Autodesk Fusion 360

Inhoud

Introduction

Overview

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.

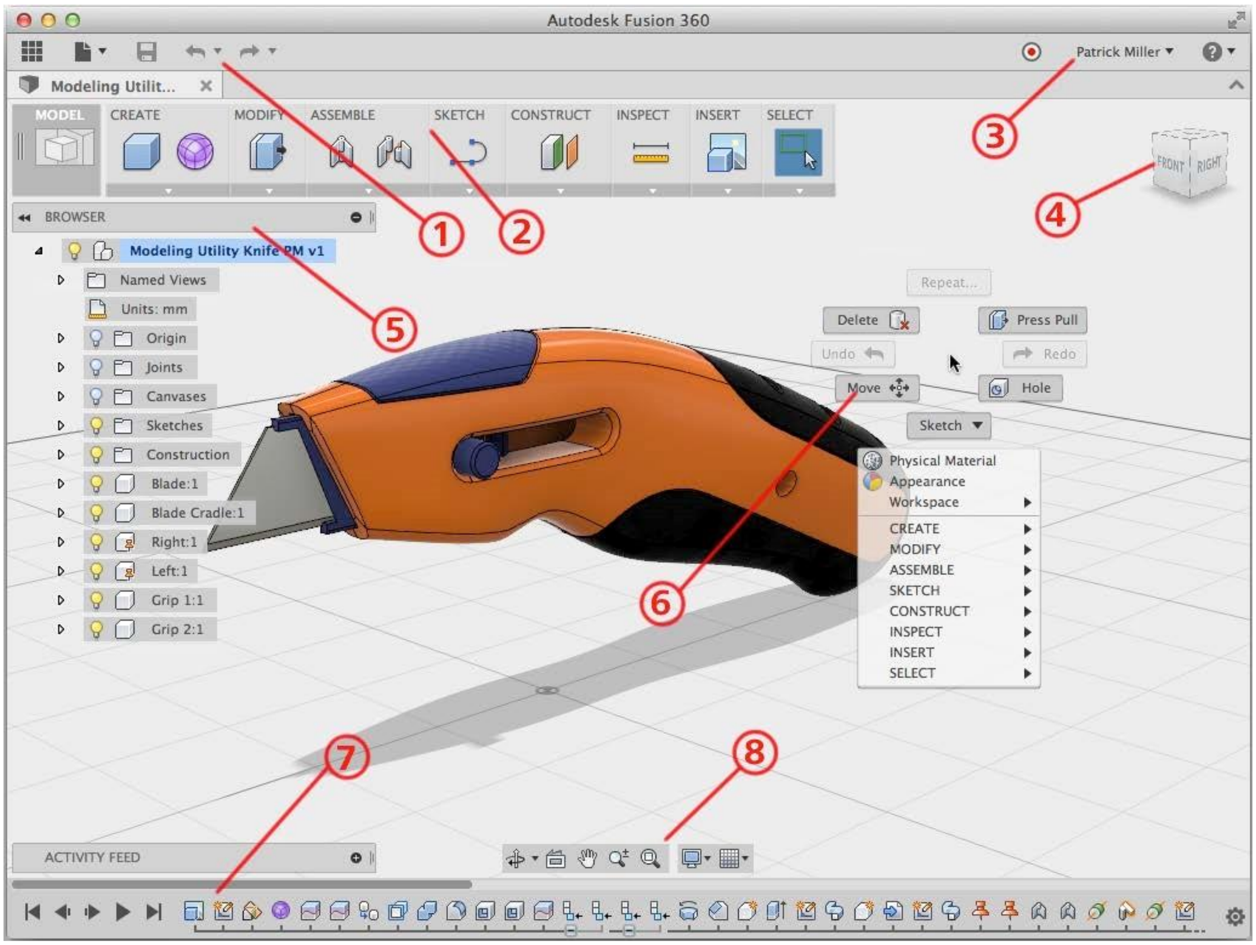
Fusion 360 enables fast and easy exploration of design ideas with an integrated concept to production toolset. Fusion lets you focus on the form, function, and fabrication of your products. Use the sculpting tools to explore form and modeling tools to create finishing features. These tools let you quickly 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 toolpaths 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 and cloud resources. For example, your design data is store 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 local 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 will get you started designing with Fusion and help you understand how it can improve your design processes.

1. Main User Interface



1. Application bar:

- Data Panel: Display or hide the data panel on the left of the interface.

- File: Access file operations such as New Design, Save, Export, and 3D Print.

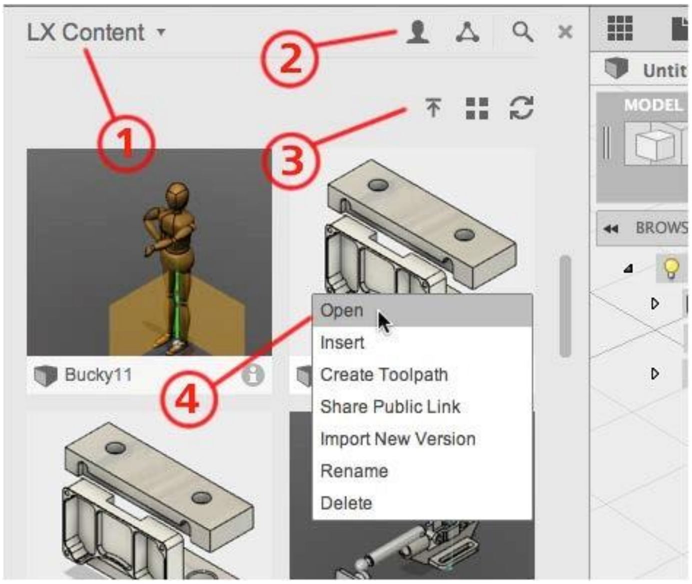
- Save: Save an untitled design or save the changes to a design as a new version.

- Undo/redo: Undo or redo operations.

2. Toolbar: Access commands in the toolbar.

3. Profile and help:
   - Profile name: Access preferences and your Autodesk profile.
   -  Help: Access help, forums, and tutorials, what’s new, and feedback.
4. ViewCube: Orbit the view and access orthographic and isometric views.
5. Browser: Lists objects in the design.
6. Marking menu: Another method to access commands. Right-click to display the marking menu.
7. Timeline: List the operations performed on a design if parametric modeling is active.
8. Navigation bar and display settings: The navigation bar contains commands to navigate the view. The display settings control the display of the design in the canvas.

2. Data Panel Interface



1. Project switcher: Select the active project.
2. Project tools

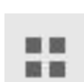
• Project Members: Invite members to the active project.

• 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 tools

•  Upload: Upload files to Autodesk A360. Many CAD data types are supported as well as standard files such as documents, spreadsheets, and presentations.

•  Data view: Select how data is displayed in the data panel.

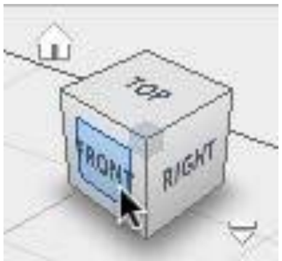
•  Refresh: Refreshes data from Autodesk A360.

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

3. View Navigation

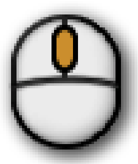
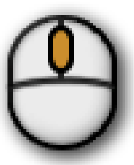
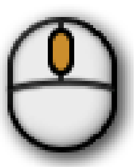
Commands

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



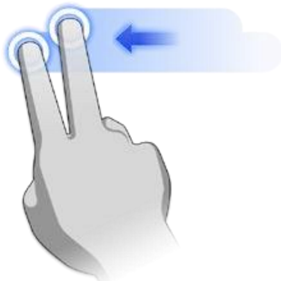
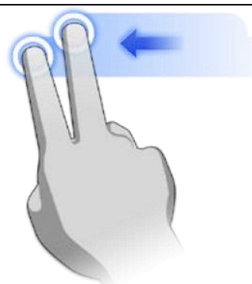
Use the commands in the Navigation bar to pan, zoom, and orbit the canvas. The menus on the right end control Display Settings and Layout Grid options.

Mouse

SCROLL		**Scroll** the middle mouse wheel to zoom in or zoom out.
HOLD		**Click and hold** the middle mouse button to pan the view.
SHIFT KEY +		Hold the **SHIFT** key and **click and hold** middle mouse button to orbit the view.

Mac Trackpad

	2 finger pinch to zoom out.

	2 finger spread to zoom in
	2 finger swipe to pan the view.
SHIFT + 	Hold SHIFT and 2 finger swipe to orbit the view.

4. Workspaces

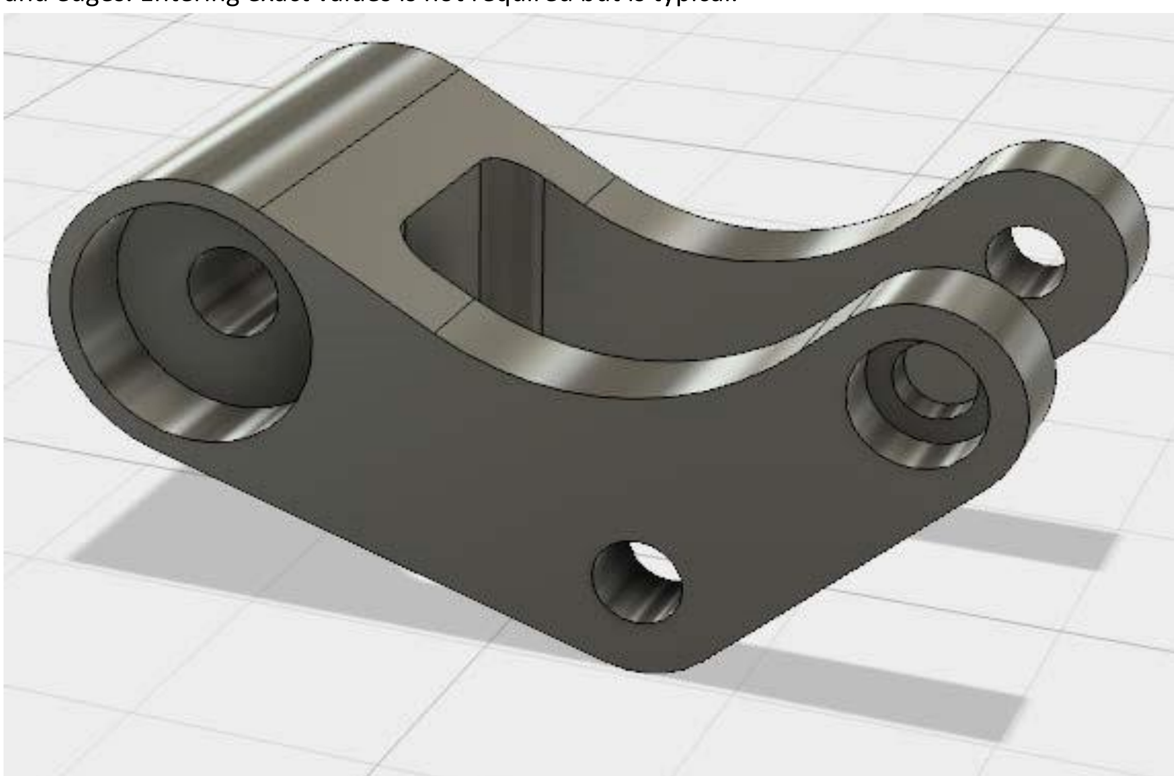
Fusion 360 uses workspaces control the commands that are available and the type of data that is created. There are multiple workspaces available depending on the work you plan to perform.

- Sculpt: create organic shapes by manipulating faces, edges, and vertices.
- Model: create solids with hard edges and flat faces.
- Patch: create open surfaces to stitch into solid bodies.
- Render: set up the environment and create photo-realistic renderings.
- CAM: create and simulate tool-paths then generate g code for subtractive manufacturing.
- Drawing: generate 2D manufacturing drawings.

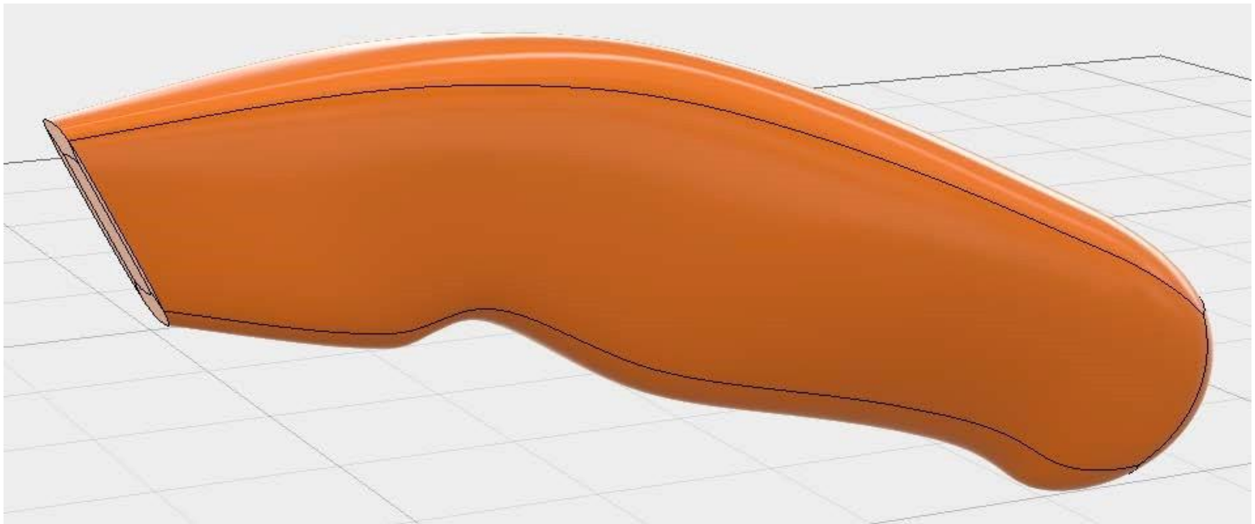
You also have a drawing workspace for documentation, render workspace for creating photo-realistic renders, CAM workspace for creating toolpaths.

It’s obvious when to use some workspaces. If you need a 2D manufacturing drawing, you use the drawing workspace. What about model and sculpt? They are both used to create 3D designs so how do you choose to use one over the other?

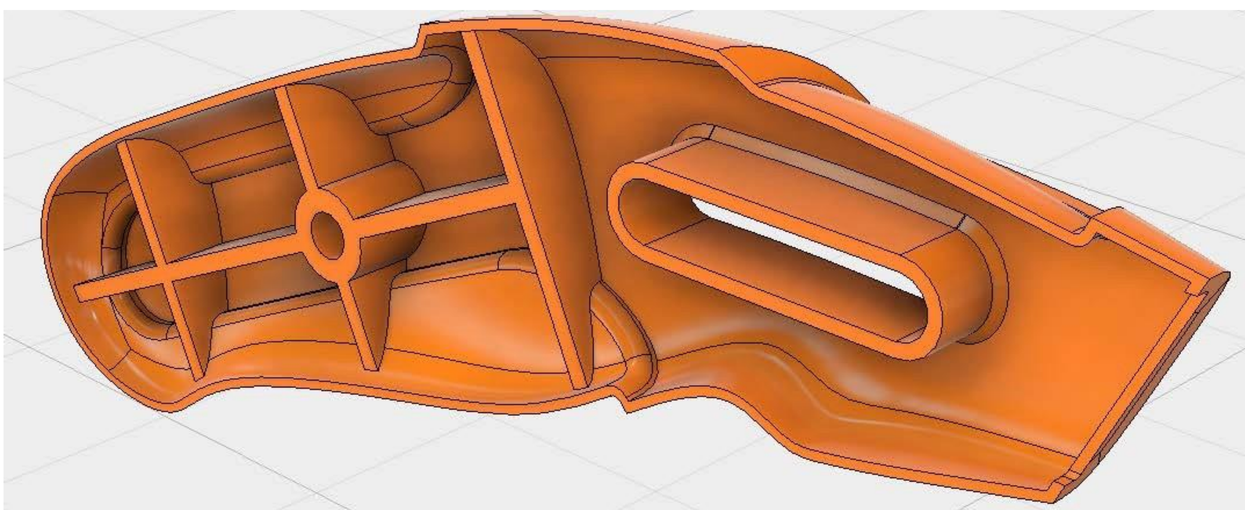
Use model to create designs with hard edges and flat faces. Model creates bodies requiring exact sizes and edges. Entering exact values is not required but is typical.



Use the sculpt workspace to create bodies with organic shapes. Sculpt bodies are highly curved and the shape is more critical than exact size.



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. You can work entirely in sculpt, entirely in model, or you can combine the two. You can combine sculpt and model to create the shape required as well as precise manufacturing features. Create the organic shape in sculpt then use model for manufacturing features afterwards.



5. Design History

Fusion 360 can work with or without recording design history. Design history refers to the operations you perform on the design to create and modify geometry. Operations are recoded in the timeline at the bottom of the interface.

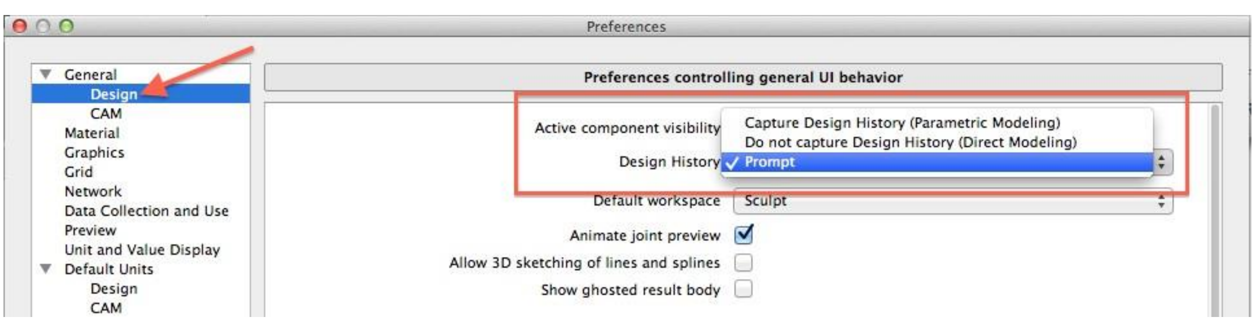
When using parametric modeling, the design history is captured in the timeline at the bottom of the interface. Operations are captured in the order they are performed in. History is captured for commands in the Model and Patch workspaces. You edit the operations in the timeline to make changes to your design.

When using direct modeling, design history is not captured. The same commands are used from the toolbar but there is no timeline. You use commands like Press Pull or Move to move faces and change your design.

So, why use one over the other? Using history allows you to make precise predictable edits to one or many components and allow the model to rebuild reliably. History is also useful if you plan to go switch between the model and sculpt workspaces. This allows you to create your outer shape, then create model operations (shell, split, hole, etc.) then go back and change your shape. If history is enabled, the model operations will recalculate to fit the new shape.

With direct modeling (history is off), you change geometry by moving faces. There are no operations to edit and therefore, no relationships between features in the design. Direct modeling works well for quick concept design or when working the imported data.

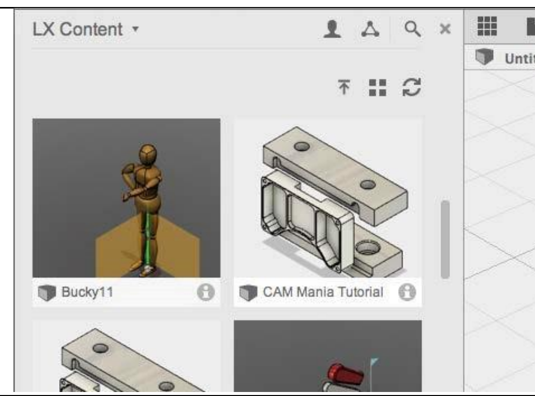
You can control the default behavior for new designs using preferences or you can turn design history on/off in the browser after a model is created.



6. Autodesk A360

Fusion 360 uses Autodesk A360 to manage data and collaborate with teams. When you access your designs in Fusion, you are actually using A360 under the hood. You can also access your data in a web browser using A360. A360 provides access and management of the versions of your designs. You can also upload other supporting design documentation. A360 lets you manage who can access your design data. All these tools within A360 make it easy to collaborate with team members.

Get Started: To get started with Fusion 360, we will log in, create a project, and save a design to the project.

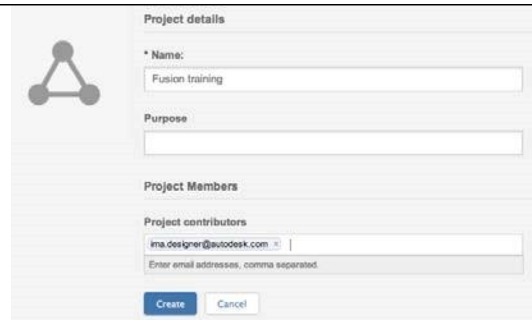
Step 2 – Log in to Fusion 1. Launch Fusion 360. 2. Use your Autodesk ID to log in to Fusion. If you do not have an Autodesk ID, you can create an account for free.	
Step 3 – Open the Data Panel 1. Click  to display the Data Panel. 2. The data panel slides open on the left.	

Step 4 – Create a new project

1. Click  to go to Autodesk A360 in your Internet browser.
2. Move the mouse over 

   then click 

   to begin creating a new project.
3. Enter **Fusion Training** in the Name field.
4. Enter email address in the Project Contributors field to invite people to this project.



7. Hot Keys

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 doubleclick a third face	Select two faces then doubleclick 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

Sketching

Overview

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.

Learning Objectives

In this section you will learn how to:

- Create a 2D sketch
- Create geometry in a sketch
- Use constraints to position geometry
- Use dimensions to set the size of geometry

A Fusion design can contain multiples sketches. The sketch is an object that contains the geometry to define profiles. Your sketches are listed in the browser. Sketches are also listed in the timeline in parametric designs.

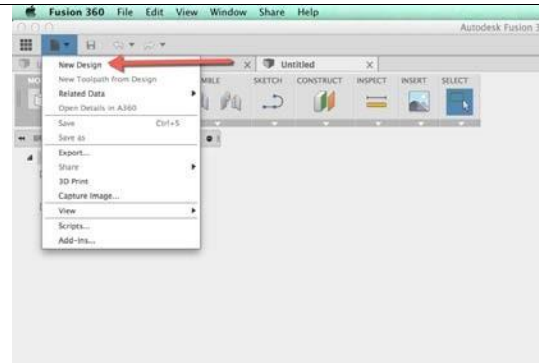
Sketches must be created on a plane. You can use the origin planes, construction planes, or a flat model face to define a sketch.

8. Create Sketch

In this exercise, you create a new Fusion design then create a new sketch in the design.

<table>
<tr>
<td>Step 1 – Create a new design

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

This creates a new design where you will create geometry.</td>
<td>
</td>
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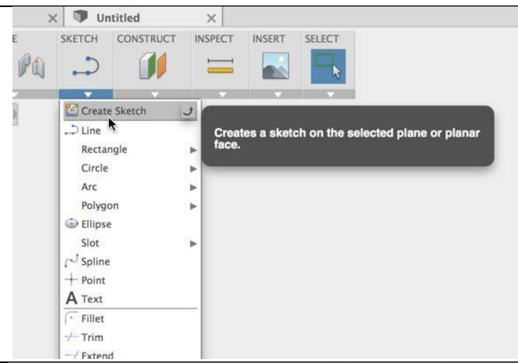
<tr>
<td>Step 2 – Create a new sketch

1. Select Sketch > Create Sketch.</td>
<td>
</td>
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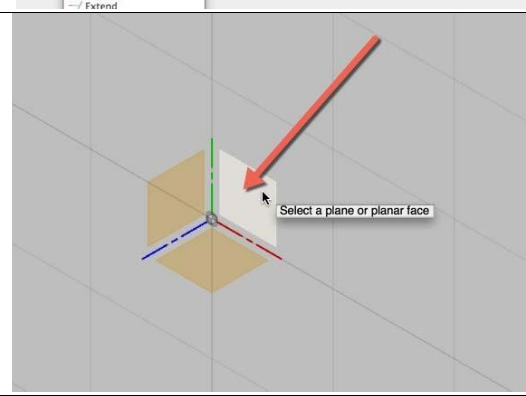
<tr>
<td>Step 3 – Select the sketch plane

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

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.</td>
<td>
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Sketches contain vector data. They are made of lines, circles, arcs and other curves. They are not artistic sketches or renderings.

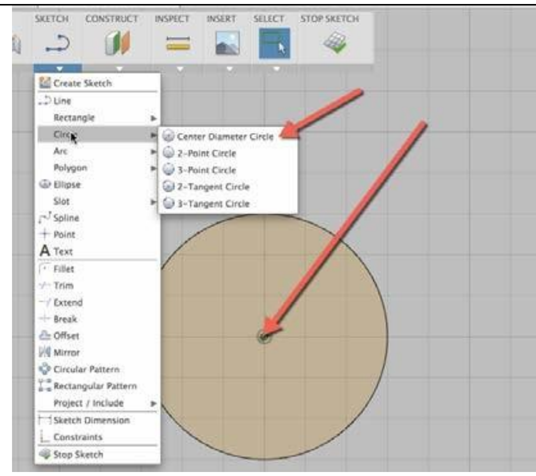
There are many commands available to create and edit sketch geometry.

Basics: This exercise uses a few sketch commands to get you comfortable. We will delete the geometry later.

Step 1 – Create a circle

1. Select **Sketch > Circle > Center Diameter Circle**.
2. Select anywhere in the screen to define the center point.
3. Move the mouse out until you have a size you like.
4. Click again to complete the command.

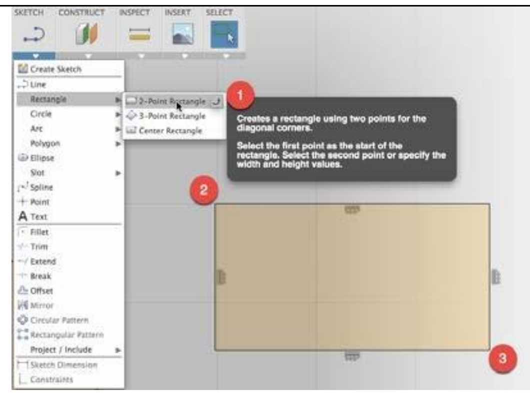
Sketches usually consist of many types of "primitive" geometry added together.



Step 2 – Create a rectangle

1. Select **Sketch > Rectangle > 2-Point Rectangle**.
2. Select anywhere in the screen to start the rectangle.
3. Move the mouse out until you have the size you want.
4. Click again to complete the command.

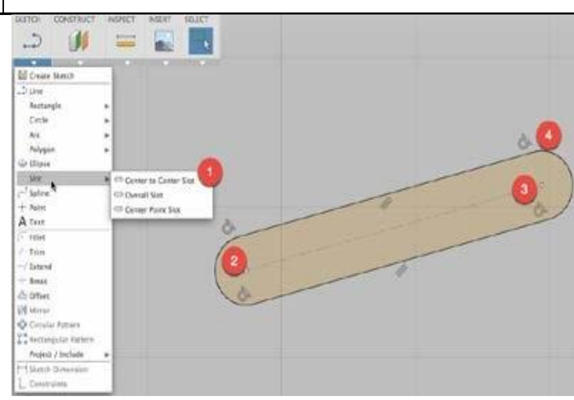
There are different ways to create circles and rectangles. Experiment with some of the other options in the menu. See more here:



Step 3 – Create a slot

1. Select **Sketch > Slot > Center to Center Slot**.
2. Select anywhere in the screen to start the slot.
3. Move out until you have a length you like then click to establish the slot length.
4. Now move out to define the width of the slot.
5. Click again to complete the command.

You will notice the various symbols that are being created on the geometry. These are "constraints." They control the relationships between the entities in the sketch.

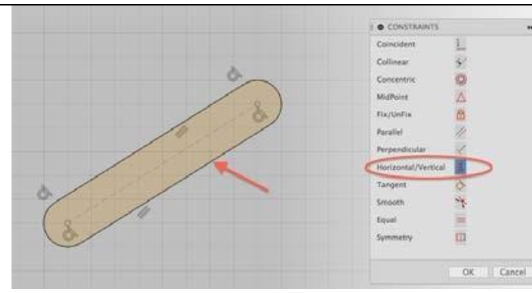


Step 4 – Constrain the slot

1. Select **Sketch > Constraints**.
2. Select one of the straight lines in the slot.
3. Select **Horizontal/Vertical**.

The Horizontal/Vertical constraint sets the geometry to whichever is closer: horizontal or vertical.

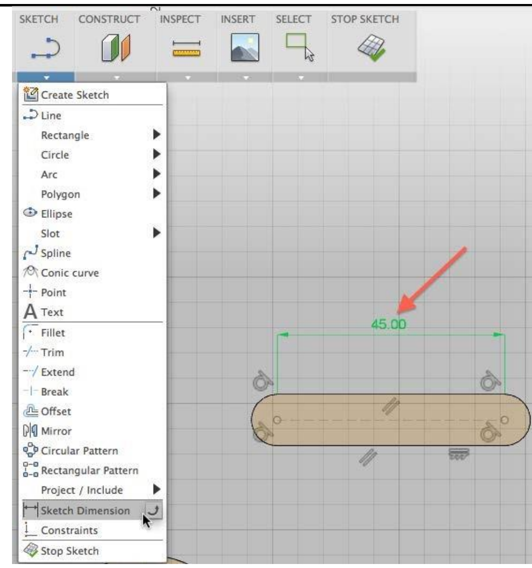
By adding a horizontal relationship to the slot we have "fixed" its movement. Try dragging on some of the points to see the behavior.



Step 5 – Dimension the width of the slot

1. Select **Sketch > Sketch Dimension**.
2. Select one of the horizontals lines.
3. Move the cursor out and click to place the dimension.
4. Enter a value for the dimension.

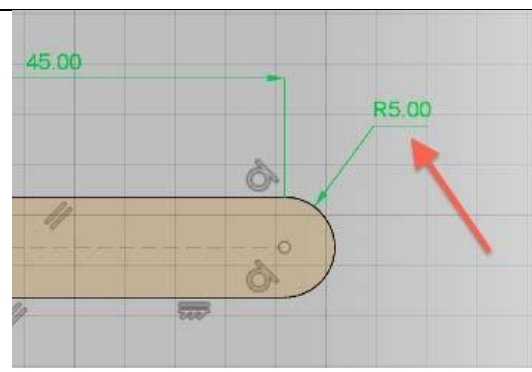
Typically we place dimensions after placing geometry.
This dimension now controls the length of the slot. You can double click it to change the value.

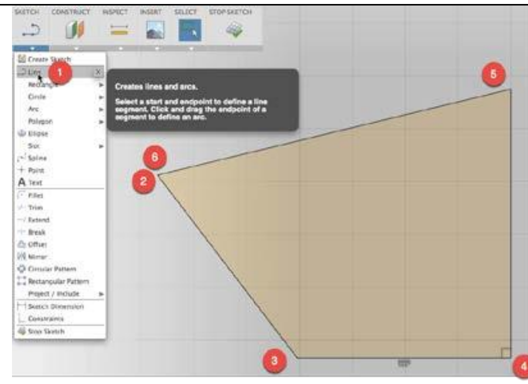
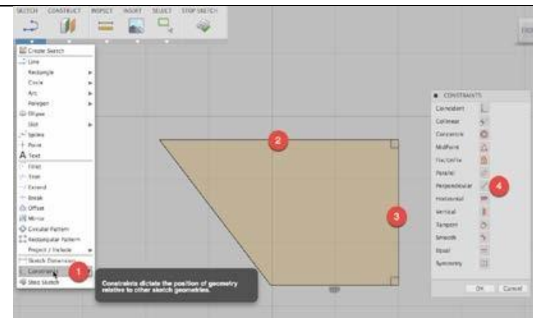
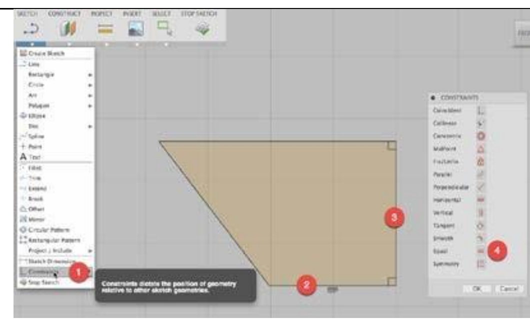
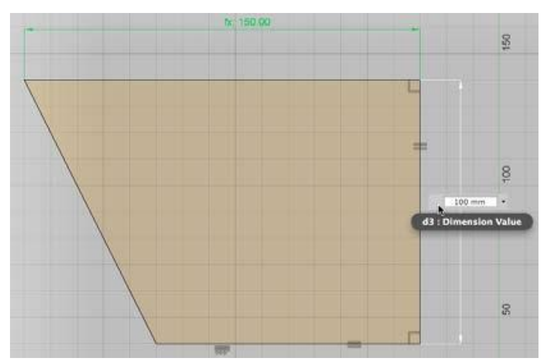


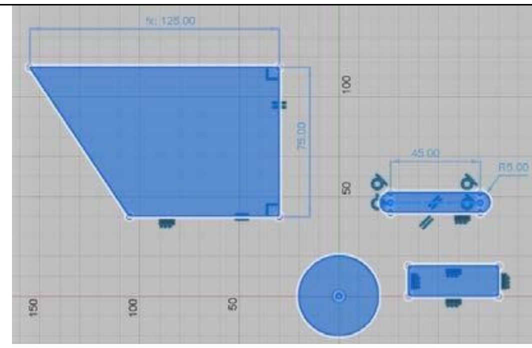
Step 6 – Dimension the radius of the slot

1. Now select the right arc (the dimension command should still be active).
2. Place the dimension.
3. Press **Esc** to end the command.

Notice that you cannot place a second dimension on the left arc. This is because it would conflict with the other constraints we have created.



Step 7 – Sketch a profile 1. Select **Sketch > Line**. 2. Draw the following shape by clicking in this order. 3. Make sure on the last line you "connect" it to the start point. If you finish it correctly you will see that the shape is shaded in. Notice that some constraints were created for you if you picked it right.	
Step 8 – Constrain the profile 1. Select **Sketch > Constraints**. 2. Select these two lines. 3. Select **Perpendicular**. Feel free to experiment with other constraints between the lines.	
Step 9 – Constrain the profile 1. Select Sketch > Constraints. 2. Select these two lines. 3. Select **Equal**. You see how you can create more intelligence in the models with relationships	
Step 10 – Dimension the profile 1. Select **Sketch > Sketch Dimension**. 2. Place a dimension on the vertical line. 3. Enter **100 mm** for the value and note the name of the dimension, d3 in this case. 4. Place a dimension on the top horizontal line. 5. Enter **d3 + 50** for the value. 6. Now change the value of the vertical dimension and see the horizontal update. There is another method to make relationships. To modify these relationships you can go to: Modify > Change Parameters	

Step 11 – Delete the sketch profiles 1. Zoom out to see everything. 2. Select everything in the sketch by dragging a selection box. 3. Press **Delete**. You should now have an empty sketch to start the next part of the lesson.	

Overview

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 help you to nurture sketching tools to create and edit a 2D sketch.

9. Base Sketch

Now we will begin to create some real sketch geometry that will be used to create 3D geometry.

<table>
<tr>
<td>
Step 1 – Sketch a circle

1. Select Sketch > Circle > Center Diameter Circle.
2. Select the origin (center point of the grid).
3. Drag out a circle about 30 mm in Diameter. We will add the specific size later.
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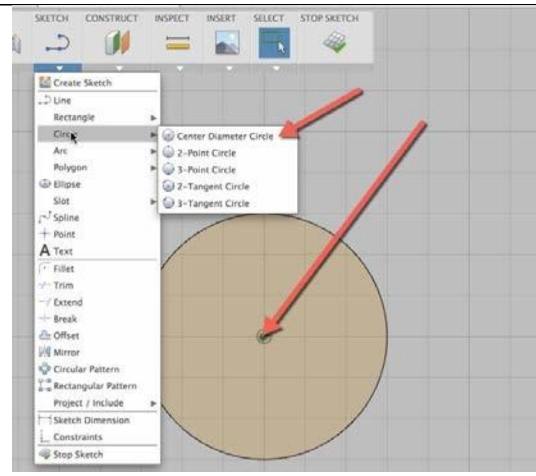
<td>


</td>
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<tr>
<td>
Step 2 – Sketch another circle

1. Select Sketch > Circle > Center Diameter Circle.
2. Create another circle in roughly this position.

Don’t worry about a specific size or position now we will fix that later.
</td>
<td>


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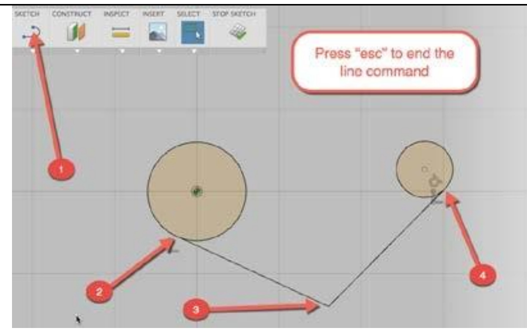
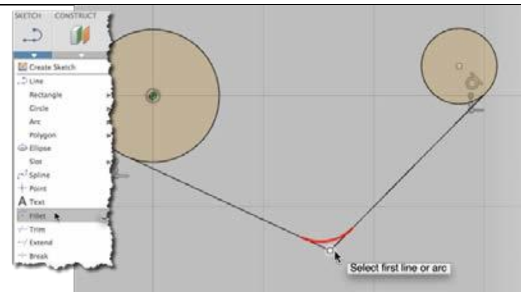
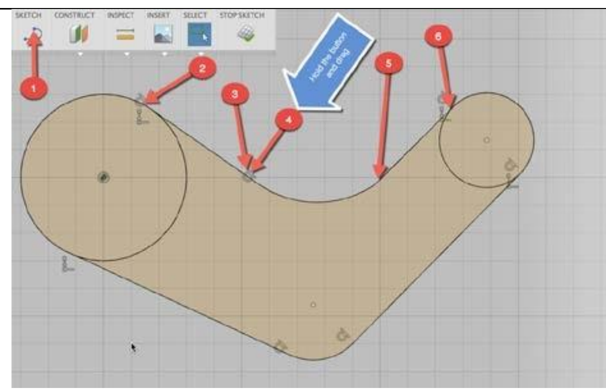
</table>

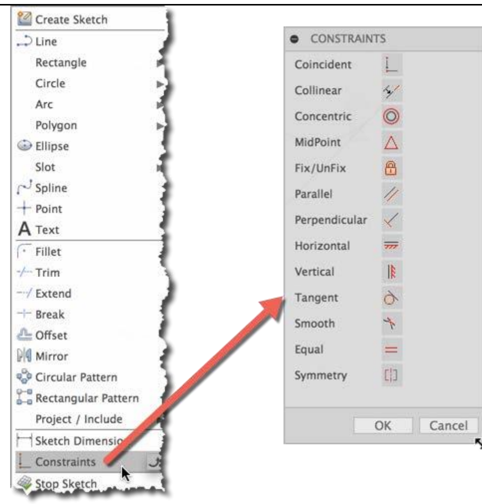
Step 3 – Create lines 1. Select **Sketch > Line**. 2. Click on the left circle to start the line. 3. Hold **Shift** then click again where shown to make the first line. Holding shift locks the line tangent to the circle. 4. Click once more on the right circle at the position where the line is tangent to the circle. 5. Press **Esc** to end the command.	
Step 4 – Fillet the lines 1. Select **Sketch > Fillet**. 2. Select the intersection of the two lines. 3. Type **10 mm** into the input box. 4. Press **Enter**. When you create a fillet "tangent" relations are added to the sketch.	
Step 5 – Complete the profile 1. Select **Sketch > Line**. 2. Select the left circle (2) as shown. 3. Hold **Shift** then click near the number (3) as shown. 4. Hold down the left button on the end point of the line and drag to create an arc. 5. Click (5). 6. Click on the right circle (6). If you make a mistake just feel free to delete the lines and try again.	

Relationships: Now we will see how to create dimensions and constraints for our sketch.

Step 1 – Display the Constraints dialog box

1. Select **Sketch > Constraints**.

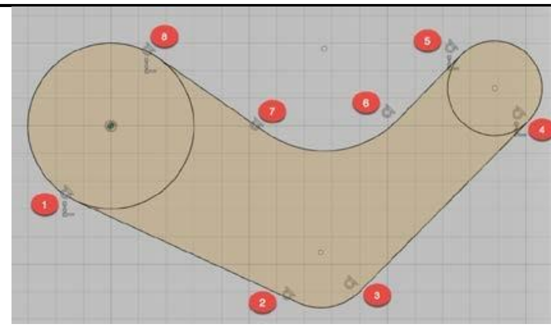
This brings up the dialog box where you can create relationships between sketch entities. Constraints will determine how your sketch behaves. Once a relationship like tangent or perpendicular is applied then it is persistent as you continue to modify the sketch.



Step 2 – Add tangent constraints

1. Select the **Tangent** constraint.
2. Apply tangent constraints at the locations shown if they do not already exist.
3. Make sure you have 8 tangent symbols on your sketch.

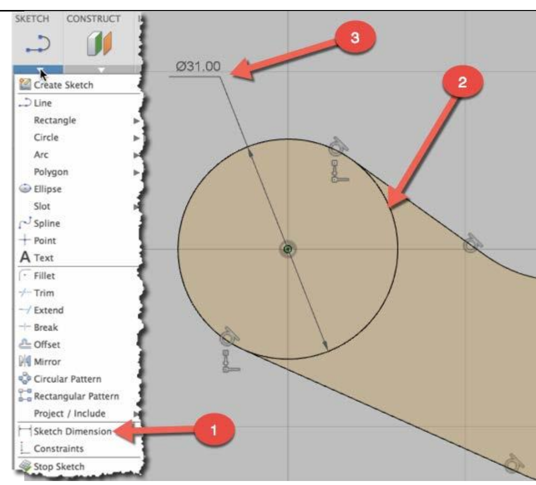
You may have noticed that there are already some of these created as you were sketching. Many commands in Fusion 360 allow you to create relationships "on-the-fly." As you become more familiar with the tool this will make more sense.



Step 3 – Dimension the circle

1. Close the constraints dialog box.
2. Select **Sketch > Sketch Dimension**.
3. Select the left circle.
4. Click again to place the dimension.
5. Change the value to **31mm**.
6. Press **Enter**.

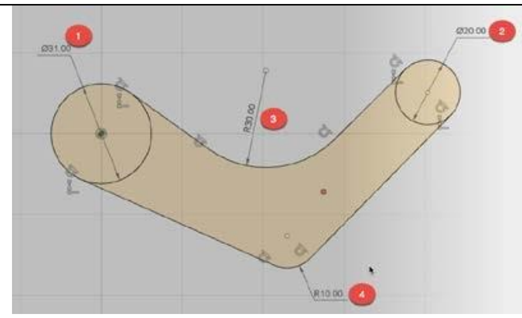
This is how we size our sketch. It is very common to draw things out in rough position then go back and add sizing information. Later we will show how to add this at creation.



Step 4 – Dimension the profile

1. Repeat the process for the right circle and two arcs.
2. Change the values to match those shown here.

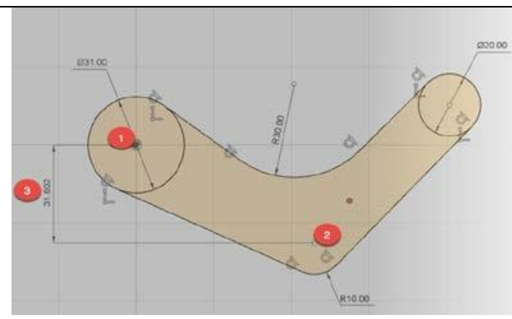
Once you have started the dimension command it stays active. You do not have to go to the menu each time to activate it.



Step 5 – Add more dimensions

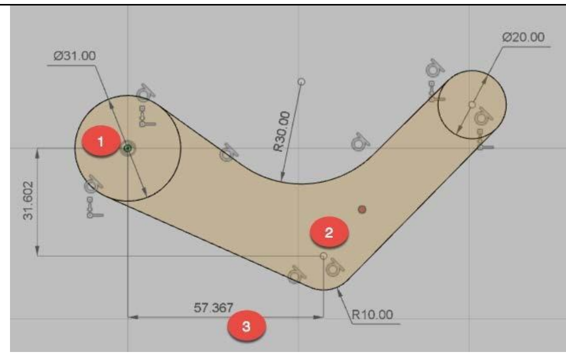
1. Make sure the dimension command is still active.
2. Select the center point of the left circle.
3. Select the center point of the lower arc.
4. Place the dimension to the left center of the two points.
5. Don't change the value yet (Don't worry if your value is different).

Depending on where you place the dimension you get different results. You can make horizontal, vertical and shortest distance dimensions by where you drag the dimension when you are placing it. Once the dimension is placed you can't change it. If you make a mistake delete the dimension and start again.



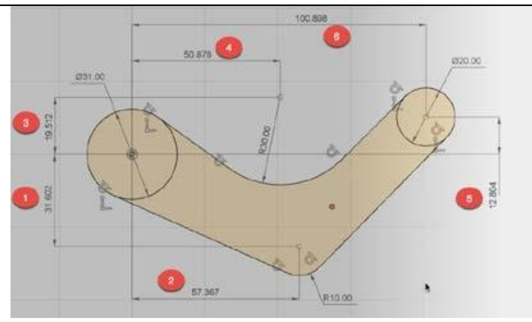
<table>
<tr>
<td>
Step 6 – Add more dimensions

1. Select the center point of the left circle.
2. Select the center point of the lower arc.
3. This time place the dimension to the lower center position to create a horizontal dimension.
4. Place the dimension but don't edit the values.

Sometimes it is better to place your dimensions first before changing the values. When some geometry doesn't have any relationships yet it can behave in unexpected ways.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 7 – Add more dimensions

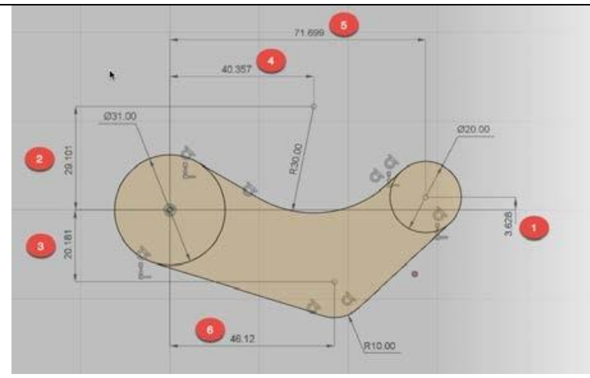
1. Repeat the process for the upper arc and the left circle.
2. Create two dimensions between the left circle and the center point of the right circle.

Your sketch should look like this although you will have different values for the dimensions. We will change the values in the next step.
</td>
<td>

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

Step 8 – Set the dimension values

1. Double click a dimension to change its value.
2. Change the values of the dimensions to match those shown.
3. Follow the order of the numbered balloons for the best results.

What is happening under the hood is the software is “solving” all the relationships at once. It is possible that you will try to enter a dimension that conflicts with the other values. As you become more familiar with the behavior you will get a sense of the best order to makes changes for the desired result.



The end result of a sketch is often to create a 3D feature. Fusion automatically discovers closed profiles that can be used in 3D features.

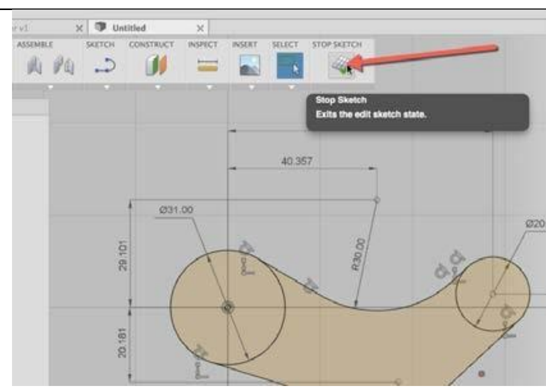
Creating Geometry: Next we will use this sketch to create 3D geometry.

Step 1 – Leave the sketch environment

1. Select **Stop Sketch**.

This completes the sketch. If you need to edit the sketch again you can select it from the sketch folder in the browser on the left of your screen.

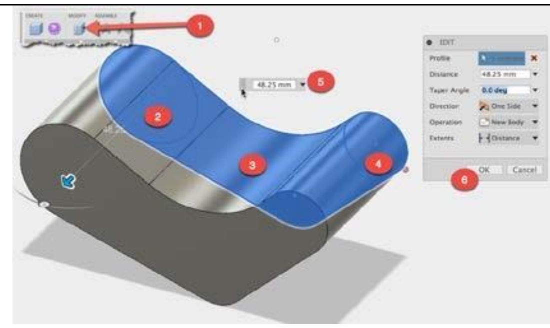
You can also rename it from there.



Step 2 – Create 3D geometry

1. Select **Modify > Press Pull**.
2. Select the three profiles of the sketch so that the entire area is highlighted.
3. Enter a value of **48.25 mm**.
4. Press **OK**.

You have now created a base 3D extrusion

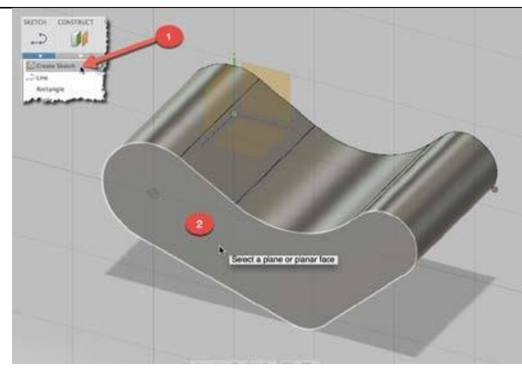


Creating Sketch References: Here will use the geometry we just created to make more sketches.

Step 1 – Create a new sketch

1. Select **Sketch > Create Sketch**.
2. Select this face of the geometry.

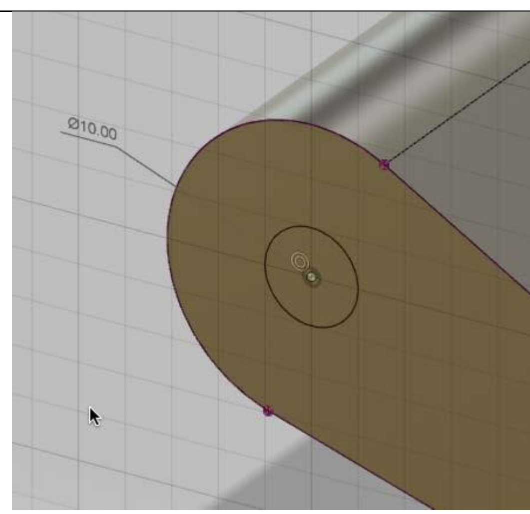
Previously we sketched on an origin plane. Now we are sketching directly on the model geometry.

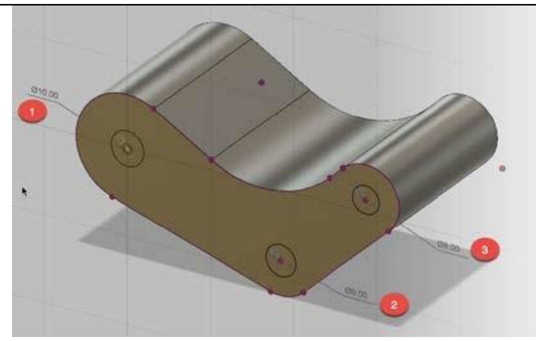
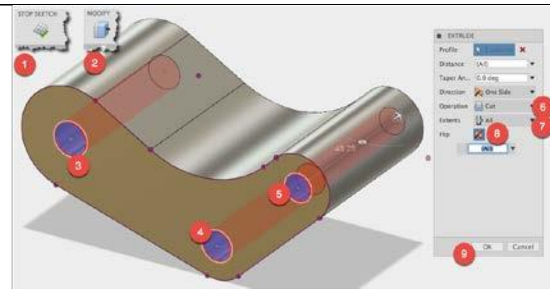


Step 2 – Create a circle

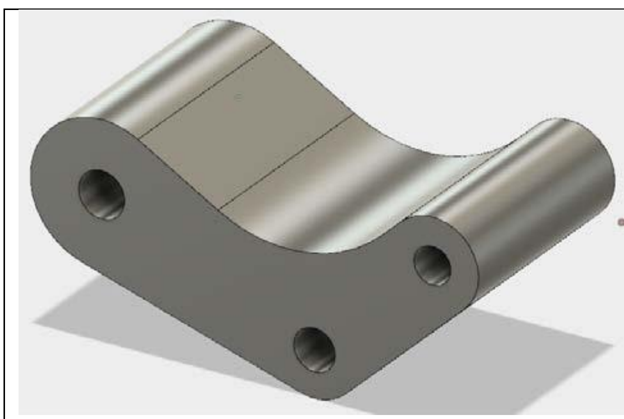
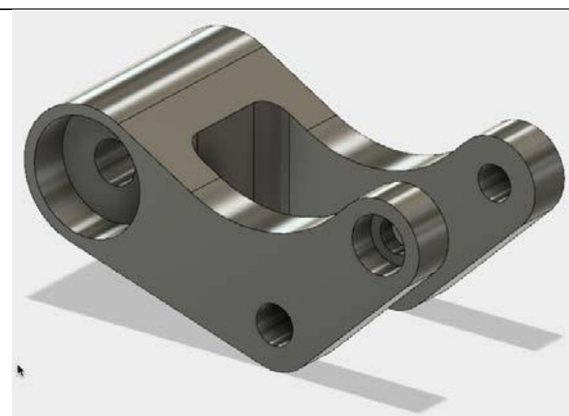
1. Select **Sketch > Circle > Center Diameter**.
2. Select the center point of the right circle.
3. Drag the circle then type the value **10 mm**.
4. Press **Enter** twice.
5. You can move the dimension out if you want as well.

This circle is now “locked” to the center of the original boss feature

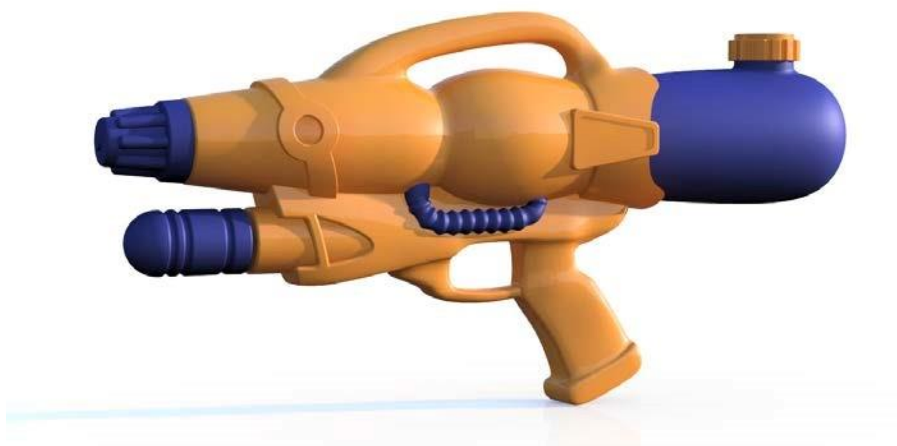


Step 3 – Create additional circles 1. Repeat the process for two more circles. 2. Make sure to attach the center to the center of the lower arc and the left circle. 3. Use a diameter of **8 mm** for both circles. 4. Select **Stop Sketch**. This circle is now "locked" to the center of the original boss feature.	
Step 4 – Cut holes in the body 1. Select **Modify > Press Pull**. 2. Select the three circles. 3. Select: **Cut**, **All**, and **Flip Direction**. 4. Select **OK**. You created a cut in the geometry that references the position of the base arcs.	 Launch Video

Challenge: Congratulations you have finished the model! If you want to challenge yourself try to use the techniques in here to finish the model. Otherwise you will learn a slightly different method of creating this part in the modeling lesson.

 What we've done so far	 Completed Rocker Arm Model

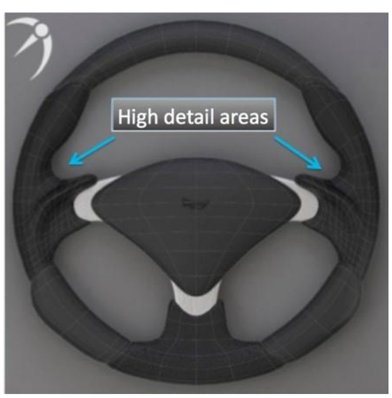
Sculpting

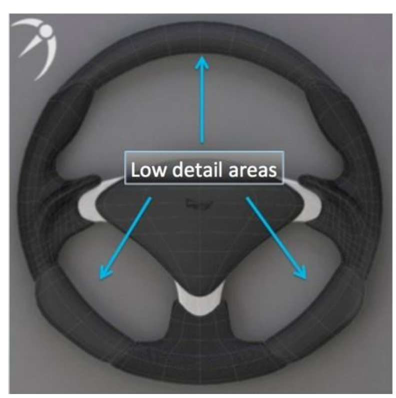


Overview

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 TSplines is the ability to add detail only where necessary.





This may not sound like much, but this is one of the biggest challenges in most subdivision 3D modeling tools. By only adding data in necessary locations, a single T-Spline surface can be incredibly smooth, while still having areas of high detail and remaining easy to manipulate.

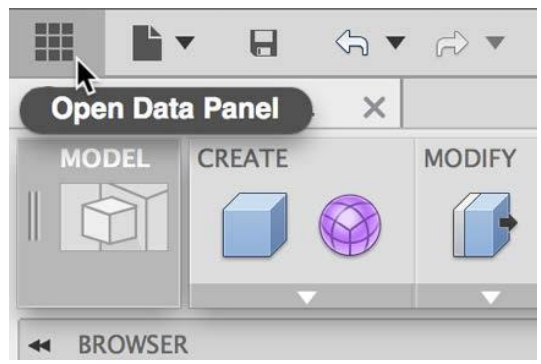
Learning Objectives

In this section you will learn how to:

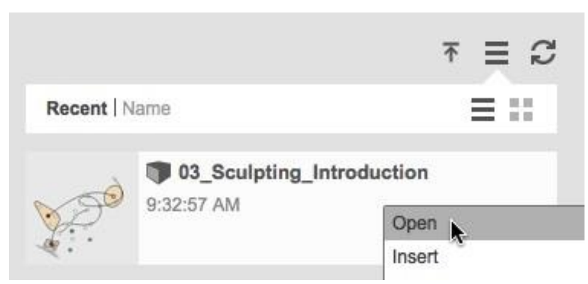
- Create a T-Spline form
- Modify a T-Spline form
- Add details to a T-Spline form
- Create a T-Spline form based on a reference image **Creating T-Spline Forms**

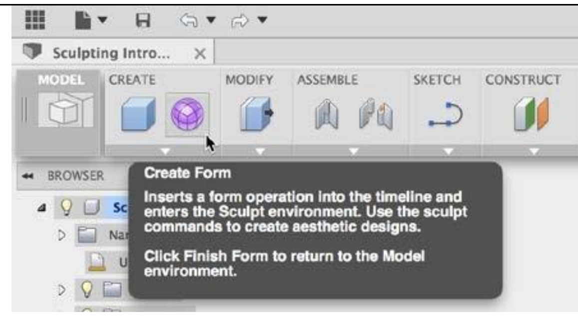
Open Fusion 360 design file and enter the Sculpt Workspace: In this section you open the introductory design file and go to the Sculpt workspace to create T-Spline forms.

<table>
<tr><td>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.</td><td>
</td></tr>
<tr><td>Step 2 – Open the design

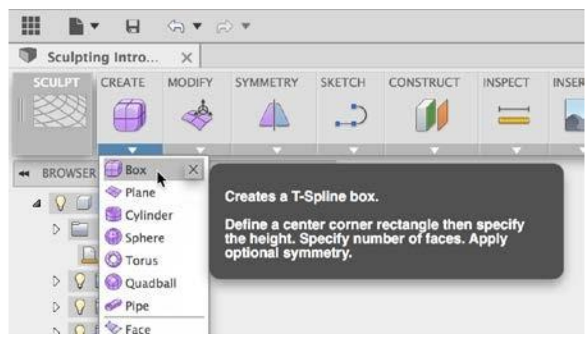
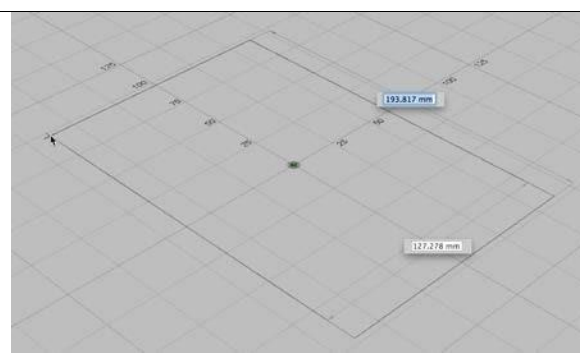
In this module we will be using the 03_Sculpting_Introduction.f3d file to complete the exercise. If you haven’t set up a new project and uploaded the necessary designs, please follow the steps in the Introduction module.

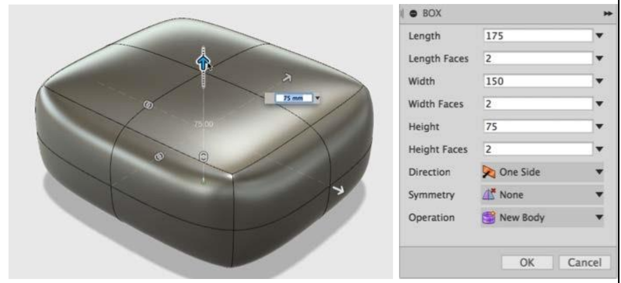
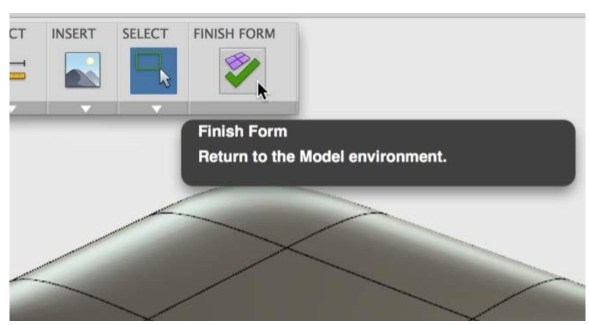
1. At the top left of the Data Panel, select the project where you uploaded the 03_Sculpting_Introduction.f3d file.
2. Navigate to this design and either doubleclick or right-click and select open.
3. When the design has opened in your modeling window, click on the icon to close the Data Panel.</td><td>
</td></tr>
</table>

Step 3 – Go to the Sculpt workspace 1. Click **Create > Create Form** to enter the Sculpt workspace. 2. A dialog box appears, telling you to click Finish Form to return to the model workspace whenever you're finished sculpting. 3. Click **OK**. You will notice that the top ribbon will change to include commands specific to sculpting.	

10.Create a T-Spline Primitive Form

In this section you learn how to create the most basic T-Spline form: a Primitive.

Step 1 – Select the T-Spline primitive 1. Click the drop-down arrow under **Create** to expand the list of creation commands. 2. You will see a list of T-Spline primitives: Box, Plane, Cylinder, Sphere, Torus, and Quadball. 3. Click on **Box** to create a T-Spline box.	
Step 2 – Position the box 1. Click the bottom plane to place the bottom of the box on this plane. 2. Click on the origin to specify the center point of our box's 2D profile. 3. Move your mouse and click again to specify the size of the rectangle.	

Step 3 – Dimension the box 1. In the new 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**. When setting the height, if it is changing at too large of increments, simply zoom in to reduce the size of each step.	
Step 4 – Subdivide the box 1. In the dialog window set the length faces equal to **3.** 2. Drag the set of arrows pointing up in the **positive** direction to increase the number of width faces, and set it equal to **3**. 3. Click **OK** to complete the primitive setup.	
Step 5 – Finish form 1. The last step is to tell Fusion 360 that we are – *temporarily* - finished sculpting our first Form. 2. At the top right of the Ribbon, click **Finish Form**. 3. You will return to the Model workspace. With our first T-Spline primitive complete, it's time to learn more methods for creating T-Spline forms.	

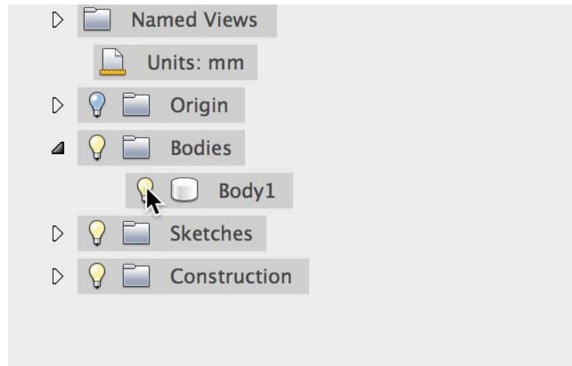
11.T-Spline Form Creation – Revolve

In this section you learn how to create a T-spline form using the revolve command and a sketch. The Revolve command creates a form by rotation 2D geometry about a fixed axis.

Step 1 – Hide the previous T-spline form

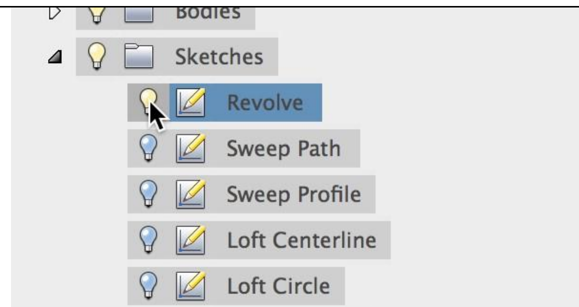
1. In the Browser, click the drop-down arrow next to your **Bodies** folder
2. Click the light bulb next to **Body1** to turn off the visibility.
3. Click **Create > Create Form**.

We hide our T-Spline primitive in order to keep things neat, but we'll be returning to it later on.



Step 2 – View the revolve sketch

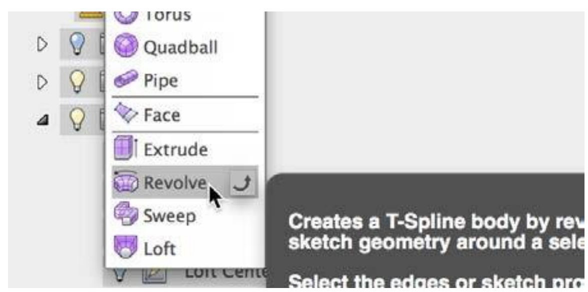
1. In the Browser, click the drop-down arrow next to your **Sketches** folder.
2. Click the light bulb next to **Revolve** to turn on the visibility of the Revolve sketch.



Step 3 – Start the Revolve Command

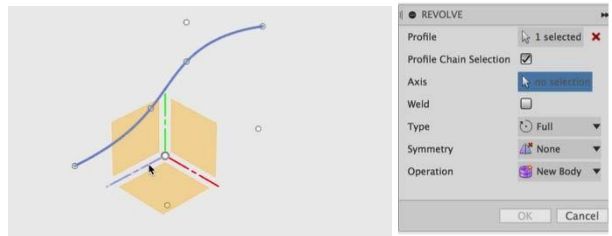
1. Click **Create > Revolve**

In addition to Revolve, you will see three more sketch-based form creation commands: Extrude, Sweep, and Loft.

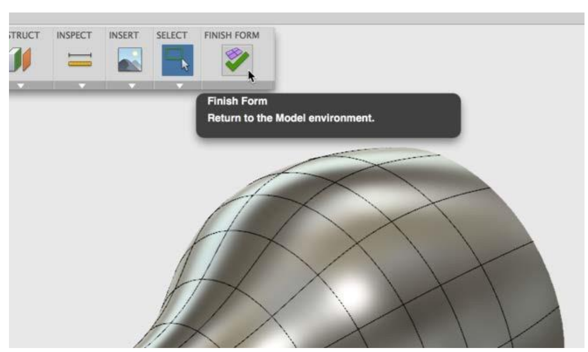


<table>
<tr>
<td>Step 4 – Select the profile and axis

1. Click on the sketch curve shown in the workspace to specify the Profile.
2. Click no selection in the dialog window next to Axis to enable selection.
3. Click on the Z-axis shown in Blue.

If you'd like Z to be "up," this can be changed under the preferences.</td>
<td></td>
</tr>
<tr>
<td>Step 5 – Explore the revolve details

1. In the Revolve dialog window, change the Type from Full to Angle.
2. Enter 90 degrees for the angle.
3. Change the Direction from One Side to Symmetric.
4. Click OK.</td>
<td></td>
</tr>
<tr>
<td>Step 6 – Finish form

1. Click Finish Form to complete this Sculpting session.
2. You return to the Model workspace.
3. The bottom timeline populates with our second T-Spline form!</td>
<td></td>
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</table>

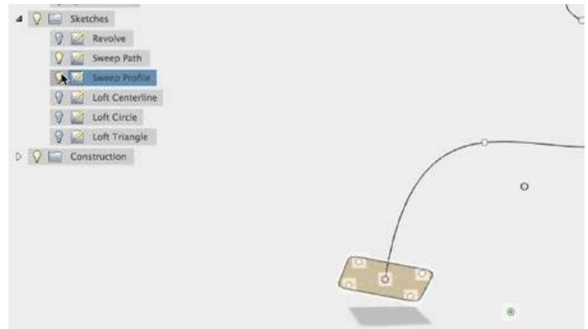
12.T-Spline Form Creation – Sweep

In this section you learn how to create a T-spline form using the Sweep command and two sketches. The Sweep command will create a form by moving a 2D profile on a particular path.

Step 1 – Toggle visibility

1. Hide the previously created **Body**.
2. Hide the **Revolve Sketch**.
3. Show the sketches called **Sweep Path** and **Sweep Profile**.

Use the light bulb icons to control the visibility of objects.

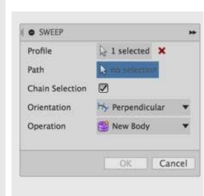


Step 2 – Start the sweep command

1. Click **Create > Create Form**.
2. Click **Create > Sweep**.

Step 3 – Select the Profile and Path

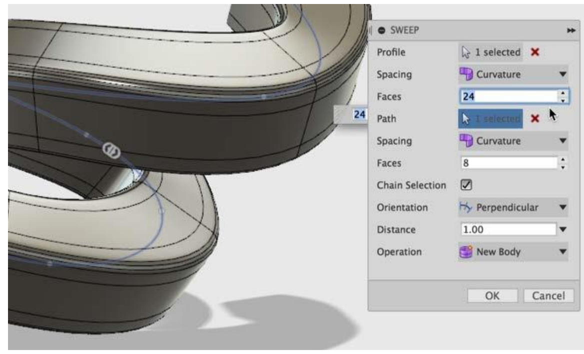
1. Click the **Square** profile visible in the workspace to specify the profile.
2. Click **no selection** next to **Path** to begin selecting a path to sweep along.
3. Select the curve profile for the Path.



Step 4 – Closer match the profile

1. To closer match our rounded-square profile, we need to increase the number of faces for the profile.
2. In the dialog window, set the number of faces for the **Profile** equal to **24**.

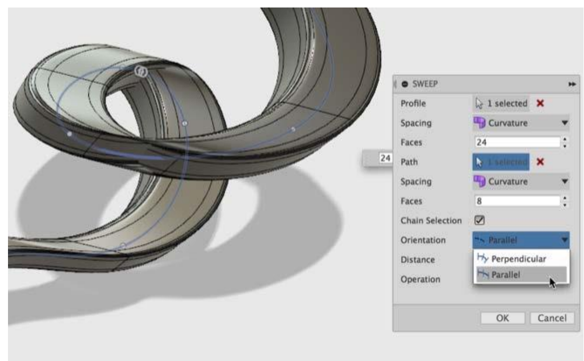
The greater number of faces, the closer the body matches the path. You need to balance the number of faces with the accuracy of the body.



Step 5 – Change the sweep orientation

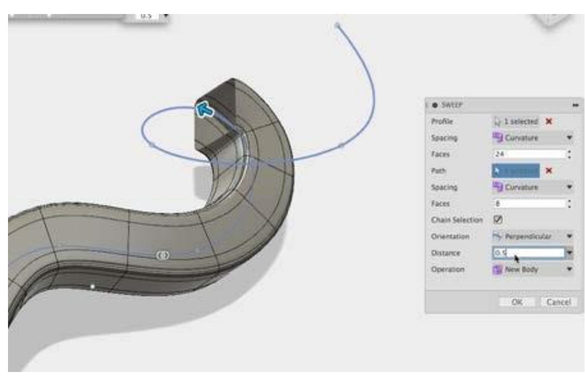
1. In the dialog window, change the Orientation from Perpendicular to **Parallel**.
2. Observe how the sweep behavior drastically changes.
3. Change the Orientation **back** to **Perpendicular**.

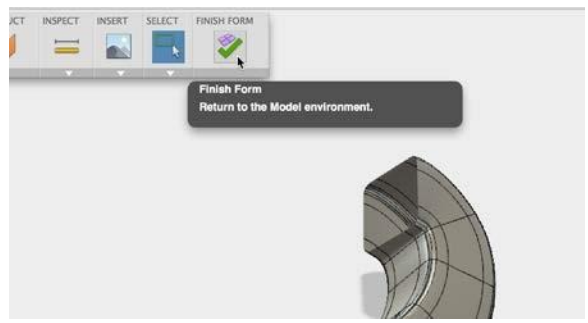
For a sweep, the optimal orientation is more often than not Perpendicular.



Step 6 – Specify the sweep distance

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.
4. Click **OK**.



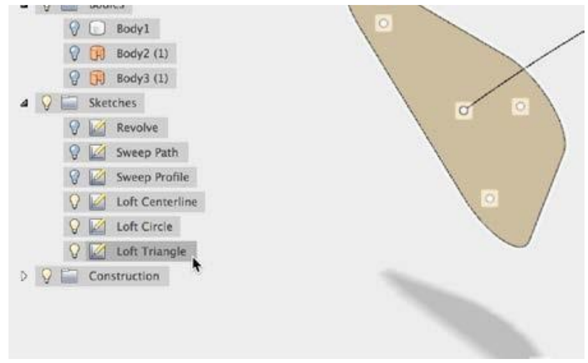
Step 7 – Finish form 1. Click **Finish Form** to complete this Sculpting session. 2. You return to the Model workspace. 3. The bottom timeline populates with our third T-Spline form!	

13.T-Spline Form Creation – Loft

In this section you learn how to create a T-spline form using the Loft command and three sketches. As you will see, the Loft command creates a transitional form between two profiles.

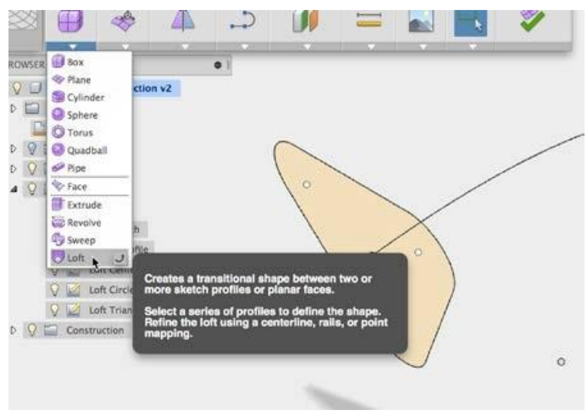
Step 1 – Toggle visibility

1. Hide the previously created **Body**.
2. Hide the two Sweep sketches.
3. Show the sketches called **Loft Centerline, Circle,** and **Triangle**.



Step 2 – Start the Loft command

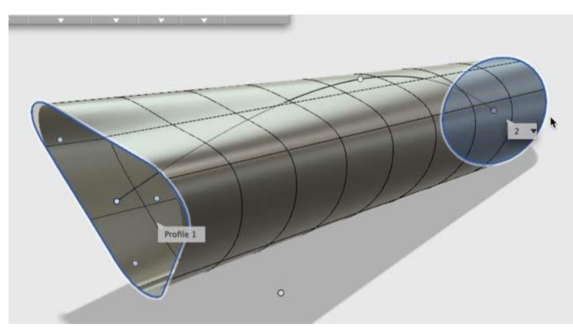
1. Click **Create > Create Form**.
2. Click **Create > Loft.**



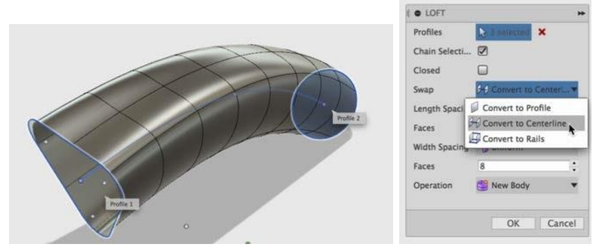
Step 3 – Select the profiles

1. Click the **Triangular** profile in the canvas.
2. Click the **Circular** profile in the canvas.
3. A transitional shape between the triangle and the circle is previewed.

This creates a form in a straight-line between our two profiles. In the next step, we add an optional centerline for more control.

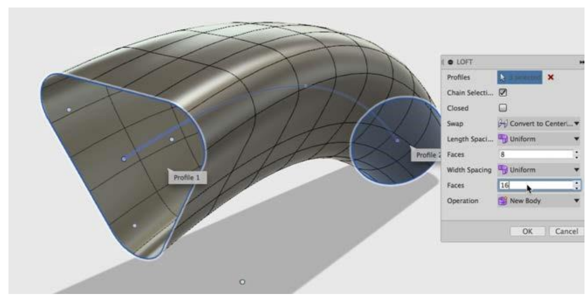


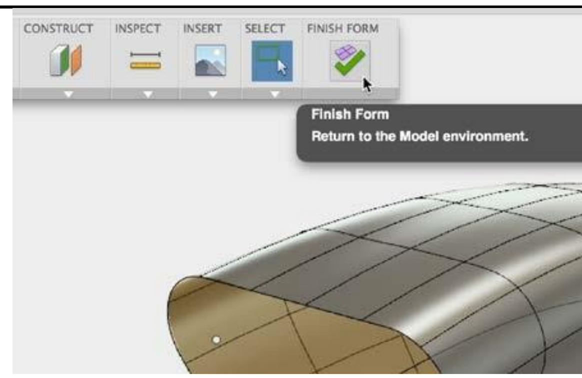
<table>
<tr>
<td>
Step 4 – Specify the centerline

1. Click the **centerline** curve visible in the canvas.
2. You receive an error because Fusion thinks this is a *third* profile.
3. To specify this curve as a centerline select **Convert to Centerline** under *Swap* in the dialog window.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 5 – Closer match the profile

1. To closer match our triangular profile, we need to increase the number of faces for the profile.
2. In the dialog window, set the number of faces for the **Width** equal to **16**.
3. Click **OK**.

The greater number of faces, the closer the body matches the profile.
</td>
<td>

</td>
</tr>
<tr>
<td>
Step 6 – Finish form

1. Click **Finish Form** to complete this Sculpting session.
2. You return to the Model workspace.
3. The bottom timeline populates with our fourth T-Spline form!
</td>
<td>


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

14.Modify a T-Spline Form

Now that we learned different methods for creating T-Spline forms, the next step is to learn how to *modify* them. Being that sculpting in Fusion 360 allows for freeform shape manipulation, it's **very** rare that a T-spline form does not require any editing. If it did not, then it might as well have been performed in the Model or Patch workspaces. It's important to create forms in the Sculpt workspace that you intend to be organic.

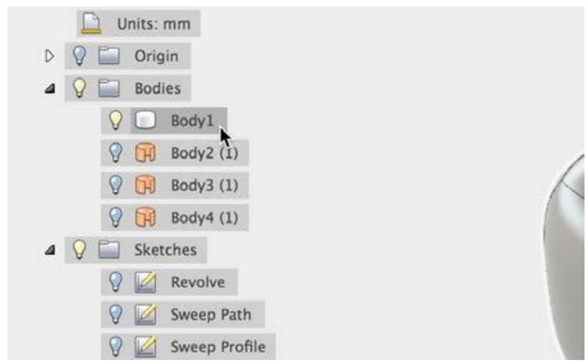
Edit Form: In this section you learn how to modify a T-Spline body with the primary T-Spline modifying command, Edit Form. We start by editing the first T-Spline primitive that we created, the Box.

<table>
<tr>
<td>

Step 1 – Toggle visibility

1. Hide the previously created **Body.**
2. Hide any visible sketches.
3. Turn on the visibility of our primitive TSpline Box – **Body 1**.

</td>
<td>



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<td>

Step 2 – Edit form operation

1. In the parametric timeline, locate the 4 TSpline form icons.
2. Hover over the first icon and confirm that the primitive **Box** becomes highlighted.
3. Right-click on this form icon in the timeline and select **Edit**.
4. You roll back to that point in time and enter the Sculpt workspace again.

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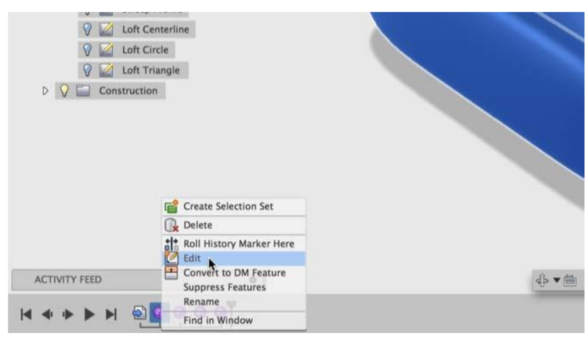

</td>
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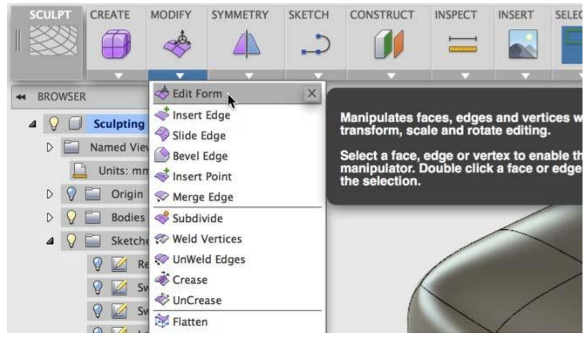
</table>

Step 3 – Start the Edit Form command

1. Click **Modify > Edit Form**. There are many modifying commands in the Sculpt workspace.

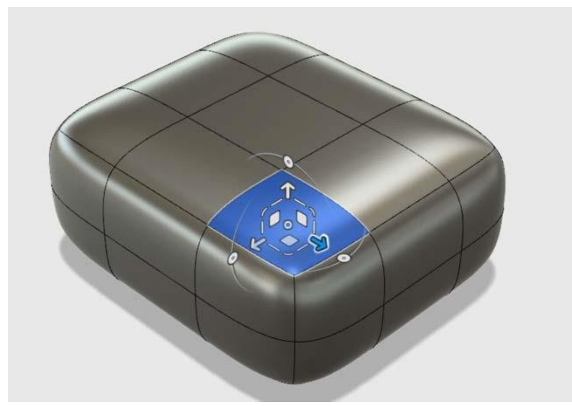
Alternatively, you can click the large Edit Form icon that resides in the ribbon by default.



Step 4 – Select geometry to edit

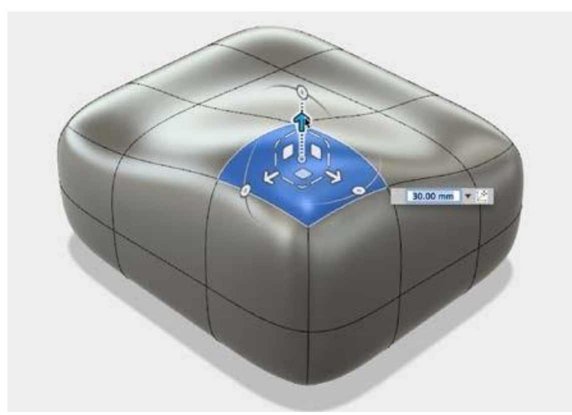
1. The Edit Form command can directly manipulate **Faces**, **Edges**, and **Vertices**.
2. Click on a top face located at one of the corners of the box.
3. The manipulator displays on the face with an assortment of tools.

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



Step 5 – Single direction translation

1. Click and drag on the **arrow** pointing **up** to translate the selected face up by **30 mm**.
2. The surrounding faces move to maintain continuity.



<table>
<tr>
<td>
Step 6 – Planar translation

1. Click and drag on one of the white boxes to translate the selected face on a plane.
2. More surrounding faces move to maintain continuity throughout the form.

Planar translation is incredibly useful, as you will often want to move geometry in all directions but one.
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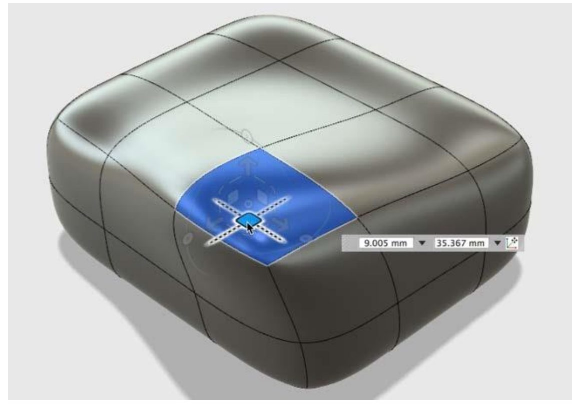
<td>


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</table>

<table>
<tr>
<td>
Step 7 – More translation

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

In the following sections we cover the manipulators: scale and rotate.
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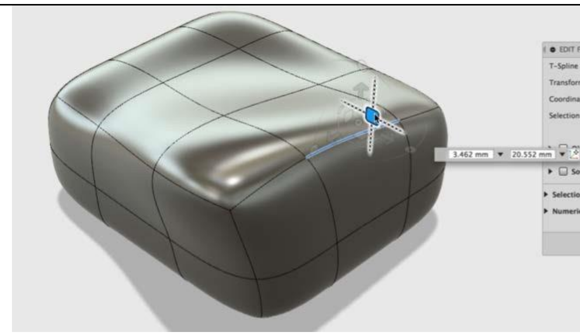
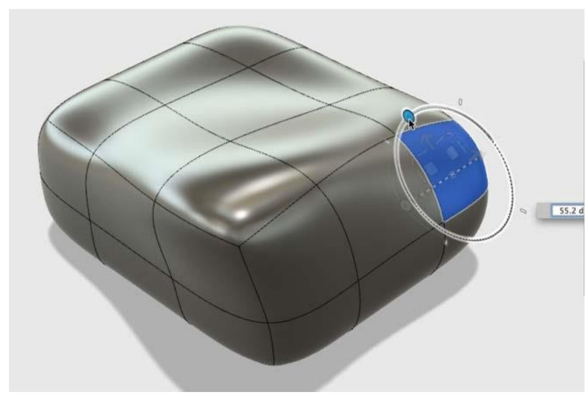

</td>
</tr>
</table>

Edit Form – Rotate and Scale: In this section you learn how to modify T-Spline geometry using the rotate and three scale manipulators.

<table>
<tr>
<td>
Step 1 – Rotation

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.

Be careful not to rotate geometry too far, as selfintersecting faces, or geometry that twists through itself will not result in a watertight solid body.
</td>
<td>

</td>
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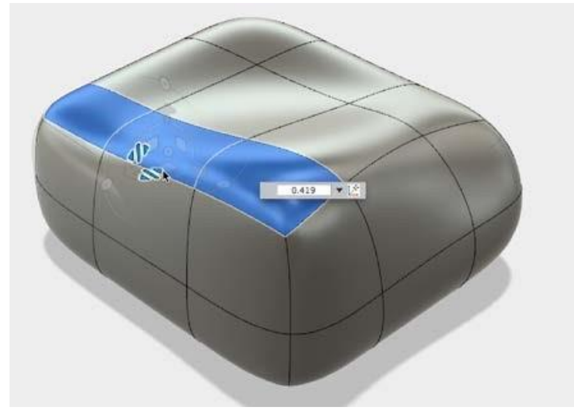
</table>

<table>
<tr>
<td>
Step 2 – 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.
3. You will see that scaling in each axis will have a significantly different effect.
</td>
<td>


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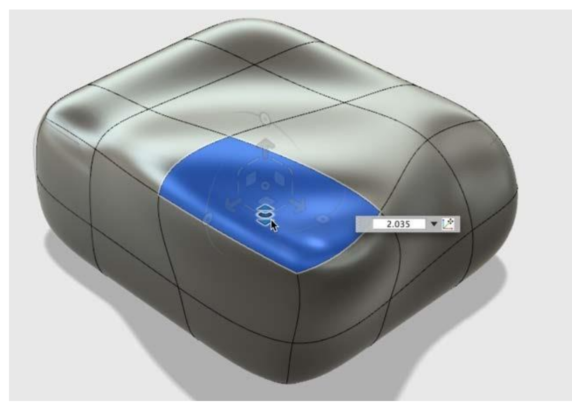
<tr>
<td>
Step 3 – Planar scaling

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.

Similar to planar translation, it is often very useful to scale geometry in two directions.
</td>
<td>


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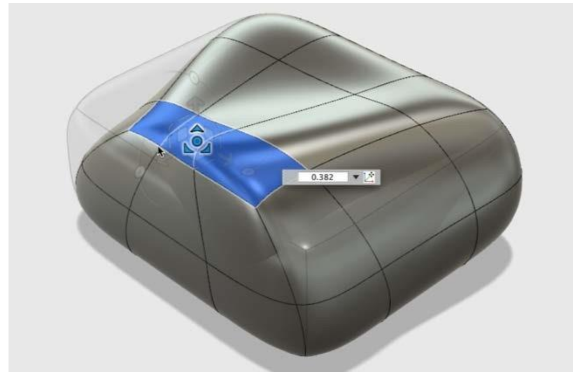
</table>

Step 4 – 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**.

With the basics of editing a T-Spline form covered, we move on to a very useful function of the Edit Form command: Adding additional geometry.

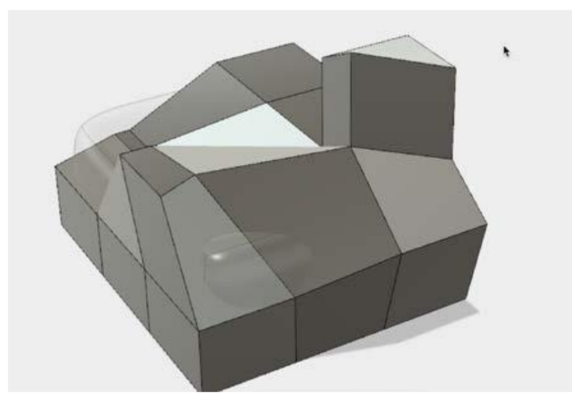


Edit Form – Add Geometry: In this section you learn how to use the Edit Form command to add geometry to your T-Spline form, rather than “stretching” it.

Step 2 – Box Mode

1. We will now turn on **Box Mode**, which displays a simplified control cage of our TSpline form.
2. On a Mac: hold **control** and press **1**.
3. On a PC: hold **alt** and press **1**.
4. The control frame for the T-Spline body is displayed.

Box mode improves performance and can be useful in finding problem areas.



<table>
<tr>
<td>

Step 3 – Add geometry in Box Mode

1. With Box mode enabled, hold **altoption/alt**.
2. **Translate** outwards in a single direction again.
3. You will see a similar effect as before, but with slightly faster performance.

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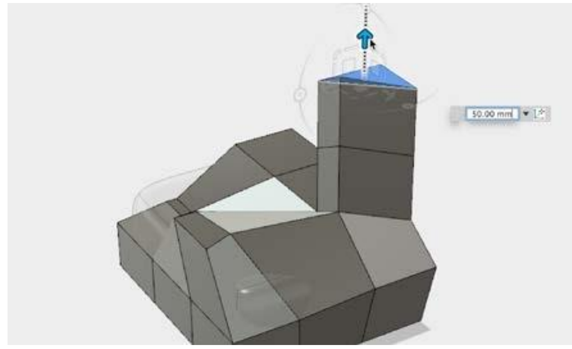

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Step 4 – Return to Smooth Mode

1. To view how a T-Spline form truly looks, we will return to **Smooth Mode**.
2. On a Mac: hold **control** and press **3**.
3. On a PC: hold **alt** and press **3**.

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Step 5 – Finish form

1. We'll temporarily finish editing our TSpline form.
2. Click **Finish Form** in the Ribbon.
3. You will return to the model workspace.

That wraps up our introduction to editing a T-Spline form! Next, we take a look at adding additional detail to a T-Spline form by inserting and deleting edges.

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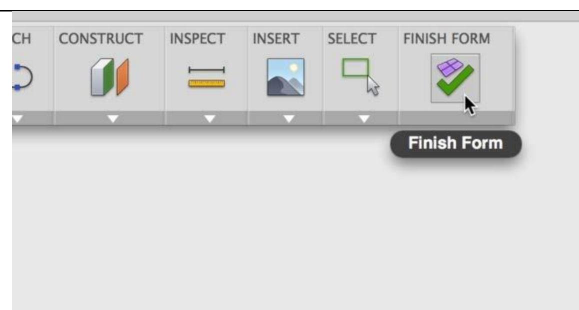

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</table>

15.Add Details to a T-Spline Form

T-Spline forms are flexible in the fact that edges can be inserted at particular locations, as well as deleted. It is very common to add additional edges to a T-Spline form in the early design stages and remove excess edges towards the end to ensure a very smooth form.

Insert T-Spline Edge: In this section you will learn how to insert an edge in a T-Spline form and how this affects the shape.

<table>
<tr>
<td>Step 1 – Create a New T-Spline form

Enter the Sculpt workspace by clicking Create > Create Form.
Create another Box primitive whose length, width, and height are 100mm, 100mm, and 200mm.
Set the number of length, width, and height faces equal to 4, 2, and 2.
Click OK.

</td>
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</tr>
<tr>
<td>Step 2 – Start the Insert Edge command

Click Modify > Insert Edge.
Double-click on one of the middle edges to select the entire middle loop.

The input for the Insert Edge command tells Fusion what reference edge you’d like to add. Selecting a single edge adds one edge, while selecting a loop adds a second loop.
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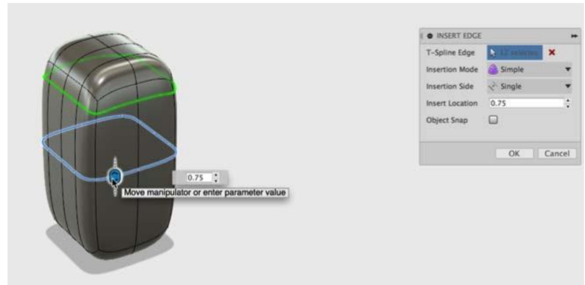
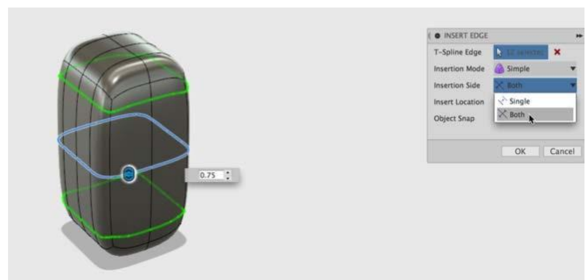
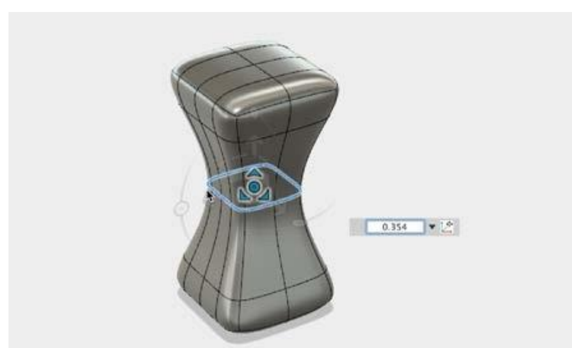
</table>

Step 3 – Adjust the insert location 1. Click and drag the manipulator to adjust the position of the inserted edge. 2. The value between 0 and 1 can be thought of as the percent between two adjacent edges. 3. By hand, or with the text field, set the **Insert Location** equal to 0.75.	
Step 4 – Adjust the insert specifics 1. With the Insert Mode set to Simple, the edge will be added, but the form will change, and this is okay for this application. 2. Change the **Insert Side** from Single to **Both**. 3. Click **OK**. Tip*:* If you want to add an edge, but maintain the existing form, set the Insert Mode to Exact.	
Step 5 – Explore the result 1. After the edges are inserted, the form will certainly change. The top and bottom edges become sharper due to the additional edge. 2. Double click on the **middle edge loop.** 3. Start the **Edit Form** command. 4. **Universal scale** this edge loop inward using the center manipulator. 5. Click **OK**. As you can see, having additional edges allows for more complex form creation. In the next section we take a look at deleting an edge, and the effect that has on the form.	

16.Delete T-Spline Edge

In this section you learn how to remove an edge in a T-Spline body and how this affects the shape.

<table>
<tr>
<td>
Step 1 – Select an edge loop

1. Select the recently added upper edge loop by double-clicking on one of its edges.
2. Click Modify > Delete.
3. Click Finish Form.

The form has again drastically changed, as there now must be a smooth transition between the middle and top loops.
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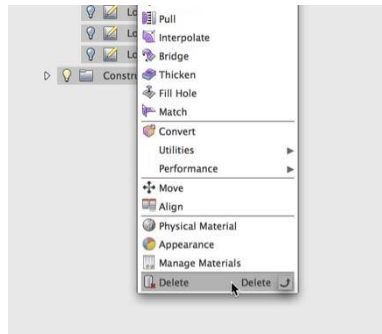


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</table>

There are many more commands for adding addition detail to a T-Spline form, but knowing how to insert and delete edges is at the core of sculpting in Fusion 360.

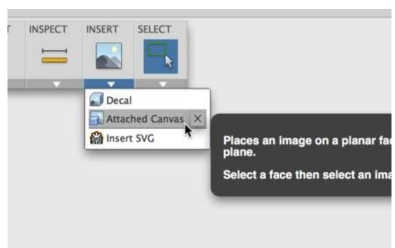
17.Create a T-Spline Form from a Reference Image

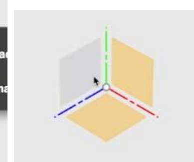
In this section you learn how to create a T-Spline form based on a calibrated reference image. With the freeform capabilities that come along with sculpting Fusion 360, this is a very common workflow. Let's get started by creating with a blank slate by creating a **New Design**.

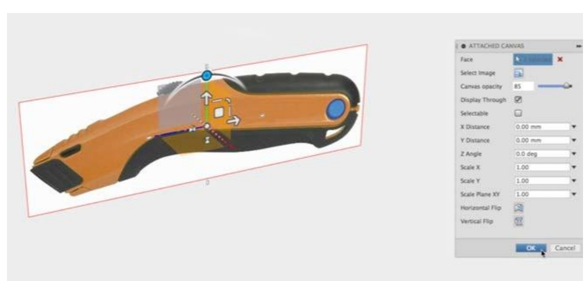
Design Setup – Attach Canvas: The first step is to learn how to attach a reference image to a particular plane and calibrate it to ensure an accurate design.

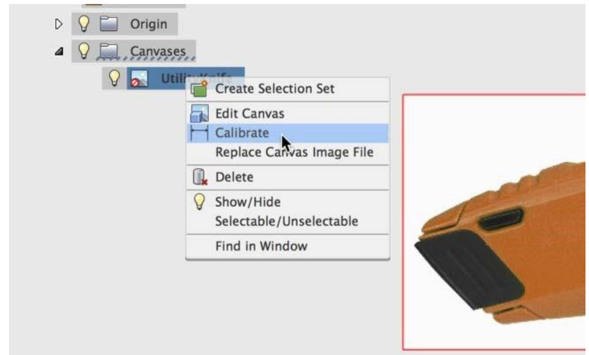
<table>
<tr>
<td>Step 1 – Attach a canvas

1. Click Insert > Attached Canvas.
2. Select the YZ Plane (between the green and blue axes) to set the Canvas' orientation.
3. In the dialog window, click the Select Image button and navigate to the

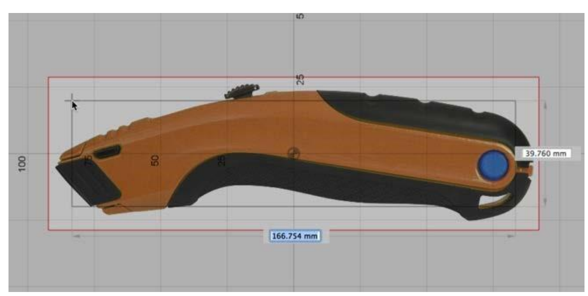
03_UtilityKnife.jpg file in the downloaded .zip folder.</td>
<td>

</td>
</tr>
<tr>
<td>Step 2 – Setup the canvas

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

Step 3 – Start the calibrate command 1. We need to calibrate our canvas to make sure our utility knife fits in our palm, and not on our fingertip! 2. In the Browser, click the drop-down arrow next to the **Canvases** folder. 3. Right-click on **UtilityKnife** and select **Calibrate**.	
Step 4 – Calibrate the canvas 1. Click **Right** on the ViewCube to view the utility knife from the side. 2. Click once at the **front** of the utility knife. 3. Click once at the **back** of the utility knife. 4. Enter the approximate length, **180 mm**. 5. The canvas will scale up accordingly.	

Create Primitive Form: With the canvas in place, the next step is to create a T-Spline primitive form on the proper plane.

Step 1 – Start the 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 to draw its 2D profile.	

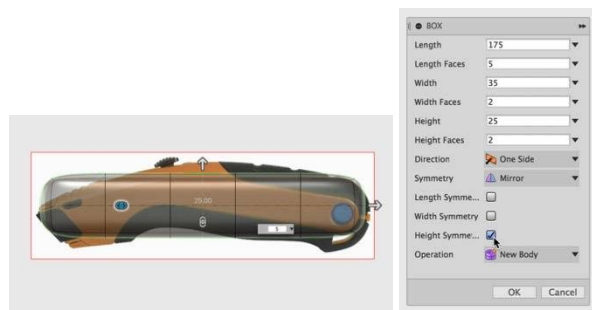
Step 2 – Specify the box primitive

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**.

Looking at the form of the utility knife, the complexity is along the length of the knife, so we set additional faces in that direction. We can always add or remove these later on.

Step 3 – Add symmetry

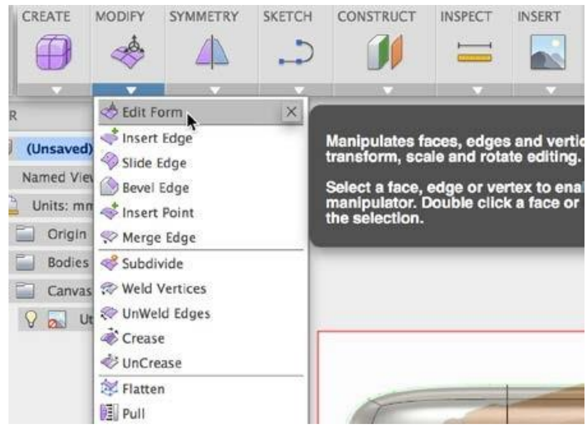
1. In the dialog window, change the **Symmetry** from None to **Mirror**.
2. Check the box for **Height Symmetry**.
3. A green line is displayed that indicates where we have symmetry set up.
4. With no more symmetry to add to our form, click **OK**.

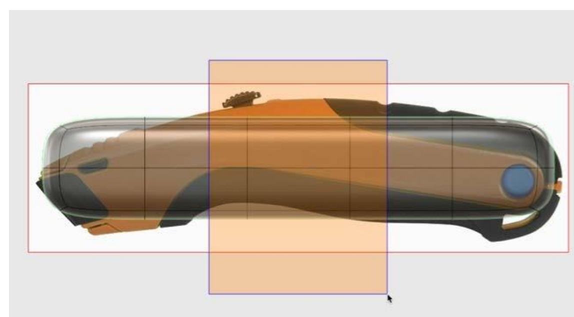


Edit the T-Spline Form: Our T-Spline primitive is now in place, but we need to edit its geometry to better match our reference picture. In the following steps, we'll edit our existing geometry to match the canvas as best as we can. After that, we can add and subtract more edges to fine-tune our design.

<table>
<tr>
<td>Step 1 – Start the Edit Form command

1. Click Modify > Edit Form.
2. For simplicity, ensure you're looking at the form from the Right view. To set this, you can click Right on the view cube.

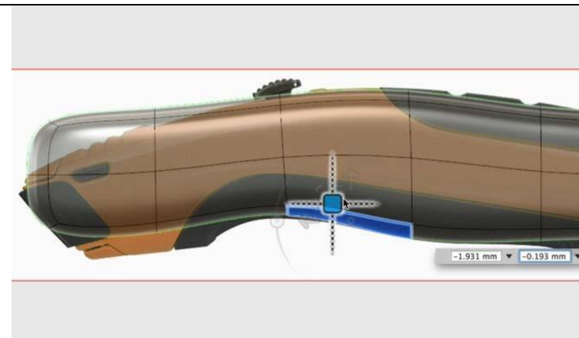
Common to most workflows, we stick to modifying our form from just one view, proceeding to 3D manipulations as a final step.</td>
<td>
</td>
</tr>
<tr>
<td>Step 2 – Select a loop of faces

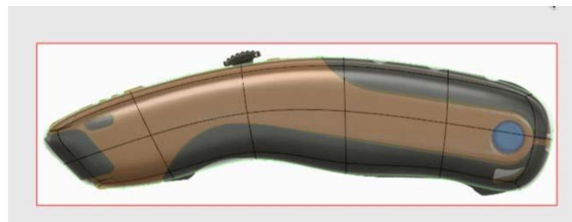
1. Select the middle loop of faces going down the length of the utility knife with a window selection (left-click and hold), as shown in the picture.
2. Dragging left to right will select all the geometry that is fully captured by the window, while right to left will capture everything that touches the window.</td>
<td></td>
</tr>
</table>

Step 3 – Start modifying the form

1. 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.
2. To align the bottom in this section, select the bottom face and use the planar translation and rotate manipulators.
3. 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.
4. For more controlled editing, try modifying individual edges.

   Don't worry if the finer details of the contour are not matched, as we will accomplish this in the next step by adding and subtracting edges. Shoot for the image on the right!



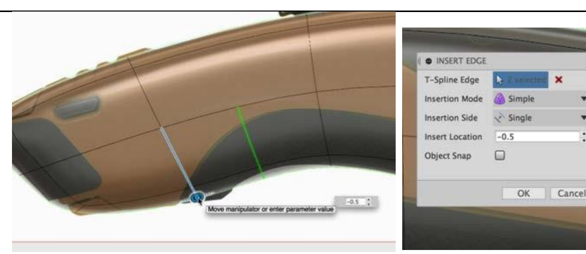


Add Additional Details – Insert Edge: Our T-Spline form is starting to resemble our reference image, but there are without a doubt some details that we need to add! As the form is right now, there simply are not enough edges available to capture all of the details that we would like. To combat this, you implement the earlier-discussed method of inserting and deleting edges into a T-Spline form.

Step 1 – Insert additional edges

1. Hold **Shift** then select the edges shown.
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**.

We'll now repeat the previous steps to insert another set of edges.



<table>
<tr>
<td>
Step 2 – Insert more edges

1. Holding **Shift** then select the edges shown.
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**.

The form will change due to the additional edges.

In the next step we will edit our form to our liking.
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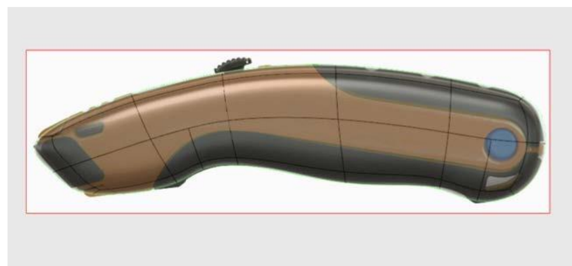
<tr>
<td>
Step 3 – Edit Form

1. Use the **Edit Form** command to manipulate the recently inserted edges to achieve the result shown on the right.
2. The planar translation manipulator will be extremely useful.

As you can see, the reference image will help us roughly capture the correct form, but the fine details are entirely up to us.
</td>
<td>

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</table>

Add Additional Details – Insert Point: To insert the final two edges we need, we'll actually learn a new command, the Insert Point command. Slightly different from Insert Edge, the Insert Point command will easily insert an edge by connecting two points together.

Step 1 – Start the Insert Point command

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.

Step 2 – Insert Point details

1. Leave the **Insert Mode** set to **Simple**.
2. Click **OK**.

An Insert Mode of **Simple** will add the desired edge, but the form will change slightly. An Insert Mode of **Exact** will add the desired edge, as well as *additional edges* to maintain the previous form. As

you can tell, this setting will be a trade-off between maintaining form and reducing the number of edges.

Step 3 – Repeat Insert Point

1. Click the **Modify > Insert Point**.
2. Construct an additional edge as shown to the right.
3. Leave the **Insert Mode** set to **Simple**.
4. Click **OK**.

Step 4 – Edit Form

1. Use the **Edit Form** command to manipulate the recently inserted edges (as well as the surrounding geometry) to achieve the result shown on the right.
2. Click **Finish Form**.

Congratulations! You've completed your first sculpting workflow in Fusion 360. Feel free to sculpt the utility knife's sides for even more detail. With a more complex form, a similar workflow can be utilized, but with *multiple* calibrated canvases.

Solid Modeling

Overview

Modeling techniques in Fusion 360

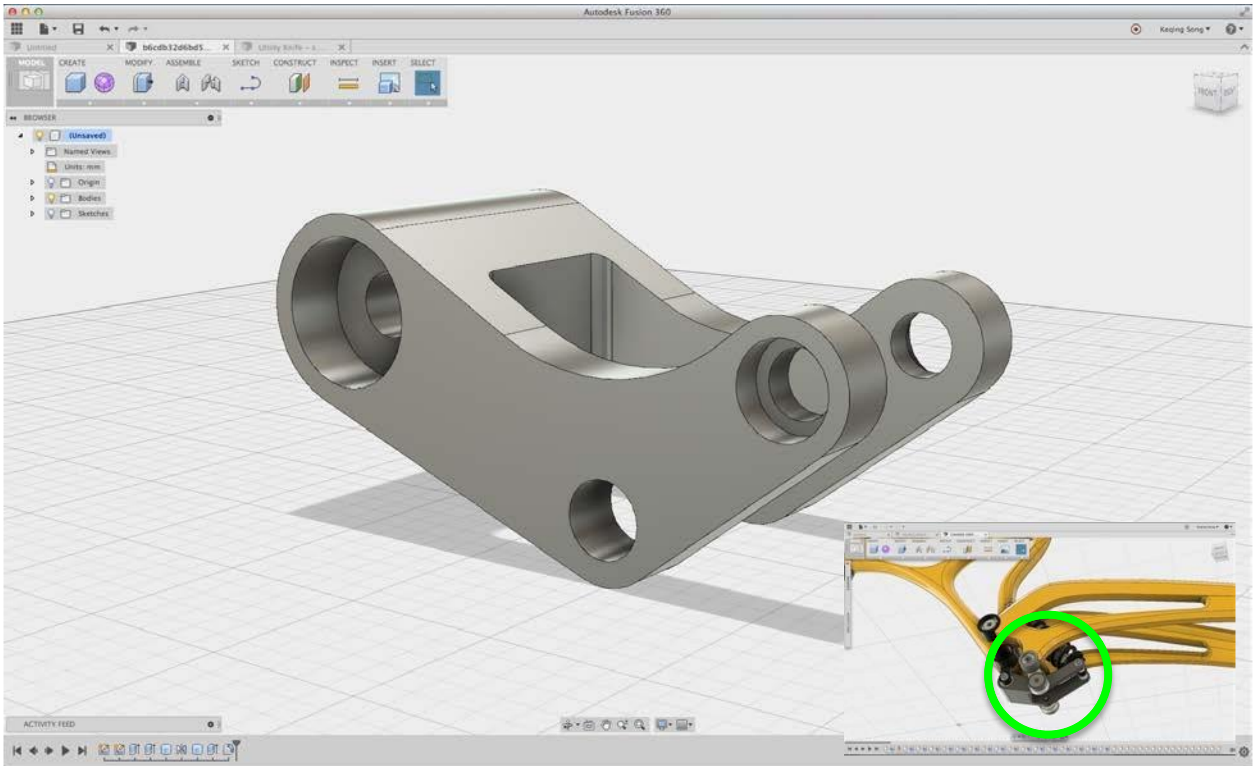
Modeling in Fusion 360 is quite a different experience from how you would model in conventional history-based CAD software. Some users have expressed that it is a different mindset, but once they get it, it makes so much more sense to them. Modeling in Fusion 360 is essentially a series of workflows that include a whole bunch of different commands, and when they're used together, it makes the experience faster, easier, and more intuitive. In many cases, bodies, sketches, and planes in Fusion 360 can be used not only to help create additional geometry, but also help subtract geometry. In this module, you are introduced to this mindset.

Learning Objectives

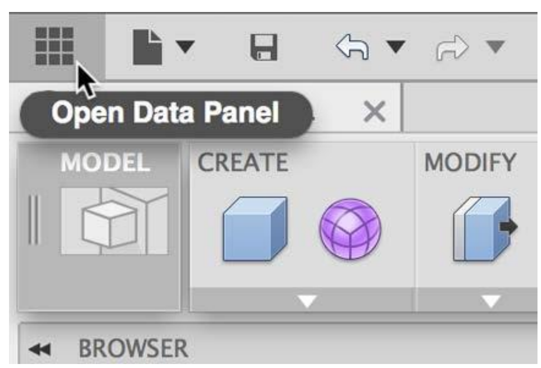
In this section you will learn how to:

- Create a new design in the model workspace
- Create bodies
- Modify your design
- Add features to a sculpted body

Bicycle rocker arm



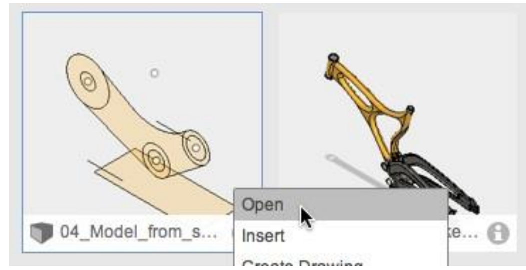
Before moving on, make sure you have uploaded **04_Model_from_sketch** design to your A360 site. If **Open Fusion 360 design file**: In this section you will open the introductory design file.

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 **04_Model_from_sketch.*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 **04_Model_from_sketch.f3d** file.
2. Navigate to this design and either **doubleclick** 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.



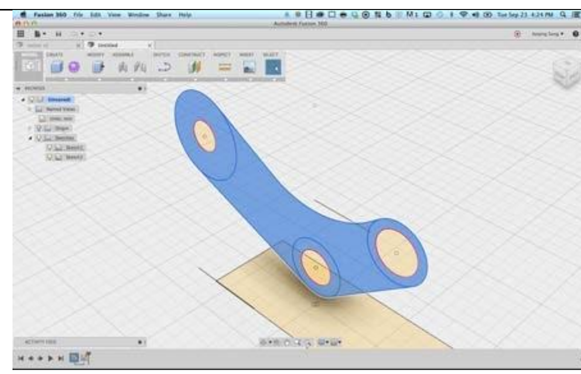
18.Create solid body

Let's start with this sketch of the rocker arm. We're going to use this to create a solid body.

Step 1 – Select profiles

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.

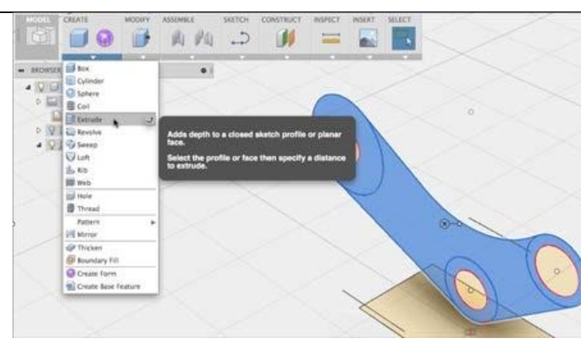
   Note: If you are having trouble selecting certain profiles, zoom in closer and that should make it easier to select.

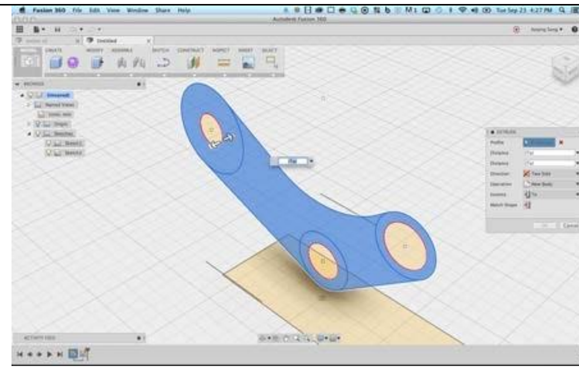


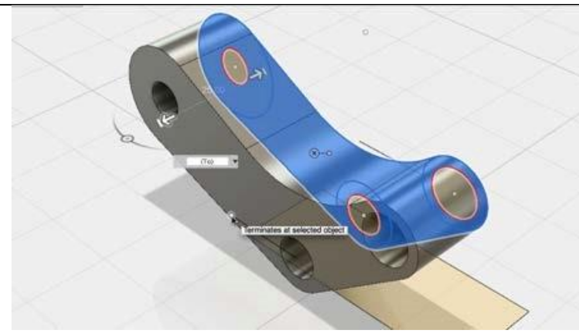
Step 2 – Start the Extrude command

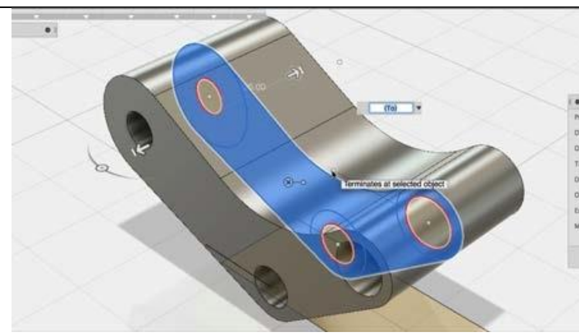
1. Click **Create > Extrude**.

We're going to extrude the selected profiles.

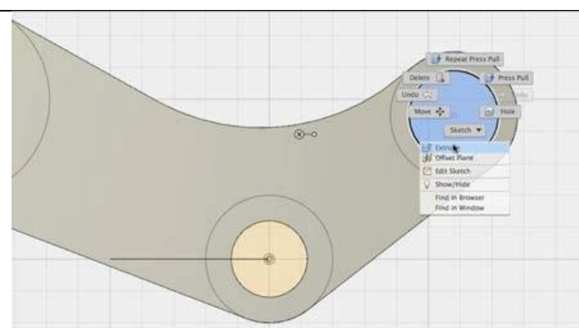


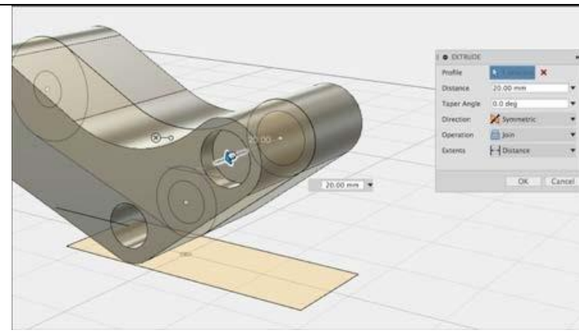
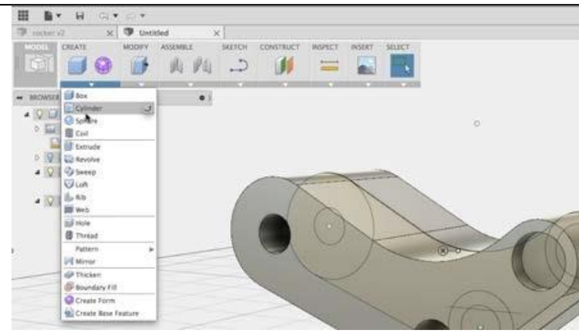
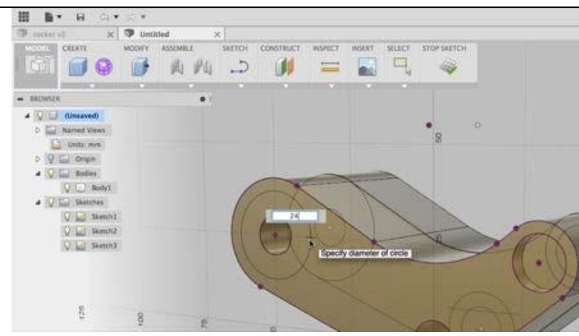
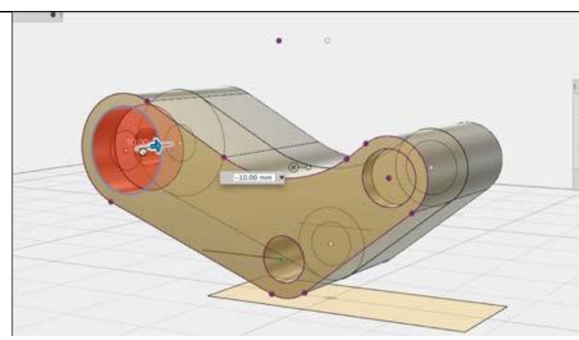
<table>
<tr>
<td>Step 3 – Set the extrude options
1. Set Direction to Two Side.
2. Set Extents to To.</td>
<td></td>
</tr>
<tr>
<td>Step 4 – Set the distance for the left side
1. Click once on the left arrow manipulator
2. Now hover over the line sketch on the left side and click on the end point.

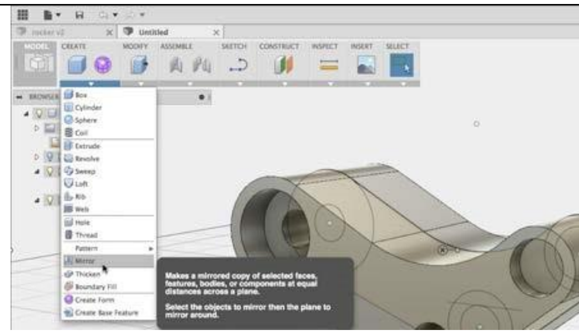
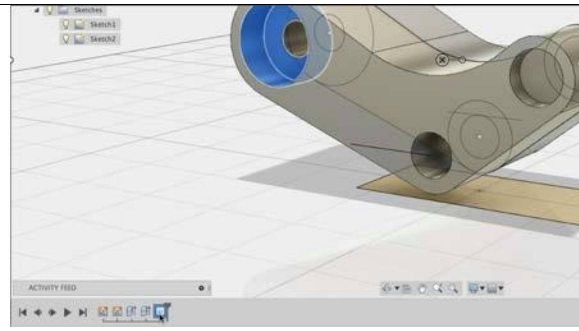
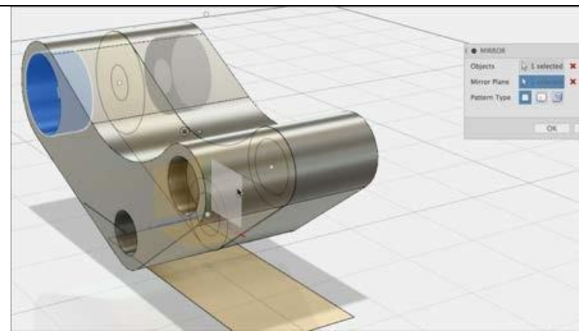
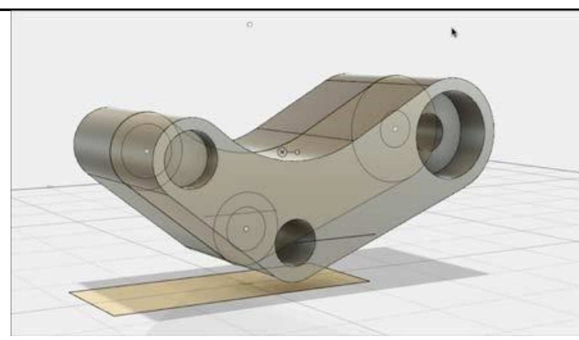
Note: Make sure you select the line sketch and not the rectangle sketch. When you’ve done this, the extrusion will automatically terminate at that point, hence why we selected the Extents as To.</td>
<td>
</td>
</tr>
<tr>
<td>Step 5 – Set the distance for the right side
1. Repeat Step 4, but now for the right side.
2. Click OK to finish the extrusion.
You now should have the basic shape of the rocker arm.

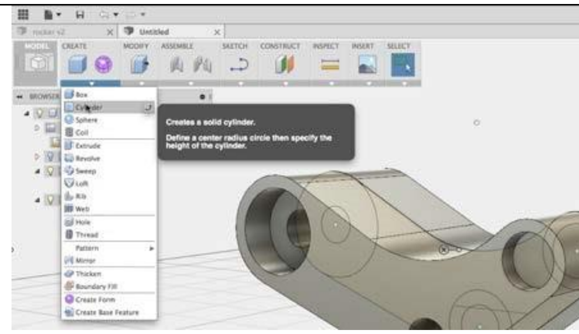
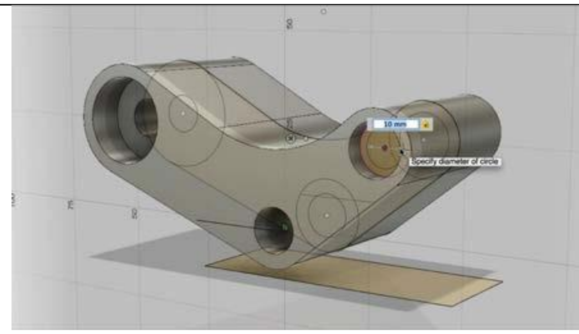
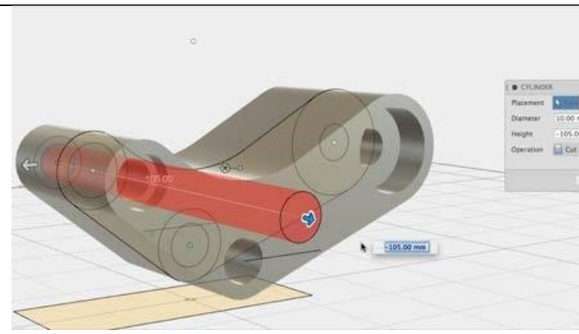
Note: Line sketches can be used for a variety of different tasks, such as reference lines for other tasks, as well as creating geometry.</td>
<td>
</td>
</tr>
</table>

Cut holes: In this section you use the sketch profiles to cut holes in the body.

<table>
<tr>
<td>Step 1 – Start the Extrude command
1. Go to the browser and click the light bulb next to Sketches to turn the visibility on.
2. Go to the ViewCube and select the FRONT view.
3. Click on the right-most circle sketch profile so that it is selected.
4. Right-click and select the Extrude command.</td>
<td>
</td>
</tr>
</table>

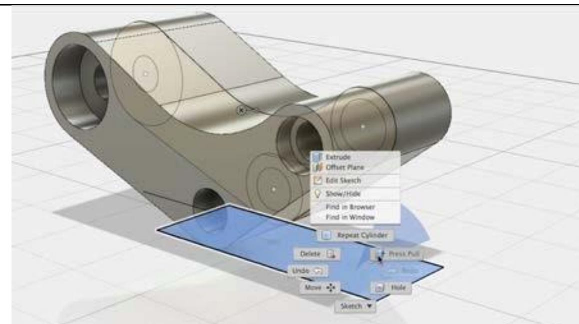
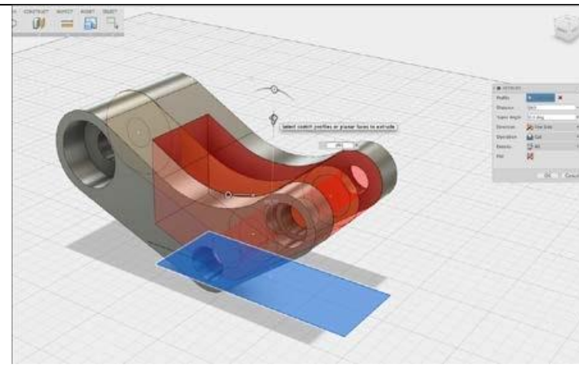
Step 2 – Set the extrude options 1. Set Direction to **Symmetric** 2. Set Operation to **Join** 3. Set Extents to **Distance** 4. Use the arrow manipulator and drag the arrow out to **20.00 mm**. 5. Click **OK** to finish.	
Step 3 – Start the Cylinder command 1. Click Create > Cylinder. We're going to use the **Cylinder** command to cut a counter-bore for the hole on the far left.	
Step 4 – Define the cylinder 1. Click the outer most surface to place your cylinder. 2. Hover over the left circle sketch profile until you see a small blue circle snap on the center point of the circle sketch. 3. Click once and move the cursor outward until you reach **24 mm**. 4. Click one more time to set the diameter. Note: You can also enter the value and then hit **Enter** twice.	
Step 5 – Set the cut distance 1. Use arrow manipulator and drag it inward to **– 10 mm**. 2. Click **OK** to finish the cut. Note: This naturally became a cut because the software recognized that the cylinder body is intersecting with an existing body, thus assumed that you wanted a cut.	

Step 6 – Start the mirror command Now let's mirror this cut on the other side. 1. Click **Create > Mirror**.	
Step 7 – Select the operation to mirror 1. Go to the timeline at the bottom and select the **cylinder operation** we just created as the feature to mirror.	
Step 8 – Select the mirror plane 1. Go to the browser and click the light bulb next to Origin to turn the visibility on. 2. In the dialog box, make sure **Mirror Plane** is selected. 3. Select the plane that is in the **middle of the rocker body** as the mirror plane. Note: If you're having trouble selecting the plane, hover over it, click and hold the click. A dialog will display and allow you to choose what you want to select.	
Notice that the other side has been successfully mirrored to have the same counter bore hole.	

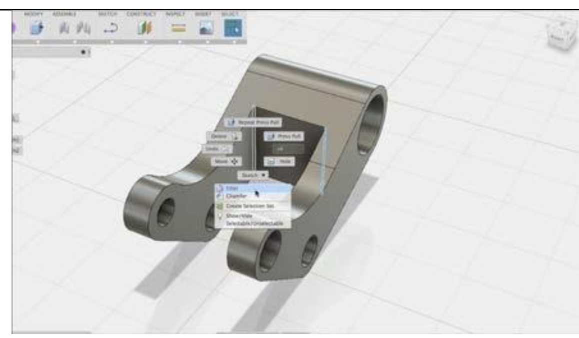
Step 9 – Start the Cylinder command Now let’s punch a hole through the far right circular cut. 1. Click **Create > Cylinder**. Note: The Cylinder command is one of many versatile tools where it can be used for a number of tasks – new bodies as well as Boolean cuts.	
Step 10 – Set the diameter of the cylinder 1. Place the cylinder at the **center point** of the inner circle. 2. Click once to confirm the placement of the cylinder. 3. Move the cursor outward until you reach **10 mm**. Click once to confirm the size. Note: You can also enter the value and then hit **Enter** twice.	
Step 11 1. Use the arrow manipulator and drag it across to the other side. Don’t worry about the depth of the cut, as long as it is through the entire body. 2. Click **OK** to finish.	

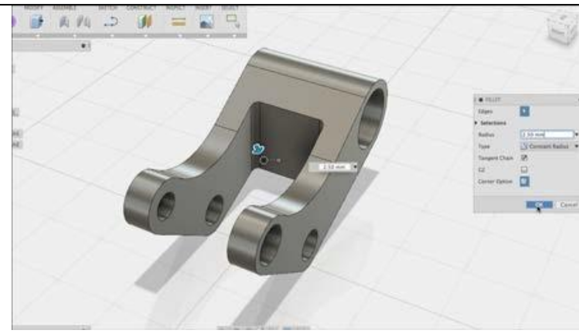
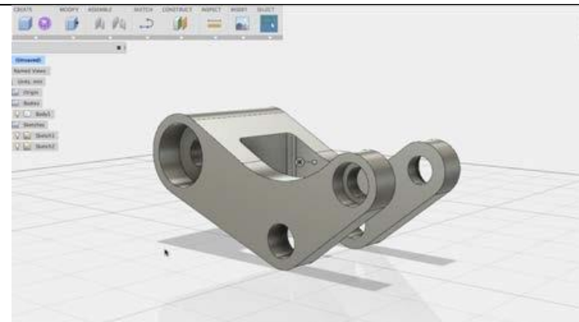
19.Remove geometry for a slot

In this section you use a sketch to cut material from the body, creating a slot.

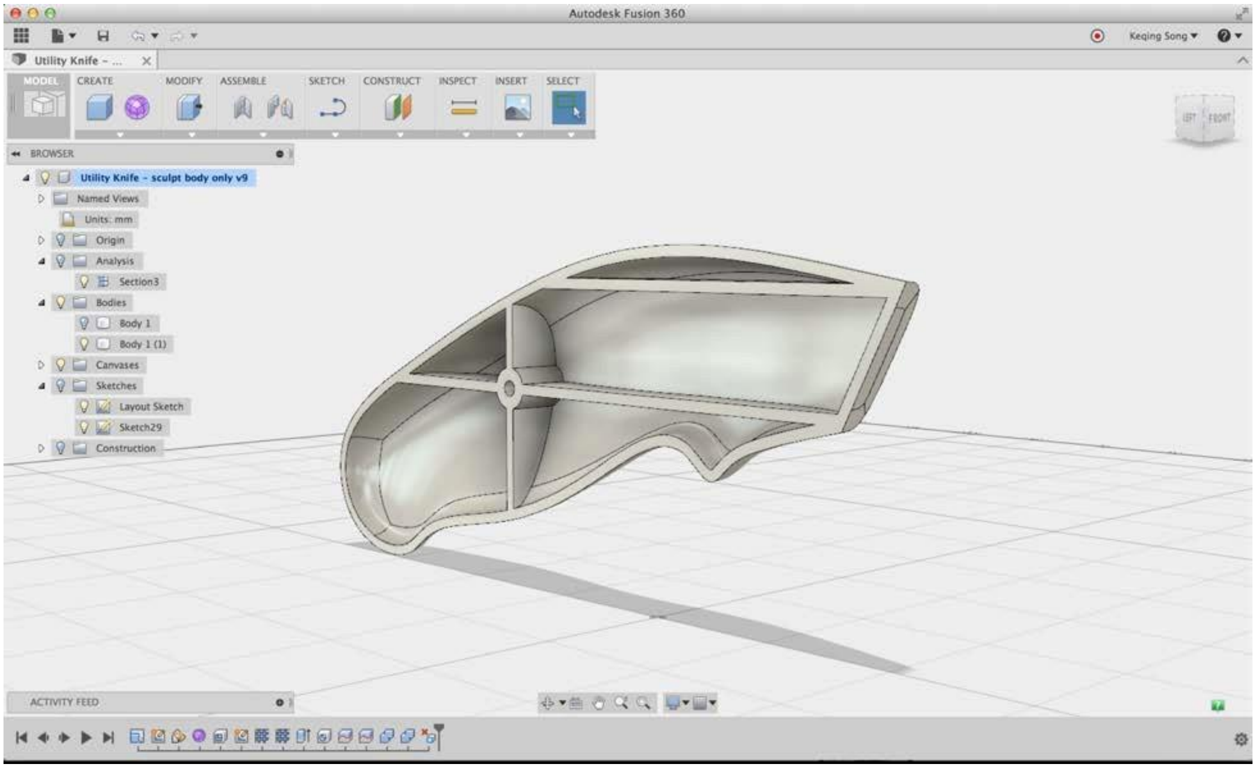
Step 1 – Start Extrude using Press Pull 1. We’re now going to use the rectangle sketch to cut the arms out. Select the rectangular sketch. 2. Right-click and select **Press Pull**. Note: Press Pull is similar to Extrude, but it is somewhat of a hybrid command, where it is aware of what you want to Press Pull, and will turn into the appropriate command for that task.	
Step 2 – Set the extrude options 1. Set Operation to **Cut**. 2. Set Extents to **All**. 3. Click **OK** to finish the command. This will use the rectangle sketch profile and cut the rocker all the way through.	

Fillet sharp edges: Now we finish off the design by adding fillets to round off sharp corners.

Step 1 – Start the Fillet command We’re now going to add a couple fillets on the inner edges of the rocker arm. 1. Hold the **Shift** key and select the two edges shown in the image. 2. Right-click and select **Fillet**.	

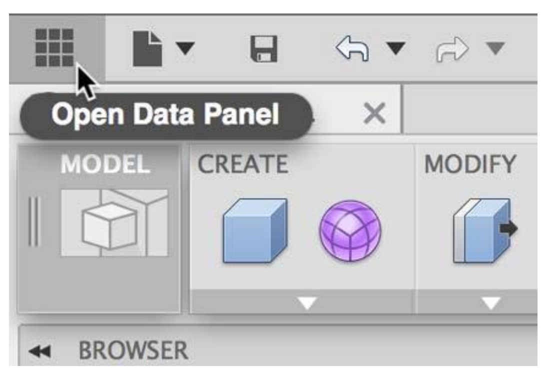
Step 2 – Set the fillet radius 1. Use the arrow manipulator and drag it to **2.50 mm**. 2. Click **OK** to finish.	
Good job! We've successfully modeled the rocker arm from a sketch. You're now ready to move onto the next part.	

20.Model from a Sculpted body



In this lesson we're going to work on a sculpted utility knife handle. We're going to look at how to create mechanical features based on a sculpted body. We'll be using tools that we used in the previous lesson, as well as learn some new ones.

Before moving on, make sure you have the **04_model_from_sculpted_body** design open and in your design environment.

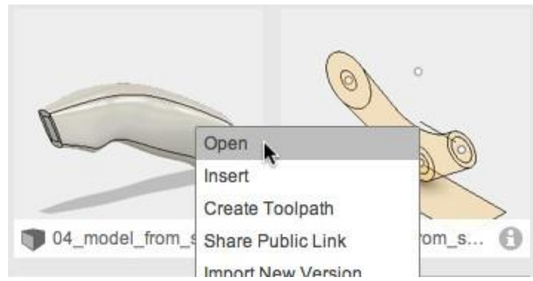
Step 1 – Open the Data Panel 1. Open the Data Panel by clicking on the icon located at the top left of the menu bar. 2. The Data Panel will slide open.	

Step 2 – Open the design

In this module we will be using the

04_model_from_sculpted_body.*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 **04_Model_from_sculpted_body.f3d** file.
2. Navigate to this design and either **doubleclick** 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.

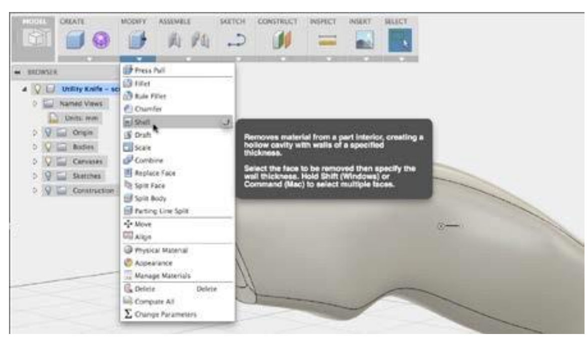


Shell a sculpted body: In this section you hollow out a body using the shell command.

Step 1 – Start the Shell command

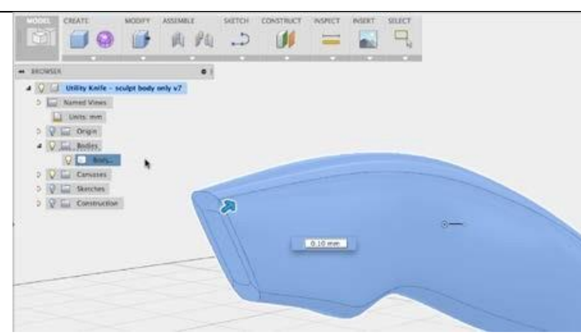
Let's start by first shelling the body.

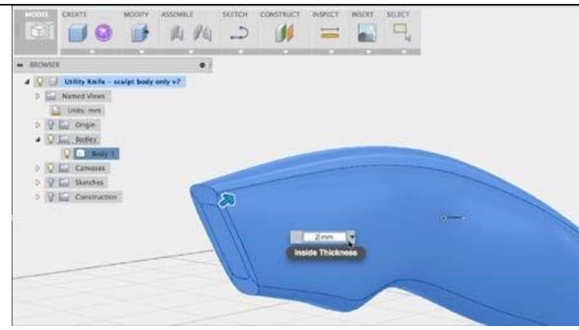
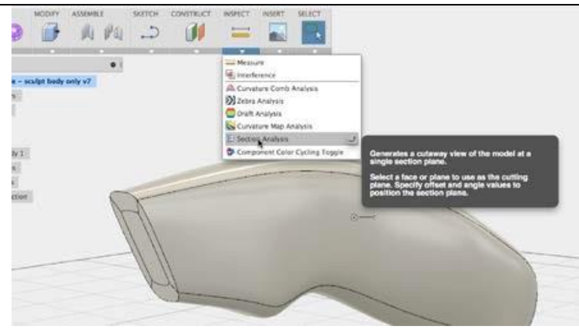
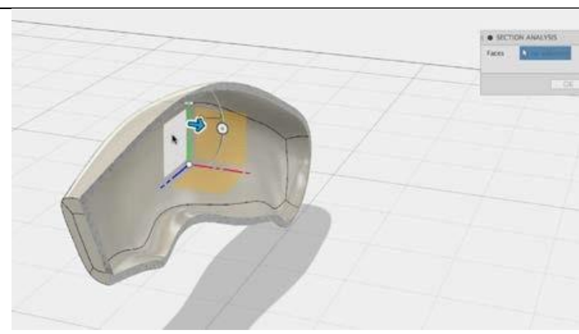
1. Click **Modify > Shell**.



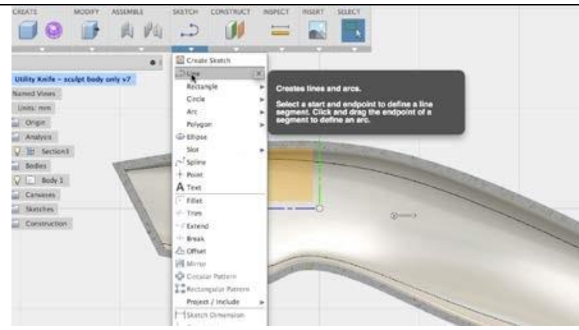
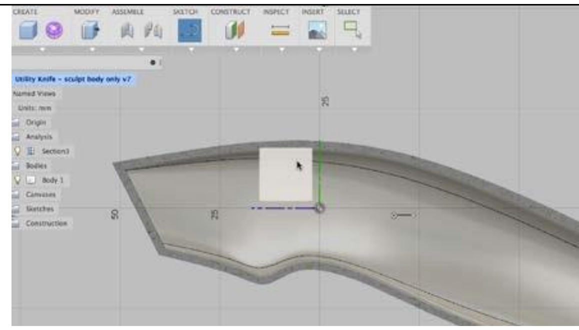
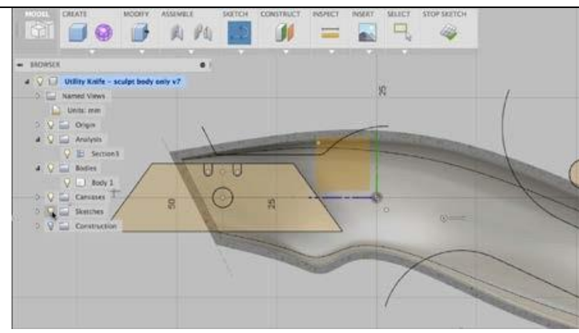
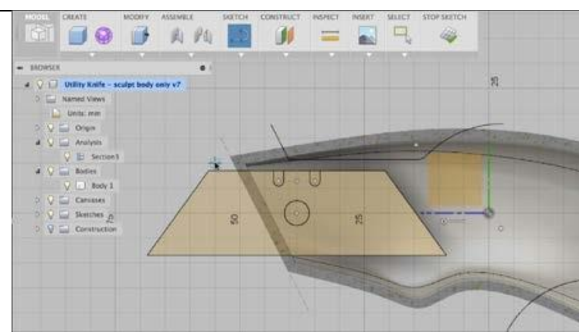
Step 2 – Select the body

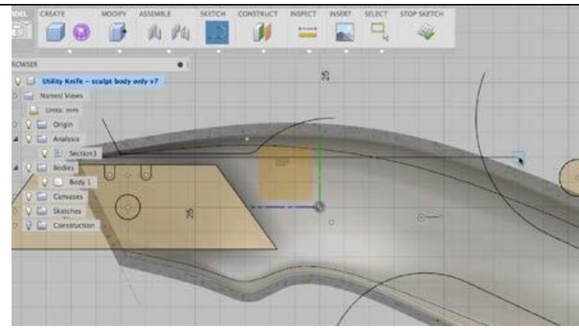
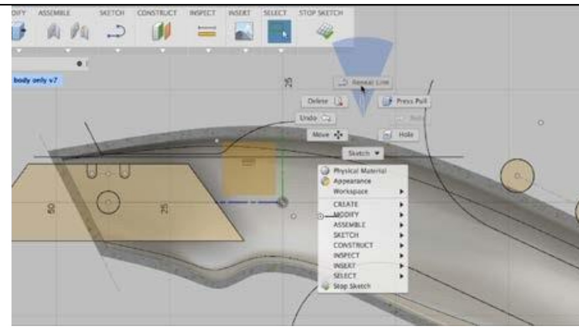
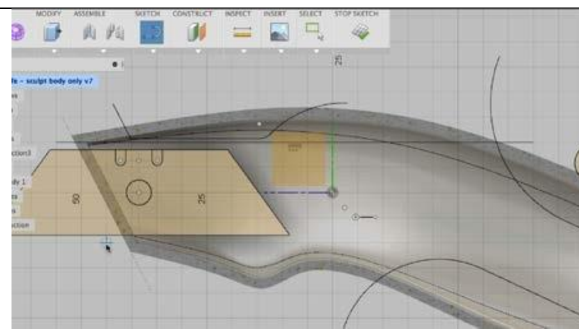
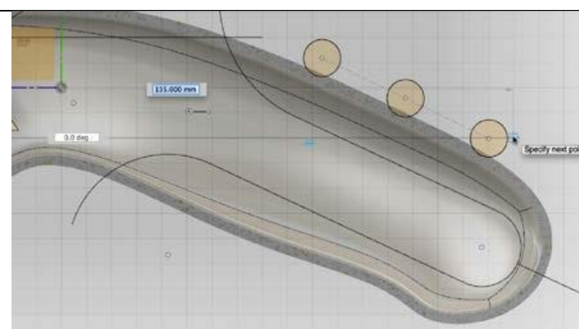
1. Go to the browser and locate the knife body.
2. Click it once to select it as the body to shell.

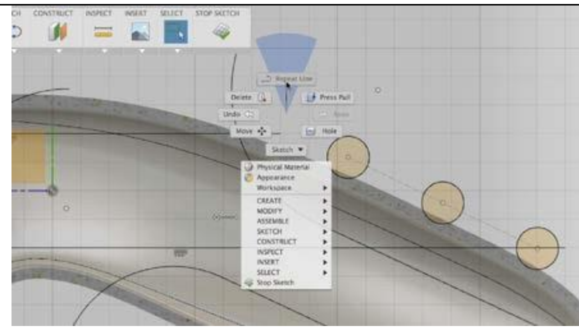
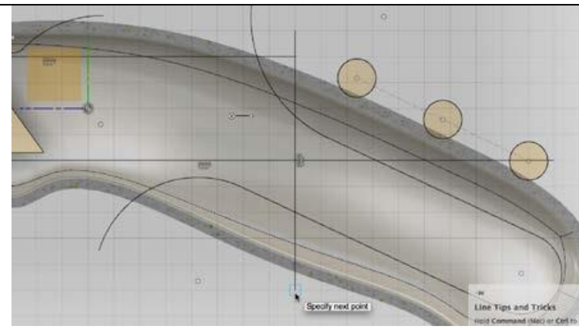


Step 3 – Set the shell thickness 1. Instead of using the arrow manipulator, go to the floating command dialog and change the value to **2 mm**. 2. Press **Enter** to finish the command. Now that the body is shelled, we're going to begin creating features on the inside.	
Step 4 – Create a section view 1. To see the inside, click **Inspect > Section Analysis**. Note: Section Analysis let's you see a section of the model based on a reference plane or surface. It does not affect the geometry of the body.	
Step 5 – Select the section plane 1. Go to the browser and turn on the **Origin** planes. 2. Select the plane that is in the middle of the knife body, parallel to the length of the model. Click **OK** to confirm. You should see half of the shelled model.	

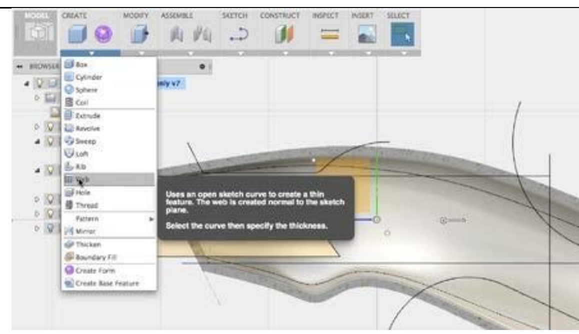
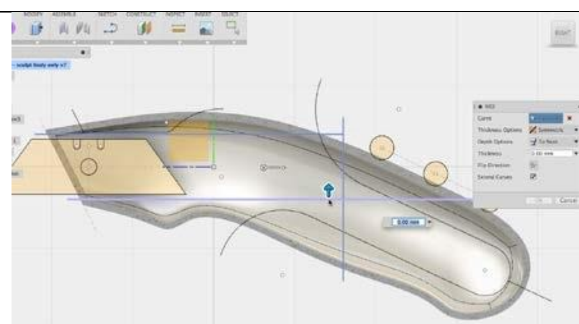
Create a sketch: In this section you create a 2D sketch that will be used for web features.

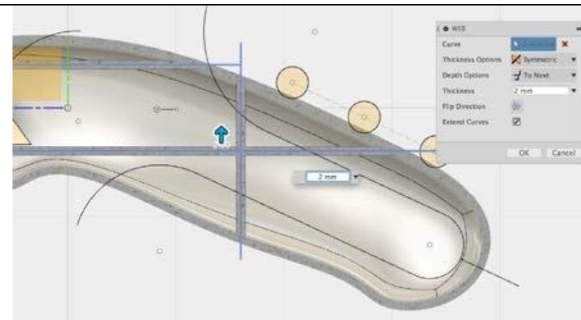
Step 1 – Create a sketch We're now going to set up some web features. 1. Go to the **RIGHT** view on the ViewCube. 2. Click **Sketch > Line**.	
Step 2 – Select the sketch plane 1. Choose the plane that is parallel to the section view.	
Step 3 – Turn on the visibility of sketches 1. Go to the browser and turn on **Sketches**. You should see some profile sketches that have been already created. We're going to use them as references for our line sketch.	
Step 4 – Pick the start point of the line 1. Hover over to the top of the blade profile until your cursor snaps to a grid intersection. Make sure that it is outside the knife body. 2. Click once to place the start of your line. If you're having trouble snapping to the grid intersection, zoom in and the grid will scale accordingly.	

Step 5 – Pick the end point of the line 1. Stretch line across the knife body until it is on the other side. 2. Click to place the end point of the line. 3. Press **ESC** key to end the command.	
Step 6 – Repeat the line command 1. Right click and select **Repeat Line** to reuse the Line command.	
Step 7 – Pick the start point 1. Now repeat the same task at the bottom of the blade sketch profile. Snap to a grid intersection. 2. Click to place the starting point of the line.	
Step 8 – Pick the end point 1. Extend the line across the knife body until it reaches past the last circle sketch profile. Snap to a grid intersection 2. Click to place the end point of the line. 3. Press **Esc** to end the command.	

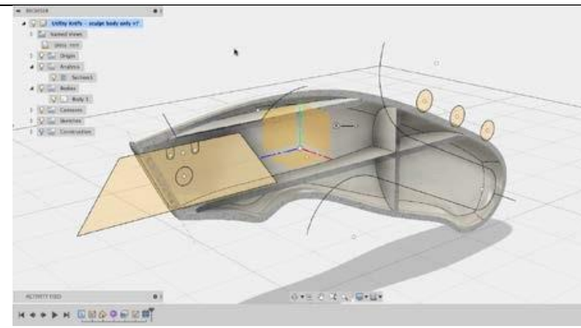
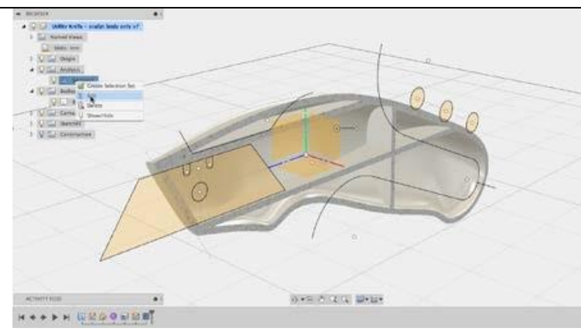
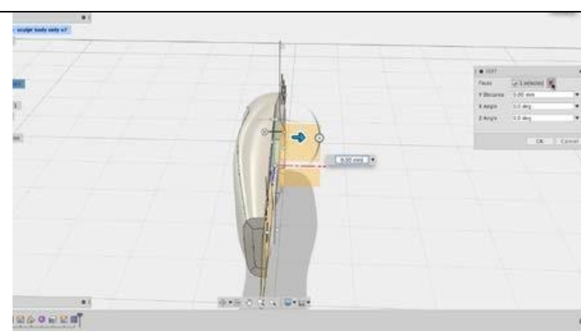
Step 9 – Repeat the line command 1. Right click and select **Repeat Line** to reuse the Line command.	
Step 10 – Pick the start point and end point 1. Draw a line that is perpendicular to the first and second line, making sure that it also extends past the knife body. 2. When you're done placing the end point of the 3rd line, press **STOP SKETCH** to end sketching.	

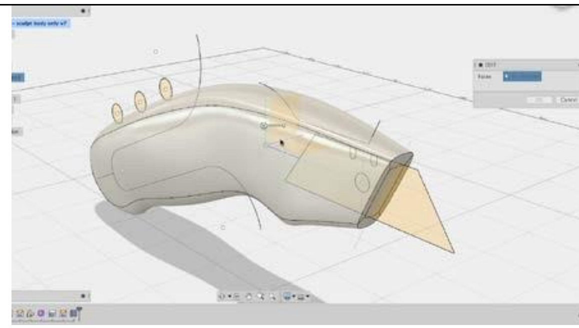
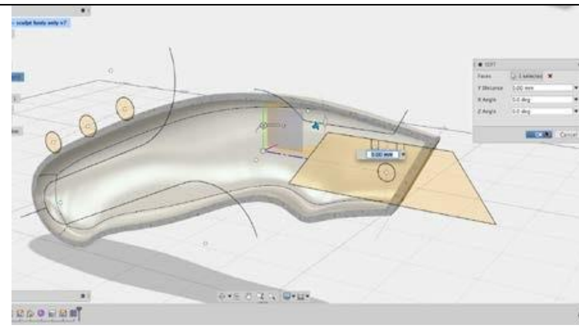
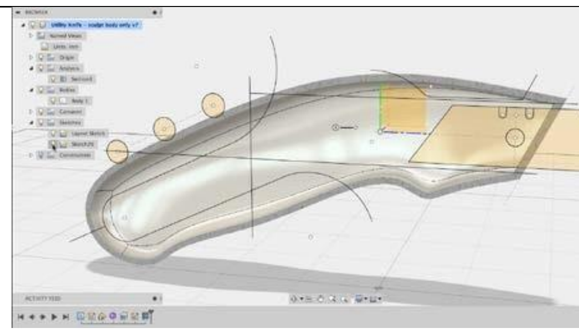
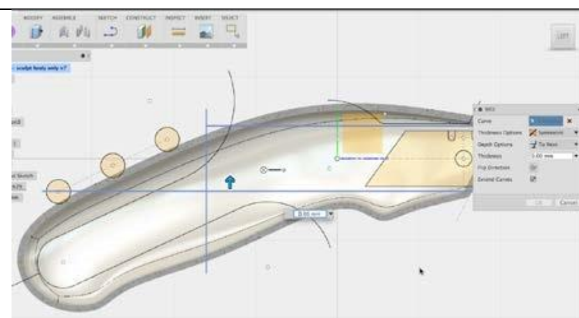
Create a web: In this section you create a strengthening web from a 2D sketch.

Step 1 – Start the Web command We're now ready to model the webs. 1. Click **Create > Web**.	
Step 2 – Select the web profiles 1. Click on the 3 line sketches you just made so that they are selected as the reference lines for your web.	

Step 3 – Set the web thickness 1. Change the thickness value to **2 mm**. 2. Press **Enter** to finish the command.	

Create another web: In this section you create a web on the opposite side of the design.

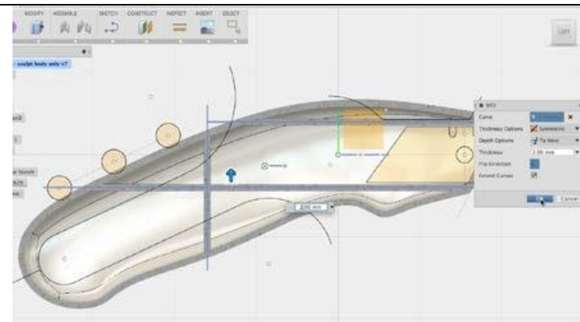
Step 1 – Find the section analysis in the browser Now that you've created webs for one side, you'll need to duplicate the task on the other. We're going to do this by editing the **Section Analysis.** 1. Go to the browser and expand the Analysis folder.	
Step 2 – Edit the section analysis 1. Right click on the analysis and select **Edit**.	
Step 3 – De-select the section plane 1. In the dialog box, click the red **X** to deselect previously selected plane.	

Step 4 – Select a new section plane 1. Rotate the model around to the Left side. 2. Select the left side of the middle plane.	
Step 5 – Finish the edit 1. You should be seeing the other half without the webs. Click **OK** to finish.	
Step 6 – Display the web sketch 1. Go to the browser and turn on the sketch for the web.	
Step 7 – Create another web 1. Click **Create > Web**. 2. Select the 3 line sketches. 3. Before setting a thickness, click **Flip Direction** in the dialog box so that the webs will be going in the right direction.	

Step 8 – Set the web thickness

1. Set the thickness to **2 mm**.
2. Click **OK** to finish.

You now have webs on both sides of the model.

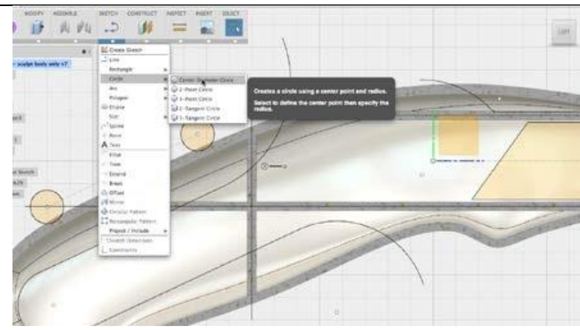


Create a boss: In this section you create a boss on the web features.

Step 1 – Sketch a circle

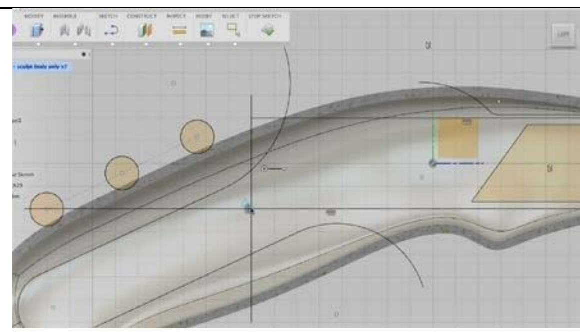
We're now going to create a boss hole right in the middle of where the webs intersect.

1. Click **Sketch > Circle> Center Diameter Circle**.



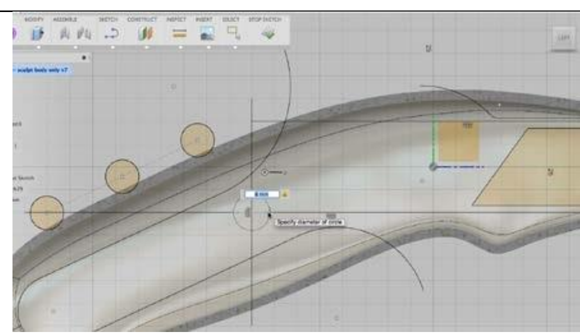
Step 2 – Select the sketch plane

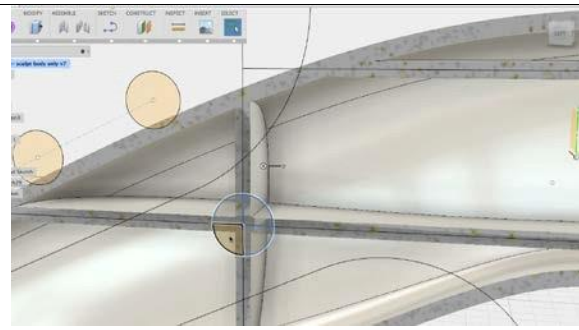
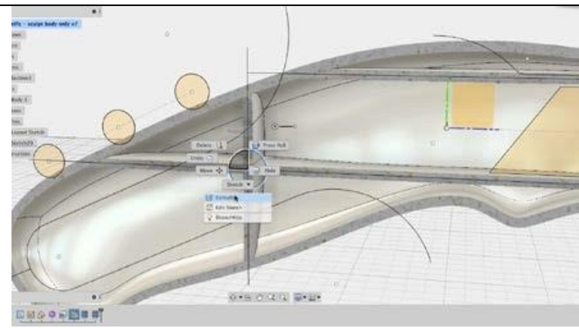
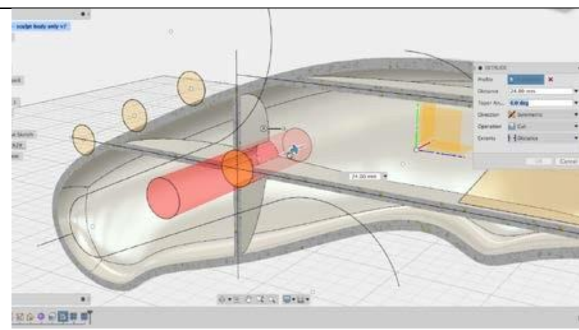
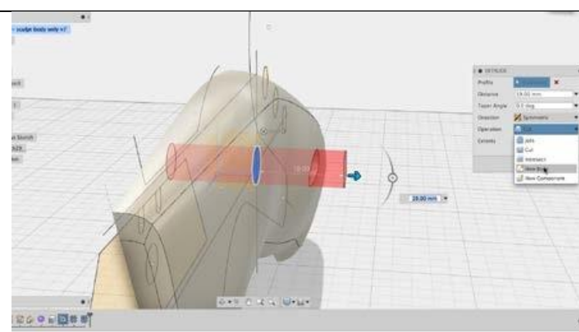
1. Click on the plane where the web line sketches were drawn
2. Hover the cursor at the intersection until the cursor snaps to the center. Click to place your circle there.

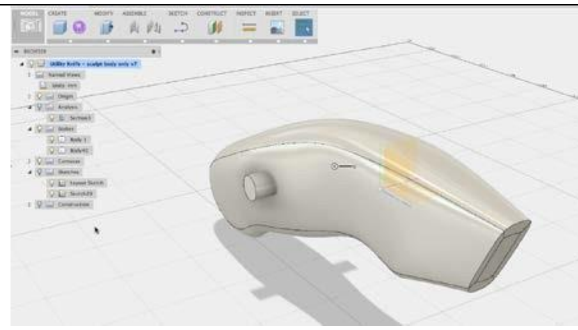
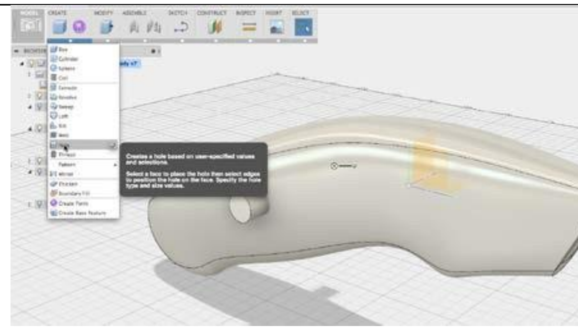
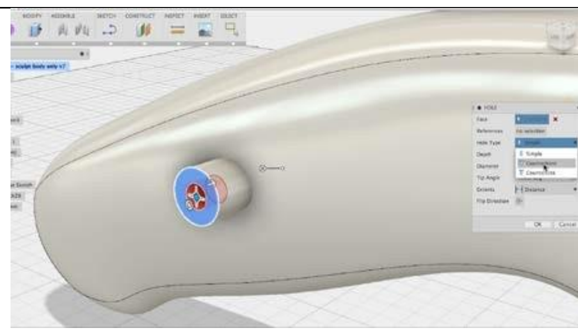


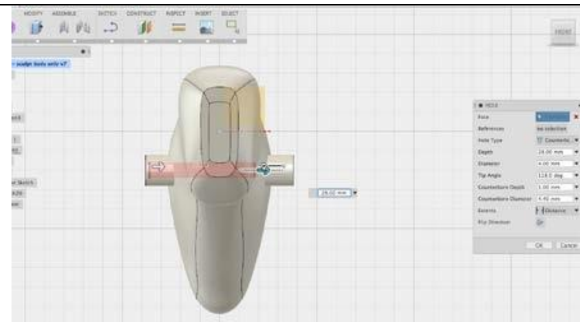
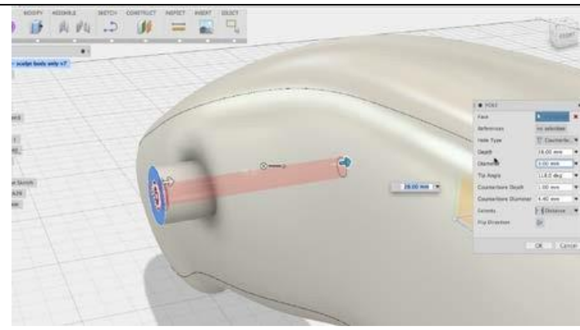
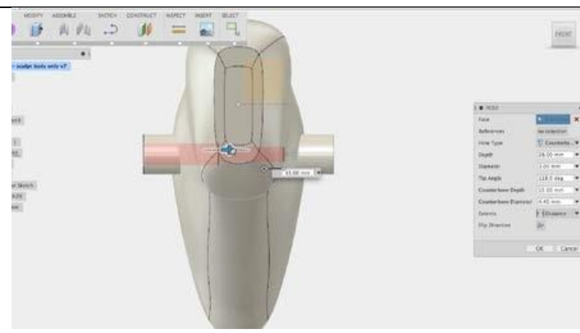
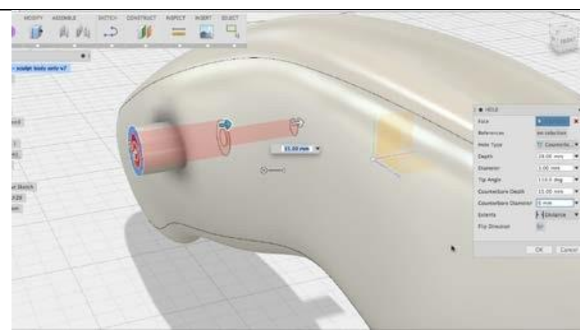
Step 3 – Finish the sketch

1. Enter a value of **8 mm** as the diameter.
2. Press **Enter** to confirm.
3. Click **STOP SKETCH** to exit the Sketch environment.



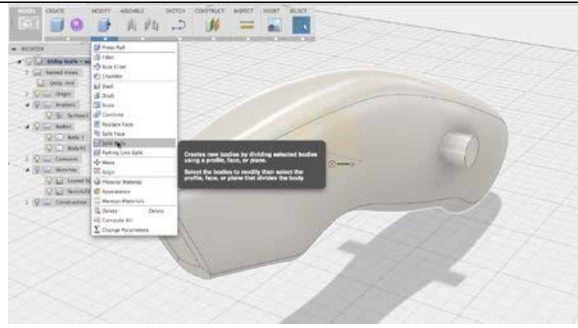
Step 4 – Select the sketch profile We're now going to create a cylinder using the circle sketch profile. 1. Select the entire circle profile. Hold the **Shift** key to add onto each selection.	
Step 5 – Start the Extrude command 1. Right click on the selected circle profile and select **Extrude**.	
Step 6 – Set extrude options 1. In the dialog box, set the Direction to **Symmetric**. 2. Drag the arrow manipulator to a value of **19 mm**.	
Step 7 – Set extrude options 1. Change the Operation from Cut to **New Body**. 2. Click **OK** to finish.	

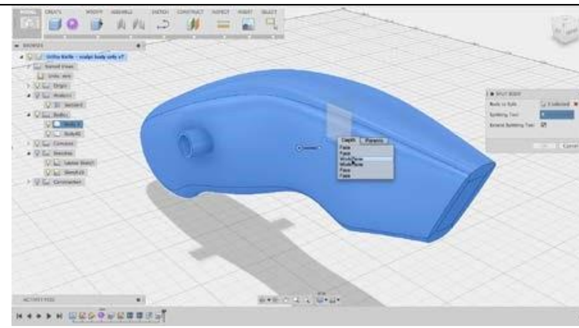
Step 8 – Turn off the analysis 1. In the browser, click the light bulb next to **Analysis** to turn off all analysis views. You should now see a cylinder body protruding from both sides of the knife body.	
Step 9 – Start the hole command We're now going to create a counter-bore hole through the cylinder. 1. Click **Create > Hole**.	
Step 10 – Position the view to select the center 1. Rotate around the model so you see the left side of the knife. 2. Hover over the surface of the cylinder body until you see a center point appear.	
Step 11 – Select the center and set the type 1. Click on the surface with the center-point visible. This will snap the hole to the center-point. 2. In the dialog box, change the Hole Type to **Counterbore**.	

Step 12 – Set the hole depth 1. Drag the arrow that determines the depth of the hole to **28 mm**.	
Step 13 – Set the diameter 1. In the dialog box, set the diameter of the hole to **3 mm**.	
Step 14 – Set the counterbore depth 1. Now drag the arrow that determines the depth of the counterbore to **15 mm**.	
Step 15 – Set the counterbore diameter 1. In the dialog box, change the counterbore diameter to **6 mm**. 2. Click **OK** to finish.	

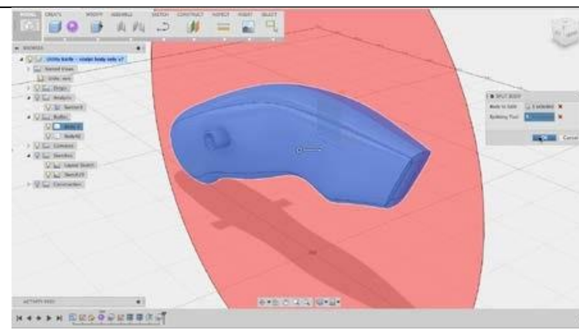
<table>
<tr>
<td>Step 16 – Start the Split Body command

We’re now ready to split the body into 2 halves.

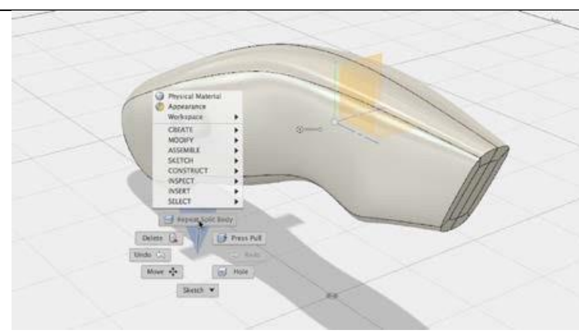
1. Click Modify > Split Body.</td>
<td></td>
</tr>
<tr>
<td>Step 17 – Select the body and split plane

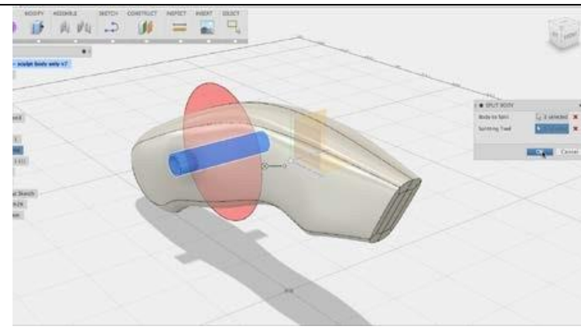
1. Click on the knife body as the Body to Split.
2. Hover over the middle plane, click and hold the click until a selection dialog is displayed.
3. Choose the first Work Plane as the Splitting Tool.</td>
<td></td>
</tr>
<tr>
<td>Step 18 – Finish the command

You should see this as a result.

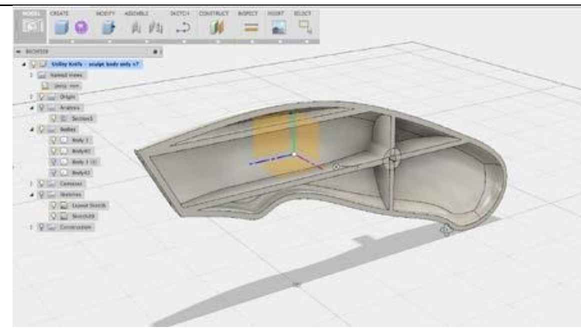
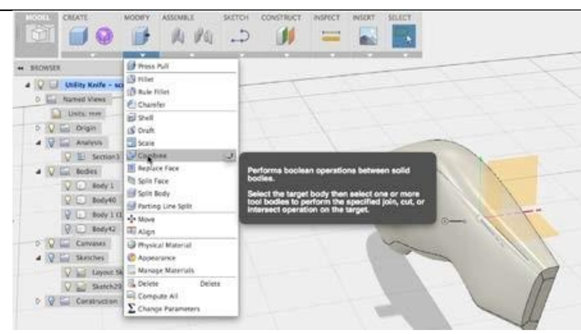
1. Click OK to finish.</td>
<td></td>
</tr>
<tr>
<td>Step 19 – Start the Split Body command

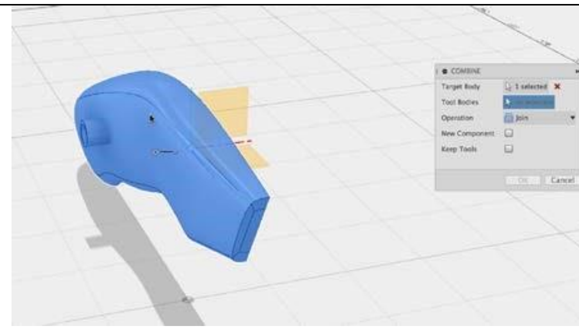
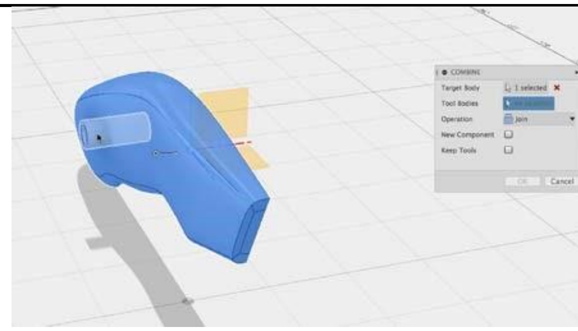
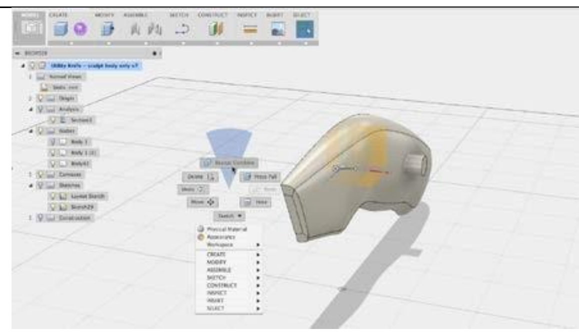
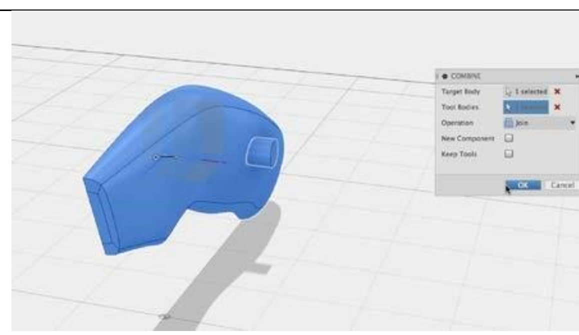
Now that we split the knife body into two pieces, let’s split the cylinder body as well.

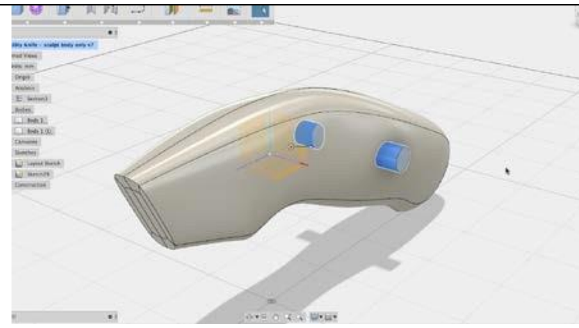
1. Right click and select Repeat Split Body.</td>
<td></td>
</tr>
</table>

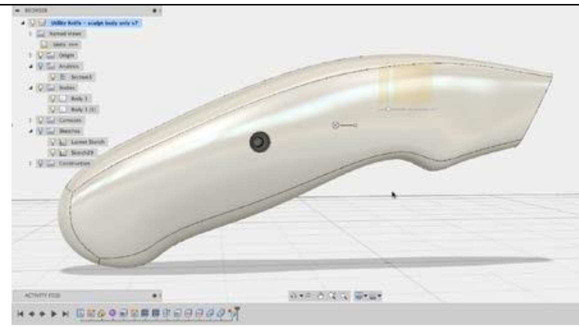
Step 20 – Select the body and split plane 1. Click on the cylinder body as the Body to Split. 2. Select the middle origin plane as the Splitting Tool. 3. Click **OK** to finish.	

Combine bodies: Now you combine bodies. The end result is two bodies.

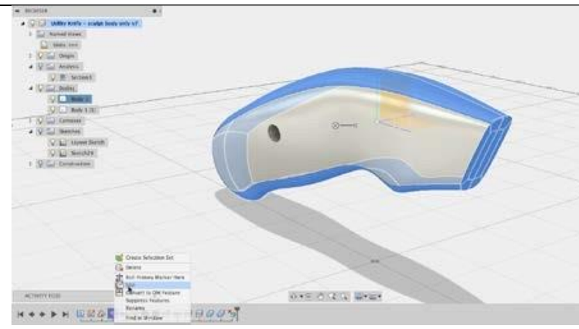
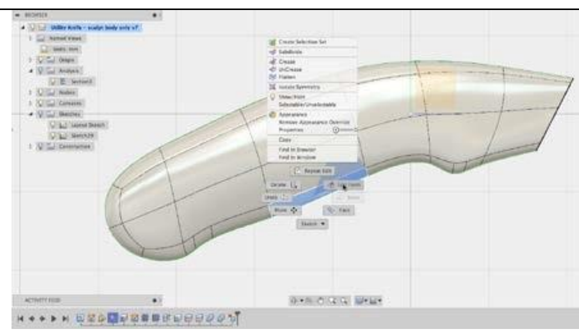
Step 1 – Hide one side of the design After the split body commands, you'll see that you have 4 bodies in your Bodies folder in the browser. We'll want to combine the left cylinder with the left knife body, and the right cylinder with the right knife body so that we are left with two bodies total. 1. In the browser, use the light bulbs to hide the right side bodies so only the left side is visible.	
Step 2 – Start the Combine command 1. Click **Modify > Combine**. We're going to use this command to join the knife body with the cylinder body where we have our counter-bore hole.	

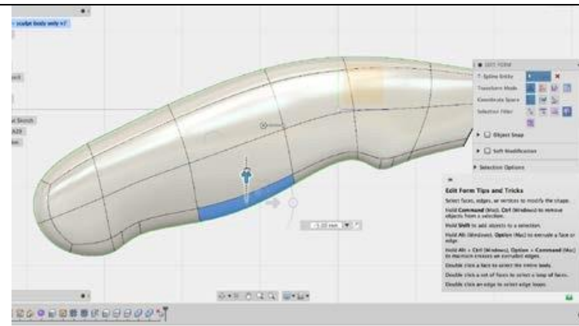
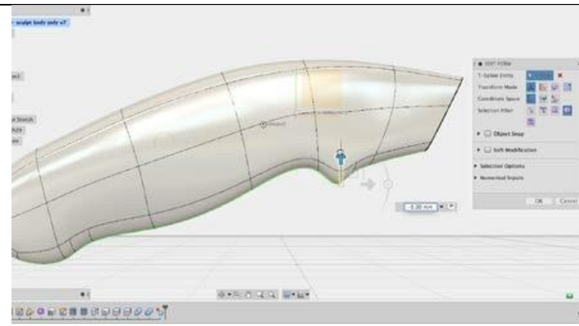
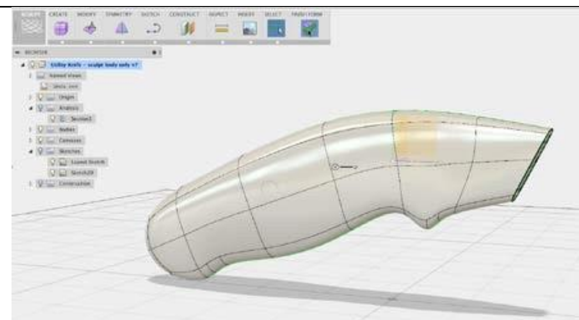
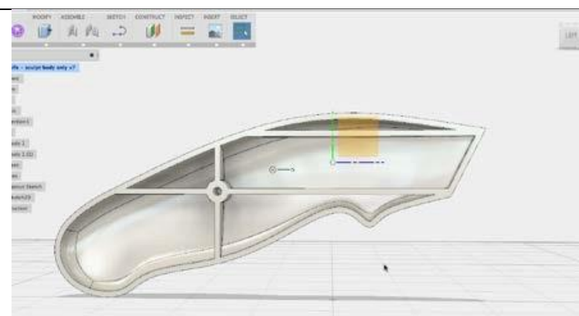
Step 3 – Select the body to keep 1. First, select the utility knife body as the Target Body.	
Step 4 – Select the tool body 1. Select the cylinder body as the Tool Body for the target body to combine with. 2. Leave the Operation as **Join**. 3. Click **OK** to finish.	
Step 5 – Repeat for the right side 1. In the browser, use the light bulbs to hide the left side and make the right side bodies visible. 2. Right-click and select **Repeat Combine** to reuse the last used command.	
Step 6 – Set the Combine options We're going to repeat the last combine steps, but now for the right side bodies. 1. Select the knife handle as the Target Body. 2. Select the cylinder as the Tool Body. 3. Leave the Operation as **Join.** 4. Click **OK** to finish.	

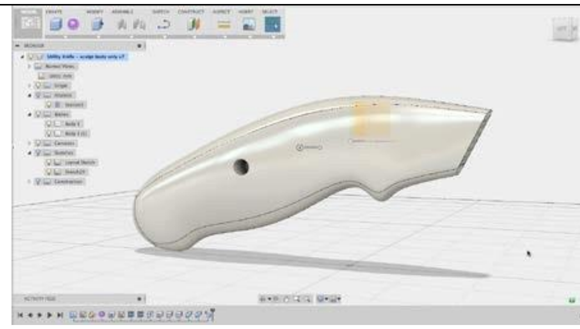
Step 7 – Delete bodies Let’s get rid of the two protruding cylinders. 1. Hold **Shift** and then select the two cylinder bodies. 2. Press **Delete** or **Backspace** on your keyboard and the two bodies should just go away.	

Step 8 – Finished webs and holes Now you can see that the utility knife has a counter-bored hole that extends across the split bodies, just the way it would be manufactured.	

Modify shape: Finally, you edit the sculpted body to see how this affects downstream operations.

Step 1 – Edit sculpted form Let's make a change! 1. Locate the sculpt operation in the timeline at the bottom of the canvas. 2. Right-click and select **Edit**. This allows you to get back into the sculpt environment and make change to the knife body itself.	
Step 2 – Select the faces to modify 1. Select **LEFT** view on the ViewCube. 2. Click on the bottom face. 3. Right-click and select **Edit Form**.	

Step 3 – Move the face 1. Use the vertical arrow manipulator and drag the surface down to **– 5 mm**.	
Step 4 – Move an edge 1. Select the lower edge of the finger guard. 2. Use the arrow manipulator and drag that edge down **– 5 mm**. 3. Click **OK** to confirm.	
Step 5 – Finish form 1. Click **FINISH FORM** to finish the changes. 2. Once clicked, the model updates automatically with all your downstream features still intact.	
Step 6 – View the inside of the design 1. Hide one of the bodies and notice that the webs and hole updated along with the change you made to the sculpted body.	

And we're done! Now that you know how the timeline works, you can select any of the commands used and make appropriate changes, such as web thickness, hole depth and diameter, shell thickness, as well as sketch dimensions.	

Manage and Collaborate

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.

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).

21.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.

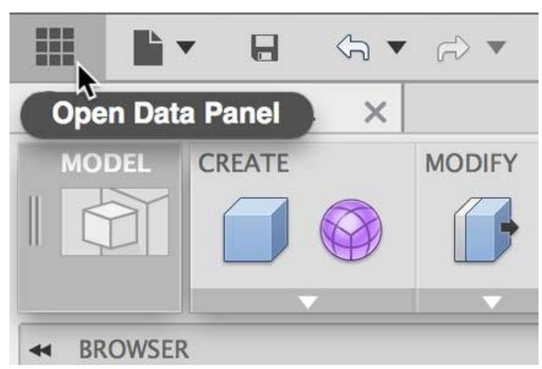
Fusion 360 allows you to securely access data from anywhere. You also control who else has access. The main way Fusion 360 controls access is by using "group projects." Group projects are the control mechanism for how you define who has access to specific sets of information. Group projects ensure that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

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<table>
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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.

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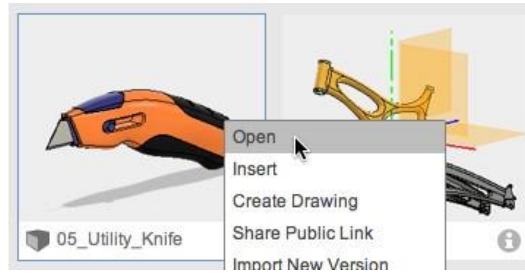

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<td>

Step 3 – Start the Save As command

1. Select **File > Save As.**

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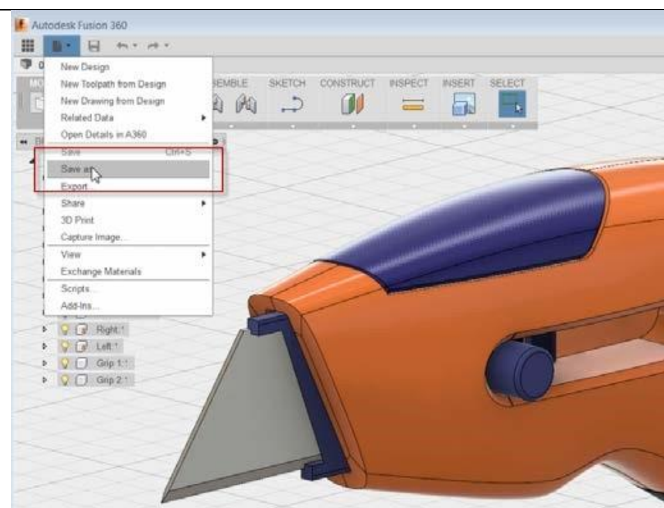
<td>



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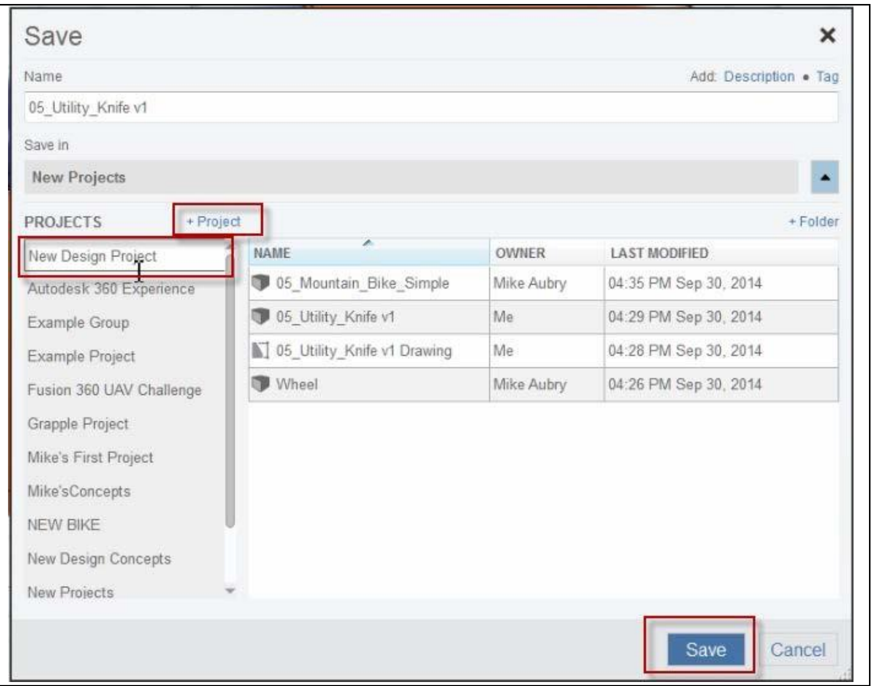
</table>

Step 4 – Create a new project

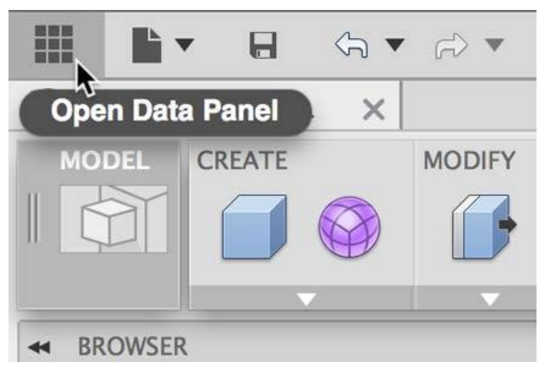
1. Click **+Project** to create a new project.
2. Name the project "**New Design Project**".
3. Select **Save**.



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

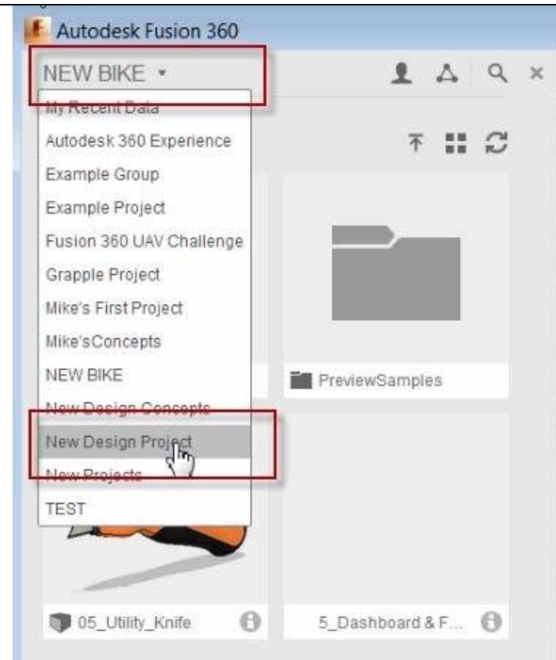
1. Open the Data Panel by clicking on the icon located at the top left of the menu bar.
2. The Data Panel will slide open.



4

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.



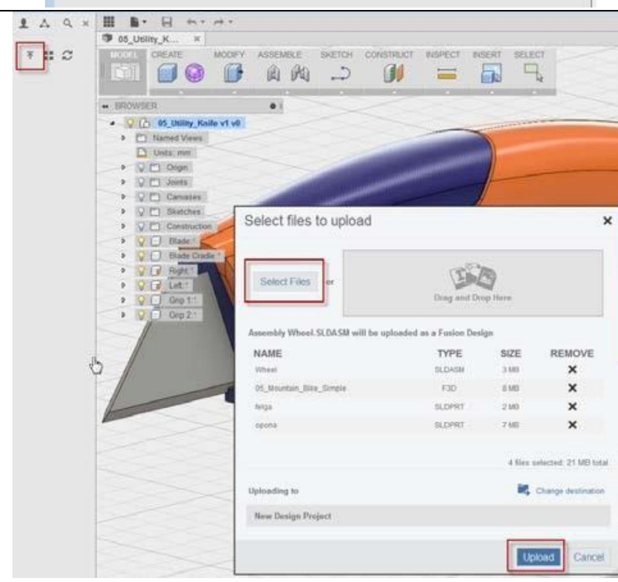
Step 3 – Upload data

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**.

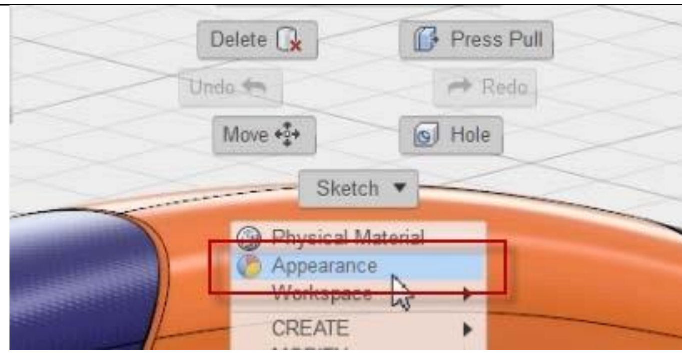
Uploading will continue in the background while you work.



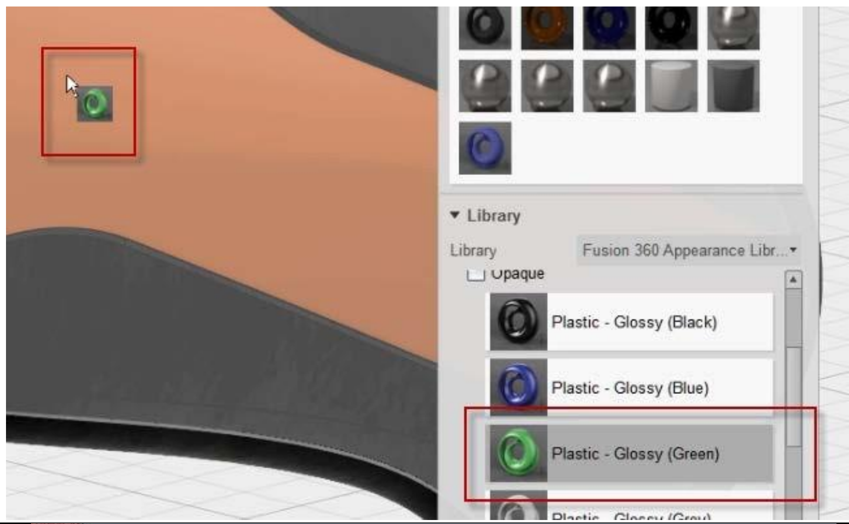
22.Create new versions

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

<table>
<tr>
<td>
Step 1 – Start the Appearance command

1. Right-click and select **Appearance**.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 2 – Change the appearance

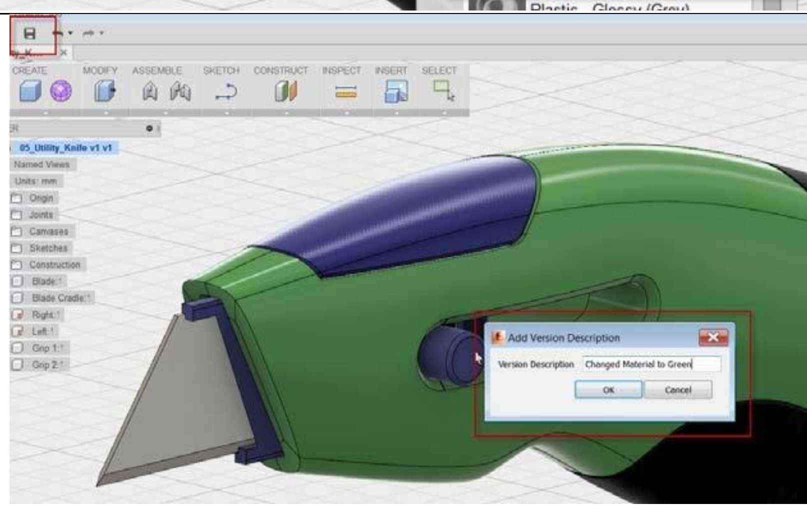
1. Select **Plastic > Opaque > Plastic – Glossy (Green)**.
2. Drag and drop the appearance over the orange cover.
3. Click **Close** to dismiss the appearance command.
</td>
<td>


</td>
</tr>
<tr>
<td>
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.
</td>
<td>


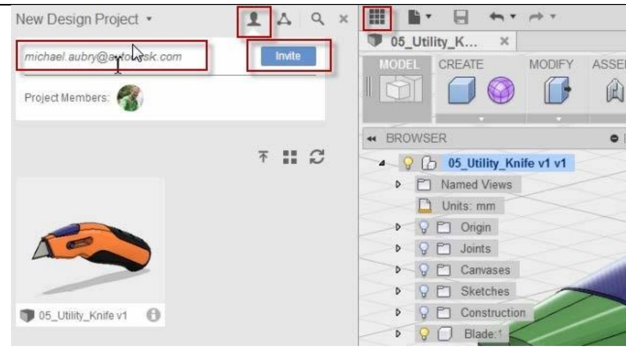
</td>
</tr>
</table>

23.Add a user to your project

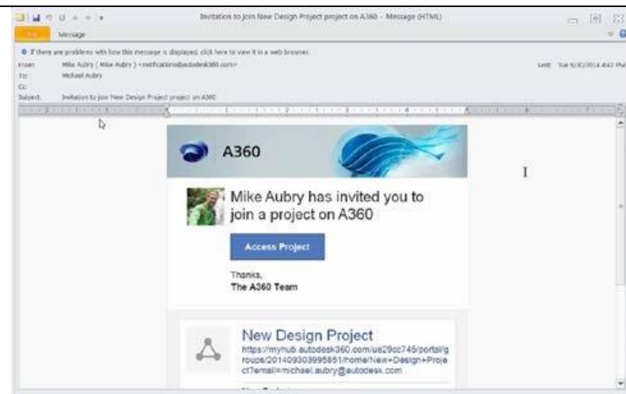
In this section you will open the introductory design file.

<table>
<tr>
<td>
Step 1 – Add a project member

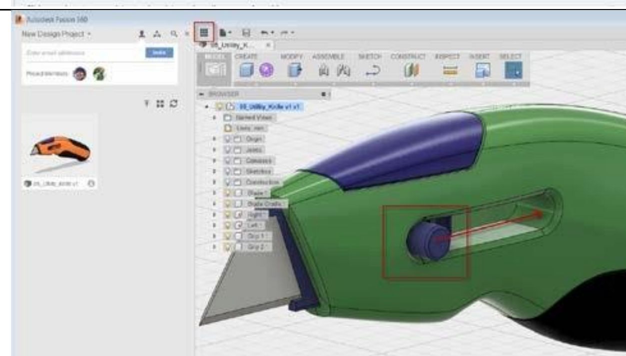
Fusion 360 allows you to control who has access to design and other data within projects. Invite another collaborator to access your project.

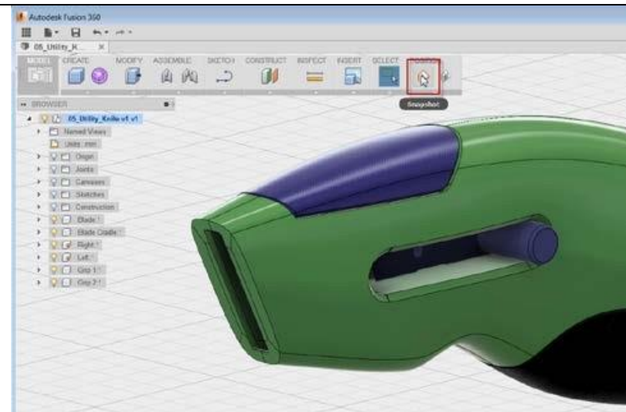
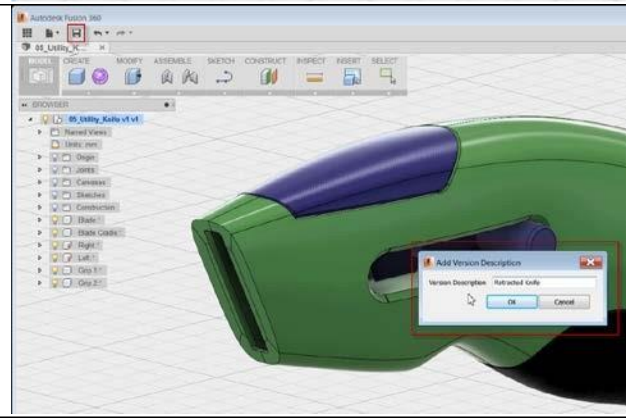
1. In the Data Panel, click the Project Members button (the head shaped icon).
2. Enter an email address of another Fusion user who you would like to have access. If you don’t have one feel free to use michael.aubry@autodesk.com.
3. Click Invite.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 2 – Confirmation email

1. Once added, the user you will receive an email notifying him or her of their addition.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 3 – Make a design change

1. Drag the slider to the right to retract the blade.
</td>
<td>

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

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.	

24.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.

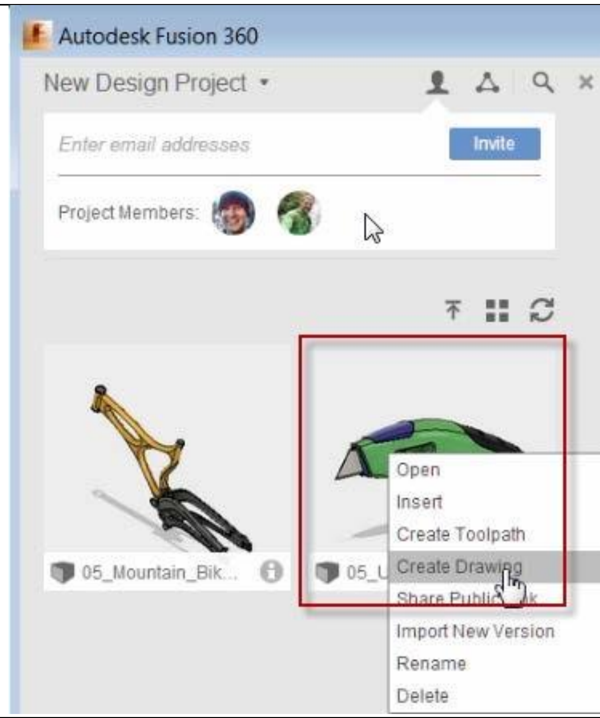
<table>
<tr>
<td>

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.**

</td>
<td>

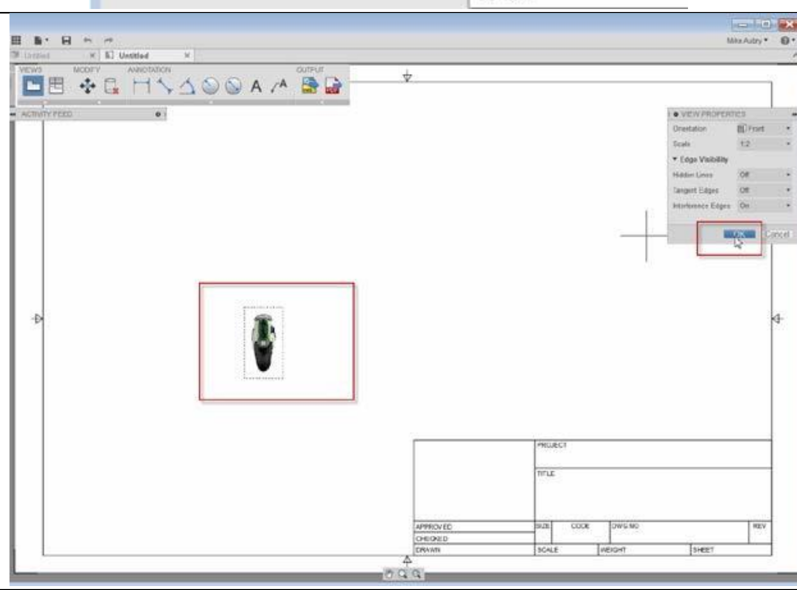


</td>
</tr>
<tr>
<td>

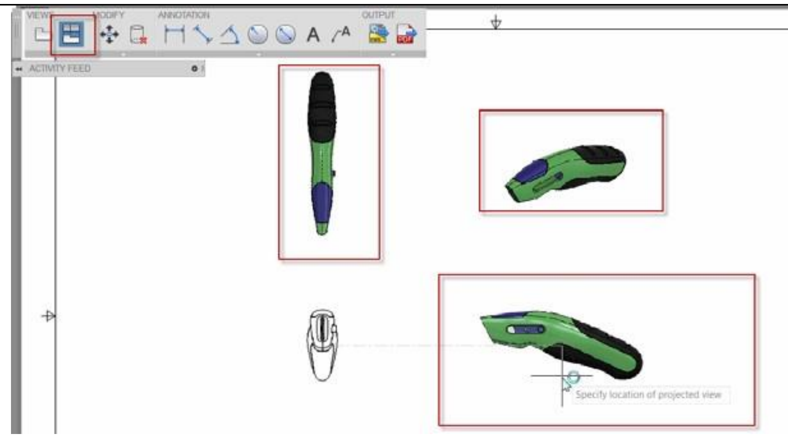
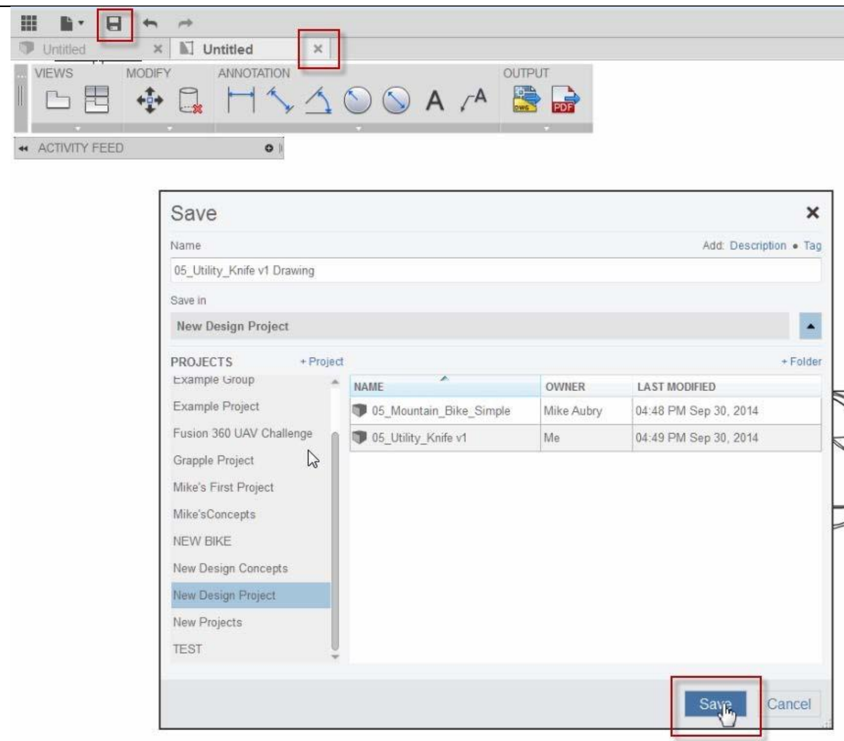
Step 2 – Place the base view

1. Left-click on the drawing sheet to place the base view.
2. Click **OK** to create the view.

</td>
<td>



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

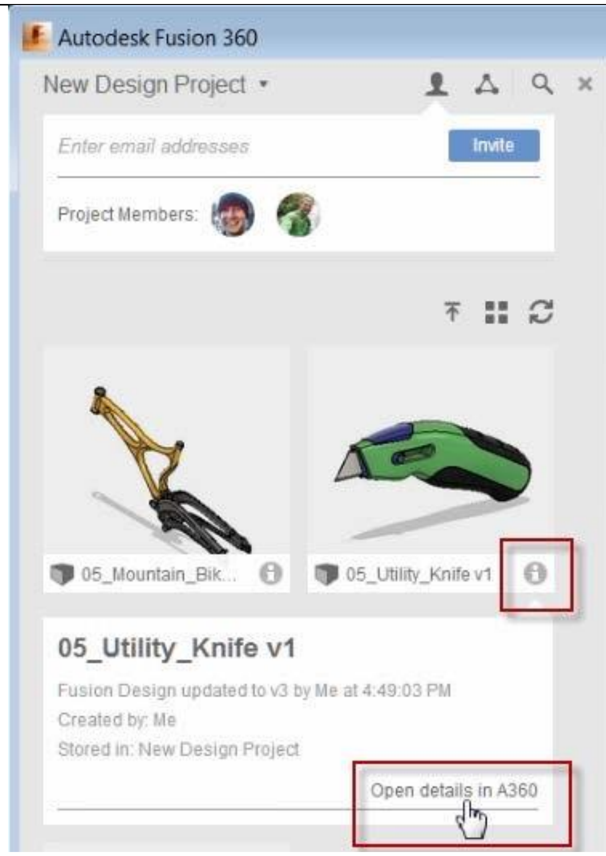
Step 3 – Add projected views 1. Click **View > Projected View**. 2. Click on the base view. 3. Click in the three areas shown to create top, right, and isometric views. 4. Right-click and click OK to create the views.	
Step 4 – Save and close 1. Click the **Save** icon. 2. Click **Save** to accept the default name and project. 3. Close the drawing. You will see the association in a later step.	

25.Access data in a web browser

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

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


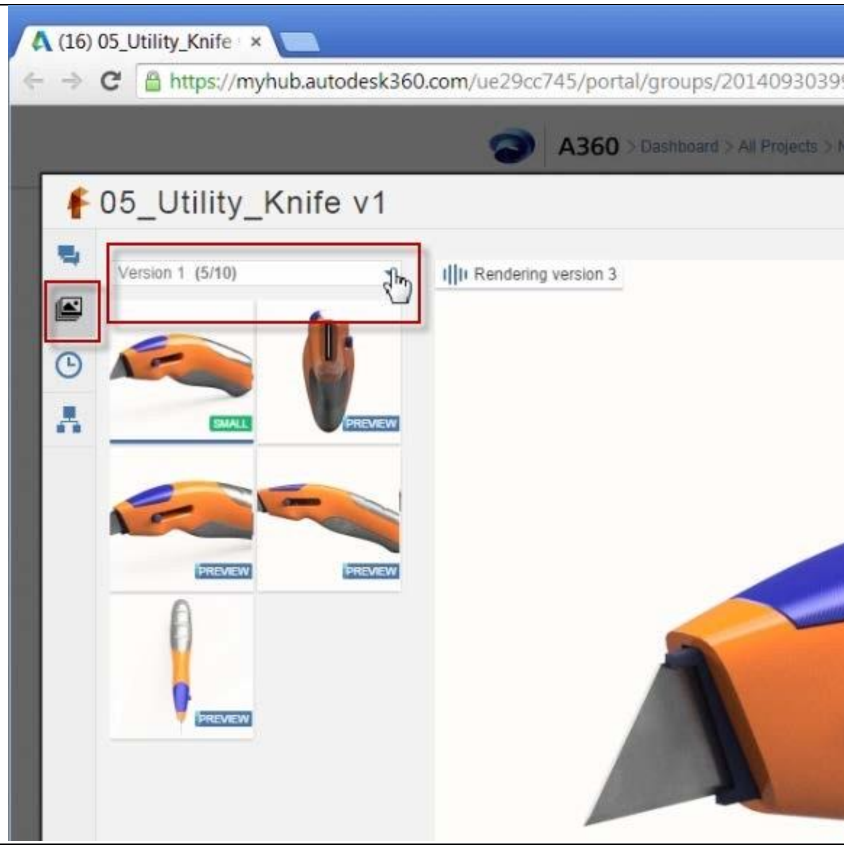
</td>
</tr>
</table>

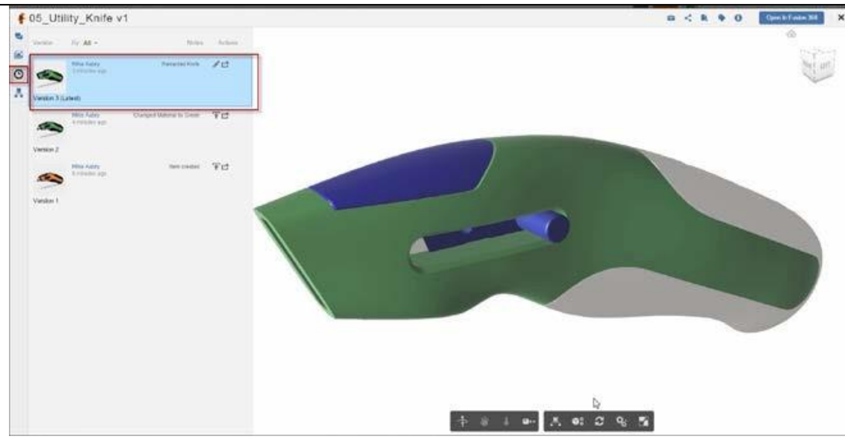
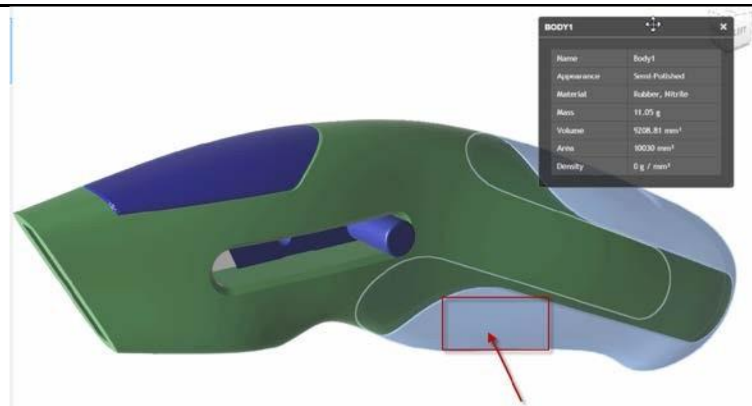
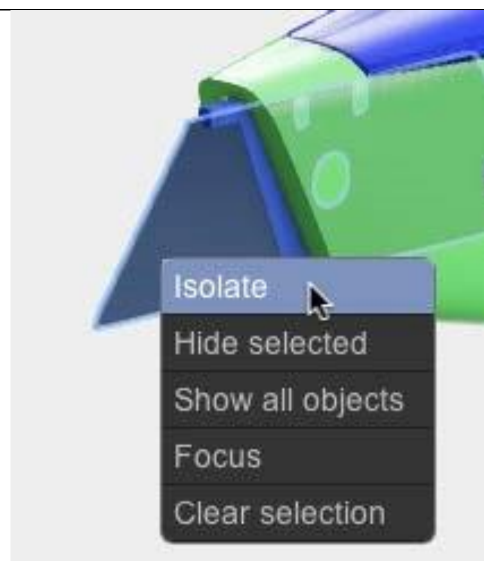
Display versions and use the viewer: In this section, you use the viewer to display the different versions of the utility knife. You also use the viewer to investigate the design.

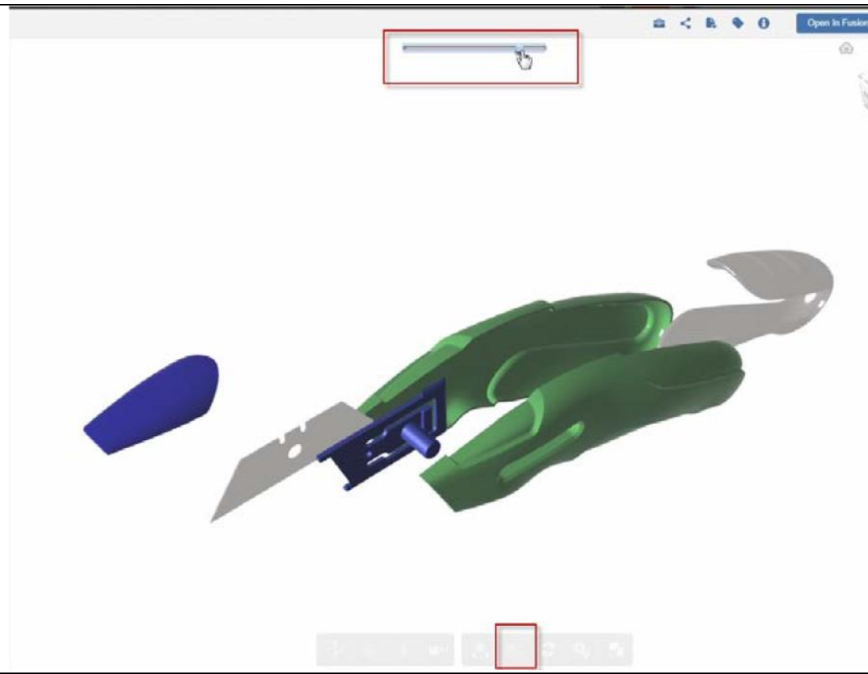
<table>
<tr><td colspan="2">Step 1 –View renderings

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.</td><td></td></tr>
<tr><td>2.</td><td>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.</td><td>
</td></tr>
</table>

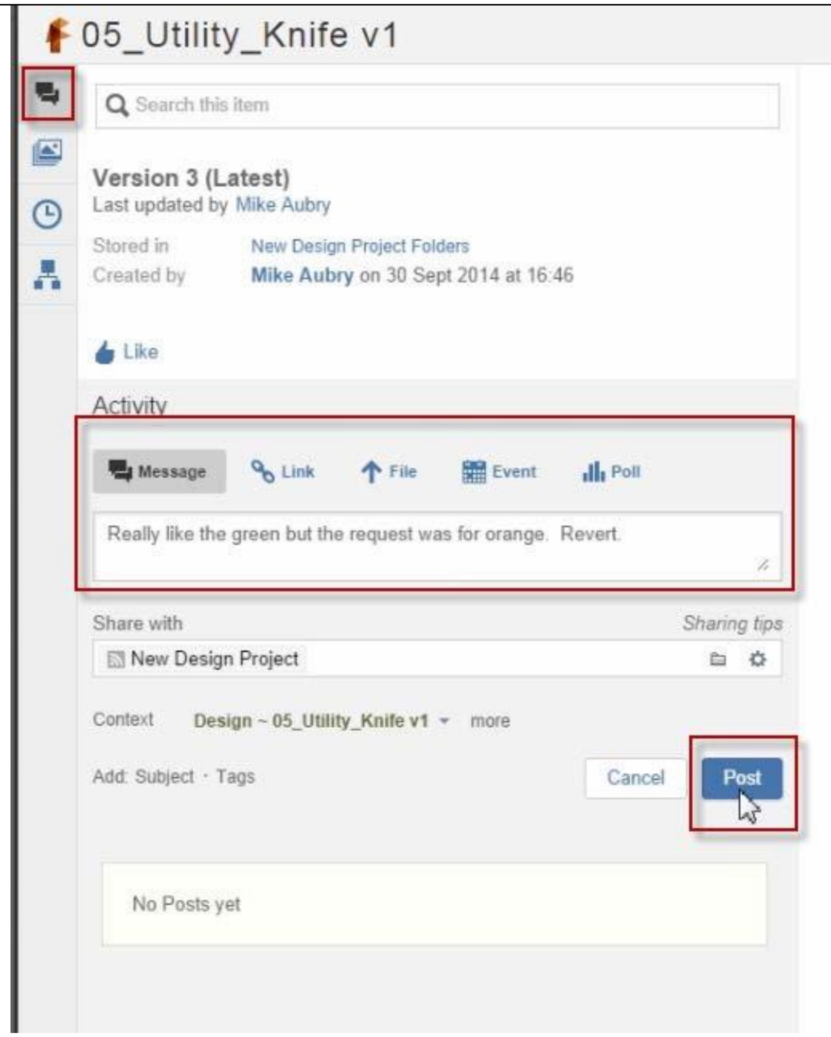
Step 2 – View versions 1. Click the **Versions** icon (the watch shaped icon). 2. Click the different versions to view 3D previews. Notice the comments that were entered in previous steps are also available to view.	
Step 3 – View the 3D design 1. Hold the left mouse button to rotate the 3D model in 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.

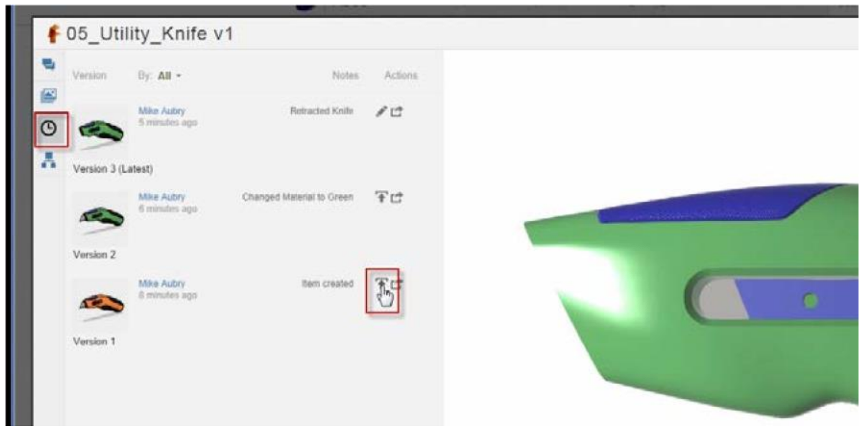
<table>
<tr>
<td>
Step 1 – Document activity

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.

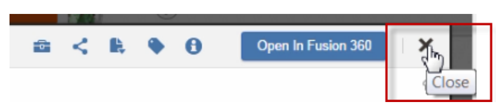
1. Click the Preview icon.
2. Click in the message field and enter: “Really like the green but the request was for orange. Revert.”
3. Click Post to post the message.
</td>
<td>


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

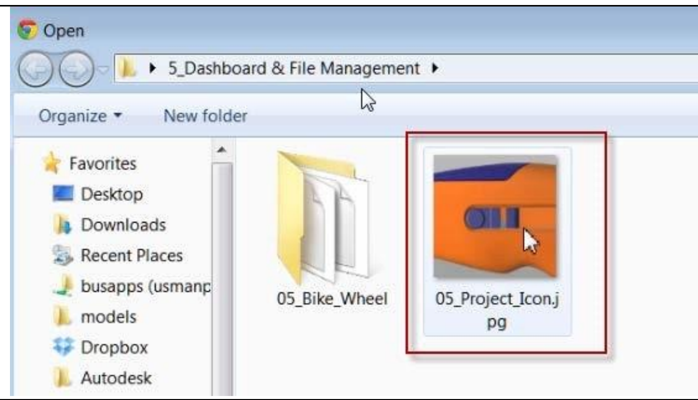
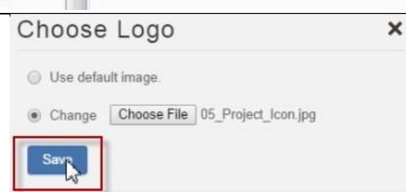
Promote a previous version: In this section, you promote a previous version to be the current version. This let's you "rollback" your design.

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.	

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.

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.	
Step 2 – Close the viewer 1. Click the X to close the viewer.	

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

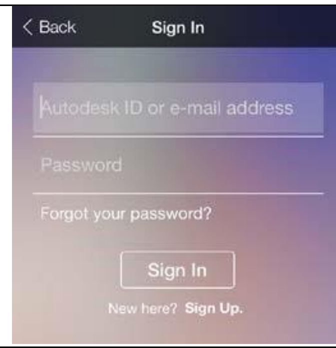
Step 12 –Add a logo to your project 1. Select **Add Logo** in the A360 web browser. 2. Select **Choose File**.	
Step 2 – Select the image 1. Navigate to the png image provided in the chapter 5 dataset or use any jpg or png under 200x200 pixels. 2. Select the image then click **Choose**.	
Step 3 – Save the change 1. Click **Save.**	

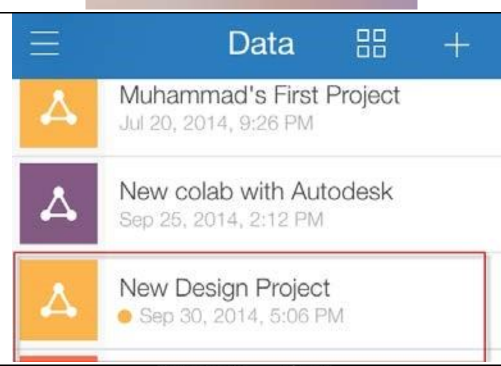
Access data from a mobile device: In this section, you access the project from a mobile device.

<table>
<tr><td>Step 1 – Access data from a mobile device

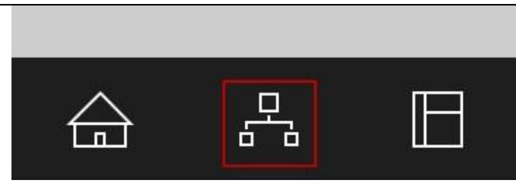
Fusion 360 is accessible through your mobile devices.

1. To access data from your apple mobile device install the free Autodesk A360 App via the Mac App Store or Google Play Store.</td><td>
</td></tr>
<tr><td>Step 2 – Sign in

1. Sign in to the App using your Autodesk ID (it is the same account you use for Fusion 360).</td><td>
</td></tr>
<tr><td>Step 3 – Select the project

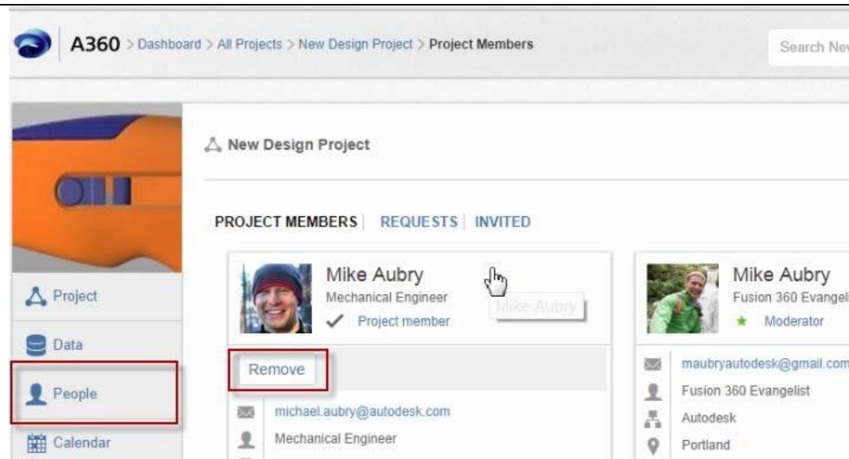
1. Scroll to the New Design Project and select it.</td><td>
</td></tr>
<tr><td>Step 4 – Select the design

1. Open the Utility Knife.</td><td>
</td></tr>
</table>

Step 5 – Select the action 1. Select the **Isolate** icon.	
Step 6 – Select the object 1. Choose **Blade Cradle** to isolate the blade.	
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.	

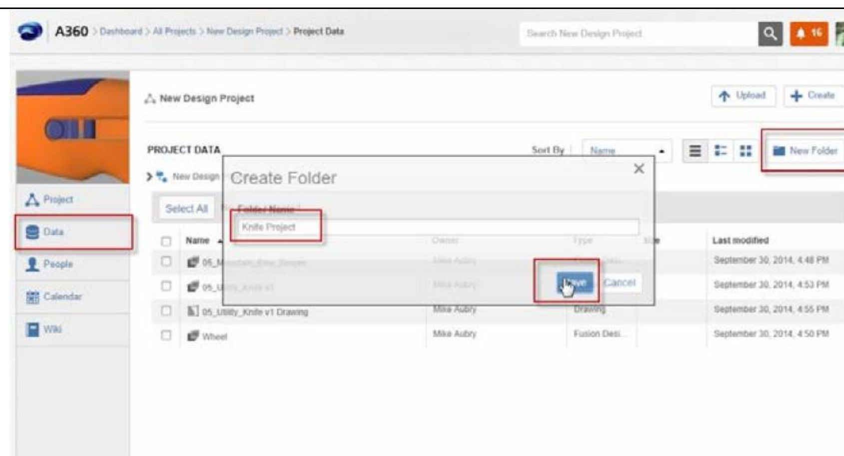
Revoke access to the project: Fusion 360 allows the moderator of a project to add or remove access to that project. Revoke the access of a project member.

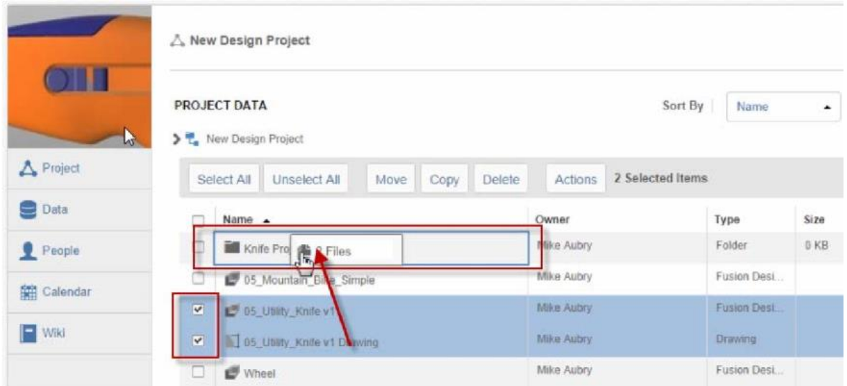
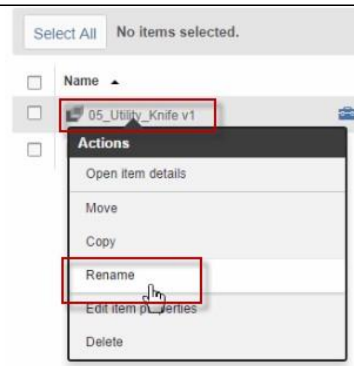
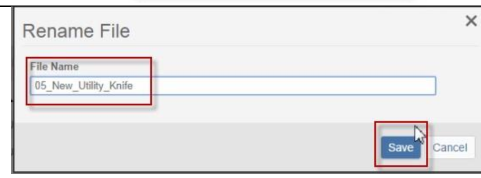
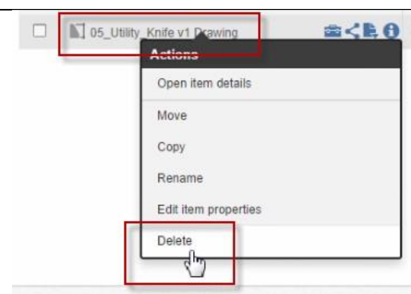
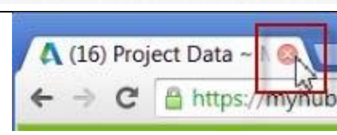
<table>
<tr>
<td>Step 1 – Revoke access from a project

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

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

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

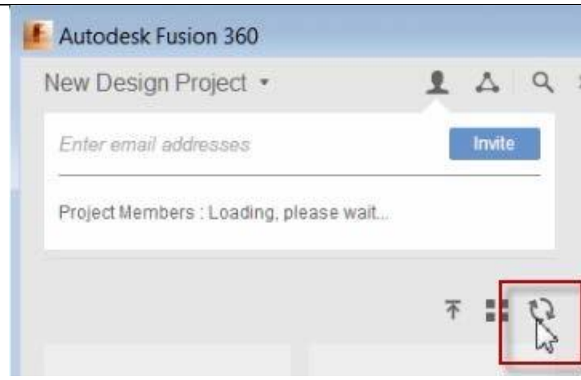
Step 2 – Move data 1. Check the boxes next to the design and drawing. 2. Drag and drop the items into the folder to move them. 3. Open the **Knife Project** folder.	
Step 3 – Rename a design 1. Right-click on the **05_Utility_Knife** design and select **Rename**.	
Step 4 – Enter new name 1. Rename to: "**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.	

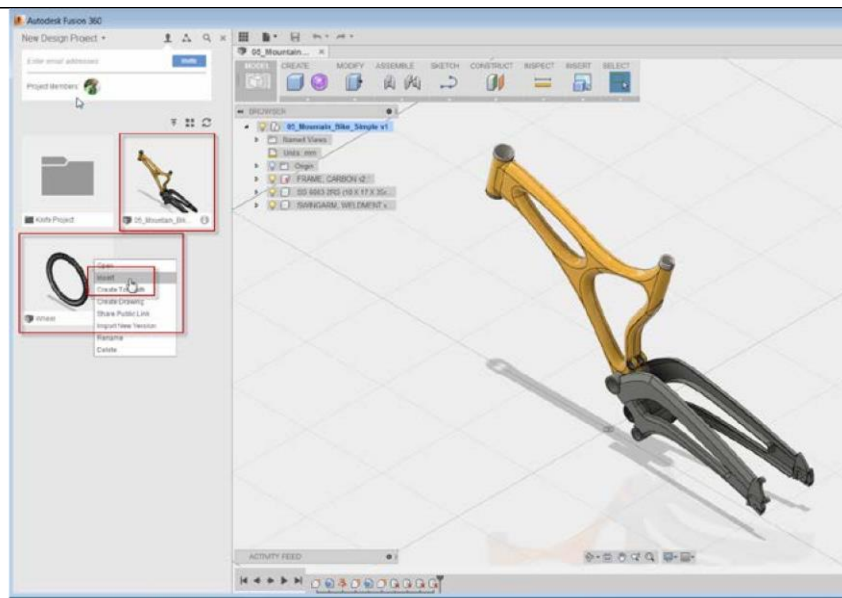
26.Insert designs into other designs

In this section, you insert a design into an assembly design.

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

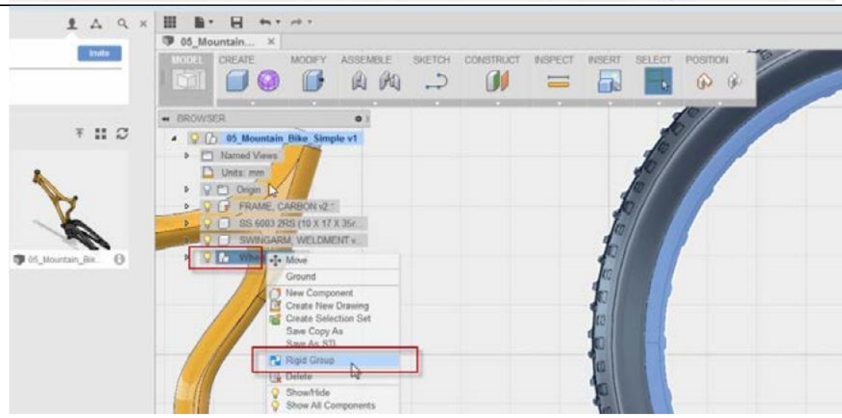
Step 3 – Place the wheel

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

Step 4 – Lock the wheel component

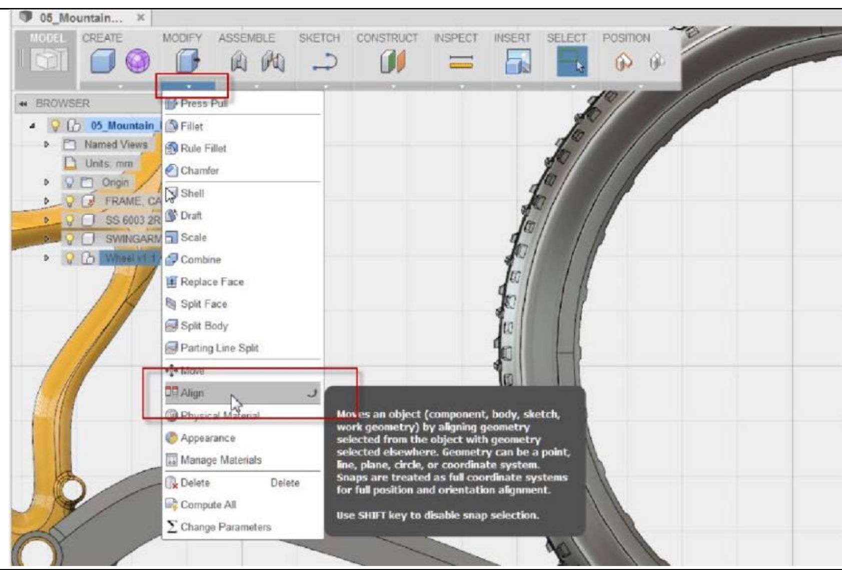
1. Right-click **Wheel** in the browser and select **Rigid Group**.

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

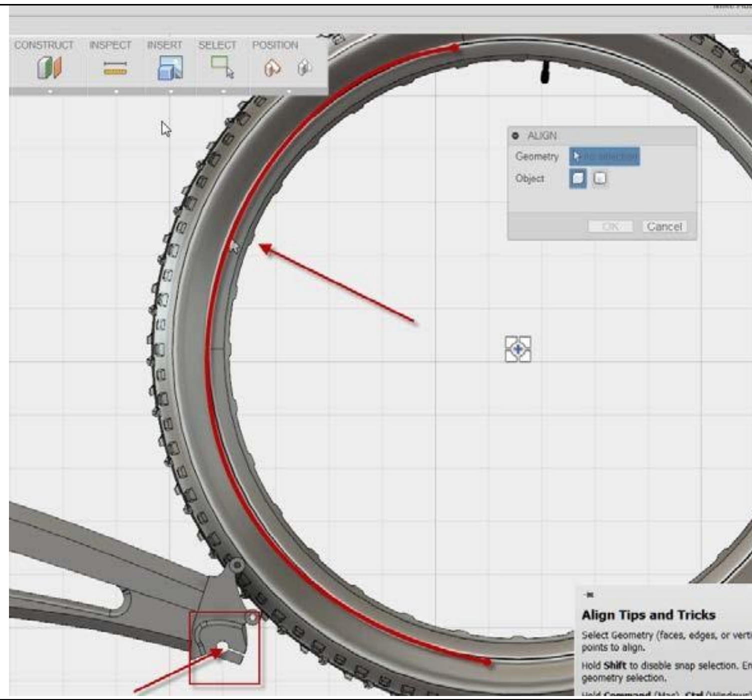


Step 5 – Start the Align command

1. Click **Modify > Align**.

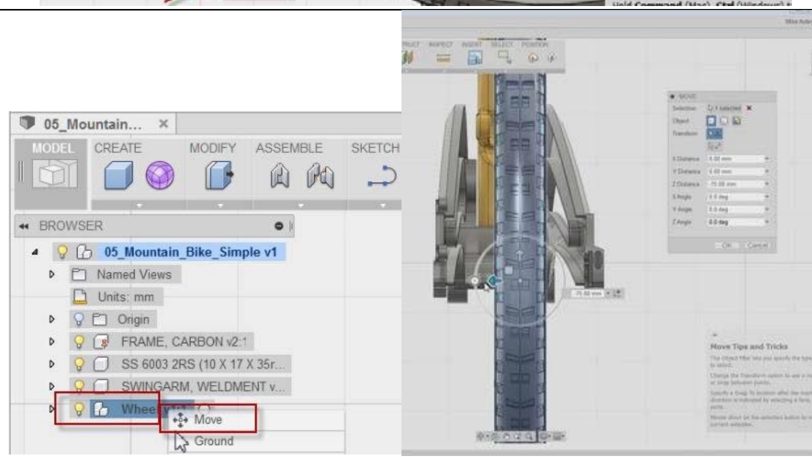


<table>
<tr>
<td>
Step 6 – Select areas to align

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>


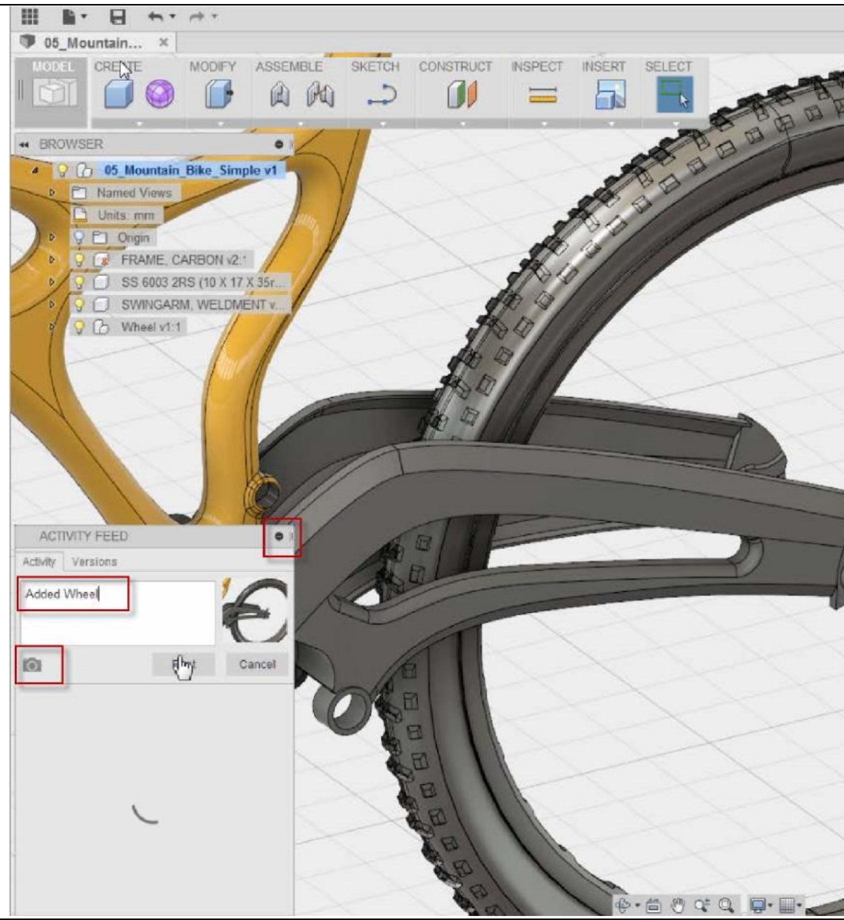
</td>
</tr>
</table>

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**.

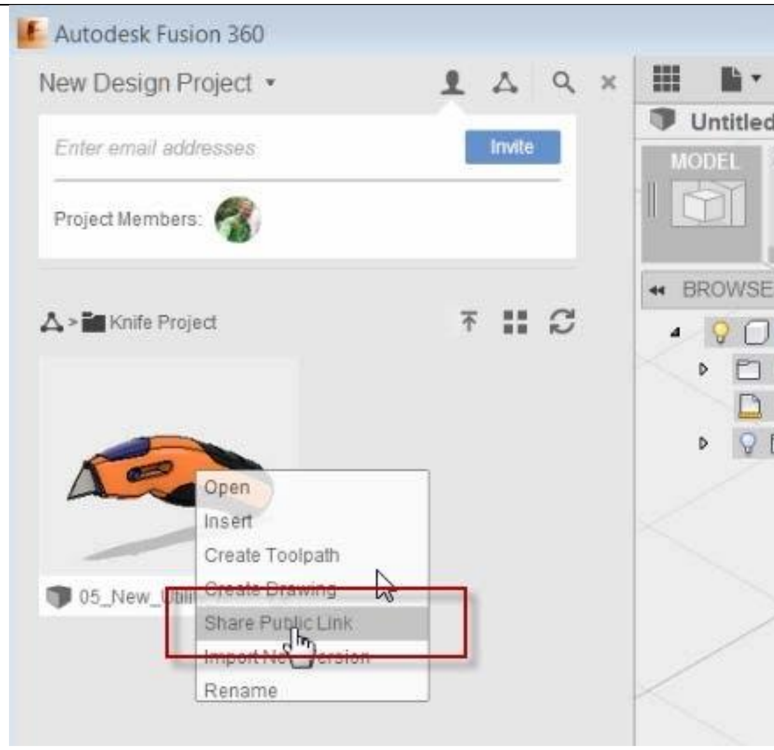


27.Share designs

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

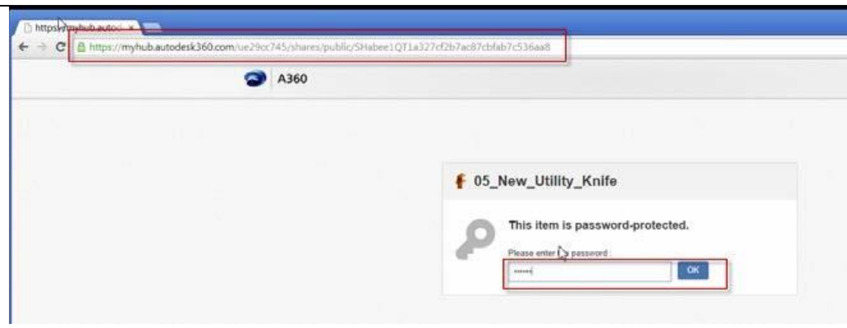
Regardless of whether a user has an Autodesk ID, you are still able to share and make designs available for download. Share a design using a public web link.

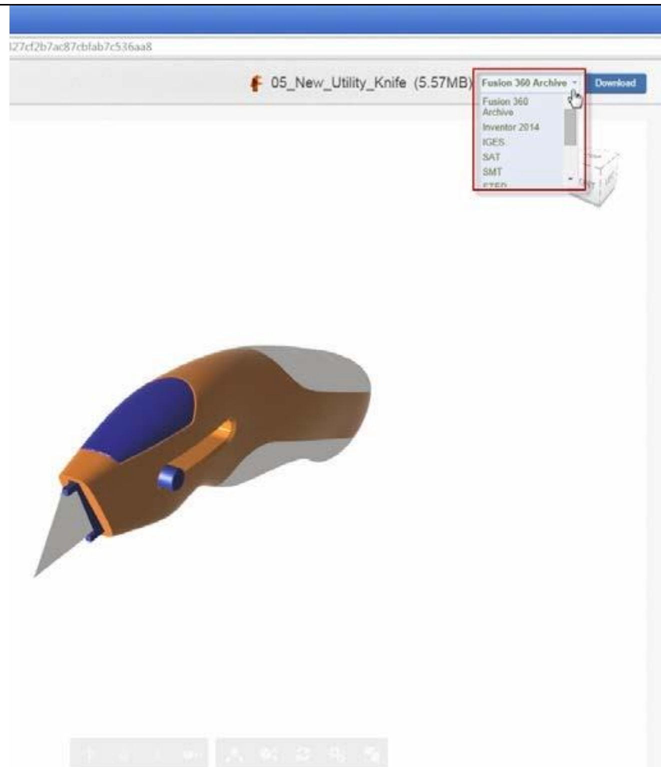
1. Open the Data Panel and browse to the 05_Utility_Knife.
2. Right-click on the Utility Knife and choose Share Public Link.
</td>
<td>


</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.
</td>
<td>


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

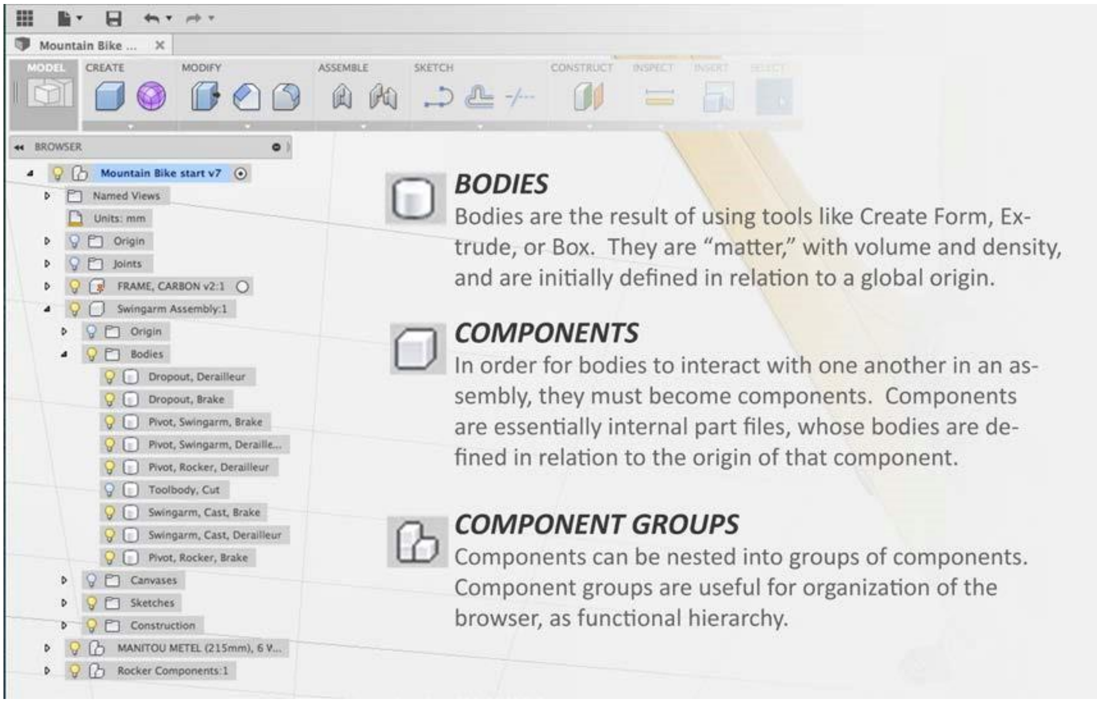
4. Close the Share Public Link dialog box.	
Step 3 – Use the link 1. Open your web browser. 2. Paste in the copied URL. 3. Enter the password, "**Fusion!**".	

Step 4 – View and download 1. A 3D preview is displayed in the browser. 2. Select a CAD format to download. 3. Click **Download**.	

Congratulations! You completed the Manage and Collaborate module.

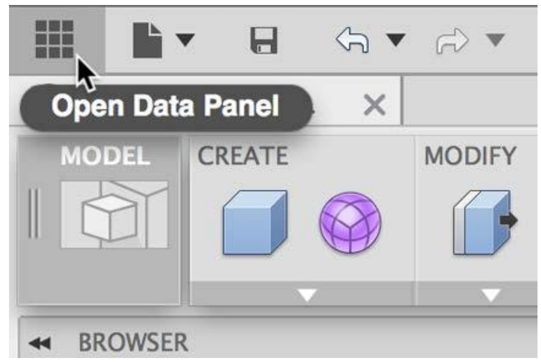
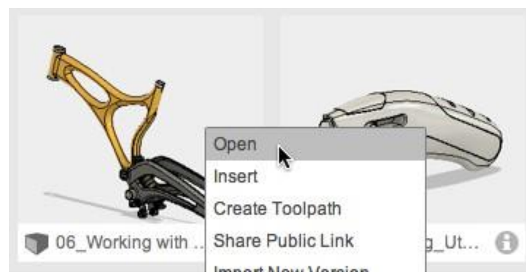
Assembly Design

Bodies and Components

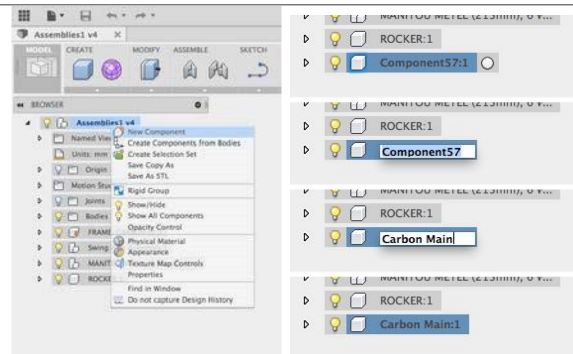
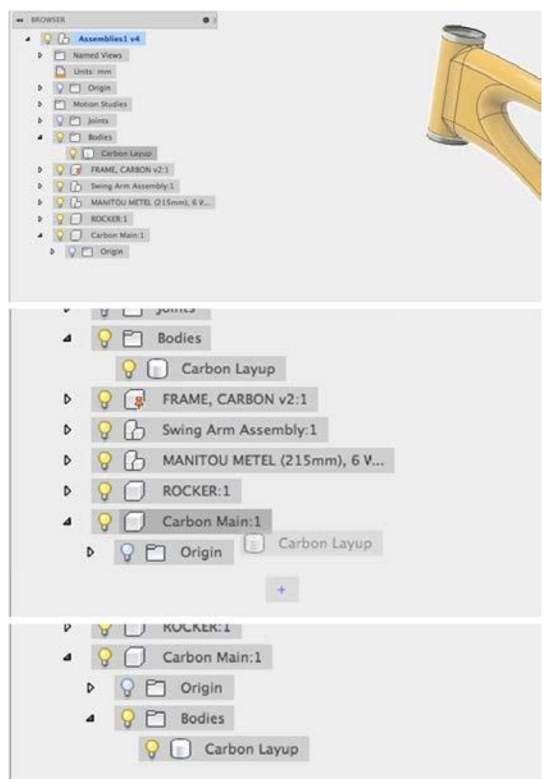


If you're more familiar with a CAD system that references external parts in an assembly, the first thing to know is that Fusion's equivalent to a "part file" is a "component," and all components exist in the same working Fusion file – there are no external references. Component groups act like sub-assemblies, and bodies are physical objects that exist either in the global space, or in a component. There can be multiple copies or instances of one component, and in that case, modifying one will modify all other instances similarly.

Open Fusion 360 design file: In this section you will open the first 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* **06_Working with Components.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 **06_Working with Components.f3d** file. 2. Navigate to this design and either **doubleclick** 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.	

Working with Components: In this exercise, we'll be exploring the different tools used work with components and component groups.

Step 1 – Creating components 1. Right-click on the top node in the browser. 2. Select "**New Component**." 3. Click on the name of the new component in the browser. 4. Click on the name again to edit. Enter "**Carbon Main**."	
Step 2 – Converting to Components 1. Expand the main "Bodies" folder in the browser. 2. Drag the body "**Carbon Layup**" to the new "**Carbon Main**" component. The Body "Carbon Layup" is now defined in relation to the component origin, not the global origin. This is now effectively a "part" in an "assembly." Alternatively, right-click on the body in the tree and choose "Create Components from Bodies" creates a new component that includes that body. To pull any body or other object out of a component, drag it to its desired destination, or the top node in the browser.	

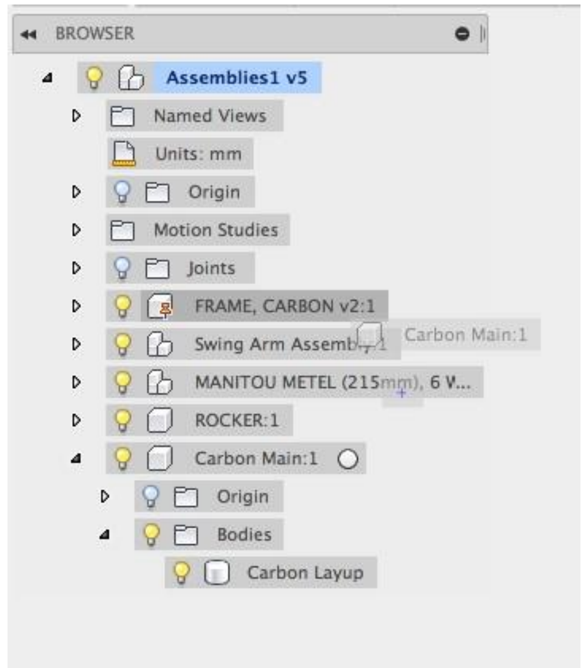
Step 3 – Component Groups

1. We want to move this component into a component group.
2. Click on and drag the new “**Carbon Main**” component, and release it over the “**Frame, Carbon**” Component Group.
3. We’ve now moved it into a component group.

By moving it into a component group, we’ve defined its origin in relation to the subgroup’s origin, which is in turn defined by the global origin.

By moving any component A to any component B, component B will become a component group that contains its original content, as well as component A.

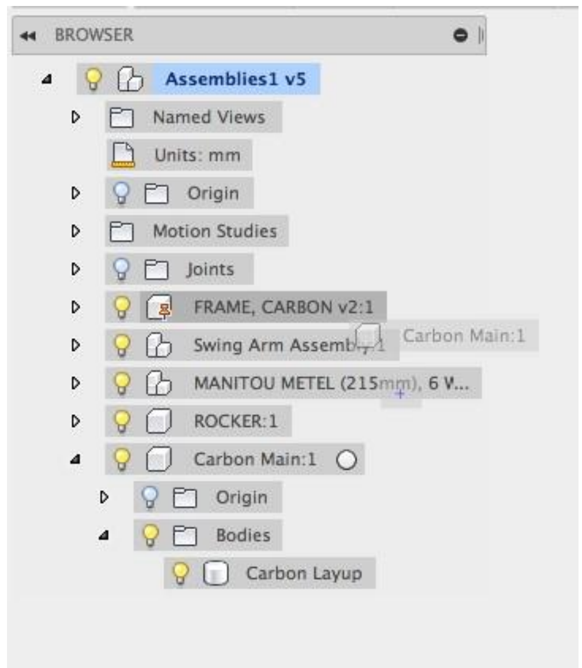
If a component group is moved into another component group, it creates a nested component group, accordingly.



Step 4 – Instances/Referencing

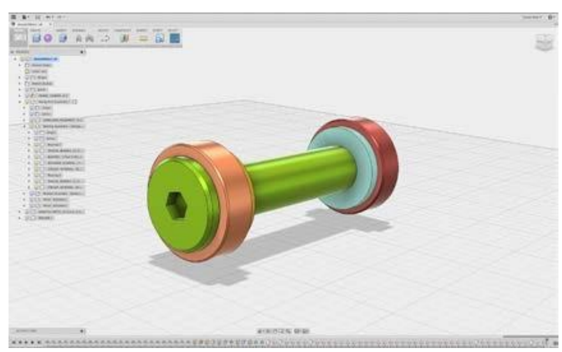
1. Expand the component group “**Swingarm Assembly**” in the browser.
2. In the group, expand the nested component group “**Bearing Assembly – Swingarm to Frame**”
3. Rename the first component that’s listed. Change “SS 6003 2RS (10 X 17 X 35mm)” to “**Bearing**.”

Note that the change also changed another component listed in the “Bearing Assembly – Swingarm to Frame” group. This is because the component that is now “Bearing: 2” is actually just another instance of the bearing component. Making any change (renaming, materials, or any modeling change) to one will make the same change to any other instance of that component. Instances are denoted with a :x, x being the 2nd, 3rd, 4th, etc. instance of that component.



Step 5 – Component Color Cycling

1. Click the “**Home**” button on the ViewCube.
2. Click **Inspect > Component Color Cycling.**
3. Expand the group “**Swingarm Assembly**” in the browser.
4. Right-click on “**Bearing Assembly – Swingarm to Frame**” and select **Isolate**.
5. Notice that even though the two identical instances of the “Bearing” component are shown, they are different colors to distinguish them.
6. Click **Inspect > Component Color Cycling** to turn off component color cycling.



28.Move and Align

In this exercise, we move and align components in space. Moving and aligning position components, but do not lock them into their new location. Joints (in an upcoming exercise) move and align components, but also restrict their movement based on that definition. Moving and aligning simply shifts position in the global space.

<table>
<tr>
<td>
Step 1 – Move

1. Display the Data Panel.
2. Open “**06_Moving and Aligning”**.
3. Hover over the workspace switcher and select **Model**.
4. Click the **Home** icon on the ViewCube.
5. Expand the group “**Swingarm Assembly**”.
6. Right-click on “**Bearing Assembly – Swingarm to Frame**” and select **Isolate**.
7. Right-click on “**Bushing 17mm X 85L**” in the browser then select **Move**.
8. Type **– 20 mm** as the Z Distance in the dialog box.
9. Click **OK**.

We’ve moved a component, and not just a body. Therefore, its origin has moved with it, and has been redefined by the global origin.
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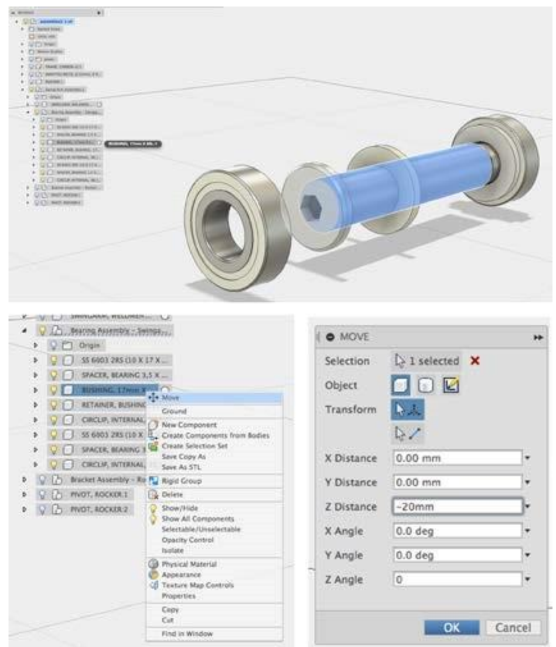

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<tr>
<td>
Step 2 – Align Component

1. Select the front side face of the bearing shown at right.
2. Hold **Shift** and select the inside back side face of the bushing head.
3. Click **Modify > Align**.
4. In the dialog box, make sure “**Align Components**” is selected for the Object selector.
5. Click **OK**.

When aligning, the geometry that is clicked first will move to the geometry that is clicked second. Here, we’ve aligned two full components, not just bodies.
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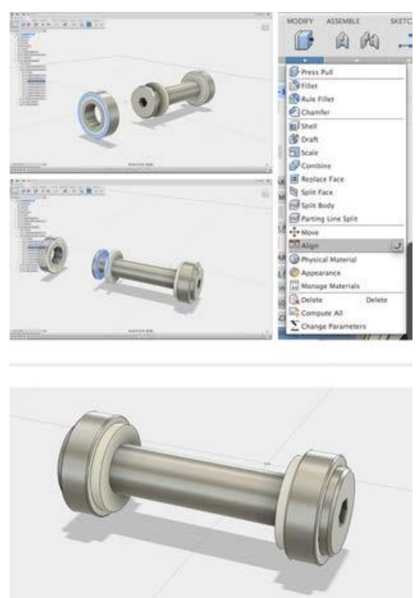
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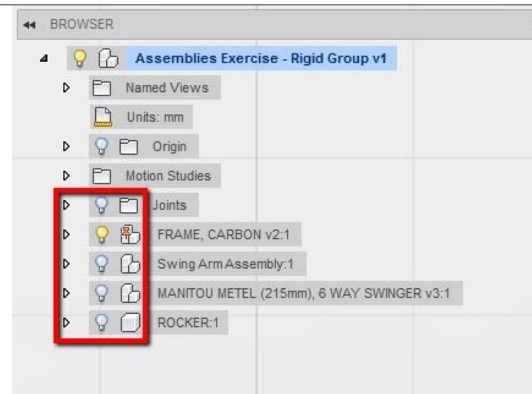
29.Create a Rigid Group

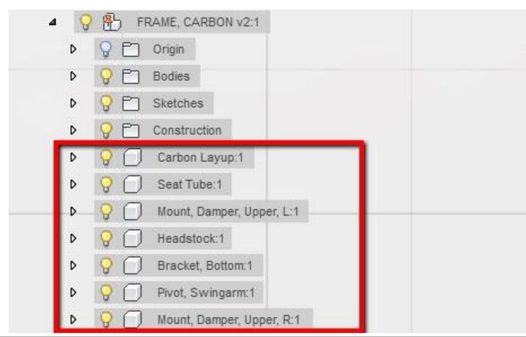
We create Rigid Groups in Fusion when we need to constrain multiple bodies to each other that have no capability of movement relative to themselves. So instead of creating multiple

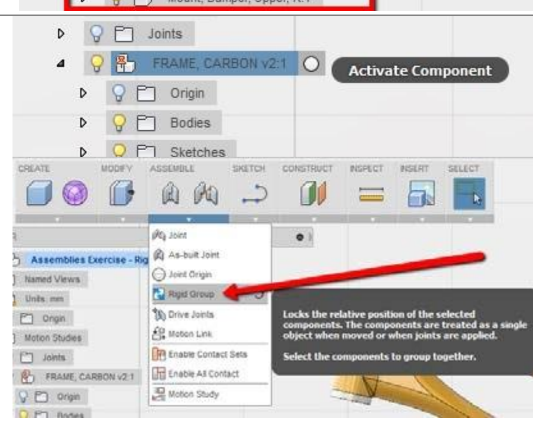
Rigid joints, it's easy to use the Rigid Group command and constrain multiple objects at once. The Rigid Group function locks the relative position of the selected components. The components are then treated as a single object when moved or when joints are applied.

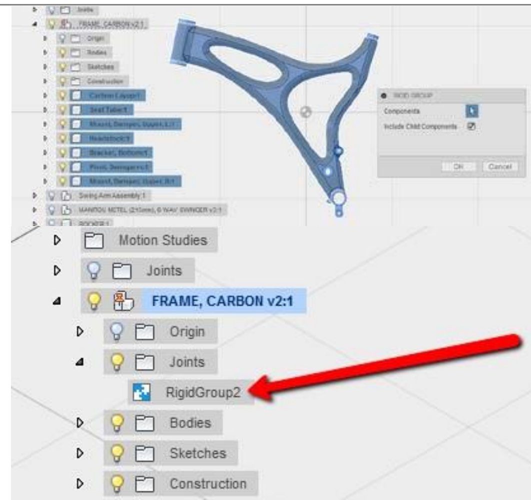
<table>
<tr><td>Step 1 – Open a design

1. Display the Data Panel.
2. Open the 06_Rigid Groups design.
3. Notice that the visibility of all other components except the Frame is turned off.</td><td>

Launch Video</td></tr>
<tr><td>Step 2 – Expand a component group

1. Expand the Frame, Carbon component group.
2. Notice that there are multiple components within it. We need these components to function as a single object and so we will use the Rigid Group function to constrain them.</td><td>
</td></tr>
<tr><td>Step 3 – Start the Rigid Group command

1. Hover over Frame, Carbon then click the dot next to the name to activate the component. We do this because we want the Joint to associate with this component group rather than the top level assembly.
2. Click Assemble > Rigid Group.</td><td>
</td></tr>
</table>

Step 4 – Select objects 1. Select **Carbon Layout**, hold **Shift** then select **Mount, Damper, Upper, R**. This selects all components between those two. 2. Click **OK** and all these components will now function as one single object. 3. The Rigid Group is listed in the browser under the Joints node.	

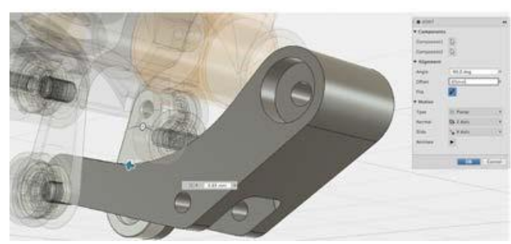
30.Joints

In this exercise, we'll use the joints tool to align a component to others in an assembly. These joints will also define the degrees of freedom by which these parts can move. Joints are enacted between components, but are defined by certain features within the component, like a body face or edge. They ultimate define how components can move and animate, and they drive motion studies.

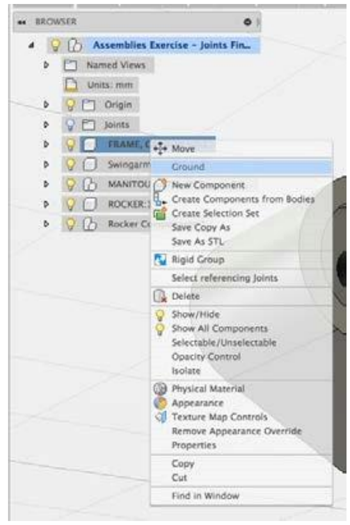
Step 1 – Add Planar Joint

1. Display the Data Panel.
2. Open the **06_Joints** design.
3. Hover over the workspace switcher and select **Model**.
4. Select **Assemble > Joint**.
5. For Component 1, select the front face of the rocker and snap to a point near the hole in the center.
6. Rotate the model to view the back of the rocker pivot (shown). Select the back face of the rocker pivot and snap to the point in the center of the face.
7. You'll see Component 1 move to Component 2, with an animation of the degrees of freedom of the joint.
8. Under Motion Type, select "**Planar**." The animation shows the two selected planes exiting on the same plane, with the freedom to move along that plane.
9. Enter an offset of **0.65 mm** to a space between the two components. (Click the Left View of the ViewCube to see this gap)
10. Click **OK**.

The animations you see are assuming no other joints in the assembly. The animations ONLY show the added constraints of the joint.



<table>
<tr>
<td>Step 2 – Ground a component

1. Hold Alt then drag the rocker component to move it around in space a bit.
2. Notice that everything in the assembly moves with it. Something in the assembly needs to be grounded – fixed in space.
3. Click Undo to put the components back in the original positions.
4. Right-click the component “Frame, Carbon” and select “Ground.” You’ll see a pin  appear on the icon in the browser. 5. With the frame now fixed in space, hold Alt and drag the rocker to move it around a little. You will see the planar joint in action.</td>
<td>
</td>
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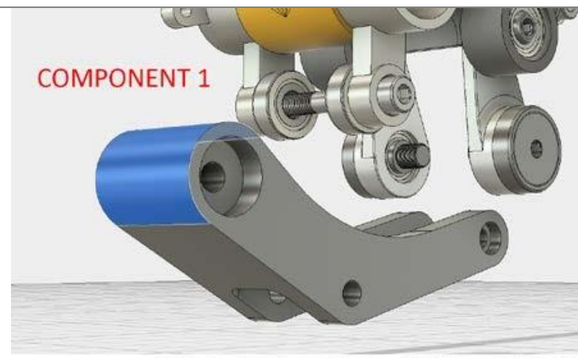
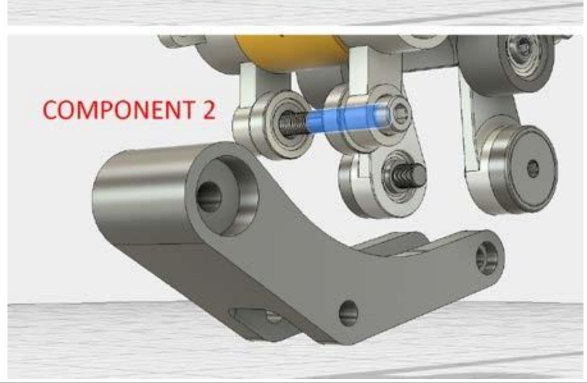
<tr>
<td>Step 3 – Add first of three cylindrical joints

You might need to move the rocker around or rotate the model to select some geometry.

1. Select Assemble > Joint.
2. Set the Motion Type to Cylindrical.
3. For Component 1, select the midpoint of the rocker face shown.
4. For Component 2, select the midpoint of the bolt face shown.
5. Click OK.</td>
<td>


</td>
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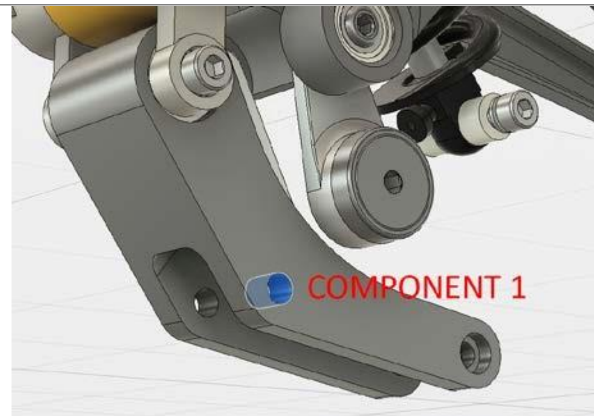
</table>

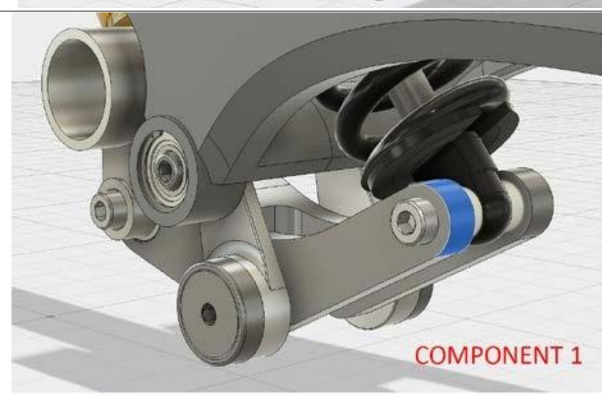
Step 4 – Add the next joint

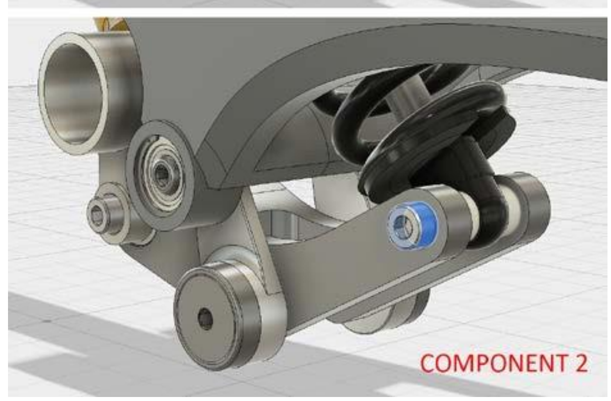
1. Right-click and select **Repeat Joint**.
2. Make sure the Motion Type is set to **Cylindrical**.
3. For Component 1, choose the rocker face shown.
4. For Component 2, choose the pivot’s edge shown. Be sure to select the edge and not the face.
5. Click **OK**.



Step 5 – Add a cylindrical joint

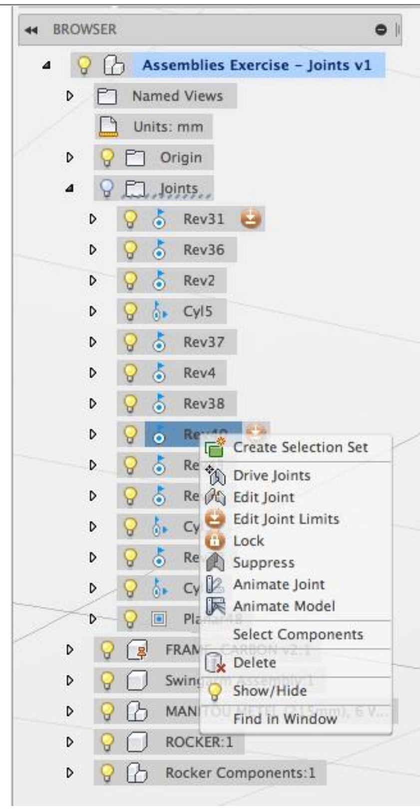
1. Right-click and select **Repeat Joint**.
2. Make sure the Motion Type is set to **Cylindrical**.
3. For component 1, choose the rocker face shown.
4. For component 2, choose the bolt face shown.
5. Click **OK**.





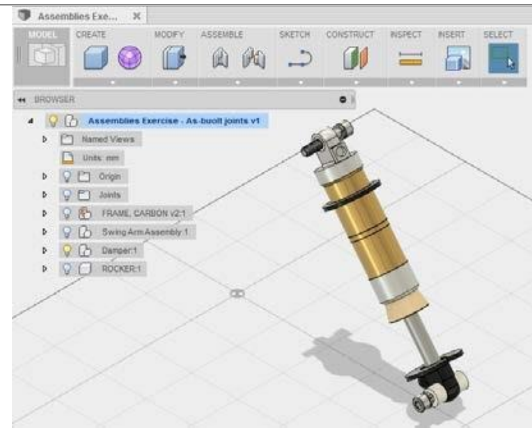
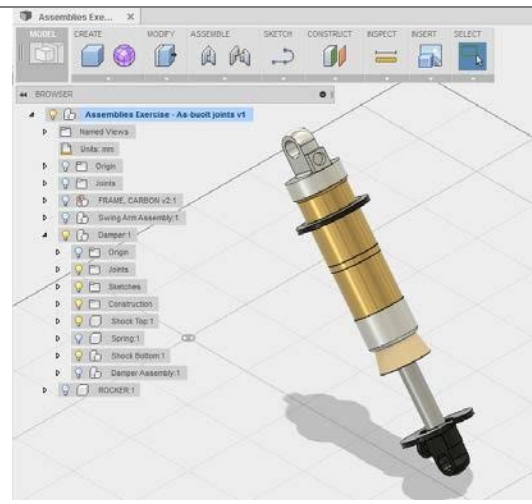
Step 6 – Visualize joints

1. Hold **Alt** and drag the swingarm. Notice the joints are honored as the parts move throughout space.
2. You'll also find folders in the browser that contain joints in the global space, as well as within component groups. The light bulbs next to the folders and individual joints will show/hide the joint icons in the model, but will not suppress the joints.
3. Right-click on a joint to suppress it, edit it, definite its limits, animate it, or lock it.



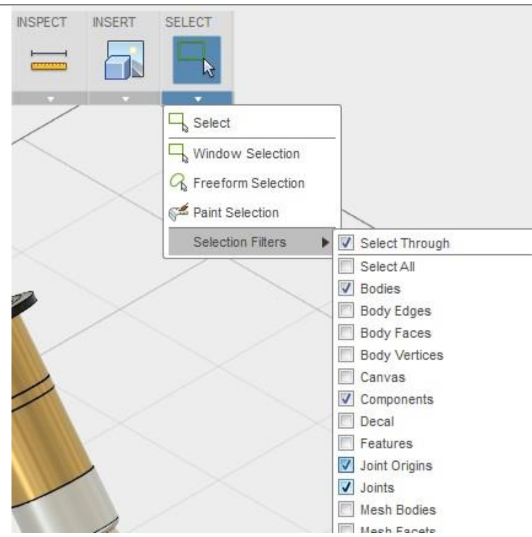
31.As-built Joints

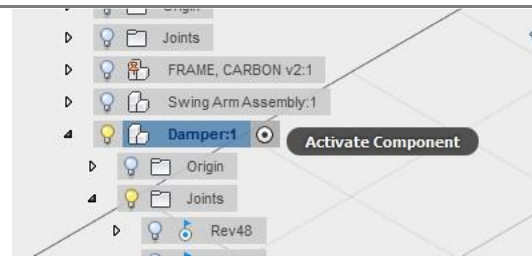
An as-built joint is used in the case of imported geometry or top-down design when the components to be constrained are in the correct positions relative to each other, i.e. they don’t need to be moved. An as-built joint maintains the position, and defines the relative motion.

Step 1 – Open the design and set visibility 1. Open **06_As Built Joints**. 2. Use the light bulbs to turn off the visibility of the **Frame, Carbon** and the **Swing Arm Assembly** so that you only see the Damper	
Step 2 – Component structure 1. Expand the **Damper** component group and you will see that it is made of several different components and components groups itself. 2. We are going to constrain these components in order to define how the damper will function.	

Step 3 – Set your selection

1. Make sure your selection filters are set to select only **Bodies**, **Components**, **Joint Origins** and **Joints**. This will make it easier to select the right components to constrain.
2. Hover over **Damper** in the browser then click the dot next to the name to activate the component. Once it is activated, the text should appear **Bold**.

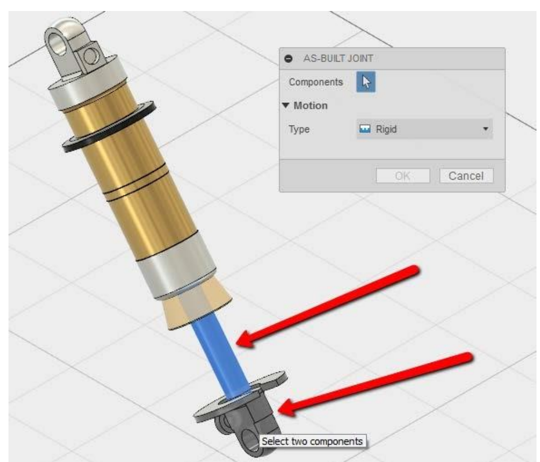




Step 4 – Create a rigid joint

The first joint we want to create is a Rigid joint between the Shock Retainer and the Shaft.

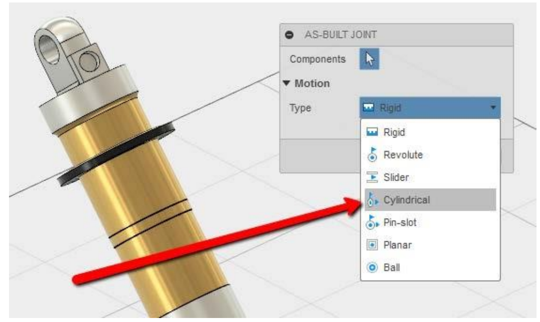
1. Click **Assemble > As-built Joint**.
2. Select the **Shaft** and the **Shock Retainer** as the two components (you can do this in the window or the browser under the Shock Bottom component group). Make sure that the Type of joint is Rigid, and you will see an animation that depicts the Rigid joint
3. Click **OK**.

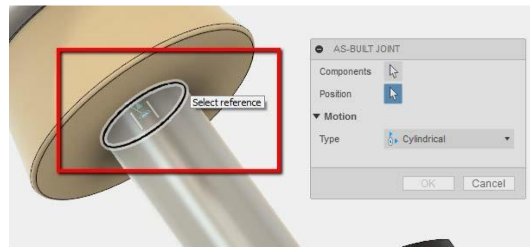


Step 5 – Create a cylindrical joint

The next joint we will create is a cylindrical as-built joint between the Shock Top and the Shaft.

1. Click **Assemble > As-Built Joint**.
2. Set the Type to **Cylindrical**.
3. Select the **Shaft** and the **Shock Top** (in that order) as the two components.
4. Select the profile of the shaft where they both meet as the Position reference.
5. You will see an animation of how this join will function. If this looks right, click **OK**.





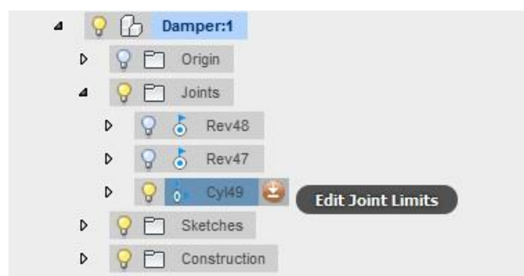
Step 6 – Joint limits

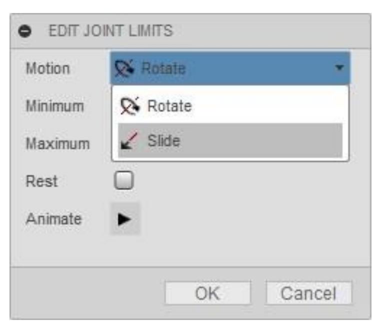
The next thing we want to do is define limits for this cylindrical joint.

1. Hover over the joint you just created and click the **Edit Joint Limits** icon. You will need to adjust these limits until you get to the appropriate limits which best define the motion of the damper.
2. Set the Motion to **Slide**.
3. Check the boxes for **Minimum** and **Maximum** limits.
4. Enter a Maximum limit of **30 mm** and you will see the Shaft move in to represent that limit. If this looks right, click **OK.**

If the shaft moved in the opposite direct (away from the Stock Top), change the minimum limit to -30 mm and the maximum limit to 0 mm.

You will now be able to click and drag the Shock Retainer (under Shock Bottom) and will be able to see the motion of the damper based on the joints you just set up.





32.Contact Sets

Contact sets designate which components do not interfere once they contact each other. A contact set can be used to define the limits of motion allowed because the motion stops when the components come into contact.

<table>
<tr>
<td>
Step 1 – Continue with 06_As built joints

1. Continue working with the **06_As built joints** design.
2. Make sure the visibility of the **Frame, Carbon** and the **Swing Arm Assembly** is turned off so that you only see the Damper.
3. Make sure Damper is activated.
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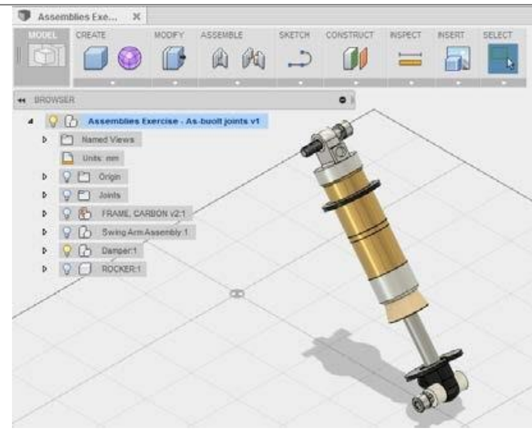
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<td>
Step 2 – Remove max joint limit

1. Hover over the cylindrical joint in the joints folder and click the **Edit Joints Limits** icon.
2. Set the Motion to **Slide**
3. Uncheck the **Maximum** check box. Leave the minimum limit as 0.00 mm.
4. Click **OK**.
5. Now when you drag the Shock Retainer upwards, you will see that it interferes with the Shock Top since there is no maximum limit of motion. We need to set up a contact set, so that when the Shock Retainer contacts the Shock Top, it knows that it cannot go any further.
6. **Undo** any movement you might have done.
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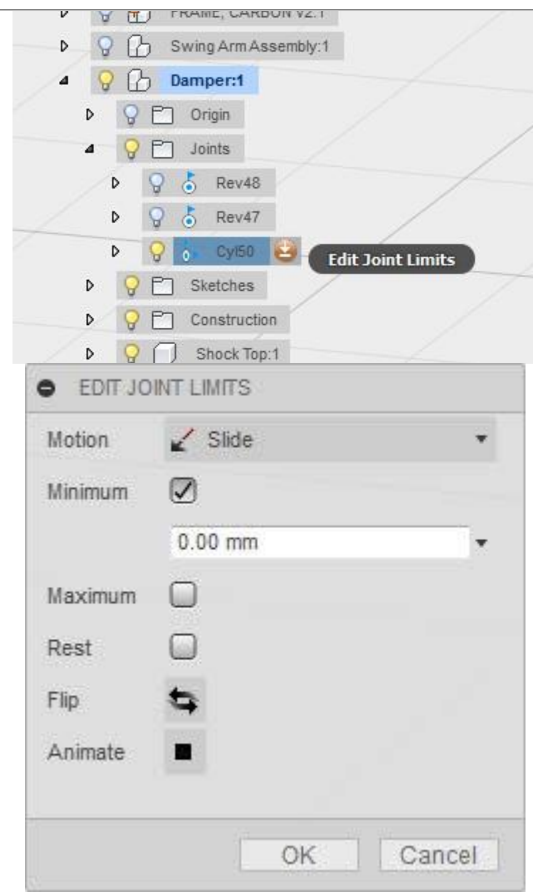

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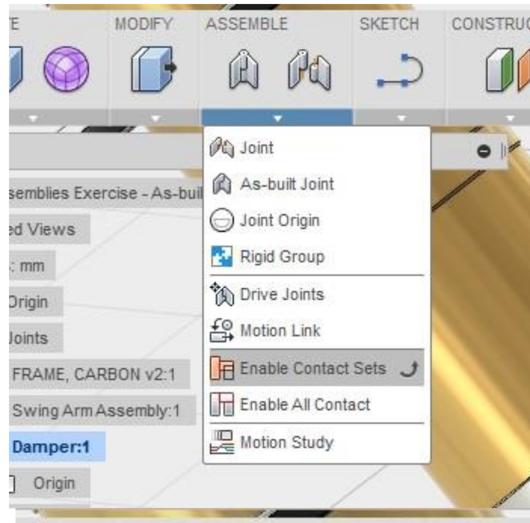
</table>

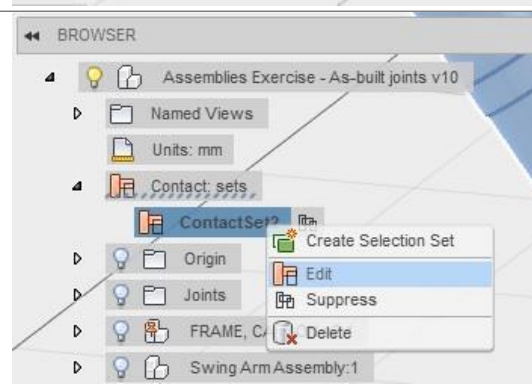
Step 3 – Set up contact

1. Click **Assemble > Enable Contact Sets**.
2. Click **Assemble > New Contact Set**. The New Contact Set dialog box displays.
3. Select the **Shock Retainer** and the **Shock Top** as the two bodies. Click **OK**.
4. Drag the Shock Retainer towards the Shock Top and you will see that it stops as soon as it comes in contact with the Shock Top.
5. **Undo** any dragging to return the components to their original position.



Step 4 – Edit the contact set

1. To Edit or Suppress the contact set at any point, right-click on the contact set in the Browser and click **Edit** or **Suppress**.



33.Motion Study

A motion study in Fusion 360 allows the user to animate the motion of the design based on the joints and limits placed.

<table>
<tr>
<td>
Step 1 – Open the design

1. Open **06_Motion Study.**
2. Use the ViewCube to view the **Front** view.
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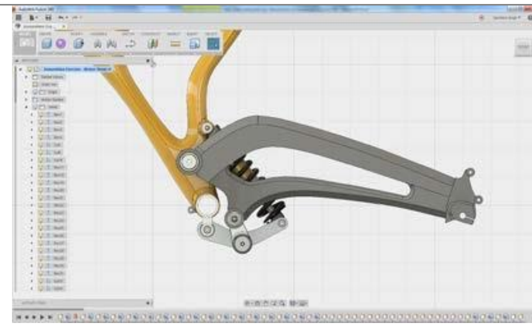
<td>

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<td>
Step 2 – Start a motion study

1. Click **Assemble > Motion Study**. The Motion Study dialog box displays.
2. The first thing you need to do is select a joint to animate. Expand the Joints folder, and pick **Rev31** as the joint to animate. The joint is added to the list of joints in the Motion Study dialogues box.

In the Motion study dialog box, you create a chart of the motion to be animated. The X Axis on this chart denotes steps or time. The Y Axis denotes the extents of the motion. In the case of a revolute joint the Y Axis denotes degrees, but in the case of a slider joint, it denotes distance in in/mm.
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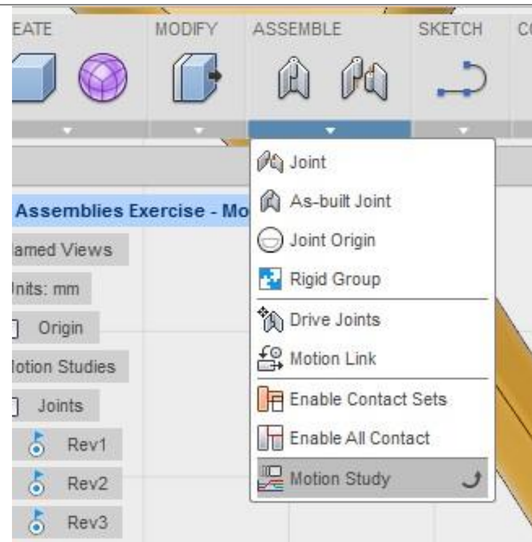

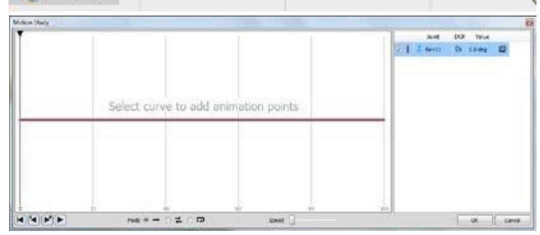

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<table>
<tr>
<td>
Step 3 – Set up motion

1. Click on the line to add a point.
2. Enter these values:
Angle: 5
Step: 10
3. Click another point on the line.
4. Enter in the following values.
Angle: 10
Step: 15
5. Continue putting in the following values for the next 5 points Angle: 2 Step: 25
Angle: 7 Step: 30
Angle: 12 Step: 40
Angle: - 3 Step: 45
Angle: - 7 Step: 50

The goal of putting in these random numbers is to simulate the motion of the bike as it goes over different terrain.
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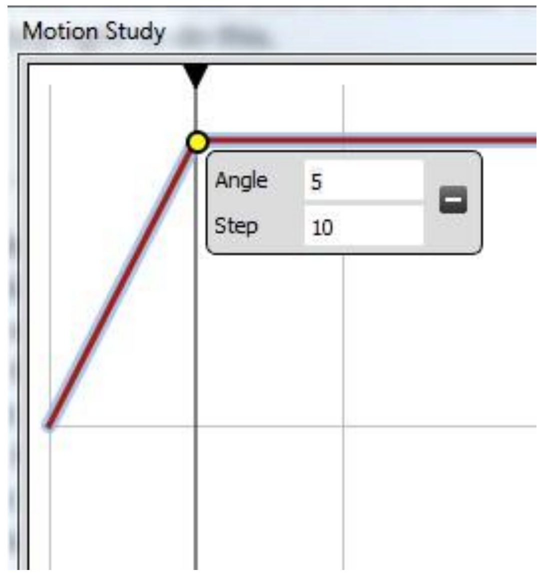

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<td>
Step 4 – Play the motion

1. Your chart will look like the one on the right.
2. Change the Mode to Loop
3. Click the Restart button to bring the counter back to Zero.
4. Click Play.

You will see the complete assembly reacting to the motion that you just built in. All the joints in the assembly are working to drive this motion. A motion study is a way for the user to verify the function of their designs.
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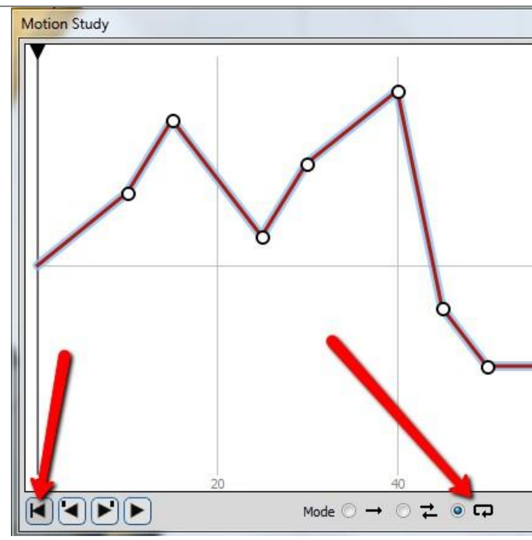

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Top-down Design Methodology

Fusion uses a top-down design approach, which is essentially the breaking down of a system to gain insight into its compositional sub-systems. In a top-down approach an overview of the assembly is formulated, specifying but not detailing any base level parts. Each subassembly and part is then refined in yet greater detail, sometimes in many additional levels, until the entire specification is reduced to base elements.

In top-down assembly design, one or more features of a part are defined by something in an assembly, such as a layout sketch or the geometry of another part. The design intent (sizes of features, placement of components in the assembly, proximity to other parts, etc.) comes from the top (the assembly) and moves down (into the parts), hence the phrase "top-down".

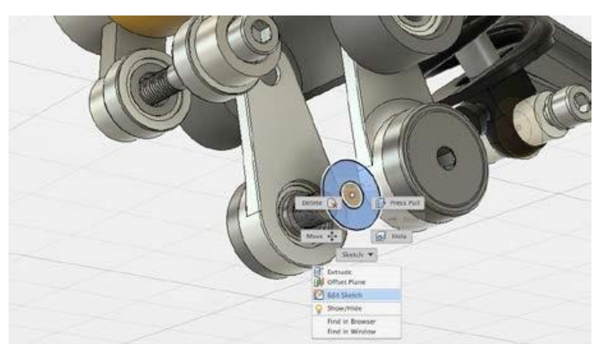
34.Using existing geometry to drive sketch curves

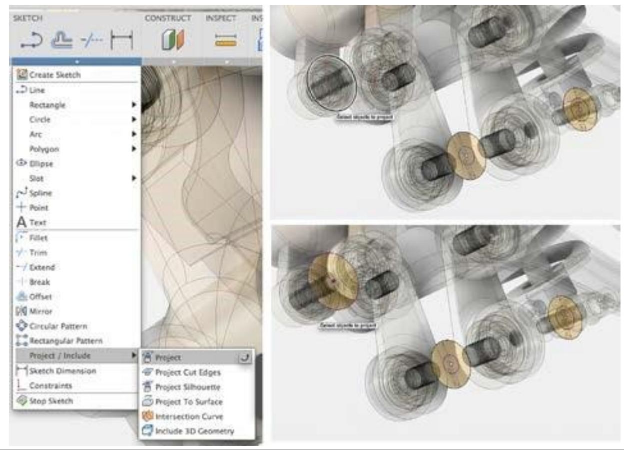
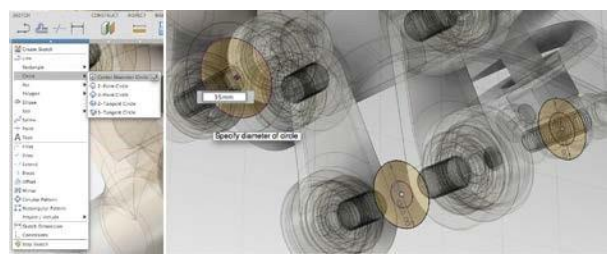
In this exercise, we'll be designing a rocker and applying as-built joints to dynamically connect it with other parts of the assembly. We'll be working with existing geometry from existing components to sketch and extrude a new component. By designing in one space alongside existing components, we eliminate the need to toggle back and forth between part files, as we can easily drive component features based on the existing assembly. We can also easily add an as-built joint to define the relationship between the parts.

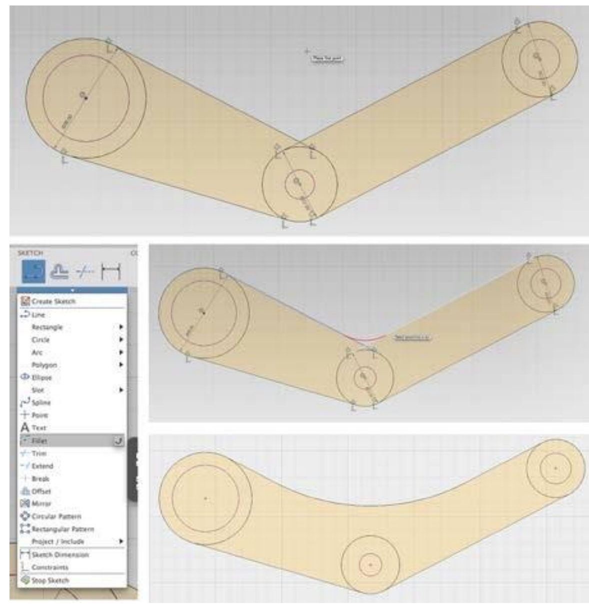
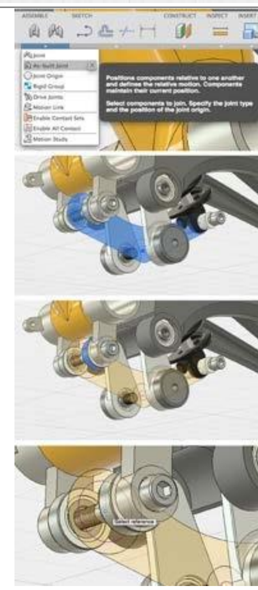
Step 1 – Edit the sketch

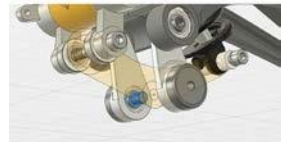
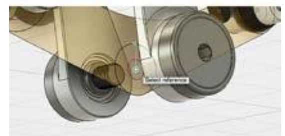
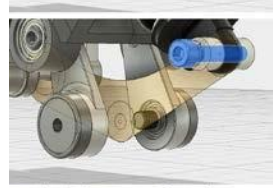
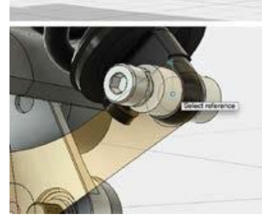
1. Open the **06_Top Down Design** design.
2. Right-click the visible sketch at the bottom of the assembly.
3. Choose "Edit Sketch."

We'll being using the provided sketch, but adding sketch curves based on the geometry of other parts of the model.



Step 2 – Project geometry 1. Click the **Home** icon on the ViewCube to orient the model as shown. 2. Click **Sketch > Project/Include > Project**. 3. Hover over a circular profile from one of the bolts or bearings that align with the hinge of the rocker. You'll notice the preview of the projection in red. Click to project the circular profile. Project will take a profile of any outside component, body, sketch, or construction feature and project its profile onto the current sketch. • Note that the sketch profile projected is purple. This indicates that it is locked in and defined by geometry outside of the sketch.	
Step 3 – Build off of the project geometry 1. Click **Sketch > Circle > Center Diameter Circle**. 2. Use the center of the projected circle as the center for a new circle. 3. Type in a diameter of **35 mm**. We created a relationship between the sketch circle and the body used to project the geometry.	

Step 4 – Complete the Rocker Profile 1. Click FRONT on the ViewCube. 2. Right-click the **Rocker** component in the browser, and choose “Isolate” to hide the other components. 3. Click **Sketch > Line**. 4. Draw tangent lines between the three circles, as shown at right. To automatically create the tangent constraint, pick a point on one circle then hold **Shift** and click a point on another circle. 5. Click Sketch > Fillet. Select the top two lines. Use a radius of **77 mm**. 6. Click **Stop Sketch**. 7. Right-click on **Rocker** and select **Unisolate.** Click Sketch > Constraints to manually apply constraints. See module on Sketching for more details.	
Step 5 – Add revolute as-built joint 1. Click **Assemble > As-Built Joint**. 2. Set the Type to **Revolute**. 3. For the two components, click once on the sketch, and once on the bearing. 4. For the position, click the center point of the sketch circle. 5. Click **OK**.	

Step 6 – Add cylindrical as-built joint 1. Click **Assemble > As-Built Joint**. 2. Set the Type to **Cylindrical**. 3. For the two components, click once on the sketch, and once on the bolt. 4. For the position, choose the center point of the sketch circle. 5. Click **OK**.	  
Step 6 – Add a revolute as-built joint 1. Click **Assemble > As-Built Joint**. 2. Set the Type to **Revolute**. 3. For the two components, click once on the sketch, and once on the bolt. 4. For the position, choose the center point of the sketch circle. 5. Click **OK**.	  

35.Extrude the sketch and interface with other parts

Now that we have a sketch and some as-built joints, we can extrude the sketch into a 3D rocker.

<table>
<tr>
<td>
Step 1 – Extrude the sketch into a body

1. Continue with 06_Top Down Design.
2. Hover over Rocker and click the dot next to the name to activate the component.
3. Click the Home view on the ViewCube.
4. Click Create > Extrude.
5. Click on all profiles of the sketch.
6. Set Direction to Symmetric.
7. Set the Distance to 24.125 mm.
8. Set the Operation to New Body.
9. Click OK.
10. If the body is not in the “Rocker” component, be sure to find it in the browser and drag it into the “Rocker” Component.
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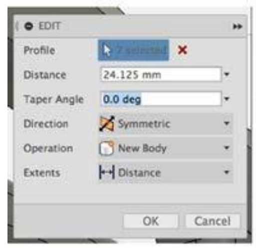



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<td>
Step 2 – Extrude-cut a split in the rocker

1. Right-Click Rocker and select Isolate.
2. Click the light bulb next to the Split sketch to make it visible.
3. Click Create > Extrude.
4. Select the profile in the Split sketch
5. Set the Operation to Cut.
6. Set the Extents to All.
7. Click OK.

The Split sketch contains projected geometry from other components. This is another use of topdown design in the sketching environment.
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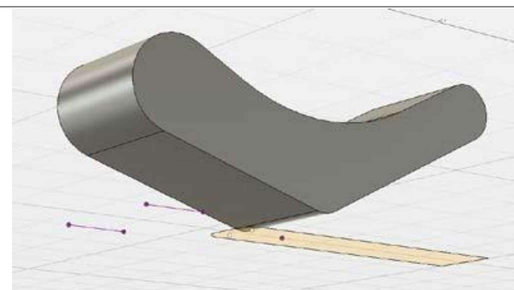
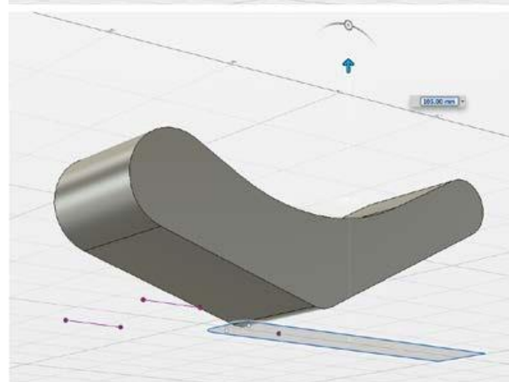

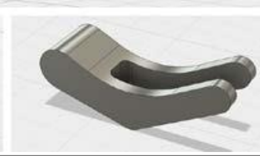
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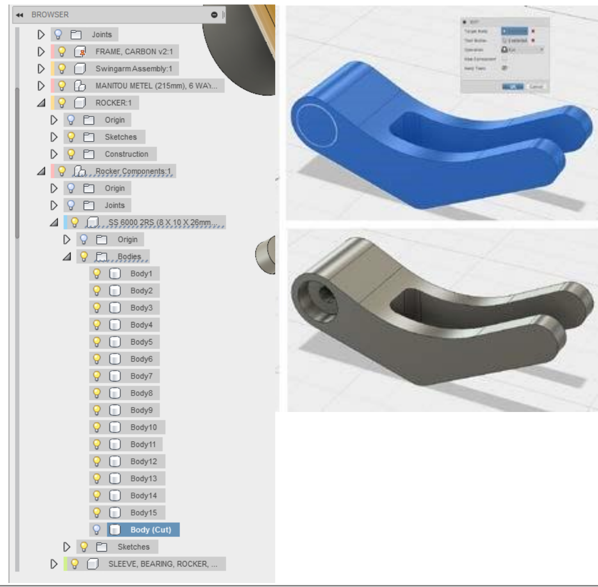
Step 3 – Boolean operation

1. Click **Modify > Combine**.
2. Select the rocker body as the Target Body.
3. Select Body Cut in the browser for the Tool Body. (Component SS 6000 2RS)
4. Set the Operation to **Cut**.
5. Click **OK**.
6. Right-click on Rocker and select **Unisolate** to display all components.



Challenge: Add two additional holes to the rocker based on the provided geometries and add any filets to complete the rocker in the assembly. Add a brushed steel appearance, and render the full assembly.

Rendering

Overview

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.

Learning Objectives

In this section you will learn how to:

- Assign materials to your model
- Edit and replace materials
- Place decals on your model
- Change the environment settings and background color
- Change environment effects
- Render an image using the Real Time Ray Tracer
- Create images using the A360 Cloud Render feature

36.Open Fusion360 file and go to Render Workspace

In this section you will open the design file for the Utility Knife and go the Render workspace so that you can apply materials.

<table>
<tr>
<td>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. The Data Panel will slide open.</td>
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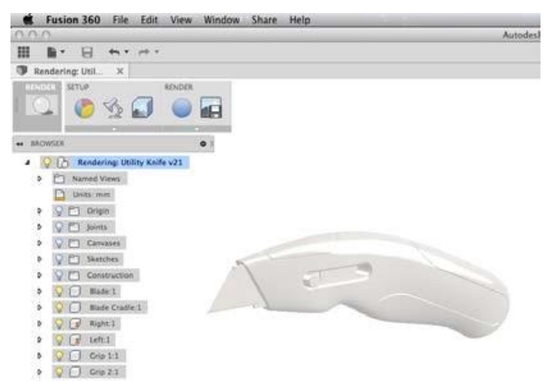
</tr>
<tr>
<td>Step 2 – Open the design

1. At the top right of the Data Panel, select the project from the project drop down list.
2. Double-click on the design called 07_Rendering_UtilityKnife to open the design in Fusion 360.
3. When the design has opened in your modeling window, click on the icon to close the Data Panel.</td>
<td></td>
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Step 3 – Go to Render Workspace

1. Click on the Model icon in the left of the toolbar to view other available workspaces.
2. Select the **Render** workspace.

You may notice that the environment changes slightly, this happens because the environments you use for modeling are different than the environments that have been created specifically for rendering.

The Rendering workspace toolbar contains tools to **Setup** your render and to create a **Render**.

In the **Setup** section you have access to

- Appearance materials library to apply materials to your design
- Environment settings to change the background and how lights are cast on your design
- Decal tools for applying decals to your design

In the **Render** section you have access to

- Start/Stop a RRT Render
- Save an image of your design

37.Apply Materials

Now that the model is in the Rendering Workspace you can begin assigning **appearance materials**. There are two types of materials in Fusion360:

Physical Materials dictate what the object is made of and is used in mass calculations. In the absence of any appearance materials that have been applied to your design, you will see the default physical material. The default physical material can be changed in your preferences.

Appearance Materials dictate how the object will look when rendered.

Note: in absence of Appearance material the Physical material will be shown in the render.

<table>
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<td>

Step 1 – Open Appearance tool

1. In the **Render Workspace** click on **Setup > Appearance.**
2. The Appearance dialog box opens.

Appearance dialog box has several sections to it:

Apply To – Allows you switch between applying materials to bodies/components or to individual faces

In This Design – Shows which materials have been assigned to parts of your design.

Library – In this section you can switch between the new Fusion 360 Appearance Library and your personal Favorites Library. It also contains the folders and sub-folders of materials broken down by common categories and example swatches of the materials.

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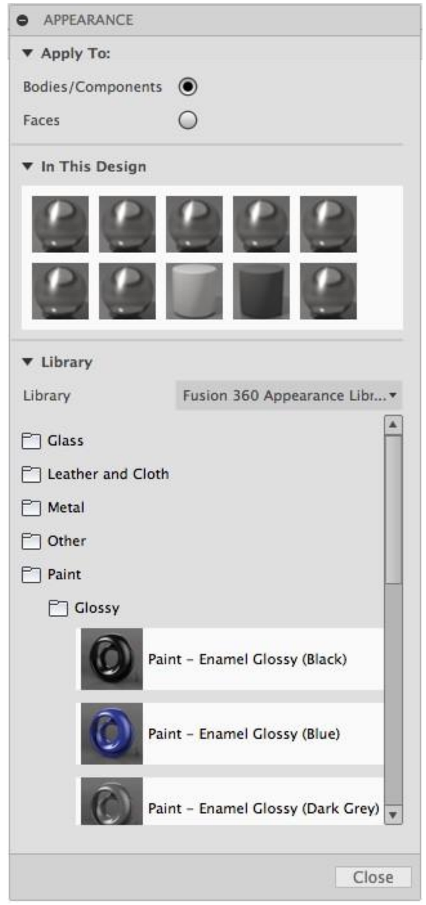

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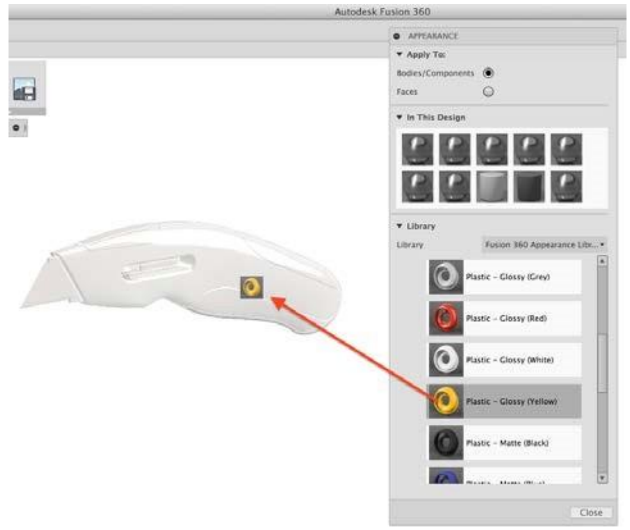
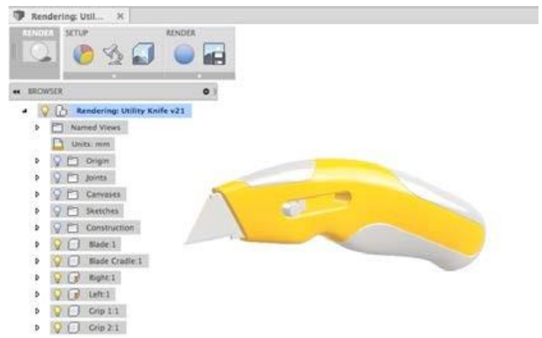
</table>

Step 2 – Apply a Material directly to geometry

1. In the Appearance dialog box scroll down to **Plastic > Opaque > Plastic – Glossy (Yellow).**
2. Click and hold on the **Plastic – Glossy (Yellow)** swatch icon and drag it on to the main side body of the utility knife.
    a. The material on the part changes to **Plastic – Glossy (Yellow)**
3. Repeat these steps so that **Plastic – Glossy (Yellow)** is assigned to both sides of the utility knife.

In the Appearance dialog box you will notice that there is only one swatch for Plastic – Glossy (Yellow) even though that material has been assigned to two separate bodies. 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. Editing the one material will affect all of the bodies that have that material assigned.

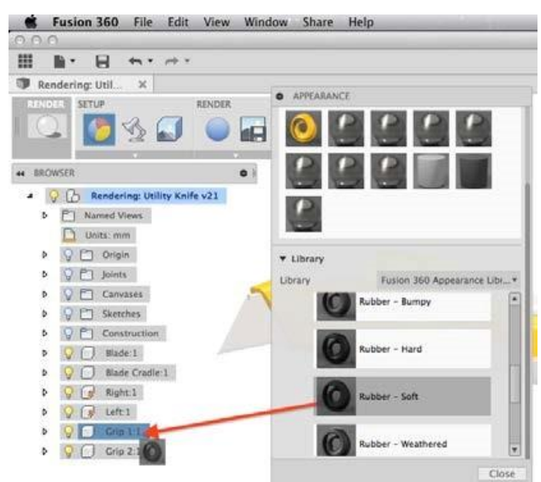




Step 4 – Apply a material to a body in the browser

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. Click and hold on the **Rubber – Soft** swatch icon and drag it on to **Grip 1:1 > Bodies > Body 1** in the browser
    a. The material on the grip changes to **Rubber – Soft.**

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



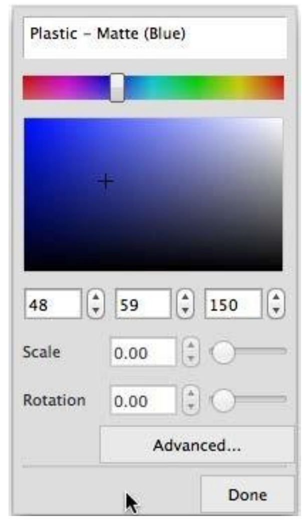
Step 5 – Apply additional materials 1. Using the method of your choice apply the following materials to the parts listed • **Plastic – Textured – Polka** to **Grip 2:1** • **Plastic – Translucent – Matte (Blue)** to **Blade Cradle:1** • **Metal - Stainless Steel – Satin** to **Blade:1** If you apply 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.	

38.Editing Materials

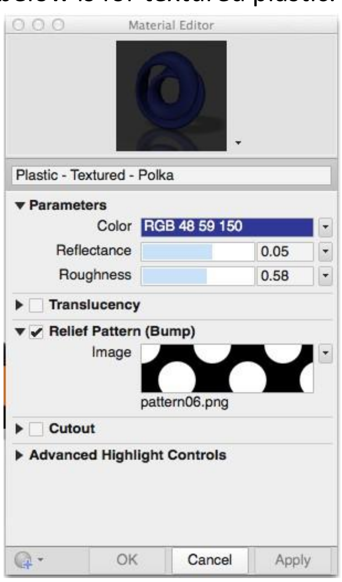
Now that you have all the base materials applied to your design, you can customize the materials to look the way you want.

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



The options in the advanced editor will differ depending on the material you are editing. The example below is for textured plastic.



Texture Mapping is a process in which a 2-D image called a *texture map* is wrapped around a 3-D object. In the physical world this is similar to applying wallpaper or veneer to a real object. The texture map can be used to change various properties of a material including the color of the material as is seen in the Wood materials, or the way highlights hit a surface as in the Rubber materials.

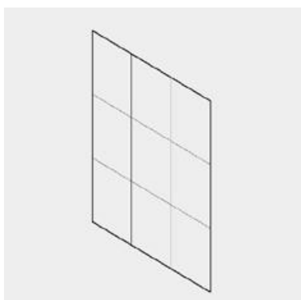
Bump Mapping is a technique for giving a 3-D surface the appearance of deformities and depth (e.g.: wrinkles or bumps). Although a surface that has a *bump map* applied will appear to have real depth, the surface of the underlying object is not actually changed. The textured plastic materials all use bump maps.

You can use the scale and rotate tools in the material editor to change size and orientation of the images that control the texture and bump results.

Projection Mapping is a method for placing one texture across several separate faces in order to give the illusion that it is one solid or continuous object. The texture map is applied to the projection type and "pushed through" the surfaces it projects on to. Fusion 360 has 4 projection types to choose from:

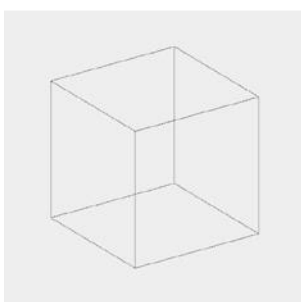
Planar

The texture map is projected from a plane in a user defined direction



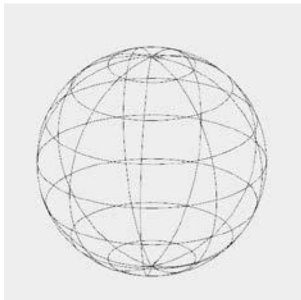
Box

The texture map is applied to a cube that surrounds the object and creates 6 planar projections



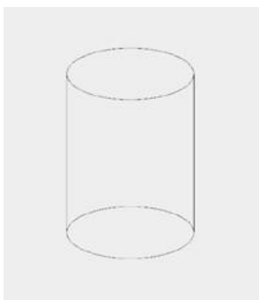
Sphere

The texture map is applied to a sphere that surrounds the object. This projection type will show a poll pinch point depending on the axis defined by the user.



Cylindrical

The texture map is applied to a cylinder that surrounds the object.



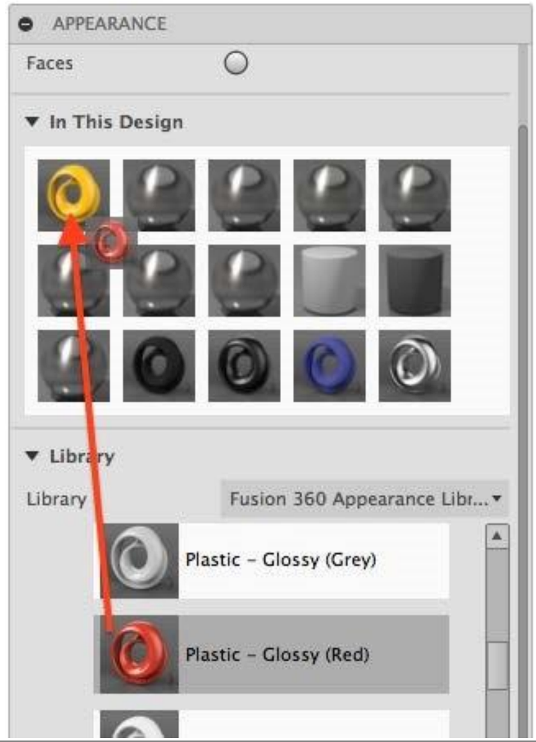
To change the way the maps are projected on the surface right-click the body of the object in the browser and select **Texture Map Controls** from the drop down menu.

In this section you will swap out one material for another and change the parameters for several of the materials.

Step 1 – Replace the Yellow Plastic Material

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.

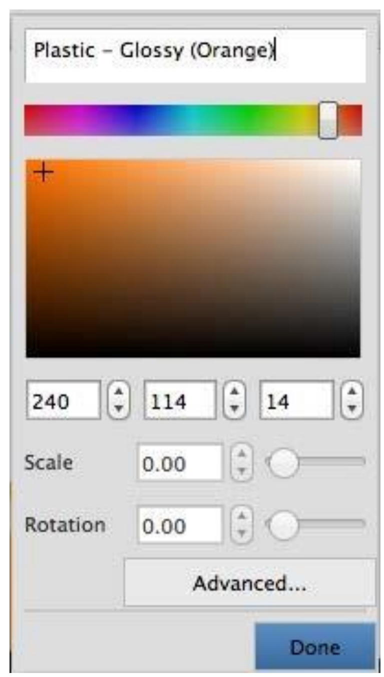
All the bodies that had Plastic – Glossy (Yellow) applied now have Plastic – Glossy (Red) applied. This is a quick method for swapping out materials in your design.



Step 2 – Edit Plastic – Glossy (Red)

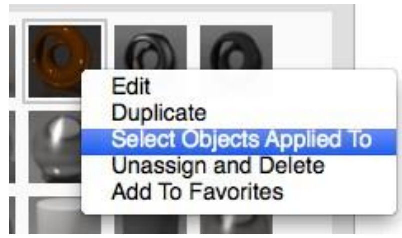
1. Double click on the swatch for **Plastic – Glossy (Red)** in the "In this design" section of the Appearance dialog box to open the editor window for this material.
2. At the top of the dialog box is the current name assigned to the material – double click in the name field and change the name to **Plastic – Glossy (Orange)**
    a. This will make it easier to find the material later if required.
3. In the middle of the dialog box there is section that allows you to enter RBG values for a specific color.
4. Enter **240, 114, 14** to change the color of the plastic material to orange.
    a. You can also use the sliders to change the base color and the tone of the color.
5. Click the **Done** button.

You should now see the swatch for the edited material in the **In This Design** section of the



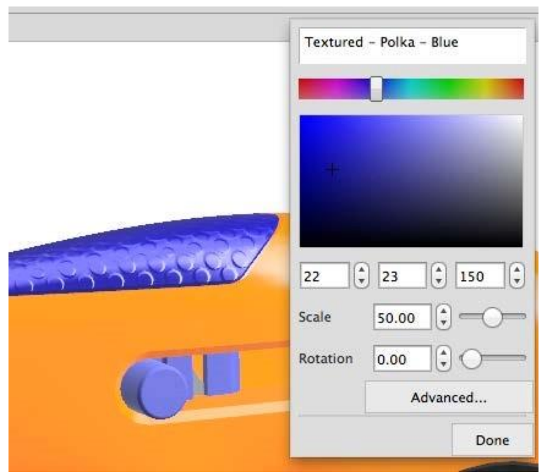
Appearance dialog box. If you hover over the swatch the modified name will appear.

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.
    a. If you wish to match the blue of the blade cradle enter the RGB values of **48, 59, 150**
3. Change the scale of the texture map to **41**.
    a. The slider can be used to interactive change the size of the texture map.
4. Move the slider next to Rotate to interactively change the orientation of the texture map. When you are satisfied with the orientation, click **Done**.

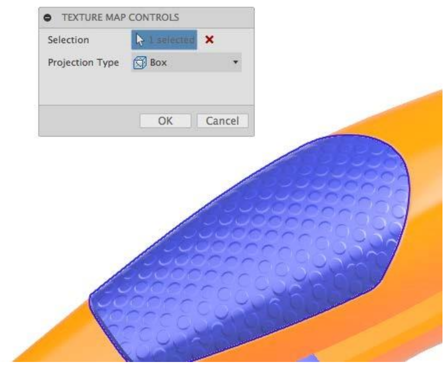


Step 4 – Change the texture projection method for Grip 2:1

Texture Map Controls are used to change the type of projection method used on the object with a texture map. Fusion 360 will automatically choose the best projection method when a material with a texture map is applied, if you are not satisfied with the look of the texture you can change the projection setting manually.

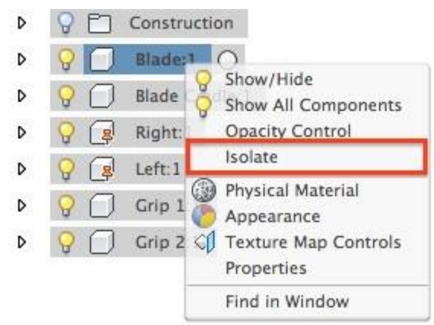
1. Close the **Appearance** dialog box by clicking on the **Close** button.
2. In the browser, locate the component labeled Grip 2:1 and right-click on it.
3. Select Find in Window to bring the part in the center of the screen.
4. Zoom in further so that you can clearly see the texture map on the surface.
5. Right-click on component Grip 2:1 and select Texture Map Controls from the dropdown menu to open the Texture Map Controls dialog box.
6. In the dialog box, change the Projection Type to Box
7. Click OK to accept the change.

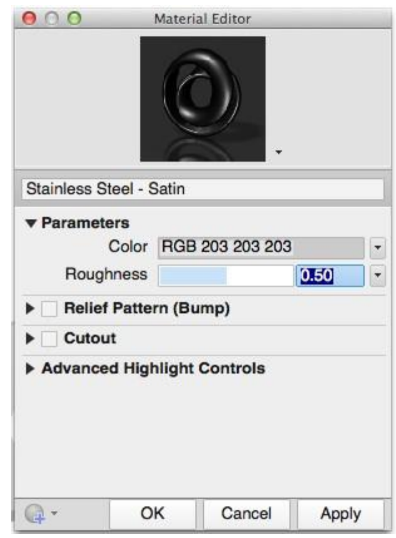
Try selecting the other projection types to see the different results. In some cases you may have to choose an axis for a projection direction. In those instances an axis widget will appear, simply click on the axis that best matches the direction you would like to project.



Step 5 – Edit and duplicate a material

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.
    a. All of the other components disappear leaving Blade:1 in the window.
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**.
10. Right click on **Stainless Steele – Blade** and

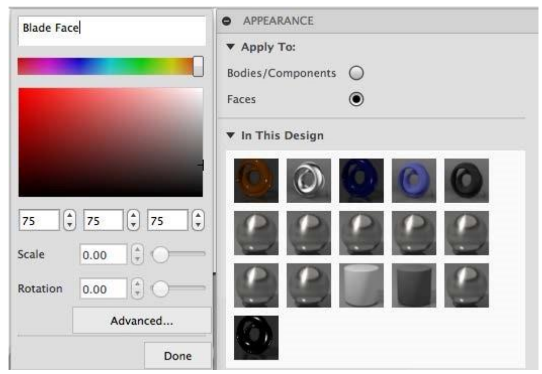




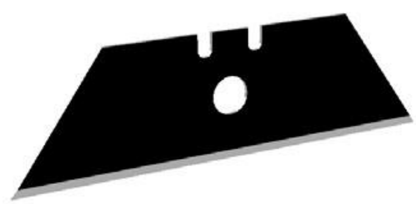
select **Duplicate** from the drop down menu.

a. This creates a second material called Stainless Steele – Blade that has the exact same settings as the original.

11. Double click on the copy **Stainless Steele – Blade** material to open the Material Editor.
12. Change the name of the material to **Blade Face**
13. Change the color to **75,75,75**.
14. In the Appearance dialog box change the **Apply To:** setting from **Bodies/Components** to **Faces**
    a. Now you can only apply materials to selected faces on a body/component



15. Drag the Blade Face material to the side face of **Blade:1**
16. Repeat this step on the other side of **Blade:1** so that both side faces have Blade Face applied.
17. Close the **Appearance** dialog box.
18. Right-click on **Blade:1** in the browser and select **Unisolate** to show the rest of the design.



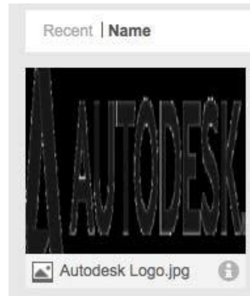
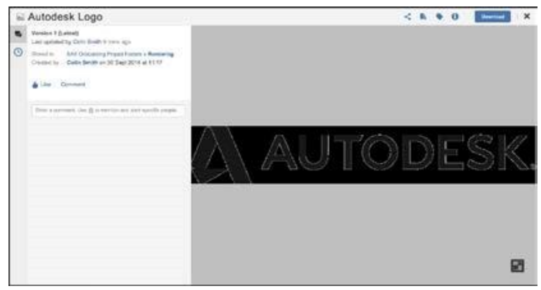
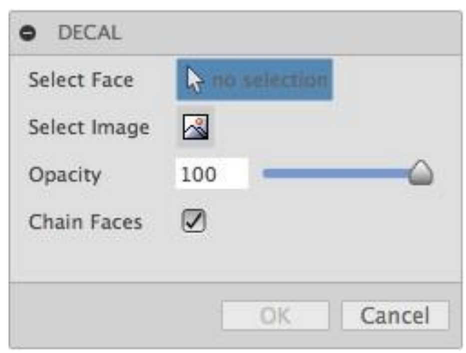
NOTE: If you apply a material a body that already has a material(s) applied to one or more of it's faces you will be presented with a choice to keep or remove appearances applied to the faces:

Keep - all of the existing materials on the face will not be replaced with the new material you applied to the body.

Remove - all of the materials on the faces will be replaced with the new material you applied to the body.

39.Apply A Decal

In this section you will apply an image of the Autodesk logo to the body of the utility knife using the decal tool.

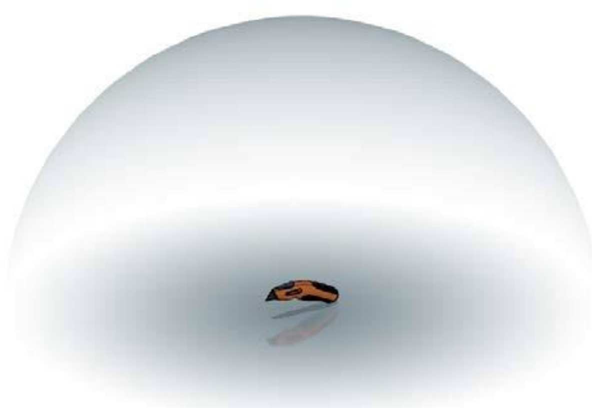
Step 1 – Download the image file 1. In the **Data Panel** find the item named **Autodesk Logo.jpg.** 2. Double-click on the thumbnail to launch A360. 3. Click on the blue **Download** button on the upper right side of the window. 4. Place the file in a location where you can easily find it. Your desktop for example.	 
Step 2 – Apply the decal 1. Select **Setup > Decal** a. The Decal dialog box opens. 2. Click on the body of the utility knife to highlight it. This is the surface you will apply the decal to. 3. In the **Decal** dialog box, 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**. a. The image appears on the face you selected with a move manipulator similar to the one you use for moving objects. b. The Decal dialog box also expands to show additional inputs for distance, angle and scale.	 

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 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.
    a. You can also input **0.85** in the **Scale Plane XY** field in the **Decal** dialog box.
4. Click **OK** to accept the decal location.

40.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 surrounds the 3-D model. These images reflect in the surface of your model and are used to simulate lighting.

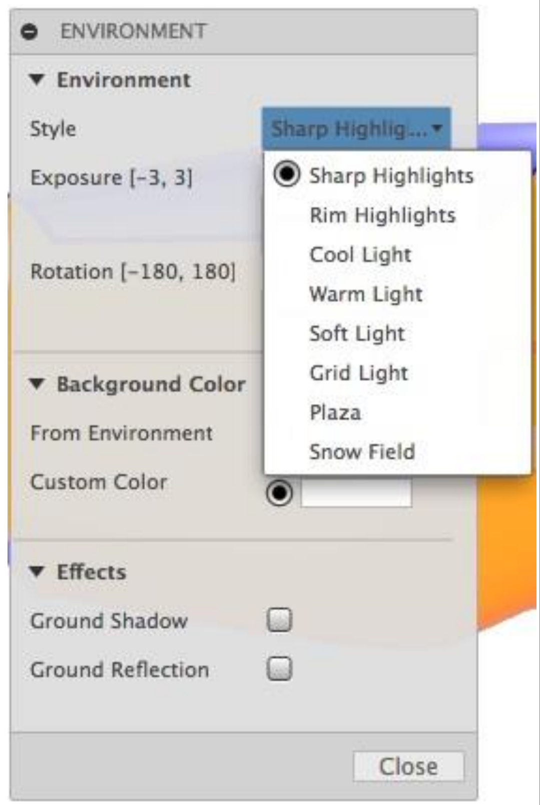


In this section we will choose the environment, change the background color and turn on effects.

Step 1 – Change the Environment settings

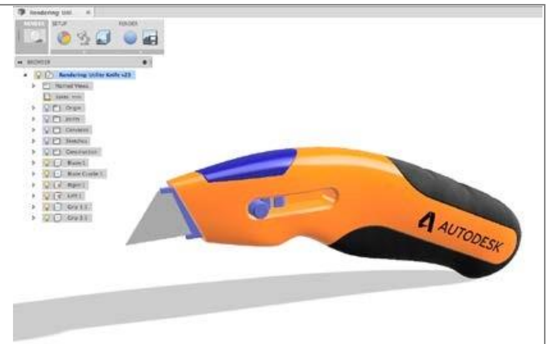
In the Environment section you can pick the type of environment map you want to use. There are eight maps to choose from. Six of them are designed to simulate a photo studio set up. The remaining two (Plaza and Snowfield) are HDR photos of outdoor environments.

5. Click on **Setup > Environment**.
    a. There are 3 main sections to this tool:
        i. Environment
        ii. Background Color iii.
        Effects
6. Select several styles from the dropdown list. Notice that the reflections change as well as the pre-assigned background color for each style.
7. From the Style dropdown list select **Sharp Highlights**.
8. Use the slider next to Exposure to change the light level in the scene. Pick an exposure setting you like.
    a. If you want to go back to the default setting type 0.0 in the field next to the slider
9. Use the slider next to **Rotation** to rotate the environment image around the dome.
    b. As you move the slider you will see reflected highlights change on the design.
10. Rotate the environment until you see a highlight across the right side of the knife.
    c. You can also type **57** 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**.
    a. The background changes to white.
2. Click on the white swatch next to **Custom Color** to open the color chooser dialog box.
    a. This dialog box is similar to the color chooser in the material editor. You can pick colors by using the sliders or enter an RGB value.
3. Keep the background color as white.	

Step 3 – Change the Ground Effects



You have the option to have your design cast a shadow or to reflect your model on the ground plane of the environment.

1. Click on the button next to **Ground Shadow**.
    a. Now you can see a shadow being cast on the ground plane.
2. In the **Environment** section change the rotation value.
    a. Notice that the shadow location is changing based on the location of the light sources in the chosen environment style.
3. Click on the button next to **Ground Shadow** to turn it off.
4. Click on the button next to **Ground Reflection**.
    a. Now you can see a reflection of your design on the ground plane.

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

In the Display Settings at the bottom of your screen there are Effects setting that can affect your rendering. In particular:
Ground Plane – turning off the ground plane will turn off ground shadows and ground reflections.

Ground Shadows and **Ground Reflections** – are the same commands that are in the **Environment > Effects section**.

All the other commands in Display Settings only affect the GPU rendering in your modeling window (See Rendering section for definition of GPU rendering) and will not affect the results of Rapid Ray Tracer or Cloud renderings.

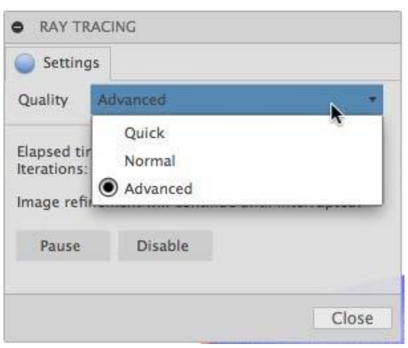
41.Rendering

Now that the design has materials applied and the environment is set correctly it is time to create a rendered image. There are 2 types of rendering methods in Fusion 360:

Graphics Processing Unit rendering (GPU) is the rendering you see in the modeling window and uses your computer's graphics card to show the materials and lighting that have been assigned to your model and is similar to the technology used in computer gaming. This method uses the least amount of resources on your computer to create the images. It shows the materials assigned to your design in real time but is not photo-realistic. The analysis tools in the **Model >Inspect** dropdown (zebra, draft and curvature map) also use GPU rendering to show the results.

Rapid Ray Tracer (RRT) in the **Rendering Workspace** can be used to create photo realistic images from your Fusion 360 models. 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.   The Real Time Ray Tracer requires you to let the rendering engine complete multiple iterations in order to get a high quality image – during this time you cannot interact with the workspace or the rendering will restart.

Under **Render – Ray Tracing** there are 3 quality settings **Quick**, **Normal** and **Advanced**

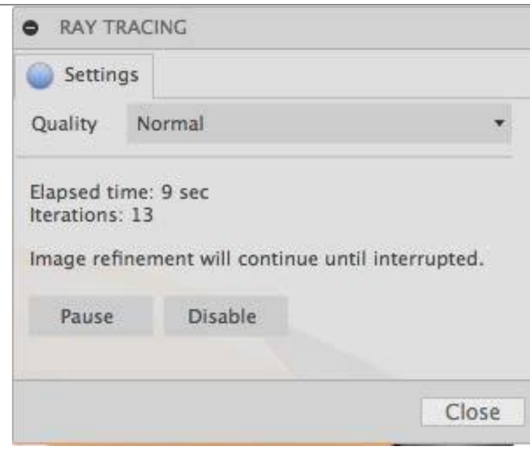


Quick: At this setting the materials in the scene are approximated as either completely polished or completely diffuse and completely ignores indirect light. In the rendered image you will see reflections but you don't get any indirect light. This can be considered as a type of enhanced GPU mode.

Normal: At this setting the materials are approximated as either completely polished or completely diffuse and the indirect light in the scene is also approximated. This setting allows you to get to a noise free image quickly and get a good view of the total light in the scene.

Advanced: This setting does a full physically based rendering with full and accurate simulation of direct and indirect light and a full simulation of all material properties. In this mode the image will start off noisy before the image has converged and can take a long time to generate a photo realistic image.

Explore Fusion 360 rendering: In this section you will change the settings of the **Render > Ray Tracing** quality settings to observe the differences, and use **Render > Capture Image** to save a PNG file to your desktop.

Step 1 – Start the Rapid Ray Tracer 1. From the Render toolbar select **Render > Enable Ray Tracing**. a. The **Rapid Ray** Tracer starts and the **Ray Tracer** dialog box opens.	 (Sample screenshot)
Step 2 – Change the Quality setting 1. Set the **Quality** setting to **Quick** a. The image clears up rapidly but the image is not high quality 2. Change the **Quality** to **Normal** a. 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. 3. Change the **Quality** to **Advanced**. a. The image is very noisy to start and will continue to clear up over time.	

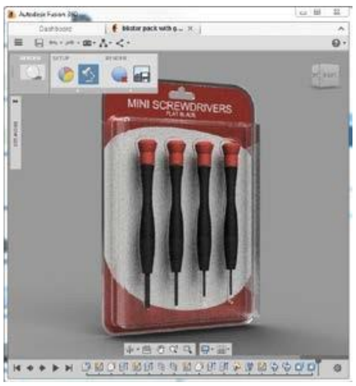
The **Rapid Ray Tracer** is a real-time ray tracer, meaning that as soon as you click on the **Enable Ray Tracing** icon your computer will start rendering the image immediately. The image will start off noisy and will start clearing up. If you change the orientation of the model or change materials and environment the Rapid Ray 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.

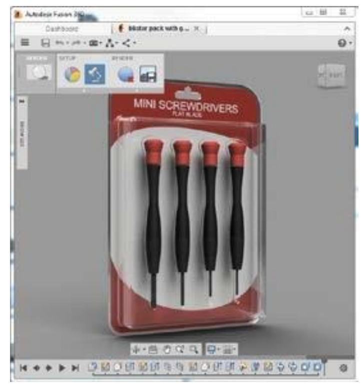
Because materials and lighting are approximated, **Quick mode** creates an image in a short amount of time.

For the **Normal mode** you usually only have to run around 10 iterations to get nice anti-aliased edges and get the noise in shadows to disappear. This will vary somewhat depending on materials and lighting but in general you would need quite few iterations. Once the image is free of noisy shadows or aliasing the image will not get better by letting it render longer.

For the **Advanced mode** the amount of time needed to create a good image will vary a lot more. Some scenes and some materials will take a lot longer to get noise free. Frosted Glass is one such material for instance. In general you usually need a couple of hundred iterations for a relatively complex scene to get totally noise free. The length of time needed is totally scene and material dependent so really complex scenes may need up to 500 iterations or more. In the **Advanced mode** you can generally see if that if an image looks noisy, it will benefit from longer rendering time.



Advanced Mode – 50 iterations



Advanced Mode – 500 iterations

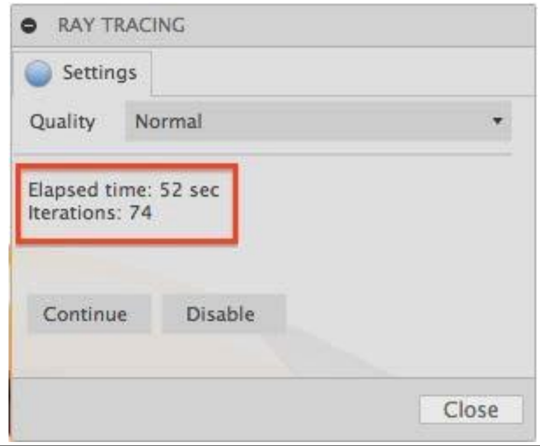
Advanced Mode – 5000 iterations

For the image above a range of 1500 – 2000 iterations should be sufficient to create a high-quality image.

Step 3 – Pause and Disable the Ray Tracer

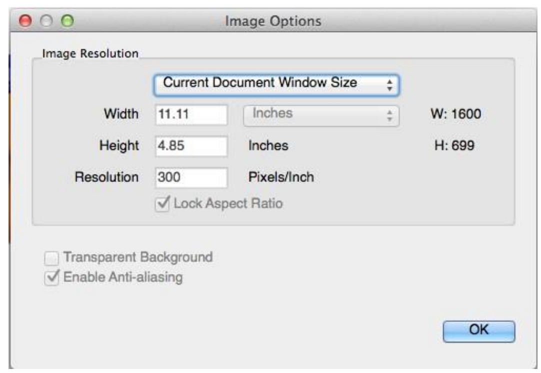
1. In the **Ray Tracer** dialog box – click on the **Pause** button
    a. The render pauses and both the **Elapsed Time** and **Iterations** stop counting.
2. Click the **Continue** button to let the **Ray Tracer** continue rendering where it left off.
3. Click on the **Disable** button to turn off the **Ray Tracer.**

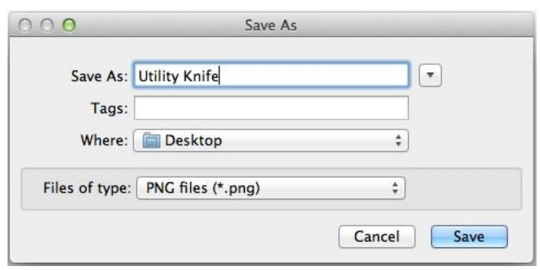
You can also click on the **Disable Ray Tracing** icon in the tool bar to disable the render.



Step 4 – Create a Normal quality Ray Trace and Capture an image.

1. Select **Render > Enable Ray Tracing**.
2. Change the Quality setting to **Normal**
    a. Let the Ray Tracer run for about **120 seconds** or until you are satisfied with the look of the rendering.
3. Hit the **Pause** button in the **Ray Tracing** dialog box.
4. Select **Render > Capture An Image**
    a. The Image Options dialog box opens
    b. In the Image Options you can change the size of the image you want to save and have an option to create a transparent background.
5. Leave the Image Options at the default and click on **OK**.
6. In the **Save As** box enter a name for the file and set a location to save the image.
7. Click on the **Save** button.





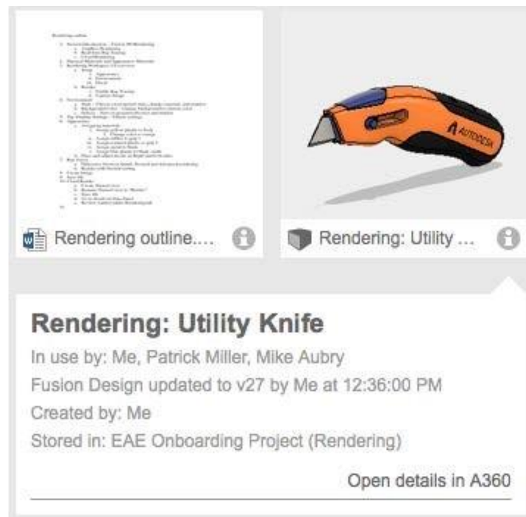
A360 Cloud Render is also a ray trace renderer that uses the cloud resources in A360 to do the image calculation rather than your desktop resources so you can continue to work while the images are being created. The cloud renderer automatically creates small sized rendered images based on the Top, Front, Right and Home named views in your browser every time you save a new version or an auto save is done.

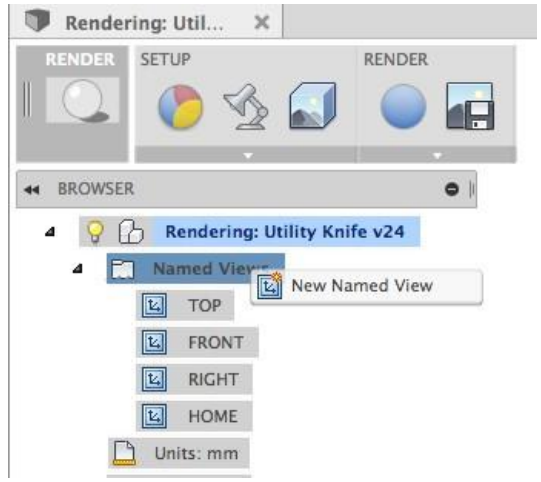
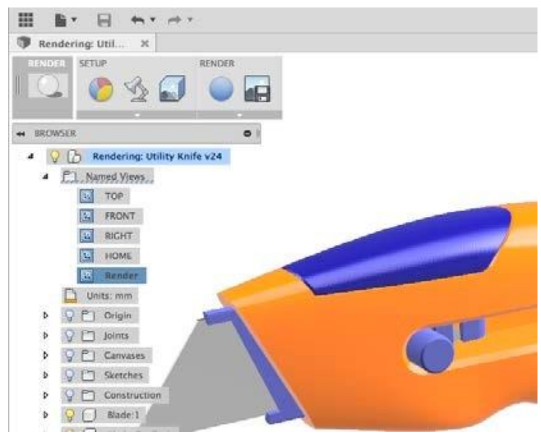
If you want to create an A360 Cloud Render of a specific view you need to create a new named view.

In this section you will locate the cloud rendered versions of the knife design and create new named view for a custom rendering.

Step 1 – Open data details in A360

1. Click on the icon in the upper left corner of the screen to open the Data Panel.
2. Find the **Rendering: Utility Knife** file in the list and click on the information icon in the bottom right.
   a. A window opens below the file to show additional information.
3. Click on **Open details in A360**
   a. Your default browser will launch and take you the item details page of the A360 project
   b. By default a lite 3D version of the model is shown
4. Click on the picture icon on the left hand side of the window to show the renderings
   a. You are now able to see the renderings that were automatically generated using the A360 cloud render. By default you can see the last version you saved. Thumbnails of the rendered named views are visible on the left side of the window. Clicking on any of them will show you larger version in the main window.
5. Hover your cursor on any of the thumbnails to show additional information about the image.
6. Close the window by clicking the **X** in the right corner.
7. Close your browser window.
8. Return to Fusion360.
9. Close the **Data Panel**

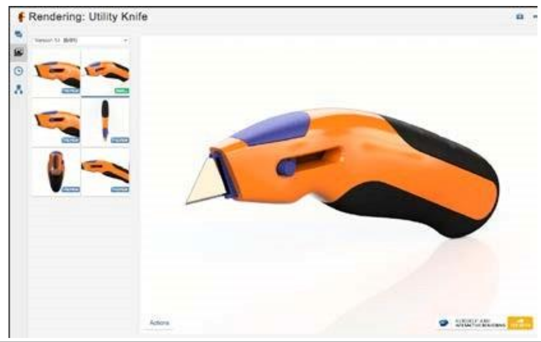


Step 2 – Create new Named View
 	
1. In the browser click on the small arrow next to Named Views to expand the list of current named views.
   a. Named Views are basically preset camera shots. By default there are Top, Front, Side and Home
2. Click on any of the named views to see that preset.
3. Arrange your design in the main window in a way that you would like your A360 Cloud Rendered image to look.
4. Right click on the Named View heading in the browser.
5. Click on **New Named View** in the drop down menu.
   a. A new named view is created based on the current window configuration with the label **NamedView**
6. Double click on **NamedView** to highlight it and type **Render** to change the label and hit enter
   a. The label has changed to **Render**
7. Click on the **Save** icon to save a new version of the design.
 	
Name views can also be deleted or updated to a new camera position by right clicking on the named view and selecting **Delete** or **Update Named View**.
	
	

Step 3 – Download the image from A360

1. Following the previous steps navigate back to the thumbnails of the **A360 Cloud Render**.
    a. A new thumbnail has been added called **Render.**
2. Select any of the rendered thumbnails to see the larger image in the middle of the screen.
3. Click on the Actions button at the bottom of the window.
4. Select Download Image from the menu.

A360 Cloud Rendering service offers you the opportunity to re-render any of your current thumbnail images with new render settings. Depending on the type of entitlement you have some of these options will cost cloud credits.



Drawings

Overview

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

Learning Objectives

- About Fusion 360 Drawings
- Create a Drawing of a model
- Create Views
- Create & Edit Annotations
- Drawing Settings and Preferences
- Output the Drawing

Introduction to Drawing Views

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.

Projected views inherit the properties of the base view by default. When you change the properties of the base view, the projected view properties also change. However if you override a property of a projected view, that property stops following the changes you make to the base view.

Note: When creating a drawing, the system picks up settings such as the projection angle, annotation format, and the drawing border and title block from Preferences.

42.Create a Drawing

In this section you will open the design file for the Utility Knife and learn how to create a new drawing of the assembly.

<table>
<tr>
<td>
Step 1 – Open the Data Panel

In this module we will be using the 08_Drawings Utility Knife 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.

Click on the icon in the upper left to open the Data Panel.
The Data Panel will slide open

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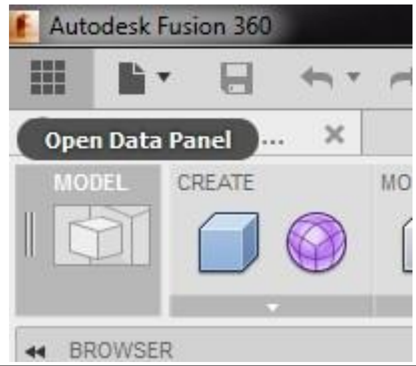
<td>


</td>
</tr>
<tr>
<td>
Step 2 – Open the design

At the top left of the Data Panel, select the project where you uploaded 08_Drawings Utility Knife.
Double-click on the design called 08_Drawings Utility Knife to open the design in Fusion 360.
When the design has opened in your modeling window, click on the 'x' icon to close the Data Panel.

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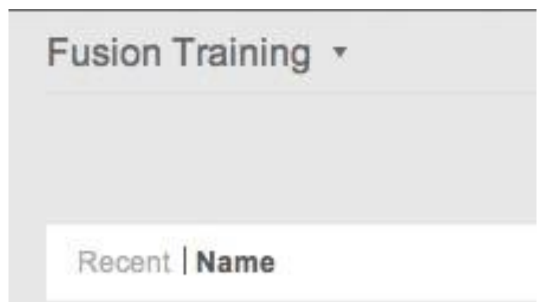
<td>


</td>
</tr>
<tr>
<td>
Step 3 – Initiate a new Drawing

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

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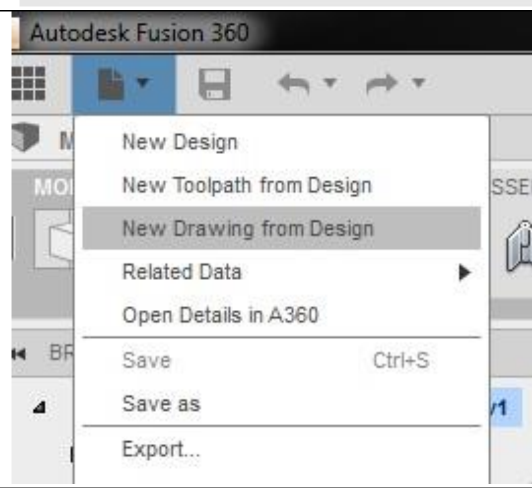
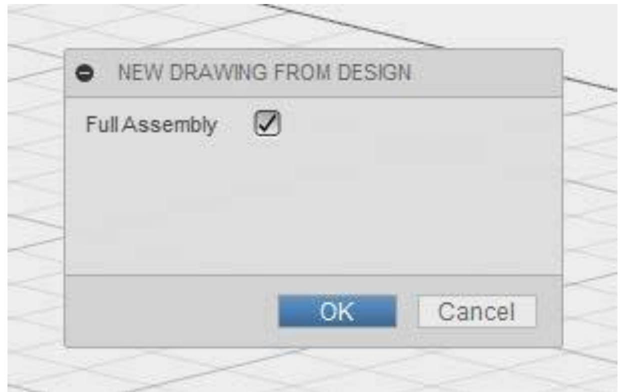
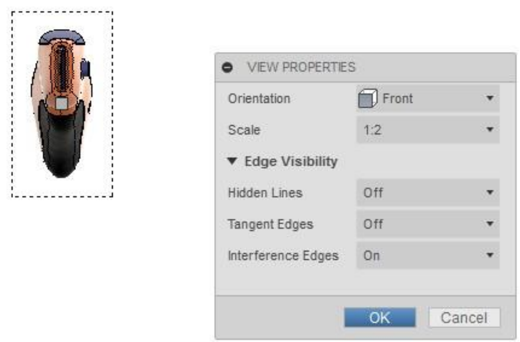
<td>


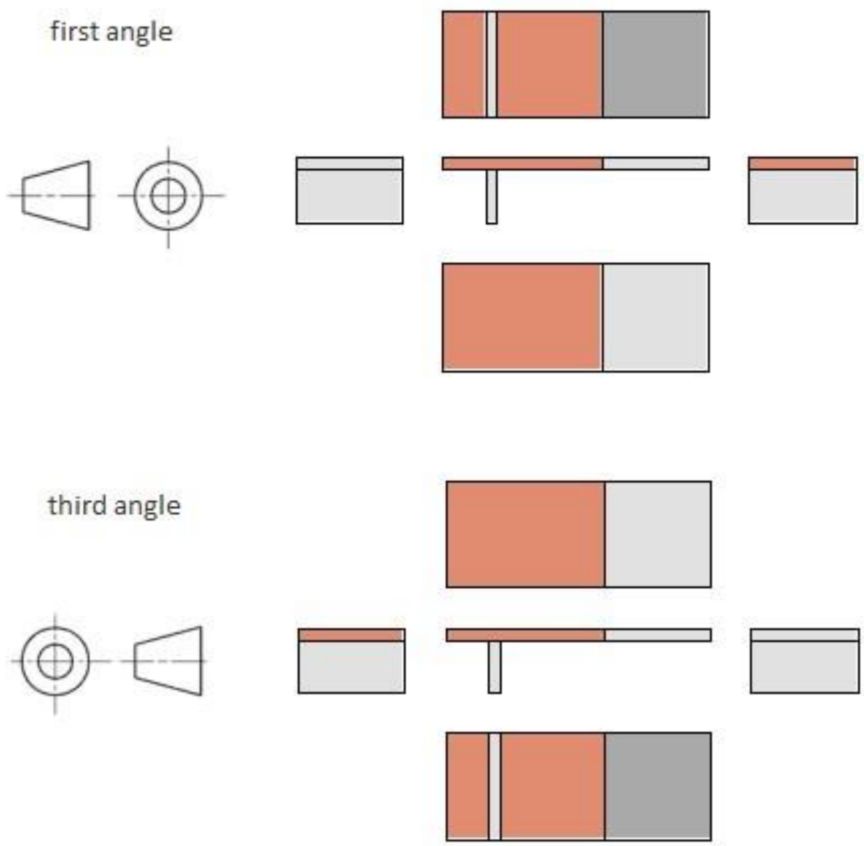
</td>
</tr>
</table>

Step 4 – Choose Assembly 1. Select **Full Assembly** from the dialog and click **OK** to initiate the drawing. 2. Notice that a new tab is automatically generated in Fusion 360 of the Drawing workspace. Note: If you un-check “Full Assembly” from the New Drawing dialog, you have additional controls to pick any set of components to create a drawing from if the full assembly is not what you are looking to document.	
Step 4 – Commit a Base View 1. Move your cursor around the screen and see 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.	

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 Projection

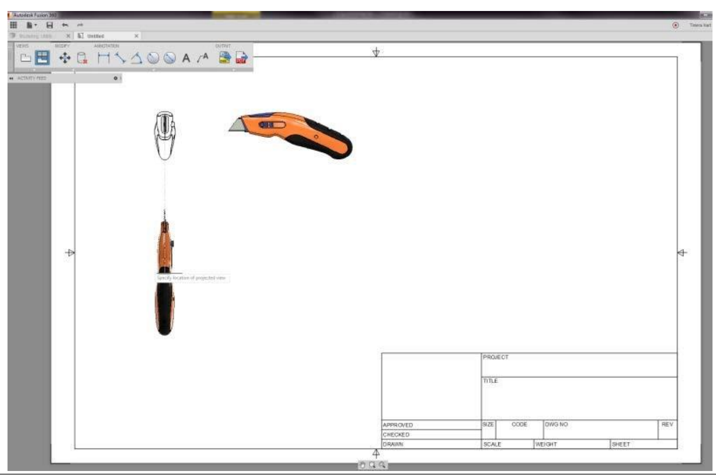
When you use first angle projection, projected views placed to the right of a base view depict the appearance when viewing it from the left. Projected views placed below the base view depict the appearance from above. The ISO drafting standard specifies that drawings use first angle projection.

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 ANSI drafting standard specifies that drawings use third angle projection.

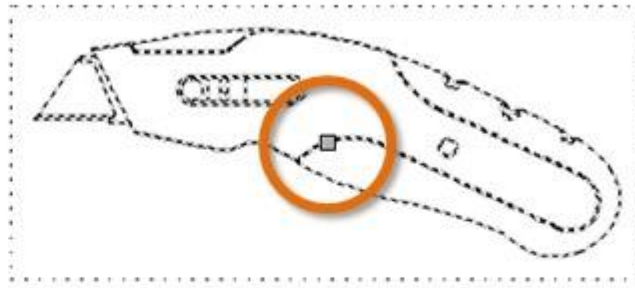
43.Finish the layout:

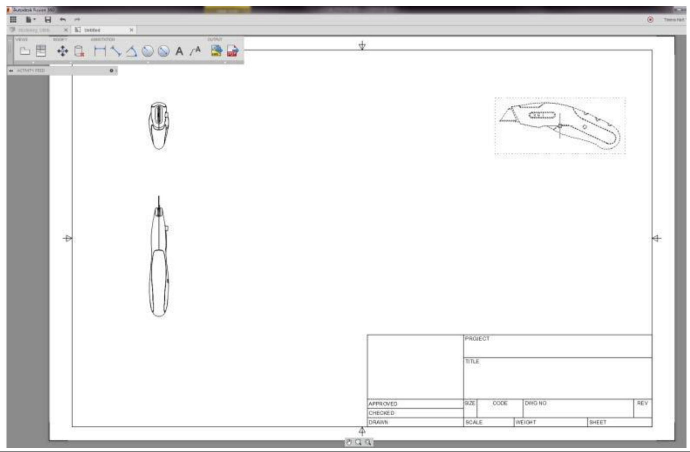
Now that you have created a Base View of the model assembly, you will learn to create projected views and edit their properties to create a complete drawing layout.

Step 1 – Initiate the **Projected View** command 1. Click **Views > Projected View**. 2. Select the existing Base View as the parent view that the projected views will be created from and associated to.	
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: Projected views inherit all its properties from the parent. When the properties of the parent view change, the corresponding properties on the projected view also change.	

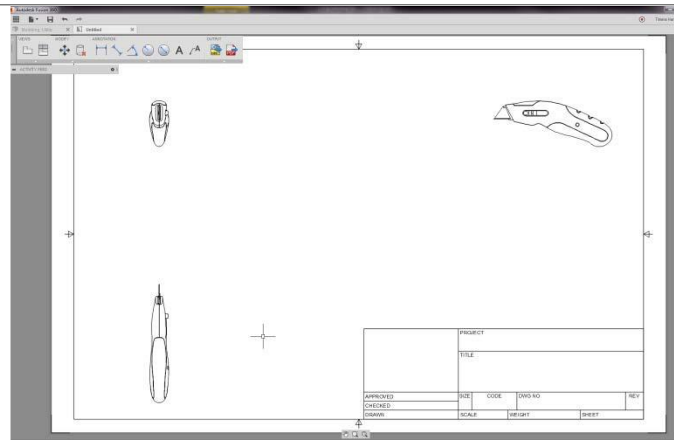
Step 3 – Move the right view

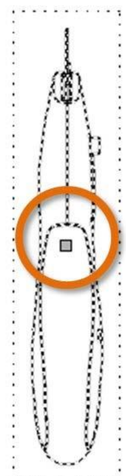
1. Click anywhere inside the selection boundary of the right projected view to activate it.
2. Click the **center grip** to drag the view.
3. Move the view to a new location at the far right of the layout.

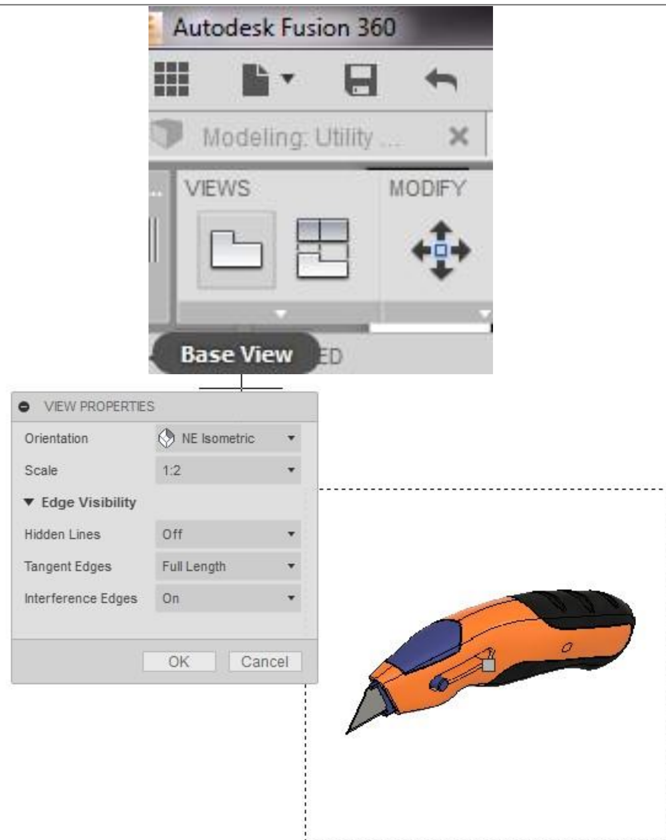




<table>
<tr>
<td>
Step 4 – Move the bottom view

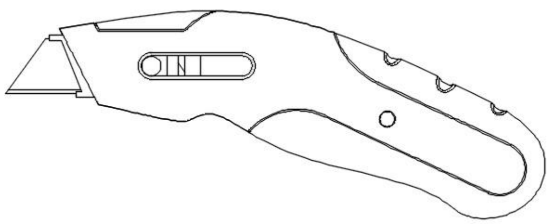
1. Click anywhere inside the selection boundary of the bottom projected view to activate it.
2. Click the center grip to drag the view.
3. Move the view to a new location at the left bottom of the layout.
</td>
<td>



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

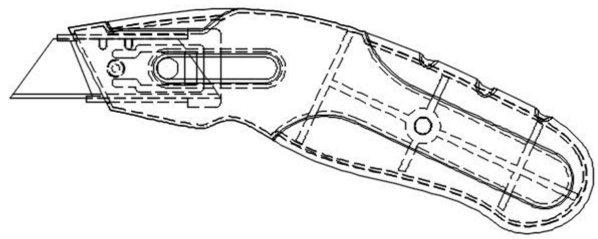
Step 5 – Create an isometric base view 1. Click **Views > Base View**. 2. Click to place the view in the center of the sheet layout. 3. Set the Orientation to **NE Isometric.** 4. Click **OK** to commit the view	

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.

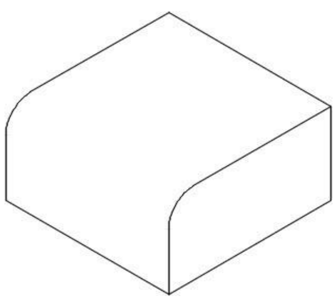


Hidden Lines OFF

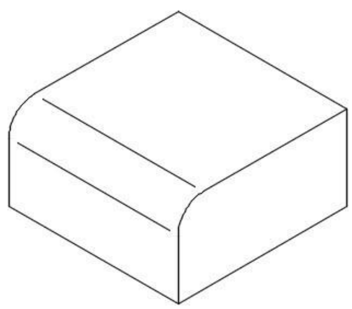


Hidden Lines ON

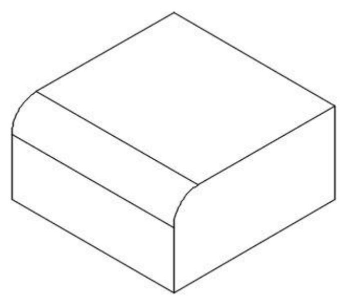
Tangent Edges – Select Full length, Shortened or Off from the drop-down list to display Tangent edges within the selected base view. Tangent edges mark the transition between a flat surface and a rounded edge, most commonly seen as filleted edges. Tangent edges can be set to Full Length, Shortened, or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



Tangent Edges Off

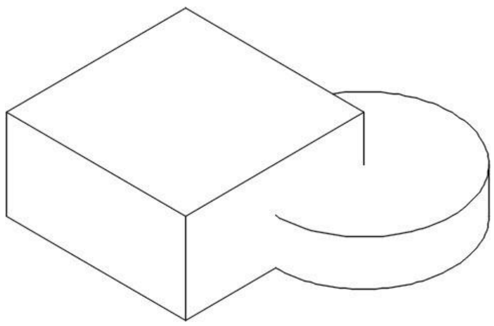


Tangent Edges Shortened

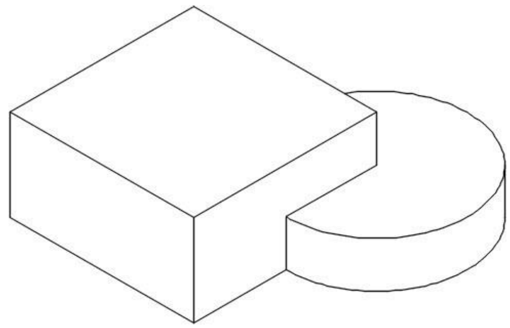


Tangent Edges Full Length

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).



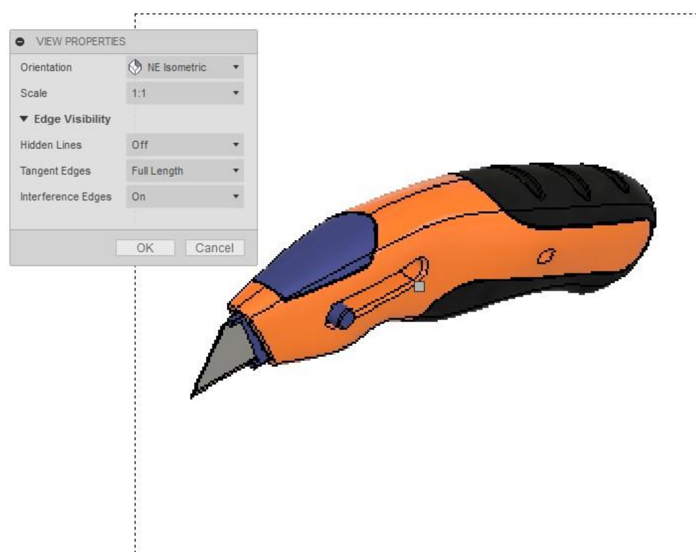
Interference Edges Off



Interference Edges ON

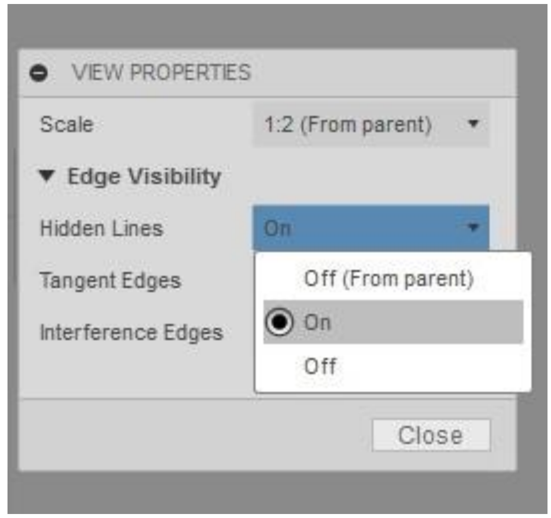
44.Edit the layout views

Now that you have created a base view and several projected views of the model assembly, you will practice using the View Properties settings to further customize the view layouts.

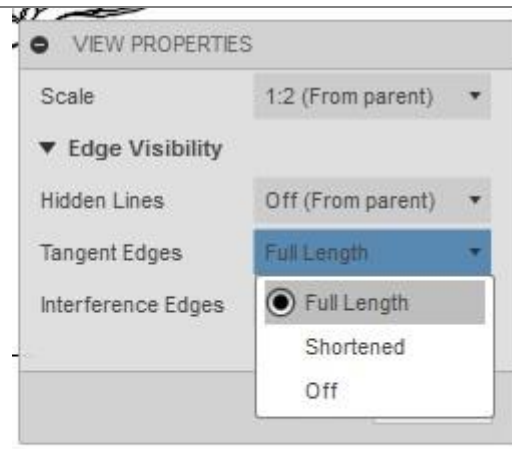
Step 1 – Edit the isometric base view 1. Double-click anywhere inside the selection boundary of the isometric view to activate it. 2. Select the **Scale** ratio in the View Properties dialog box to change the scale. 3. Change the Scale to: **1:1** 4. Click **OK** to accept the drawing view changes. If the projected view properties are changed, they no longer inherit the settings of the base view.	

<table>
<tr>
<td>
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, click Hidden Lines and select **On**.
3. Click **Close** to accept the drawing view changes.

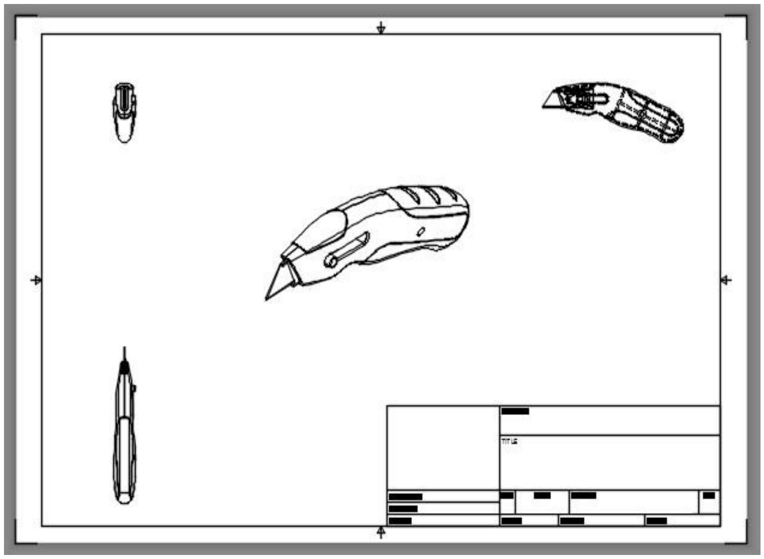
When the projected view properties are changed, they no longer inherit the settings of the base view.
</td>
<td>


</td>
</tr>
<tr>
<td>
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, change Tangent Edges to **Full Length**.
3. Click **Close** to accept the drawing view changes.
</td>
<td>


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

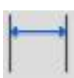
By now, your drawing should look something like this:



45.Text and Annotations

At any point in the drawing creation, you can add a variety of text and annotations to the drawing and the views to document additional detailed information. The tools for adding annotation and text to a drawing include:

Dimension: Linear

 Creates a horizontal or vertical dimension.

Dimension: Aligned

 Creates a linear dimension that is aligned with the origin points of the extension lines.

Dimension: Angular

Measures the angle between selected geometric objects or 3 points.

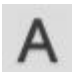
Dimension: Diameter

Creates a diameter dimension for a circle or an arc.

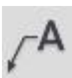
Dimension: Radius

 Creates a radius dimension for a circle or an arc.

Text

 Creates single and multi-line text.

Leader

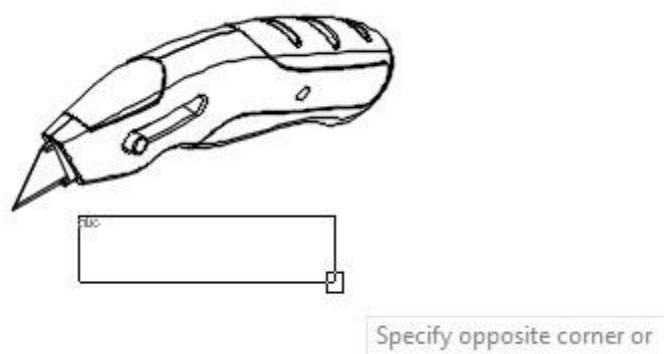
 Creates a leader note.

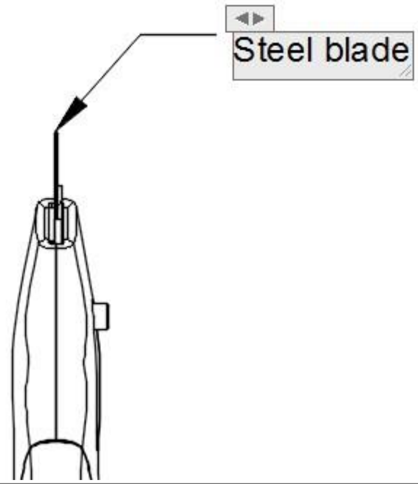
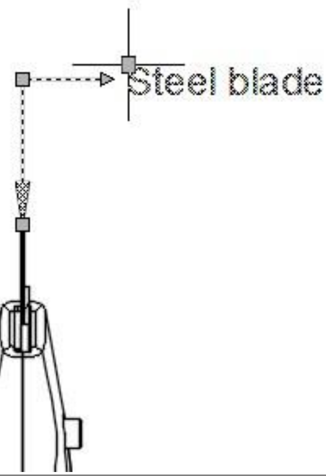
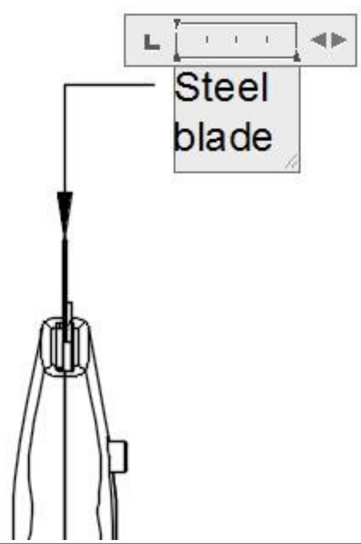
46.Text and Leader Notes:

In this section you will learn how to create, reposition and edit text and leader notes in the drawing.

<table>
<tr>
<td>Step 1 – Create Text

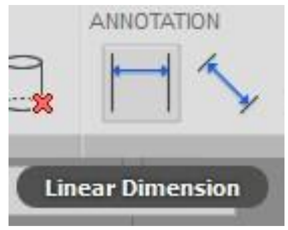
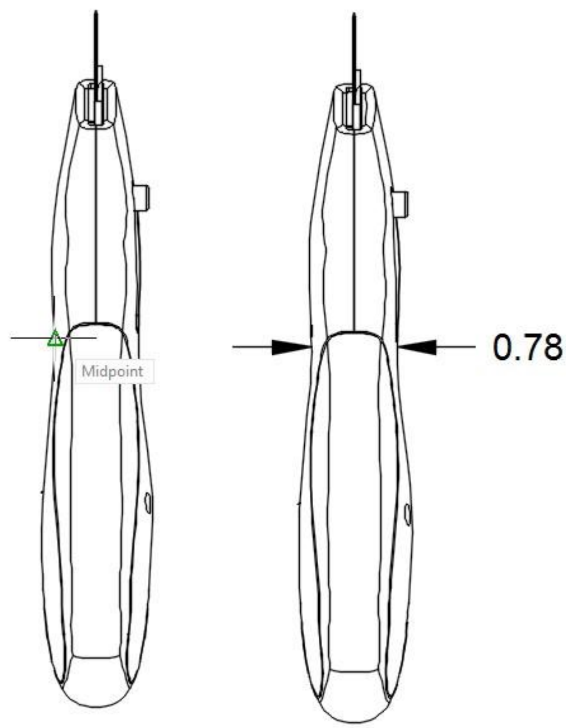
1. Click Annotation > Text.
2. Select two corners below the Isometric Base View to create a text box.
3. Type the following text into the text box: NE Isometric View
4. Select anywhere outside of the text box to commit the action.
5. Repeat process to add text below the to the initial base view with the text: Front View
6. Repeat process to add text below the right projected view with the text: Right View
7. Repeat process to add text below the bottom projected view with the text: Bottom View</td>
<td>

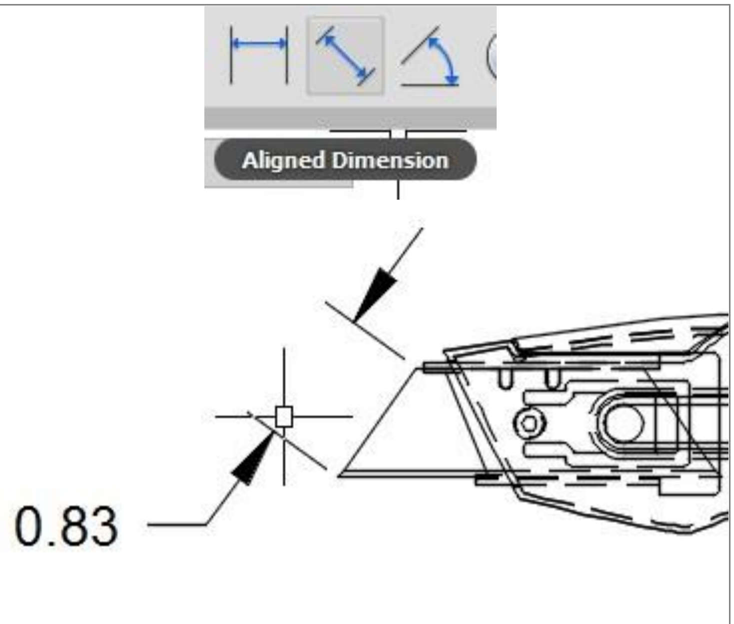
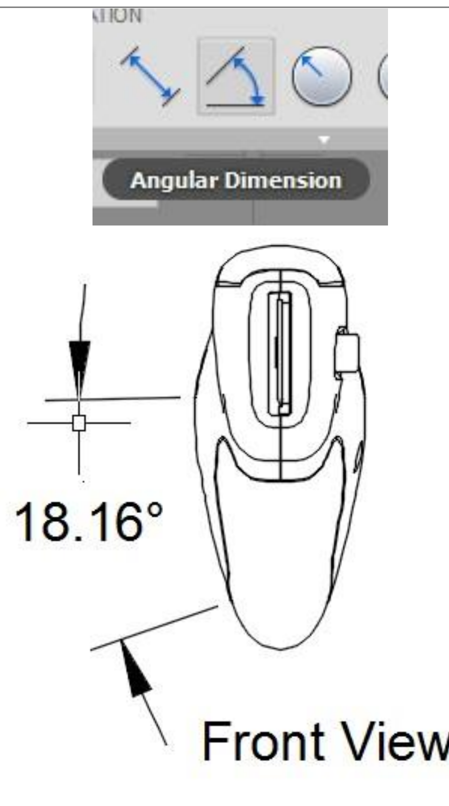

</td>
</tr>
</table>

Step 2 – Create Leader Notes 1. Click **Annotation > Leader**. 2. Click near 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.	
Step 3 – Reposition Leader Notes 1. Click on the leader note to activate. 2. Drag the leader note using the text grip to the left so that the leader is at a 90 degree elbow. 3. Press the **Esc** key to commit the changes and exit the command.	
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. Select anywhere outside of the text box to commit the action.	

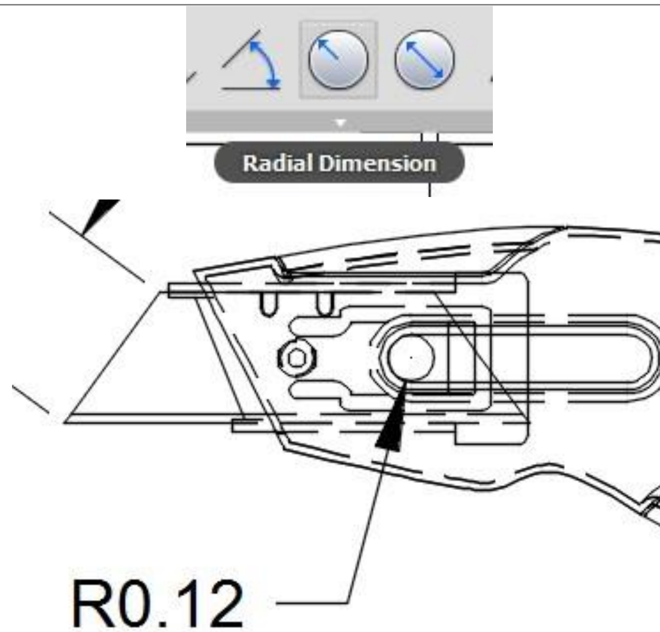
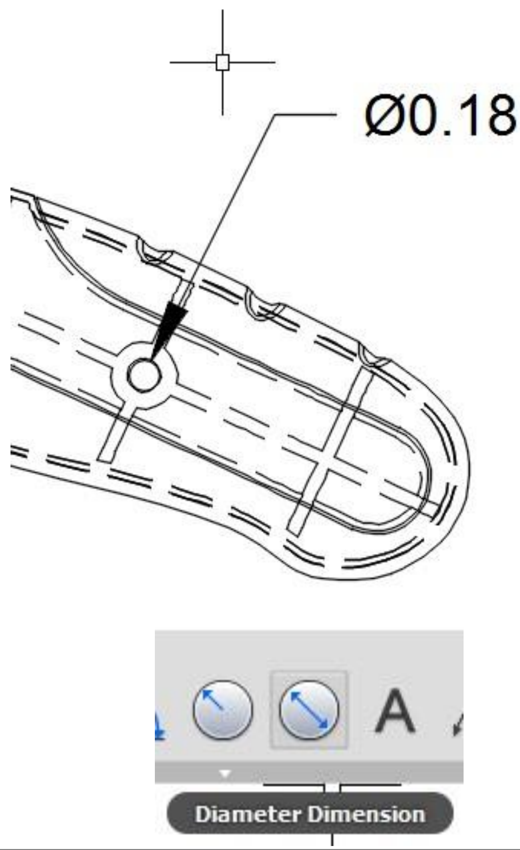
47.Dimensions

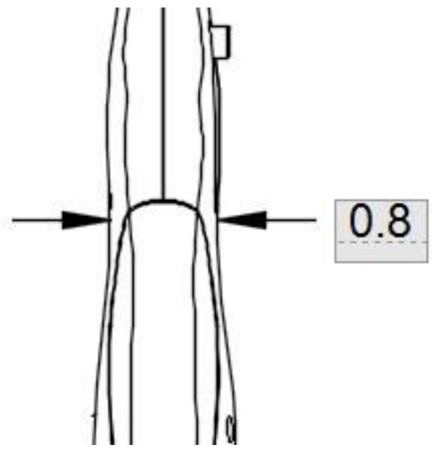
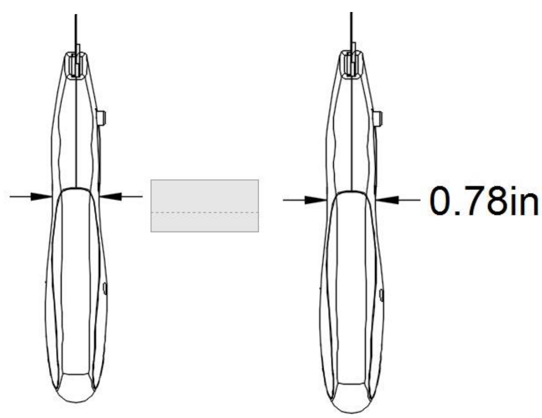
In this section you will learn how to create, reposition and edit a variety of dimension types in the drawing.

Step 1 – Create linear dimensions 1. Click **Annotation > Linear Dimension**. 2. Click the two midpoints of the bottom view and a preview is displayed on your curser. 3. Click again to place the dimension.	 

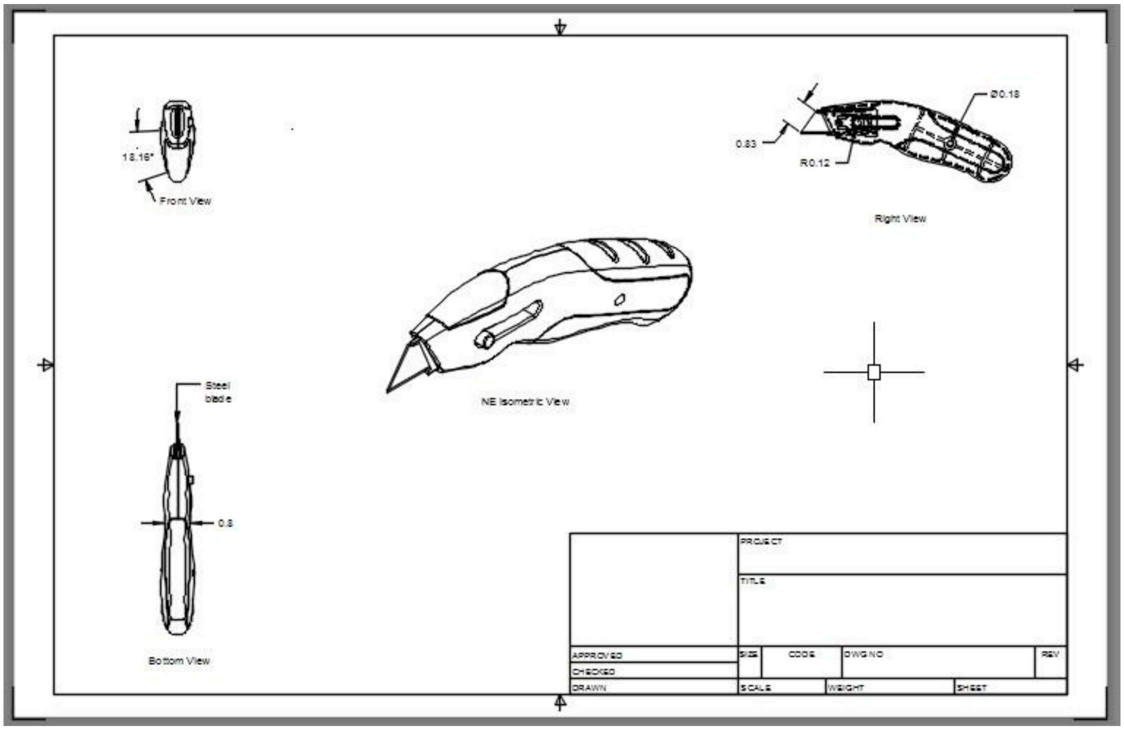
Step 2 – Create aligned dimensions 1. Click **Annotation > Aligned Dimension**. 2. Click the top edge and bottom point of the cutting blade of the right view. 3. Move the cursor out to the left and see a preview of the dimension. 4. Click again to place the dimension and finish the command.	
Step 3 – Create angular dimensions 1. Click **Annotation > Angular Dimension**. 2. Select the left curved edge of the front view. 3. Move the cursor out to the left and see a preview of the dimension. 4. Click again to place the dimension and finish the command.	

<table>
<tr>
<td>Step 4 – Create Radial Dimensions

1. Click Annotation > Radial Dimension.
2. Select the blade slider of the right view as the circle to dimension.
3. Move the cursor down and to the left and see a preview of the dimension.
4. Click again to place the dimension and finish the command</td>
<td>
</td>
</tr>
<tr>
<td>Step 5 – Create diameter dimensions 1. Click Annotation > Diameter Dimension.
2. Select the circular boss in the webbing of the right view as the circle to dimension.
3. Move the cursor up and to the right and see a preview of the dimension.
4. Click again to place the dimension and finish the command.</td>
<td>
</td>
</tr>
</table>

Step 7 – Edit dimensions 1. Double-click the linear dimension on the bottom view to activate it. 4. Replace the current value with **0.80**. 5. Select anywhere outside of the text box to commit the action and see the override	
Step 8 – Return dimension to measured value 1. Double-click the linear dimension you just edited. 2. Highlight the text and delete the "0.80" text. 3. Select anywhere outside of the text box to commit the action and see the initial associated value (0.78 in).	

By now, your drawing should look something like this:



48.Drawing Settings & Preferences

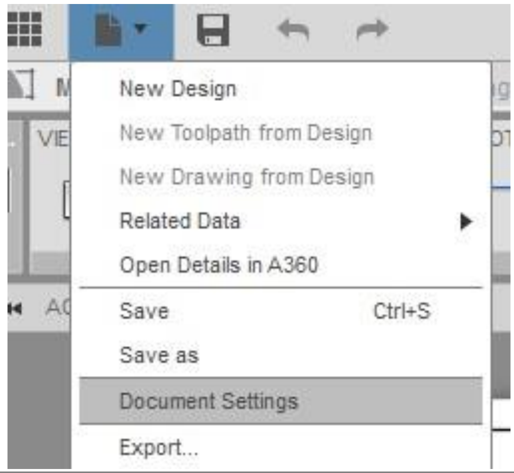
You can change settings such as the default projection angle, sheet size, title block, annotation properties and dimension precision.

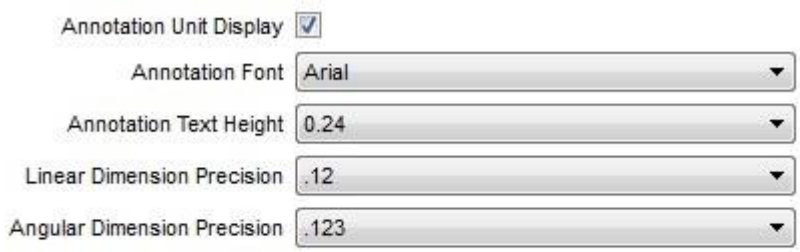
The default settings are applied whenever you create a new drawing. If required, you can override some of the settings once the drawing is created.

Change Drawing Settings: Making adjustments to the Drawing Settings are local and drawing-specific. These preferences will be used as the default override for the active drawing only.

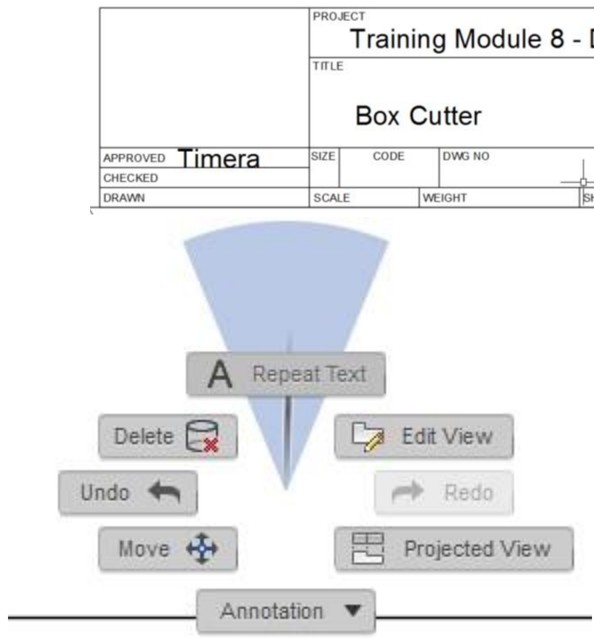
Note: If you would like any of these settings to permeate as a default in future drawings, you can use the same workflow as below in the Preferences dialog. The Preferences dialog can be found in the application bar under Your Name:

<table>
<tr>
<td>Step 1 – Open Document Settings

1. On the application bar, click the file dropdown menu.
2. Select Document Settings to display the Document Settings dialog box.</td>
<td>
</td>
</tr>
<tr>
<td>Step 2 – Edit annotation preferences

1. Check the box for Annotation Unit Display.
2. Change the Annotation Text Height to 0.24.
3. Change the Angular Dimension Precision to .123.
4. Click OK to accept the changes</td>
<td>
</td>
</tr>
</table>

<table>
<tr>
<td>
Step 3 – Edit title block

1. Click Annotation > Text.
2. Add a description to the Project section of the title block.
3. Right-click and select Repeat Text.
4. Add a description to the Title section of the title block.
5. Right-click and select Repeat Text.
6. Add a description to the Approved section of the title block.
</td>
<td>


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

Associatively Update the Drawing

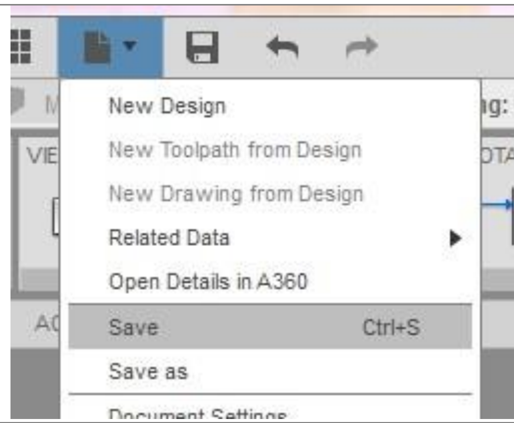
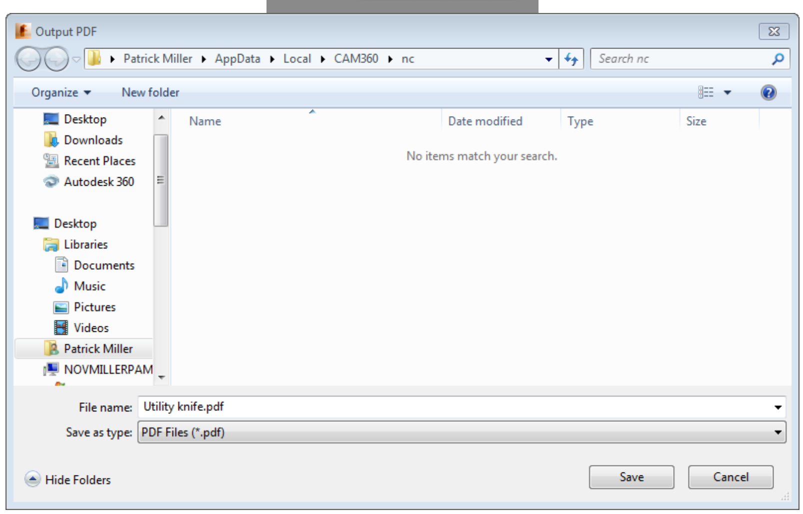
Any change you make to the model's geometry, the drawing views are immediately updated to reflect the changes.

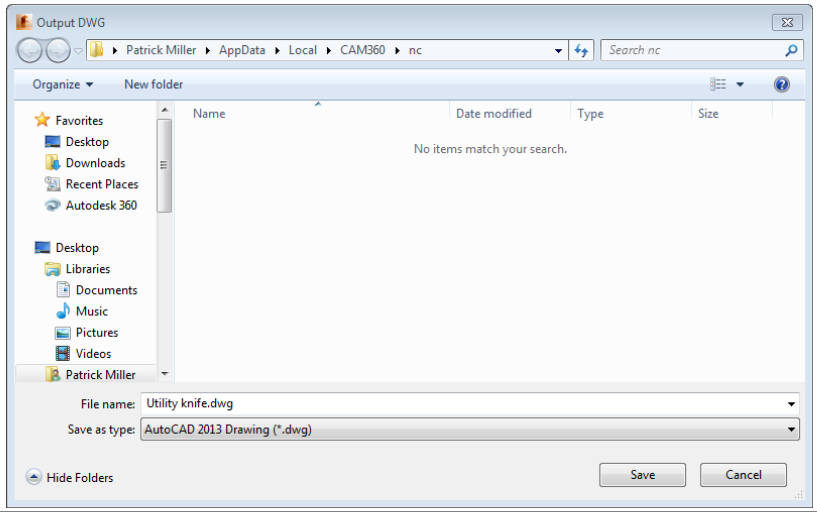
The 3D models are likely to change even after drawing views are created and annotated.

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.

49.Output the Drawing

When the drawing is completed, you have the ability to output the layout to either a PDF or DWG. Both of these options creates a copy of the drawing and prompts you to save it locally onto your machine.

Step 1 – Save the drawing 1. Click **File > Save**. 2. Enter **Utility Knife** in the Name field. 3. Click **OK**.	
Step 1 – Output a PDF 1. Click **Output > Output PDF**. 2. Navigate to a desired local location on your computer. 3. Click **Save**.	

Step 2 – Output a DWG 1. Click **Output > Output DWG**. 2. Navigate to a desired local location on your computer. 3. Click **Save**.	 

Notes

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Notes

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