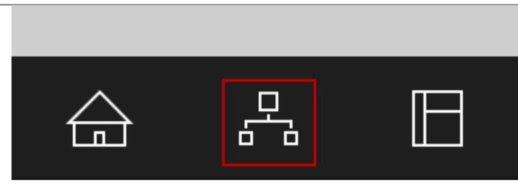
1. Navigate to the png image provided in the chapter 5 dataset or use any jpg or png under 200x200 pixels.
2. Select the image then click Choose.</td>
<td>
</td>
</tr>
<tr>
<td>Step 3 – Save the change

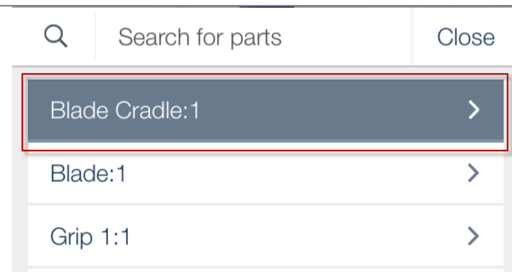
1. Click Save.</td>
<td>
</td>
</tr>
</table>

Access data from a mobile device: In this section, you access the project from a mobile device.

Step 1 – Access data from a mobile device Fusion 360 is accessible through your mobile devices. 1. To access data from your apple mobile device install the free Autodesk A360 App via the Mac App Store or Google Play Store.	
Step 2 – Sign in 1. Sign in to the App using your Autodesk ID (it is the same account you use for Fusion 360).	
Step 3 – Select the project 1. Scroll to the **New Design Project** and select it.	
Step 4 – Select the design 1. Open the **Utility Knife**.	

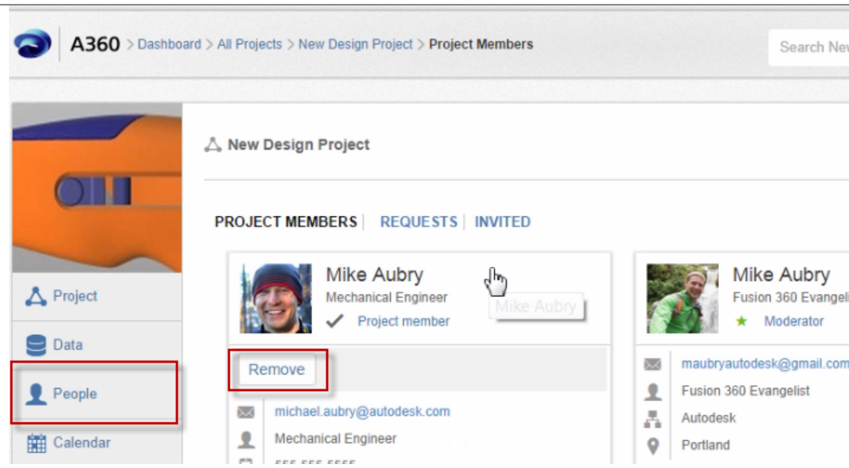
<table>
<tr>
<td>Step 5 – Select the action

1. Select the **Isolate** icon.</td>
<td></td>
</tr>
<tr>
<td>Step 6 – Select the object

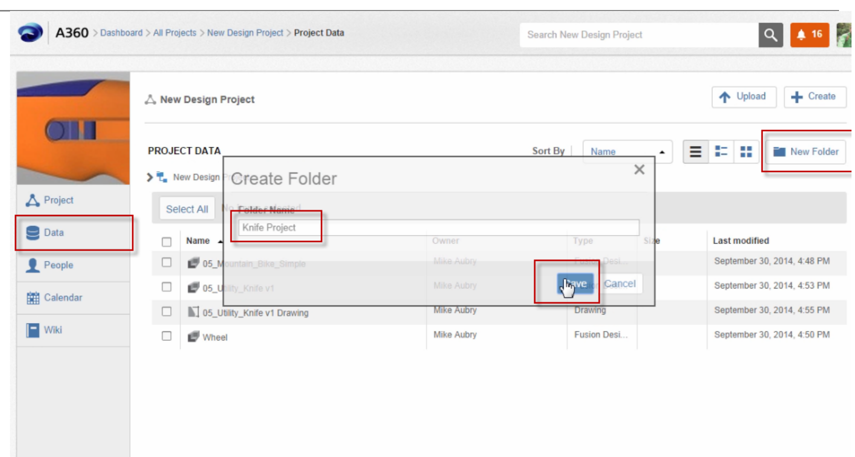
1. Choose **Blade Cradle** to isolate the blade.</td>
<td>
</td>
</tr>
<tr>
<td>Step 7 – View the object

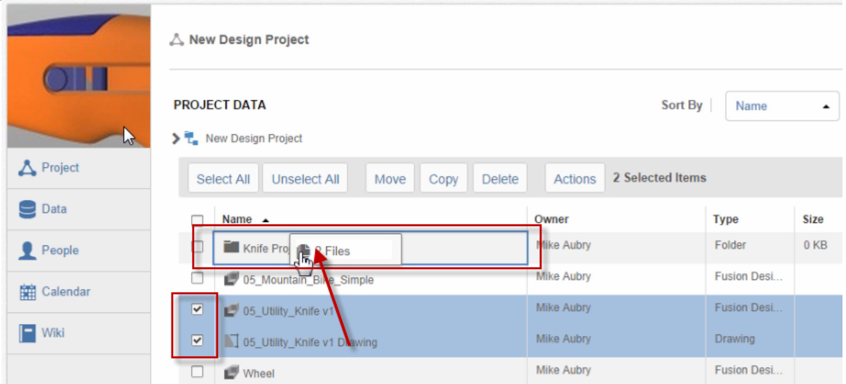
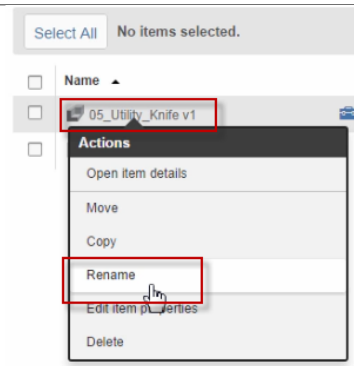
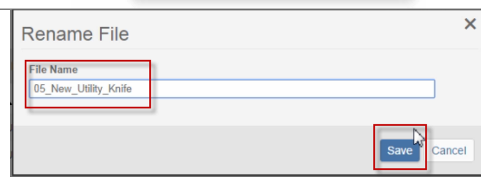
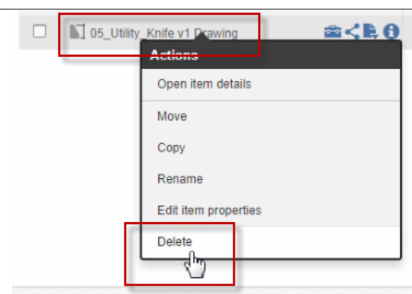
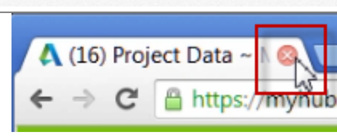
1. Rotate the model by placing a finger and moving on the screen.
2. Double tap on the model to change the rotation pivot point, which will appear as a green sphere.
3. Exit the app and return to your laptop to complete the remaining exercises.</td>
<td>
</td>
</tr>
</table>

Revoke access to the project: Fusion 360 allows the moderator of a project to add or remove access to that project. Revoke the access of a project member.

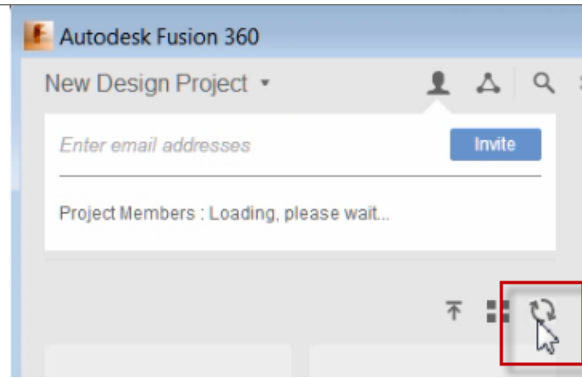
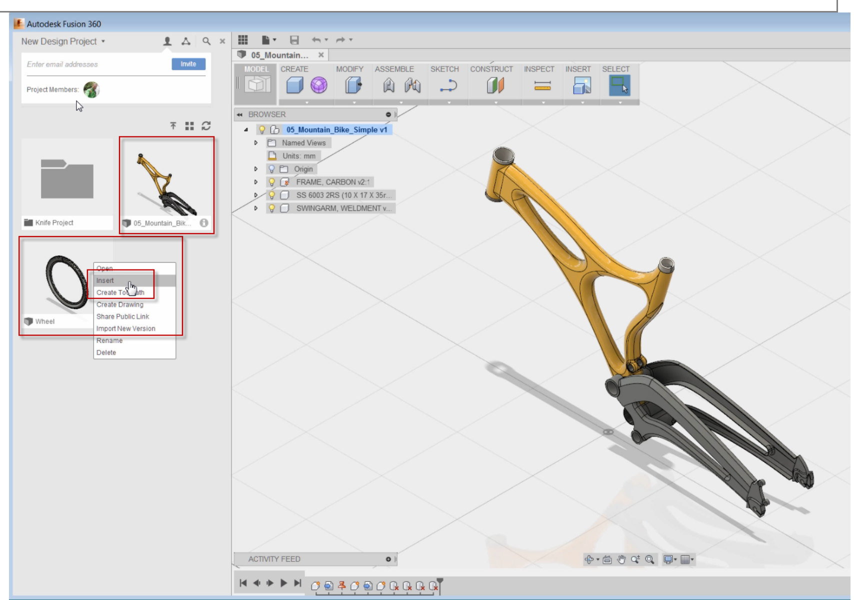
Step 1 – Revoke access from a project 1. Go to the **New Design Project** in the A360 web interface. 2. Select **People**. This shows all the members of the project. 3. Select **Remove**. This removes the access of the user you added earlier in an earlier step. The user no longer has access to the project from any device.	

Manage data: Familiarize yourself with common data management operations like move, rename and delete by moving the Utility Knife into a new folder.

Step 1 – Create a new folder 1. Select **Data** to explore the folder hierarchy of the project. 2. Click **New Folder**. 3. Enter "**Knife Project**" then click **Save**.	

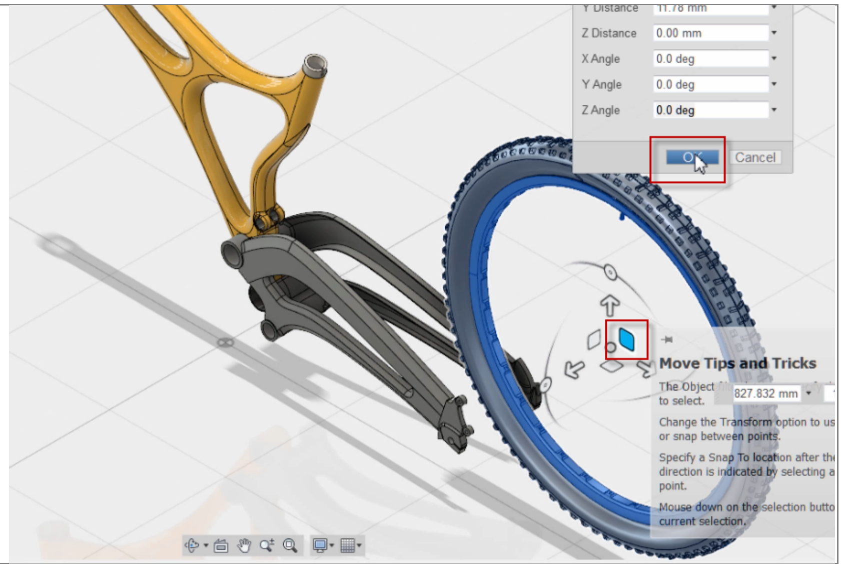
Step 2 – Move data 1. Check the boxes next to the design and drawing. 2. Drag and drop the items into the folder to move them. 3. Open the **Knife Project** folder.	
Step 3 – Rename a design 1. Right-click on the **05_Utility_Knife** design and select **Rename**.	
Step 4 – Enter new name 1. Rename to: "**05_New_Utility_Knife**". 2. Select **Save**.	
Step 5 – Delete an item 1. Right-click on the **05_Utility_Knife** drawing and select **Delete**.	
Step 6 – Close your browser 1. Close the web browser.	

Insert designs into other designs: In this section, you insert a design into an assembly design.

Step 1 – Refresh data 1. Return to Fusion 360. 2. Select **Refresh** in the Data Panel. This will get the most current data and reflect the changes made in the web browser in the previous step. Notice that while you were completing the previous steps that Fusion 360 has fully uploaded the bike dataset and SolidWorks wheel dataset. The wheel is also combined into one single Fusion design.	
Step 2 – Insert a design into an assembly 1. Double-click the **Mountain Bike** design to open it. 2. Right-click on the **Wheel** design and select **Insert** to insert the Wheel into the Bike Design.	

Step 3 – Place the wheel

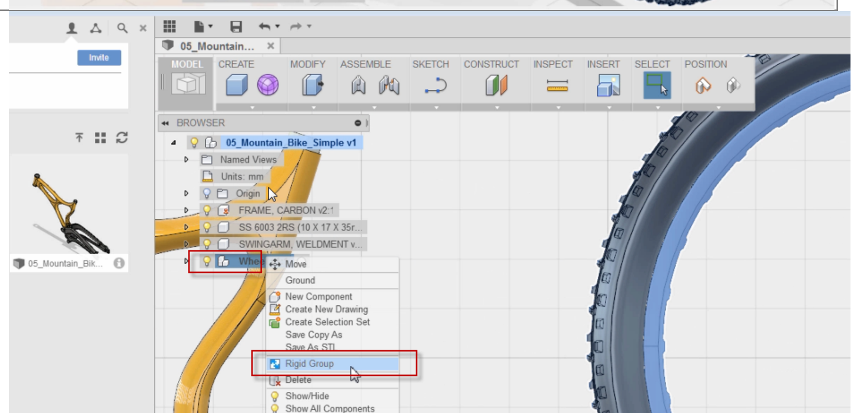
1. Use the manipulator to position the wheel in the bike assembly.
2. Click **OK** when done.



Step 4 – Lock the wheel component

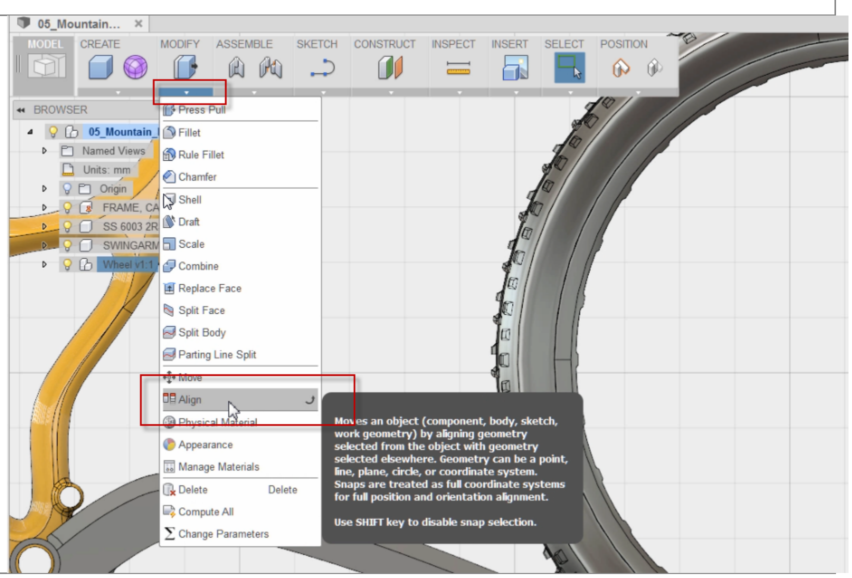
1. Right-click **Wheel** in the browser and select **Rigid Group**.

Rigid groups move as one object. This will make it easier to align to the wheel with the bike in the next step.

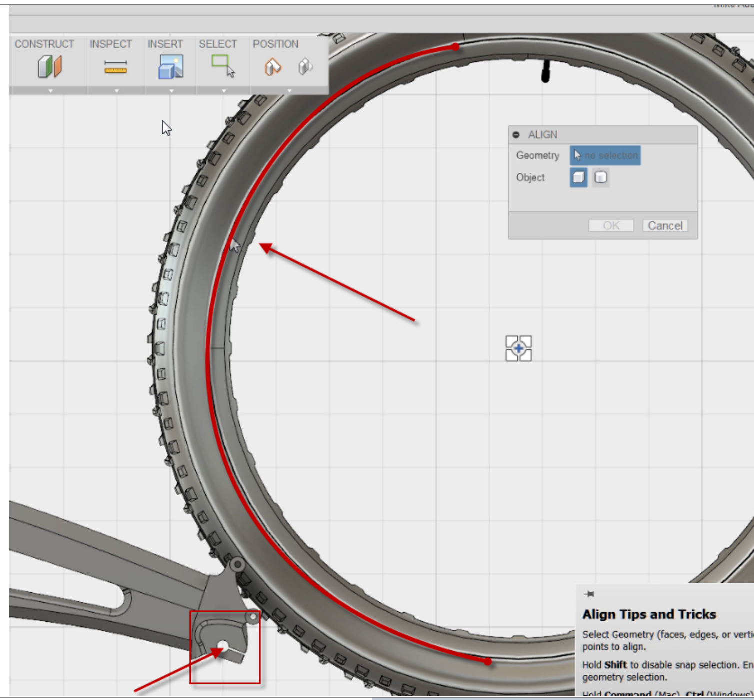


Step 5 – Start the Align command

1. Click **Modify > Align**.

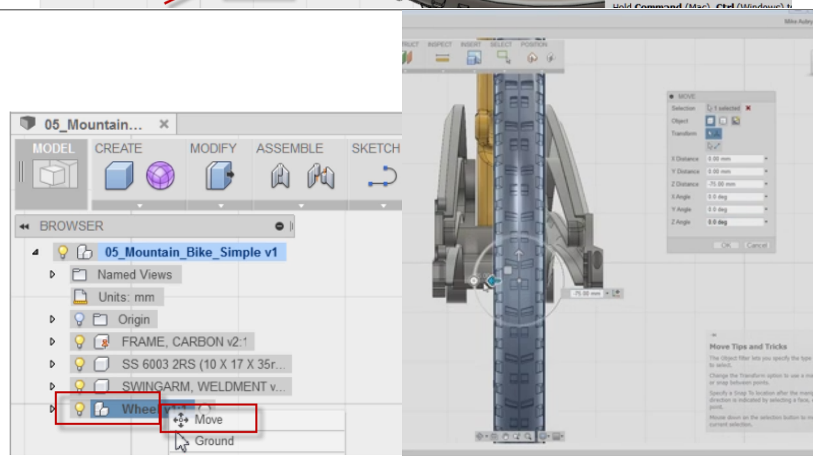


<table>
<tr>
<td>
Step 6 – Select areas to align

1. Select the circumference of the wheel and select the circumference of the bike frame's dropout. The wheel will axially align with the bike dropout.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 7 – Center the wheel

1. Right-click Wheel in the browser and select **Move**.
2. Drag the manipulator to approximately center the wheel.
3. Click **OK**.

The Assemblies tutorial will show how to be more precise using joints.
</td>
<td>


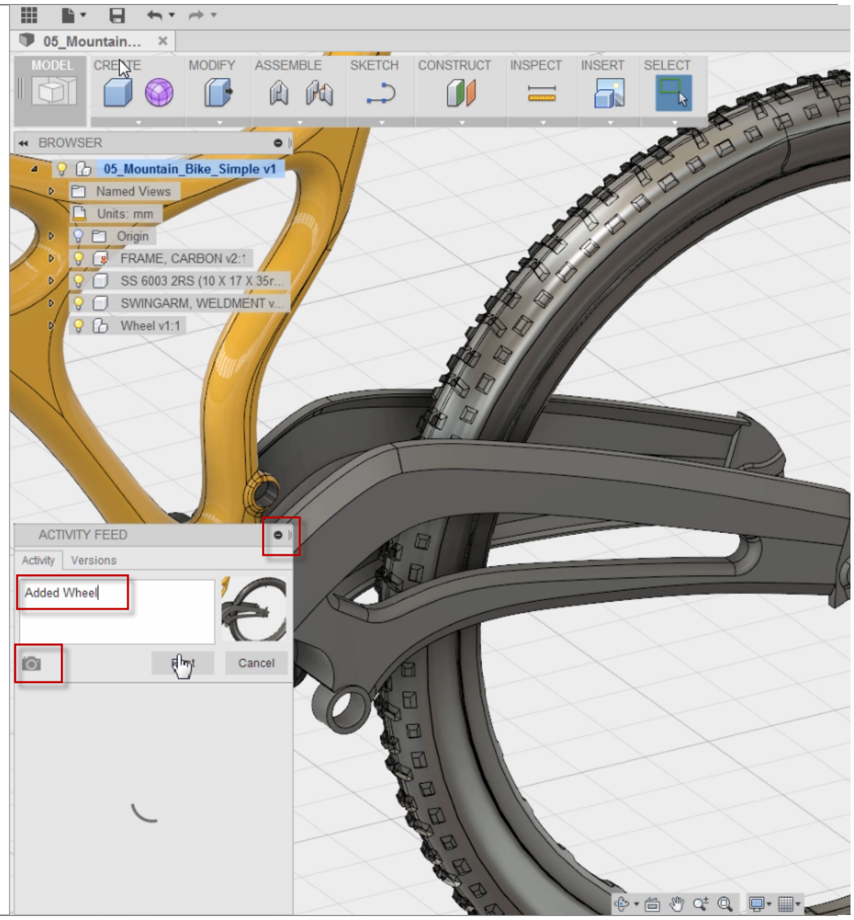
</td>
</tr>
</table>

Step 8 – Activity feed

1. Click the "**+**" to expand the Activity Feed.

The activity feed is where you see recent activity in the design.

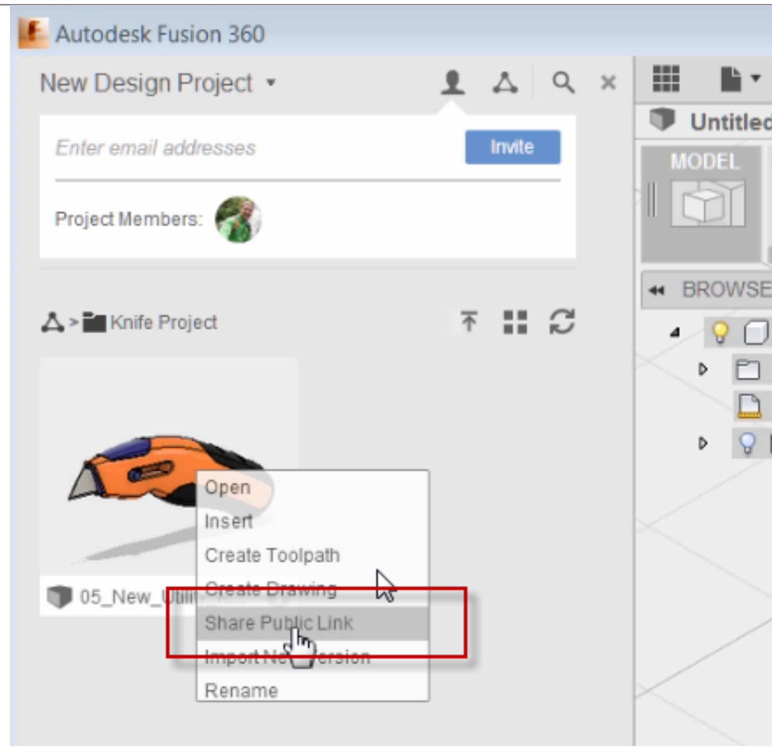
2. Click the **Camera** icon to create a snapshot of the screen to show progress you have made on the design.
3. Enter "**Added Wheel**" in the comment field.
4. Click **Post**.



Share designs: In this section, you create a public link to share your design. Then paste the link into a browser to see what other stake holders see when using the link. Finally, you download the file from the public link.

<table>
<tr>
<td>
Step 1 – Share public link

Regardless of whether a user has an Autodesk ID, you are still able to share and make designs available for download. Share a design using a public web link.

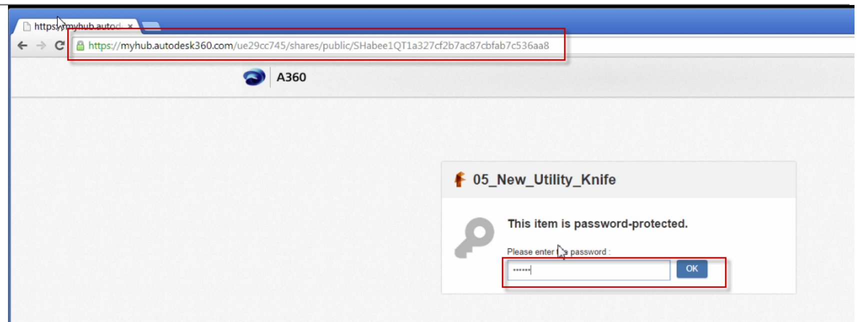
1. Open the Data Panel and browse to the 05_Utility_Knife.
2. Right-click on the Utility Knife and choose Share Public Link.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 2 – Configure the link

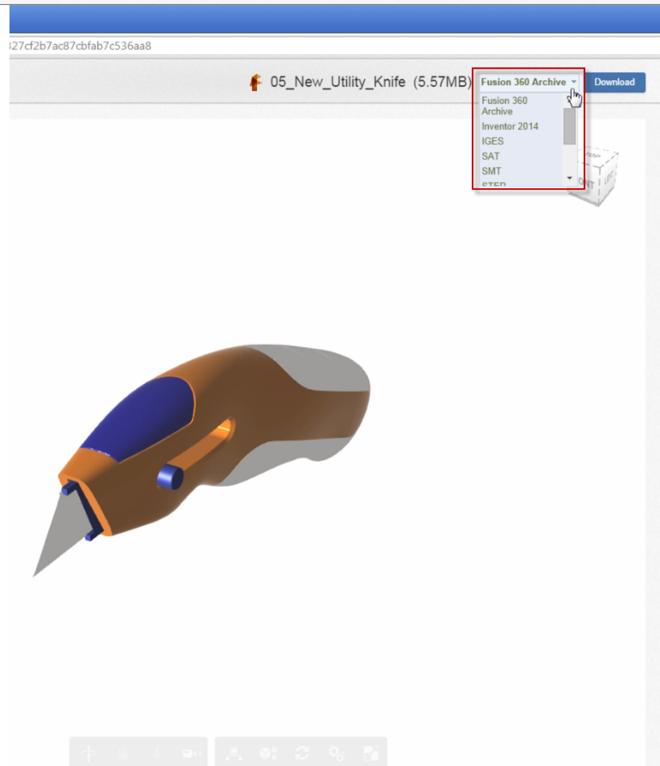
1. Check the following:
• Share the latest version with anyone using this public link
• Allow the item to be downloaded
• Require a password
2. Enter “Fusion!” as your password.
3. Click Copy to copy the link to your clipboard.
4. Close the Share Public Link dialog box.
</td>
<td>


</td>
</tr>
</table>

<table>
<tr>
<td>
Step 3 – Use the link

1. Open your web browser.
2. Paste in the copied URL.
3. Enter the password, “**Fusion!**”.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 4 – View and download

1. A 3D preview is displayed in the browser.
2. Select a CAD format to download.
3. Click **Download**.
</td>
<td>


</td>
</tr>
</table>

Congratulations! You completed the Manage and Collaborate module.

06: Working with Assemblies

Assemblies

Fusion 360 supports several ways of designing assemblies. This module takes a look at a few common workflows.

Joints

Joints control the position and motion between components. Joints are similar to assembly constraints in traditional parametric modelers in that they both are used to control the position of components in an assembly. Joints are different because they indicate the allowable motion between components. Assembly constraints remove degrees of freedom instead of allowing motion.

Fusion 360 has two types of joint commands. Use the As-built Joint command when components are already in position and you need to join them. As-built Joint is often used with top-down design. Use the Joint command when components are out of position. Joint is typically used with distributed designs.

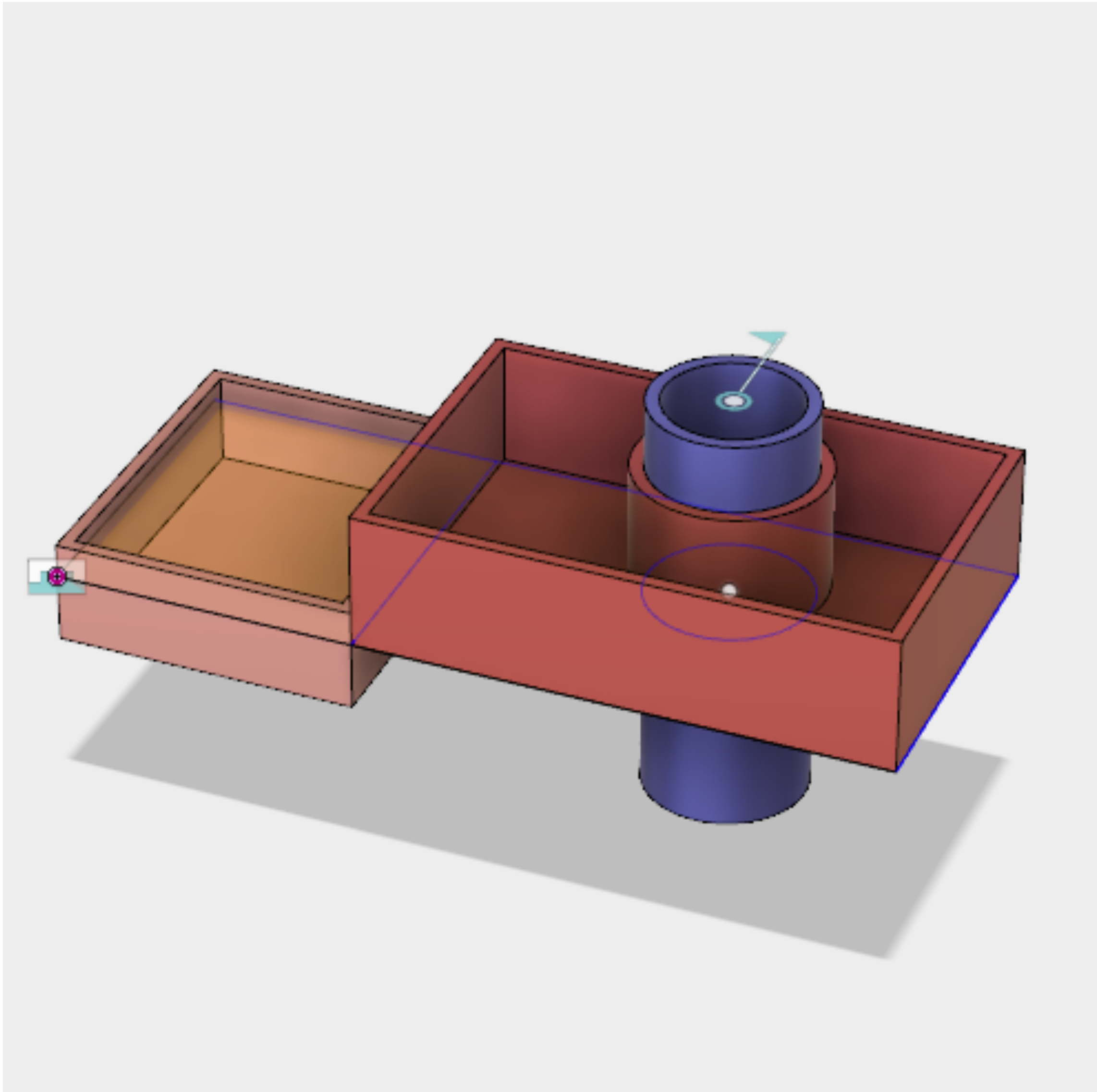
Top-down design

Top-down design is a design method where you create your components in the same assembly design. Top-down design makes it easy to create and manage relationships between parts.

Distributed design

Distributed design or bottom-up modeling is a design method where components are created individually then added to an assembly. Using distributed design allows you to reuse components and maintain the relationship back to the original component. Use distributed design when you use the same part in multiple assembly designs.

Lesson 1: Top-down Design



Learning Objectives

1. Create components in an assembly.
2. Modify components using the driving sketch.
3. Use as-built joints to position the components.

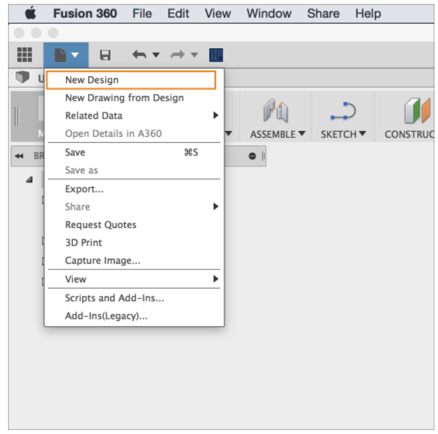
Datasets

This lesson uses a new design for the exercise.

Step-by-step Guides

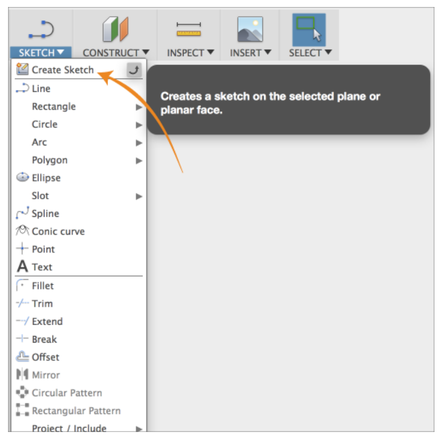
Step 1: Let's start with creating a new design. We are going to create an assembly in this design.

1. Launch Fusion 360.
2. Click **File > New Design** to start a new design.



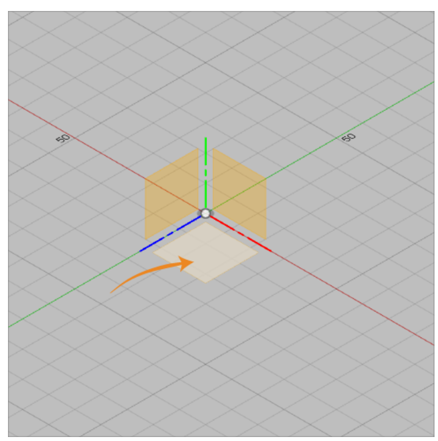
Step 2: Start the Create Sketch command.

1. Select **Model > Sketch > Create Sketch.**



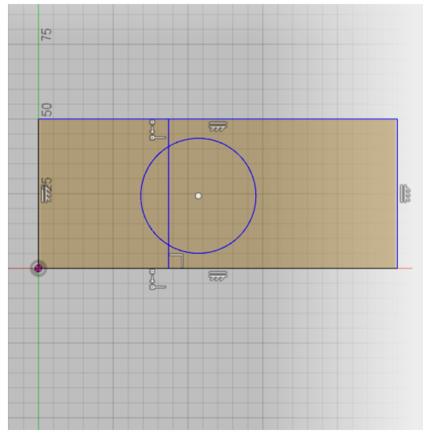
Step 3: Select the sketch plane.

1. You are prompted to select the plane you want to sketch on.
2. Select the bottom plane (XZ) plane.



Step 4: Create sketch curves.

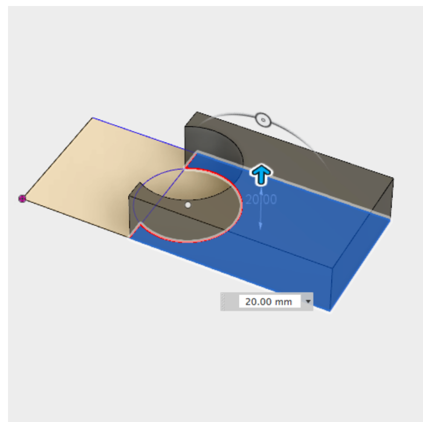
1. Draw a rectangle that starts at the origin and is **50 mm** x **120 mm**.
2. Draw a vertical line that is approximately **50 mm** from the left edge of the rectangle.
3. Draw a circle with a center point near the vertical line and a diameter approximately **40 mm**.
4. Click **Stop Sketch**.



Step 5: Create the first component.

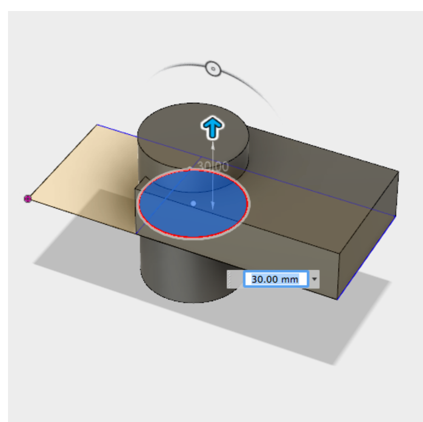
1. Click **Modify > Press Pull**.
2. Select the sketch profile on the right.
3. Enter **20 mm** for the Distance.
4. Select **New Component** for the Operation.

NOTE: Notice that a new component is added to the browser.



Step 6: Create a cylinder component.

1. Expand the Sketches node in the browser then click the light bulb next to Sketch1 to turn on the visibility.
2. Click **Modify > Press Pull**.
3. Select the two sketch profiles that make a circle.
4. Set the Direction to **Symmetric**.
5. Enter **30 mm** for the Distance.
6. Select **New Component** for the Operation.
7. Click **OK**.
8. Click **Inspect > Component Color Cycling Toggle**.

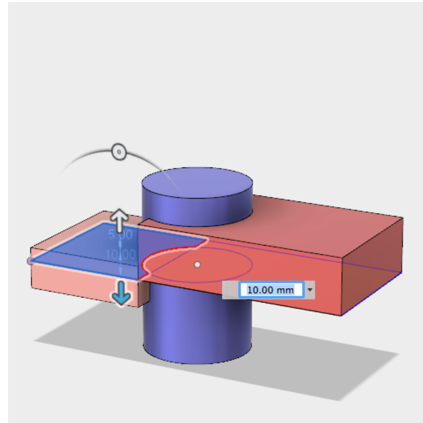


Note: Another component is added to the browser and you should see the two components displayed as different colors.

Step 7: Create the last component.

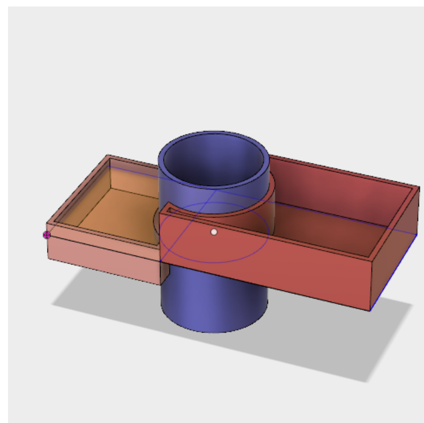
1. Click **Modify > Press Pull**.
2. Select the sketch profile on the left.
3. Set the Direction to **Two Side**.
4. Enter **5 mm** for the first Distance.
5. Enter **10 mm** for the second Distance.
6. Select **New Component** for the Operation.

Note: You now have three components in the browser.



Step 8: Shell the components.

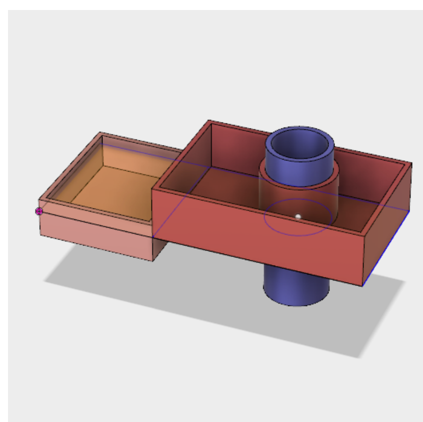
1. Click **Modify > Shell**.
2. Select the top face of the component on the left.
3. Hold ctrl (Windows) or command (Mac) then select the top faces of the other two components.
4. Enter **2 mm** for the Thickness.



Step 9: Modify the sketch to update the components.

1. Drag the edge of the sketch circle to change the diameter. Once you release the mouse button the components adjust to the change.
2. Drag the center point of the circle. Once you release the mouse button the components update again.

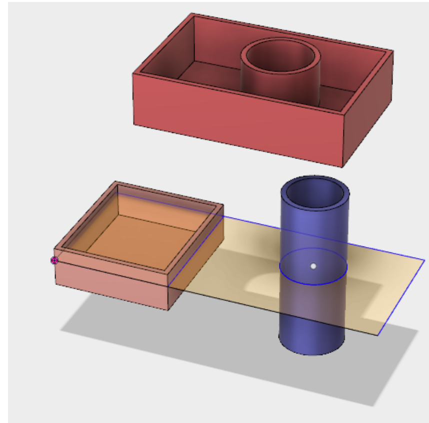
Note: The components update with sketch changes because all three components were created from the same sketch. This is top-down design.



Step 10: Move components.

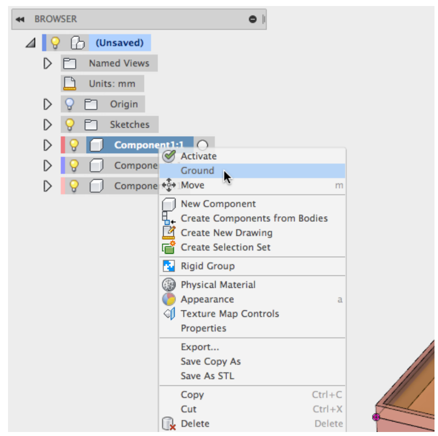
1. Drag the box on the right to move it. Notice a Position panel is displayed at the end of the toolbar.
2. Click **Position > Revert** to move the box back.

Note: All three objects are free to move because they are individual components.



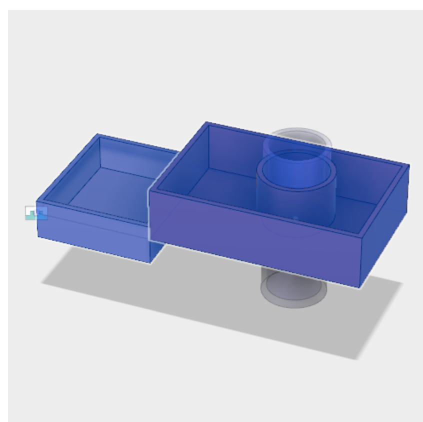
Step 11: Fix the first component in space.

1. In the Browser, right-click on Component1 then click **Ground**.
2. In the canvas, click and drag the box on the right (Component1). Now it cannot move because it is grounded.



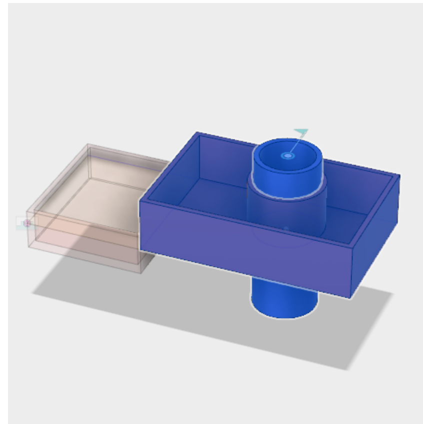
Step 12: Use joints to control the position the other box.

1. Click **Assemble > As-built Joint**.
2. Set the Type to **Rigid**.
3. Select the two boxes.
4. Click **OK**.
5. Click and drag the box on the left. It cannot move because it is rigidly joined to the other box.



Step 13: Use joints to control the position of the cylinder.

1. Click **Assemble > As-build Joint**.
2. Set the type to **Revolute**.
3. Select the cylinder and the box on the right.
4. Select the top edge of the cylinder for the Position.
5. Click **OK**.
6. Click and drag the cylinder. Notice the symbol rotates indicating that the cylinder can move.



Lesson 2: Create Joints

Learning Objectives

1. Create basic animations
2. Introduction to the animation timeline

Datasets

In the Samples section of your Data Panel, browse to:

Basic Training > 06 - Assemblies > **06_tripod**

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

Step 1: Move components to test degrees of freedom.

1. Drag some of the components to see that they are free to move. A Position panel is displayed in the in toolbar after you drag components. You can use **Snapshot** to keep the position or **Revert** to put the components back.
2. Click **Position > Revert** to return the components to their original position.

Note: This design was created in another CAD system and uploaded to Fusion. All the geometry is imported in position but there are no joints to keep them in position.

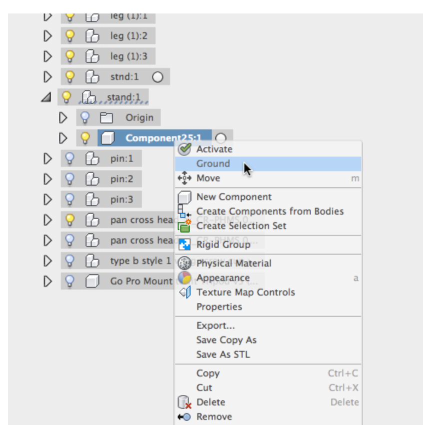
Step 2: Move components to see components behind them.

1. Drag the red stand and grey camera mount to the side.
2. Click **Position > Snapshot** to maintain this position.

Step 3: Lock the stand bracket in position.

1. In the Browser, expand **stand**.
2. Right-click on **Component25** then select **Ground**.

Note: Grounding a component locks it in the current position. Typically you will ground at least one component in an assembly.



Step 4: Add a joint to the cylindrical legs.

1. Click **Assemble > As-built Joint**. Make sure it is the "As-built Joint" command and not the "Joint" command.
2. Change the Type to **Slider**.
3. Select the two cylinders that make up one of the legs.
4. For the Position, select the lower circular edge of the white cylinder. You will see a preview of the motion set up by the joint.

Note: The legs are three instances of the same subassembly. Applying a joint to one instance adds that joint to all three instances.

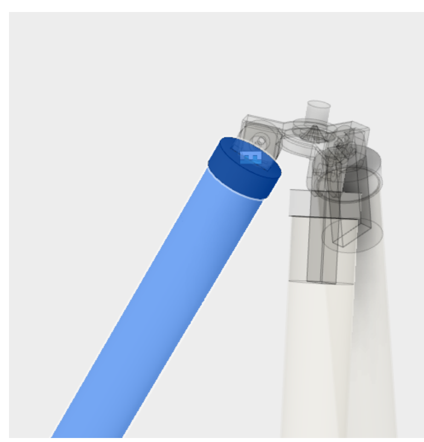
Step 5: Add a joint to the bottom end cap.

1. Click **Assemble > As-built Joint**.
2. Change the Type to **Rigid**.
3. Select one of the bottom end caps and the grey leg attached to it.

Note: The end cap is also part of the leg subassembly so applying a joint to one end cap adds the same rigid joint to all three end caps.

Step 6: Add a joint to the top end cap.

1. Click **Assemble > As-built Joint**.
2. Select one of the top end caps and the white cylinder attached to it.



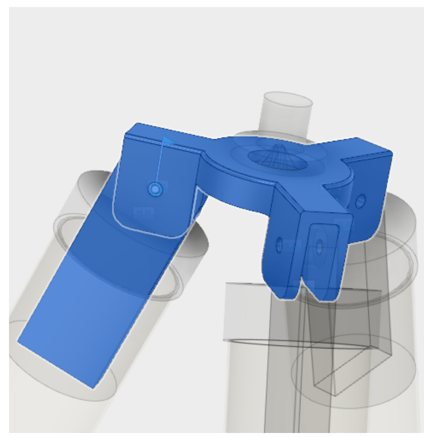
Step 7: Add a joint to the leg connector.

1. Click **Assemble > As-built Joint**.
2. Select one of the top end caps and the connector attached to it. The connector is the tombstone shaped component that connects the legs to the stand bracket.

Step 8: Add joints to connect the legs to the bracket.

1. Click **Assemble > As-built Joint**.
2. Change the Type to **Revolute**.
3. Select the connector you used in the previous joint and the bracket attached to it.
4. For the Position, select the edge of the hole in the bracket.

Note: This is a joint between one of the leg subassemblies and a component in the main assembly so it is only applied to the selected subassembly. Next we'll repeat this process to join the other two legs.

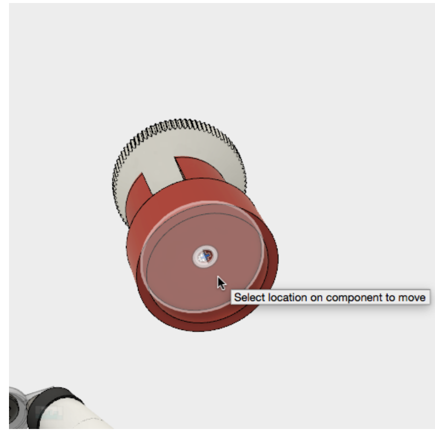


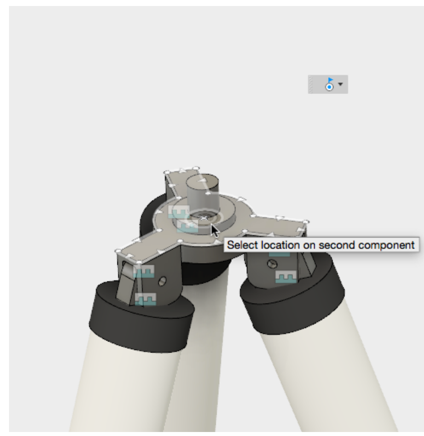
Step 9: Join the other two legs.

1. Use the **As-built Joint** command to join the other two legs to the bracket.
2. Drag the legs to see how the cylinders can move inside each other and how the legs rotate around the bracket.
3. Click **Position > Revert** to return the legs to their original position.

Step 10: Use the Joint command to join the red stand to the bracket.

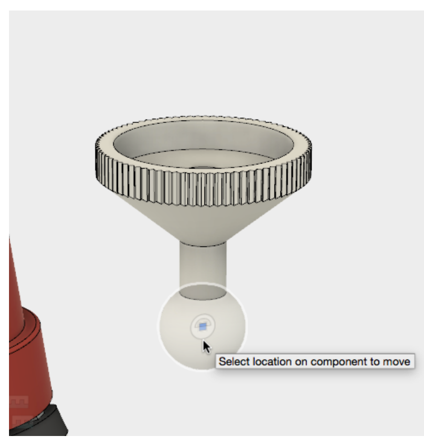
1. Click **Assemble > Joint**. Make sure you use the Joint command and not the As-built Joint we have been using.
2. Change the Type to **Rigid**.
3. Move the cursor over the bottom face of the red stand. Click when the joint glyph is displayed in the center of the hole in the stand.
4. Move the cursor over the top face of the bracket. Hold command (Mac) or ctrl (Windows) to lock on that face then click on the hole in the center.

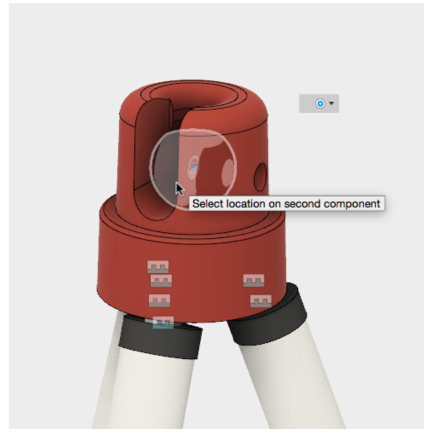




Step 11: Use the Joint command to join the stand and the camera mount.

1. Click **Assemble > Joint**.
2. Change the Type to **Ball**.
3. Select the ball on the bottom of the camera mount.
4. Select the ball cavity on the inside of the red stand.





Step 12: Test the motion of the camera mount.

1. Drag the mount to see the available motion.
2. Click Position > Revert to return the mount to the original position.

Note: Fusion does not detect material interference unless you are set up contact sets.

7.1: Rendering

Rendering is the process of generating an image by combining geometry, camera, texture, lighting and shading (also called materials) information using a computer program.

Before an image can be rendered Appearance Materials are applied to the various parts of your design to visualize how your design would look in the real word. Materials contain the visual properties of plastic, glass, metal, paint and wood (and pretty much anything else you can think of) to create photorealistic images.

Lesson 1: Applying Materials

Learning Objectives

1. Accessing the Render workspace
2. Understanding the Appearance Material window
3. Assigning appearance materials to geometry
4. Assigning appearance materials to bodies in the browser

Datasets Required

In Samples section of your Data Panel, browse to:

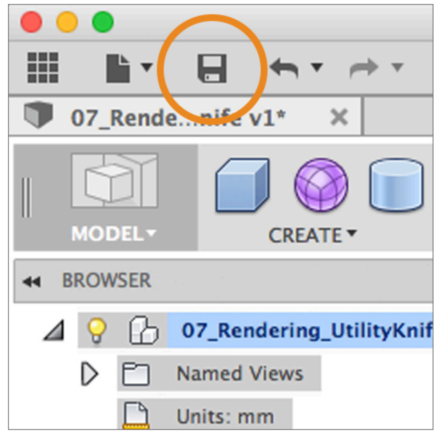
Fusion 101 Training > 07-Rendering > 07_Rendering_UtilityKnife.f3d

Open the design and follow the step-by-step guide below to get started with the lesson.

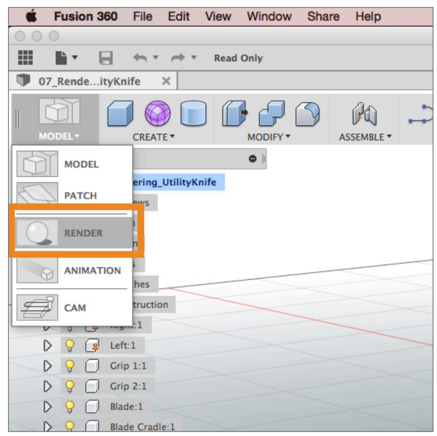
Step-by-step Guides

Step 1: Go to the Render workspace – The Rendering workspace toolbar contains tools to Setup your model and to create a Render.

1. Click on the **Save** icon on the top menu bar to copy of this sample file in to your personal project.
2. Click on the **Model** icon in the left of the toolbar to view other available workspaces.
3. Select the **Render** workspace from the dropdown list.

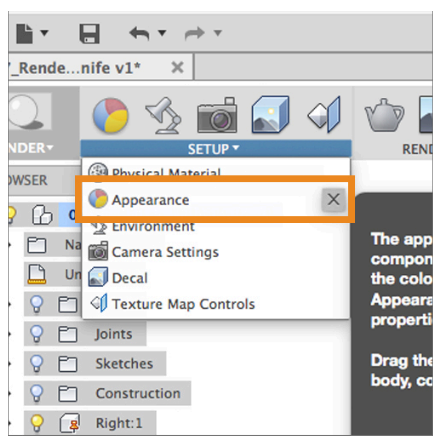


Note: You may have noticed that the environment changed slightly when you moved from Model to Render. This happens because the environment for Rendering is specifically tuned to create good-looking images.

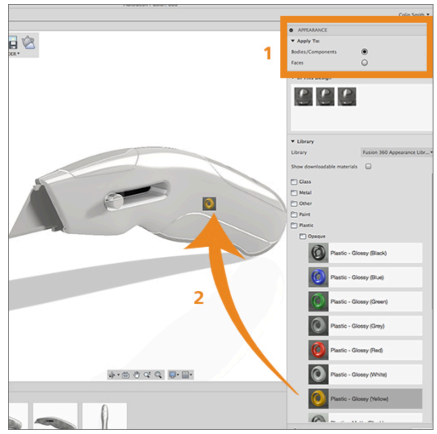


Step 2: Apply Materials to Geometry - Now that the model is in the Rendering Workspace you can begin assigning **appearance materials.**

1. In the **Render Workspace** select **Setup > Appearance** to open the **Appearance window.**
2. In the Appearance window check that the **Apply To** option is set to **Bodies/Components**
3. In the Library section of the Appearance window scroll down to **Plastic > Opaque > Plastic – Glossy (Yellow).**
4. Select and hold on the **Plastic – Glossy (Yellow)** swatch icon and drag it on to the main side body of the utility knife.
5. Repeat these steps so that **Plastic – Glossy (Yellow)** is assigned to both sides of the utility knife.



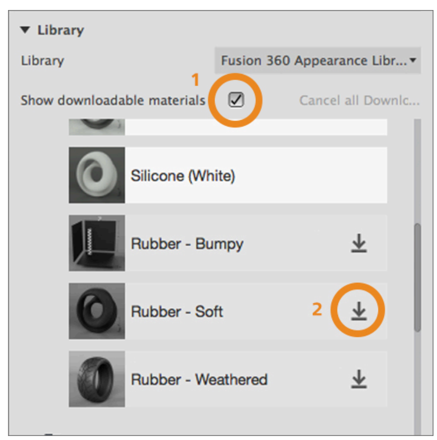
Note: If you assign the same material to several bodies in your design and do not edit them they will be automatically linked to the one material and only one material will be shown in the **In This Design** section. Editing the one material will affect all of the bodies that have that material assigned.

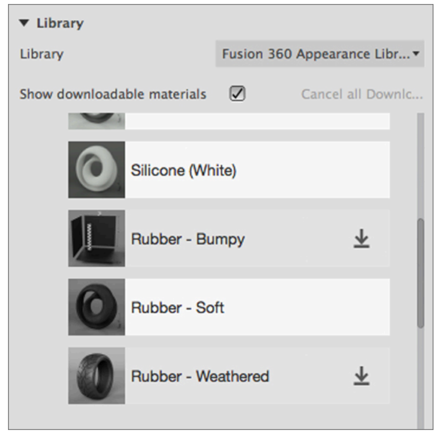


Step 3: Download two new materials. You can download new materials directly in to the Material Library.

1. In the **Appearance** window check the box next to Show **Downloadable Materials.**
2. In the Library open the folders for **Plastic > Textured** and **Other > Rubber**
3. Click on the **Download Material** icon next to **Plastic – Textured – Polka** and **Rubber - Soft**
4. After a minute the new material will be available to assign.

Note: You must be connected to the internet to download materials.

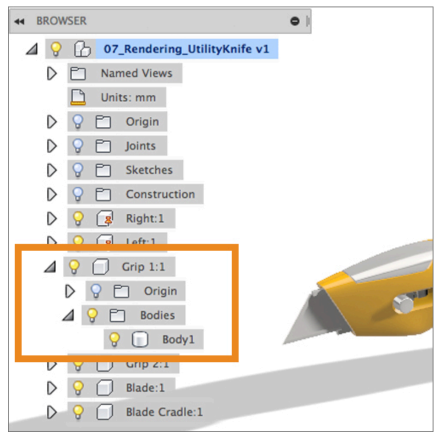


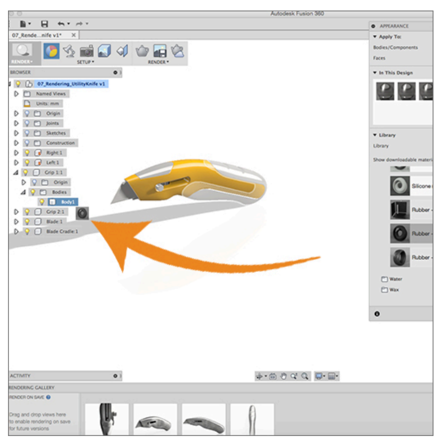


Step 4: – Apply a material to a body in the browser.

1. In the browser locate the component called **Grip 1:1.**
2. Click the arrow to the left of **Grip 1:1** to expand the contents of the component
3. Click the arrow to the left of **bodies** to show the bodies that are assigned to that component.
4. In the Appearance dialog box scroll down to **Other > Rubber > Rubber – Soft**
5. Select and hold on the **Rubber – Soft** swatch icon and drag it on to **Grip 1:1 > Bodies > Body 1** in the browser

Note: To assign a material to all of the bodies in a component, drag the material to the top-level component in the browser.





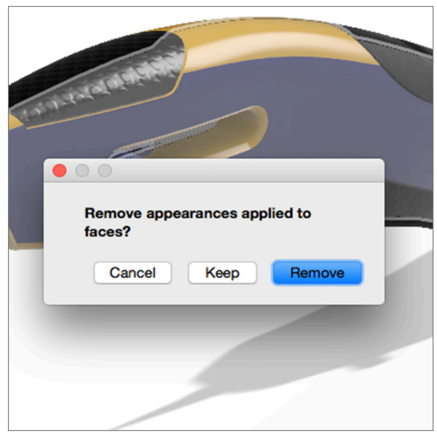
Step 5: Apply additional materials to the model

1. Using the method of your choice apply the following materials to the parts listed
   - **Plastic – Textured – Polka** to **Grip 2:1**

- **Plastic – Translucent – Matte (Blue)** to **Blade Cradle:1**
- **Metal - Stainless Steel – Satin** to **Blade:1**

Note: If you apply a material to a component and one or more of the bodies in the component already have a material applied you will be presented with and option to remove appearances applied to the bodies.

- **Keep** - only the bodies you selected that didn't already have a material applied will have the new material applied.
- **Remove** – all of the existing applied materials will be replaced with the new material you applied to the component.



Lesson 2: Editing Materials

Now that you have all the base materials applied to your design, you can customize the materials to look the way you want.

There are two levels of editing for materials. The basic or “lite” editor window enables you to quickly change:

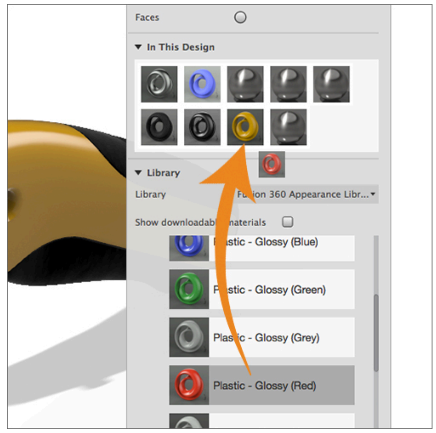
- Change the name of the material
- Edit the color – either by dragging the color sliders or by entering an RGB value.
- Change the scale of the texture or bump map that is part of the material (if appropriate)
- Rotate the texture or bump map that part of the material (if appropriate)
- Go to Advanced options

Learning Objectives

1. Assigning appearance materials to geometry
2. Assigning appearance materials to bodies in the browser
3. Editing Materials
4. Downloading New Materials to the Library
5. Assigning Materials to faces
6. Duplicating Materials
7. Adjusting Texture Map Controls

Step 1: Replace the Yellow Plastic Material. There are several ways to replace existing materials.

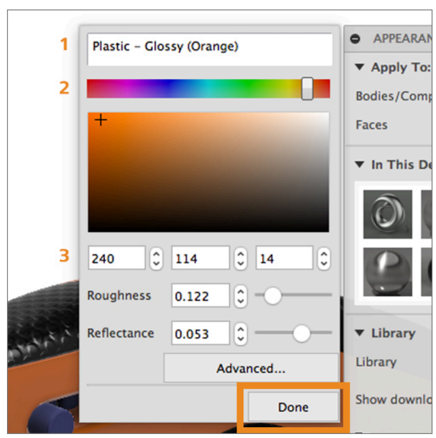
1. In the Appearance dialog box locate **Plastic – Glossy (Red).**
2. Drag the swatch from the library on to the swatch for **Plastic – Glossy (Yellow)** in the "In this design" section of the Appearance dialog box.



Note: All the bodies that had Plastic – Glossy (Yellow) applied now have Plastic – Glossy (Red) applied. This is a quick method for swapping out materials in your design.

Step 2: Edit Plastic – Glossy (Red)

1. Double click on the swatch for **Plastic – Glossy (Red)** in the **In this design** section of the Appearance window to open the editor window for this material.
2. At the top of the dialog box is the current name assigned to the material – double click in the name field and change the name to **Plastic – Glossy (Orange)**
3. Drag the color slider under the name field to an orange color of your liking.
4. In the middle of the dialog box there is section that allows you to enter RBG (Red, Blue and Green) values for a specific color.

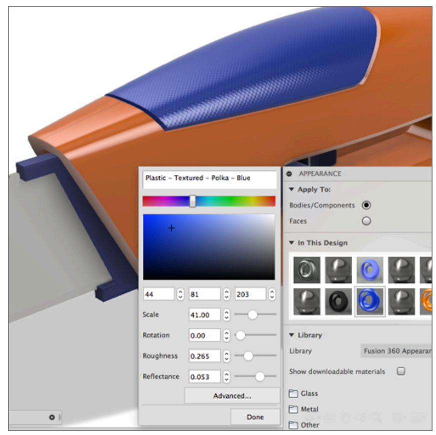


5. Enter **240, 114, 14** to change the color of the plastic material to orange.
6. Click the **Done** button.

Note: If you want to see which materials have been applied to which bodies in your design you can right click on the swatch and select **Select Objects Applied To** from the drop down list. Doing so will highlight bodes in your design that have the selected material applied.

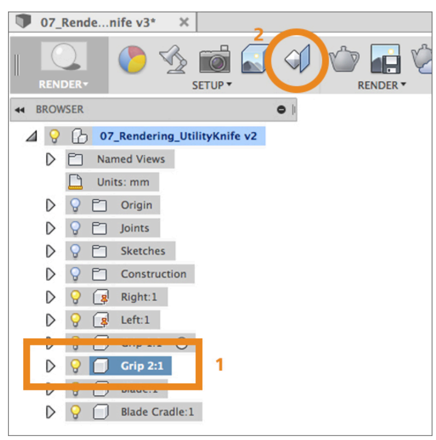
Step 3: Edit color and texture map for **Plastic – Texture – Polka.**

1. Change the name to **Textured – Polka - Blue**
2. Using the method of your choice change the color of **Plastic – Textured – Polka** to blue.
3. Change the scale of the texture map to **41**.
4. Move the slider next to **Rotate** to interactively change the orientation of the texture map.
5. When you are satisfied with the orientation, click **Done**.



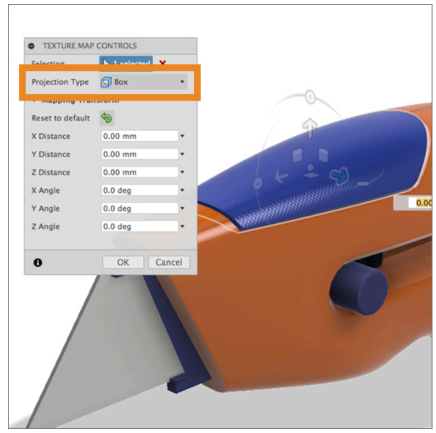
Step 4: Change the texture projection method for **Grip 2:1**. **Texture Map Controls** are used to change the type of projection method used on the object with a texture map.

1. Close the **Appearance** dialog box by clicking on the **Close** button.
2. In the browser, locate the component labeled **Grip 2:1** and right-click on it.
3. Select **Find in Window** to bring the part in the center of the screen.
4. Zoom in further so that you can clearly see the texture map on the surface.
5. In the **Render** menu bar select **Setup > Texture Map Controls**.



6. Change the **Projection Type** to **Box**
7. Click **OK** to accept the change.

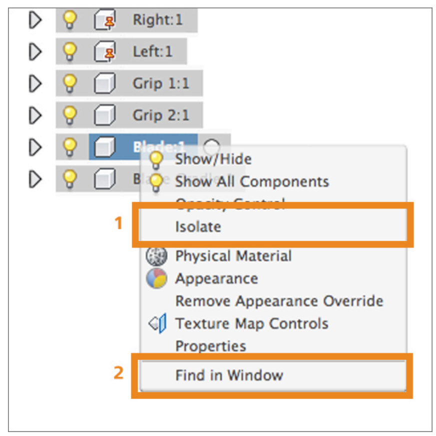
Note: Try selecting the other projection types to see the different results. In some cases you may have to choose an axis for a projection direction. In those instances an axis widget will appear, simply click on the axis that best matches the direction you would like to project. You can also change the location and rotation of the projection with the manipulator.

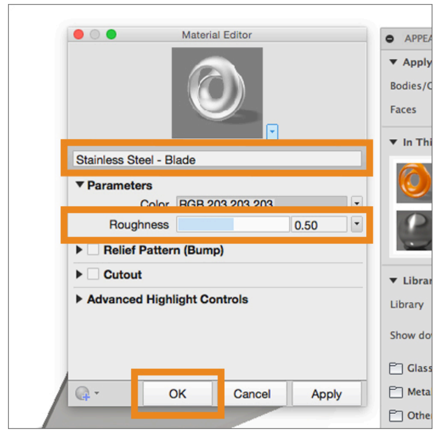


Step 5: Edit a material assigned to the Blade

1. In the browser locate the component labeled **Blade:1**
2. Right-click on **Blade:1** and select **Isolate** from the drop down menu list.
3. Right-click on **Blade:1** and select **Find In Window** from the drop down menu list.
4. Right-click on **Blade:1** and select **Appearance** from the drop down menu list to open the Appearance dialog box.
5. In the Appearance dialog box, double click on the **Stainless Steel – Satin** material in the **In This Design** section.
6. Click on the button labeled **Advanced…** to open the advanced editor window.
7. Change the name to **Stainless Steel – Blade** by click on the name field.
8. The **Roughness** setting controls the amount of reflection in the material, change the setting to **0.50**.
9. Click **OK**.

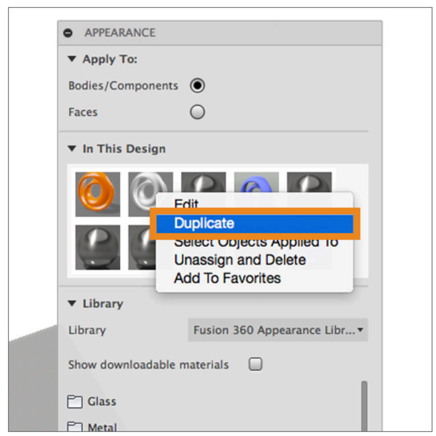
Note: The **Roughness** controls the surface finish of the material and ultimately how shiny and reflective it will look. When set to zero the surface will look like a mirror. When set to one, the surface will not reflect at all.

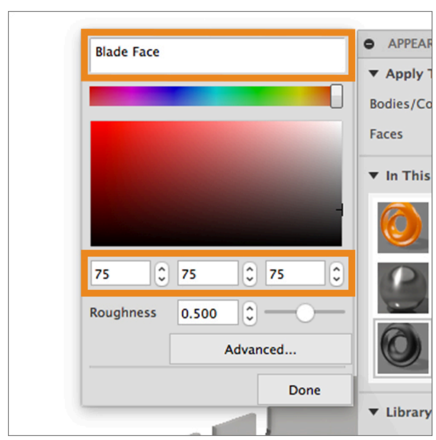


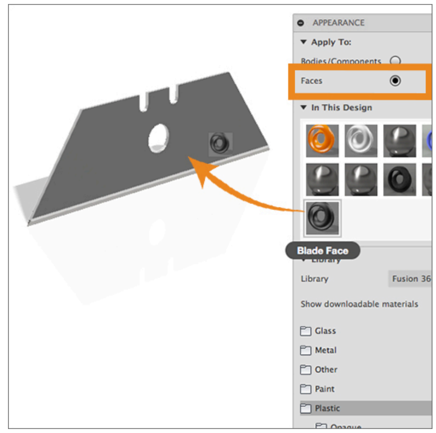


Step 6: Duplicate a material

1. Right click on **Stainless Steele – Blade** and select **Duplicate** from the drop down menu.
2. This creates a second material called **Stainless Steele – Blade(1)** that has the exact same settings as the original.
3. Double click **Stainless Steele – Blade (1)** material to open the Material Editor.
4. Change the name of the material to **Blade Face**
5. Change the color to **75,75,75**.
6. Select **Done**.
7. In the **Appearance** dialog box change the **Apply To:** setting from **Bodies/Components** to **Faces**
8. Now you can only apply materials to selected faces on a body/component
9. Drag the **Blade Face** material to the side face of **Blade:1**
10. Repeat this step on the other side of **Blade:1** so that both side faces have **Blade Face** applied.
11. Close the **Appearance** dialog box.
12. Right-click on **Blade:1** in the browser and select **Unisolate** to show the rest of the design.






APPEARANCE
Apply To:
Bodies/Components
Faces
In This Design
Blade Face
Library
Show downloadable materials
Glass
Metal
Other
Paint
Plastic

Lesson 3: Adding a Decal

A decal is used to mimic labels or transfers that appear on the surface of your model. These can be numbers on a keypad or branding and logos. Decals sit on top or the model surface and are applied differently than materials. In this lesson you will apply an image of the Autodesk logo to the body of the utility knife using the decal tool.

Learning Objectives

1. Selecting an image to use as a decal
2. Applying the decal
3. Adjusting the decal

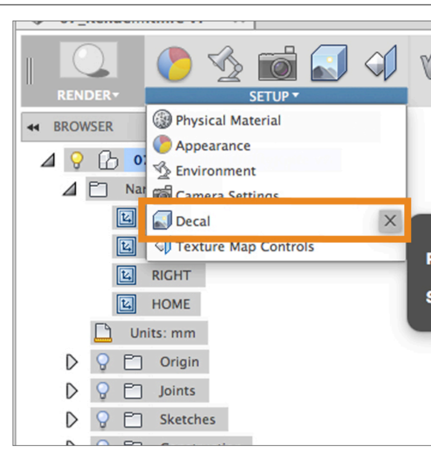
Step-by-step Guides

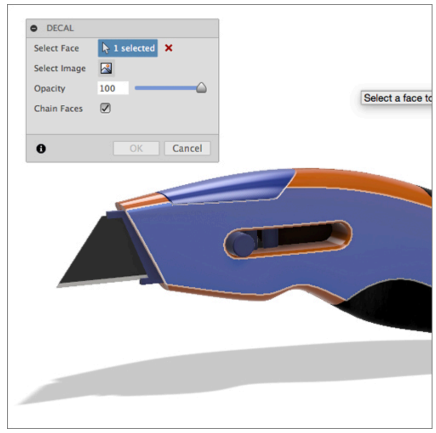
Step 1: Download the image file. Let's start by downloading the image that you will use as the decal. The Autodesk logo has been supplied in the learning materials and needs to be downloaded to your computer to be applied.

1. In the **Data Panel** find the item named **Autodesk Logo.jpg.**
2. Double-click on the thumbnail to launch **A360**.
3. Click on the blue **Download** button on the upper right side of the window.
4. Place the file in a location where you can easily find it. Your desktop for example.

Step 2: Apply the decal

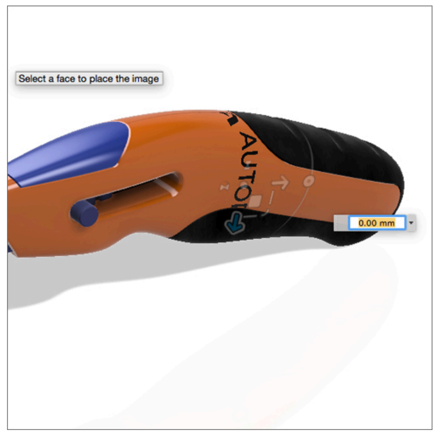
1. In the Render menu bar select **Setup > Decal**.
2. Select the body of the utility knife to highlight it. This is the surface you will apply the decal to.
3. In the **Decal** window, click on **Select Image.**
4. From the file menu go to the location where you saved **Autodesk Logo.jpg**, select the file and click **Open**.





Step 3: Adjust the decal

1. Adjust your view of the knife so that you can see the side of the handle.
2. Use the **rotate** handle on the decal manipulator to rotate the decal so that it is lined up with the handle.
3. Use the XY Plane scale handle to scale the decal down so that it looks correct.
4. Click **OK** to accept the decal location.




AUTODESK

Lesson 4: Changing the Environment Settings

The Environment Setting controls the lighting, background color, and visual effects in the rendering workspace. In Fusion 360 an environment dome with an environment image map (called a high dynamic range image or HDRI) attached to it constantly. In this lesson we will choose the environment, change the background color and turn on effects.

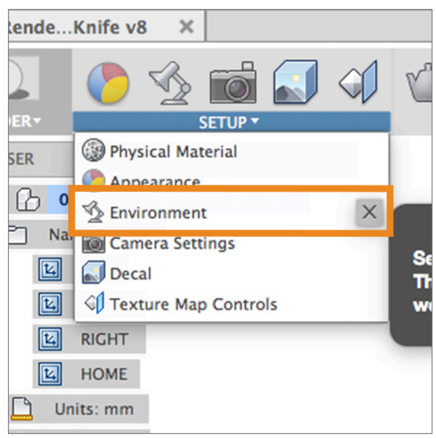
Learning Objectives

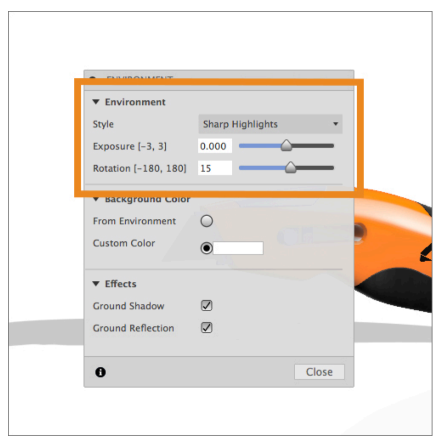
1. Changing the environment
2. Rotating the environment
3. Changing the background color
4. Changing the ground effects

Step-by-step Guides

Step 1: Change the Environment settings

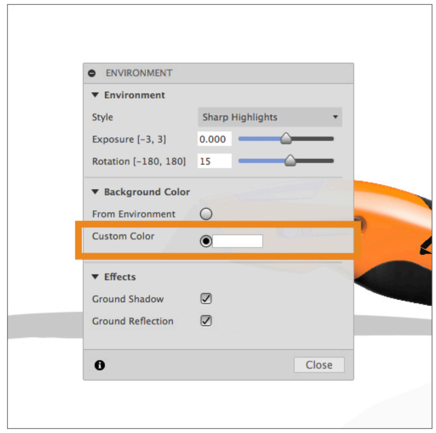
1. Click on **Setup > Environment**.
2. From the **Style** dropdown list select **Sharp Highlights**.
3. Use the slider next to **Rotation** to rotate the environment image around the model.
4. As you move the slider you will see reflected highlight change on the design and the shadows move along the ground.
5. Rotate the environment until you see a highlight across the right side of the knife.
6. Type **15** in the field next to **Rotation**.





Step 2: Choose a background color. By default the **background color** is defined by the **environment style** you choose. You have the option to change the background color to whichever color you want.

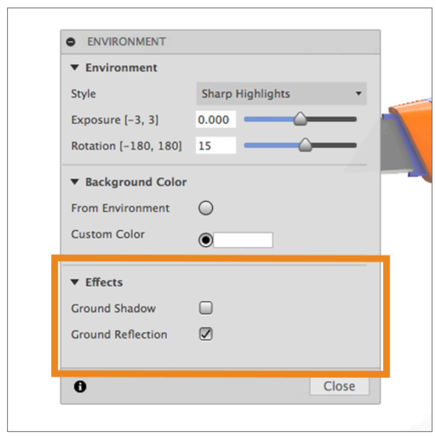
1. In the **Background Color** section click on **Custom Color**.
2. Click on the color swatch next to **Custom Color** to open the color chooser window.
3. In the RGB fields enter **255, 255, 255** to change the background color to white.
4. Select **OK** to close the color editor window and apply the changes.



Step 3: Change the **Ground Effects**. You have the option to have your model cast a shadow or to reflect your model on the ground plane of the environment.

1. Click on the button next to **Ground Shadow**.
2. In the **Environment** section change the rotation value.
3. Notice that the shadow location is changing based on the location of the light sources in the chosen environment style.
4. Click on the button next to **Ground Shadow** to turn it off.
5. Click on the button next to **Ground Reflection** to show a reflection of the model in the virtual floor.
6. Select **CLOSE**.

Note: Correct shadow computation takes a lot of time to render. If you want to speed up the render time avoid using ground shadows.



Lesson 5: Creating an image with Rendering

Now that the design has materials applied and the environment is set correctly it is time to create a rendered image. Fusion 360 uses Ray Tracing to create an image. Ray Tracing attempts to simulate the natural flow of light in your scene using a technique called Global Illumination (GI) which takes in to account not only the direct light that comes from a light source but also indirect light that reflects off of other surfaces in your scene.

There are 2 types of rendering methods that you can use in Fusion 360:

- **Local Ray Tracing:** Uses your computer CPU to create photo realistic images from your Fusion 360 models. This Ray Tracer works in is real-time, meaning that as soon as you click on the **Enable Ray Tracing** icon your computer will start rendering the image immediately. The image will start off noisy and will start clearing up. If you change the orientation of the model or change materials and environment the Tracer will restart the rendering process. The length of time needed to create the image is dependent on the **Quality** setting and the number iterations (or passes) that are needed to create the image. The size and resolution of your computer screen determines the size and resolution of your final image. You don't need to be connected to the Internet to start the Ray Tracer.

- **Cloud Rendering**: Uses a rendering engine in the Autodesk cloud to create photo realistic images from your Fusion 360 models. The size and resolution of the images can be set in advance of starting the cloud render. The advantage of using the cloud render is that it will not require any of your computer resources to create an image and create an image faster than your local computer. However, using the cloud rendering service may require Cloud Credits to create images and you must be connected to the Internet to start a cloud render.

Learning Objectives

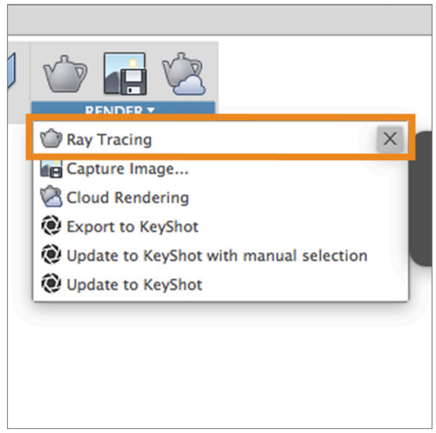
1. Using local ray tracing to create an image
2. Using cloud rendering to create an image
3. Viewing cloud rendered images in the Render Gallery

Step-by-step Guides

Step 1: Render an image using local Ray Tracing.

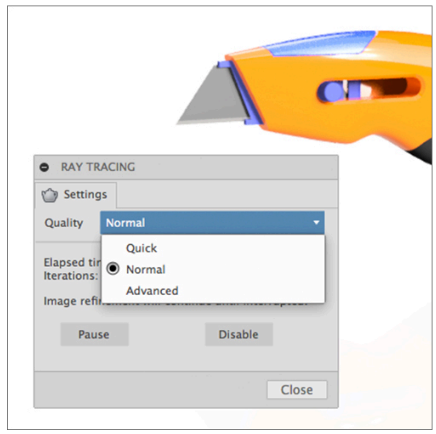
1. From the Render tool bar select **Render > Ray Tracing**.

Note: As soon as you start Ray Tracing you will notice that the screen starts to get "noisy" and starts to clear up as the Ray Tracer starts to work on the image. If you move the camera the Ray Tracer restarts the calculations.



Step 2: Change the Quality setting. There are 3 quality settings to choose from in the Ray Tracing settings. Quick, Normal and Advanced.

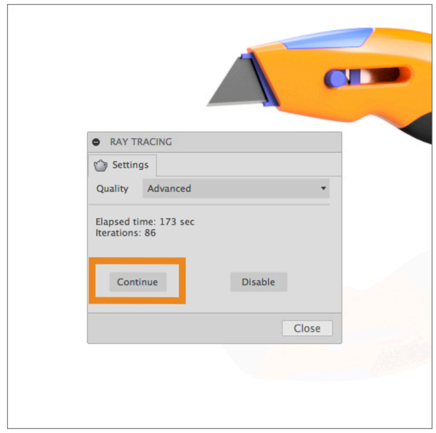
1. Set the **Quality** setting to **Quick**
2. The image clears up rapidly but the image is not high quality.
3. Change the **Quality** to **Normal**
4. The image clears quickly but you will notice that the edges of the model look jagged or pixelated and will slowly smooth out as the number of iterations rise.
5. Change the **Quality** to **Advanced**.
6. The image is very noisy to start and will continue to clear up over time.



Step 3: Pause and Disable the Ray Tracing

1. In the **Ray Tracer** window select Pause to pause the Ray Tracer.
2. Select **Continue** to let the **Ray Tracer** continue rendering from where it left off.
3. Select **Disable** to turn off the **Ray Tracer**.

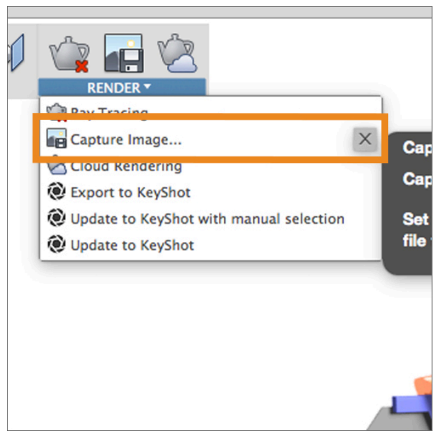
Note: Do not disable the **Ray Tracer** if you intend to capture an image of the render.

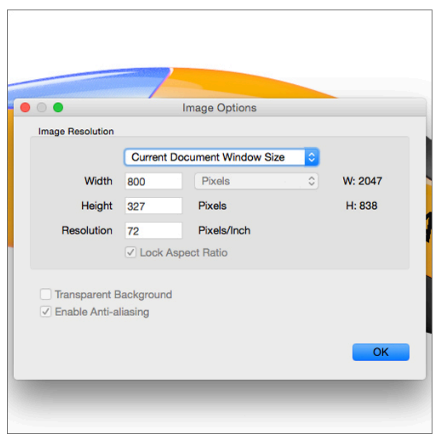


Step 4: Start the **Ray Tracer** and **Capture** an image. When rendering locally you manually capture an image when you are satisfied with the look of the image. All local **Ray Tracer** images will be based on the size and resolution of your screen.

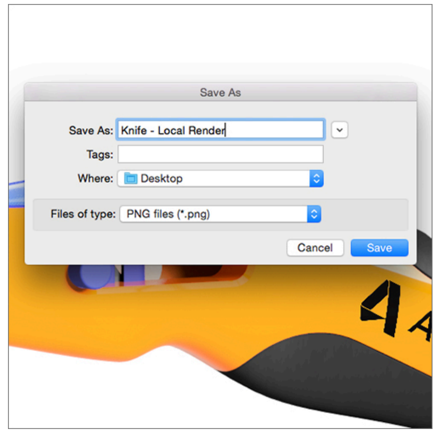
1. Select **Enable Ray Tracing** in the **Render** tool bar.
2. Change the **Quality** setting to **Normal**
3. Allow the **Ray Tracer** to run for about 120 seconds or until you are satisfied with the look of your image.
4. Select **Pause** in the Ray Tracing window
5. Select **Capture Image** from the **Render** tool bar.
6. Leave the **Image Resolution** options at the default and click on **OK**.
7. In the **Save As** window enter a name for the image file and select a location to save the image.
8. Click on the **Save** button to save the image.
9. In the **Ray Tracer** window click on the **Close** button.

Note: When creating an image using the local **Ray Tracer**. The image size and resolution is always based on the size and resolution of the screen you are using. You do not have



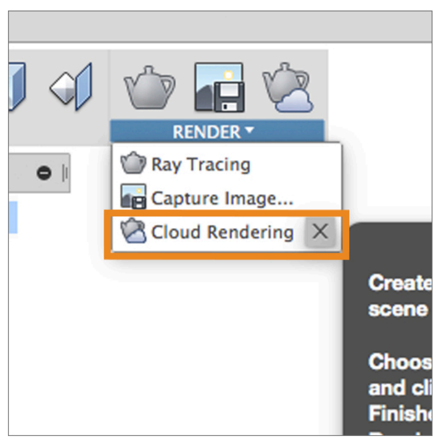


independent control over the size and resolution of the final image.



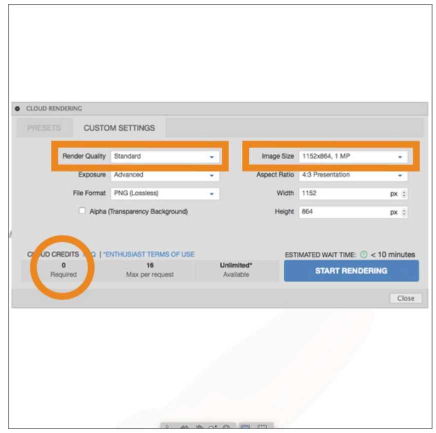
Step 5: Start a Cloud Rendering. The steps for creating an image using the Autodesk cloud is different from creating a local Ray Tracer image. With cloud rendering you have more choices for image size and resolution, the images you create are stored in the **Render Gallery** in the cloud. Images created using the the Autodesk cloud can cost **Cloud Credits**.

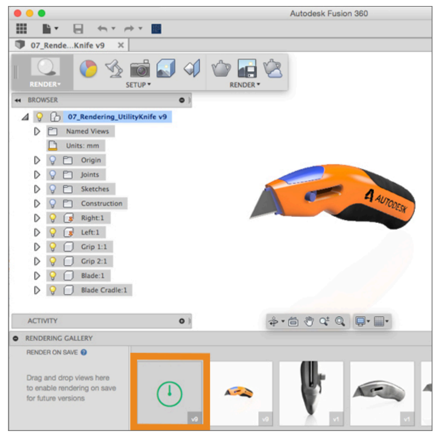
1. In the **Render** toolbar select **Render > Cloud Rendering.** To open the Cloud Render options window.
2. The default size and resolution of your image will always be based on the size and resolution of your screen. The bottom of the window shows how many Cloud Credits will be used to create the image.
3. In the **Cloud Render** window change **Render Quality** to **Standard.**

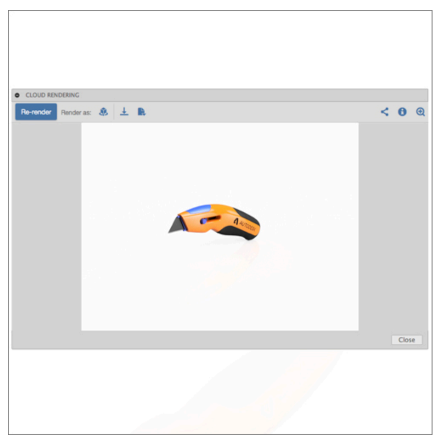


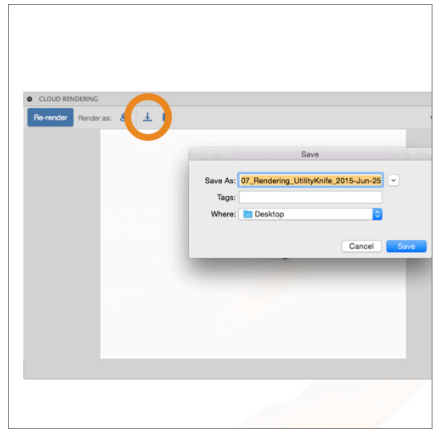
4. Change the **Image Size** to Web > 1152x864, 1 MP. At the bottom of the window the cost of the **Cloud Credit** is now **0**.
5. Click on **Start Render**.
6. In the Render Gallery you will see a green clock icon appear. This indicates that the render job has been sent to the Autodesk cloud and is the queue to be rendered to an image.
7. When the image is completed a thumbnail of the image will replace the clock icon.
8. Click on the image in the **Render Gallery** to view the image full size.
9. To download a copy of this image to your computer click on the download icon at the top of the **Cloud Render** window.
10. Enter the location where you would like to save the image and click on the **Save** button.
11. In the **Cloud Rendering** window click on the **Close** button.

Note: The Render Gallery shows you all of the cloud rendered images that have been created.



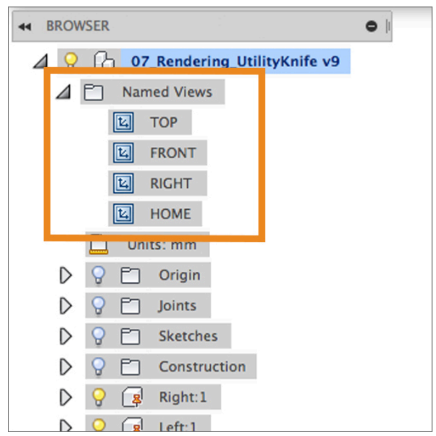


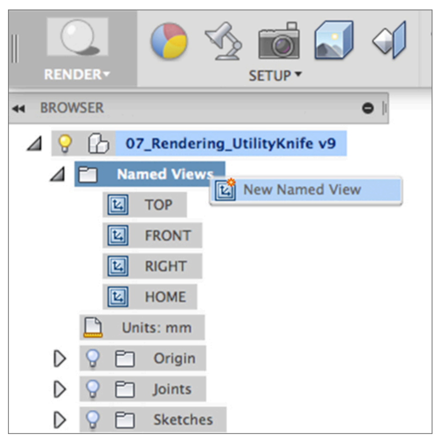




Step 5: Create a cloud render using Named Views. The cloud renderer automatically creates small sized rendered images based on the **Top**, **Front**, **Right** and **Home** named views every time you manually save a new version or an auto save is done. If you want to create automatic Cloud Render images of a specific view you can create a new named view to accomplish this.

1. In the **Browser** click on the small arrow next to **Named Views** to expand the list of current named views.
2. Arrange your design in the main window in a way that you would like your Cloud Rendered image to look.
3. In the **Browser** right mouse button on Named View and select New Named View.
4. You have now created a new named view based on the current camera angel with the label **NamedView** in the **Browser**.
5. To change the label for your custom named view double click on **NamedView** to highlight it and type **Render** and hit your Enter key.
6. Click on the **Save** icon to save a new version of the design.
12. After the save is complete a green

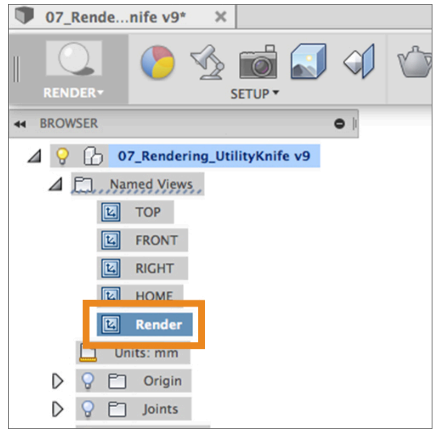


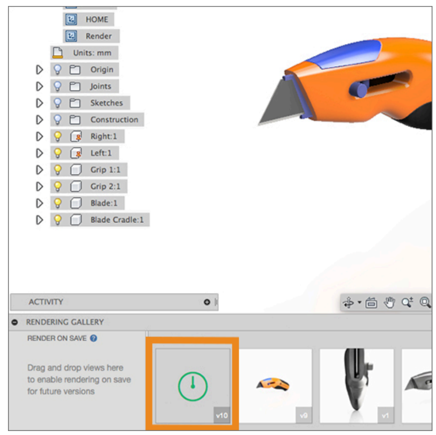


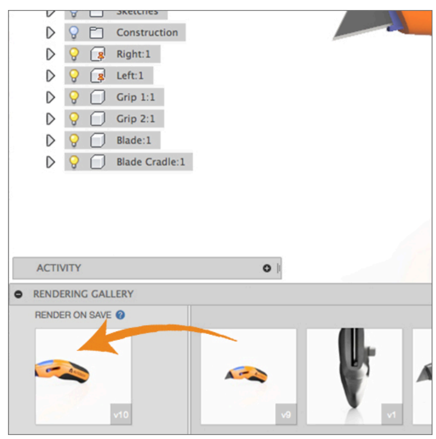
clock icon will appear in the **Render Gallery**.

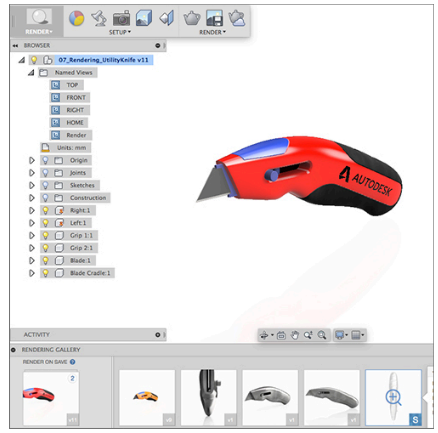
13. This indicates that the render job has been sent to the Autodesk cloud and is the queue to be rendered to an image.
14. When the image is completed a thumbnail of the image will replace the clock icon. Your custom named view is now saved.
15. Click and hold on the thumbnail image in the Render Gallery and drag it to the box on the left labeled **Render On Save**. Fusion 360 will now automatically re-render this Named View when a save or automatic save is performed.
16. Select **Setup > Appearance**.
17. Change the color of the material assigned to the body of the Knife.
18. Click the **Save** icon in the menu bar.

Note: After the save is completed, the thumbnail image of the named view you moved to **Render On Save** section of the **Render Gallery** will update. All the other images that are not in that section will will stay the same.

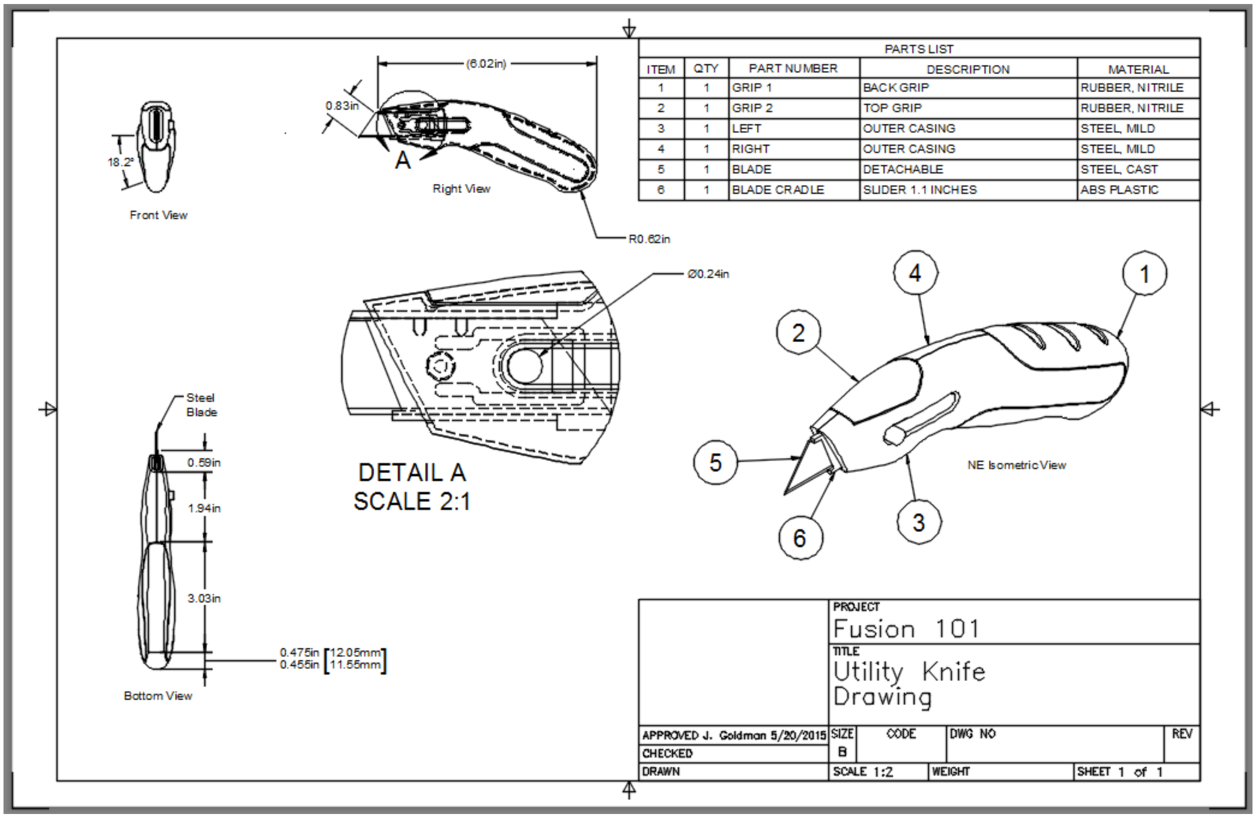








8.1: Drawings



Fusion 360 Drawings allows you to create 2D drawings from your Fusion 360 designs, which provides the ability to generate PDF and DWG documentation of your Fusion 360 model. When you create a drawing, it is generated as a derived document of a Fusion 360 model, and it shows up in the Data Panel as a unique item in the active project.

Lesson 1: Introduction to Drawing Views

Learning Objectives

1. What are Drawing Views?
2. Initiate a New Drawing
3. Place a Base View

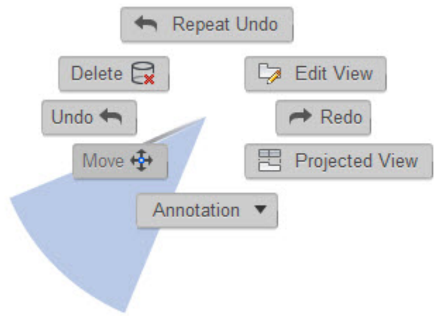
A drawing view is an object that contains a 2D projection of a 3D model.

When you create a drawing from the Fusion 360 modeling environment, the system automatically launches a new tab of the Drawing workspace and generates a 2D projection of the components you select. The drawing view generated is referred to as a **base view**. Once you place the base view in the drawing, you can generate orthogonal and isometric projected views from it. A **projected view** takes the characteristics of a base view and projects it from a different angle.

About the Marking Menu

The **marking menu** is a radial display of the most frequently used commands. It also includes an overflow menu that provides quick access to all commands found in the toolbar.

Using the marking menu can be the fastest way to input a command in any workspace. You can access the marking menu by right-clicking anywhere within the drawing canvas.



As you move the cursor from the center of the marking menu towards a command, its wedge highlights. Clicking anywhere in the wedge launches the command.

Datasets Required

In Samples section of your Data Panel, browse to:

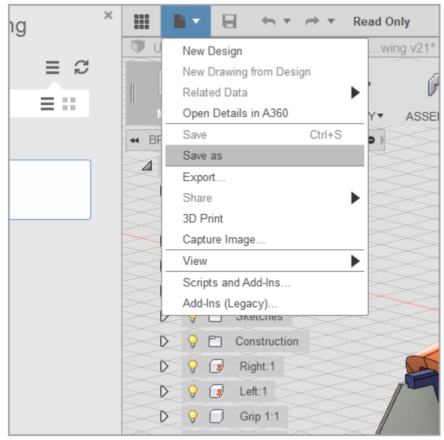
Fusion 101 Training > 08 – Drawing > **08_Drawings Utility Knife**

Open the design and follow the step-by-step guide below to get started with the lesson.

Create a Drawing

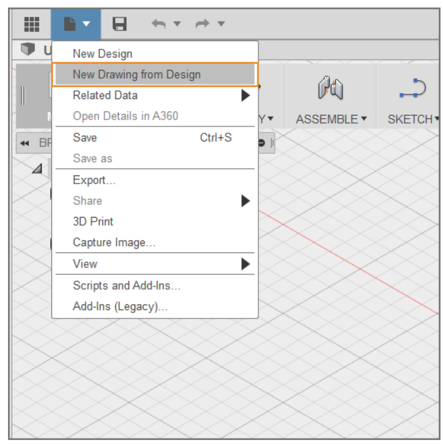
Step 1: – Save the model

1. If the model is "Read Only," go to the File Menu and click **Save As**.
2. Give the model a location and a new name, then click **Save**.



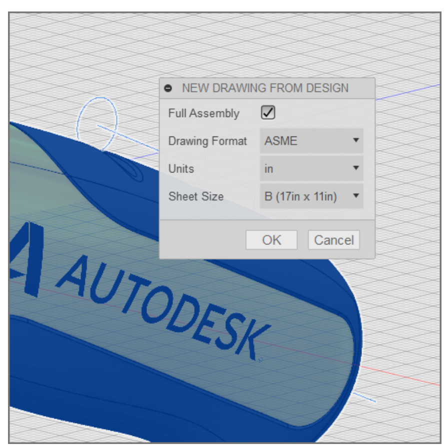
Step 2: – Initiate a New Drawing – With the Utility Knife design open, do the following:

1. Click on the File dropdown menu from the top menu bar.
2. Select **New Drawing from Design** from the file menu dropdown.



Step 3: – Choose Assembly

1. Select "Full Assembly" from the dialog and click **OK** to initiate the drawing.
2. Notice that a new file tab is automatically generated.



Note: If you un-check "Full Assembly" from the New Drawing dialog, you can select individual or multiple components to create a drawing of part of the assembly.

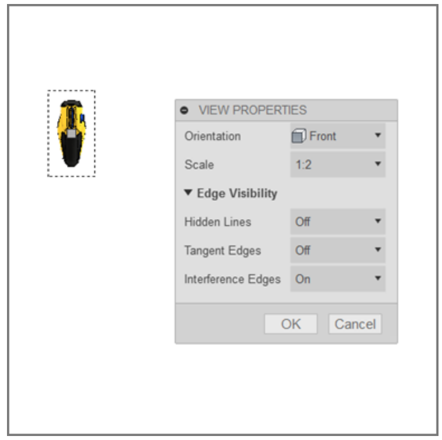
Tip: When selecting multiple components, it helps to use **Ctrl+click.**

You can set the drawing format, units, and sheet size before you create a drawing. **However, Drawing Format and Units cannot be changed once you create a new drawing.**

Step 4: – Commit a Base View

1. Move your cursor around the screen. Notice that the Base View preview is attached to the cursor.
2. Click on the top left quadrant of the sheet to place the view.
3. Click OK to commit the view.

Note: Notice that after the view is committed, the shaded preview matures into a 2D line drawing of the view.



Lesson 2: Projected Views and Detail Views

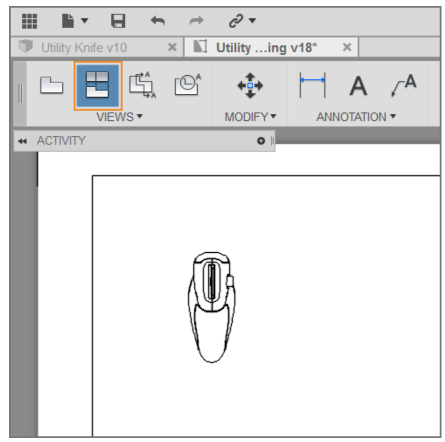
Learning Objectives

1. Create Projected Views
2. Edit View Properties
3. Create Detail Views

Add to the Layout

Step 1: – Initiate **Projected View** – Now that you we've created a base view of the model assembly, let's create projected views and edit their properties to create a complete drawing layout.

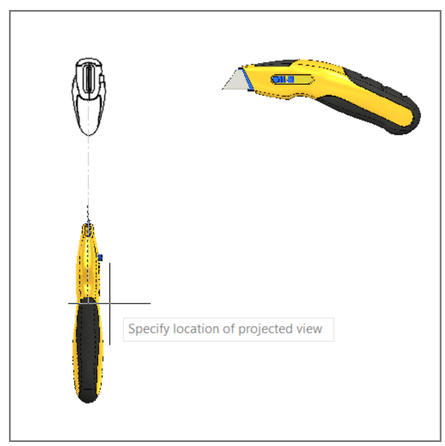
1. From the View Toolbar, select **Projected View**.
2. Click the base view to select it as the parent view from which the projected views will be created from and associated.



Step 2: – Place the views

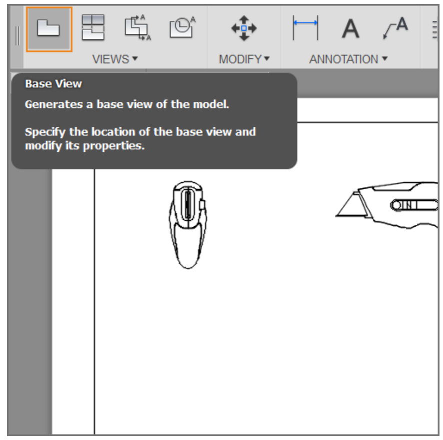
1. Drag the cursor to the right of the base view, and notice that the projected view is previewed based on this alignment.
2. Select to the right of the existing base view to place a projected view.
3. Select to the bottom of the existing base view to place a second projected view.
4. Press **Enter** to finish the task.

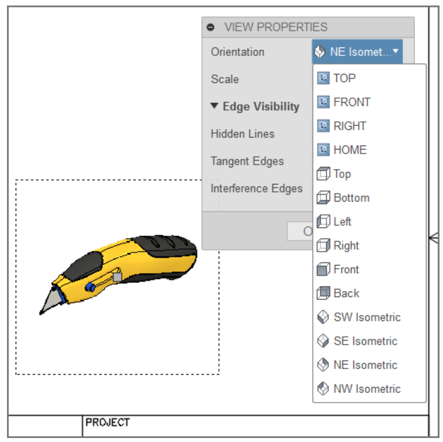
Note: A projected view inherits all its properties from the parent. When the properties of the parent view change, the corresponding properties on the projected view also change. You can change the properties of a projected view by double-clicking it.



Step 3: – Create an Isometric Base View

1. From the View Toolbar, select **Base View**.
2. Click to place the view in the lower right of the sheet layout, above the title block.
3. Set the Orientation to **NE Isometric.**
4. Press OK to commit the view.



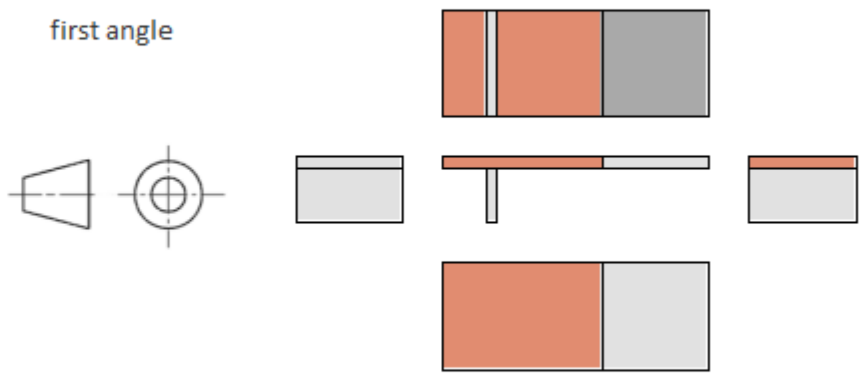


About Projected Views & View Options

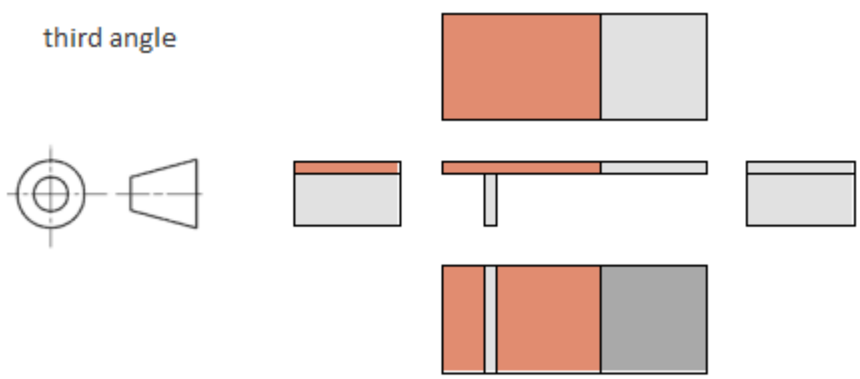
Projected views maintain a parent-child relationship with the base view it was generated from. They inherit their properties from the parent base view. If necessary, you can override them after you create the projected view.

The projection angle defines the method employed to generate projected views.

first angle



third angle



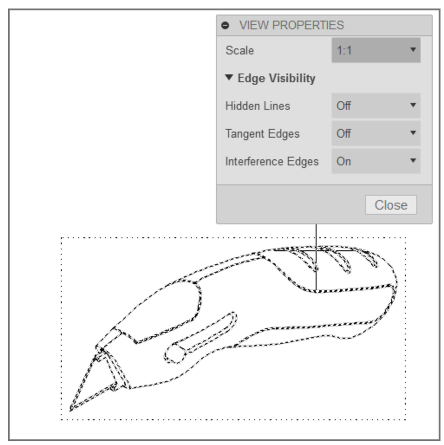
First Angle Projection – When you use first angle projection, projected views placed to the right of a base view depict the appearance when viewing it from the left. Projected views placed below the base view depict the appearance from above. The ISO drafting standard specifies that drawings use first angle projection. **By default, when the drawing format is set to ISO, the Drawings workspace will use first angle projection.**

Third Angle Projection – When you use third angle projection, projected views placed to the right of a base view depict the appearance when viewing it from the right. Projected views placed below the base view depict the appearance from below. The ASME drafting standard specifies that drawings use third angle projection. **By default, when the drawing format is set to ASME, the Drawings workspace will use third angle projection.**

Edit the Layout Views

Step 1: – Edit the isometric base view - Now that you have created a base view and several projected views of the model, let's use the View Properties settings to further customize the view layouts.

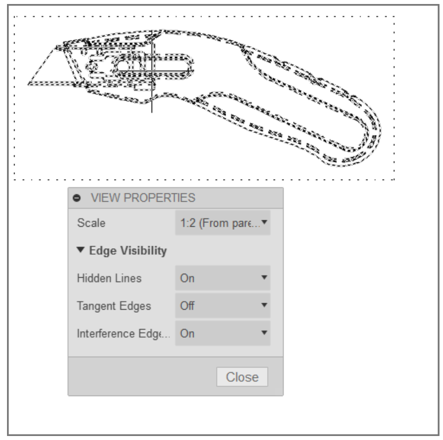
1. Double-click anywhere inside the selection boundary of the isometric view to activate it.
2. Select the Scale ratio in the View Properties dialog box to scale the base view.
3. Change the Scale to: **1:1.**
4. Click **Close** to accept the drawing view changes.



Step 2: – Edit the right projected view

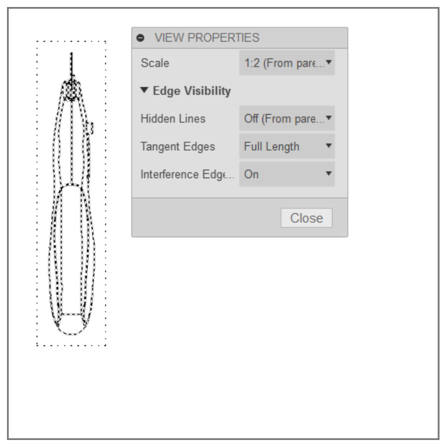
1. Double-click anywhere inside the selection boundary of the right projected view to activate it.
2. In the View Properties dialog box, edit the properties of the component.
3. For **Hidden Lines**, select **On** from the drop-down list..
4. Click **Close** to accept the drawing view changes.

Note: Once the projected view properties are changed, they no longer inherit the settings of the base view. If you change the properties back to "From parent," they will once again inherit properties from the parent view.



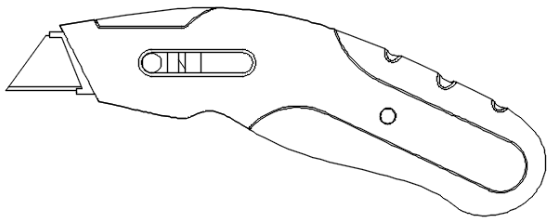
Step 3: – Edit the bottom projected view

1. Double-click anywhere inside the selection boundary of the bottom projected view to activate it.
2. In the View Properties dialog box, edit the properties of the component.
3. For **Tangent Edges,** select **Full Length** from the drop-down list.
4. Click **Close** to accept the drawing view changes.

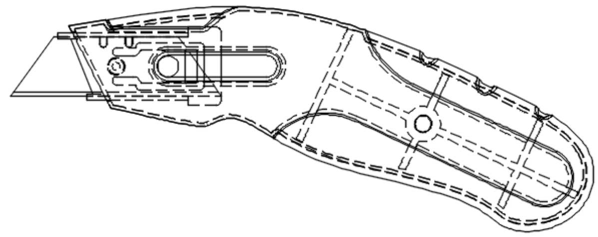


View Properties

Hidden Lines – Select On or Off from the drop-down list to display hidden lines within the selected base view. The Hidden-line representation suppresses or exposes lines, edges and other objects that are located behind other three-dimensional objects. This view property can be particularly helpful when trying to visually communicate the inner workings or dimensions of a complex assembly or part.

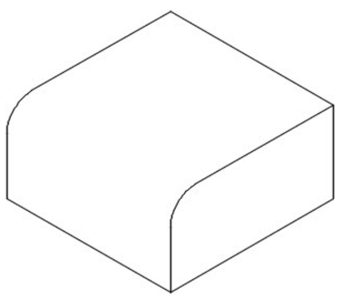


Hidden Lines OFF

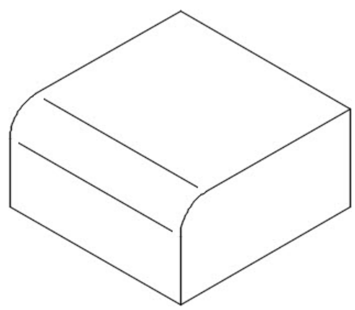


Hidden Lines ON

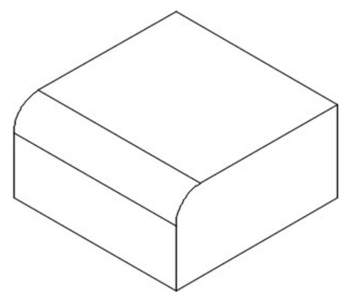
Tangent Edges – Select Full length, Shortened or Off from the drop-down list to display Tangent edges within the selected base view. Tangent edges mark the transition between a flat surface and a rounded edge, most commonly seen as filleted edges. Tangent edges can be set to Full Length, Shortened, or Off.



Tangent Edges Off

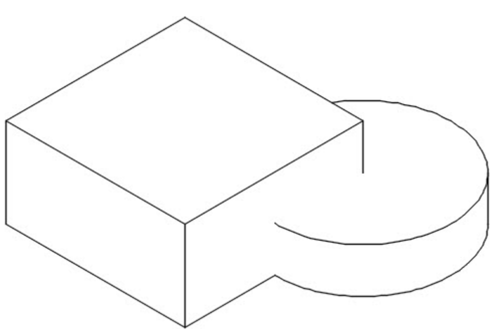


Tangent Edges Shortened

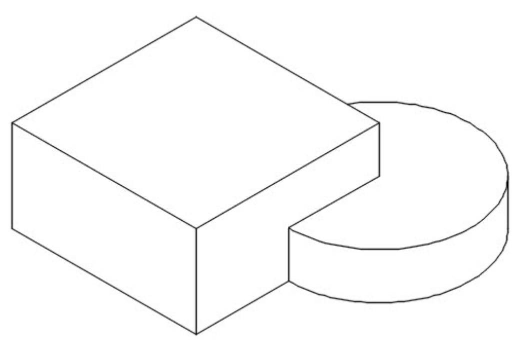


Tangent Edges Full Length

Interference Edges – Select On or Off from the drop-down list to display of Interference edges within the selected base view. An interference edge occurs when two faces of two components intersect. When Interference Edges are turned on, an edge is displayed that shows where the two components meet. When selected, associated drawing views are to display both hidden and visible edges that were previously excluded due to an interference condition (press, or interference fit conditions, threaded fasteners in tapped holes where the hole feature is modeled with the minor diameter).



Interference Edges Off



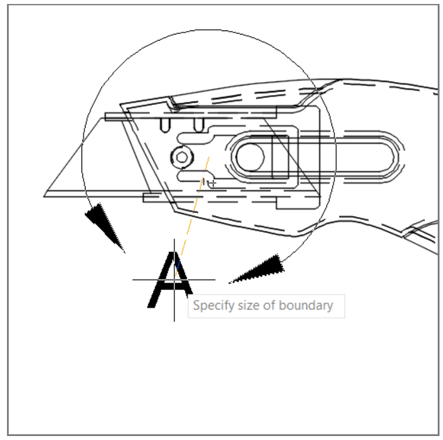
Interference Edges ON

Create a Detail View

A detail view is a projected view that shows a specific portion of a view at an enlarged scale.

Step 1: – Create a Detail View

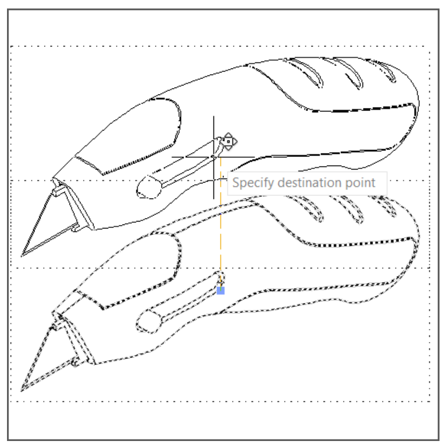
1. Select **Detail View** in the View Toolbar.
2. Select the right side view as a parent view.
3. Click the center point for the detail boundary.
4. Click again to specify the size and location of the detail boundary, and then click to place the detail view.
5. Change the Scale of the detail view to **2:1**, then press OK to generate the detail view.



Move Objects

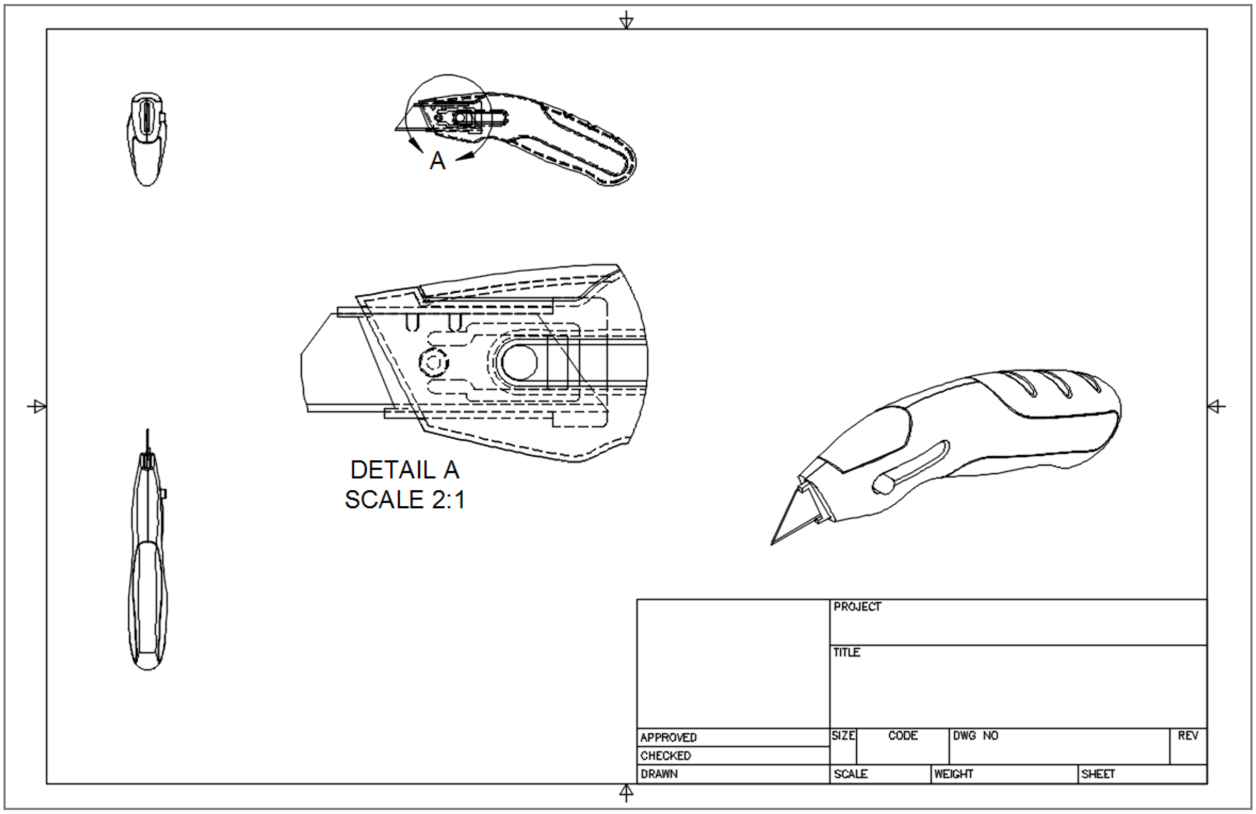
Click anywhere on an object to select it, click the gray grip in the center of the object, then click a new location for the object.

Note: The Move action works the same way for all views, text objects, dimensions, and balloons.



Move your objects around your drawing so they are nicely spaced, leaving the top right of the drawing space open (for a Parts List, to be added later).

By now, your drawing should look something like this:


A
DETAIL A
SCALE 2:1
PROJECT
TITLE
APPROVED
CHECKED
DRAWN
SIZE
CODE
DWG NO
REV
SCALE
WEIGHT
SHEET

Lesson 3: Text and Leader Notes

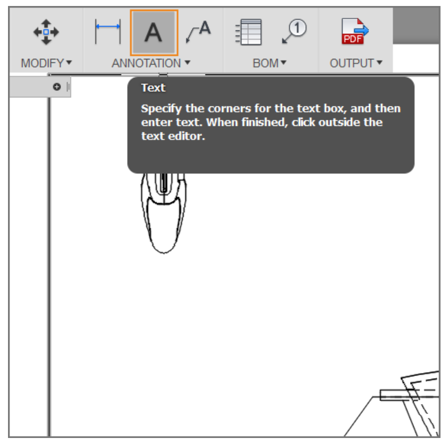
Learning Objectives

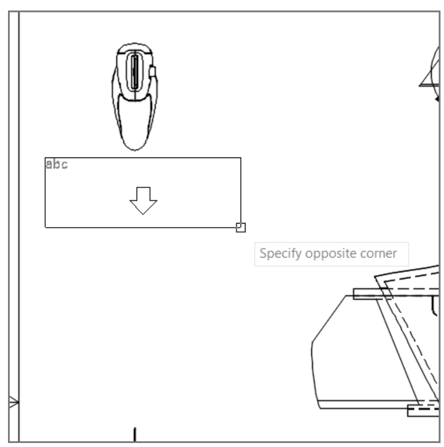
1. Create Text
2. Create Leader Notes
3. Reposition Annotation and Edit Text

Adding Text and Leaders

Step 1: – Create Text

1. Initiate **Text** from the toolbar.
2. Select two corners below the Front View to create a text box.
3. Type the following text into the text box: “Front View”.
4. Select anywhere outside of the text box to commit the action.
5. Repeat the process to add text below the to the other views, naming them “Bottom View,” “Right View,” and “NE Isometric View”.

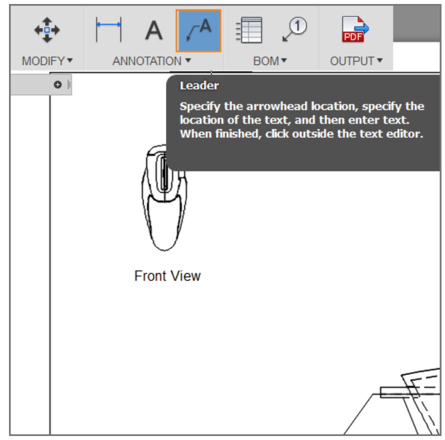




Step 2: – Create Leader Notes

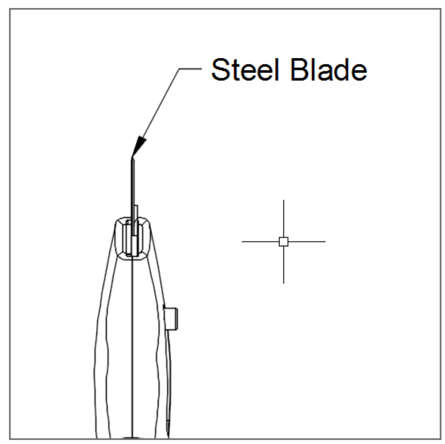
1. Initiate **Leader Note** from the toolbar.
2. Click the blade of the bottom view to place the start of the leader.
3. Click outside the view to place the end of the leader.
4. Type the following text: "Steel blade."
5. Select anywhere outside of the text box to commit the action.

Tip: When selecting a part of an object, you'll see green shapes on the object. These are called **Object Snap points,** and they help you specify a precise point on the object, like an endpoint, a midpoint, or a center point.



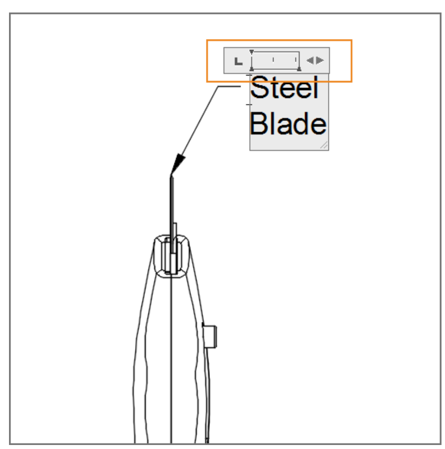
Step 3: – Reposition Leader Notes

1. Click on the leader note to activate it.
2. Click any of the gray grips to move the leader note to a new location.
3. Click anywhere outside the leader note to commit the action.



Step 4: – Edit Text

1. Double-click on the leader note to activate the text editor
2. Drag the < > to the right to format the text into 2 lines.
3. Click anywhere outside of the text box to commit the action.

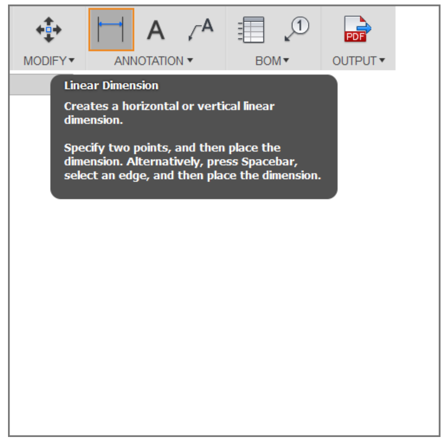


Lesson 4: Dimensions

Learning Objectives

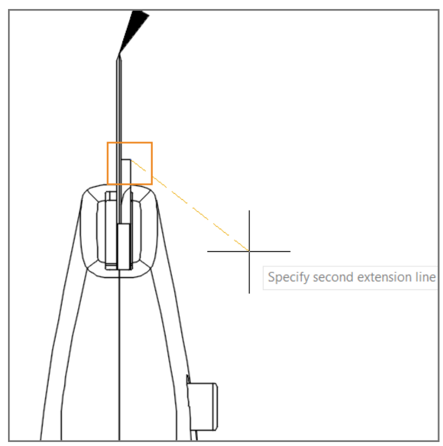
1. Create Linear, Aligned, Angular, Radial, and Diameter Dimensions
2. Reposition Dimensions
3. Use the Baseline and Chain Dimensioning tools

Create Dimensions

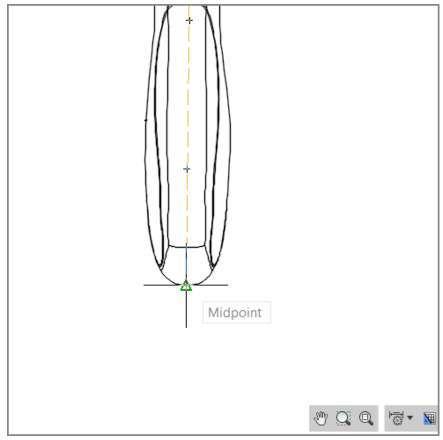


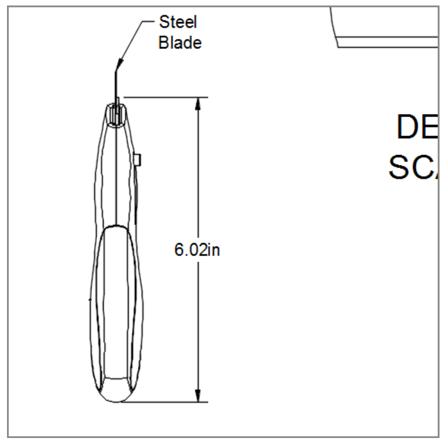
Step 1: – Create a Linear Dimension

1. Initiate **Linear Dimension**.
2. Use Object Snaps to click the two endpoints of the bottom view. A preview is displayed on your curser.
3. Drag the cursor out to the right
4. Click again to place the dimension and complete the action.



Note: You can also press **Spacebar** and then select an edge to create a dimension with only one click.

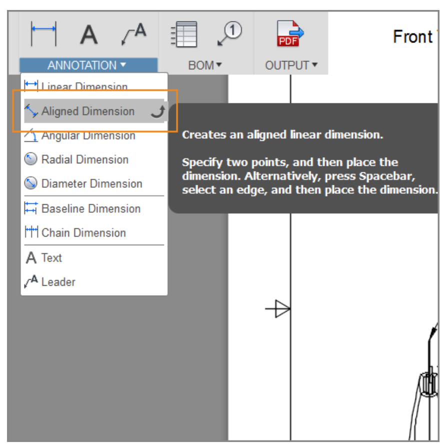


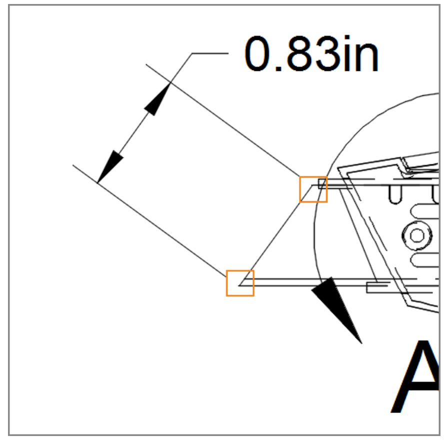


Step 2: – Create an Aligned Dimension

1. Initiate **Aligned Dimension**.
2. Using Object Snaps, click the top edge and bottom point of the cutting blade on the right view.
3. Drag the cursor out to the left to see a preview of the dimension.
4. Click again to place the dimension and finish the action.

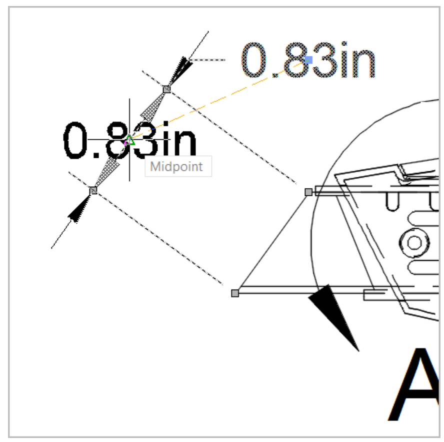
Note: Linear dimensions measure the horizontal or vertical distance between two points. Aligned dimensions measure the true distance between two points.





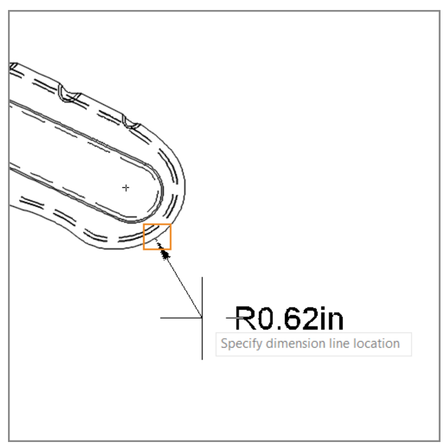
Step 3: – Reposition the Dimension

1. Click the aligned dimension.
2. Click the gray grip on the text.
3. Move the text so it is between the dimension arrows.
4. Click anywhere outside the dimension to complete the action.



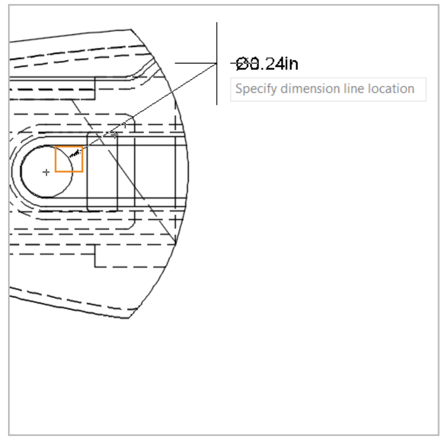
Step 4: – Create a Radial Dimension

1. Initiate **Radial Dimension**.
2. Select the curved end of the handle in the right side view.
3. Drag the cursor down and to the right to see a preview of the dimension.
4. Click again to place the dimension and complete the action.



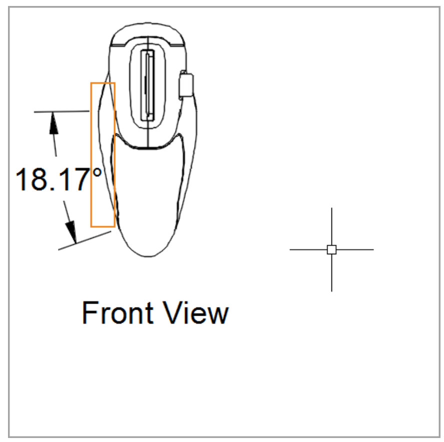
Step 5: – Create a Diameter Dimension

1. Initiate **Diameter Dimension**.
2. Select the blade slider on the detail view.
3. Drag the cursor up and to the right and see a preview of the dimension.
4. Click again to place the dimension and finish the action.



Step 6: – Create an Angular Dimension

1. Initiate **Angular Dimension**.
2. Click the side of the knife in the Front View.
3. Drag the cursor to the left to see a preview of the dimension.
4. Click again to place the dimension and finish the action.



Use Chain and Baseline Dimensions

The Baseline Dimension tool creates multiple linear dimensions measured from the same location. The Chain Dimension tool creates multiple linear dimensions placed end to end.

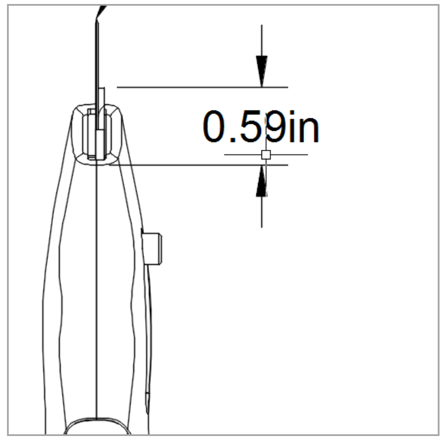
Baseline and chain dimensions derive from an original dimension. You must first create a linear dimension before you create baseline or chain dimensions.

Step 1: – Create a new Linear Dimension

1. Let's delete the Linear Dimension on the bottom view by selecting the dimension, then pressing the Delete

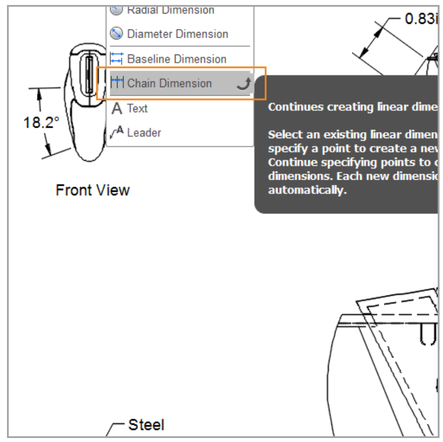
key, or by selecting Delete in the Marking Menu.

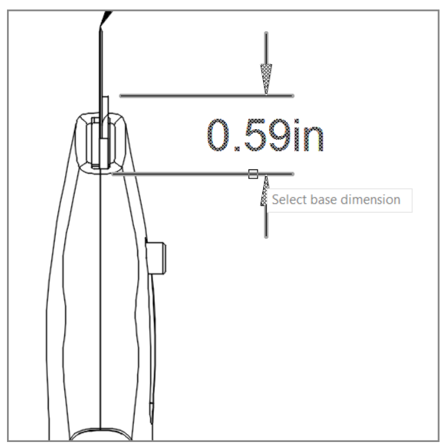
2. Create a new linear dimension at the end of the blade cradle.



Step 2: – Place Chain Dimensions

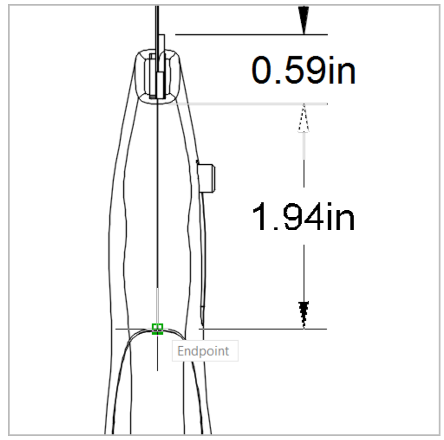
1. Select Chain Dimension in the Annotation toolbar.
2. Select the lower extension line of the newly placed dimension.

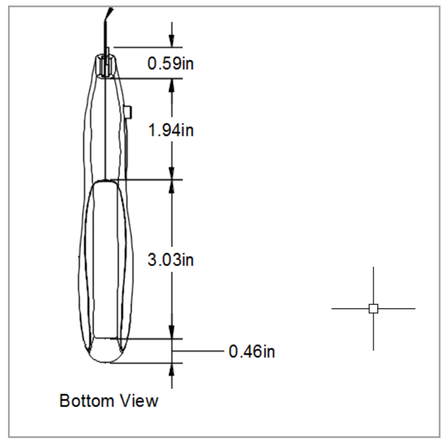




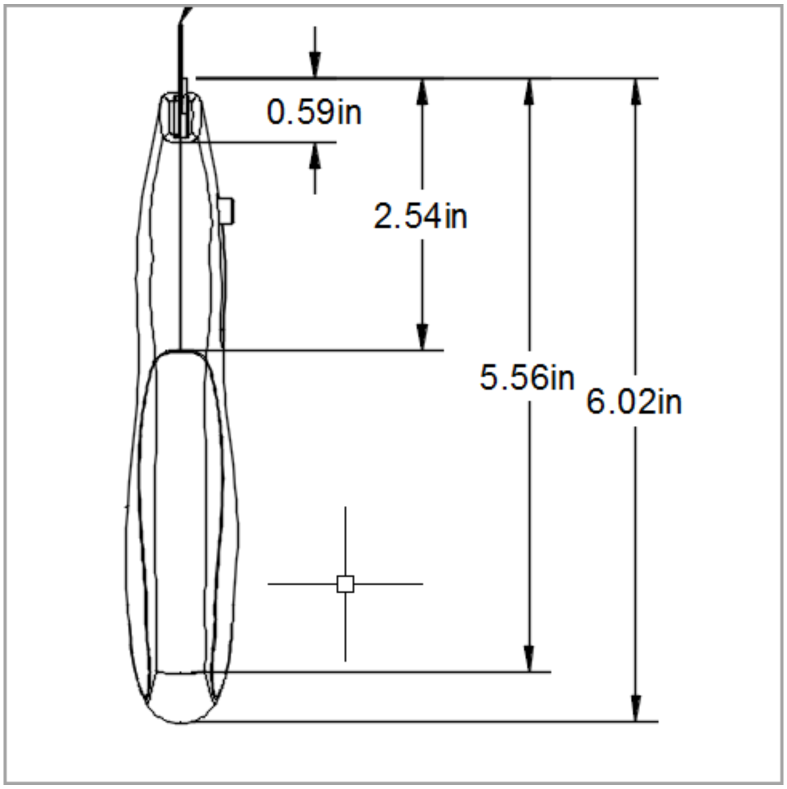
Step 3: – Complete the Chain Dimension

1. Click the end of the knife's grip to specify the endpoint of the first dimension.
2. Click again to specify another endpoint of a dimension.
3. Click once more at the very end of the bottom view.
4. To finish the action, press **Enter.**

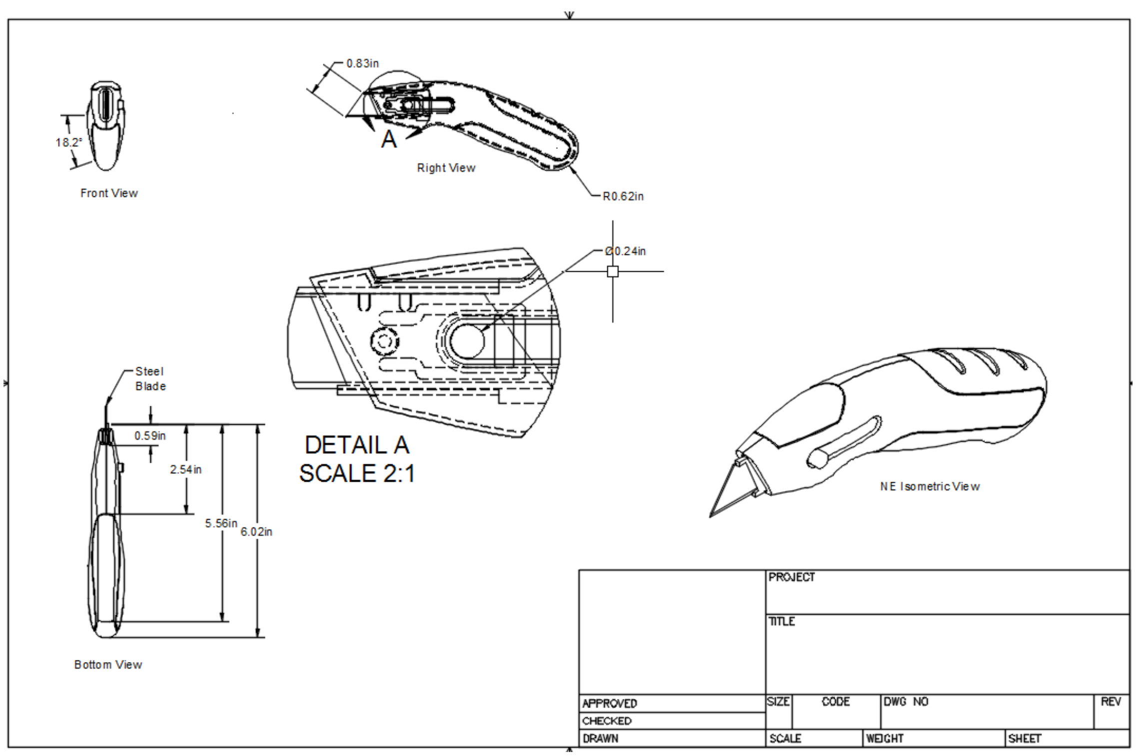




Baseline Dimensions work the same way as Chain Dimensions, only they create multiple dimensions that all begin from the same location. If you were to use Baseline Dimensions here, it would look like this:



By now, your drawing should look something like this:

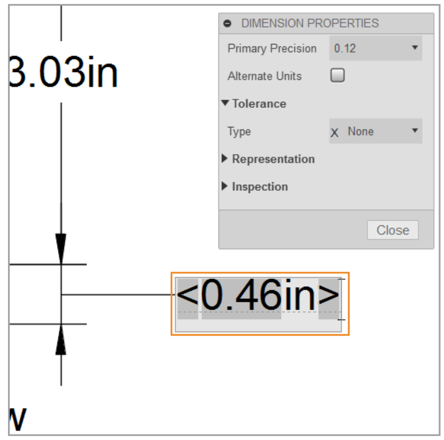


Modify the Properties of an Individual Dimension, and add Tolerance and Representation.

Step 1: – Access Dimension Properties

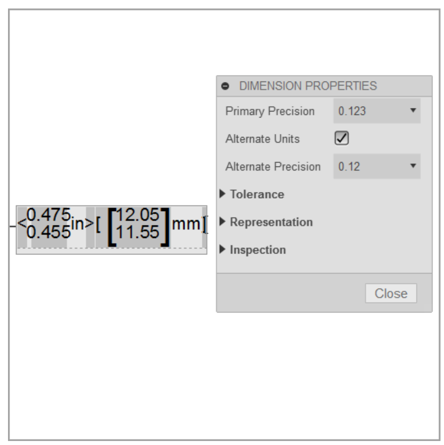
1. Double-click the dimension on the back grip of the bottom view.

The Dimension Properties dialog box opens.



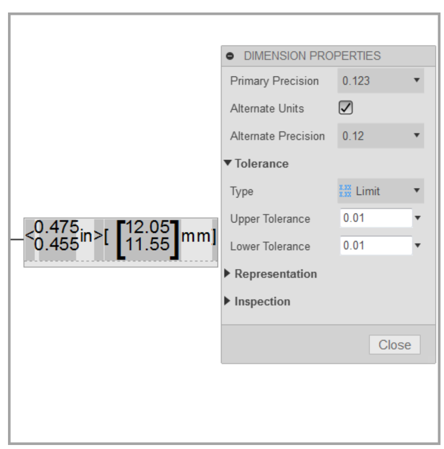
Step 2: – Modify the dimension's properties

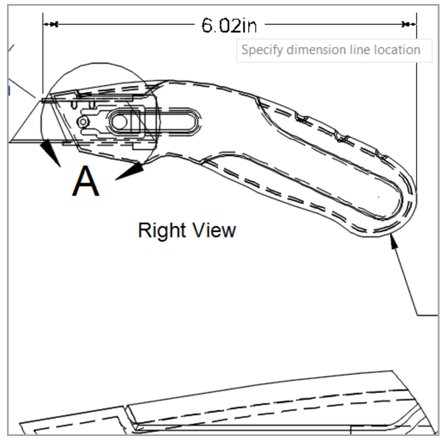
1. Change the primary precision for this dimension to **0.123**.
2. Have the dimension display Alternate Units.



Step 3: – Add tolerance to the dimension

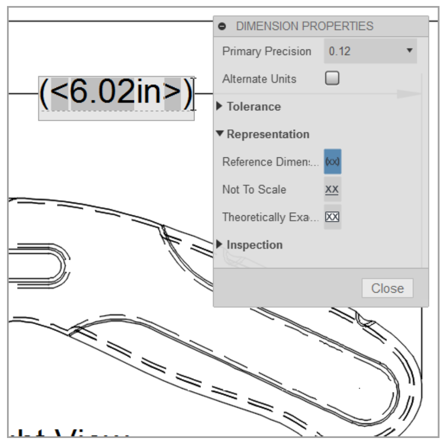
1. Under **Tolerance**, select Limit, and give it an Upper and Lower tolerance of **0.01.**
2. Press Close.





Step 4: – Create a reference dimension

1. Create a new linear dimension in the right view, using the corner of the blade cradle and the quadrant snap point on the end of the handle as snap points.
2. Under **Representation**, select Reference Dimension.



Lesson 5: Parts List, Balloons, and Title Block

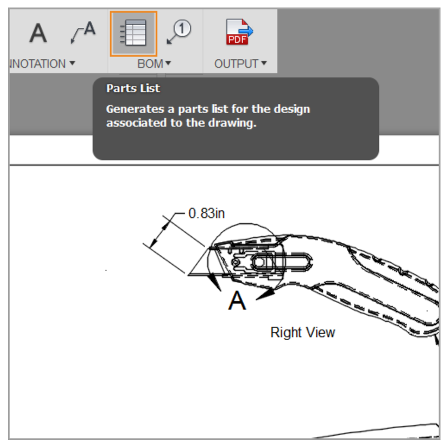
Learning Objectives

1. Insert a Parts List
2. Create Annotative Balloons
3. Populate the Title Block

A parts list is a table that catalogs the components of the design. It itemizes all components in the drawing, and includes the item number, the quantity, the part number, the description, and the material.

A balloon identifies a component included in the parts list by labeling it in the drawing.

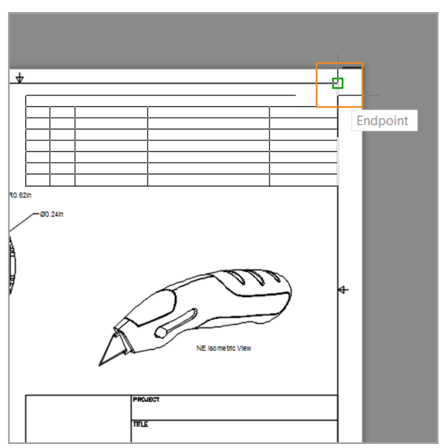
Insert a Parts List and Add Balloons

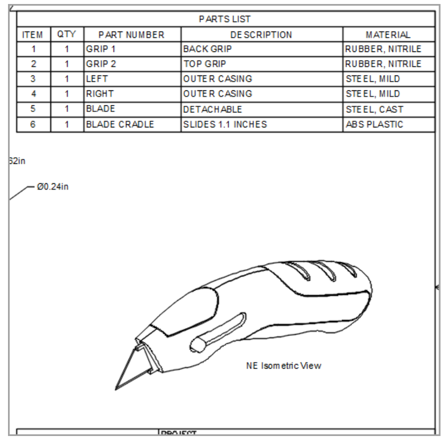


Step 1: – Insert a Parts List

1. Click **Parts List** and click the very top right corner to place the Parts List there (you may have to move some of your views over a bit, to make room).

The Parts List automatically populates, taking information from Model Space.

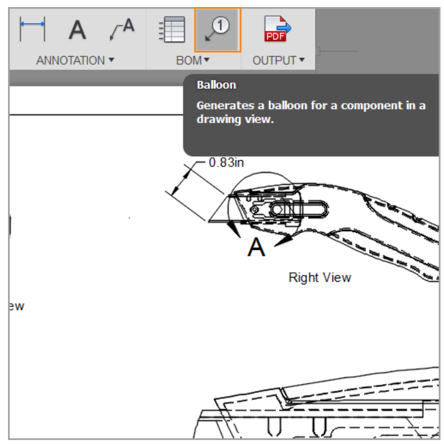


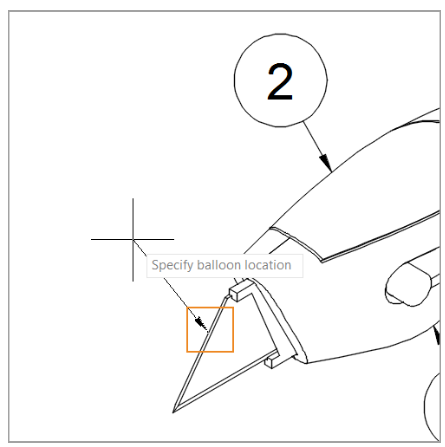


PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	GRIP 1	BACK GRIP	RUBBER, NITRILE
2	1	GRIP 2	TOP GRIP	RUBBER, NITRILE
3	1	LEFT	OUTER CASING	STEEL, MILD
4	1	RIGHT	OUTER CASING	STEEL, MILD
5	1	BLADE	DETACHABLE	STEEL, CAST
6	1	BLADE CRADLE	SLIDES 1.1 INCHES	ABS PLASTIC

Step 2: – Create Annotative Balloons

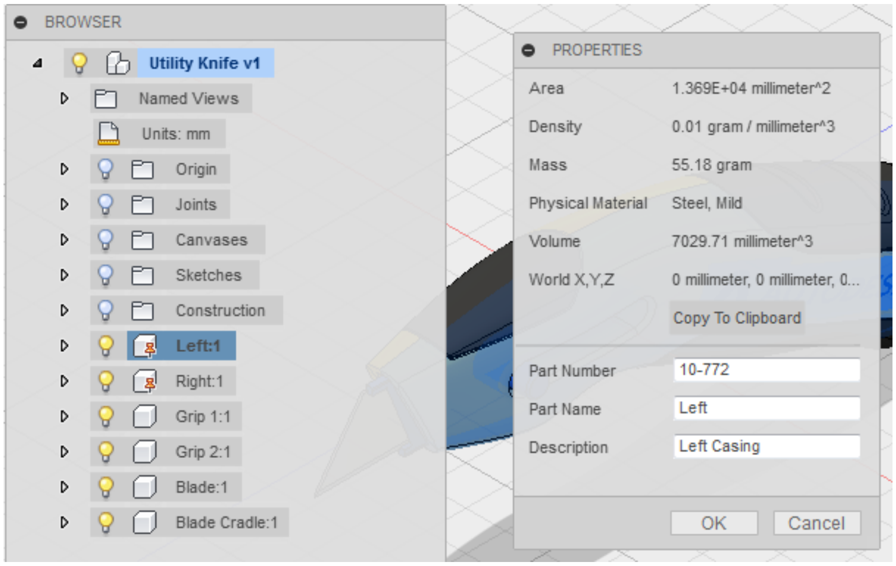
1. Click **Balloons** in the BOM toolbar.
2. On the NE isometric view, select an edge of a component.
3. Click to place the balloon.
4. Continue selecting components until all items in the Parts List are numbered.
5. When finished, press **Enter**.





About the Parts List

A component's part number and description can be modified in the Properties Panel in the Model workspace:



If you change the name or properties of a component in the Model workspace, the changes will be applied to the drawing once you associate the drawing with the saved model (see Associatively Update the Drawing).

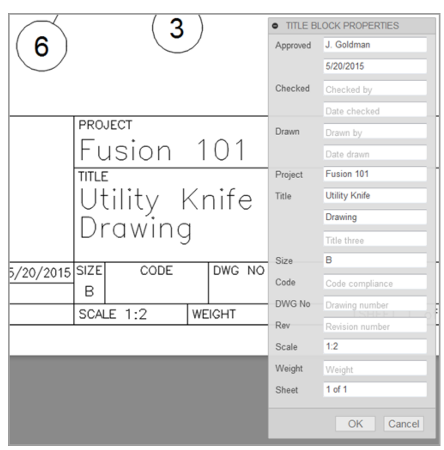
Note: The parts list table is not editable or customizable.

Populate the Title Block

To populate or edit the contents of the title block, simply double-click any boundary of the title block.

The Title Block Properties dialog box opens. Input your text, and click **OK** when finished.

Note: The title block is not customizable.



Lesson 6: Drawing Settings and Preferences

Learning Objectives

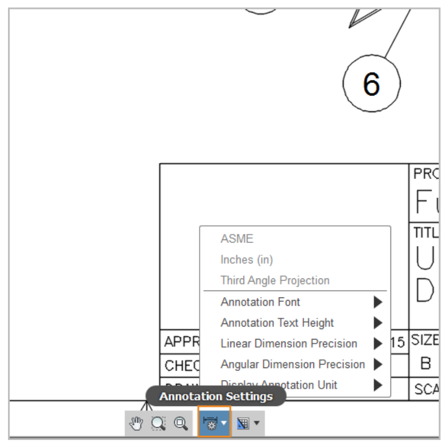
1. Modify the global annotation settings for the current drawing
2. Modify the properties of an individual dimension
3. Change the default settings for future drawings

You can modify annotation settings such as annotation font, annotation text height, dimension precision, and whether or not you want to display dimensions, by accessing **Annotation Settings** in the bottom of the drawing area. You can change the sheet size by accessing **Sheet Settings.**

Modify the Annotation Settings for the Current Drawing

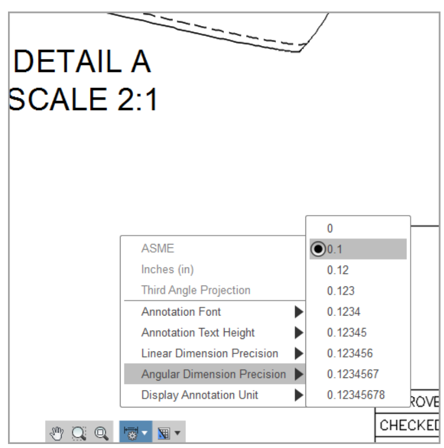
Step 1: – Access Annotation Settings

1. **Click Annotation Settings** in the bottom of the drawing area.



Step 2: – Edit the Annotation Preferences

1. Change "Display Annotation Unit" to **On** if it is not already set that way**.**
2. Change the Angular Dimension Precision to **0.1.**



The annotation settings apply to all dimensions and text in the drawing. However, you can change the properties of an individual dimension, which overrides any "global" settings you have changed.

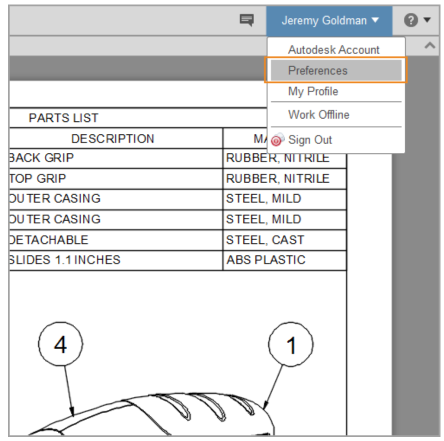
Modify the Default Settings

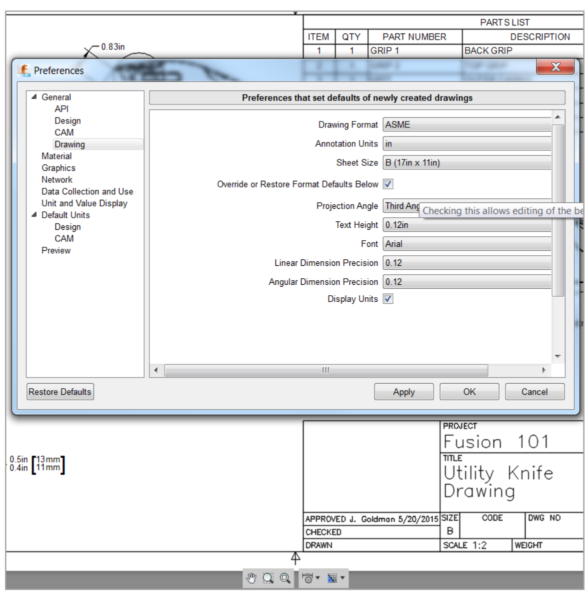
The default settings affect all future drawings.

Step 1: – Access Drawing Preferences

1. Click **Preferences** in the User Profile drop-down (your name).
2. Under General, click Drawing.

Here you will see the default settings that apply to any new drawings that you create.





Note: The default settings are applied whenever you create a ***new*** drawing. Some of the settings can be modified after you have created a drawing, but settings like Drawing Format, Units, and Projection Angle cannot be modified once a drawing is created. If you need to switch these, you will have to create a new drawing.

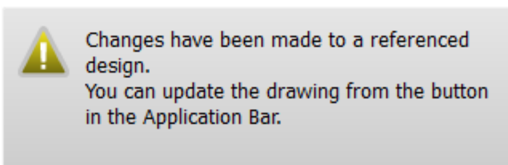
Lesson 7: Associatively Update the Drawing

Learning Objectives

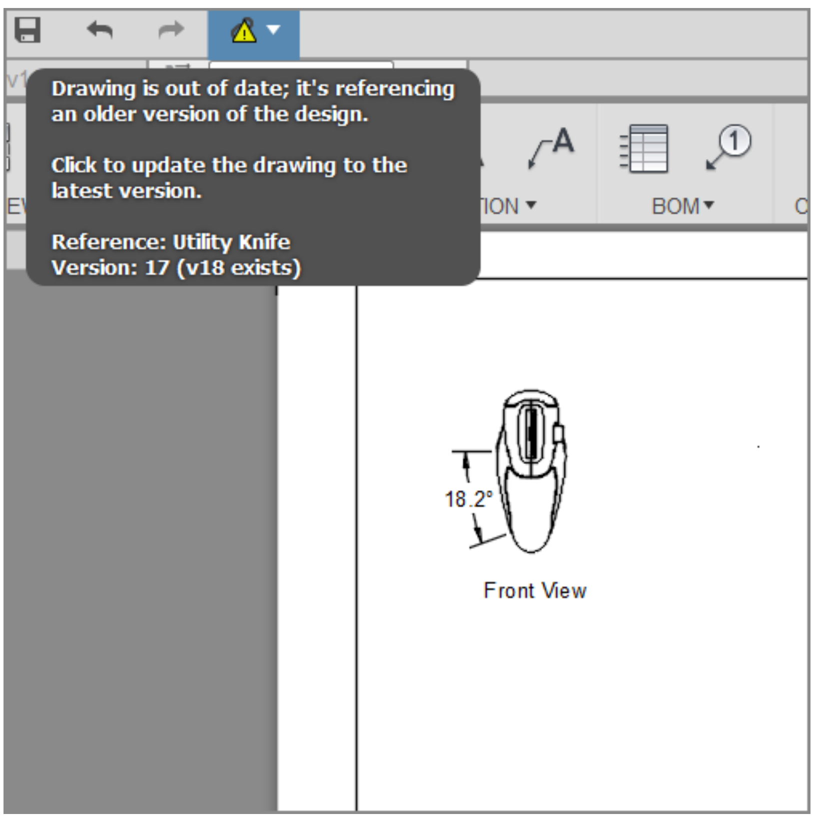
1. How to update a drawing so it reflects any changes made to the model

If you make any changes to the model's geometry and then save the model, you can update your drawing to reflect those changes by clicking "Get Latest" on the toolbar.

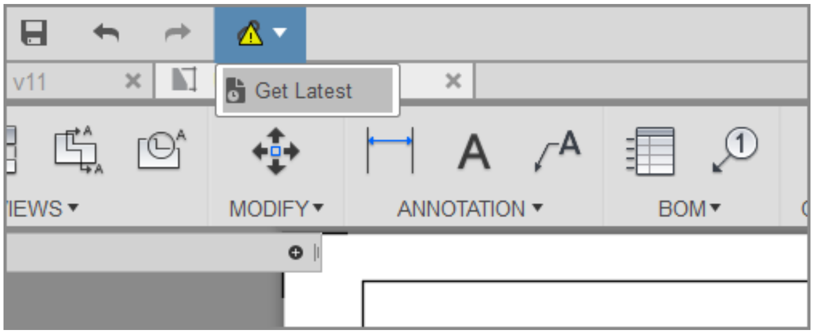
If there are saved changes to a model, you will see this message in the bottom right of your screen:



And you will see this message if you hover over the versions button:



If you click "Get Latest," the drawing will update to reflect the new changes.



If any annotations associated with the drawing view geometry get disassociated because of the model change, badges are displayed on the screen. To delete or manually re-associate these badged annotations to the view geometry, you can snap to specify the points or select the objects you want the dimension to get re-associated.

Lesson 8: Output the Drawing

Learning Objectives

1. Output your Drawing as a PDF or DWG file

When the drawing is complete, you can output the layout as either a PDF or DWG file. Both of these options creates a copy of the drawing and prompts you to save it locally.

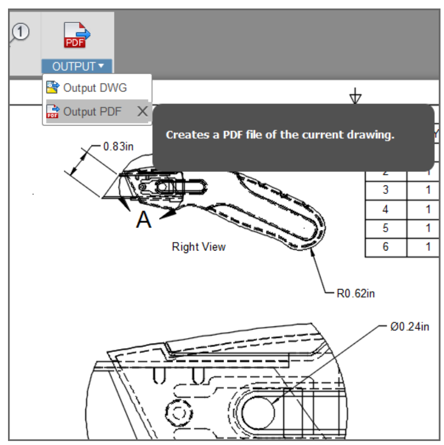
Output the Drawing

Step 1: – Save the drawing

1. Initiate the file menu dropdown and click the Save button.
2. Enter ‘Utility Knife’ in the Name field.
3. Click OK.

Step 2: – Output a PDF

1. Initiate **Output PDF**.
2. Navigate to a desired local location on your computer.
3. Press Save.



Intro to 2D Strategies | CAM for Fusion 360

CNC milling toolpaths are broadly classified as either 2D, 3D, 4-axis, and 5-axis, depending on the number of axes involved and how they move. The term, 2D, is a bit of a misnomer because all modern CNC machines control at least three axis and all three axes move at one time or another for every 2D machining operation. A more accurate term, 2-1/2D, is commonly used in CNC manufacturing. For more information, please refer to the Autodesk CNC Handbook.

2D vs. 3D Defined

2D (Prismatic) Parts

2-1/2D milling toolpaths machine only in the XY plane. The Z- axis is used only to position the tool at depth. The move to the cutting plane is a straight down feed, rapid, ramp or helical feed move.

The term, Prismatic, is a term commonly used in engineering to describe 2-1/2D parts. There are, however, prismatic parts that require 4th or 5-Axis machining, so the term is used in machining only to describe parts where all machined faces lie normal to the machine tool spindle. The XY axes are normal to the machine spindle and Z is used only to position the tool to depth (either in a feed or rapid motion).

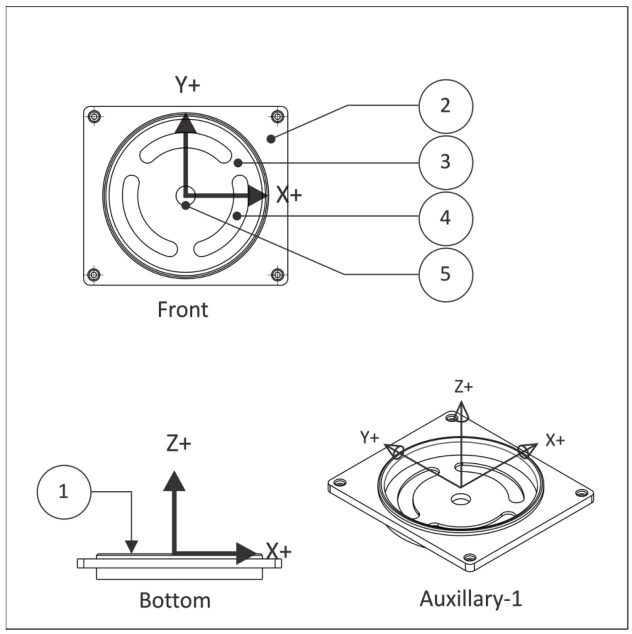


Figure 1: Prismatic Part (Orientation in CAD)

Figure 1 shows a prismatic part. All machined features lie parallel to the XY plane. Each Z-level can be machined by positioning the tool at a fixed Z-level and then moving the XY axes to remove material. Every feature can be reached with the tool approaching either from the Front or Bottom views. There are several cutting planes in this example, including the model top (1), top of the face where the holes start (2), the bottom of the pocket (3) where the slots begin, the bottom of the slots (4), and the bottom of the hole through the center (5).

Learning Objectives

Upon successful completion of this lesson, you will be able to:

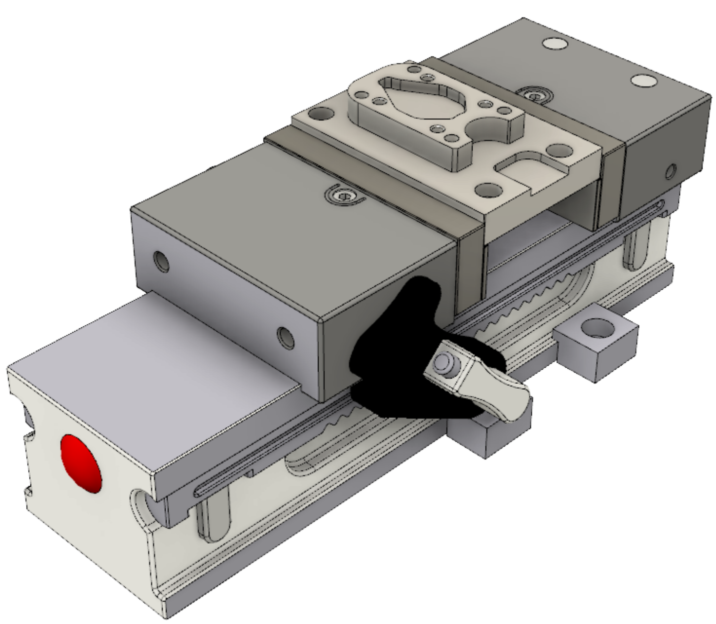
- Explain the difference between 2-1/2D and 3D machined parts.
- Explain the difference between common CAD and CAM graphics views
- Identify 2D machining features based on part geometry and your knowledge of tools and 2D toolpaths.
- Identify commonly used machining parameters for 2D tool path operations.
- Apply a Job Setup to a 2D Milled Part
- Apply a multitude of 2D Operations to a Milled Part
  - Facing Toolpaths
  - 2D Adaptive Toolpaths
  - 2D Contour Toolpaths
  - Chamfer Milling Toolpaths
  - Bore Toolpaths
- Produce Setup Sheets
- Simulate Toolpaths and Stock Material Removal
- Produce NC Code via Post Processing

Datasets Required

In Samples section of your Data Panel, browse to:

Fusion 101 Training > 09 – CAM > **09_2D_Strategies**

Open the design and follow the step-by-step guide below to get started with the lesson.



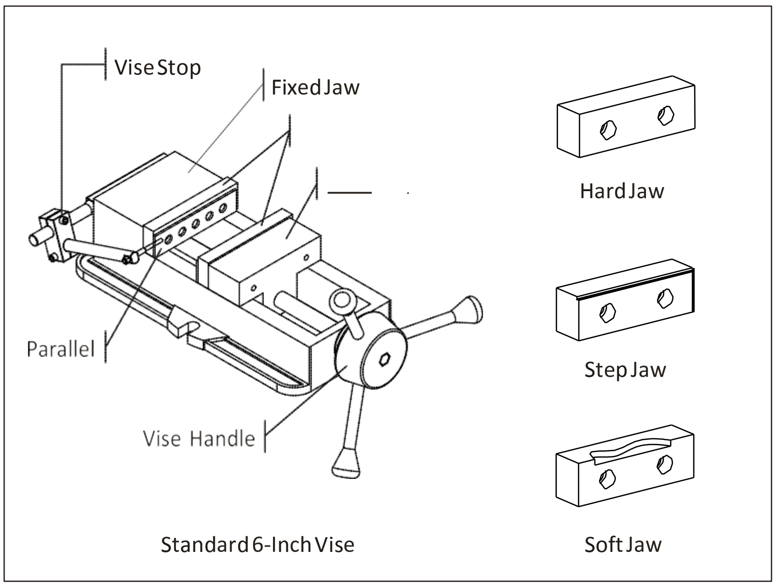
Lesson 1: Workholding & Job Setup

Fixture Component Terminology

Vise and Accessories

The CNC vise is precision engineered and manufactured with components ground flat and perpendicular to within .0002 inches. The most common is referred to as a six-inch (6”) vise, because the width of the jaws is six inches.

Once the vise is bolted to the table and aligned, parts are loaded into the vise and clamped by closing the jaws. The vise can exert tremendous force, so care is taken not to over-tighten the vise and deform fragile parts. Vise pressure must be appropriate to the part being held and expected cutting forces.

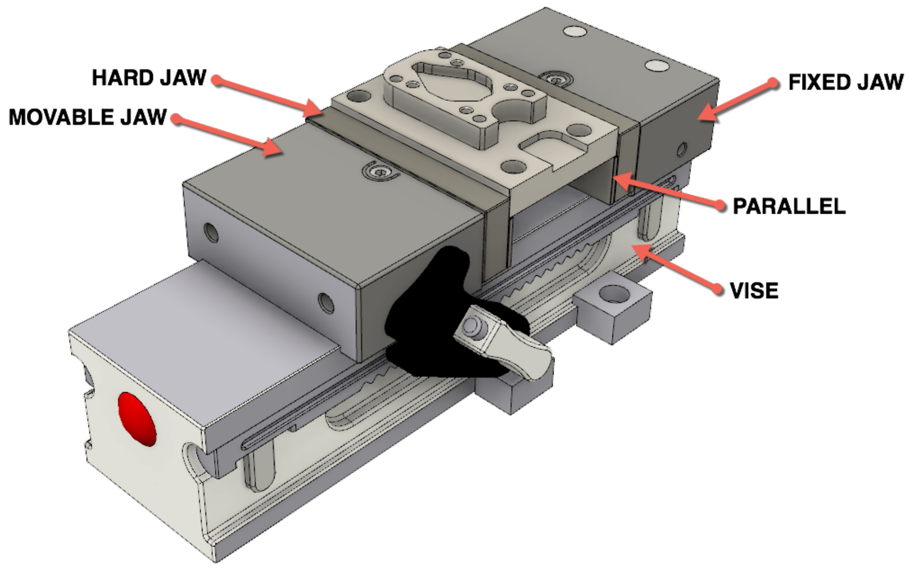


The **Fixed Jaw** remains stationary. The **Moving Jaw** opens when the **Vise Handle** is turned. It is a good practice to remove the vise handle after the jaws are closed and before running the program. This is done by simply sliding the handle off.

A **Vise Stop** is a device that allows the parts to be loaded into the vise precisely. This image shows a style of vise stop that is particularly useful because it is adjustable up-down and left-right.

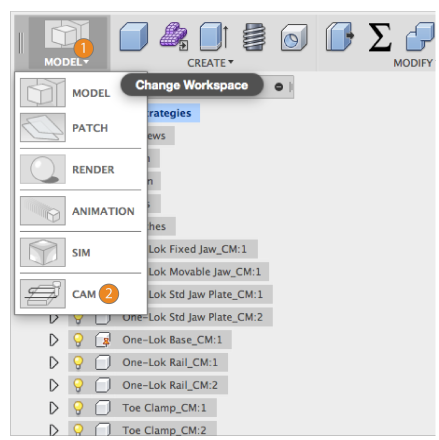
Hard Jaws are made of hardened steel and precision ground on all sides. They are usually used along with parallels.

Parallels are thin steel plates, available in various widths, used to set the grip length of the vise jaws.



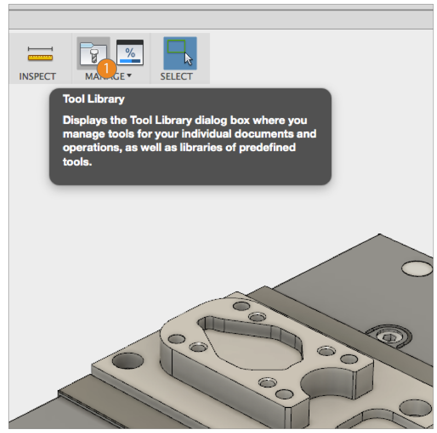
Step-by-step Guides:

Step 1: Activate the CAM Workspace.



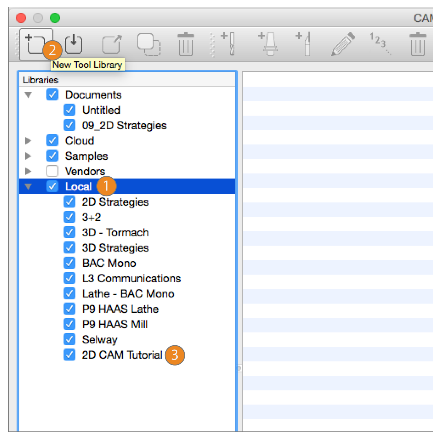
Step 2: – Start the **TOOL LIBRARY** command

1. Click **TOOL LIBRARY**



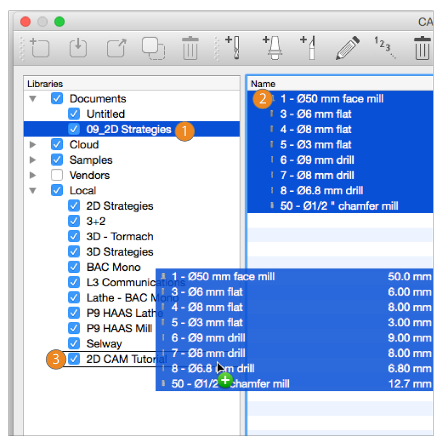
Step 3: – Create a **NEW TOOL LIBRARY**

1. Click on **Local**
2. Select the **NEW TOOL LIBRARY** Icon
3. Double Click on the **NEW TOOL LIBRARY** name and rename to **2D CAM Tutorial**.



Step 4: – Copy and Paste **TOOLS** into **NEW LIBRARY**

1. Click on **Documents** and select the Library **09_2D_Strategies**
2. Select **ALL** tools in the library, and drag and drop into your new library created.

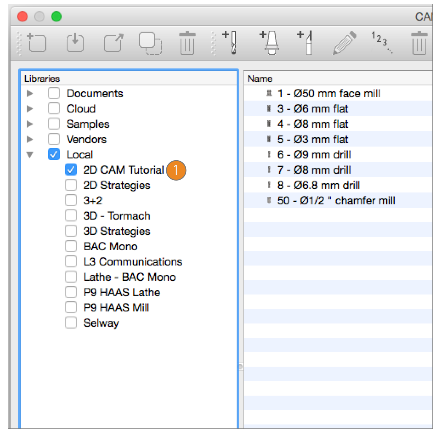


Step 5: – Turn off all other Libraries

1. Click off all other libraries and only show **2D CAM Tutorial.**

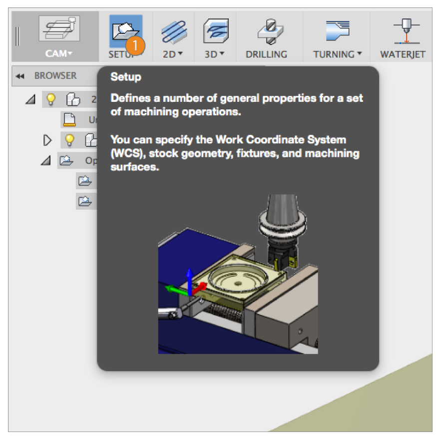
Then…

EXIT OUT OF TOOL LIBRARY



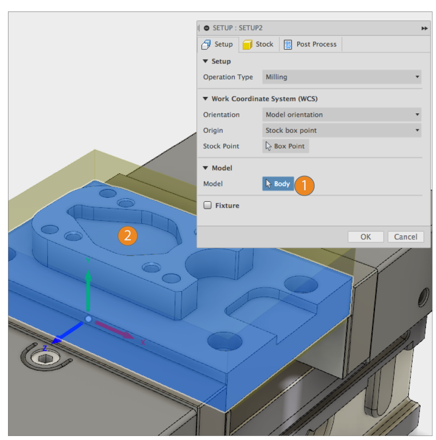
Step 6: – JOB SETUP

1. Click **SETUP**



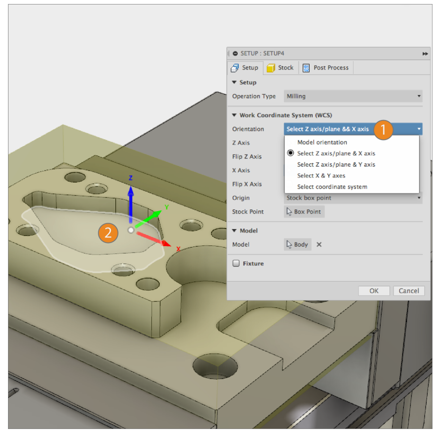
Step 7: – Select the **Part** you want to Machine

1. Under **MODEL**, active the **NOTHING** Icon
2. Select the **2D Strategies Part** in the Screen.



Step 8: – **Orientate** and **Locate** the Work Coordinate System (WCS) correctly.

1. Under Work Coordinate System (WCS), pick the Orientation drop down and select '**SELECT Z axis/plan & X axis**.'
2. Pick the **Highlighted top face**, and the WCS will orientate in the top/center of the part with **'Z' facing north**.

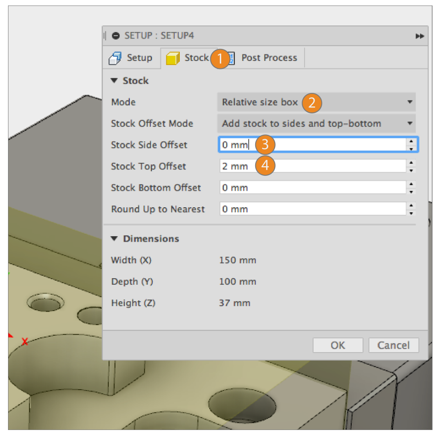


Step 9: – Change **STOCK** Options

1. Click on the **'STOCK'** Tab
2. Under **MODE**, select **Relative Size Box**
3. Under **STOCK SIDE OFFSET**, change to 0 mm
4. Under **TOP SIDE OFFSET**, change to 2 mm

THEN…

CLICK **OK** TO ACCEPT

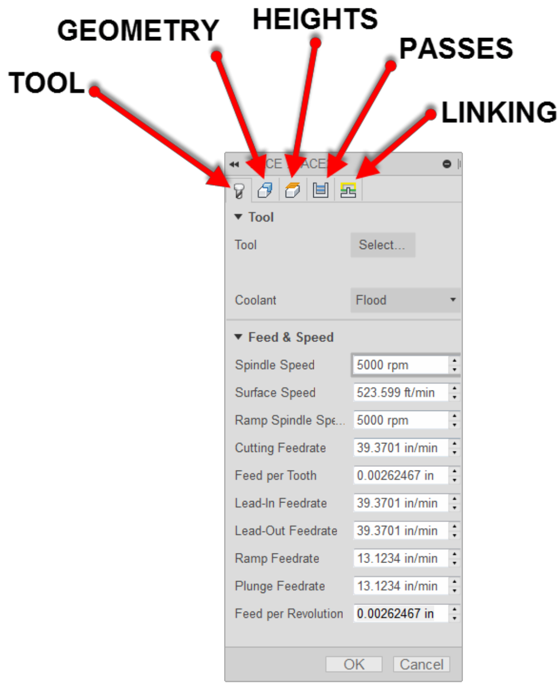


Lesson 2: Toolpath Operations

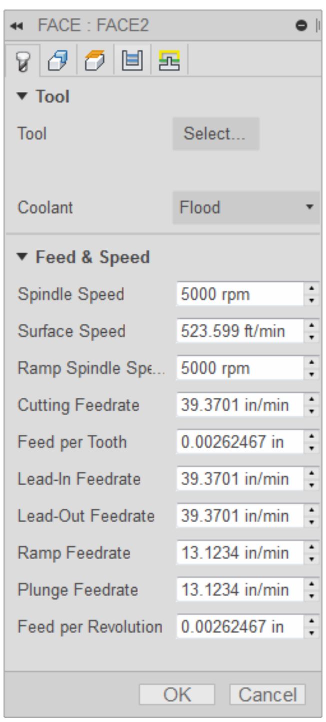
Understanding Toolpaths by Type and Use

Before going further, it is helpful to understand how 2D toolpaths are classified in most CAM software. Please refer to the Autodesk CNC Handbook for more elaborate detail.

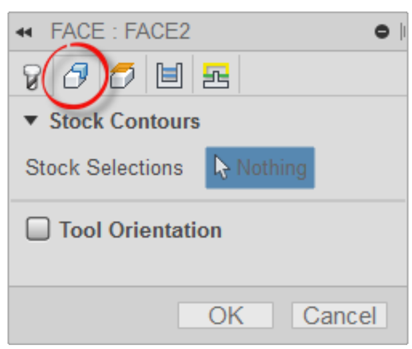
Type	Toolpath	Common Uses
Face	Face	• Finish face of part.
	Island Facing	• Finish face with open sides and bosses.
2D Contour	Contour	• Loops. • Partial loops. • Single edges. • Stick (single point) fonts. • Create dovetail, keyset, or saw cut.
	Chamfer	• Create chamfer using tapered mill or center drill. • De-burring.
	Fillet	• Creating fillet using Corner Round tool.
Pocket	Pocket	• Remove excess material. • Machining TrueType (outlined) fonts and logos.
	Slot Mill	• Straight slot. • Arc slot.
Drill	Drill	• Create spot drill, drill, tap, bore or reamed hole.
	Circular Pocket Milling	• Making holes greater than .75*in* diameter.
	Thread Mill	• Create ID threads over .75*in* diameter. • Create milled OD threads of any size.



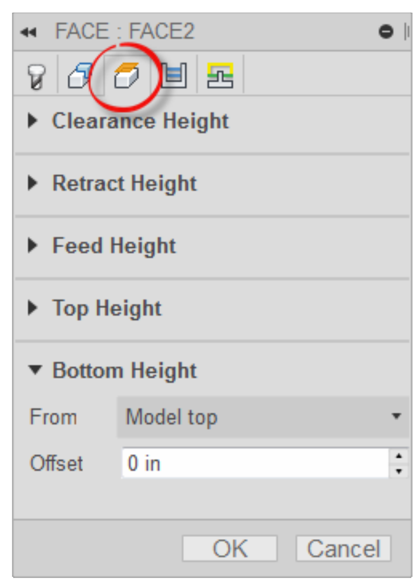
TOOL TAB– Defines the tool being used; as well as the feeds and speeds



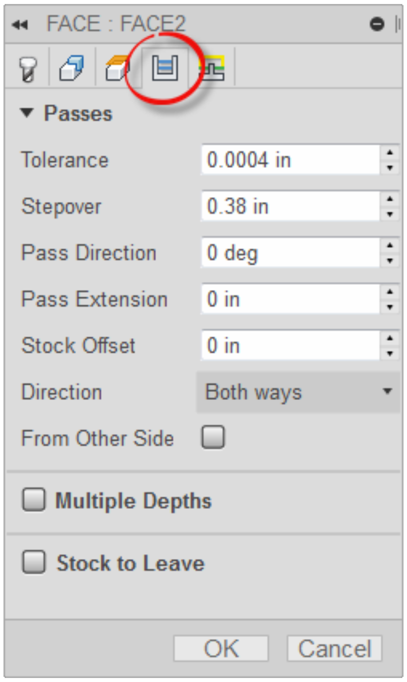
GEOMETRY TAB– Defines geometry being machined.



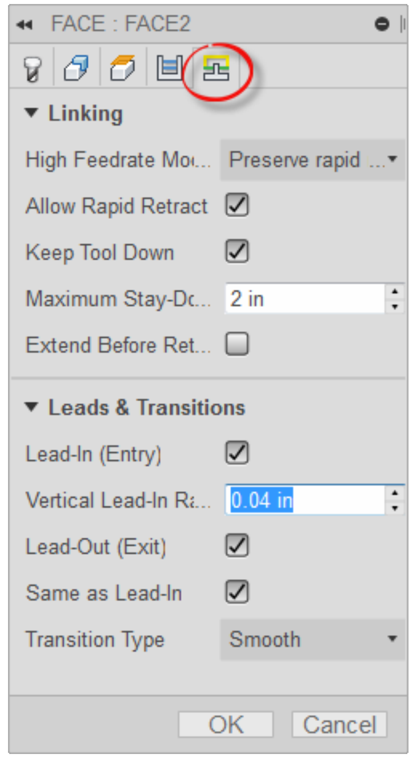
HEIGHTS TAB– Controls heights the toolpath goes to such as cut depth and retract heights



PASSES TAB– Controls how the tool will go about removing material.

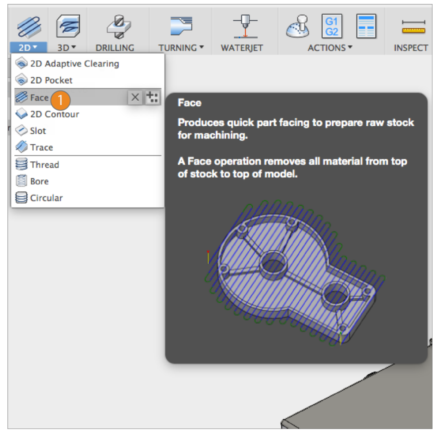


LINKING TAB– Controls how the tool enters/exits and transitions between cutting movements



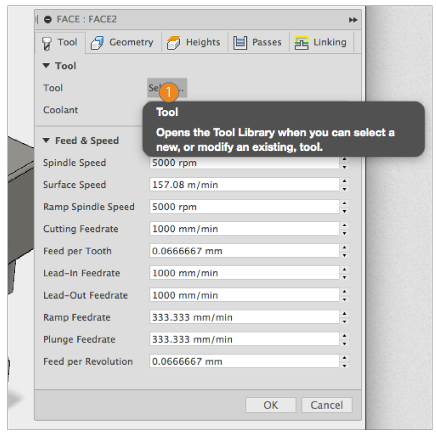
Step 10: – The **FACE** Operation

1. Under **2D Operation**, click on the **FACE Operation**



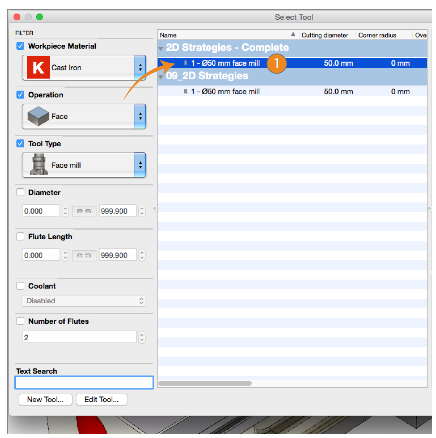
Step 11: – Access **TOOL LIBRARY**

1. Click on **SELECT** under **TOOL**



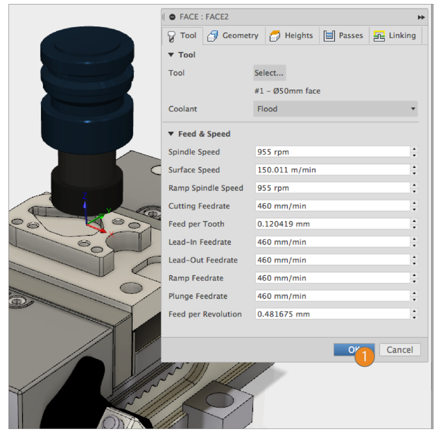
Step 12: – Select a Face Mill

1. Select the **#1 50 mm Face Mill**
2. **Click OK**

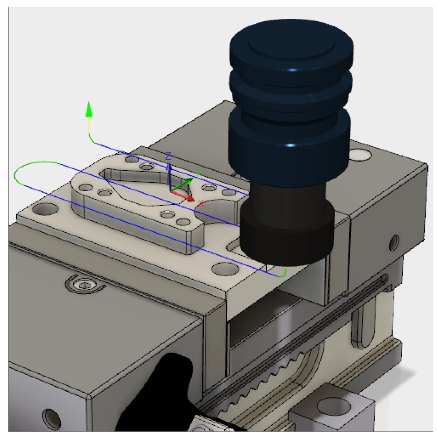


Step 13: – The **FACE** Operation

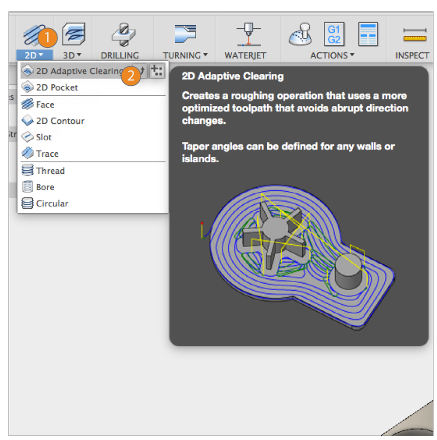
1. Click **OK**



Step 14: – **FACE Operation COMPLETE**

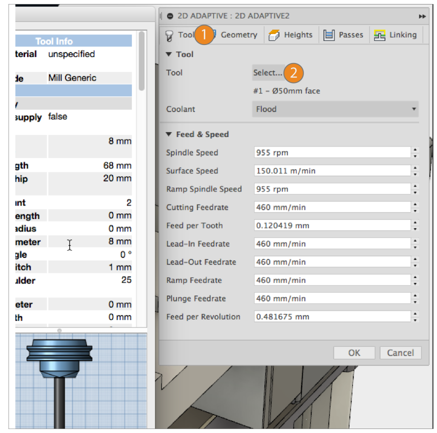


Step 15: – Apply a **2D Adaptive Clearing** Operation



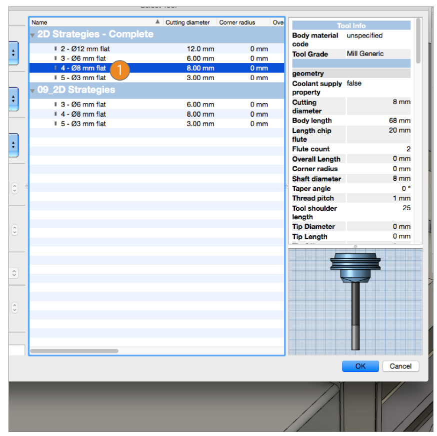
Step 16: – Select a NEW TOOL

1. Click **Tool**



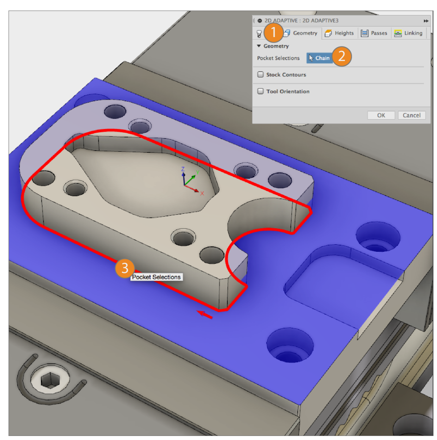
Step 17: – **Select a #4 8 mm Flat End Mill**

THEN CLICK **OK**

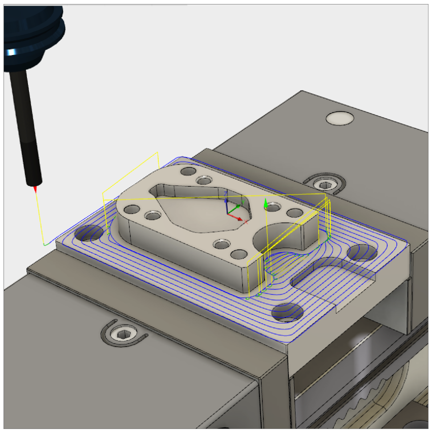


Step 18: – Select Geometry

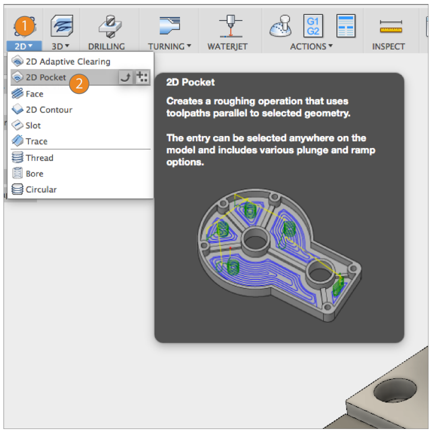
1. Click the **Geometry** Tab
2. Activate **Pocket Selection**
3. Click the outside of the Boss (IN RED)
4. CLICK **OK**



Step 19: – Toolpath Generated

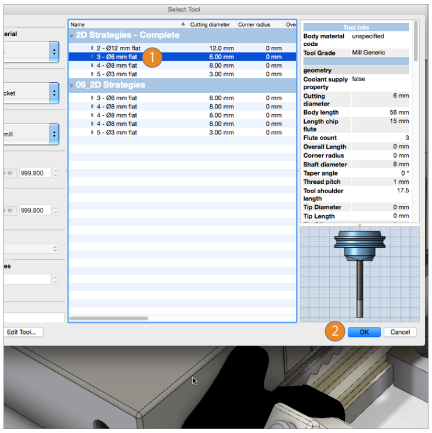


Step 20: – Apply a **2D Pocket** Operation



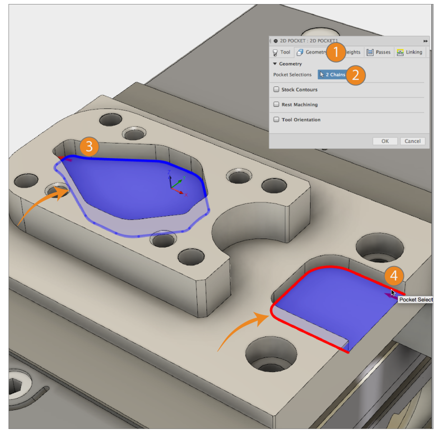
Step 21: – Pick the correct tool

1. #3 6mm Flat End Mill

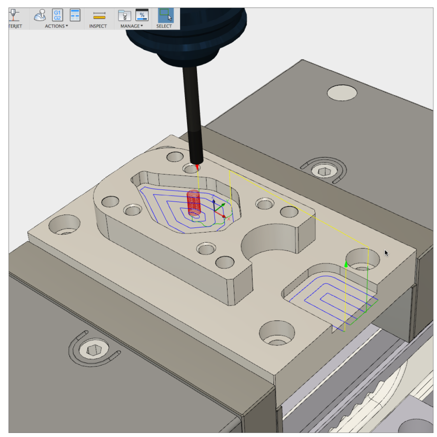


Step 22: – Select Geometry

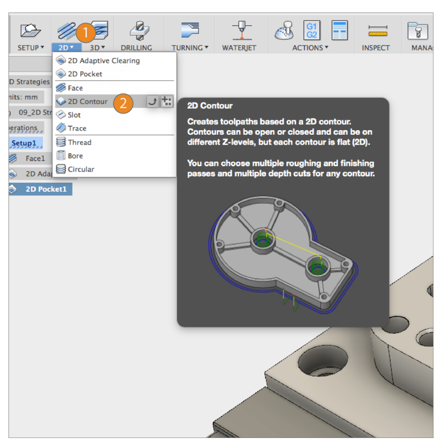
1. Click the Geometry Tab
2. Activate **Pocket Selection**
3. Click the inside embossed bottom **EDGE**
4. Click the bottom **EDGE** of the embossed open pocket (IN RED)
5. CLICK **OK**

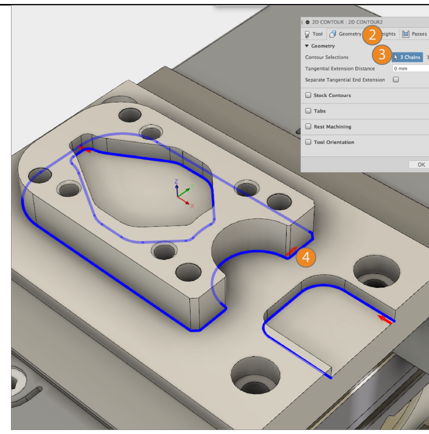
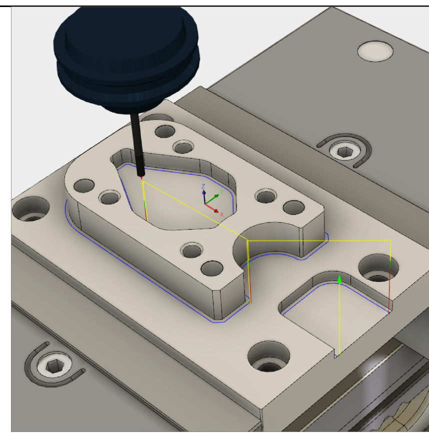
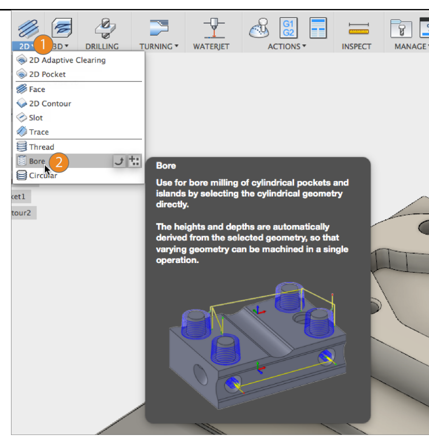


Step 23: – Toolpath Generated



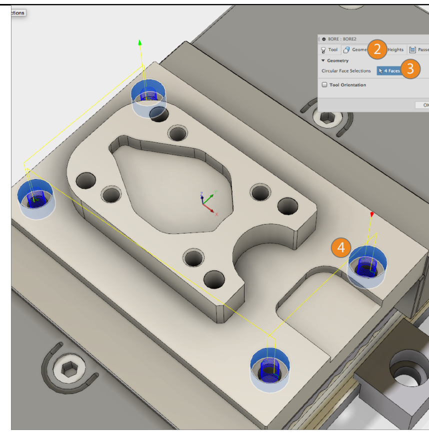
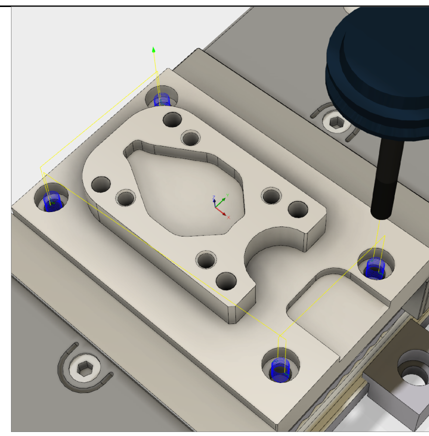
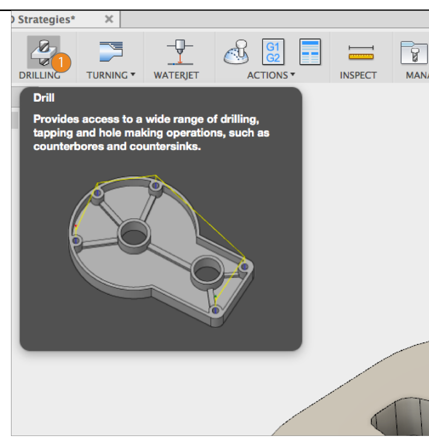
Step 24: – Apply a **2D Contour** Operation for a finishing pass.

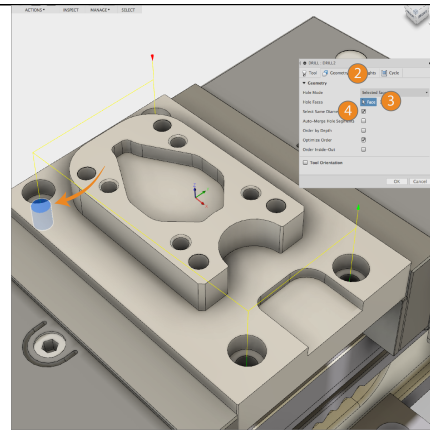
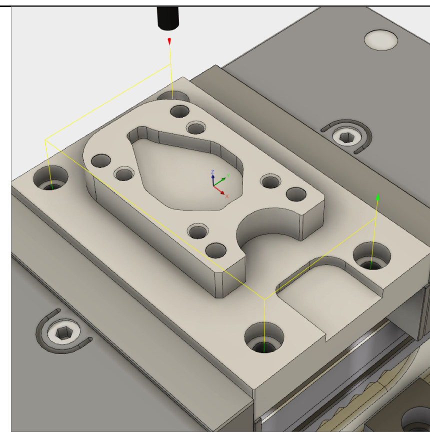


Step 25: – Select correct **TOOL** and **GEOMETRY**. 1. Under the Tool Tab, select a **#5 3mm Flat End Mill.** 2. Click the **Geometry** Tab. 3. Activate **Pocket Selection** 4. Click the **3 EDGES** shown. CLICK **OK** **TIP: CLICK ON THE RED ARROW TO HAVE THE TOOLPATH FOLLOW ON THE OUTSIDE/INSIDE OF THE BLUE LINE.**	
Step 26: – Toolpath Generated	
Step 27: – Create a **BORE** Operation	

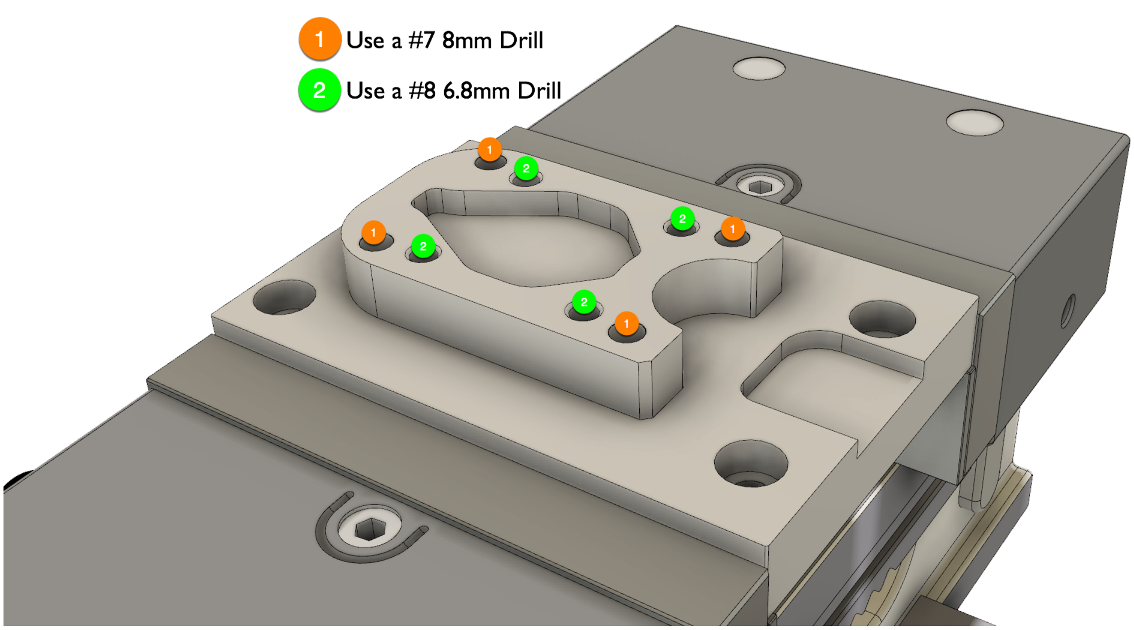
<table>
<tr>
<td>Step 28: – Select correct TOOL and GEOMETRY.

1. Under the Tool Tab, select a #4 8mm Flat End Mill.
2. Click the Geometry Tab.
3. Activate Circular Face Selections
4. Click the 4 Internal Faces of the Holes shown.

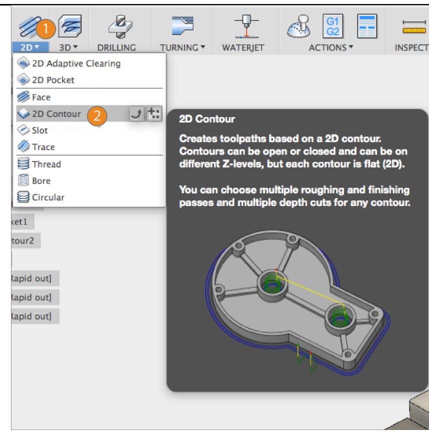
CLICK OK</td>
<td></td>
</tr>
<tr>
<td>Step 29: – Toolpath Generated</td>
<td></td>
</tr>
<tr>
<td>Step 30: – Create a DRILL Operation</td>
<td>
</td>
</tr>
</table>

Step 31: – Select correct **TOOL** and **GEOMETRY**. 1. Under the Tool Tab, select a **#6 9 mm Drill.** 2. Click the **Geometry** Tab. 3. Activate **Hole Faces** 4. Click on **'Select Same Diameter'** 5. Select hole as shown CLICK **OK**	
Step 32: – Toolpath Generated	

Step 33: – Repeat **Step 31**, and use the designed Drill bits below for the holes designated.

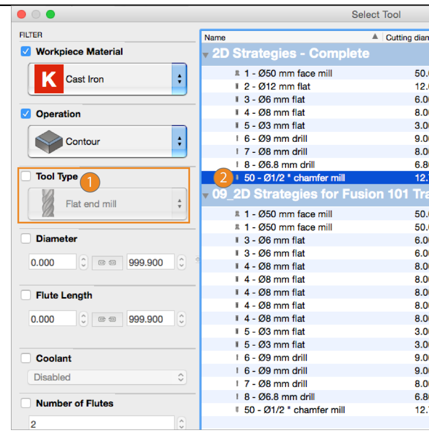


Step 34: – Create a **2D Contour** Operation



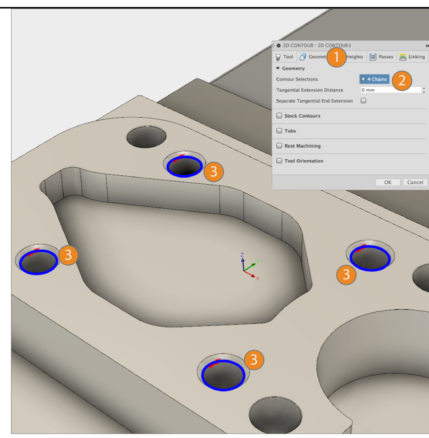
Step 35: – Pick the correct tool

1. Check **OFF** Tool Type
2. Select **#50 1/2" Chamfer Mill**



Step 36: – Select correct **GEOMETRY**.

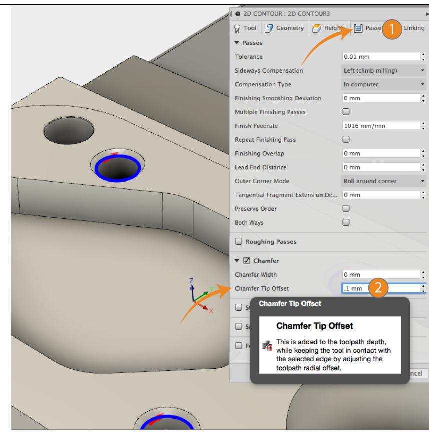
1. Click the **Geometry** Tab.
2. Activate **Contour Selections.**
3. Select edges as shown



Step 37: – Edit the **PASSES** Tab

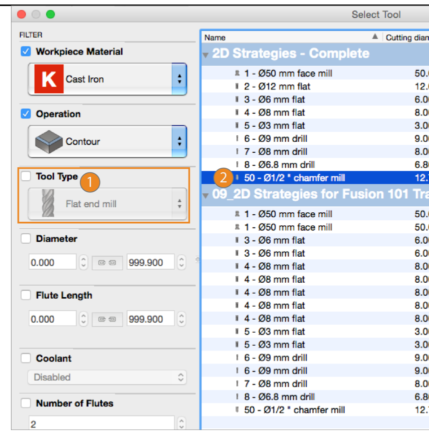
1. Click the **PASSES** Tab.
2. Under **Chamfer**, select **Chamfer Tip Offset**, and type in **0.1 mm**

CLICK OK



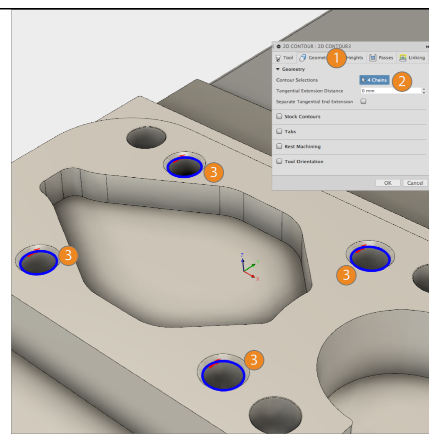
Step 38: – Pick the correct tool

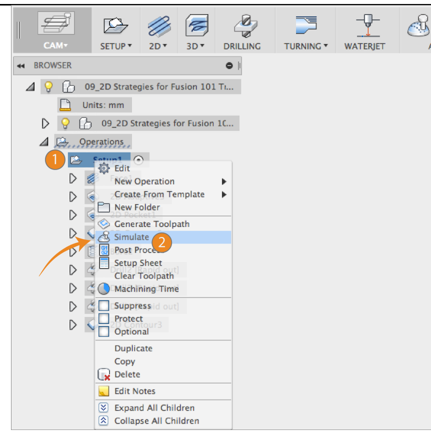
3. Check **OFF** Tool Type
4. Select **#50 1/2" Chamfer Mill**



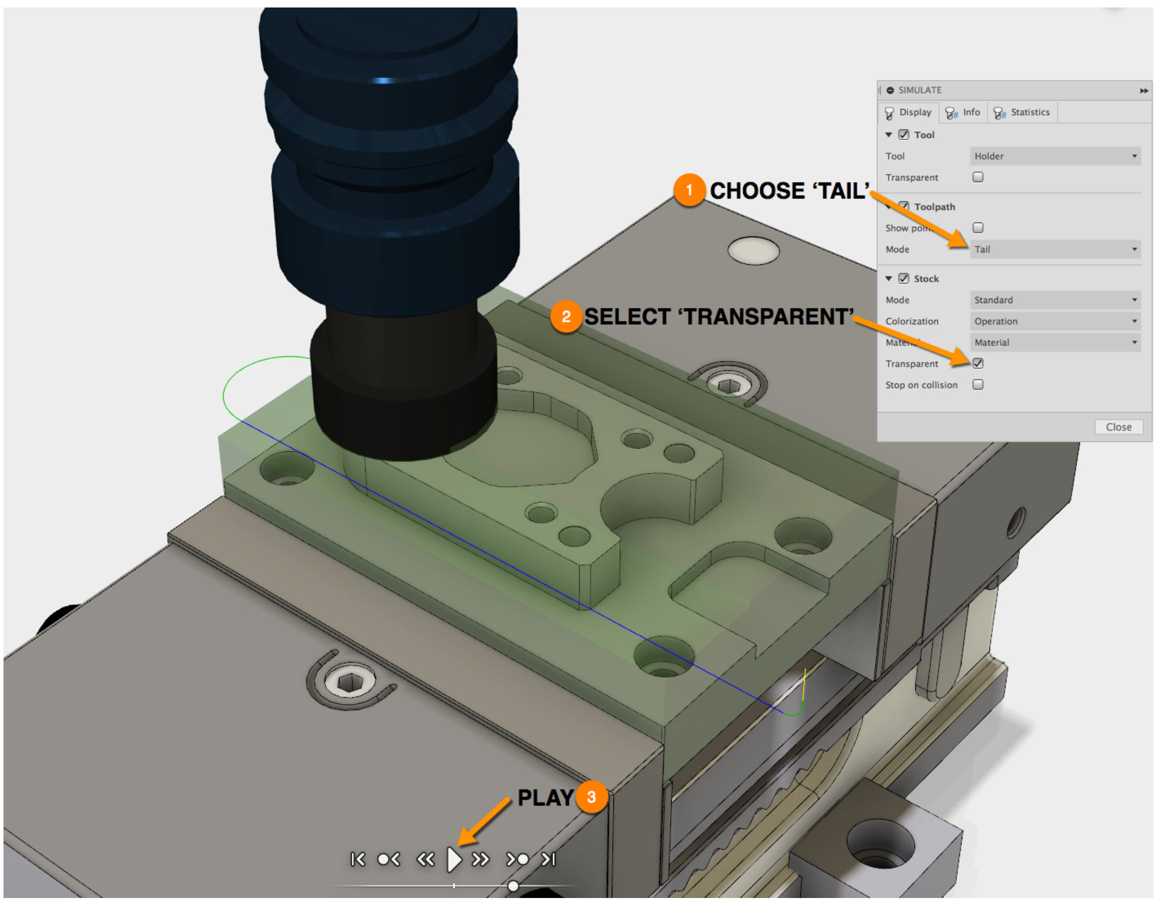
Step 39: – Select correct **GEOMETRY**.

3. Click the **Geometry** Tab.
4. Activate **Contour Selections.**
5. Select edges as shown



Step 40: – Simulate the Job 1. **RIGHT CLICK** on **SETUP** in the browser. 2. Select **SIMULATE**.	

Step 41: – Choose the following **SIMULATE** presets below:

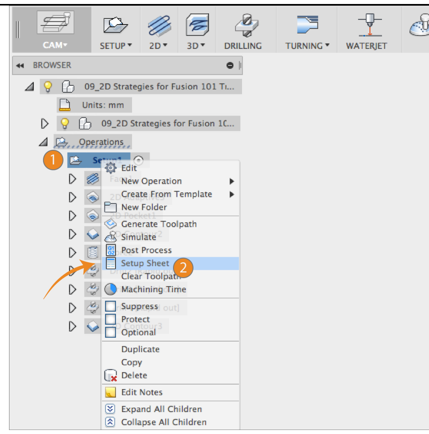


Setup Sheet

The Setup Sheet feature allows you to generate an overview of the NC program for the CNC operator. It provides tool data, stock and work piece positioning; as well as machining statistics.

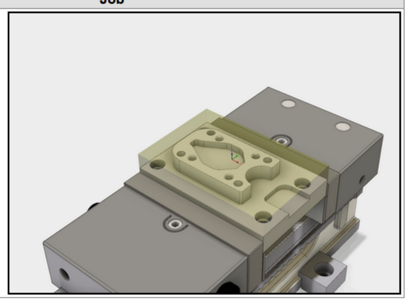
Step 42: – Create a Setup Sheet

1. **RIGHT CLICK** on **SETUP** in the browser.
2. Select **Setup Sheet**
3. Choose a location where to save it.
4. HTML will be generated of a Setup Sheet (see below).



Setup Sheet for Program 1001

Job Description: Setup1
Document Path: 09_2D Strategies for Fusion 101 Training v4

Job	
WCS: #0 Stock: DX: 150mm DY: 100mm DZ: 37mm Part: DX: 150mm DY: 100mm DZ: 35mm Stock Lower in WCS #0: X: -75mm Y: -50mm Z: -37mm Stock Upper in WCS #0: X: 75mm Y: 50mm Z: 0mm	

Total
Number Of Operations: 9 Number Of Tools: 8 Tools: T1 T3 T4 T5 T6 T7 T8 T50 Maximum Z: 15mm Minimum Z: -36mm Maximum Feedrate: 1256mm/min Maximum Spindle Speed: 10500rpm Cutting Distance: 6954.28mm Rapid Distance: 2936mm Estimated Cycle Time: 12m:25s

Operation 1/9 Description: Face1 Strategy: Facing WCS: #0 Tolerance: 0.01mm Maximum stepover: 47.5mm	Maximum Z: 15mm Minimum Z: -2mm Maximum Spindle Speed: 955rpm Maximum Feedrate: 460mm/min Cutting Distance: 600.24mm Rapid Distance: 22mm Estimated Cycle Time: 1m:19s (10.6%) Coolant: Flood	**T1** D1 L1 Type: face mill Diameter: 50mm Length: 50mm Flutes: 4 Holder: BT40 - B4C3-0040
Operation 2/9 Description: 2D Adaptive3 Strategy: Adaptive 2D WCS: #0 Tolerance: 0.1mm Stock to Leave: 0.5mm Optimal load: 3.2mm Load deviation: 0.32mm	Maximum Z: 15mm Minimum Z: -16.5mm Maximum Spindle Speed: 3900rpm Maximum Feedrate: 1256mm/min Cutting Distance: 3514.11mm Rapid Distance: 775.83mm Estimated Cycle Time: 3m:57s (31.8%) Coolant: Flood	**T4** D4 L4 Type: flat end mill Diameter: 8mm Length: 68mm Flutes: 2 Holder: Default Holder
Operation 3/9 Description: 2D Pocket1 Strategy: Pocket 2D WCS: #0 Tolerance: 0.1mm Stock to Leave: 0.5mm Maximum stepover: 5.7mm	Maximum Z: 15mm Minimum Z: -21.5mm Maximum Spindle Speed: 5233rpm Maximum Feedrate: 1256mm/min Cutting Distance: 960.78mm Rapid Distance: 159.27mm Estimated Cycle Time: 1m:47s (14.3%)	**T3** D3 L3 Type: flat end mill Diameter: 6mm Length: 56mm Flutes: 3 Holder: Default Holder

Post Processor

A post processor is essentially a printer driver for CNC machines; a unique configuration file that allows our Post Processor System to turn your programmed toolpaths into CNC programs (G-Code) that your machine control executes to cut parts.

Fusion 360 comes with a standard library of "Posts". These library posts are included because they have been proven to make good parts using standard machine defaults. As the complexity of your setups increases, and you learn more about your CNC, you will probably want modifications made to one of these library posts that produce code in a particular way or with particular options enabled. This requires a post edit. Autodesk has a dedicated Post Development Team that while not working with machine tool vendors to produce more standard library posts, helps our Autodesk CAM Resellers and end-users with post requests.

For more information on Post Processors, please review the [Autodesk Post Processor Manual.]

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Step 43: – Outputting to a Post Processor

1. **RIGHT CLICK** on **SETUP** in the browser.
2. Select **Post Process**

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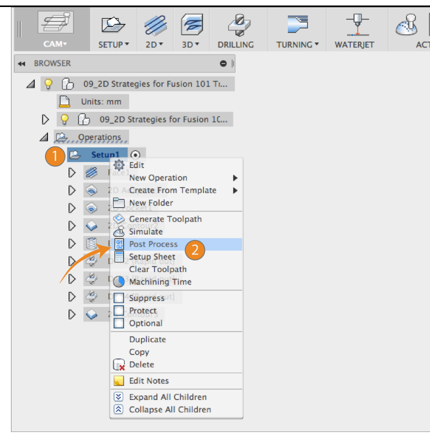

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Step 44: – Outputting to a Post Processor

1. From Post Processor, select your Machine

For Inquiries on specific posts, please email **cam.posts@autodesk.com**

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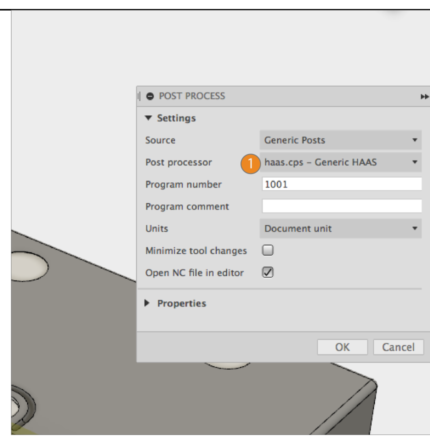

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