



3DXpert

CAD Exercise - 3

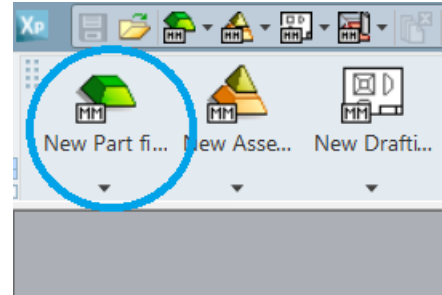
Creating a Solid model

Tutorial_V1 - Updated: 13,0200,1474,1051(SP2)

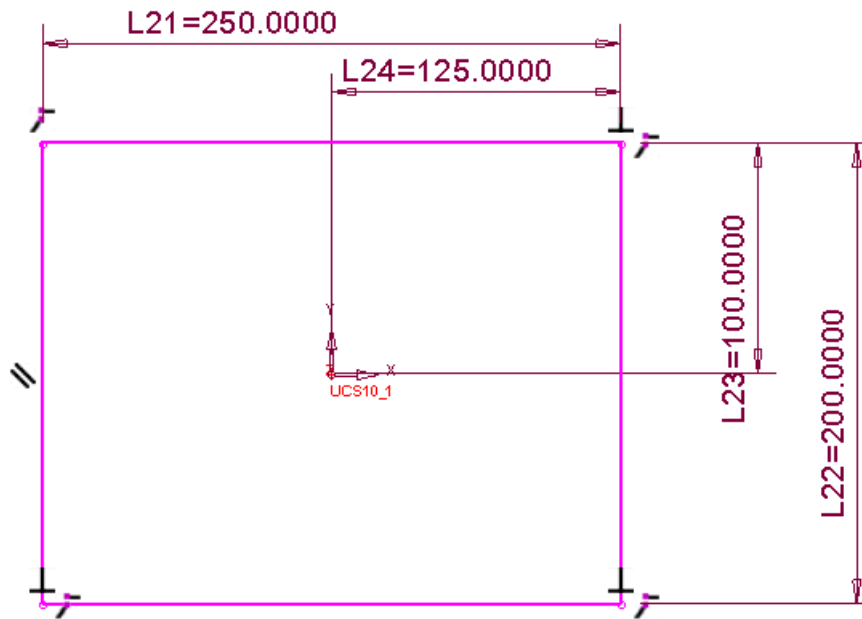
In this exercise, we will discuss 3DXpert CAD environment, specifically - the Solid modeling.

We will create a simple model.

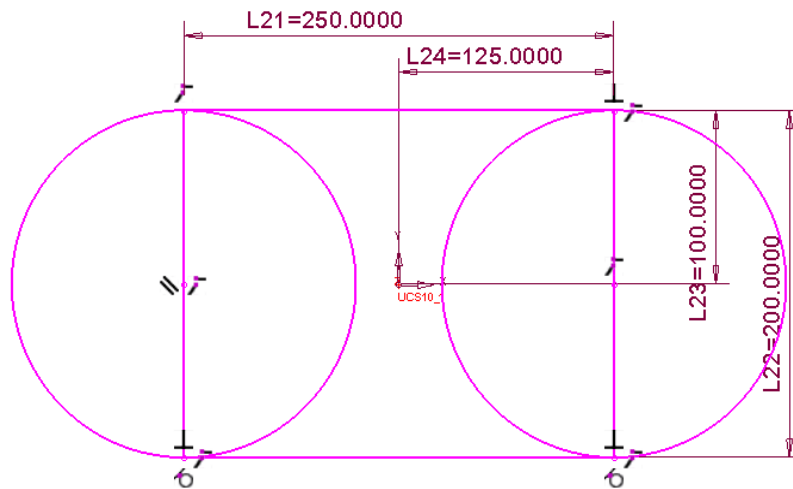
1. Launch 3DXpert and create a new Part file.
2. Enter the Sketcher and press the middle mouse button to approve the XY plane.



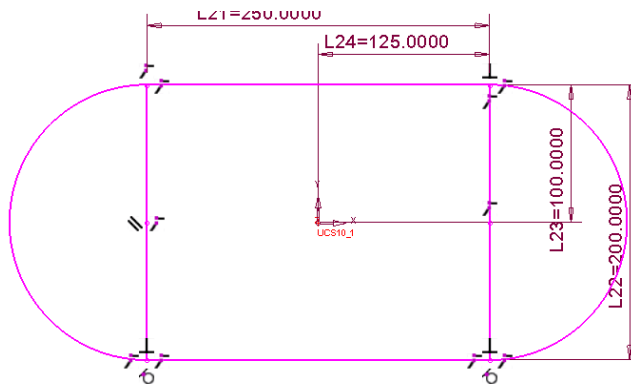
3. Create a rectangle



Add the two circle starting at the center of each vertical line, and reference them to the horizontal line.



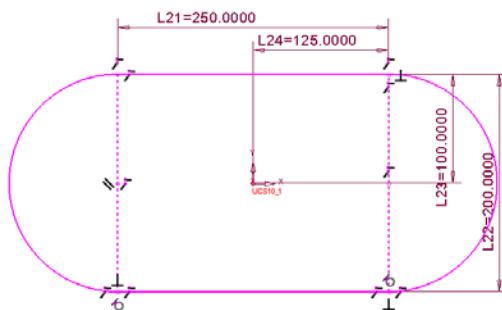
Use the Sketcher's trimming tool to remove the inner arcs



Pick the inner vertical lines and click the Turn to Reference button from the Sketcher's



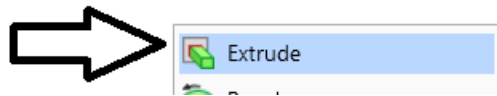
guide. Notice that the lines become dashed, these will not be used as actual construction lines.



Right mouse click on screen and click the OK option to exit the sketcher.

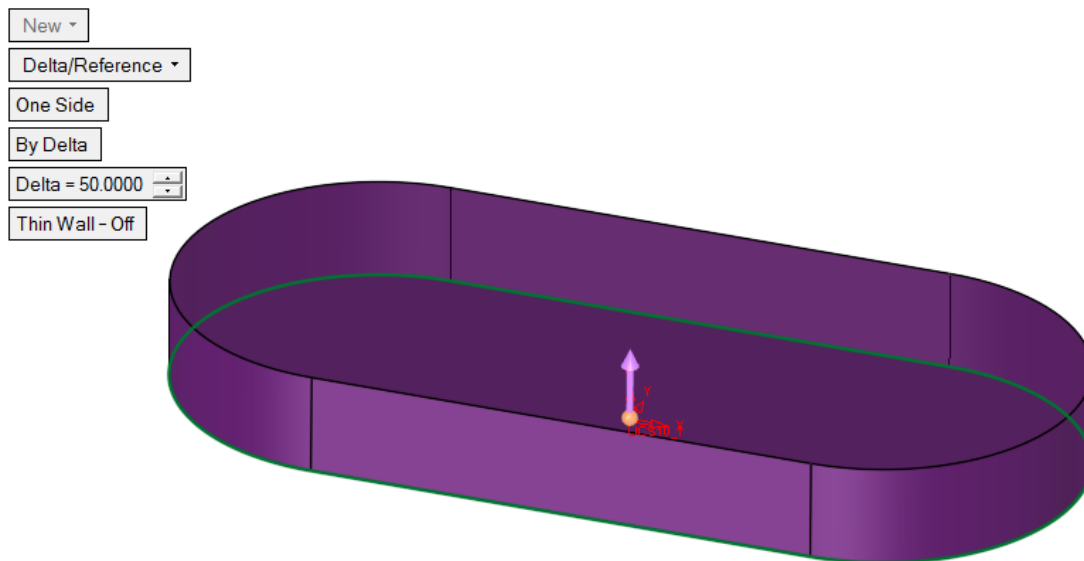


4. As we just created the contour by the sketcher, it is automatically picked (highlighted) by the system for the next operation.
If you clicked something else and the sketch is no longer highlighted, pick it from screen.
Right mouse click the display and select Extrude

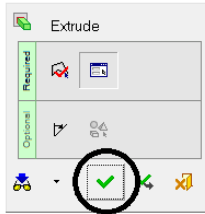


As this is the first object we are creating in this file, it is set as New.

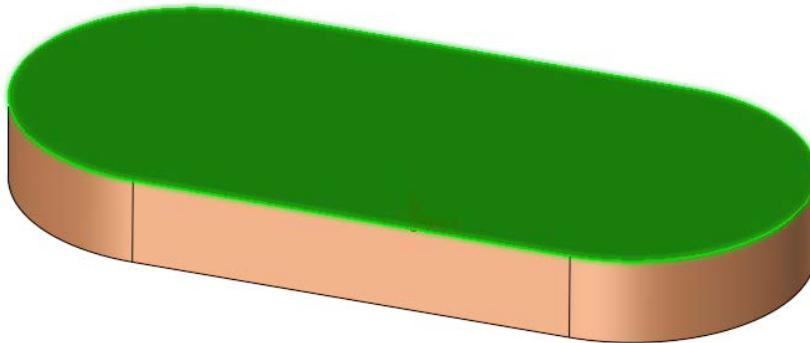
Set a 50 mm delta (height)



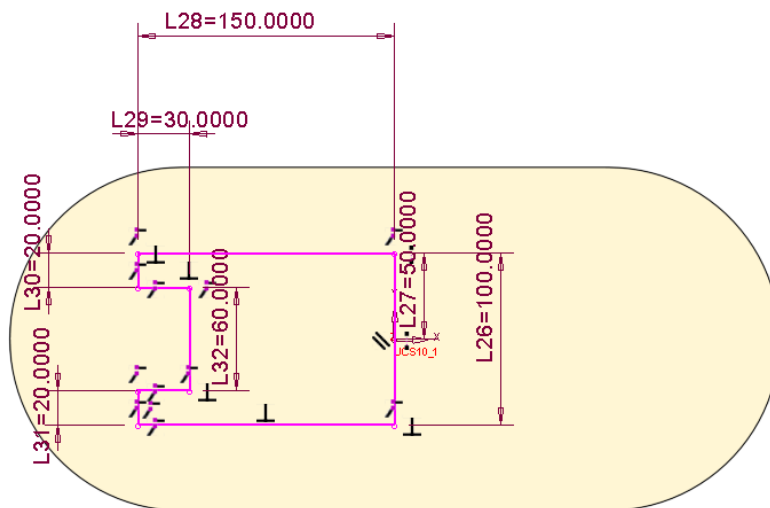
Right mouse click and press OK to approve creation.



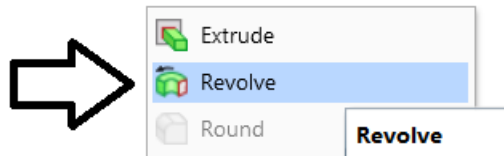
5. Pick the upper face of the object. Right mouse click and press Sketcher.



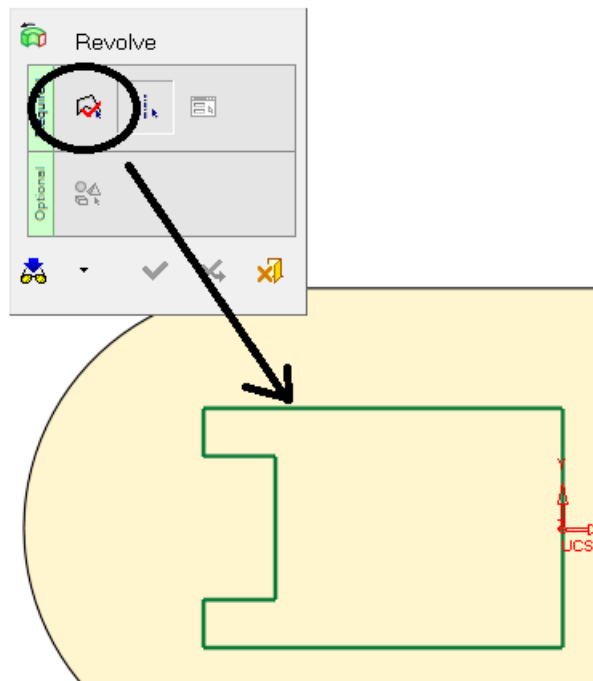
Switch to top view and create the following sketch:



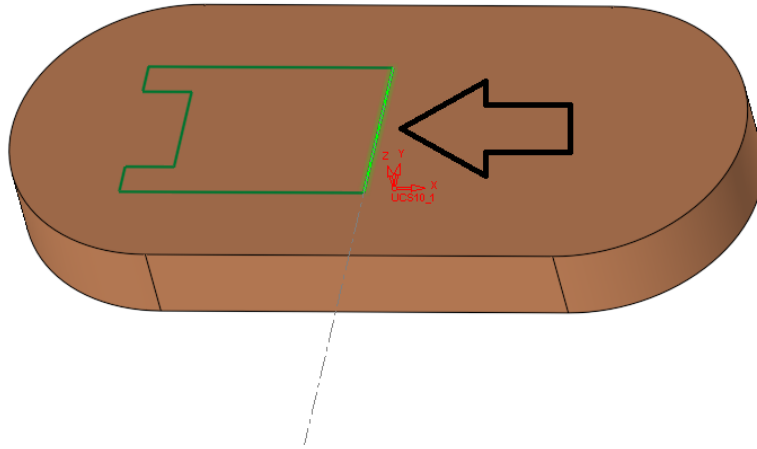
6. Right mouse click and press OK to exit the sketcher. Right mouse click on the display and select Revolve



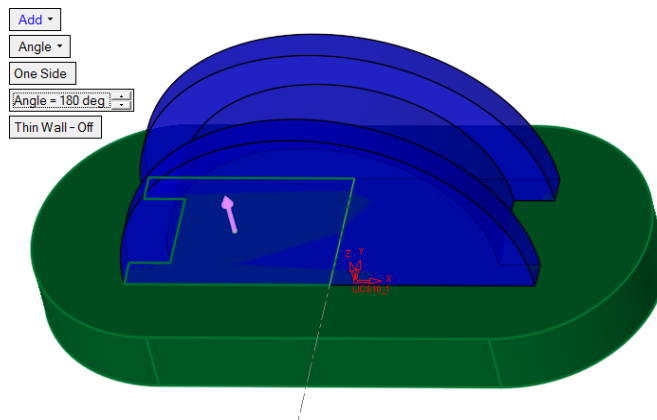
Notice that the first stage – selecting the contour is already ‘approved’. We are now in the second stage, picking the revolve axis



As the rotation axis, pick the sketch's line as shown here

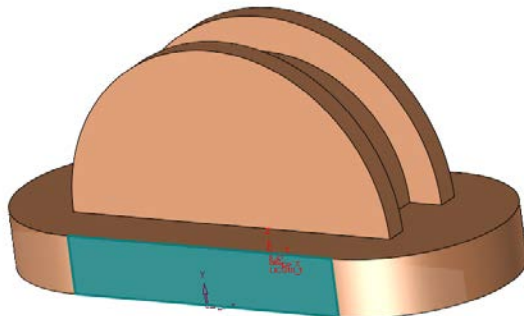



Notice that mode is Add (to existing object). Set angle as 180 degrees.

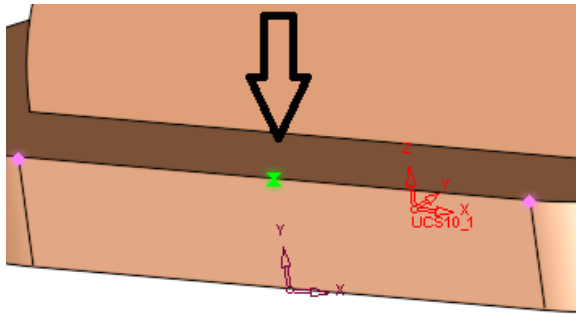


Right mouse click and press OK to approve creation.

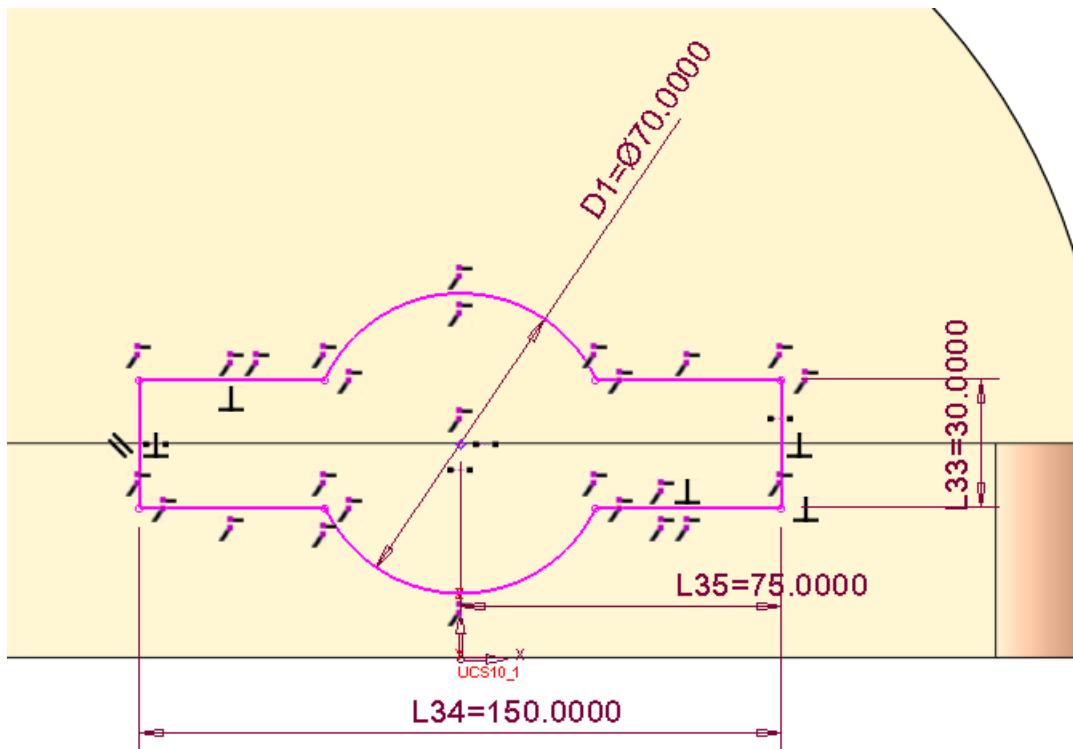
7. Create a sketch on the face shown here:



From the Sketcher's guide click the Add reference  button.
Add the center of the upper line as a reference point:

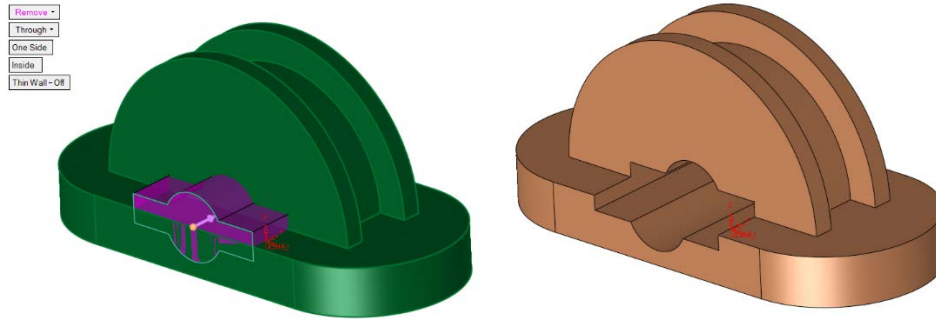


8. Create the following sketch around this point:

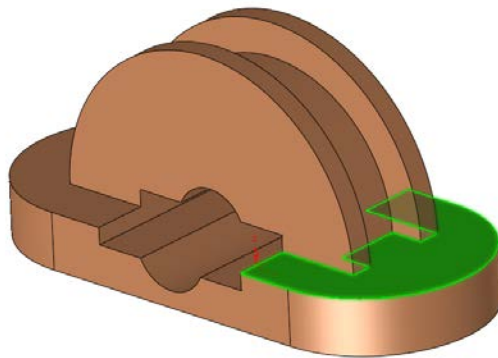


Right mouse click and press OK to exit the sketcher.

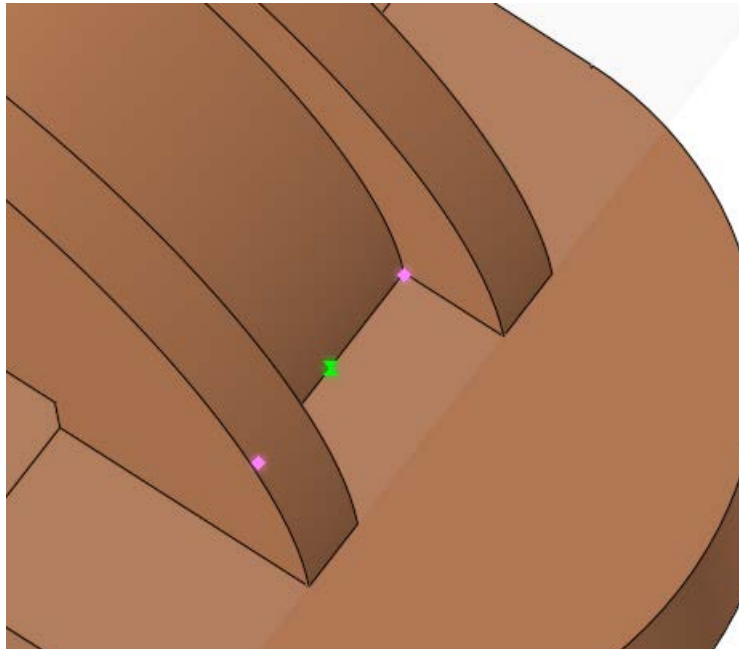
9. Right mouse click and press Extrude, switch from Add to Remove.
Switch from Delta\Reference to Through. Right mouse click and press Ok to approve.



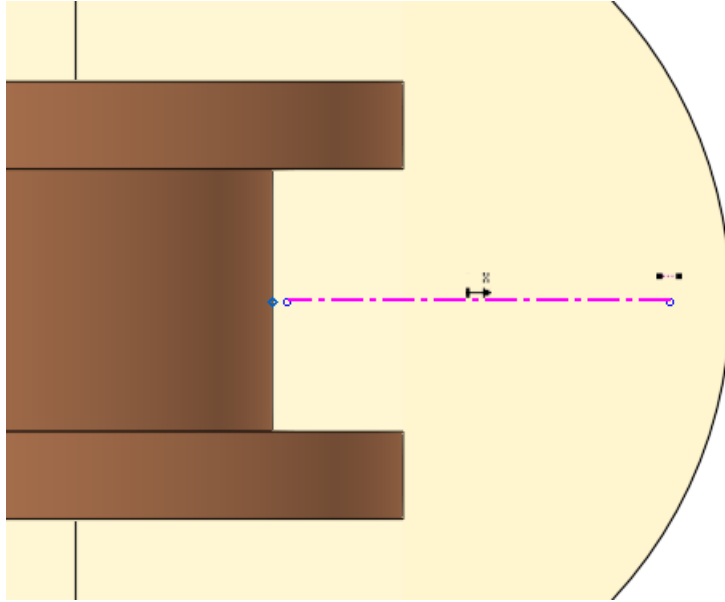
10. Select the rightmost horizontal face:



Press Add Reference and pick the point in the center of the line as shown:



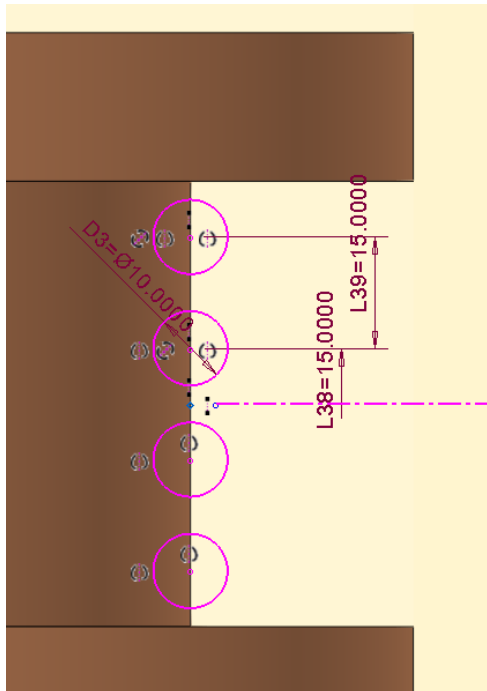
Press the symmetry button and create the symmetry line



Keep the symmetry button pressed and press the Circle button.

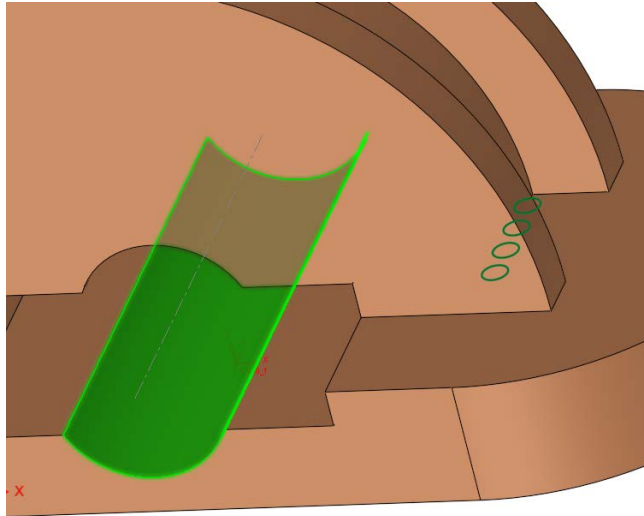
Toggle Free to Dimension and set a Diameter of 10.0mm

Add two circles on one side of the symmetry line, and set the dimensions:

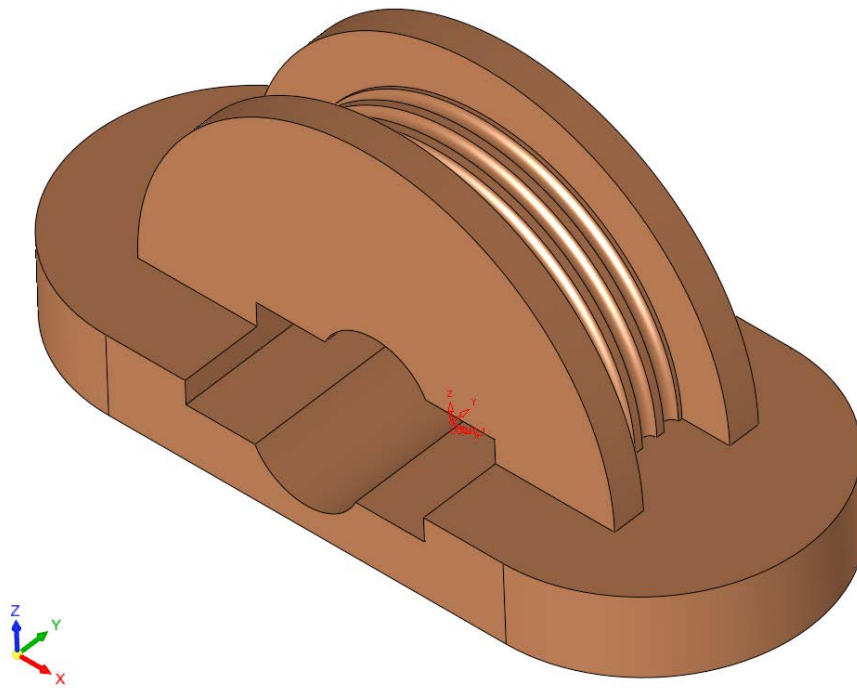


Right mouse click and press OK to exit the Sketcher.

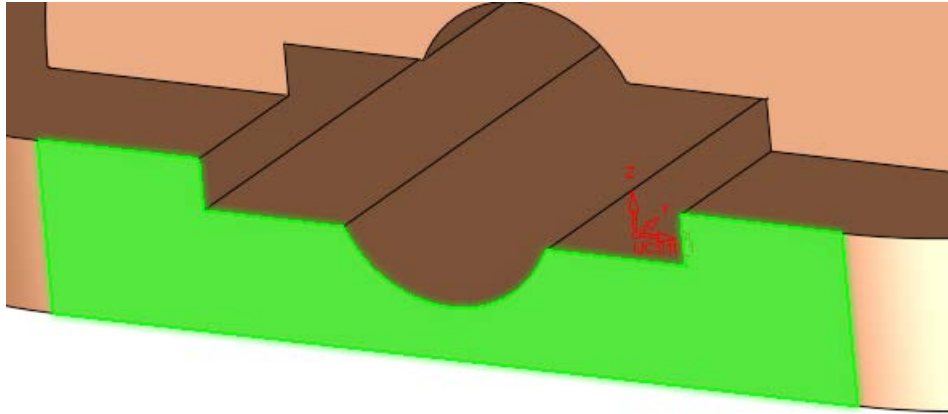
11. Enter Revolve and as the rotation axis pick the cylindrical face at the middle of the object




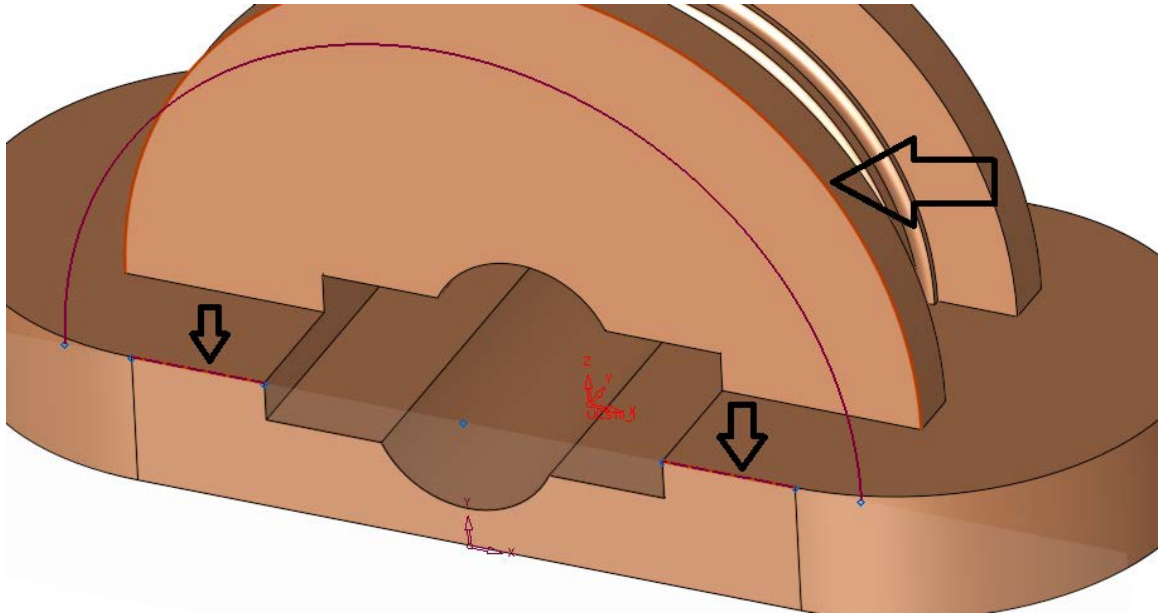
Switch from Add to Remove (keep it as 180 degrees) and press OK.



12. Create a sketch on the front face



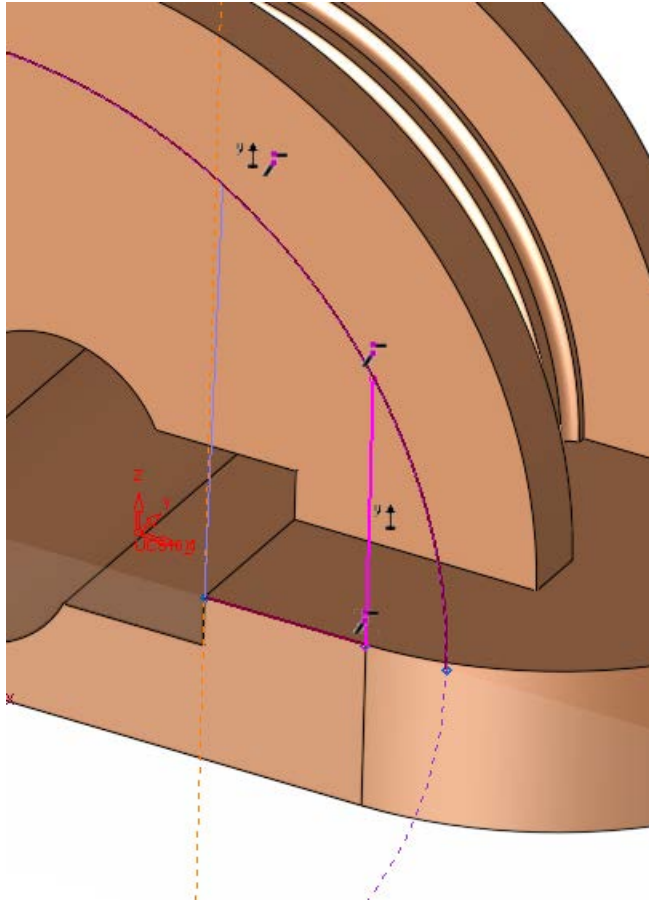
Click Add Geometry  and pick the 3 edges shown here:



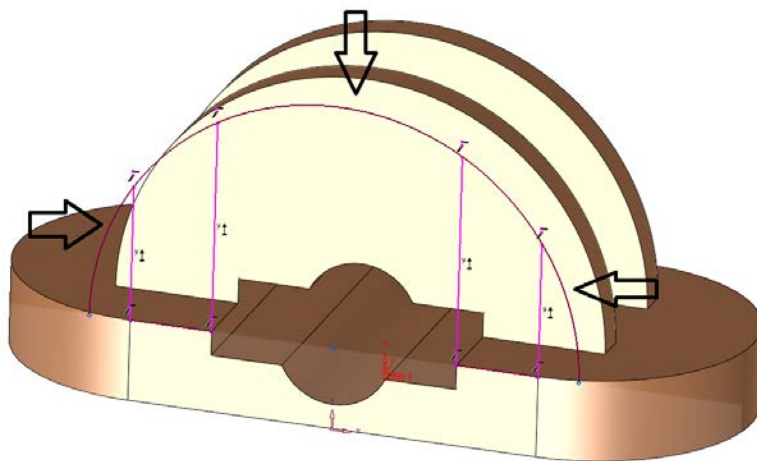
The edges are projected on the sketch plane.

Add the 4 vertical lines. Start from each lower point and left mouse click as the line is constrained with the arc. Press middle mouse button to approve each line.

If the 4 line are vertical and are constrained to the arc, they should all be colored purple.

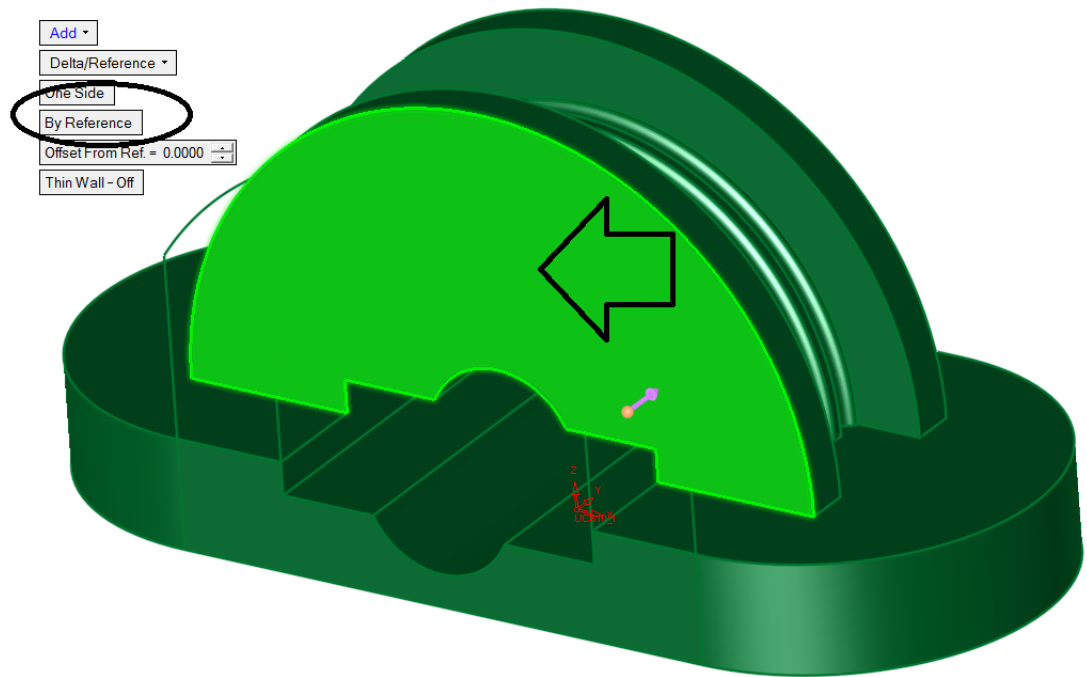


Still in the Sketcher, trim the arcs shown here:



Exit the Sketcher.

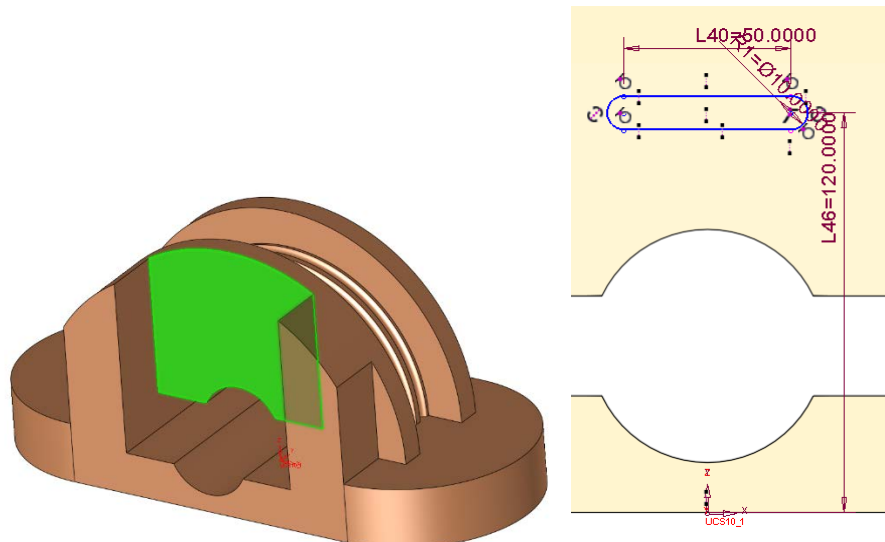
13. Enter Extrude, switch to Add and make sure that the direction points towards the object. If not, click the arrow to flip the direction. Toggle By Delta to By Reference and pick the flat face:



z

Right mouse click and Ok.

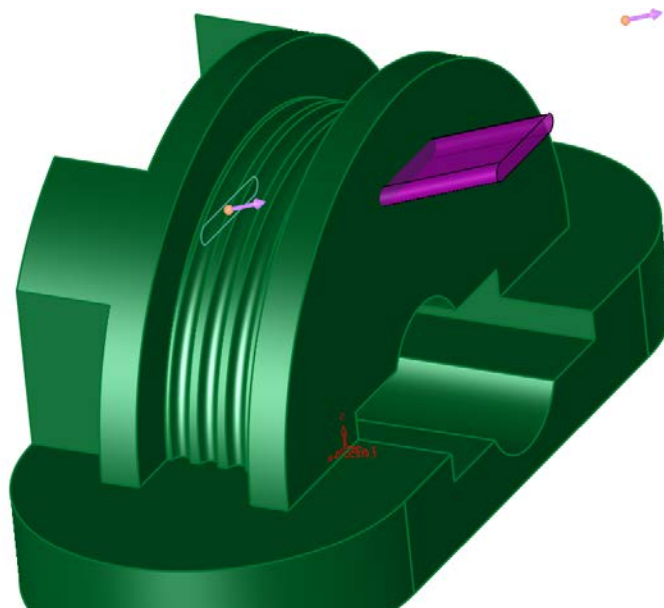
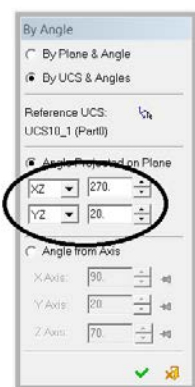
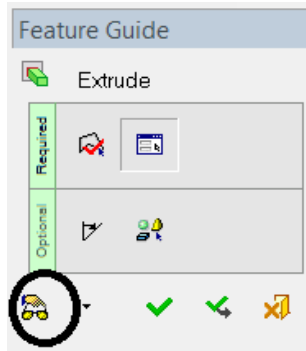
14. Pick the following face as sketch plane;



15. Extrude Remove this sketch using the Through option, however let's add an angle. Click the base of the direction arrow and select By Angle

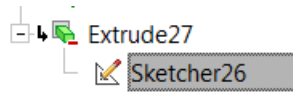


Enter the following parameters. If needed press the Preview Button on the Feature Guide

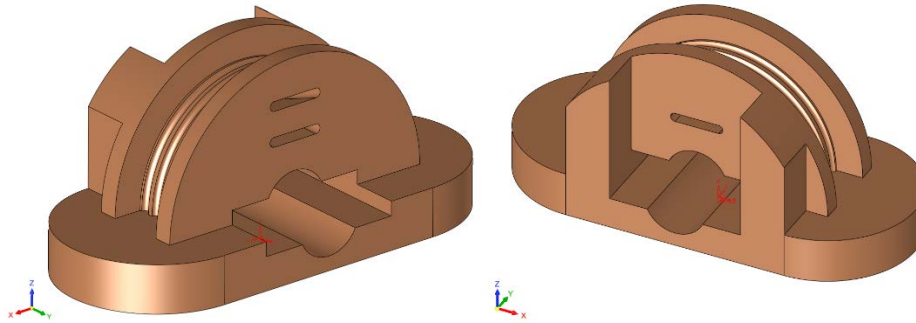


Press OK to approve and angle and then Ok to approve creation.

16. From the feature tree, click the plus sign to the left of the last Extrude feature



Right mouse click the Sketch feature below it and select Show Sketch\Contour
Click the Sketch and Extrude it again, this time keeping the default angle.



End of exercise.