CAD and 3D modeling for FTC with PTC Creo - Parts

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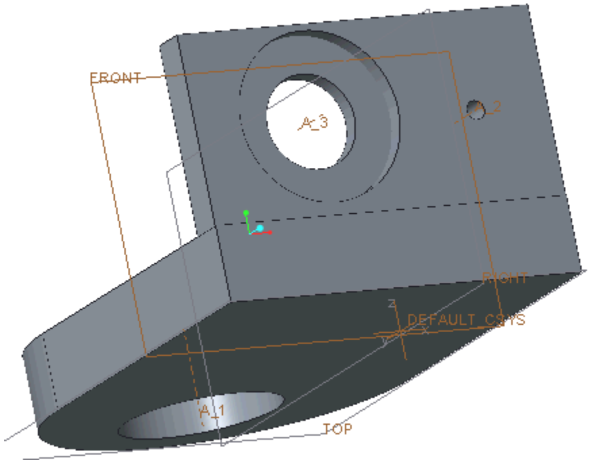
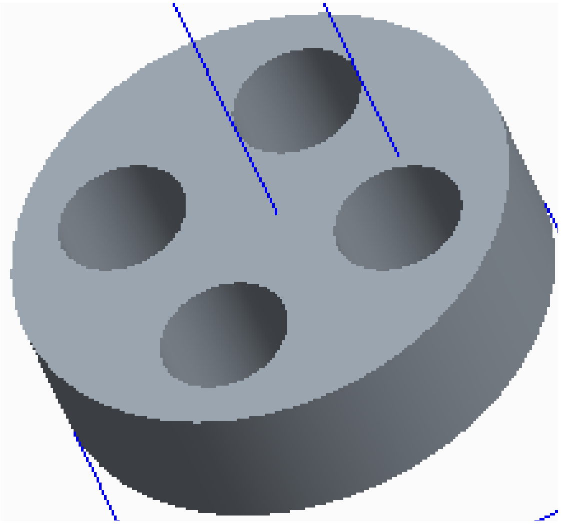
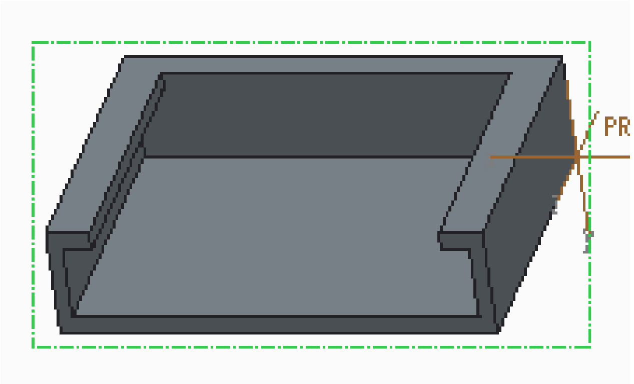
PTC Creo Parts

For this exercise we will use PTC Creo Parametric to model a variety of 3D parts.



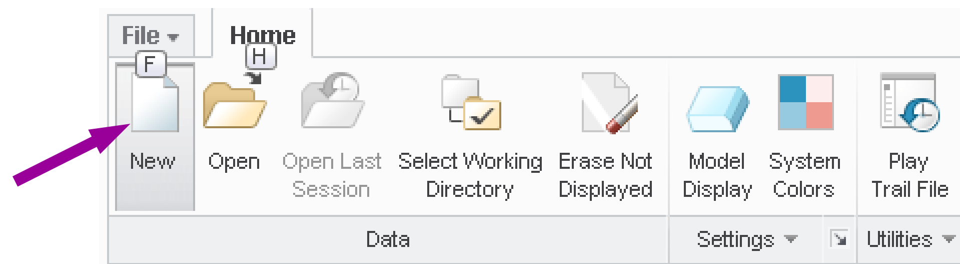




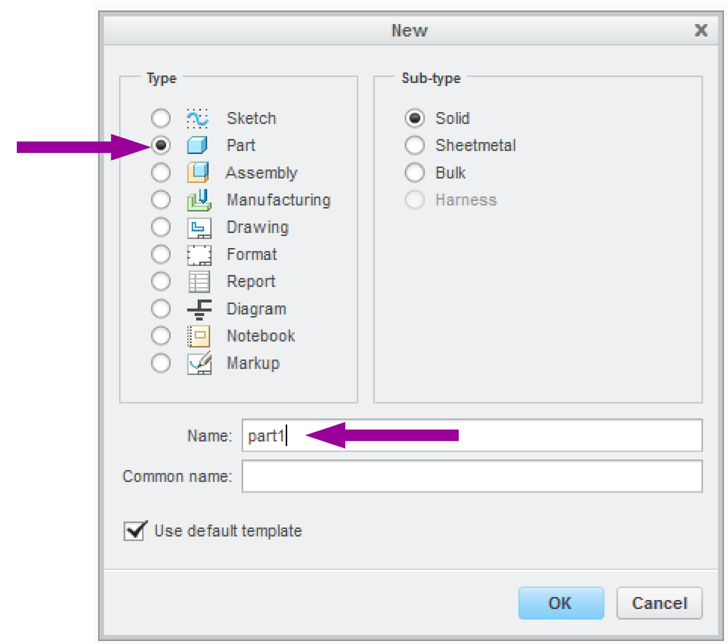
Starting with Creo Parametric

Start PTC Creo Parametric 3.0.

You may see some “Resource Center” windows after starting Creo Parametric; if you do, close them.

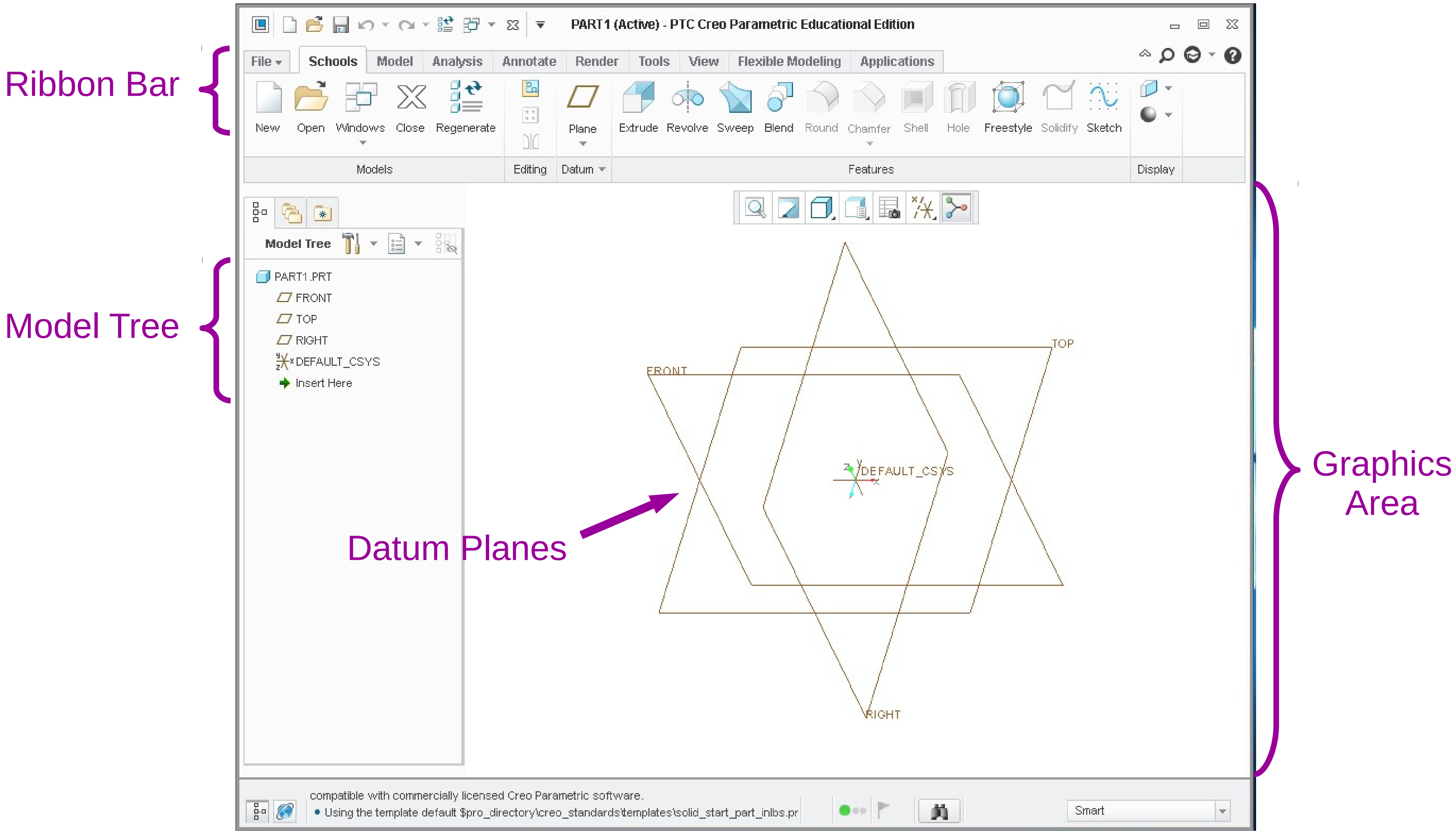


Click the **New** icon at the top left corner to create a new file.

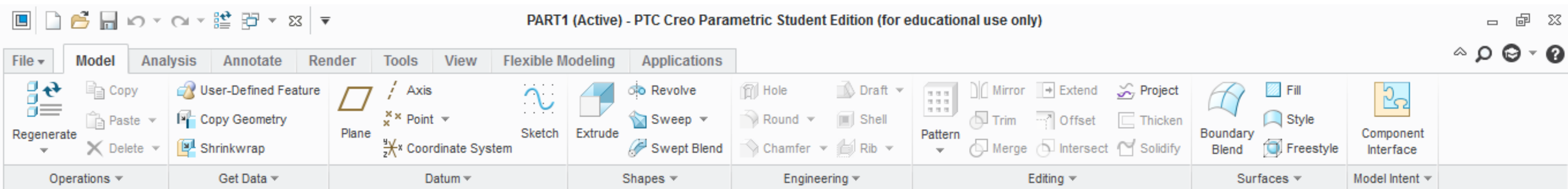


For this exercise, we are creating a *part*. Make sure **Part** is selected, then enter a name (“*part1*”), and click **OK**.

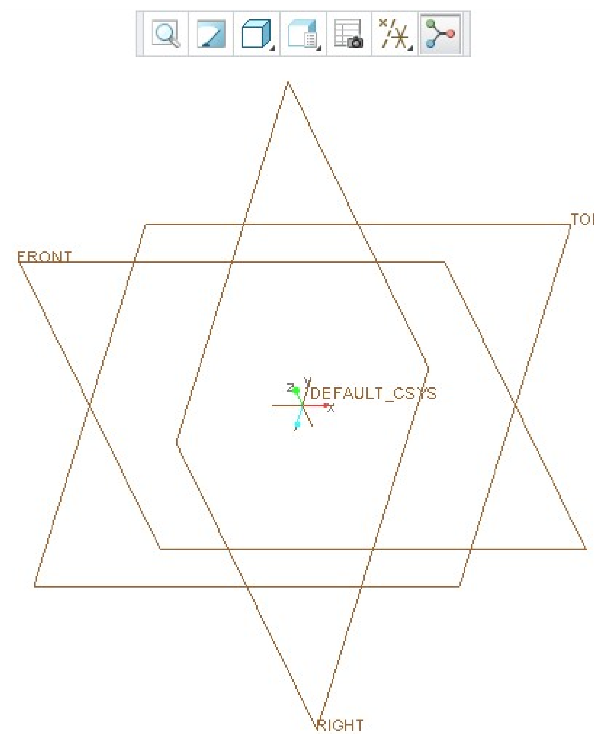
After starting a new part, your screen will look like:



The **Ribbon Bar** contains tools for making *sketches*, *extrusions*, and other model operations.

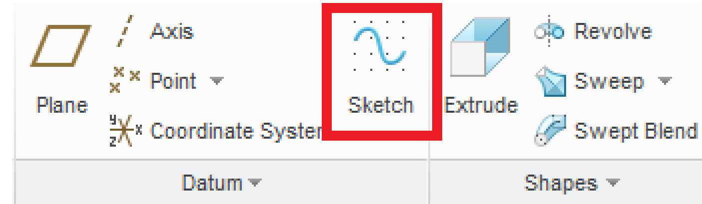


In the **Graphics Area** we see three *datum planes*. In mathematics they would be called *x*, *y*, and *z axes*; in Creo they're referred to as *Top*, *Front*, and *Right* planes.

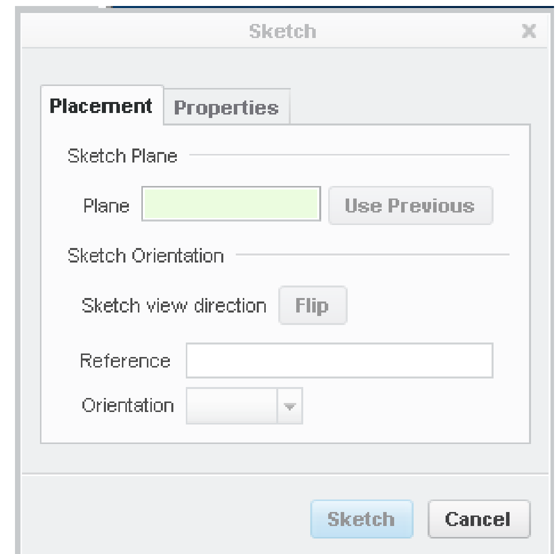


Sketching basics

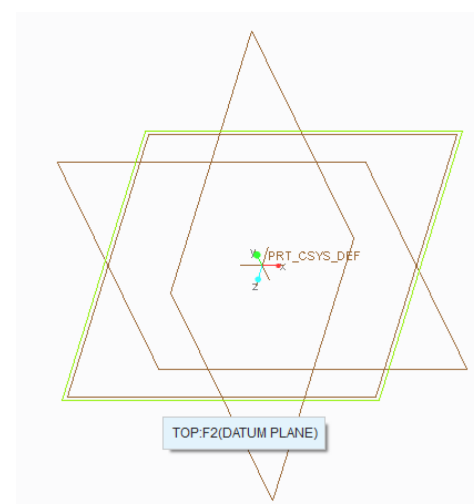
The first step to creating a model is to create a *Sketch*. A Sketch is a 2D projection of part or all of the 3D model.



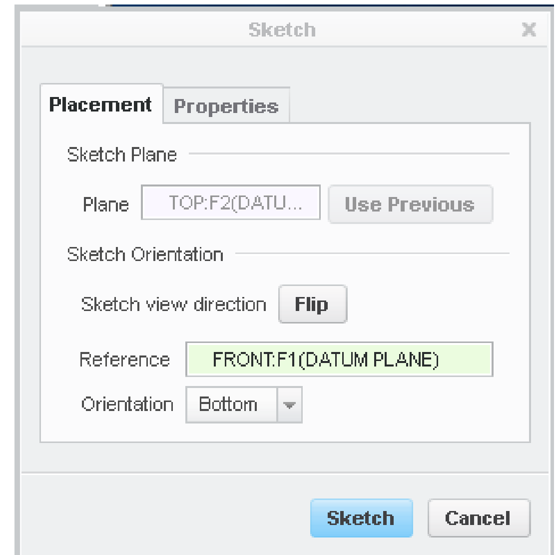
Click on **Sketch** in the ribbon bar. The Sketch dialog pops up, to determine what plane to draw the sketch on.



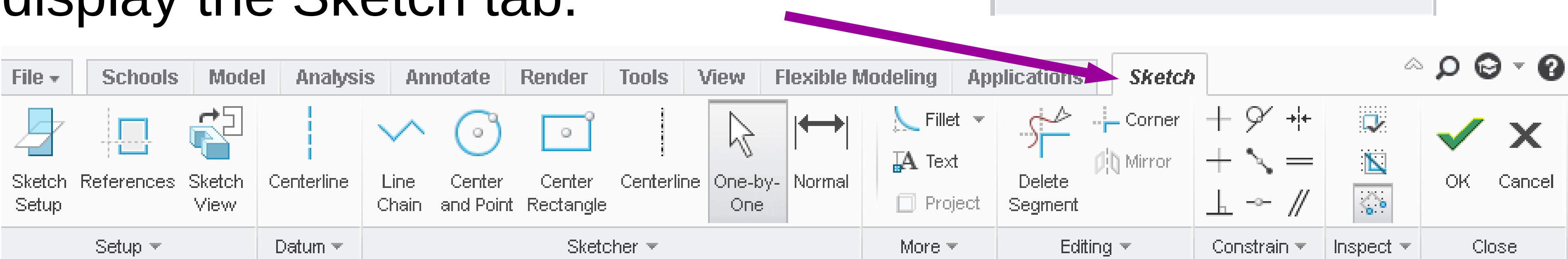
In the graphics area, hover the mouse over the border of the **Top** plane until it is outlined in green, then left-click to select it.



Click **Sketch** in the dialog box to begin sketching on the selected plane.



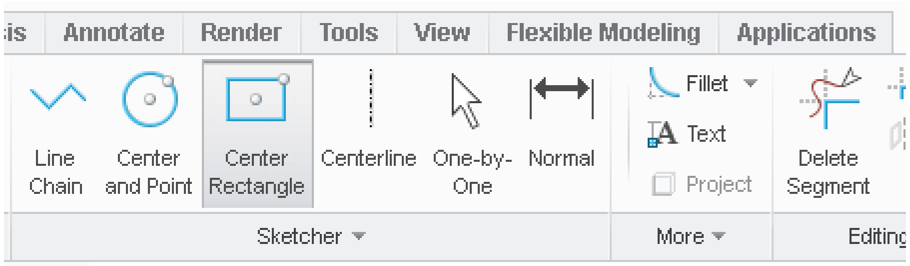
The ribbon bar changes to display the Sketch tab.



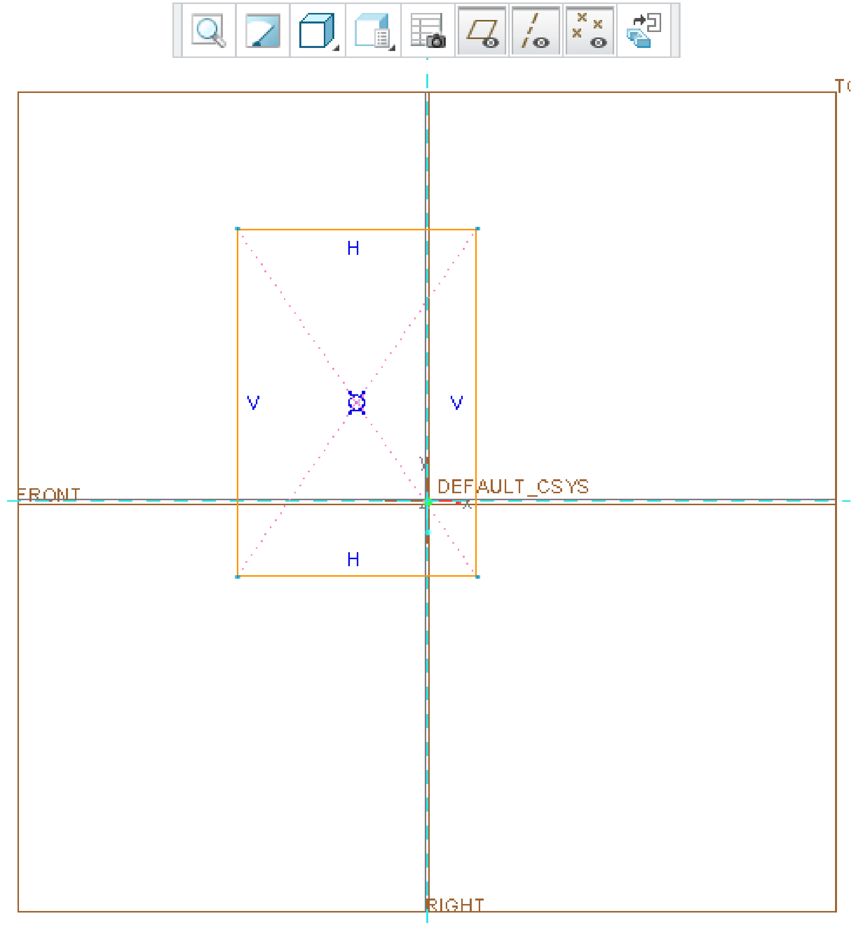
The Sketch tab contains many tools and shape primitives for creating sketches, such as lines, circles, rectangles, ellipses, etc.

Creating a shape

Select the **Center Rectangle** tool.

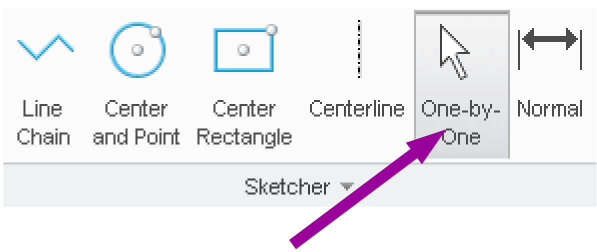


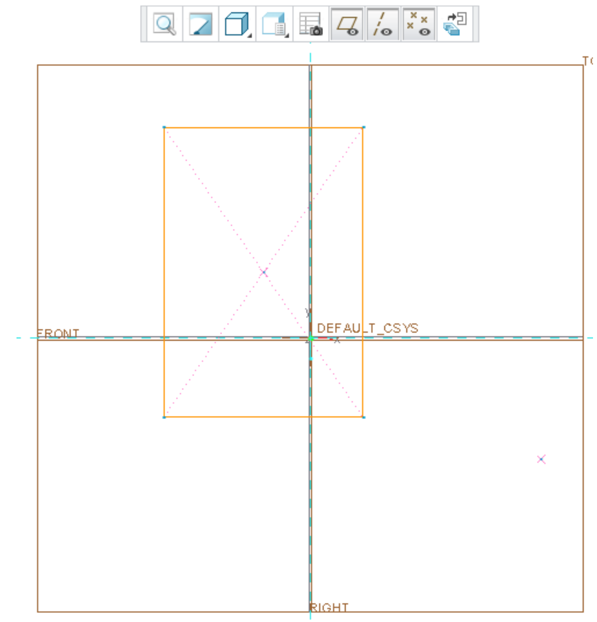
Left-click in the graphics area to set the center of a rectangle. Move the mouse pointer away from the center and click again to form the rectangle.



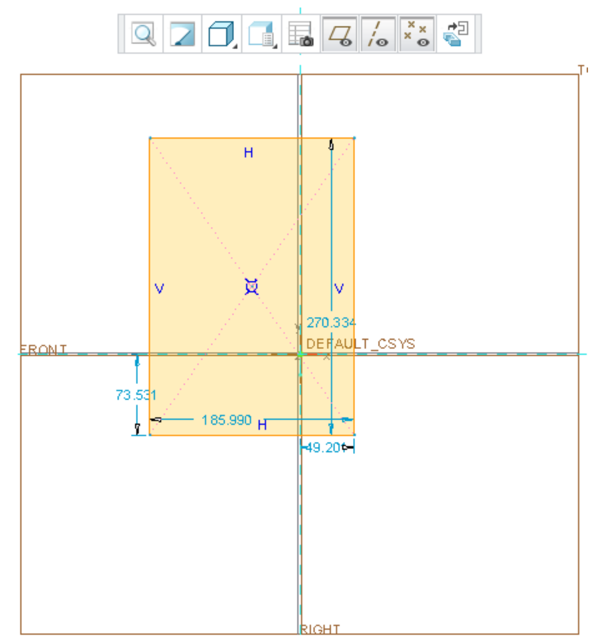
You'll be left with an orange rectangle.

Although we have a rectangle, it's not yet fully placed until it's deselected. Either left-click on the select tool, or middle-click to deselect everything.





If the shape is properly “closed”, it will turn solid orange.



Changing the model view

As with assemblies, the mouse can be used to change the view orientation of a part or sketch.

1. Dragging with the middle mouse button rotates the view.

2. SHIFT + middle mouse button pans the model view.

3. CTRL+D resets the model view to a “default” orientation.

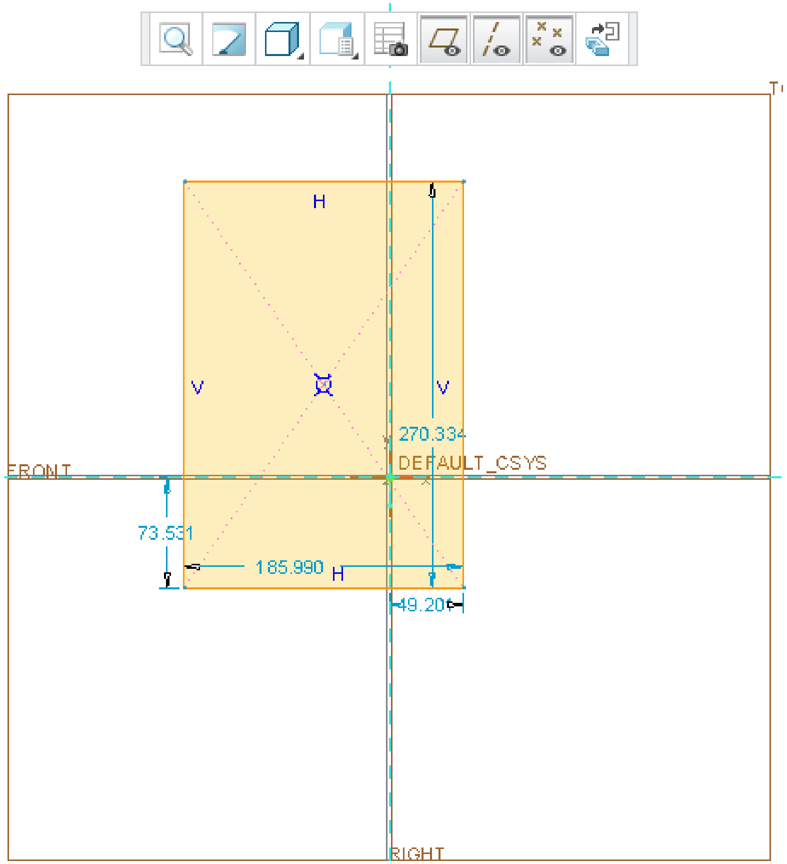
4. The mouse scroll wheel will zoom the view in and out.*

5. When sketching, the Sketch View will orient the sketch perpendicular to the screen.

*Windows 10: If the mouse scroll wheel isn't working, be sure “Scroll inactive windows” is turned off in Windows' Mouse settings.

Setting precise dimensions

When we freehand draw a shape, its dimensions are not very precise. The dimensions of our rectangle are given by four numbers here.

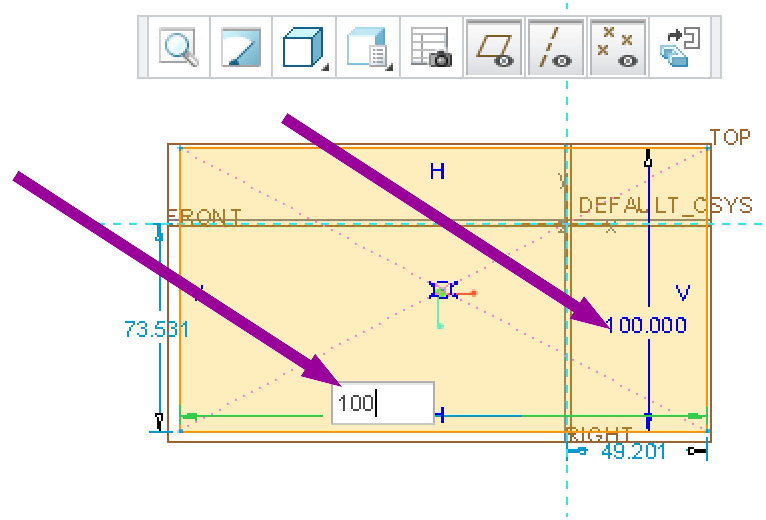


Two of the numbers represent the width and height of the rectangle.

The other two numbers are the distance between a side and a datum plane.

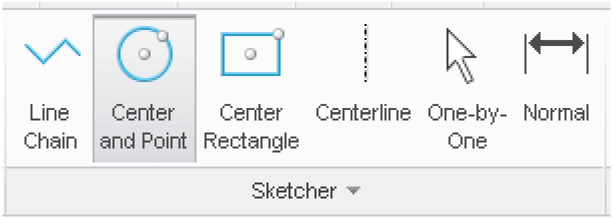
Double-click on any dimension (the blue numbers) to explicitly set that dimension to a specific value.

For this exercise, set the rectangle's width and height both to 100.

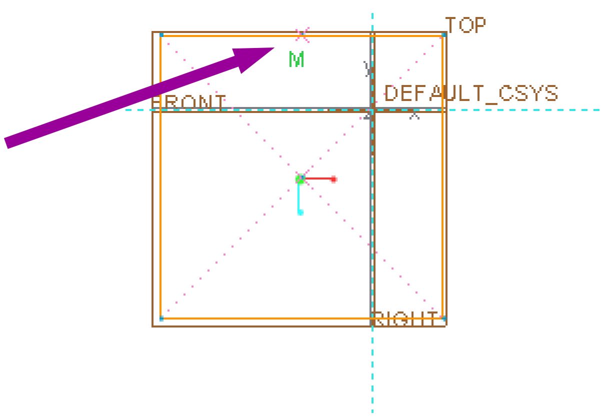


Adding to a sketch

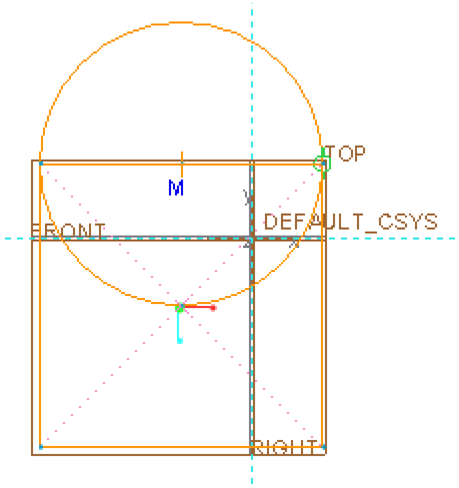
Now we will add a circle to the sketch.



Click the **Circle** tool in the ribbon bar, then hover the mouse along the top edge of the rectangle until it snaps to the middle (labeled “M”).

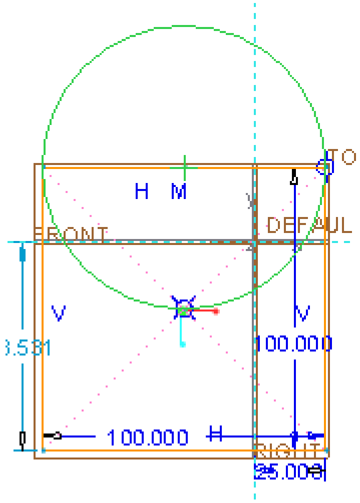


Left click to set the center of the circle at the rectangle edge's midpoint. Then drag the circle out to meet a rectangle corner such that a green circle appears and it snaps into place. Left-click again to place the circle.



Middle click to deselect the circle.

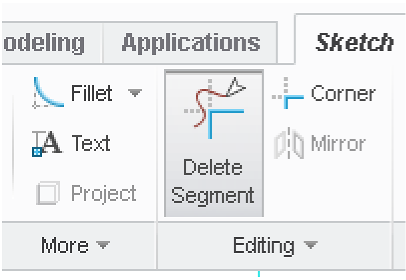
The shape is no longer closed – it has intersecting line segments – and therefore it is not solid orange.



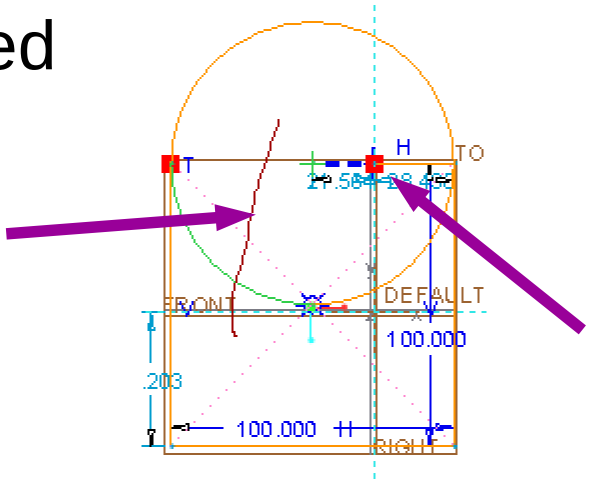
We need to remove the extra line segments.

Closing a shape

Click on the **Delete Segment** tool.

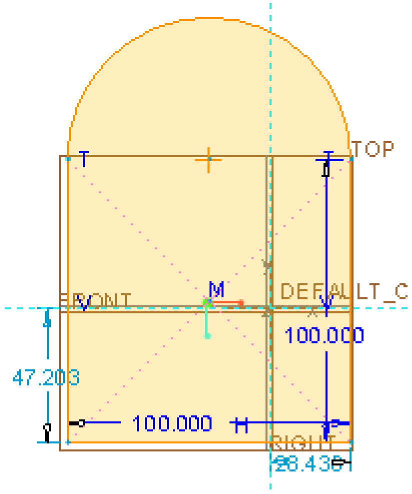


Left-click and drag the mouse over any interior lines of the sketch. Each line segment crossed will be deleted from the sketch.



You may need to delete multiple interior segments. Any segments that are unconnected or extra are marked with red boxes.

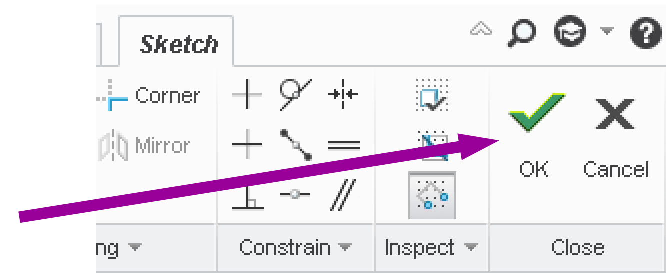
Continue until there are no more red boxes, then middle click to finish. The closed shape will fill orange.



Extruding a shape

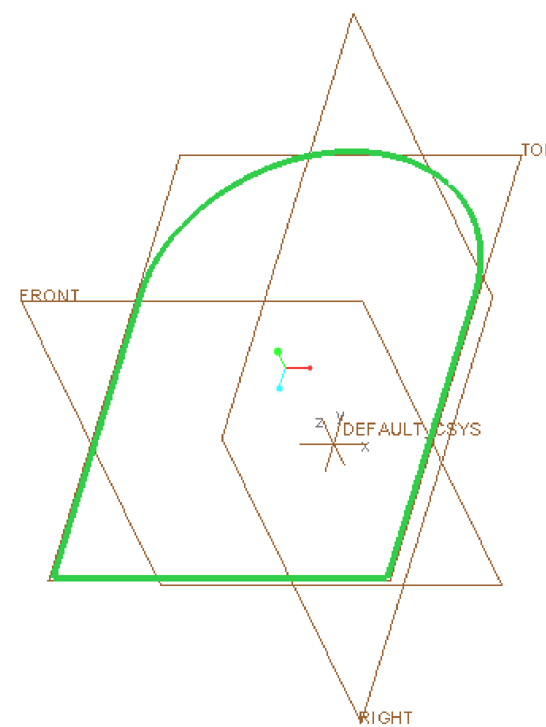
Now that our shape is finished, we can *extrude* it. Extruding a shape gives it thickness, making it three-dimensional.

Click on the **OK** box to finish editing the shape.

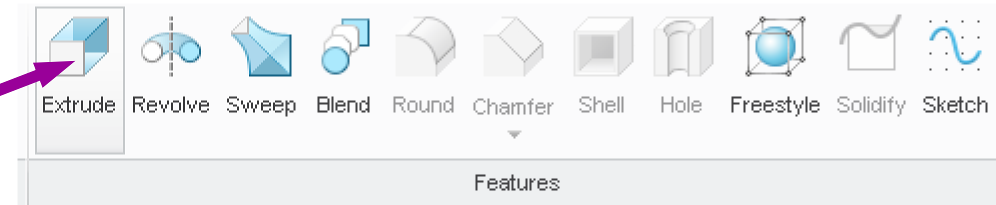


Press CTRL+D to see the shape in its datum plane.

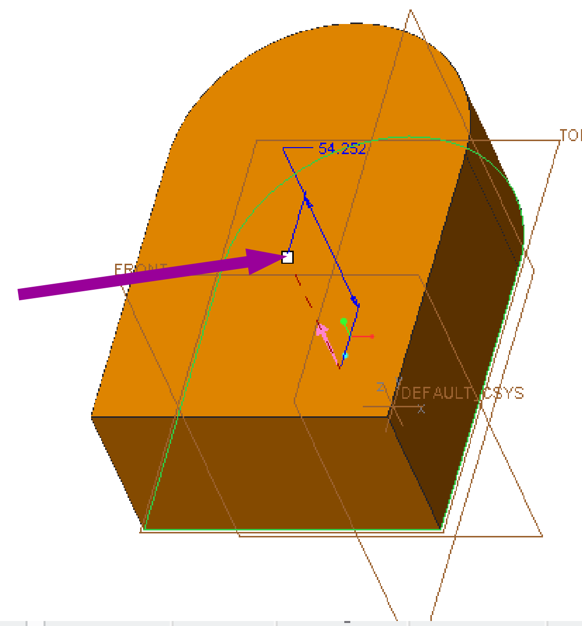
Left click on one of the shape's edges; the shape's edges will turn green to indicate it has been selected.



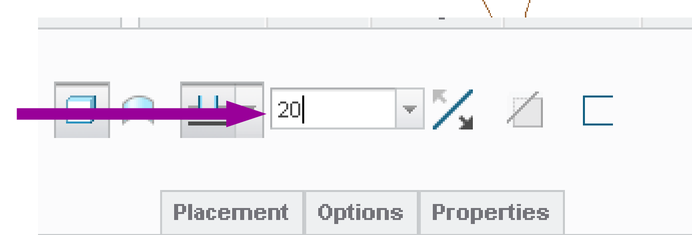
Select the **Extrude** tool.



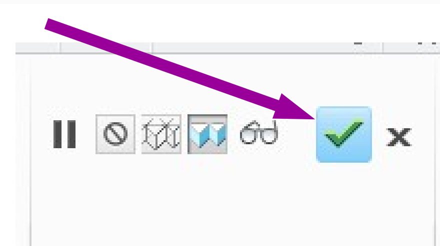
The object is now extruded into three dimensions. The depth of the extrusion is controlled by a box in the Extrude ribbon bar.



The depth can also be controlled by dragging the white box handle in the graphics area.

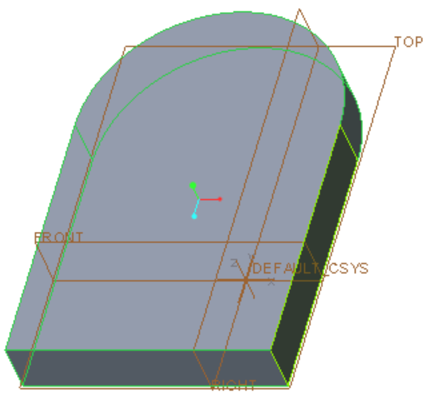


Set the extrusion depth to 20, then select **OK** in the extrude ribbon.



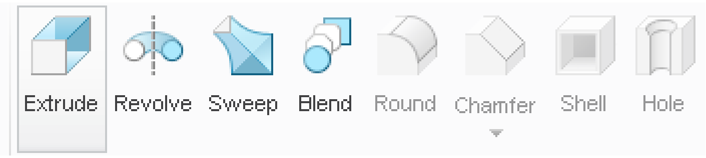
Extruding off of a face

We now have a solid part.

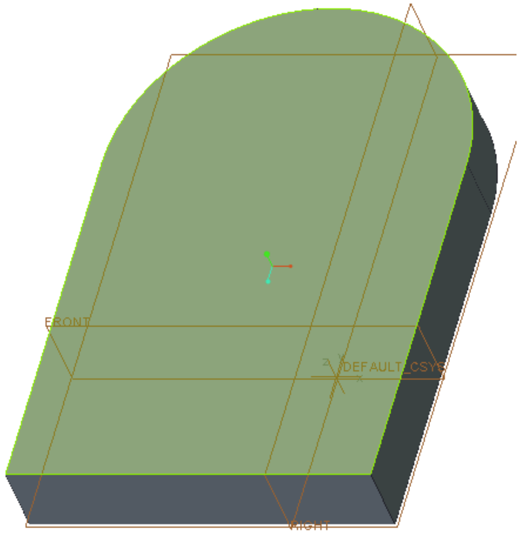


Now we are going to extrude a new feature onto our existing part.

Select the **Extrude** tool, then click the top face of the part.



Creo immediately enters the *Sketch* tool, and we are sketching on the plane defined by the top face of our part.

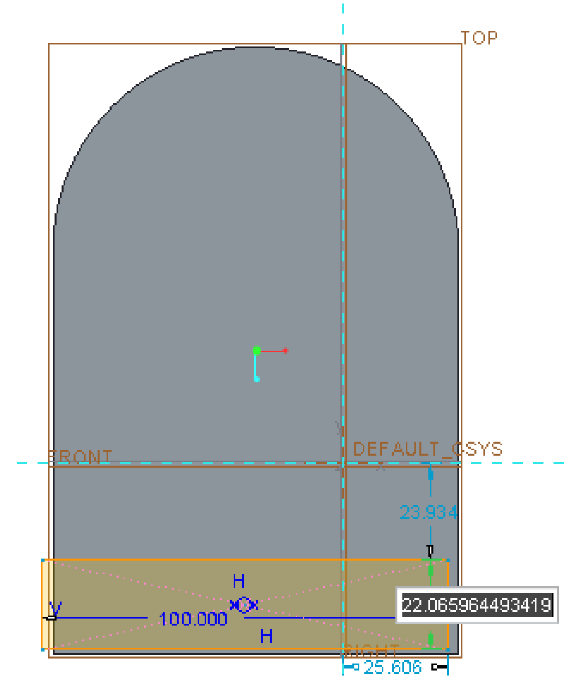


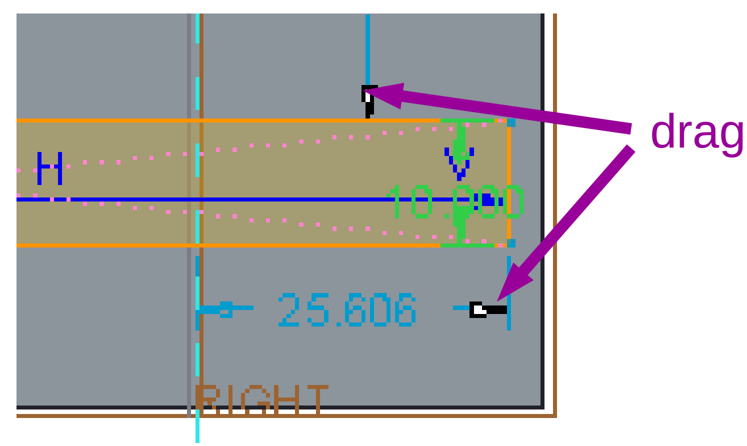
In Sketch mode, draw a 100 by 10 unit rectangle along the bottom edge of the part.

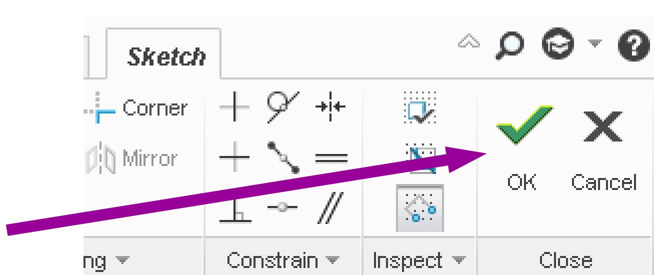
As before, double-click on dimensions to set them to exact values.

You can also drag on some dimension arrows to adjust the value.

When the 100x10 rectangle is correctly placed, click the green checkbox.

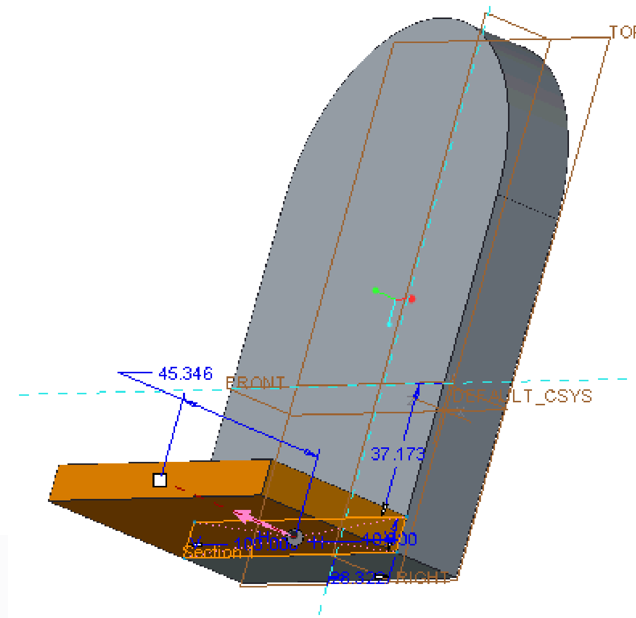


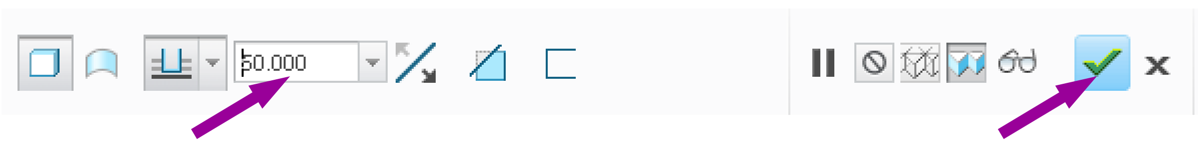




We have now added an extrusion to the top face of our part.

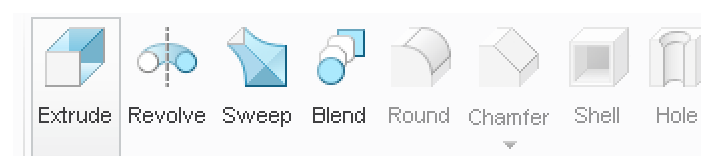
Set the depth of this extrusion to 50 and click the green checkmark to complete the extrusion.

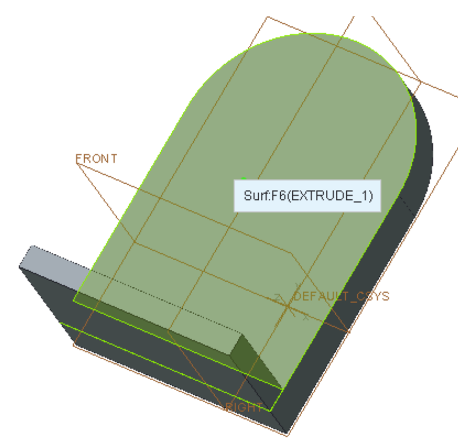




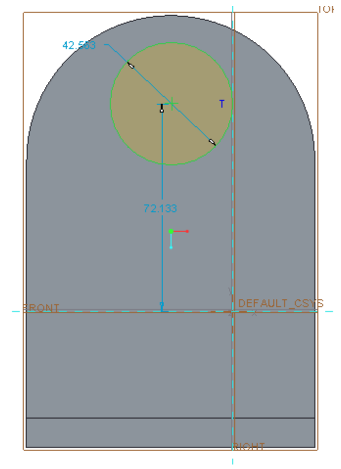
Removing material with extrusions

Now we will use an extrusion to cut a hole in our part. Again click the **Extrude** tool, and the top face of our part.



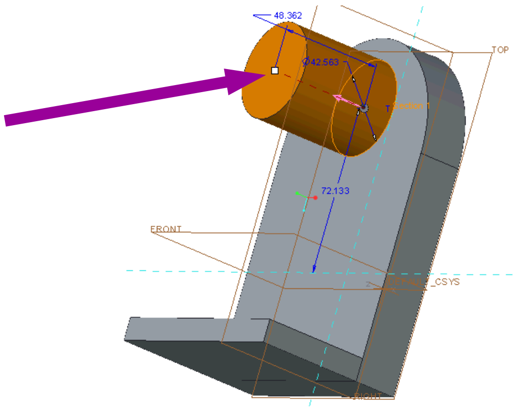


Using the **Circle** tool, sketch a circle near the rounded end of our part.

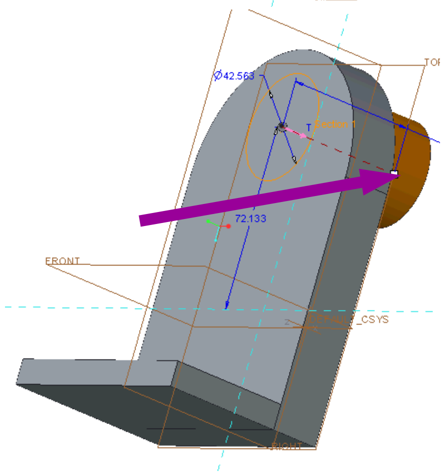


Click **OK** to complete the sketch.

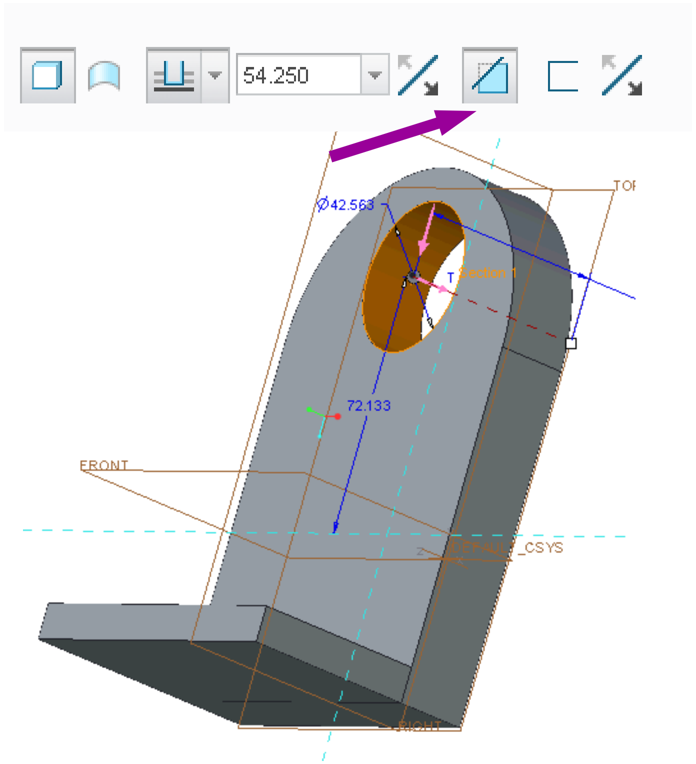
Once again, we have an extrusion on the top face of our part.



Drag on the white square to “push” the extrusion into the part base.



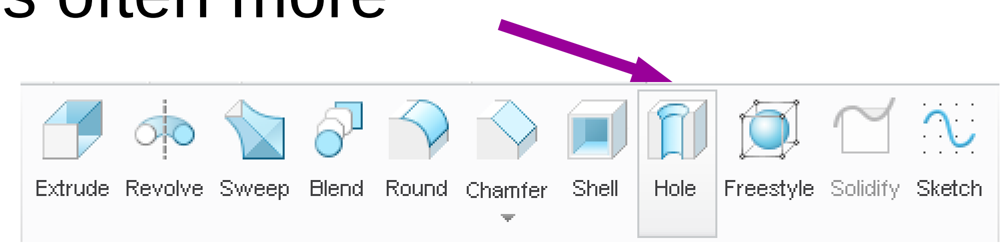
Click on the “Remove Material” toggle in the Extrude ribbon bar. This tells Creo to remove material from the part instead of adding to it.



Click the green checkmark, and the part now has a hole in it.

The Hole Tool

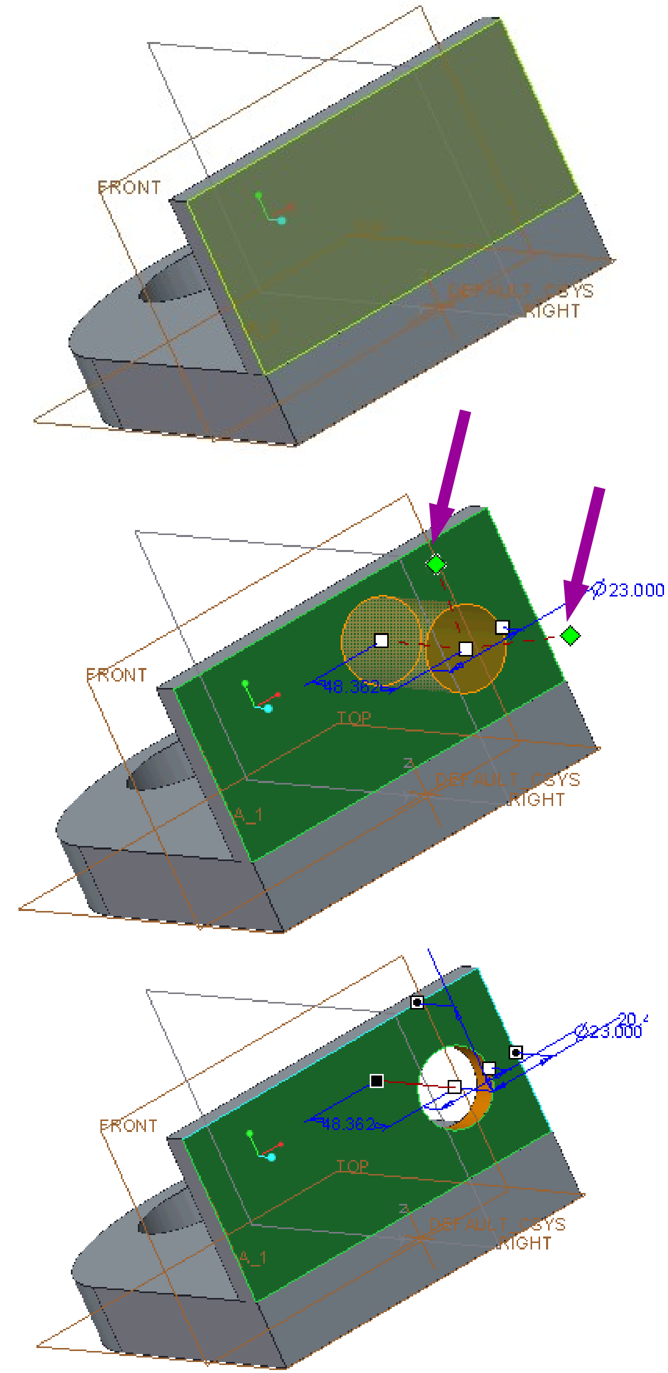
Another way to make holes is by using the **Hole Tool**. It's often more convenient.



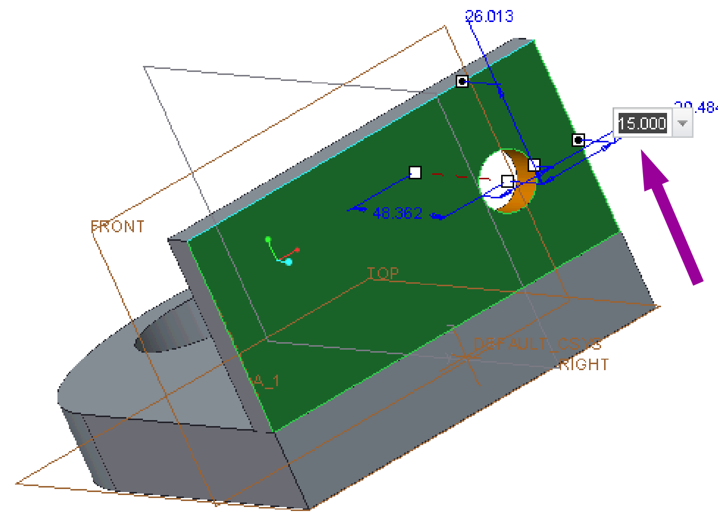
As with **Extrude**, the **Hole Tool** starts by selecting a surface on which to create the hole. Click on the face of the rectangle we extruded earlier.

Once selected, a cylinder appears where the hole is to be placed. The cylinder also has two green diamonds used to set reference points for the center of the hole.

Drag the green diamonds to edges of the part. When both are set, the orange cylinder becomes a hole.



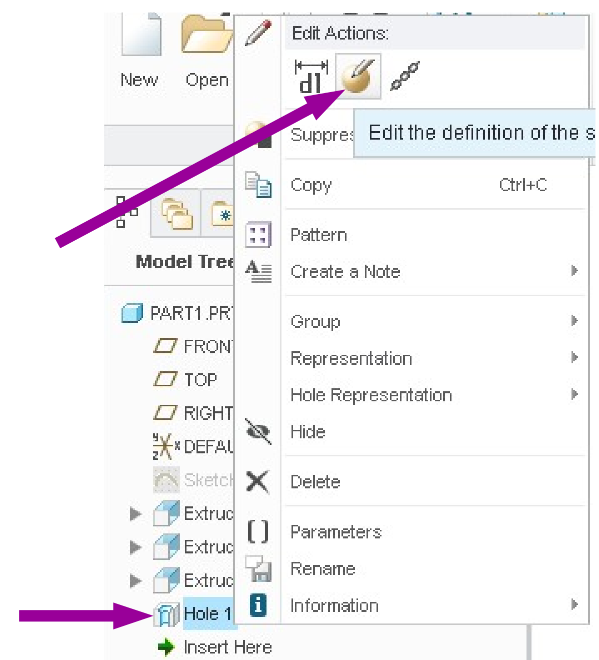
As before, we can double-click on blue dimensions to set them to exact values, or drag white squares to set dimensions.



Set the hole's diameter to 15, then click the green checkbox to finish the hole.

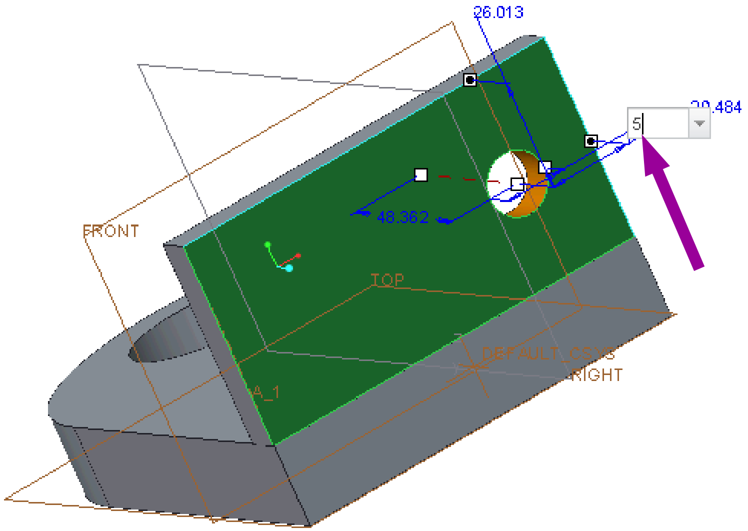
Editing existing features

On the left side of the window is the **Model Tree**. It shows all of the features in the current part.

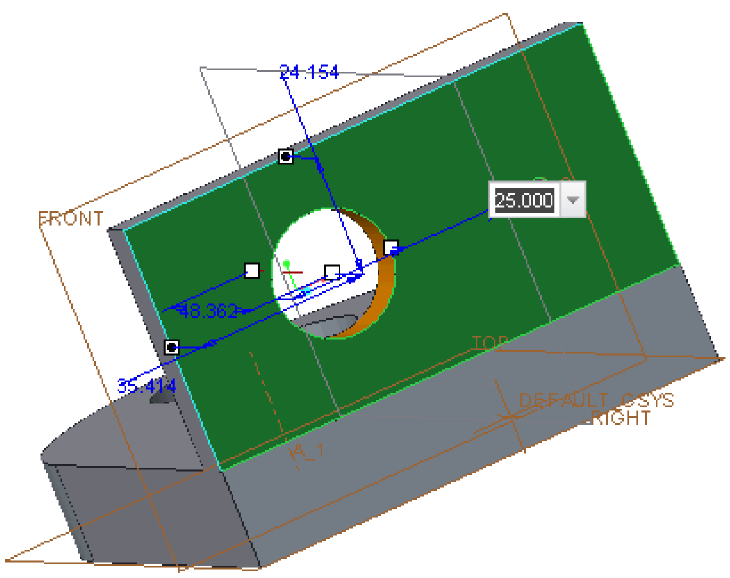


To edit an existing feature, right-click the feature and select **Edit Definition**.

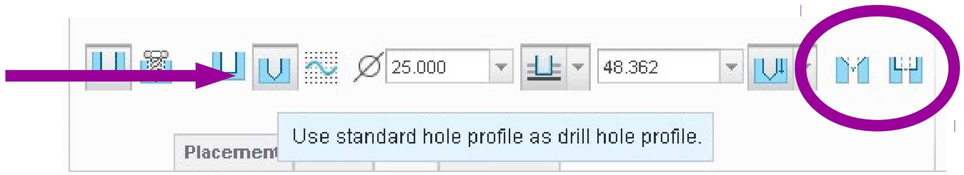
Right click on **Hole 1** in the **Model Tree**, then left click on **Edit Definition**. The hole dimensions appear again. Change the hole's diameter to 5, then click the green checkbox to finish.



Now start another hole on the same surface with diameter 25. Exact position isn't important.

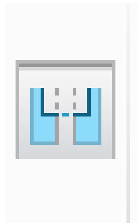


Select **Use standard hole profile** in the **Hole** ribbon bar.



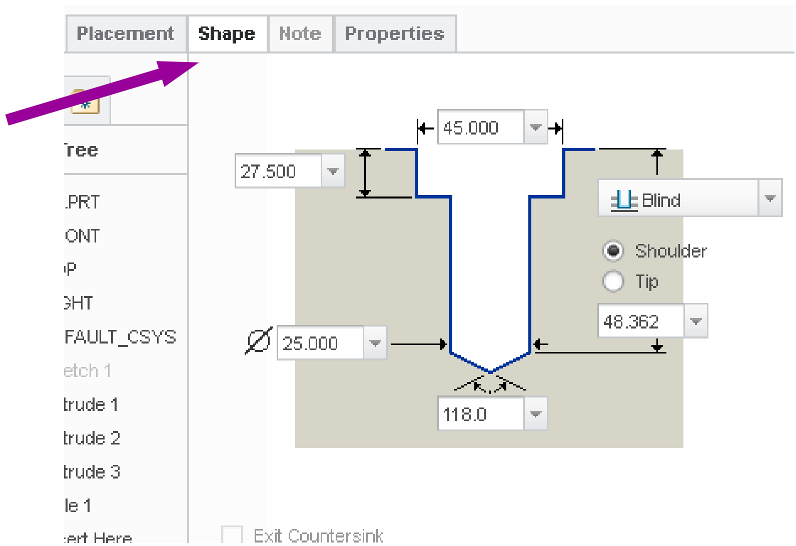
This brings up the **Countersink** and **Counterbore** options. These options cut away material around the entrace of the hole so screws or bolts can recess below the surface.

Select the **Counterbore** option in the **Hole** ribbon. The existing hole will appear larger.

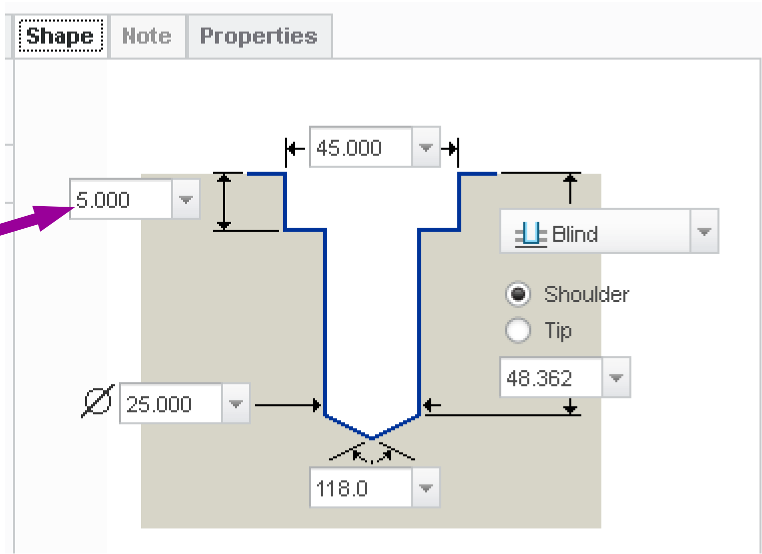


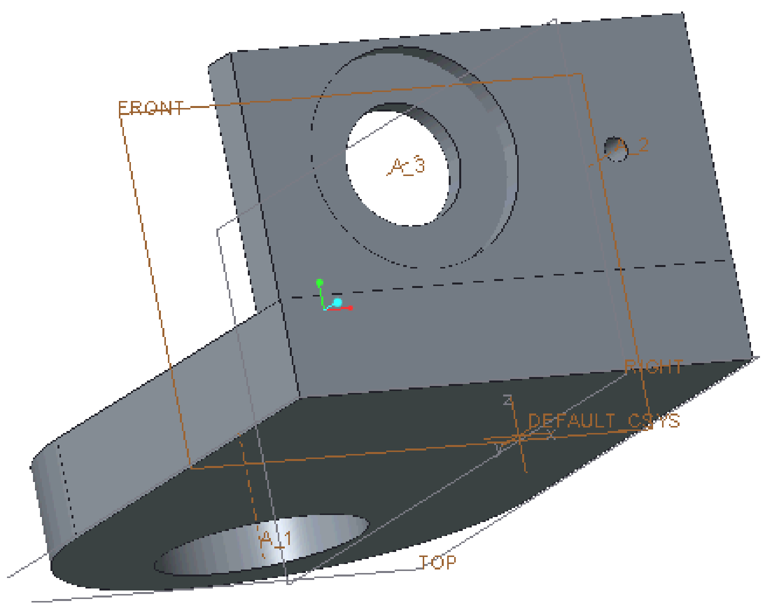
To see the hole profile, select the **Shape** tab in the **Hole** ribbon.

Here the dimensions of the counterbore can be adjusted.



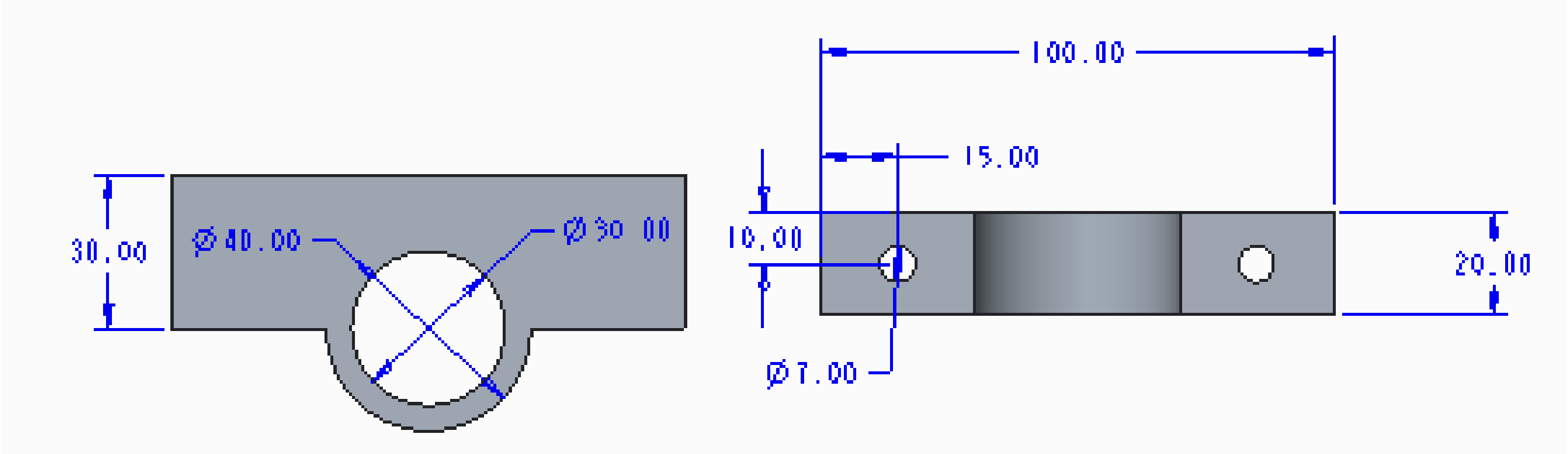
Set the counterbore depth to 5, then click the green check mark to accept the hole. This results in a counterbore hole.



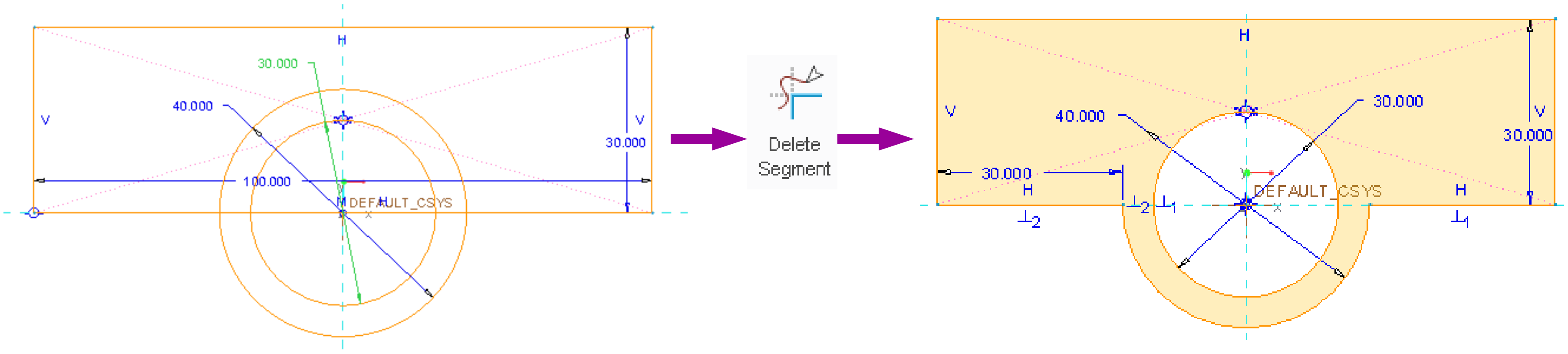


Try it yourself #1

Using what you know, try creating the part below.

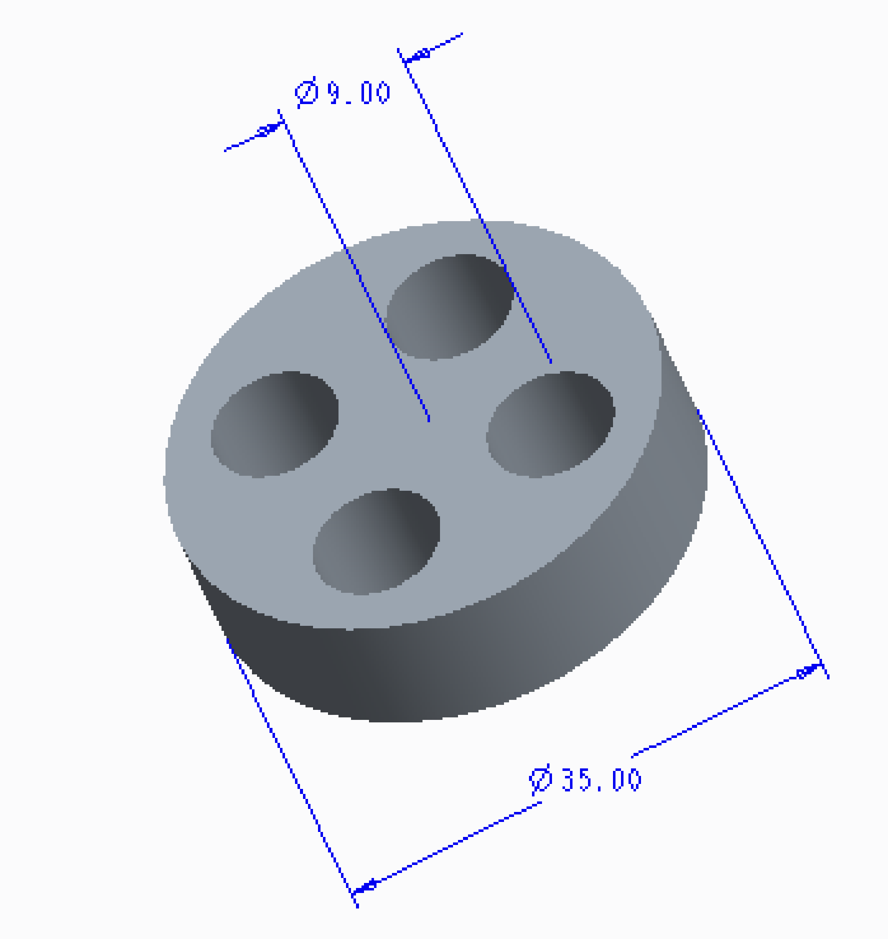


Hint: Use the **Shape** tool to create the shape on the left, **Extrude** that shape to a depth of 20, then add **Holes** in the sides.



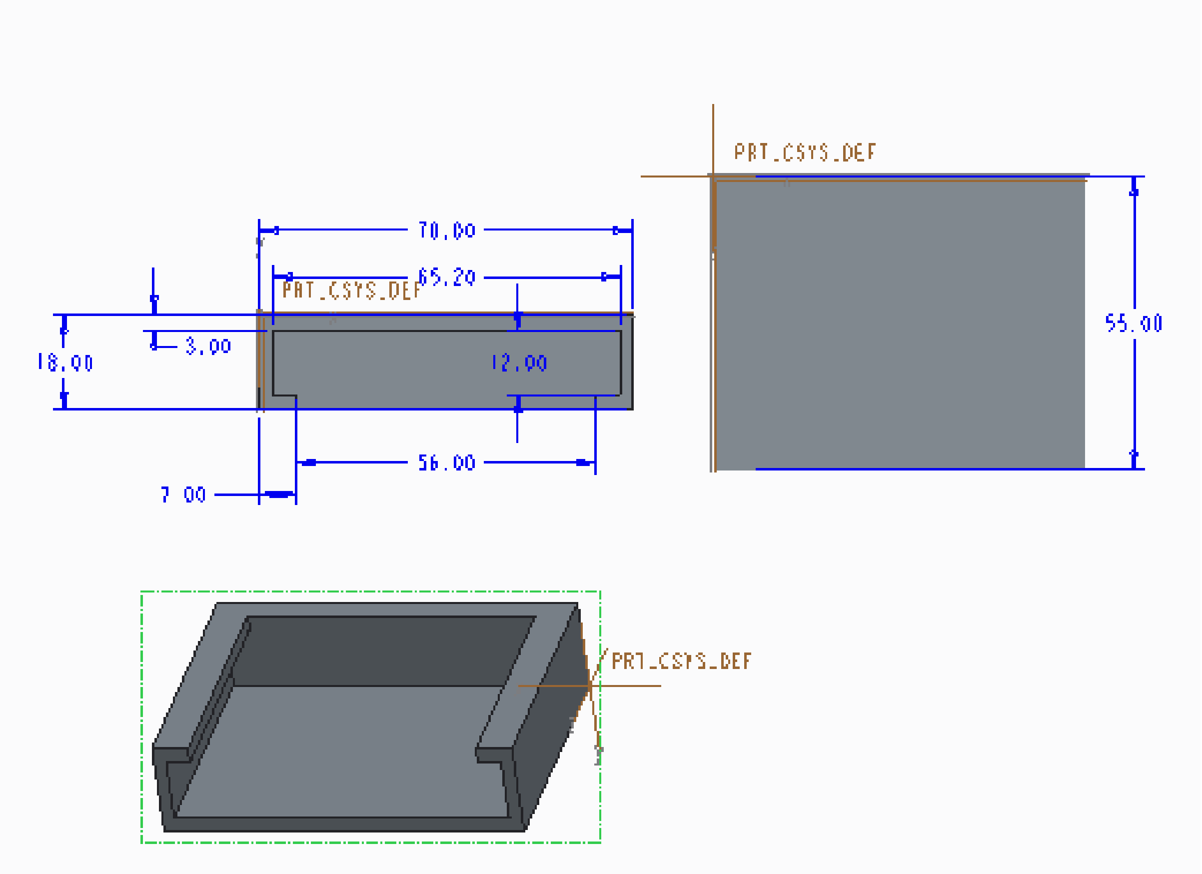
Try it yourself #2

Using what you know, try creating the part below.



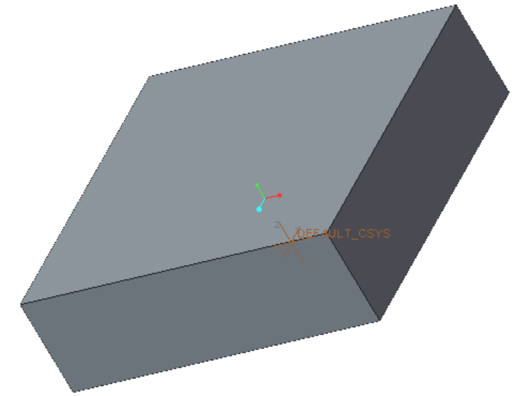
Try it yourself #3

Using what you know, try creating the part below.

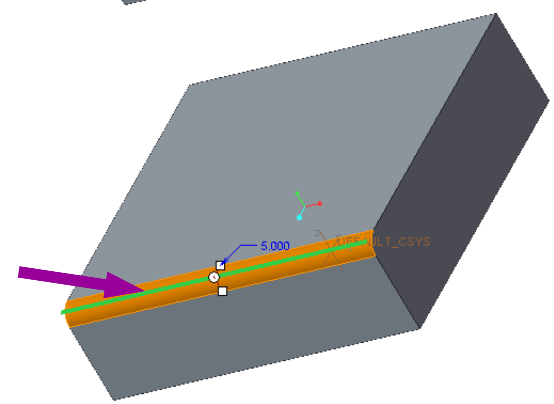


Round and Chamfer

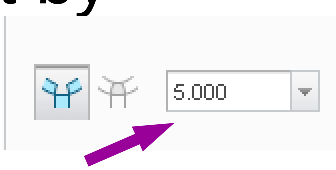
The **Round** and **Chamfer** tools work on edges of a solid, either creating a rounded or beveled edge.



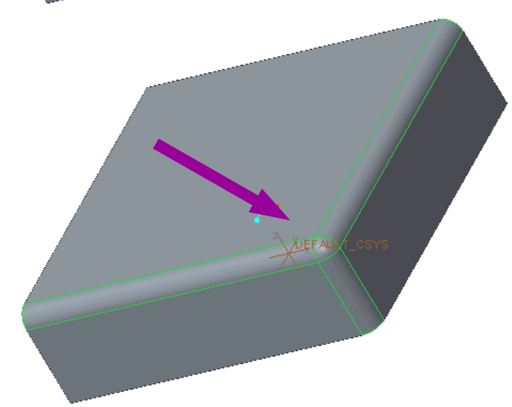
Start by creating a rectangular block. This one is 100x100x30.



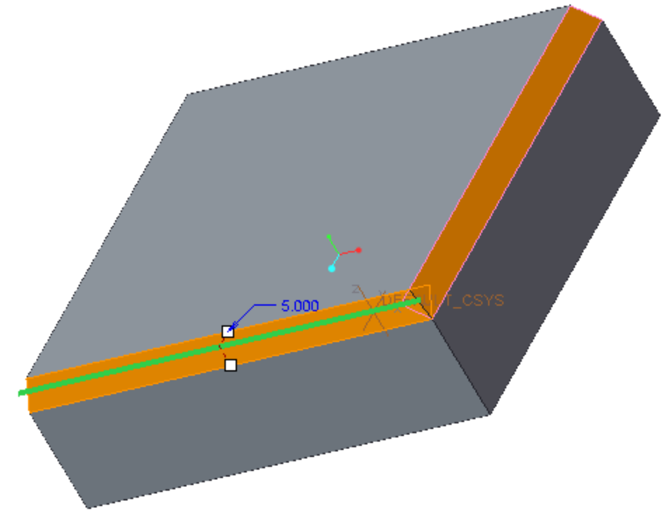
Select the **Round** tool, then click on any edges to be rounded. The amount of rounding is set by the radius box.



If multiple rounded edges meet at a corner, it's rounded also.

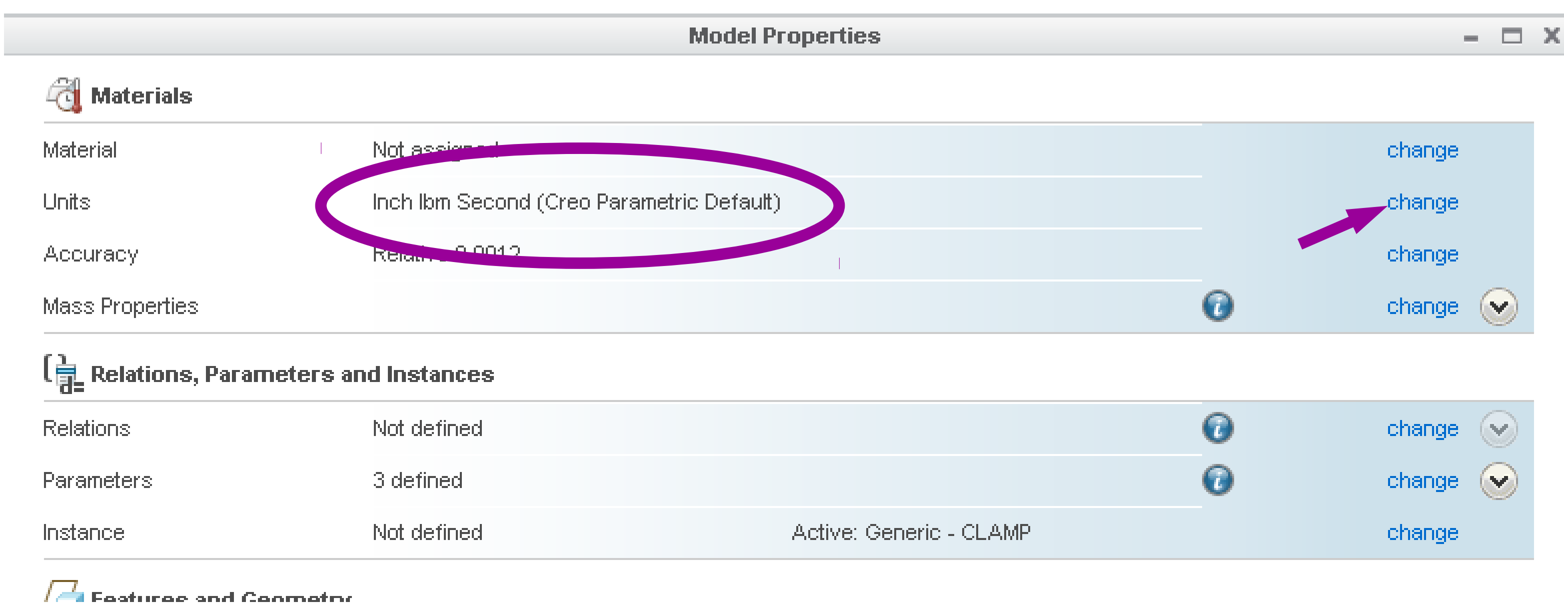


Chamfer does the same as rounding but produces beveled edges.



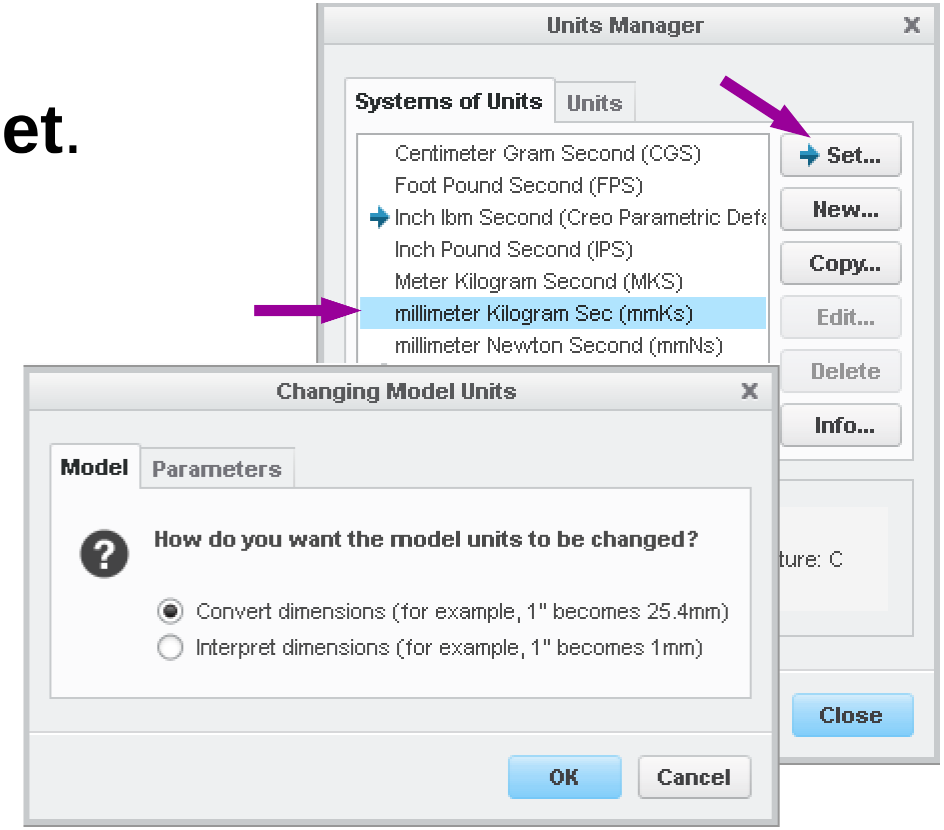
Setting model units

So far we've not assigned a unit to dimensions. To set a model's dimensions, use **File**→**Prepare**→**Model Properties**:



To change the model's units, select the new units and click **Set**.

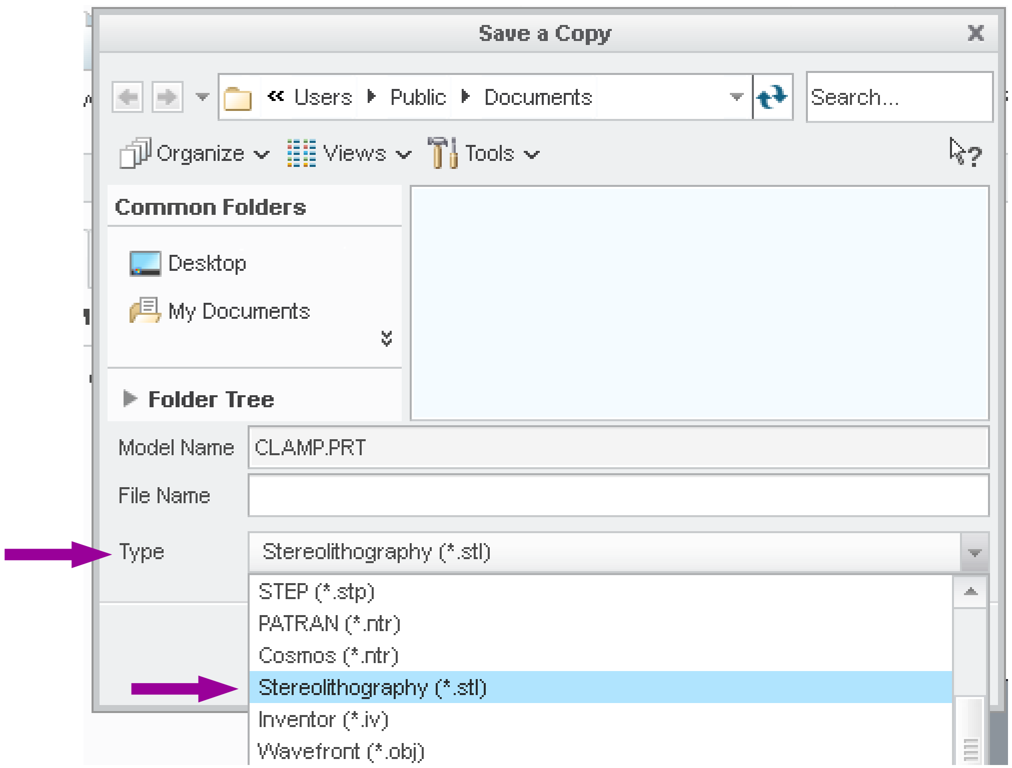
You're then given the option of how to convert or interpret existing dimensions in the model.



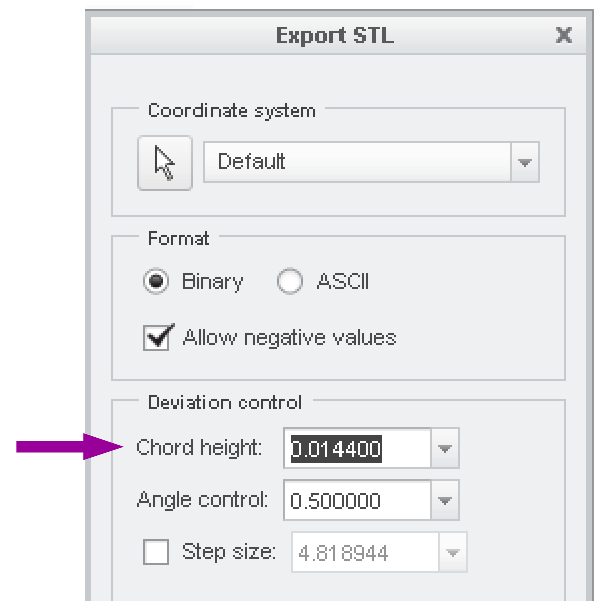
Sending a part to a 3D printer

Most 3D printing software uses Stereolithography (.stl) files.

In PTC Creo Parametric, first save the part to disk, then use **File** → **Save As** → **Save a Copy** and select **Stereolithography (*.stl)** from the **Type** dropdown menu.



In the **Export STL** dialog, enter a small value for Chord Height.



Further Resources

How to Model Almost Anything guide

- https://www.ptcusercommunity.com/docs/DOC-4618
- Section 3, Exercise 8: Sketching
- Section 3, Exercise 9: Personal Badge
- Section 3, Exercise 7: Part Model Planning