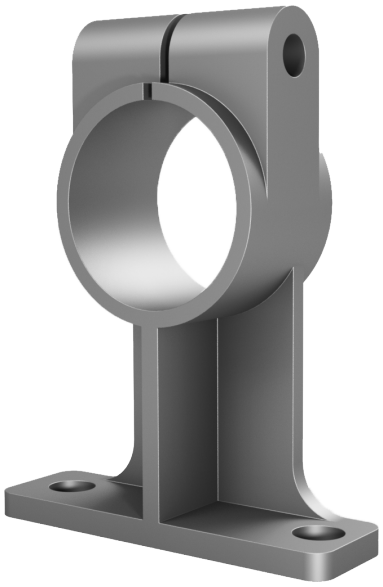
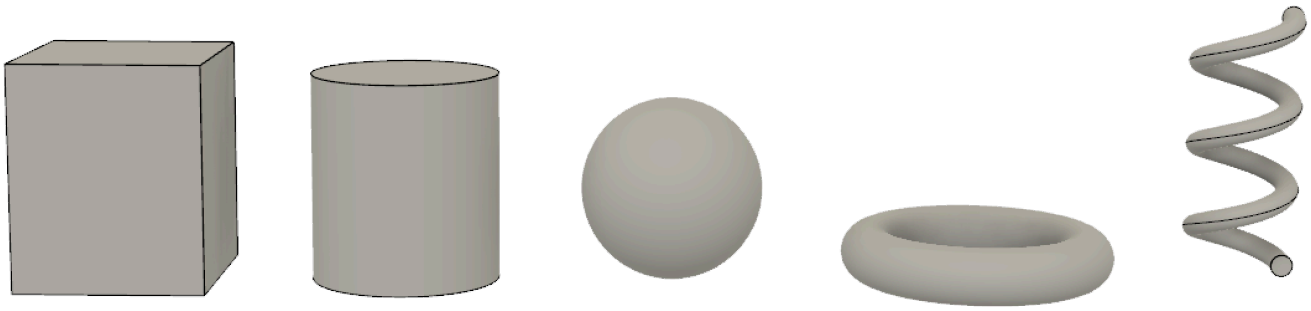
Fusion 360 | Introduction

For this lesson we'll be creating the 3D model of the split clamp shown below. When tackling any 3D design, it's essential to break down your complex part into manageable pieces. Looking at the part, we can see that a logical place to start might be the main cylindrical section of the design. By starting with this section of the design, all of the remaining features will easily fall into place!



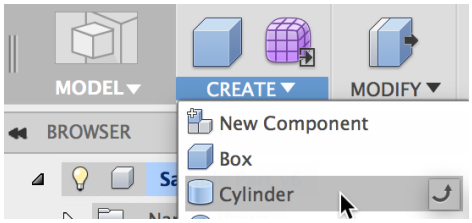
Step 1 – Start with a Primitive Cylinder

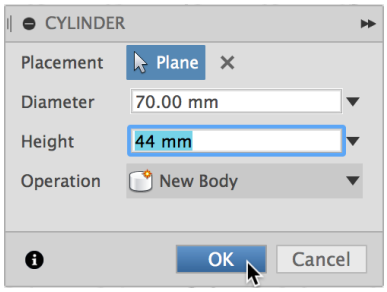
A simple way to create 3D objects within Fusion 360 is with Primitives – "building block" shapes that are easy to implement. Need to create one of the primitives seen below? There's a button for that!

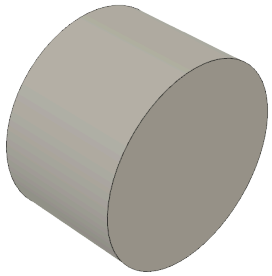


Box, Cylinder, Sphere, Torus, and Coil Primitives

To start things off, we'll click on the **Create** dropdown menu to view all of the solid model creation tools, and choose **Cylinder**. Next, we need to specify a plane to place the bottom of the cylinder on. Referencing the split clamp, we can see that the cylinder needs to rest on its side, so choose either of the orange, **vertical workplanes.** Next, we'll center the cylinder about the origin by moving the mouse onto the **origin** and **clicking once**. Next, move the cursor out to specify the diameter of the cylinder. Type **70 mm** and hit **enter** to set the cylinder's diameter. Lastly, within the Cylinder dialog window, we'll set the length to **44 mm** and click **OK**.



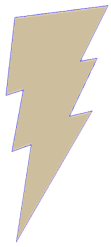




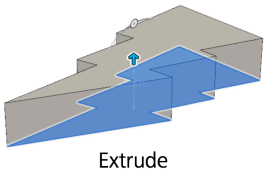
Congratulations! You've created your first 3D body inside of Fusion 360!

Step 2 – Create Your First Sketch

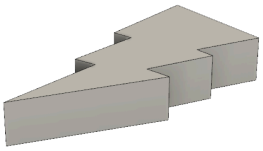
There will come a time when you want to create a 3D object that's more complex than a simple primitive. Want to make a lightning bolt? Unfortunately, there's no lightning bolt primitive command in Fusion 360. For these more complex shapes, we need to first create a 2D sketch, and then apply an operation to that 2D shape, creating a 3D object. Below are a few examples of this process.

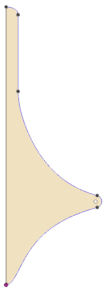


Sketch

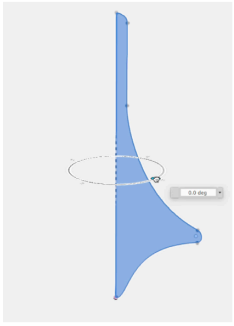


Extrude





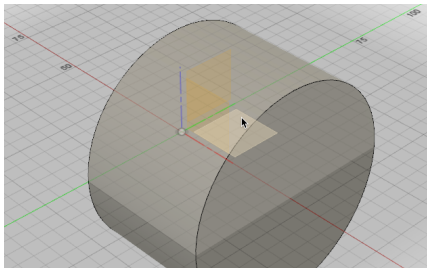
Sketch

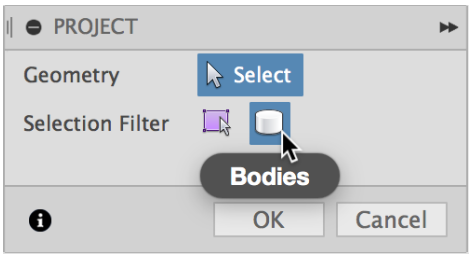


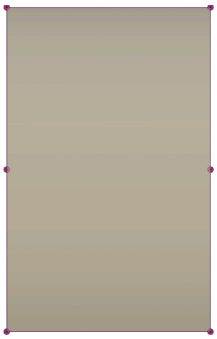
Revolve

To create the top portion of the split clamp, we'll first need a sketch. Let's create our first sketch by selecting the **Sketch** dropdown menu and choosing **Create Sketch**. Next, Fusion 360 will ask us to specify a flat surface to sketch on. This can be any of the default workplanes or any flat surfaces on any body.

For this sketch, we'll choose the **horizontal workplane**. While we can see the profile of the existing cylinder, no information is available for us to reference inside our sketch. We can "eye ball" our sketch, but we all know that's not the right way to do things. To truly reference the existing geometry, we need to project the 3D geometry into our 2D sketch plane. To do this, choose **Project/Include** from the **Sketch** dropdown menu, and select **Project**. Next, in the Project dialog window, change the **Selection Filter** to **Bodies**. This allows us to project an entire body into our sketch plane, and not just edges/faces that we select. Lastly, select the **cylinder** from the workspace to project its profile into the sketch. *Fun Fact*: From the side, a cylinder looks exactly like a rectangle.

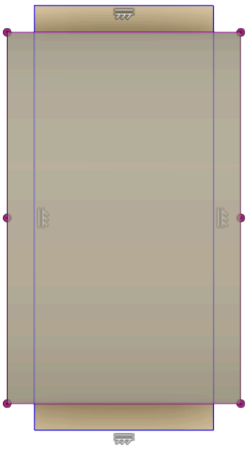






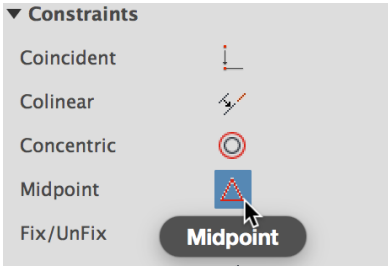
Before we create our first sketch, there's a **User Preference** I highly encourage everyone to use. Click on your name in the upper-right corner, and choose **Preferences**. Under the **Preview** tab, check the box for **Sketch – Color sketch geometry based on constraint status**. This will color code your sketches, and make them easier to understand!

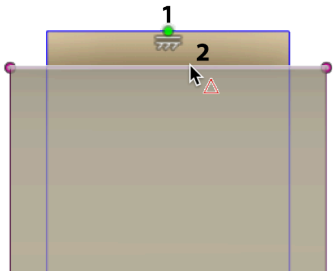
Next, to draw the rectangle shown below, we'll choose the **2-Point Rectangle** command **(Sketch > Rectangle > 2-Point Rectangle)**. With the 2-Point Rectangle tool active, **click** once to place one corner of the rectangle, move the cursor, and **click** again to complete the rectangle. To stop drawing rectangles, hit the **esc** key.

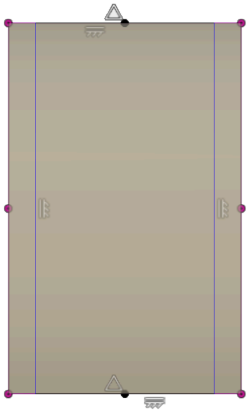


We're done drawing shapes for this sketch, and all that's left is to specify the position and size of this rectangle. We'll first lock down the rectangle's position by using **constraints** – rules between pieces of sketch geometry. The blue rectangle needs to be aligned with the middle of the projected purple geometry, and the top and bottom edges need to align, as well. To do this, choose the **Midpoint constraint** from the **Sketch Palette** on the right-hand side of the screen. Next, we'll select the piece of

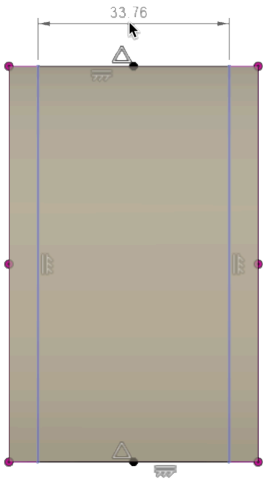
geometry that we want to move, in this case, the **green center-point** of the **top blue** line. Next, we'll select the **top purple** line, and you'll see the midpoint of the blue line has been moved to the midpoint of the purple line. **Repeat this step for the bottom lines**.







With the position set, the next step is to specify the rectangle's width with a sketch dimension – a tool for controlling the size and position of sketch objects. We'll start by choosing the **Sketch Dimension** command **(Sketch > Sketch Dimension)**. Next, select **both** of the vertical blue lines to specify the distance between them. Move the cursor to a position where the dimension is easily visible and **click** to place the dimension marker. In the text box, enter **28 mm** for the width, and hit **enter**. With the dimension in place, the rectangle is *fully defined*, and the geometry is now colored black.

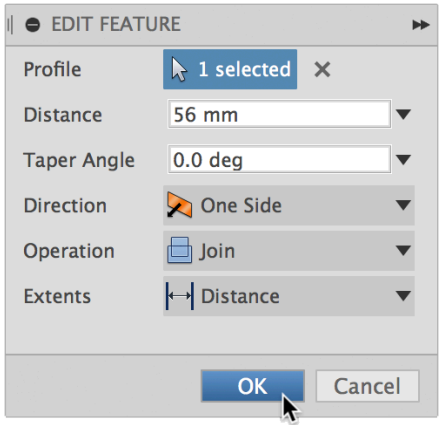


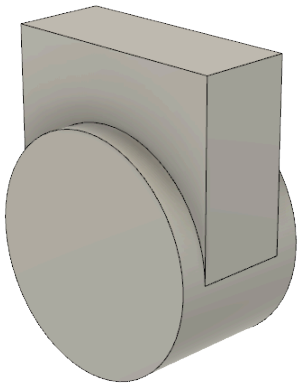
With our sketch complete, we'll choose **Stop Sketch** in the upper-right of the toolbar.

Step 3 – Create Your First Extrusion

We're now ready to extrude our rectangular sketch, turning it from a 2D sketch profile to a 3D object. To do this, choose the **Extrude** command **(Create > Extrude)**. For the **Profile** to extrude, we'll select the **rectangle** we recently sketched. Moving through the Extrude dialog window, we can now specify a **Distance** of **56 mm**. To simply extrude in one direction, we'll set the **Direction** to **One Side**. By default, Fusion 360 will assume that since we are extruding *through existing geometry*, we want to perform a

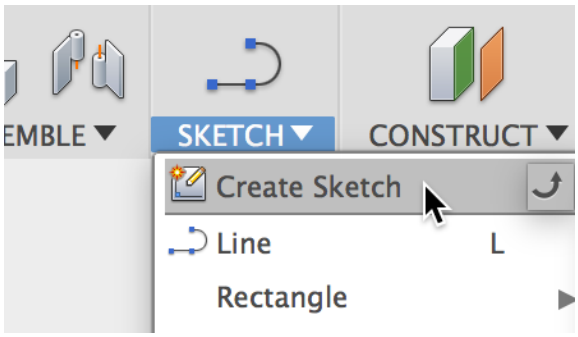
cut. While this is often the case, in this situation, we want to actually *add to* the existing geometry. To do this, we'll change the **Operation** to **Join**, and we'll be left with one single solid body that's a combination of the two shapes. Click **OK** to complete the extrude command.

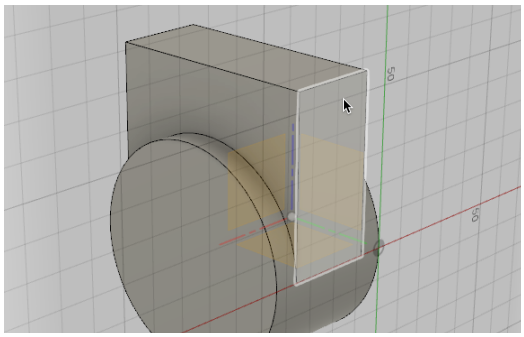




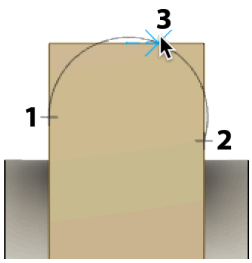
Step 4 – Sketch Top Portion

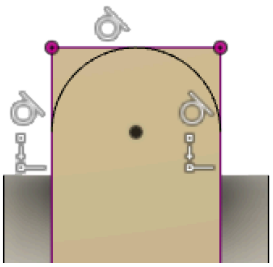
With the primary geometry in place, we're ready to modify the top portion to create the rounded top and through hole. We'll start off with the **Create Sketch** command (**Sketch > Create Sketch)**, and select either of the **side faces** of the extruded rectangle for the surface to sketch on.



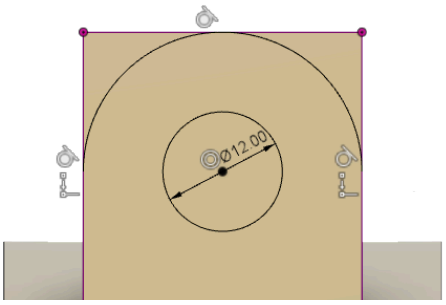


Looking at the split clamp design, we'll first draw the geometry necessary for the perfectly rounded top. To do this, we'll start with the **3-Point Arc** command **(Sketch > Arc > 3-Point Arc**). The 3-Point Arc command needs three points: a starting point, an end point, and a center-point (in that order). To draw the proper arc, we'll start by clicking on **one of the vertical edges** of the rectangle, then the **other vertical edge**, and lastly the **top horizontal line**. Next, to properly constrain the arc tangent to all three lines, we'll choose the **Tangent constraint** from the Sketch Palette. With the tangent constraint active, select the **arc**, followed by one of the three **lines**. You'll see the arc change so that it becomes tangent to the selected line. With the tangent constraint still active, repeat the last step for the **remaining two lines**. When finished, the arc should look like the picture below. Now, if our rectangular portion changes size (becomes wider or taller), the arc will *always* be the right size.





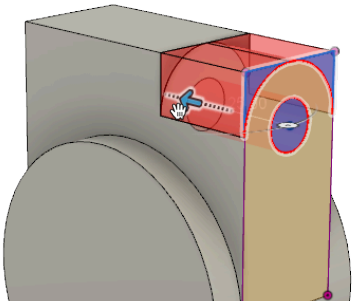
Next, for the through hole, we'll draw a circle with the **Center Diameter Circle** command **(Sketch > Circle > Center Diameter Circle)**. Click to place the center of the circle on the **center-point** of the arc shown below. Move the cursor out, type **12 mm** into the text box, and hit **enter** to set the circle's diameter.

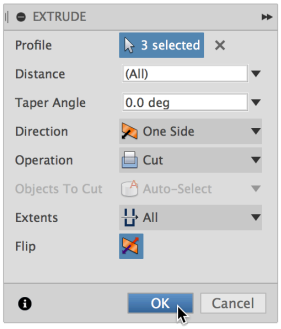


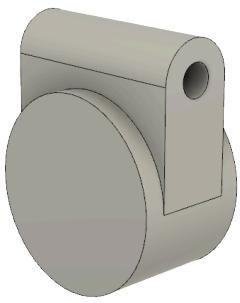
Exit the sketch by clicking **Stop Sketch**.

Step 5 – Extrude Top Portion

With the sketch complete, we're all set to make the necessary cuts to the top portion of the split clamp. We'll utilize the **Extrude** command **(Create > Extrude)** and see how it's a versatile tool for adding to or *removing from* existing geometry. With the Extrude tool active, select the **two triangular portions** of our previous sketch, as well as the **circle**. Next, using the **arrow**, drag the profiles *through* the existing geometry and see that Fusion 360 automatically performs a **cut**. Now we can drag these profiles through until they are "far enough," but this may not always do the trick! Imagine if our split clamp becomes wider – suddenly our extrusion doesn't go all the way through. To fix this, within the Extrude dialog window, change the **Extents** from Distance to **All**. Click **OK** to complete the Extrude command.

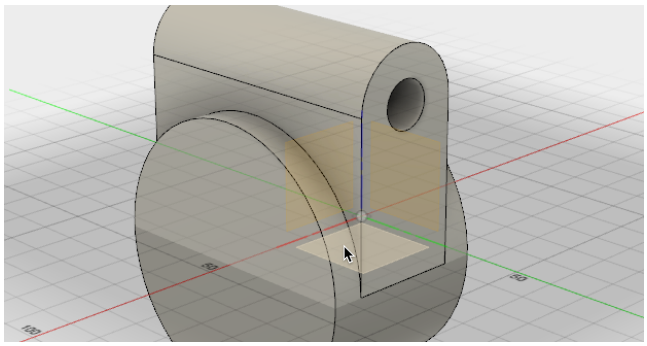


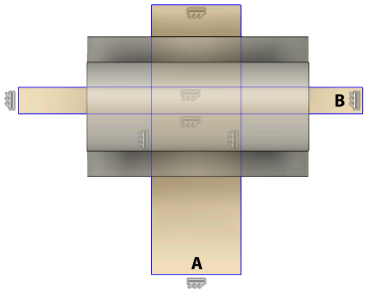




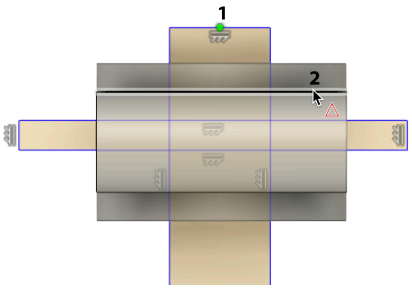
Step 6 – Sketch T-Shape Extrusion

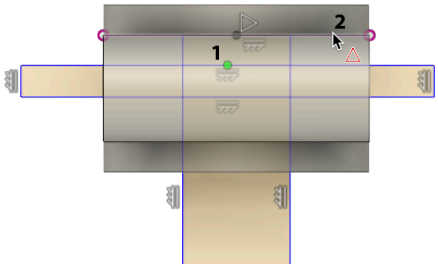
Next up is the middle portion of the split clamp, and just like before, we're going to sketch out its profile and extrude it down. We'll start things off by **Creating a Sketch** on **the horizontal workplane.** Next, choose the **2-Point Rectangle** command, and draw two separate rectangles similar those shown below.

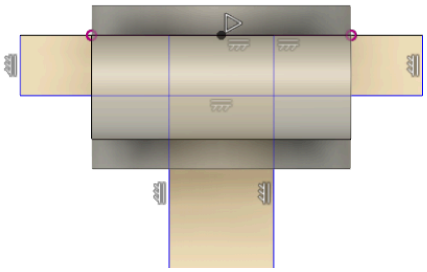




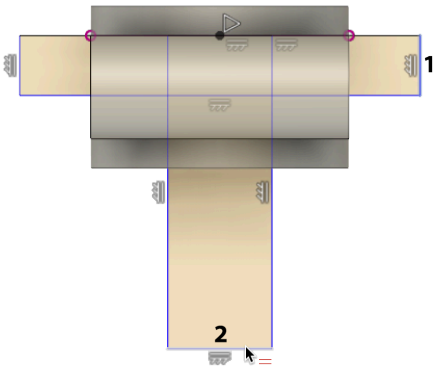
Next, we'll apply constraints to our rectangles ("*A*" and "B") to properly position them in relation to our existing geometry. Choose the **Midpoint** constraint from the Sketch Palette, select the **midpoint** of the **rectangle A**, followed by the **top edge** of the **rounded portion** of the split clamp shown below. Repeat this step for **rectangle B**.

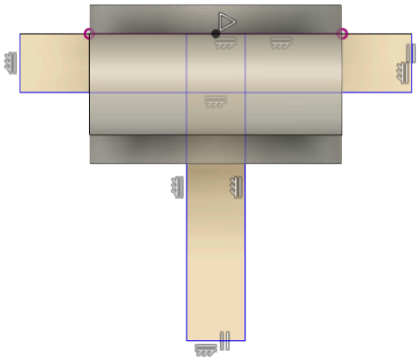






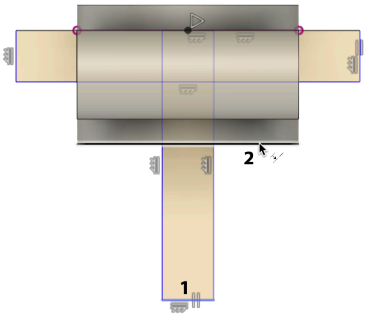
Next, we want to setup a constraint that ensures both rectangles have the same width. To do this, choose the **Equal** constraint from the Sketch Palette. Next, select edges **1** and **2** shown below, and they'll be constrained to always be the same size!

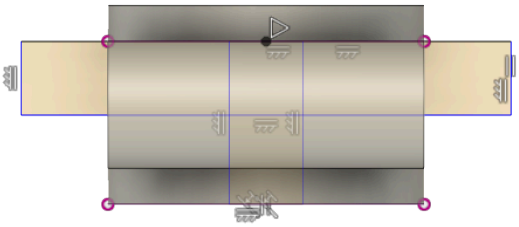




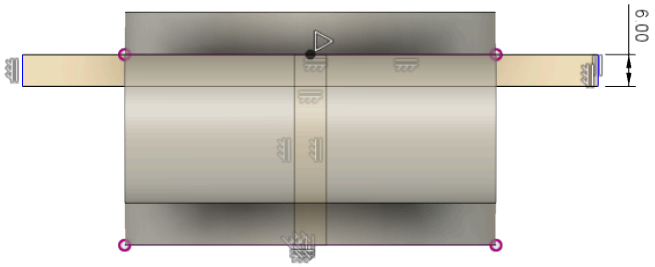
The last constraint is the align the bottom of the "T" shape with the bottom-most part of the existing geometry. To do this, choose the **Colinear** constraint from the Sketch Palette. Next, choose the **bottom**

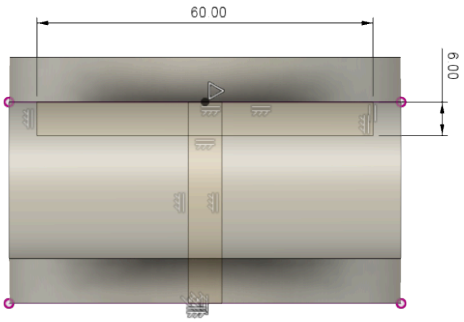
edge (1), followed by the **bottom-most edge** of the existing geometry (2) shown below. The geometry is now fully constrained and ready to be dimensioned!





With our sketch fully constrained, we can now add dimensions to properly size the sketch geometry. Let's first set the width of our rectangles to 6mm. To do this, choose the **Sketch Dimension** tool, select the **edge** shown below, and set the dimension to **6 mm**. Lastly, use the **Sketch Dimension** tool do set the length of the top rectangle to **60 mm** as shown below.

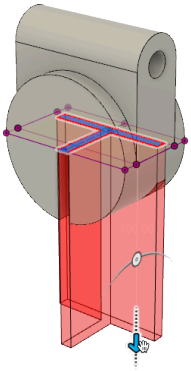


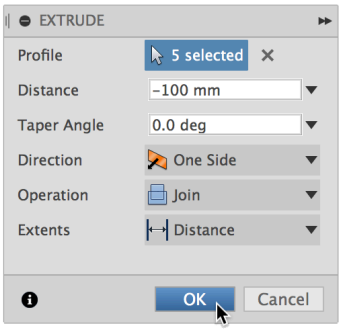


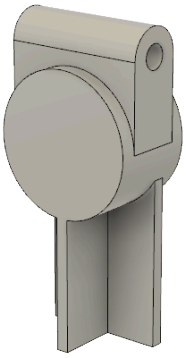
Exit the sketch by clicking **Stop Sketch.**

Step 7 – Create T-Shape Extrusion

Next up is creating solid geometry from our recent sketch. To do this, choose the **Extrude** command **(Create > Extrude)**, and select all of the profiles that make up the entire 'T' shape. If you're having a hard time selecting the profiles, you can **left-click** and **hold** to bring up a list of items under the mouse. Then, simply select the **Profile** from the list. With all of the profiles selected, enter a distance of **-100 mm** to extrude *down.* Next, in order to *combine* this extrusion with the existing geometry, change the **Operation** from Cut to **Join**. Click **OK**.

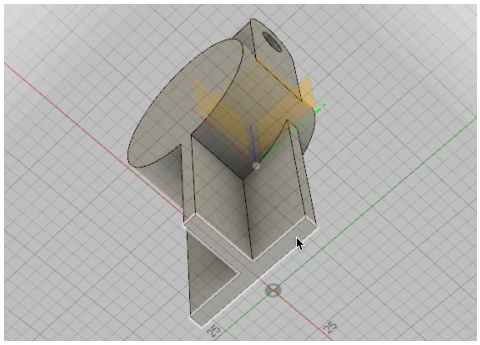


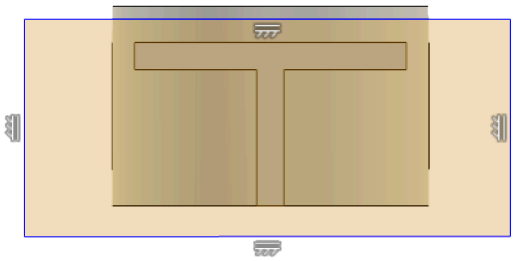




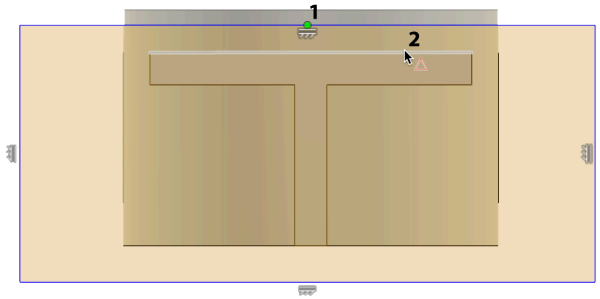
Step 8 – Sketch the Base

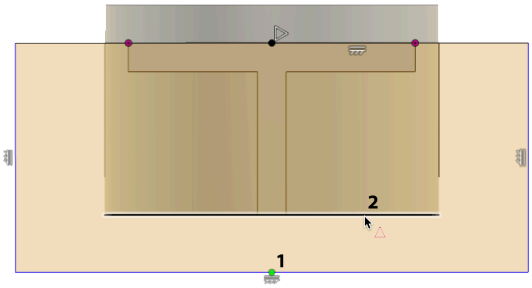
As you can probably imagine, we're going to create the base of the split clamp in a similar fashion to the previous portions of the design: sketches and extrusions. Let's start by choosing the **Create Sketch** command, and selecting the **bottom face** of the split clamp as the surface to sketch on. Next, choose the **2-Point Rectangle** command, and draw a rectangle similar to the one shown below.



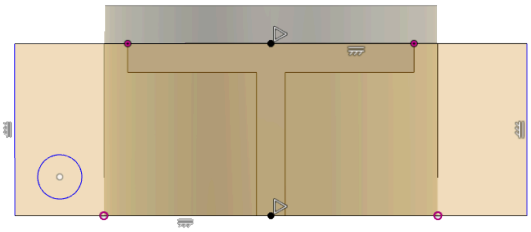


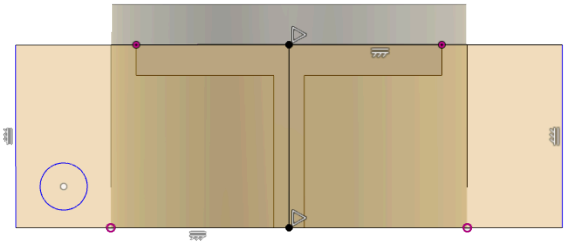
Next, with the **Midpoint** constraint, we'll center the top and bottom of the rectangle with the two existing edges shown below.



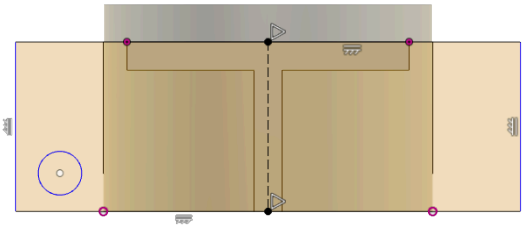


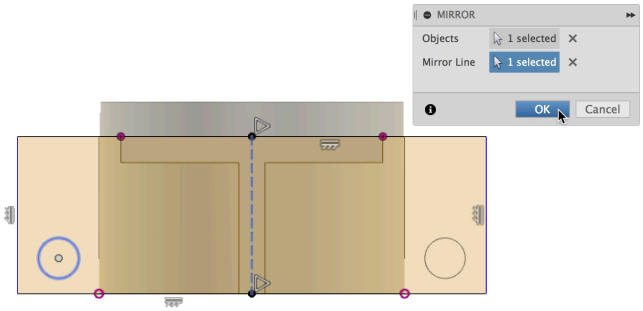
Next, we'll draw a circle that represents *one* of the holes in the base of the split clamp. Use the **Center Diameter Circle** command, and draw a circle resembling the one shown in the image below. Next, with the **line** tool, draw a line that extends from the **middle** of the **top line segment** to the **middle** of the **bottom line segment** – the line tool should automatically snap to the midpoints.



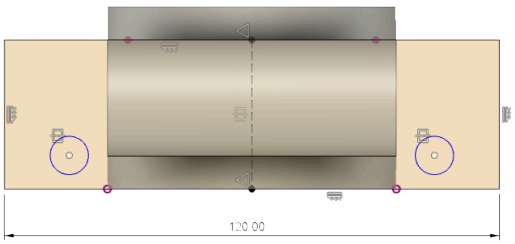


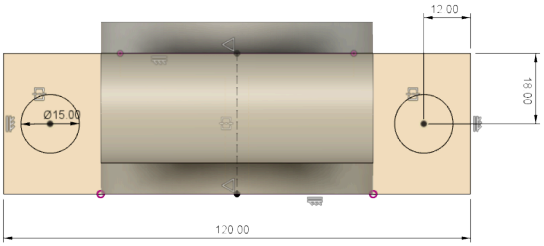
We're going to use the centerline to *mirror* the circle over to the other side, but you'll notice that it currently splits our sketch into multiple closed profiles. To use this line just for *reference*, we'll need to convert it to a Construction line. To do this, **select the line**, and choose **Normal / Construction** from the Sketch Palette. Next, to mirror the circle we'll choose the **Mirror** command **(Sketch > Mirror)**. For the **Object** to mirror, select the circle, and for the **Mirror Line**, select the center construction line. Click OK to complete the mirror command.





With all of our sketch geometry drawn and constrained, we're ready to apply dimensions. With the **Sketch Dimension** tool, we'll first set the length of the rectangle to **120 mm**. Next, we'll dimension the **diameter** of the **circle** to **15 mm.** Lastly, we'll dimension the **circle** to be **12 mm** from the side of the rectangle and **18 mm** from the top of the rectangle.

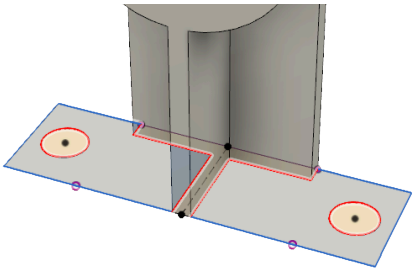


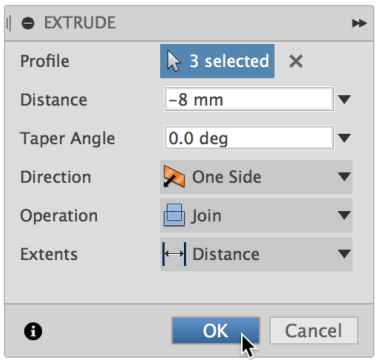


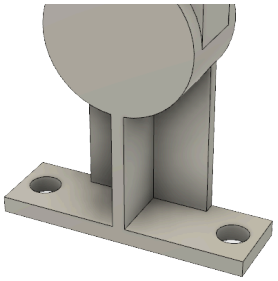
Looks like our sketch is complete! Let's exit the sketch by clicking **Stop Sketch**.

Step 9 – Extrude the Base

To turn our sketch into a 3D object, we'll choose the **Extrude** command and select the profiles shown below. Drag the **arrow** "up" to extrude the profiles towards the existing geometry – you'll see this is the *negative* direction. Because of this, for the **Distance**, enter **-8 mm**. Ensure the **operation** is set to **Join** in order to be left with a single solid body. Click **OK** to complete the Extrude command.

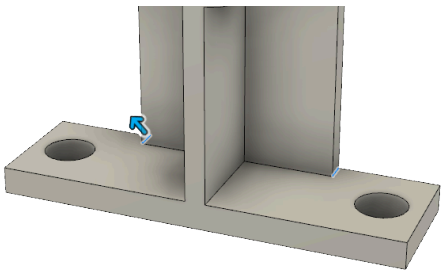


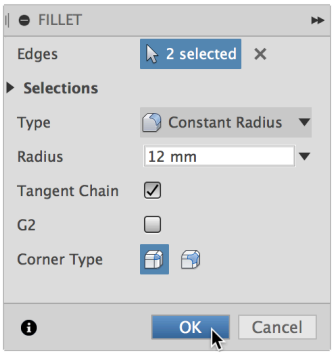


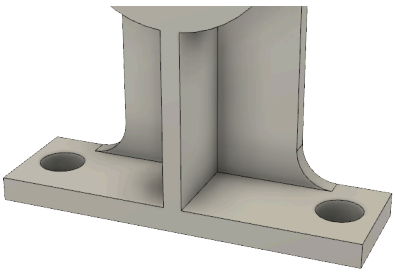


Step 10 – Round Sharp Corners

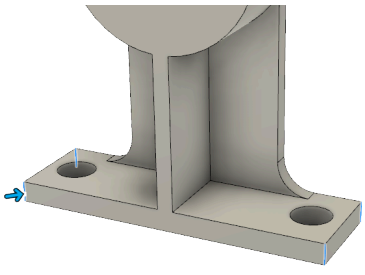
It's very common to create a smooth transition along an otherwise sharp edge when creating a 3D model, and in Fusion 360, this is referred to as a **Fillet**. In this split clamp model, we can see that we need to create a smooth transition between the base and the top of the "T" section. We'll kick things off by selecting the **Fillet** command **(Modify > Fillet)**. Being that we already have 3D geometry that we simply want to *modify*, it makes sense that this is where we'd be able to find a tool such as Fillet. Next, select the **two edges** shown below – *NOTE: you don't have to hold shift!* Lastly, we'll specify the **radius** of this smooth transition to be **12 mm**. Click **OK** to complete the Fillet command.

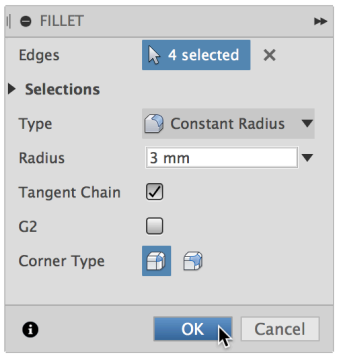


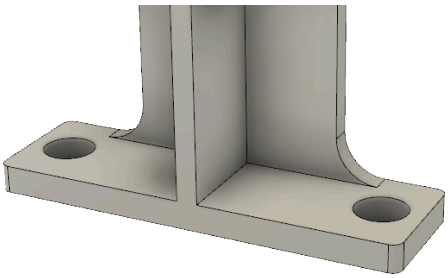




Next, we'll round off the four **outer edges** of the **base** with the **Fillet** command. Select **four edges** shown below, set the **radius** to **3 mm**, and click **OK.**

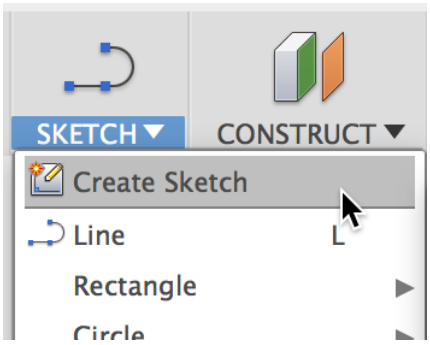


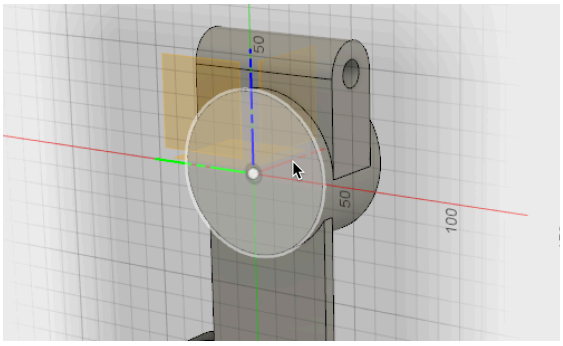




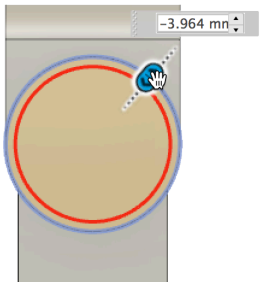
Step 11 – Sketch Hole & Split

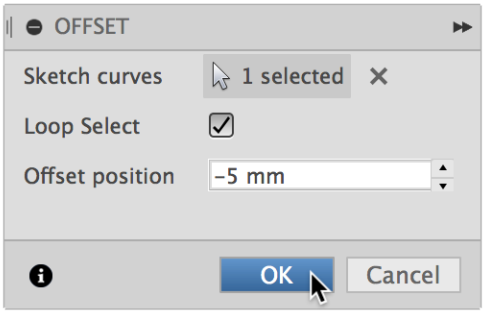
All that's left to complete the split clamp is to create the features for the main hole, as well as the slight split that allows the clamp to tighten down. Let's start by choosing the **Create Sketch** command, and selecting the **flat, circular face** on the **back** of the clamp shown below.

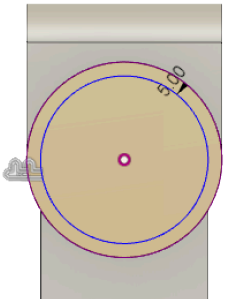




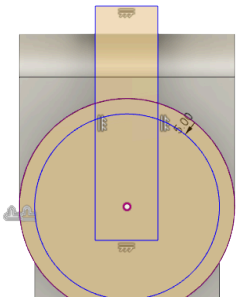
Next, we want to create a hole through the cylindrical portion that always leaves us with **5 mm** walls. To do this, choose the **Offset** command **(Sketch > Offset)**. Next, select the existing circular edge, and you'll see the offset interface appear. Drag the slider **in** to offset inwards, and notice that this results in a **negative number**. Because of this, we'll set the **Offset Position** to **-5 mm** and click **OK**.

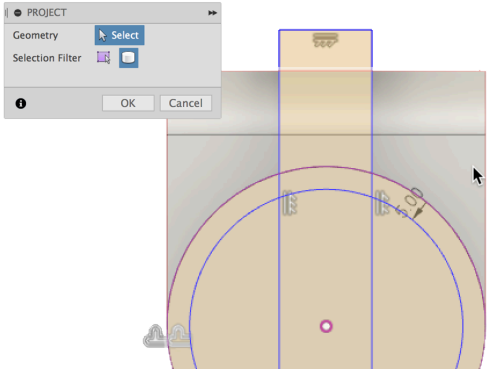


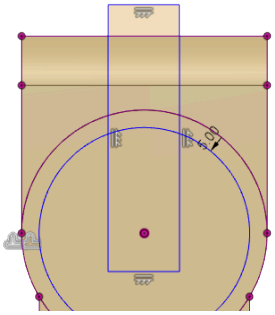




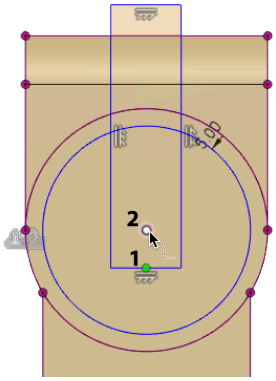
Next, we'll create the geometry necessary for the *split* in the split clamp by choosing the **2 Point Rectangle** command and drawing a rectangle similar to the one shown below. To make a robust 3D model, it would be nice to have the top of this rectangle match the very top of the existing geometry. Again, to reference geometry outside of the sketch, choose the **Project** command **(Sketch > Project / Include > Project)**. Change the **Selection Filter** to **Bodies**, and select the split clamp body. You should now be able to reference all of the existing geometry of the split clamp in this sketch.

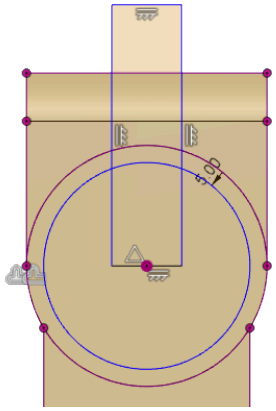




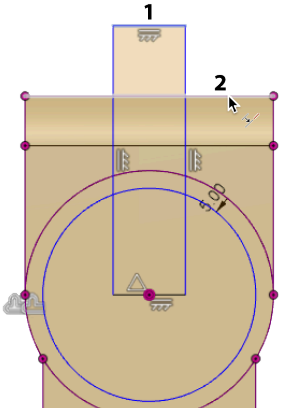


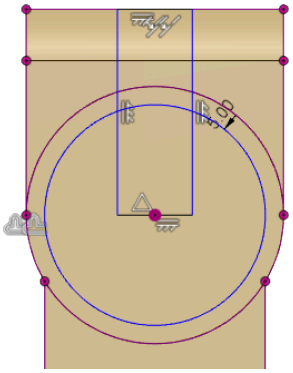
Next, we'll use **constraints** to setup the "rules" that keep the sketch geometry positioned where we want it. Let's first position the bottom of the rectangle by choosing the **Coincident** constraint in the **Sketch Palette**. Select the **green midpoint** of the **bottom** of the rectangle (1), followed by the **center point** of the circle (2). These two pieces of geometry are now positioned together!



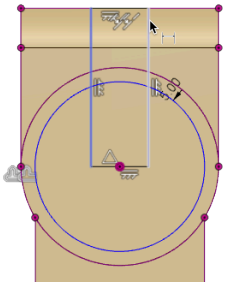


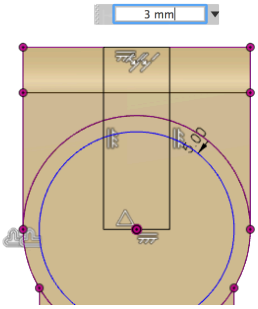
Next, we'll constrain the top of the rectangle to align with the top of the existing geometry. That way if the previous features ever change, the rectangle will parametrically update! To align these two lines, choose the **Colinear** constraint from the **Sketch Palette**. Next, choose the **top** of the rectangle (1), followed by the **top** of the existing geometry (2). These two lines are now colinear!

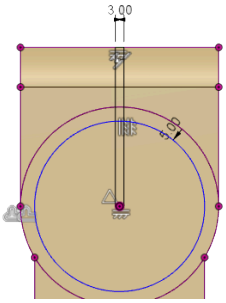




Our rectangle has all the constraints it needs, as it can now only its width can change. To drive the width, choose the **Sketch Dimension** command **(Sketch > Sketch Dimension)**. Next, select **both vertical** edges of the rectangle, and place the dimension in an easily visible location. Lastly, enter **3 mm** for the distance.



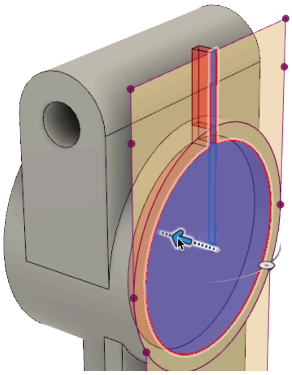


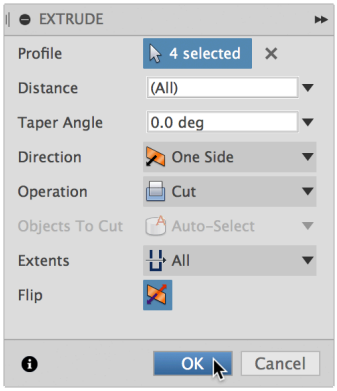


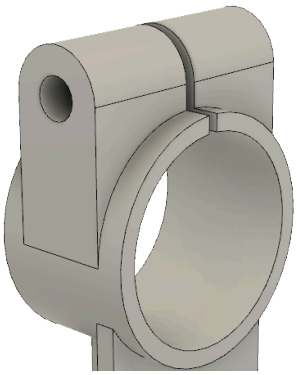
Lastly, exit the sketch by clicking **Stop Sketch**.

Step 12 – Extrude Hole & Split

With our sketch complete, we'll choose the **Extrude** command to remove the remaining geometry. Select the profiles shown below, and **drag** the Extrude **arrow** *through* the model, performing a cut. Next, in order to extrude through all of the material – regardless how much – change the **Extents** from Distance to **All**. This parametrically drives the extrusion to always cut through all of the material it sees, regardless if this is 5 mm or 5 cm. Click **OK** to finish the extrusion!

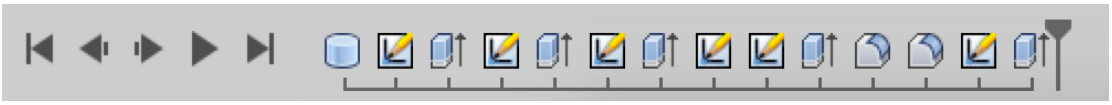




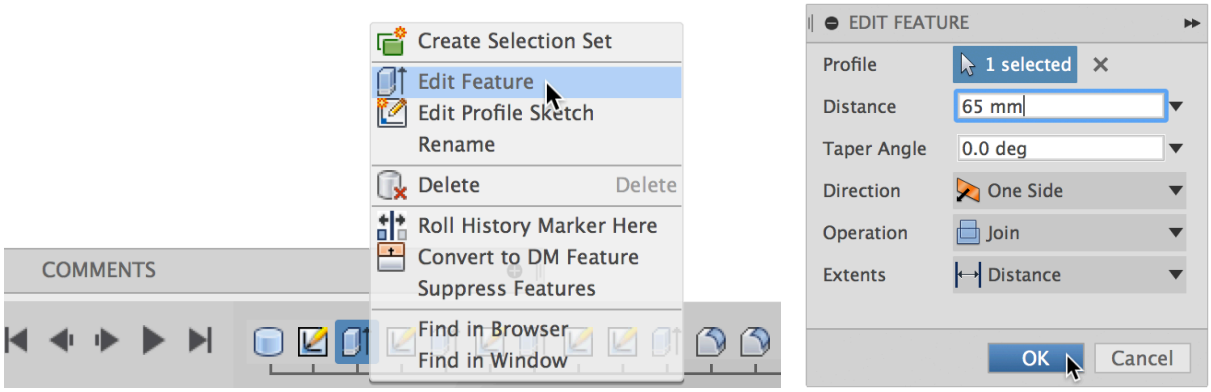


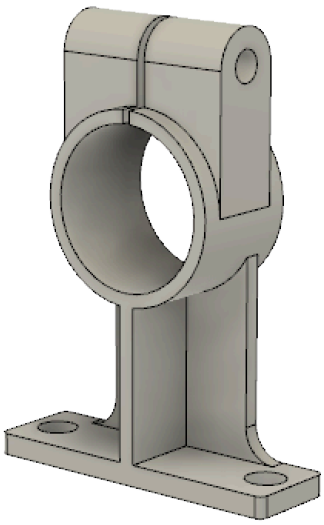
Step 13 – Modify Design with Parametric Timeline

The split clamp is finished, but there's one step that's critical to learn – how to make changes to an existing Fusion 360 design. Sure, you can pile features on top of your existing geometry, but this is a messy method that will result in a *less than robust* 3D model. To truly make changes to a model in Fusion 360, you need to take full advantage of the **Parametric Timeline**.



Let's say we want to make the top feature of the split clamp taller. What we need to do is find the feature associated with the height of the split clamp, and edit it. If you hover over a **feature** in the timeline, the associated geometry will be **highlighted** in the workspace. This is useful when you can't remember which feature was used for a certain part of a 3D model. **Hover** the cursor over the first **Extrude** feature in the timeline (the third feature) and see the extrusion highlighted in the workspace. **Right-Click** on the Extrude feature and choose **Edit Feature**. The timeline will roll back to that point in time, and allow you to make a change to that feature. Change the Extrude **Distance** from 56 mm to **65 mm**. Then click **OK**, and see how all of the subsequent "downstream" features to the right are re-calculated.





That wraps up the Fusion 360 Introduction class! You now have the fundamentals of Fusion 360 – sketching, constraining, dimensioning, extruding, modifying existing geometry, and utilizing the parametric timeline! Congratulations on being dangerous with Fusion 360.

Happy Making!