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.

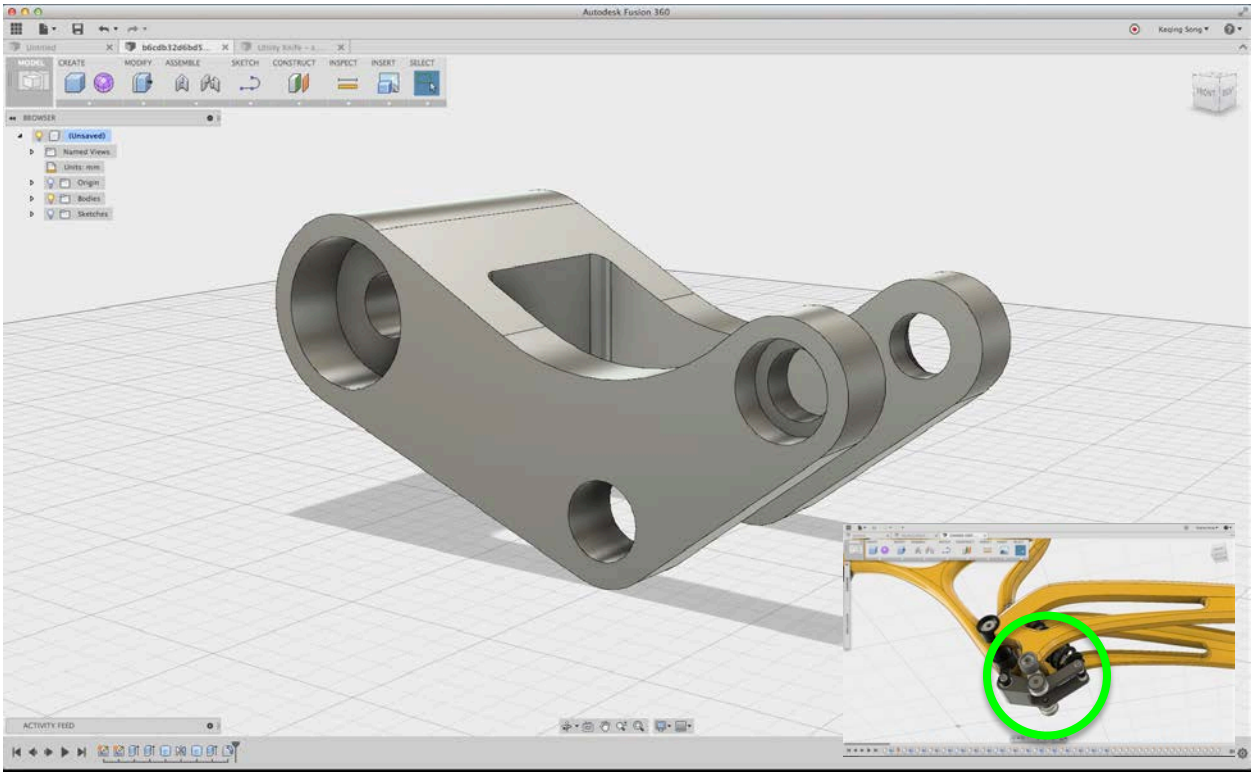
Learning Objectives

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

Modeling from a sketch

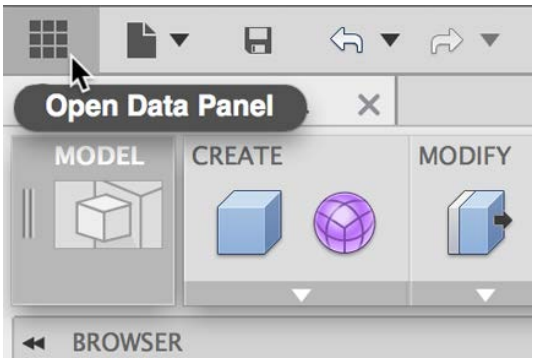
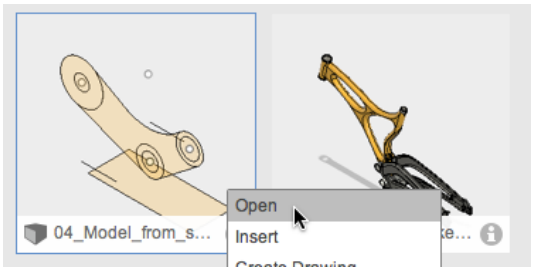
Bicycle rocker arm



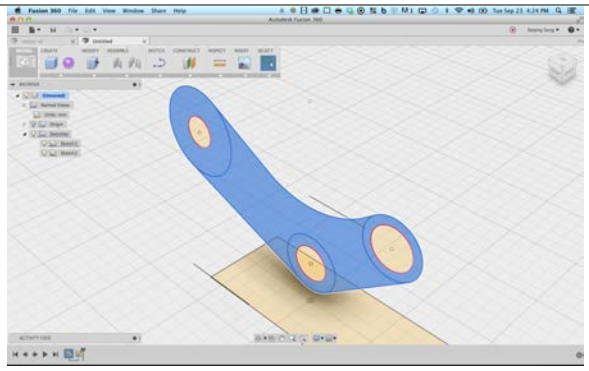
Before moving on, make sure you have uploaded **04_Model_from_sketch** design to your A360 site.

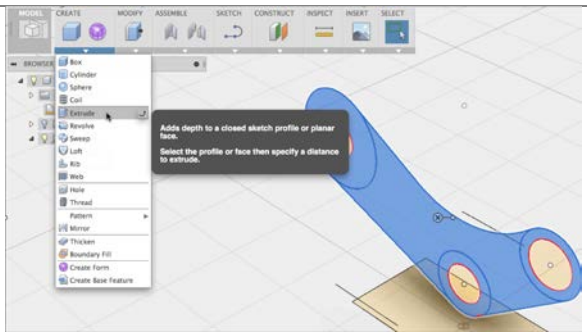
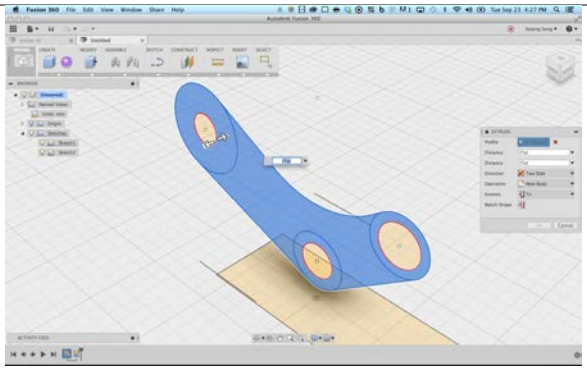
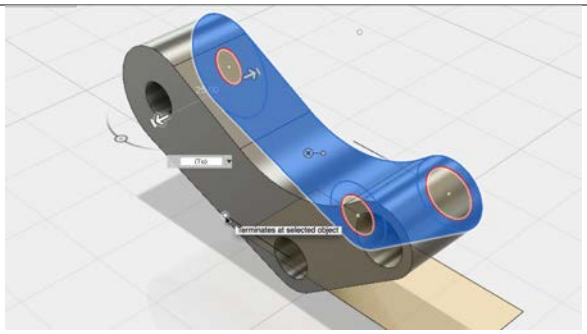
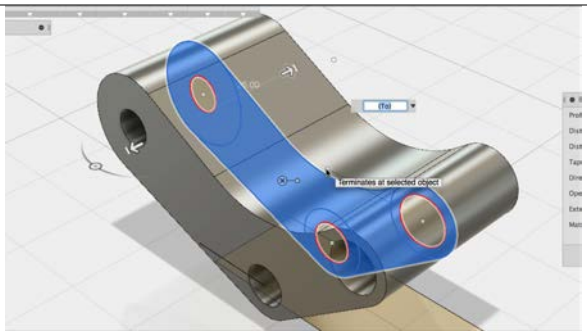
If you like to watch the video to this tutorial, click here: **Launch Video**

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

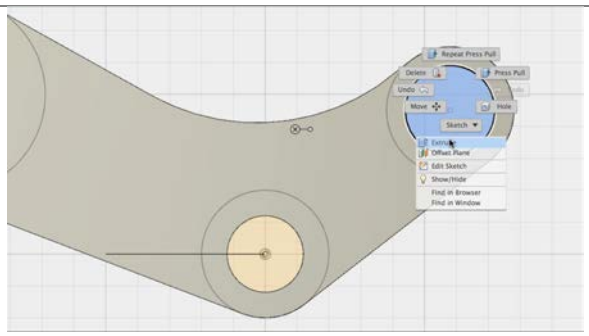
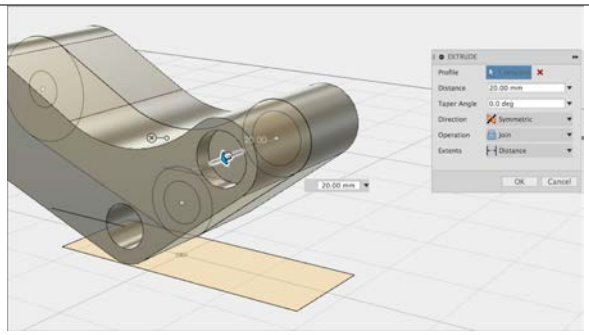
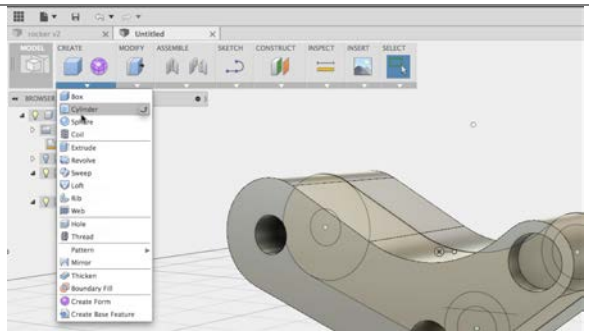
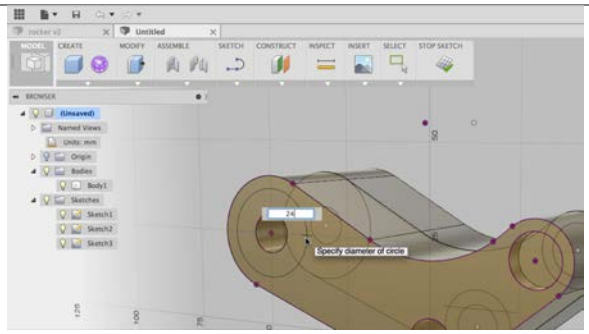
Step 1 – Open the Data Panel 1. Open the Data Panel by clicking on the icon located at the top left of the menu bar. 2. The Data Panel will slide open.	
Step 2 – Open the design *In this module we will be using the* **04_Model_from_sketch.*f3d*** *file to complete the exercise. If you haven't set up 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 **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.	

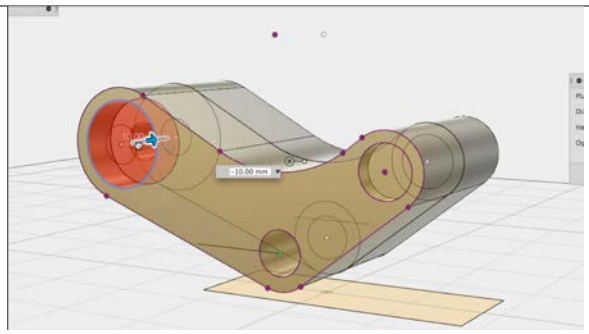
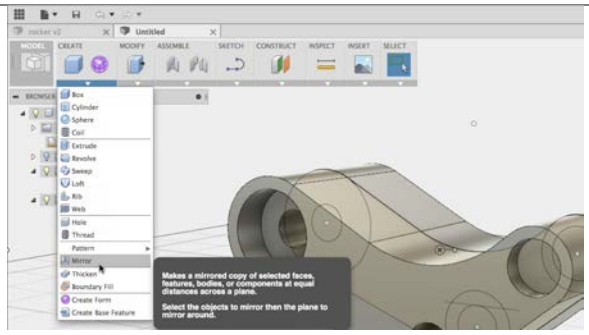
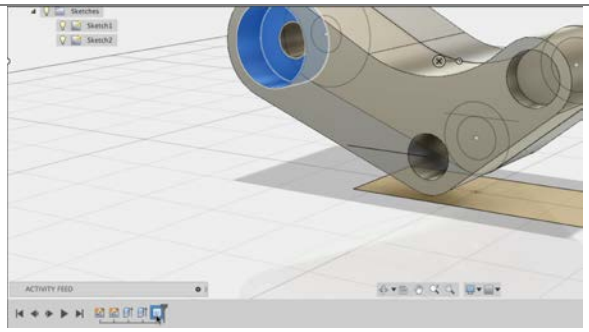
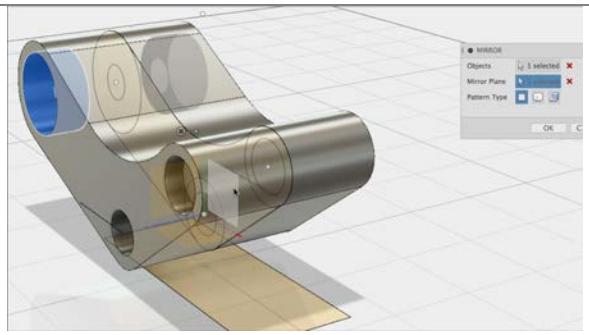
Create solid body: Let's start with this sketch of the rocker arm. We're going to use this to create a solid body.

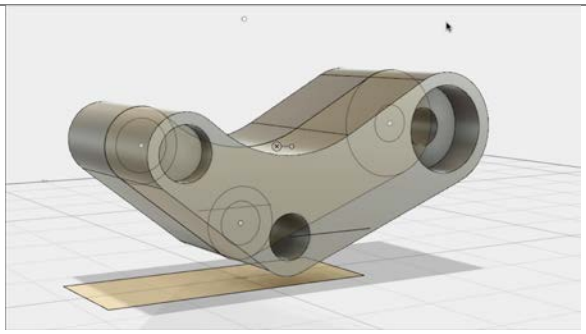
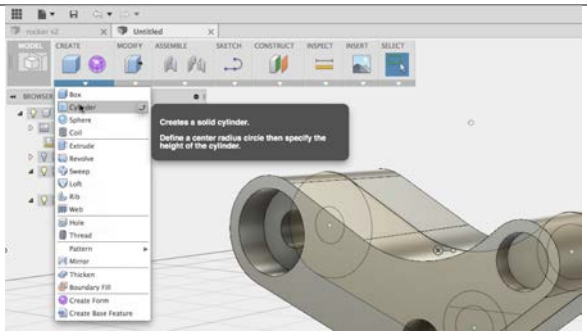
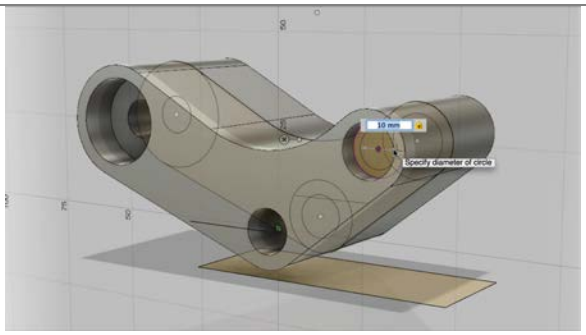
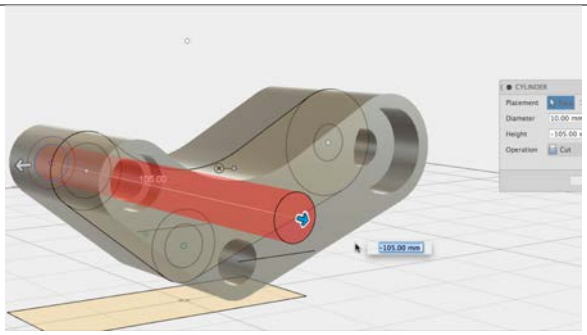
Step 1 – Select profiles 1. Hold down **Shift** and select the profiles shown in the image. Make sure that the 3 center holes are the only profiles not selected. Note: If you are having trouble selecting certain profiles, zoom in closer and that should make it easier to select.	

Step 2 – Start the Extrude command 1. Click **Create > Extrude**. We're going to extrude the selected profiles.	
Step 3 – Set the extrude options 1. Set Direction to **Two Side.** 2. Set Extents to **To.**	
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.	
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.	

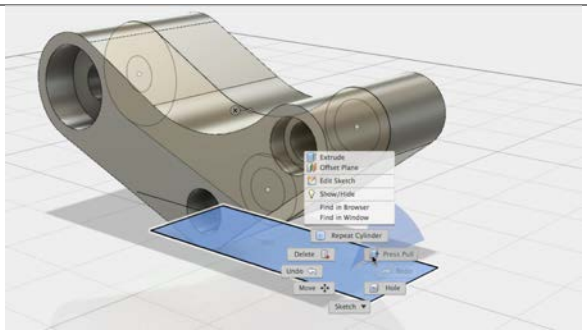
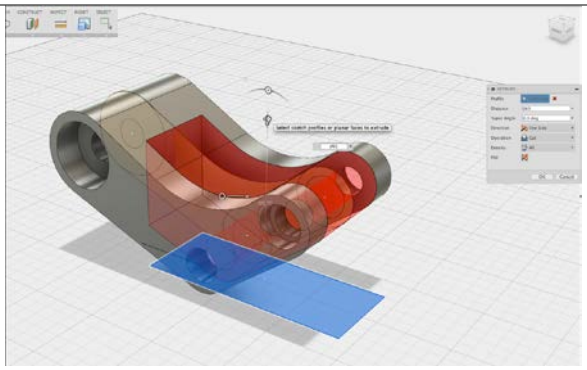
Cut holes: In this section you use the sketch profiles to cut holes in the body.

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.	
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.	

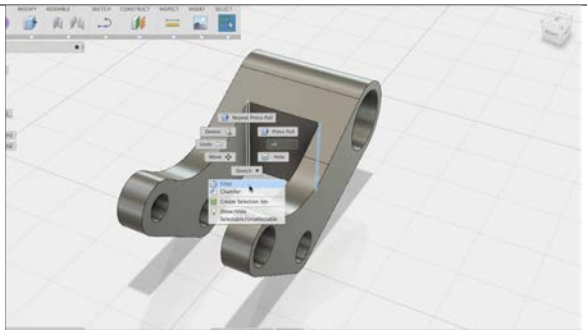
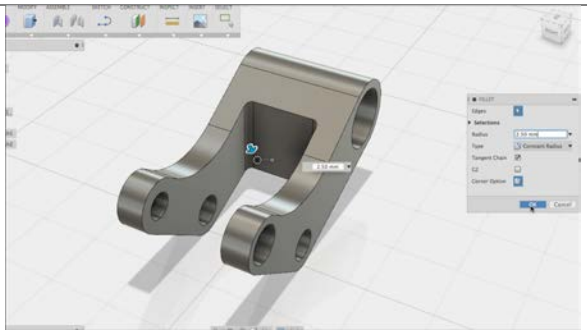
Step 5 – Set the cut distance 1. Use arrow manipulator and drag it inward to **– 10 mm**. 2. Click **OK** to finish the cut. Note: This naturally became a cut because the software recognized that the cylinder body is intersecting with an existing body, thus assumed that you wanted a cut.	
Step 6 – Start 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.	

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.