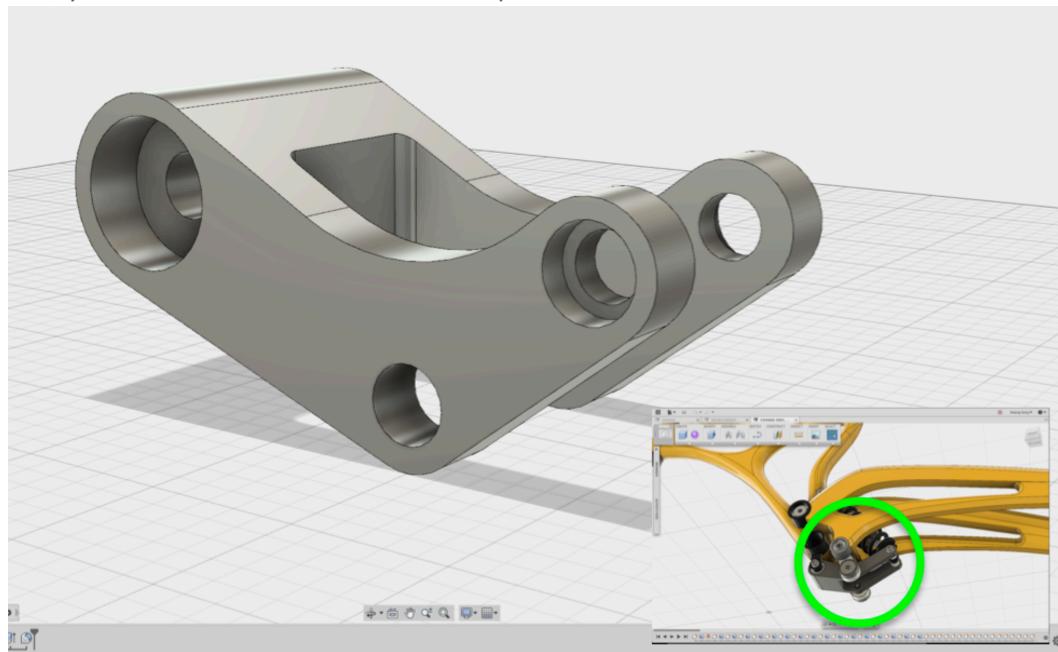


4.1: Modeling

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

Lesson 1: Modeling based on Sketches

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



Learning Objectives

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

Datasets Required

In Samples section of your Data Panel, browse to:

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

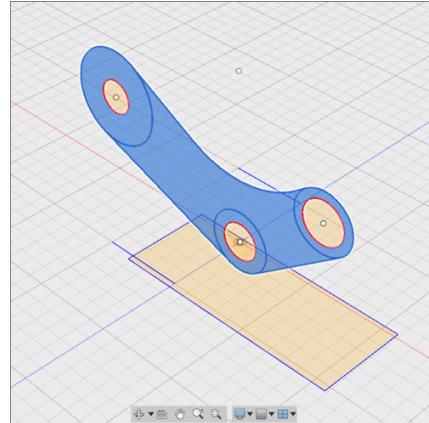
Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

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

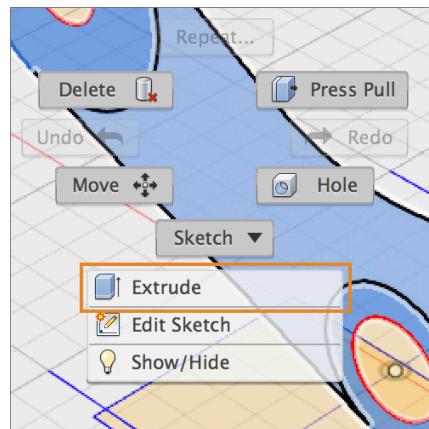
1. Hold down **Shift** and select the profiles shown in the image. Make sure that the 3 center holes are the only profiles not selected.

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



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

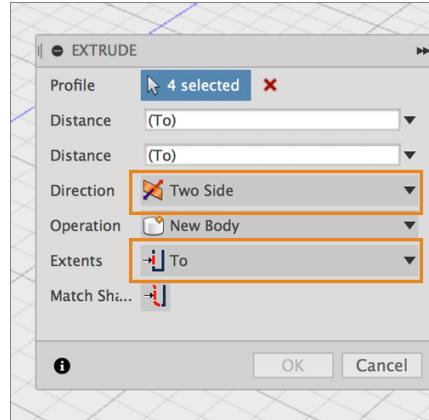
1. Right click on a selected area of the sketch and select **Extrude**. We're going to extrude the selected profiles.



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

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

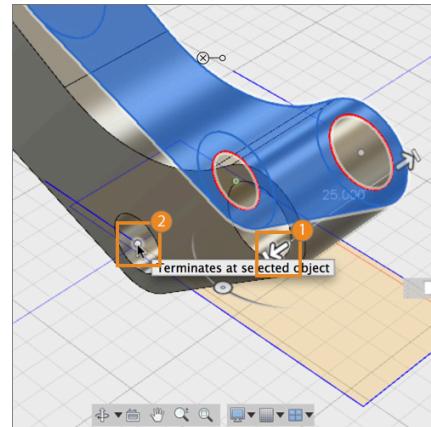
You should now see 2 arrows appear on your selected sketch profiles. We're going to use these arrows to define where we want the extrusion to go. This is especially useful when you have set geometry you can use as reference, much like our sketch here.



Step 4: – Set the distance for the left side

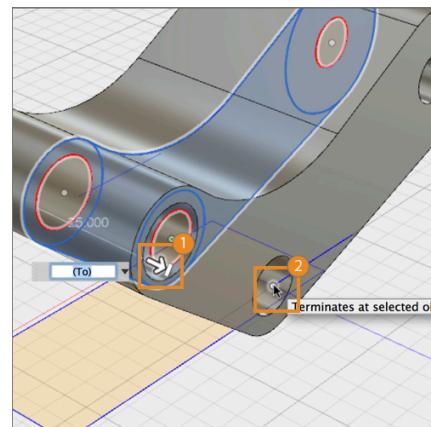
1. Click once on the **left arrow manipulator**.
2. Now hover over the **line sketch** on the left side and click on the **end point** as the extent you want to extrude to.

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



Step 5: – Set the distance for the right side

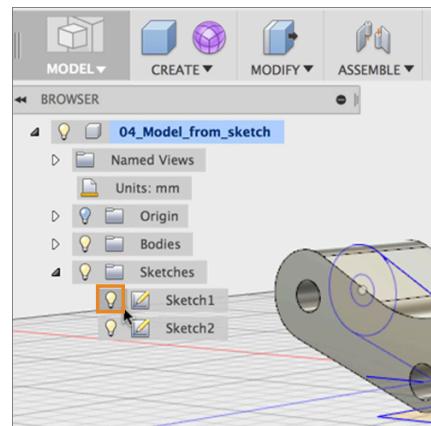
1. Repeat the same thing on the right side. Click once on the **Right arrow manipulator**.
2. Now hover over the **line sketch** on the left side and click on the **end point** as the extent you want to extrude to.
3. Click **OK** (or hit **ENTER**) to finish the command.



Step 6: – Turn sketch visibility back on

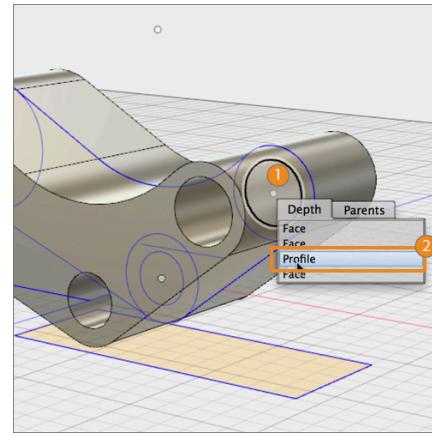
1. Let’s go to the browser and within the Sketches folder, click the light bulb icon next to **Sketch1** to turn the visibility of that sketch back on.

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



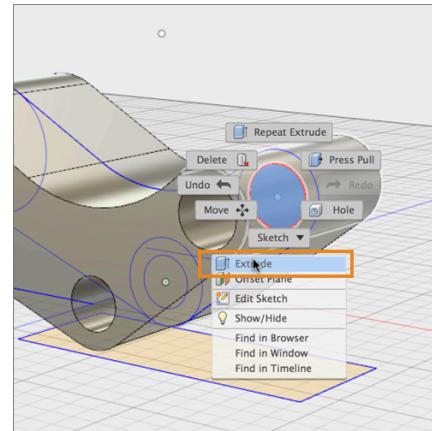
Step 7: – Select a sketch profile behind an obstruction

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



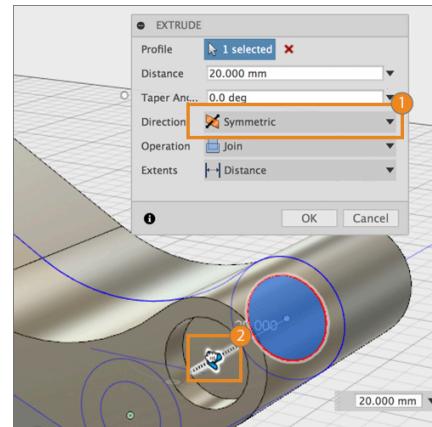
Step 8: – Extrude the circle profile

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



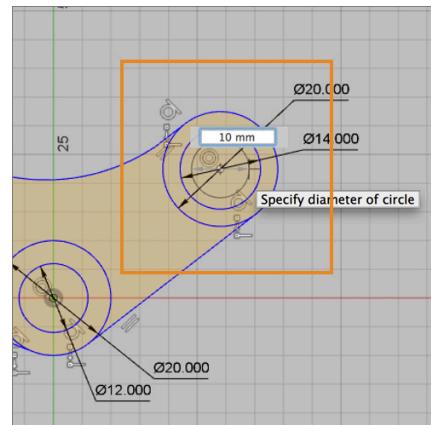
Step 9: – Join the new extruded body

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



Step 10: – Sketch a new circle profile

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

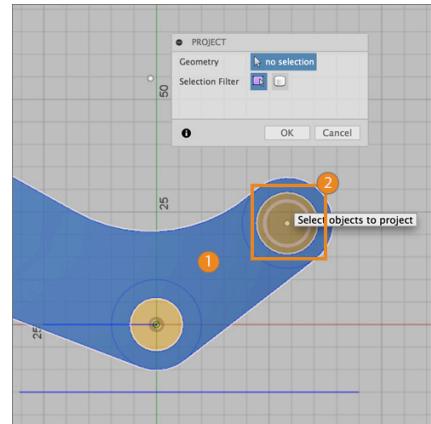


Step 11: – Project the circle onto a new face

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

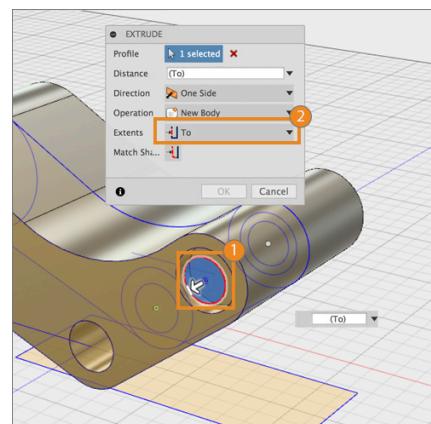
1. First select the outer face,
 2. Then select the new circle sketch we just created.
- Click **OK** to finish, and **Stop Sketch** to exit out of the sketch mode.

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



Step 12: – Extrude the circle as a cut

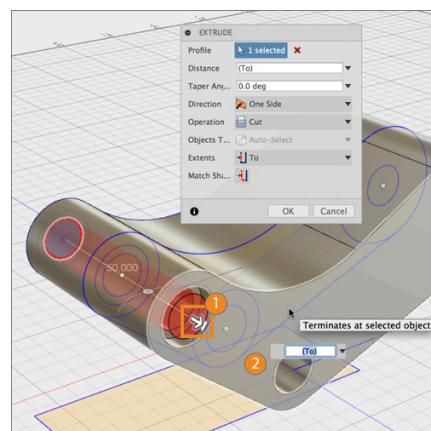
1. Let's select the project circle profile, right-click and choose **Extrude**.
3. In the command dialog, change the Extents to **To**.



Step 13: – Make the cut

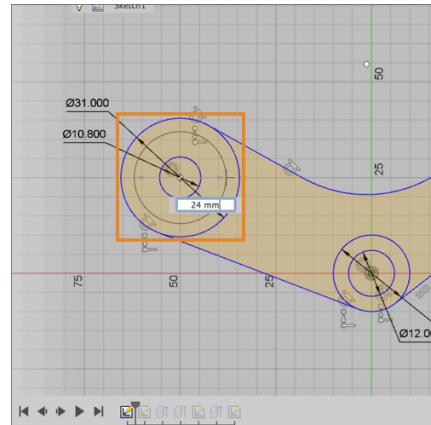
1. Click on the **Arrow Manipulator** to activate the extrusion.
2. Rotate the model to the other side so that we see the other face we want to extrude to. Click on that face and click OK to finish.

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



Step 14: – Sketch a new circle profile

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

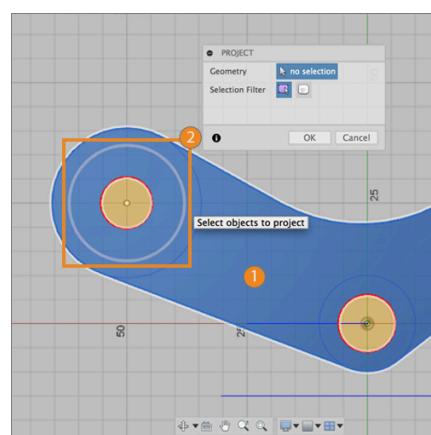


Step 15: – Repeat Project sketch workflow

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

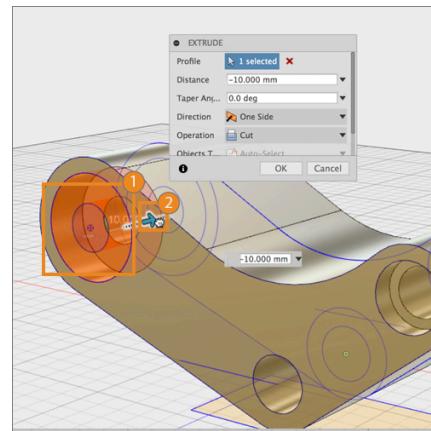
1. First select the outer face,
 2. Then select the new circle sketch we just created.
- Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

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



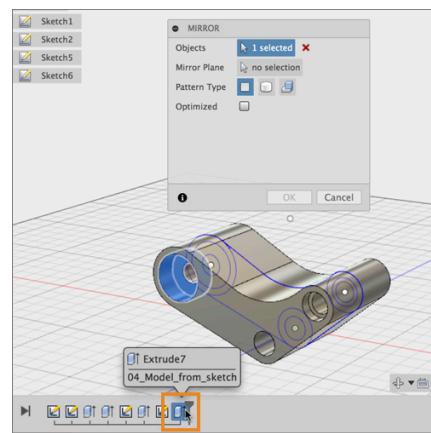
Step 16: – Extrude the circle as a cut

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



Step 17: – Mirror the cut on the other side

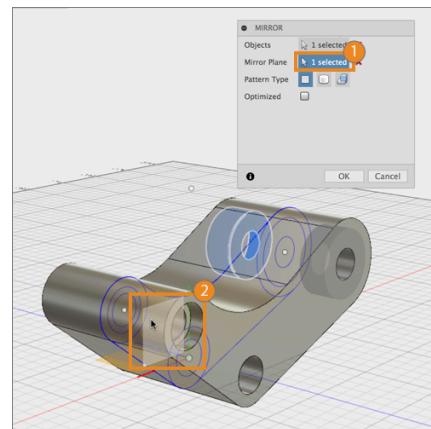
1. Now that we've made this cut, let's mirror it on the other side. Go to the Create drop-down menu and select **Mirror**.
2. Go to timeline and select the last extrusion as the object we want to mirror.



Step 18: – Mirror the cut on the other side

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

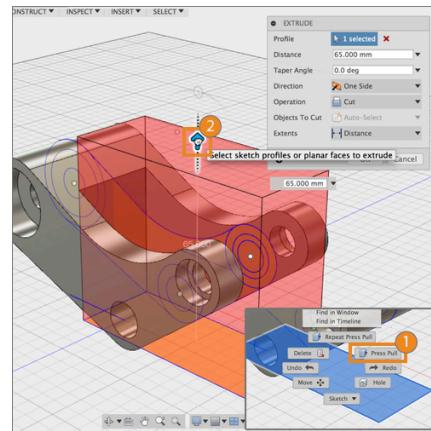
Note: If you have having trouble selecting the origin plane, remember to zoom out or click and hold to get the option to choose what you'd like to select.



Step 19: – Use Press-Pull to cut

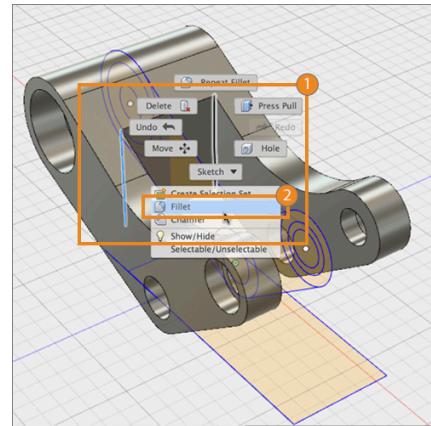
1. Now let's select the rectangle sketch at the bottom, right-click and use **Press-Pull** (on the right of your cursor).
2. Drag the **Arrow Manipulator** through the model so the cut goes all the way through.

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



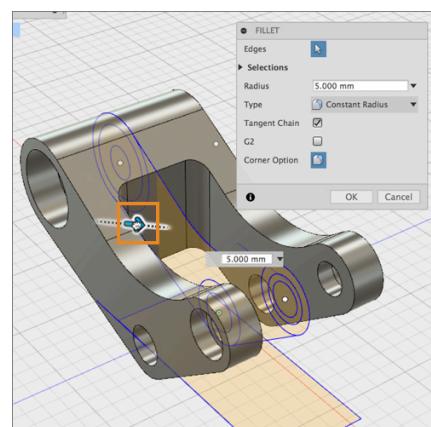
Step 20: – Add a couple fillets

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



Step 21: – Add a couple fillets

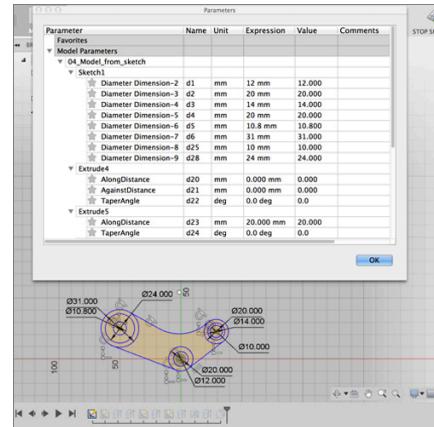
1. Drag the **Arrow Manipulator** to 5 mm
Click OK to finish.



Step 22: – Making changes to your model

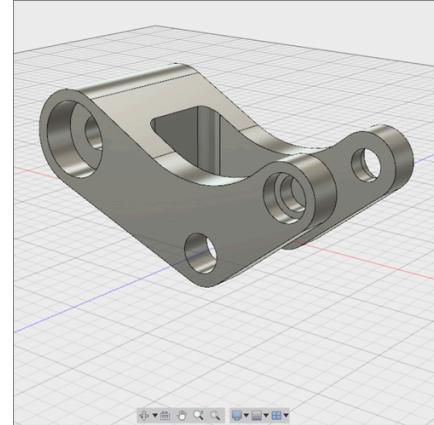
Since all the extrusions, mirror, and fillets are based on the original sketches, we can go back to **Sketch1** and **Sketch2** and any time and make dimension changes without needing to change each downstream feature or worry about any of them failing.

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



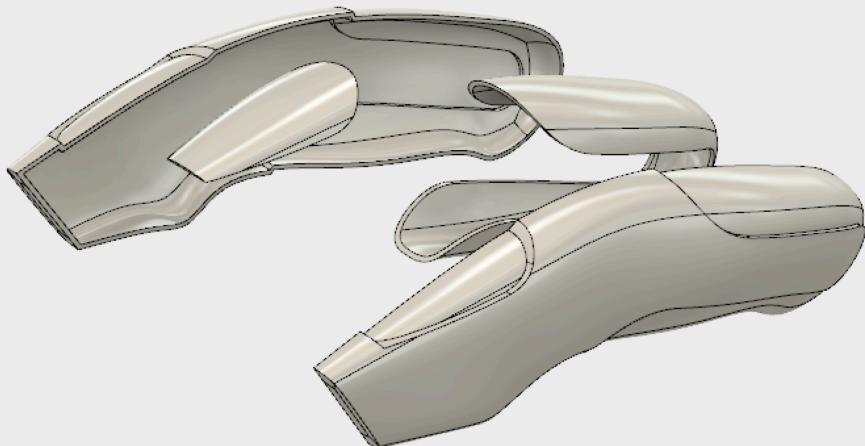
Step 23: – Model complete!

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



Lesson 2: Modeling based a sculpted body

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



Learning Objectives

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

Datasets Required

In Samples section of your Data Panel, browse to:

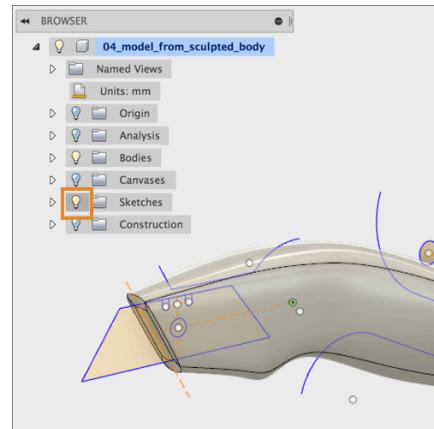
Fusion 101 Training > 04 – Modeling > **04_Model_from_sculpted_body**

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

Step-by-step Guides

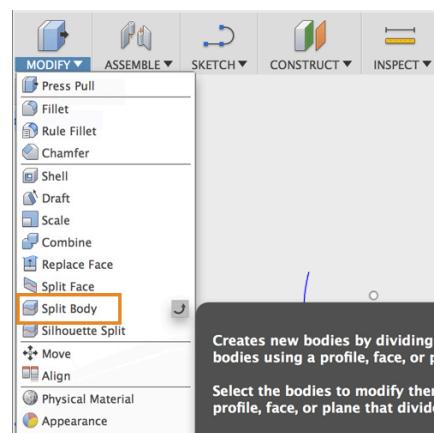
Step 1: – Turn on Sketch visibility

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



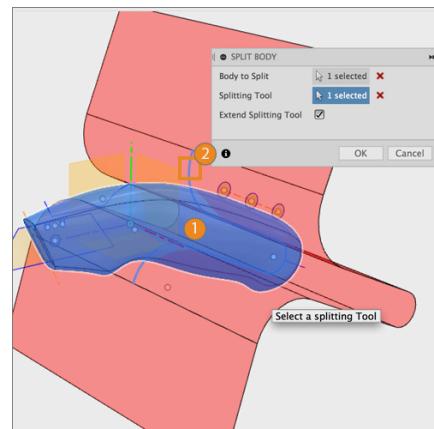
Step 2: – Split the model into 2 bodies

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



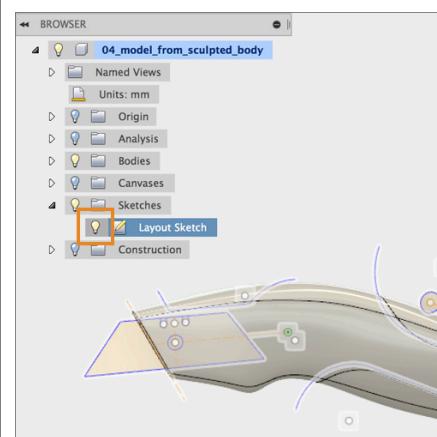
Step 3: – Make the split

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



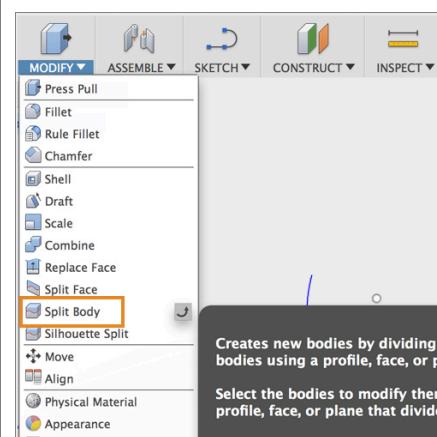
Step 4: – Repeat previous step

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



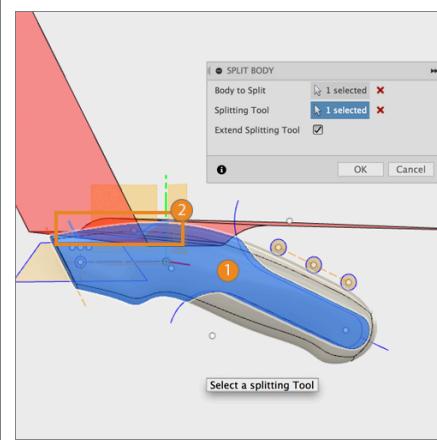
Step 5: – Split the model into 3 bodies

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



Step 6: – Make the split

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



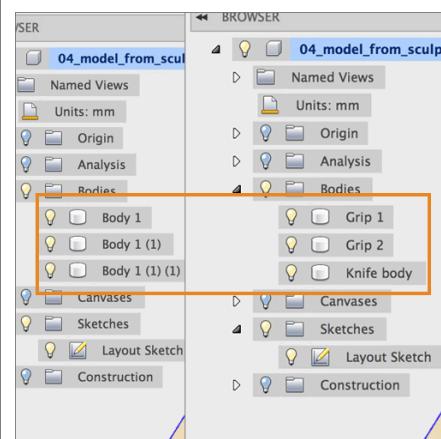
Step 7: – Rename bodies

3. In the Bodies folder, you'll notice that there are now 3 bodies. Let's rename them. **Double click** on the body and rename:

Body 1 to **Grip 1**

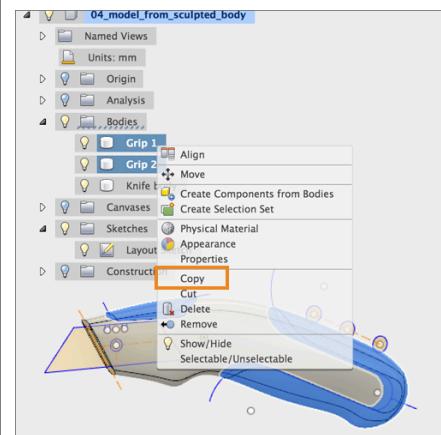
Body 1 (1) to **Grip 2**

Body 1 (1) (1) to **Knife body**



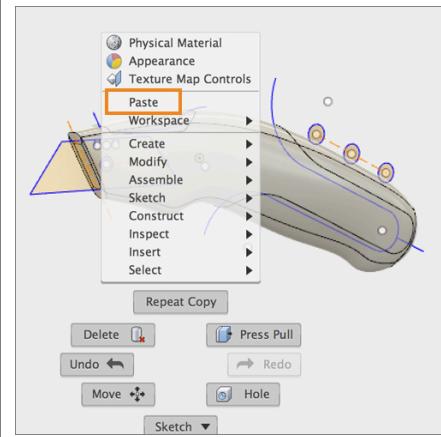
Step 9: – Copy and paste bodies

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



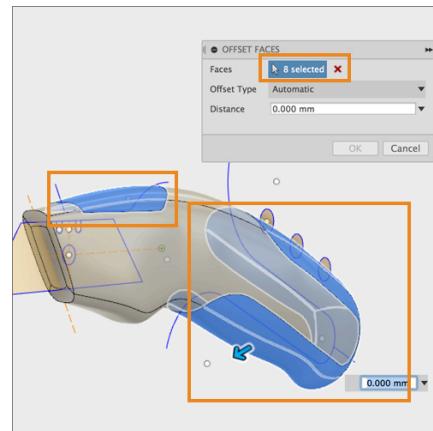
Step 10: – Copy and paste bodies

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



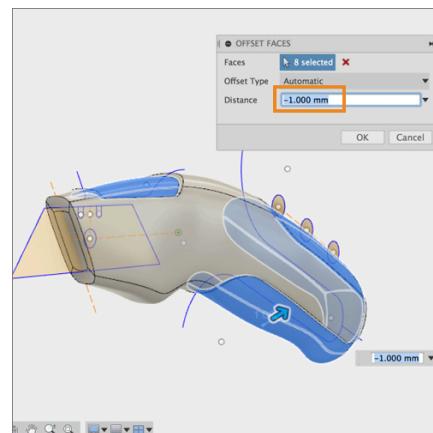
Step 11: – Offset bodies with Press-Pull

1. Turn off visibility of **Grip 1** and **Grip 2**. We're going to work on the 2 new grip bodies.
2. Right-click somewhere in canvas and select **Press-Pull**. Select all the faces of **Grip 1 (1)** and **Grip 2 (1)**. Make sure to rotate around and get all the faces. Hold **Shift** to add onto the selection. You should have a total of 8 faces selected.



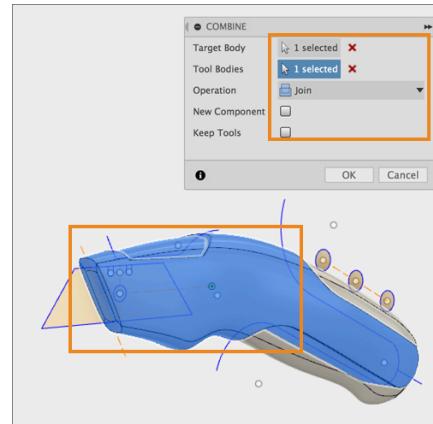
Step 12: – Offset bodies with Press-Pull

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



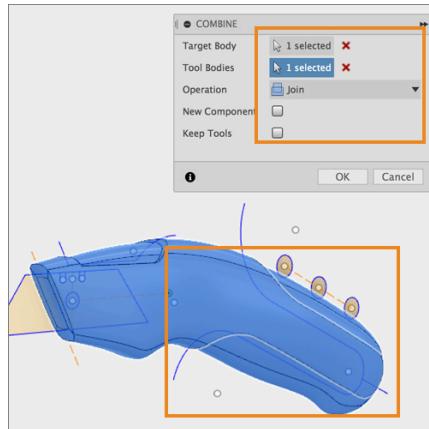
Step 13: – Join bodies with Combine tool

1. Let's now join these offset bodies to the knife body so that they are part of a whole body. Go to the Modify drop-down menu and select **Combine**.
2. In the command dialog, set:
Knife Body as the **Target Body**,
Grip 2 (1) as the **Tool Body**,
Join as the **Operation**,
Uncheck Keep Tools,
3. Click OK to finish the operation.



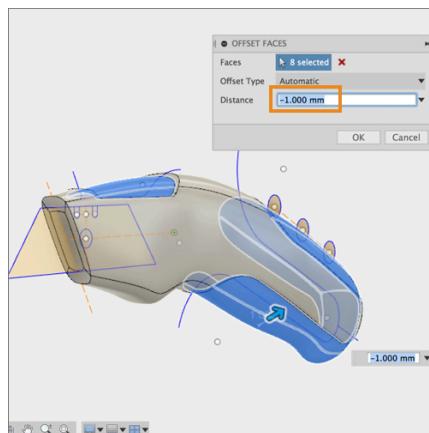
Step 14: – Repeat Combine Join

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



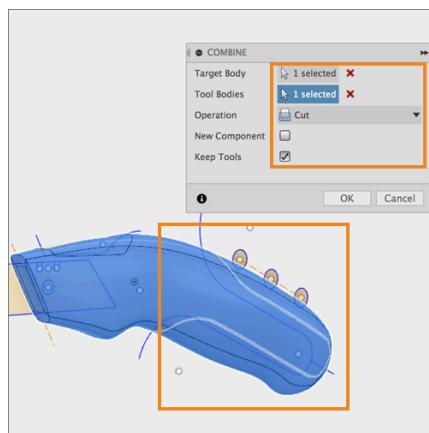
Step 15: – Cut Bodies with Combine tool

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



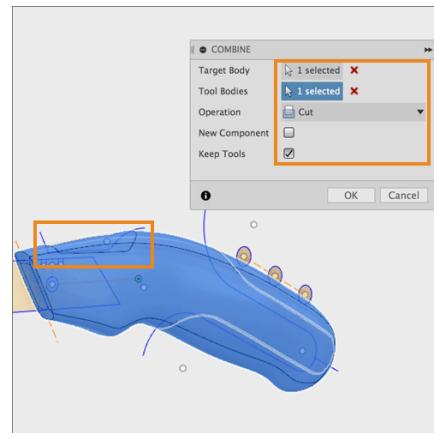
Step 16: – Cut Grip 1 with Combine tool

1. Go to the Modify drop-down menu and select **Combine**.
2. In the command dialog, set:
Grip 1 as the **Target Body**,
Knife Body as the **Tool Body**,
Cut as the **Operation**,
Check Keep Tools,
3. Click OK to finish the operation.



Step 17: – Repeat Combine Cut for Grip 2

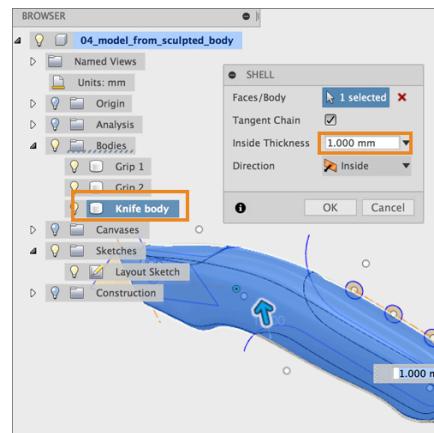
1. Repeat the previous step, but this time, cut **Grip 2** using **Knife Body**. Click OK to finish the operation.



Step 18: – Shell the knife body

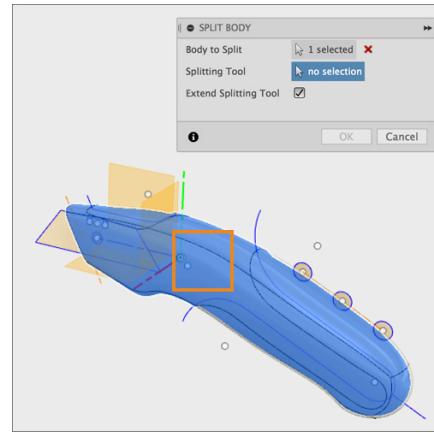
1. Now that we have our grips modeled, let's shell the inside of the knife body. Go to the Modify drop-down menu and select **Shell**.
2. Select **Knife Body** from the browser. Make sure that the shell thickness is **1 mm**. Click OK to finish.

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



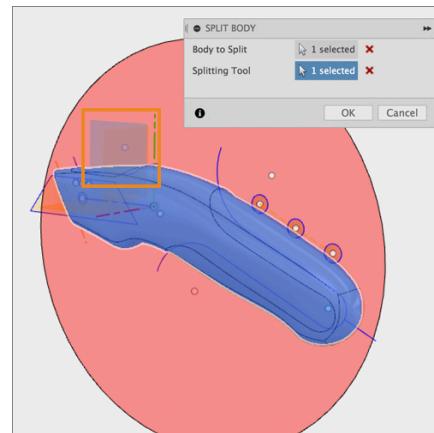
Step 19: – Split Knife Body into 2 pieces

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



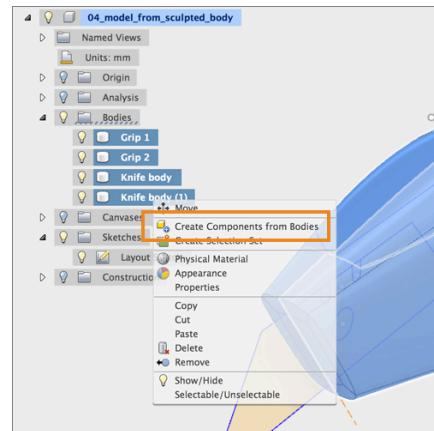
Step 20: – Split Knife Body into 2 pieces

1. Click on **Splitting Tool** to activate the selection
2. Select the plane that cuts down the middle of the **Knife body**. If you have trouble selecting the plane, zoom out until you can select it.
3. Click OK to finish the operation.



Step 21: – Convert Bodies to Components

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



Step 22: – Lesson complete!

1. You have completed the lesson!
2. You can now drag the components apart and see all the work we did around the grips and the knife body.

