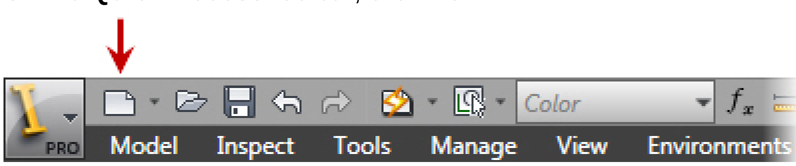
1 Project 2 - Flange Manifold Part

1.1 Instructions

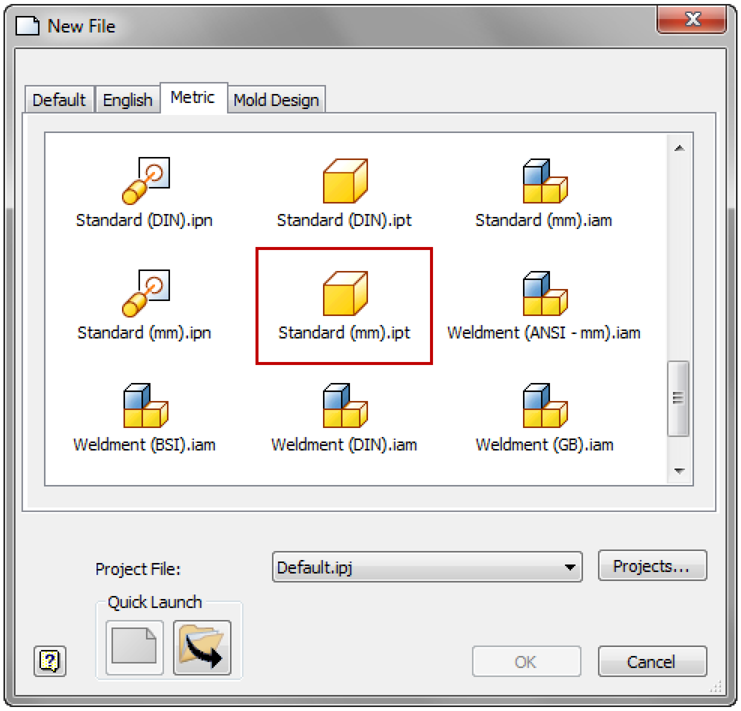
This project focuses on additional sketching methods and sketching commands. Revolve and Work features are also introduced. The part being modeled is from machine tooling used in the manufacture of an automobile engine. This flange detail would have several probes attached to it that measure the fluid flow through the flange. The challenge is that the probes need to allow for maximum clearance away from the engine.

1: Create a new part using the *Standard (mm).ipt* template.

- On the **Quick Access** toolbar, click **New**.



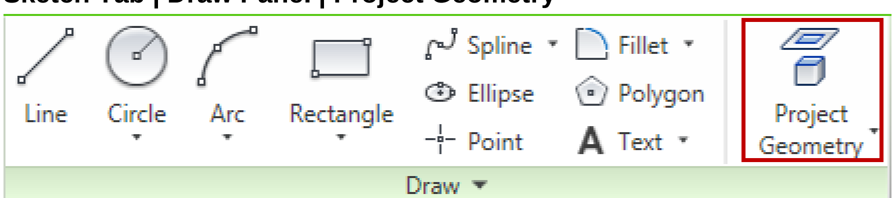
- In the **New File** dialog box, click the **Metric** tab.
- Select *Standard (mm).ipt*.



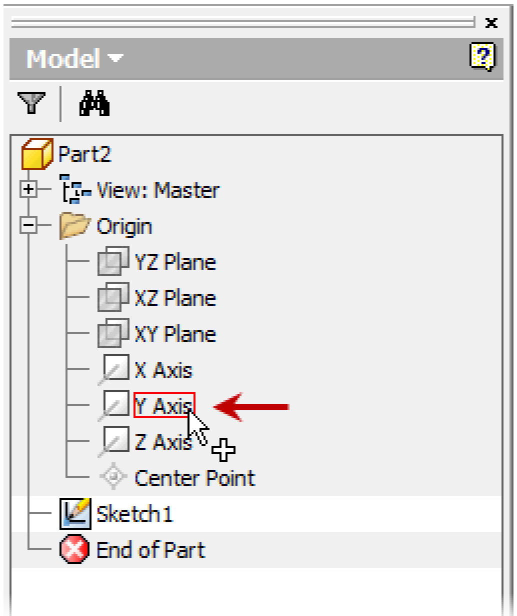
- Click **OK**.

2: Create sketch centerline

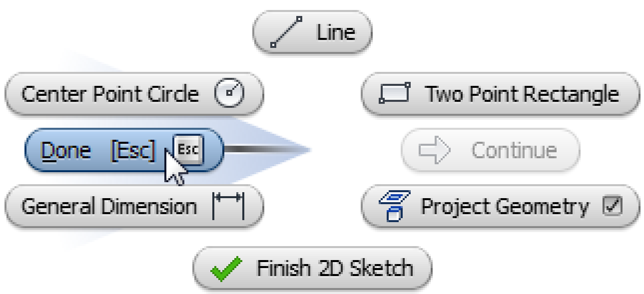
- Start the **Project Geometry** Tool
  Sketch Tab | Draw Panel | Project Geometry



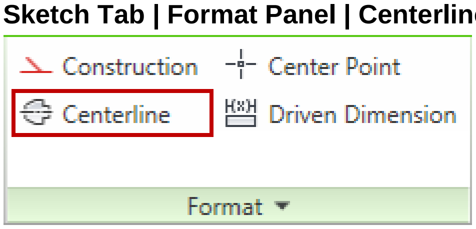
- In the browser select the **Y Axis** to project a reference line into sketch.



- Click **Done** on the Marking Menu



- Select the projected reference line in the graphics window and then change it to a centerline by selecting the **Centerline** format from the ribbon.
  Sketch Tab | Format Panel | Centerline

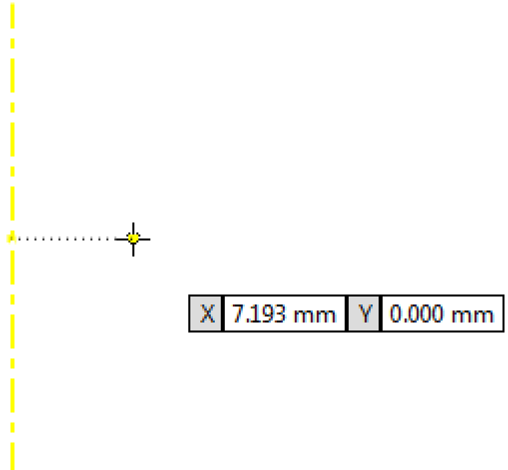


3: Sketch basic shape

- Start the Line tool.
  In this project you will sketch the approximant profile of the part completely without dimensions, and then add the required geometric and dimensional constraints to complete the sketch.

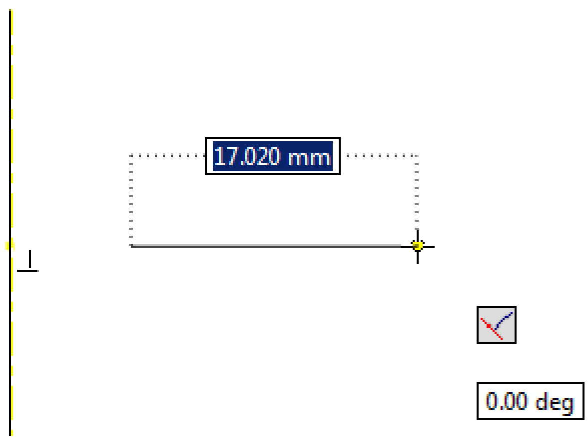
- Scrub over the projected origin point and drag to the right approximately 7 mm.
  This will line up you cursor with the origin point.

Note: The term “Scrub” refers to passing the cursor over geometry in the graphics window without clicking on it in order to use that geometry in reference to another sketch element being created. Autodesk Inventor will try to infer what areas to reference, but sometimes you may have to Scrub over a point or other sketch element before it will be referenced.

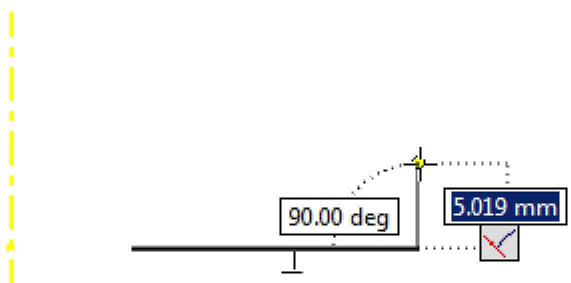


A dotted line will show that you are lined up with the point.

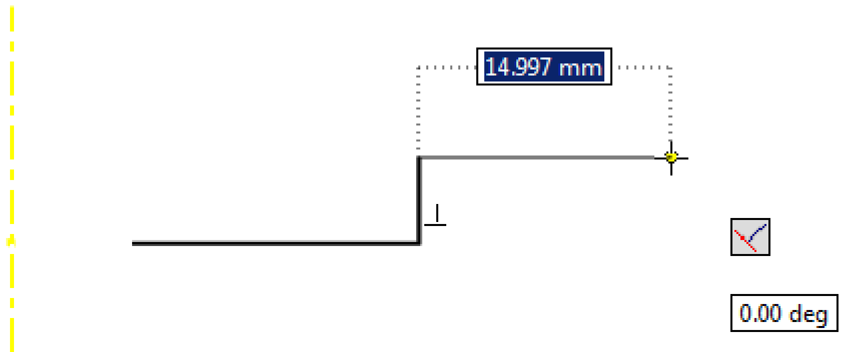
- Left-Click to create the starting point of the first line segment. Drag to the right approximately 17 mm. Click to except this line segment.



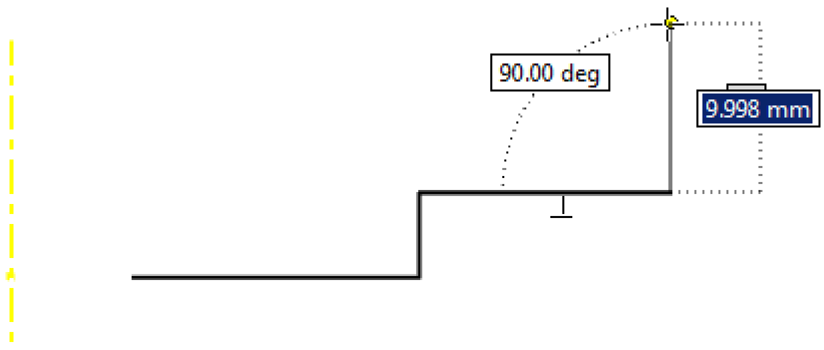
- Drag up approximately 5 mm. Click to except this line segment.



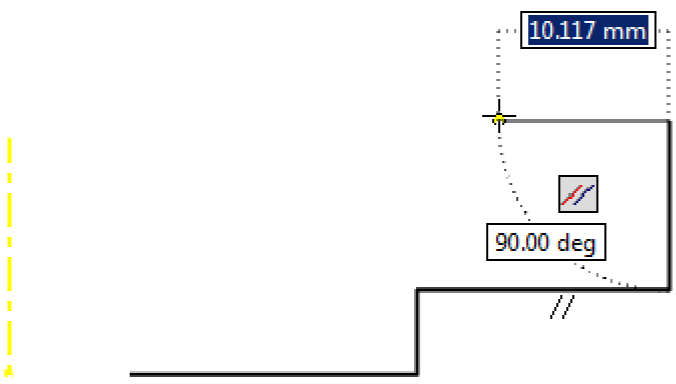
- Drag to the right approximately 15 mm. Click to except this line segment.



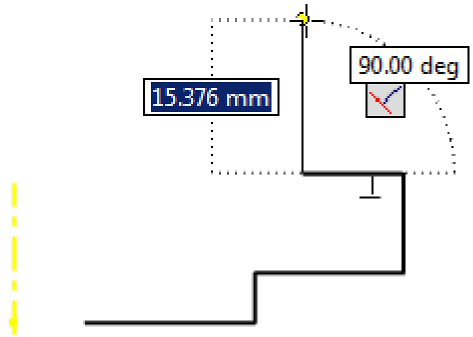
- Drag up approximately 10 mm. Click to except this line segment.



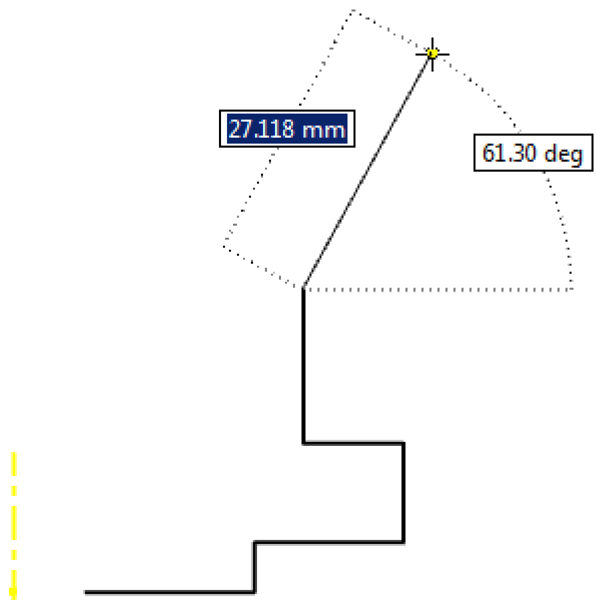
- Drag to the left approximately 10 mm. Click to except this line segment.



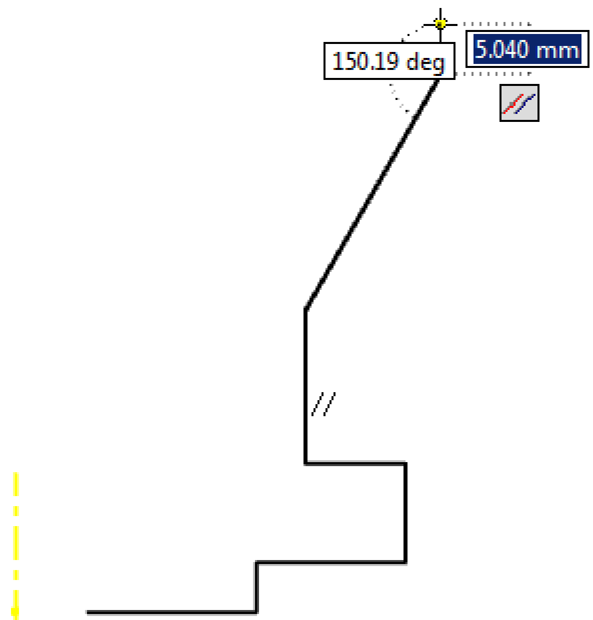
- Drag up approximately 15 mm. Click to except this line segment.



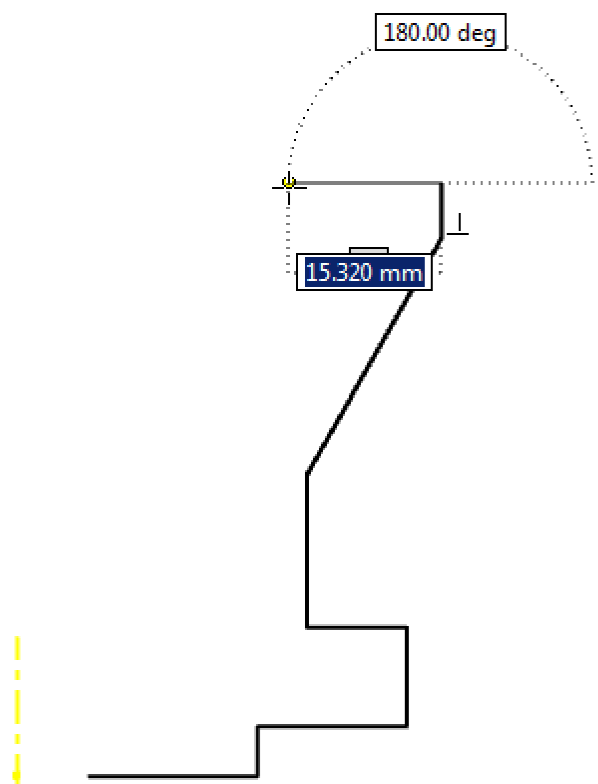
- Drag up and to the right to create an angled line segment similar to the one shown.



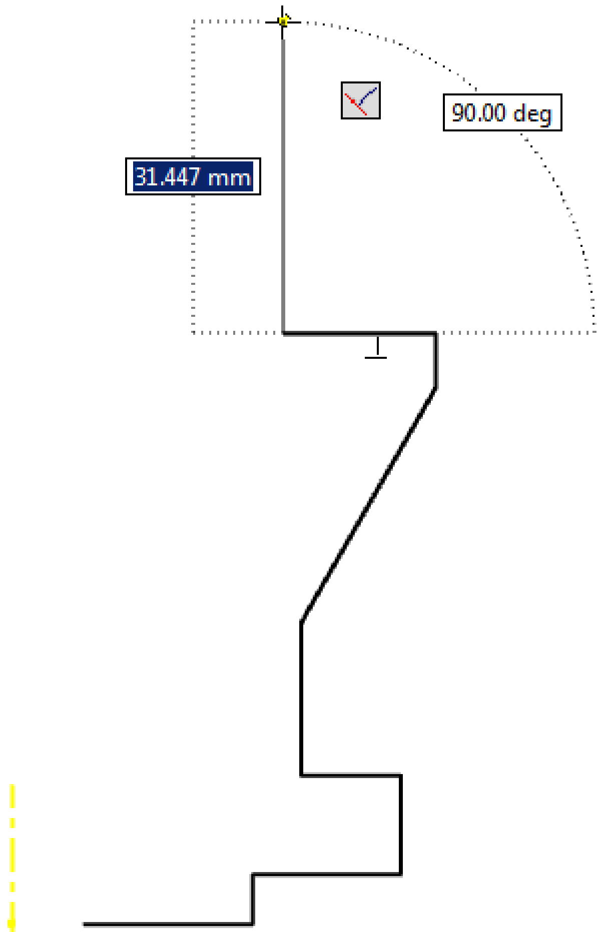
- Drag up approximately 5 mm. Make sure the Parallel Constraint glyph is shown next to the cursor to insure this line segment is parallel to the other vertical line segments. Click to except this line segment.



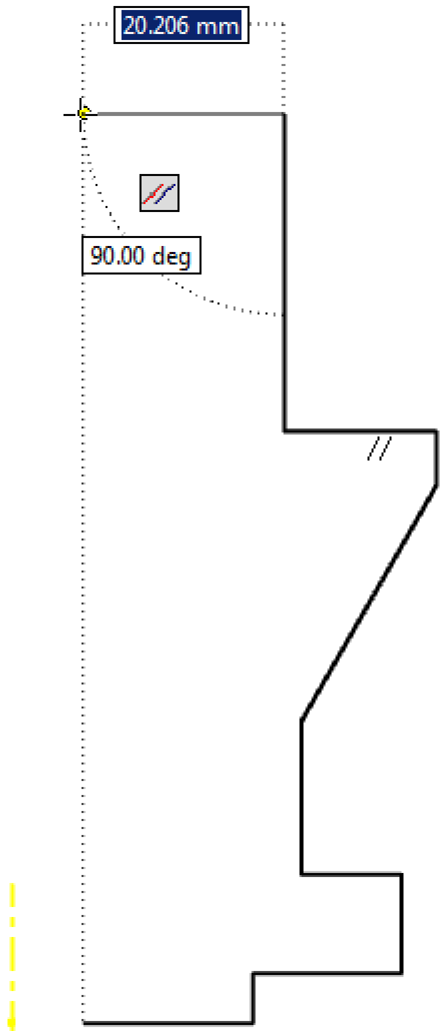
- Drag to the left approximately 15 mm. Click to except this line segment.



- Drag up approximately 31 mm. Click to except this line segment.



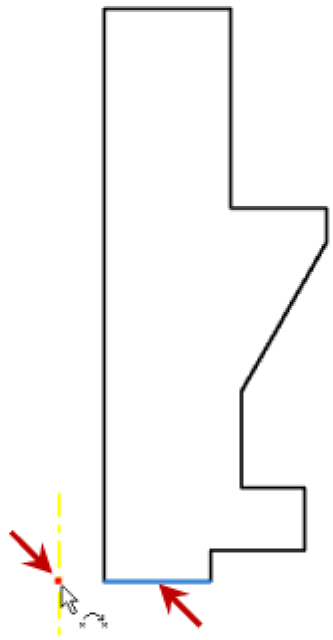
- Scrub the first starting point to line up this line segment with that point. When the dotted line is shown, and the line is parallel to the other horizontal lines (Or Perpendicular to the previous line segment) click to except.



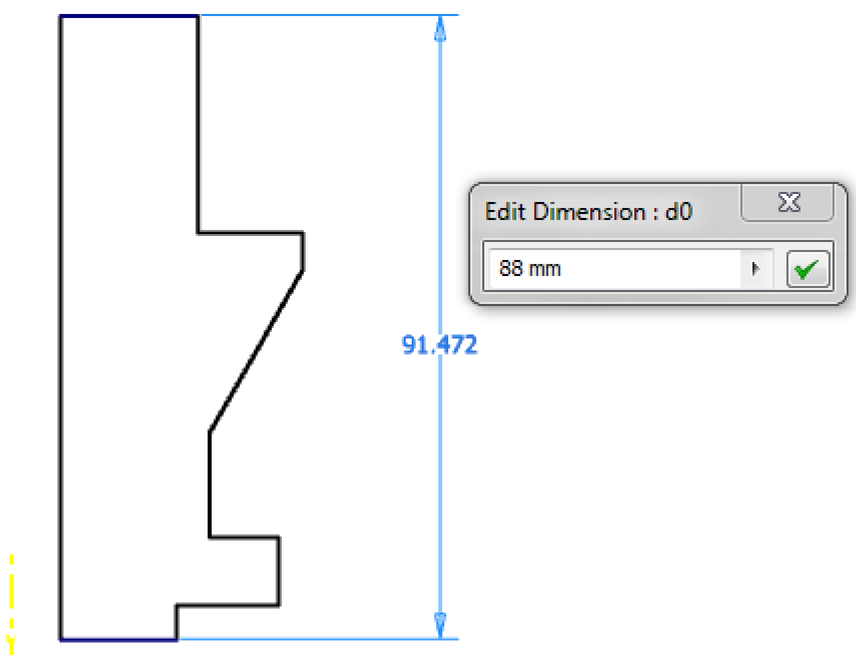
- Close the loop by picking the last line segment back to the first start point.

4: Constrain Sketch

- Place a **Coincident** Constraint between the bottom line segment and the projected Origin point.

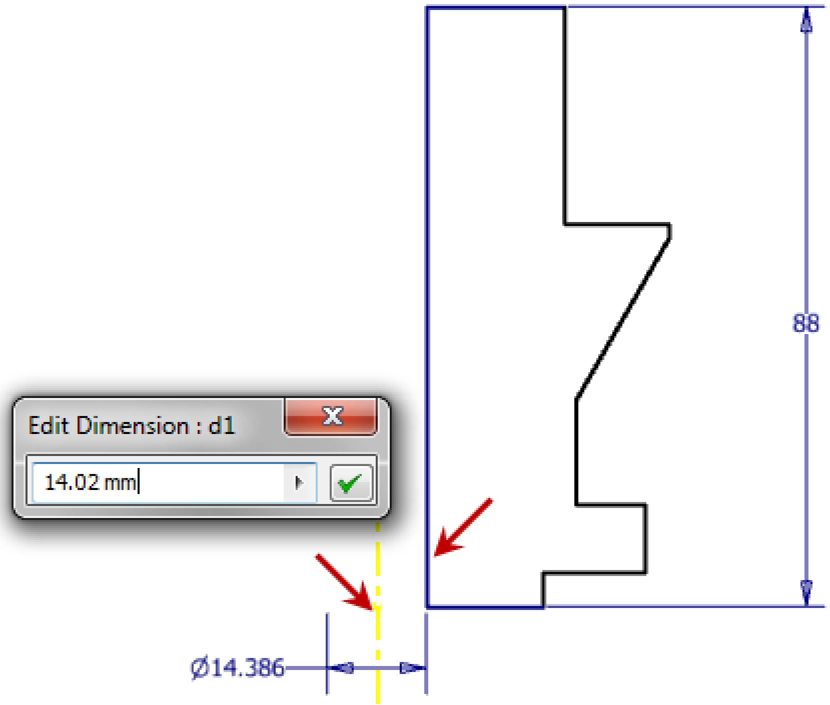


- Place linear dimensional constraint for the overall part length. Enter 88 mm in the edit dimension field.

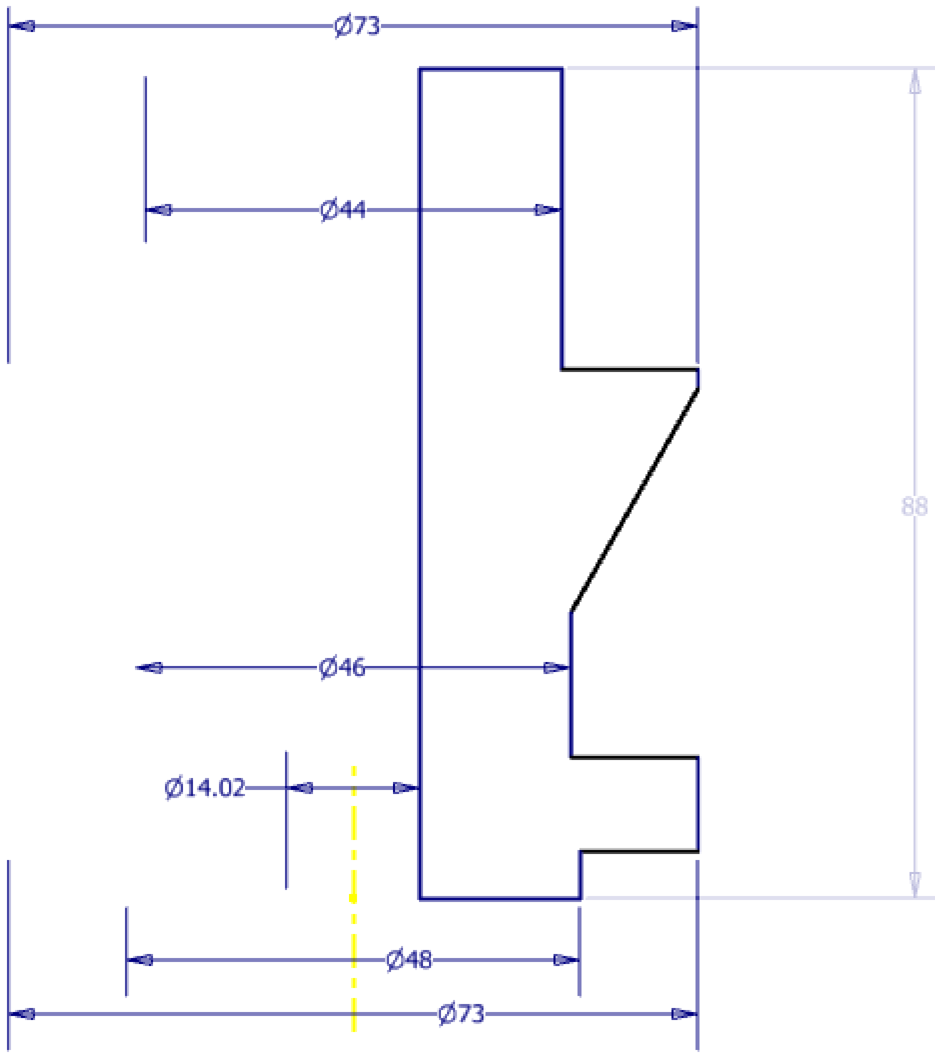


- Place an inner diameter dimensional constraint.
  Using the **General Dimension** tool from the **Sketch** tab on the ribbon, select the projected reference centerline and the inside edge of the part profile.

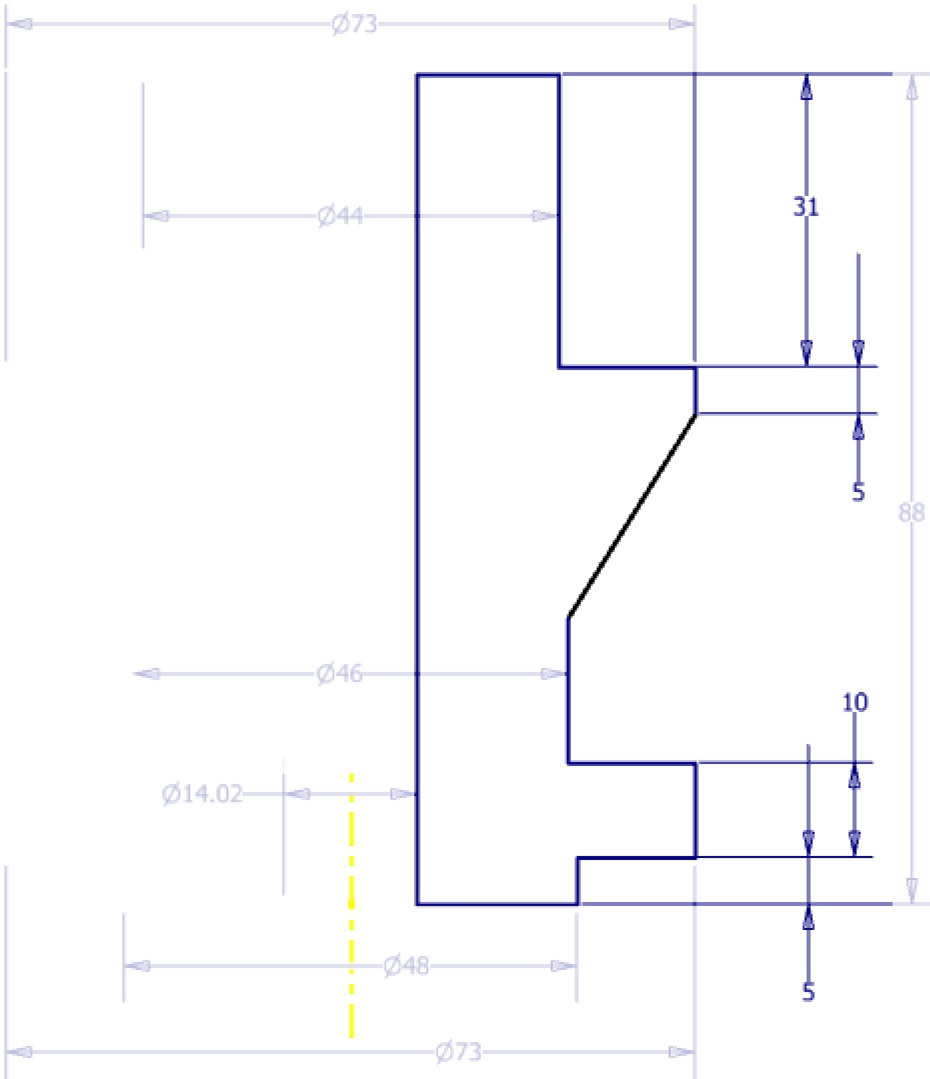
Notice that the dimension placed is a diameter.
Enter **14.02 mm** in the edit dimension field.



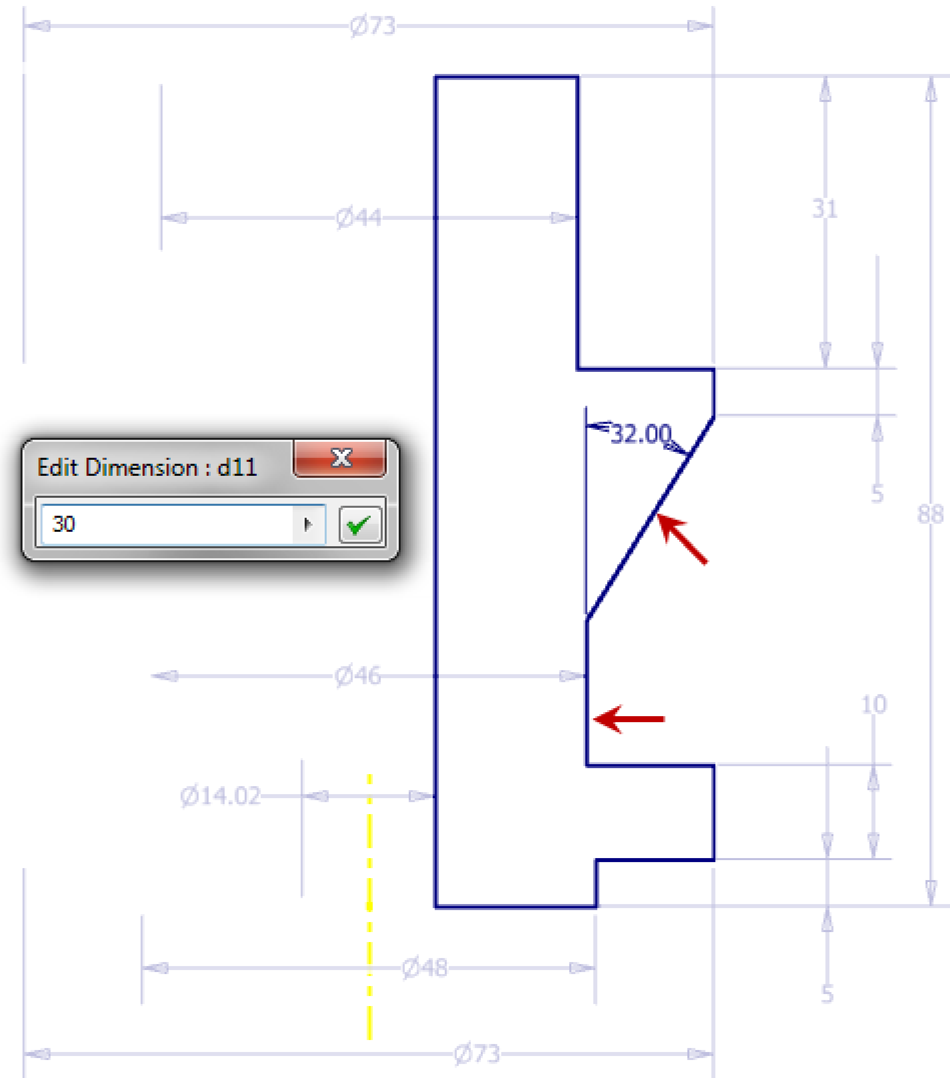
- Place the following diameter dimensional constraints.



- Place the following linear dimensional constraints.



- Place an angular dimensional constraint.
  Using the **General Dimension** tool from the **Sketch** tab on the ribbon, select the angled line segment and a vertical line segment. Enter **30** in the edit dimension field.

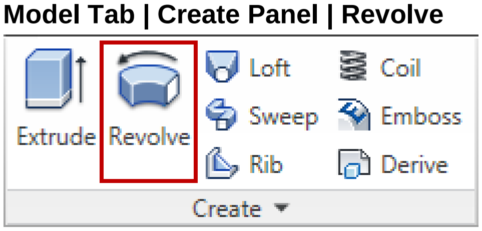


- The sketch profile is now fully constrained.

- Exit the sketch.

5: Revolve Base Feature.

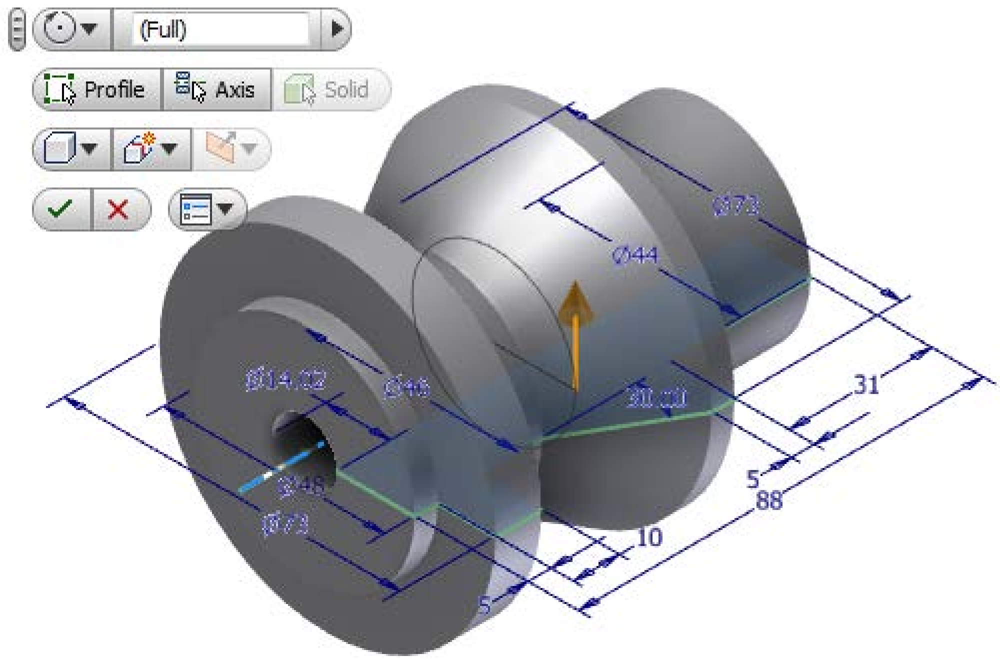
- Start the **Revolve** Tool
  Model Tab | Create Panel | Revolve



Project 2 – Flange Manifold Part

Because there is only one closed loop exists in the sketch, the sketch profile is automatically selected

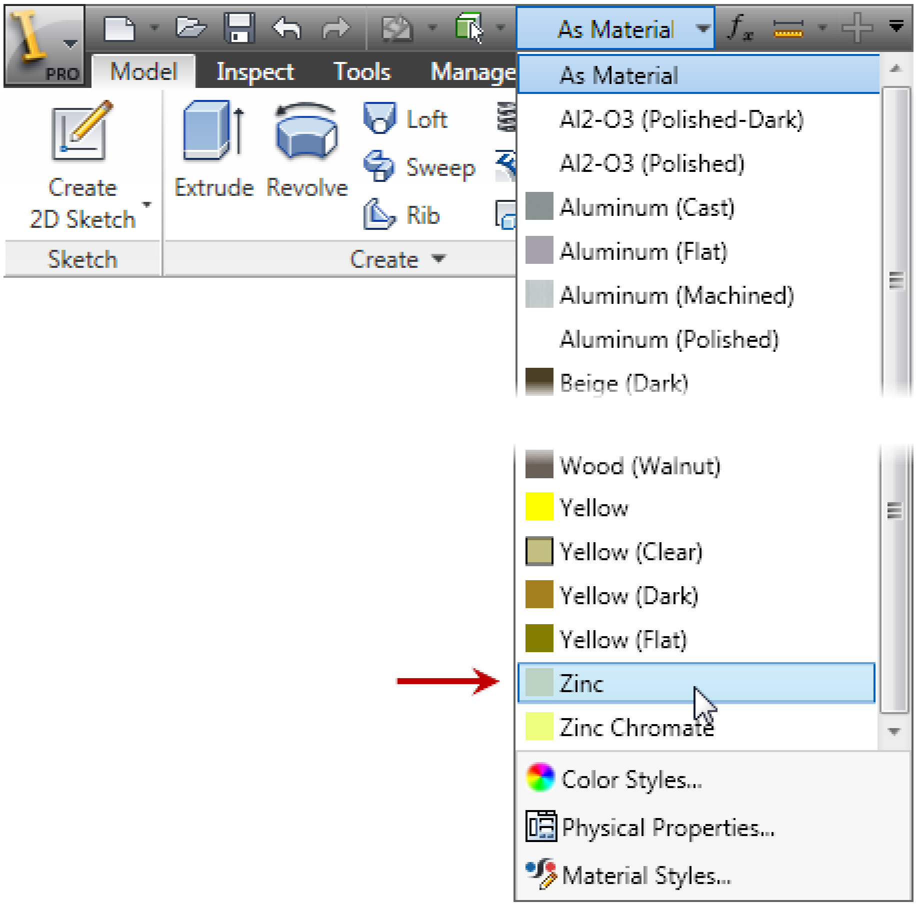
Click **OK**



6: Using the Viewing tools reposition the view as shown.

7: Change Part Color

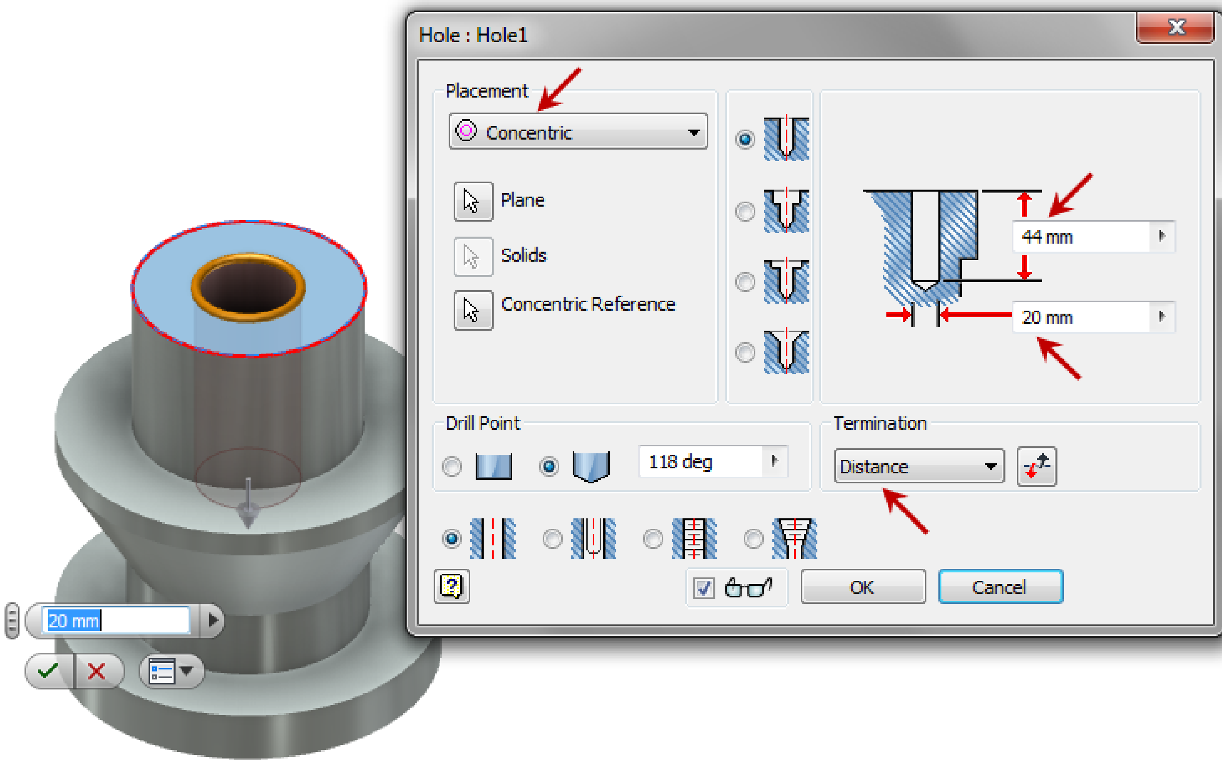
- On the Quick Access Toolbar pick **Zinc** from the part color drop down list.



8: Add Cavity Hole and End Tapped Hole

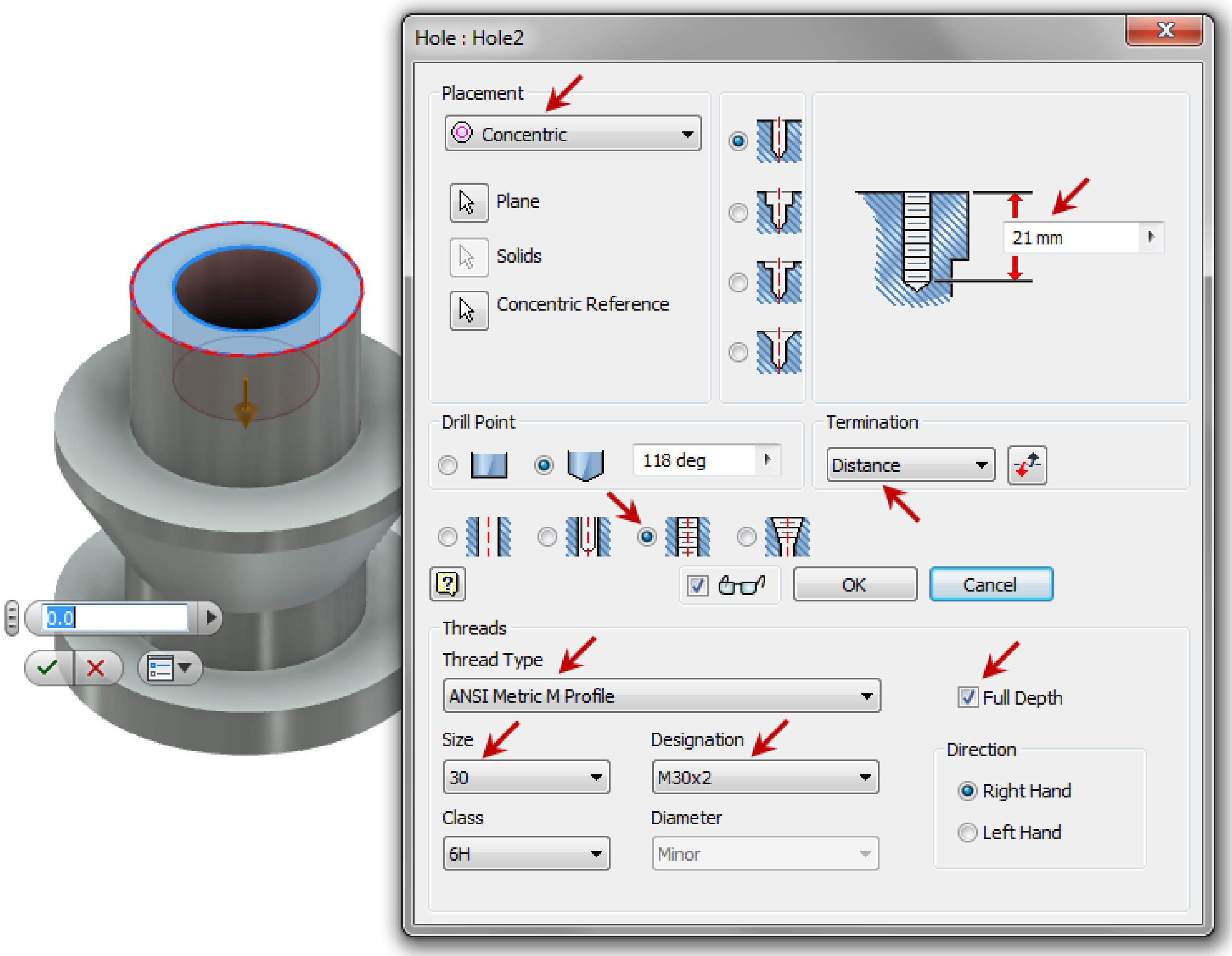
- Start the Hole Tool.
  Place a **Concentric** Hole on the top face with the following options:
  Termination: **Distance**
  Diameter: **20 mm**
  Depth: **44 mm**

  Click **OK**



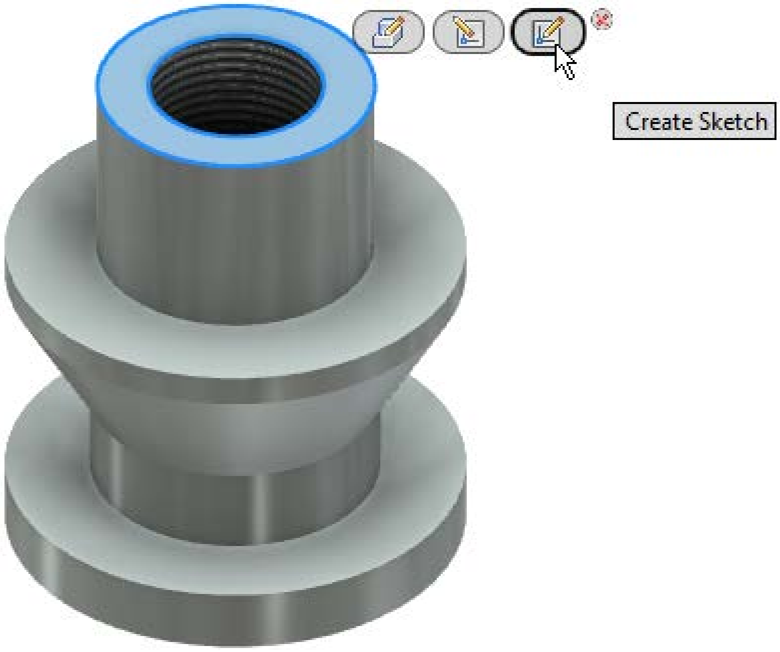
- Start **Hole** Tool
  Place a **Concentric** Hole on the top face with the following options:
  Type: **Tapped Hole**
  Thread Type: **ANSI Metric M Profile**
  Size: **30**
  Designation: **M30x2**
  Termination: **Distance**
  Depth: **21 mm**
  Full Depth

  Click **OK**

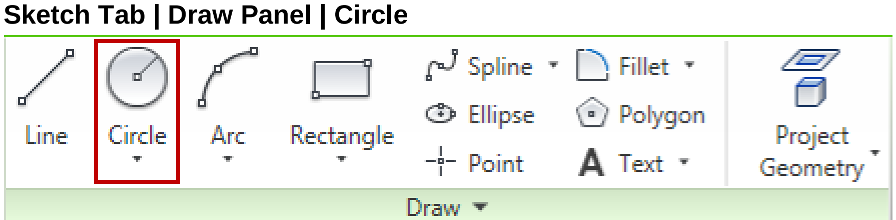


9: Add Extrusion Cut

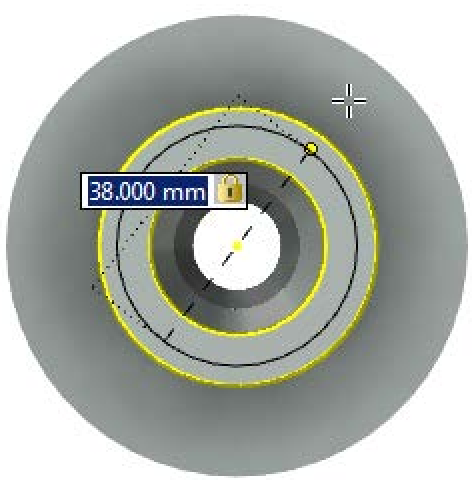
- Start a new sketch on the top surface



- Start the Center Point Circle Tool
  Sketch Tab | Draw Panel | Circle



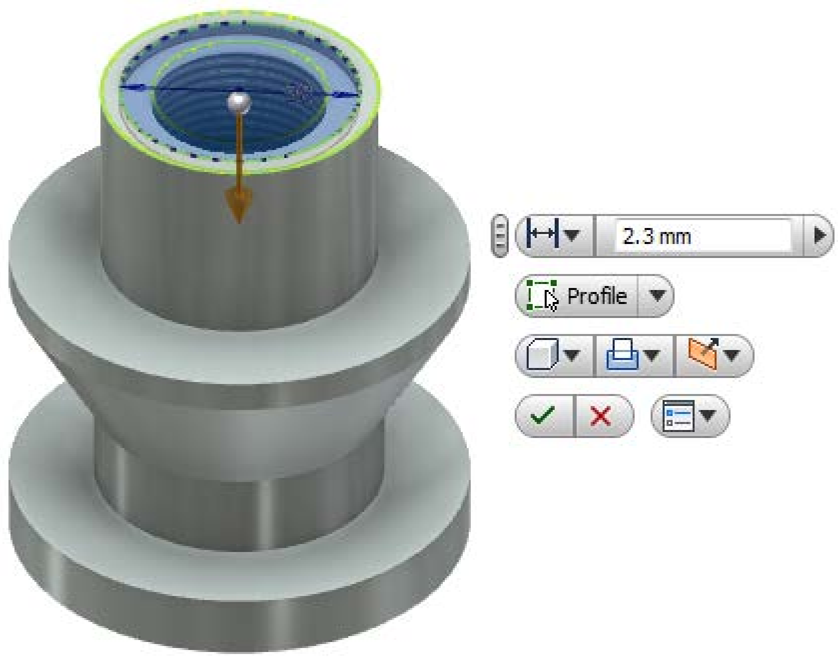
Pick the projected center point for the Center Point Circle starting point.
Drag away from the center. Enter 38 mm in the direct entry field. Press Tab to lock in the value.
Press ENTER to except the circle.



- Exit Sketch

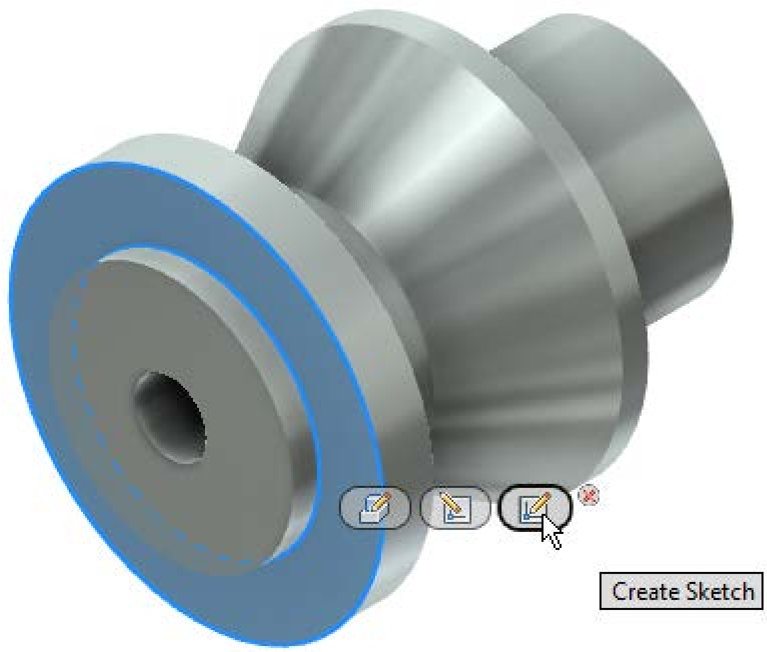
- Start the Extrude Tool
  Pick both profiles inside the sketched circle.

  Select the **Cut** option
  Enter **2.3 mm** in the entry field
  Click **OK**

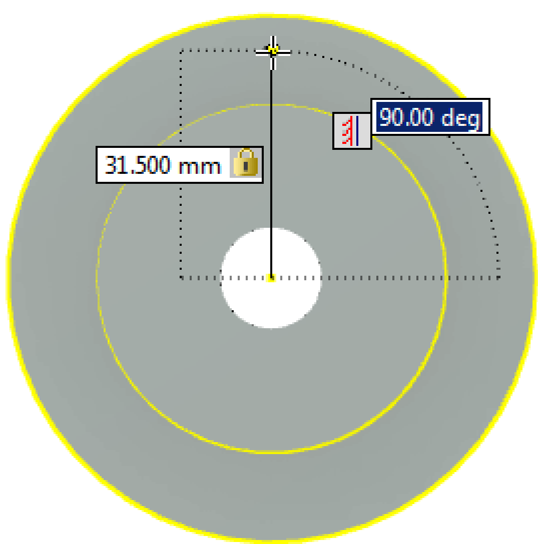


10: Add bolt pattern

- Create Sketch on bottom face shown.



- Start the **Line** Tool
  Draw vertical line starting from the center point **31.5 mm** long



- Exit the Sketch

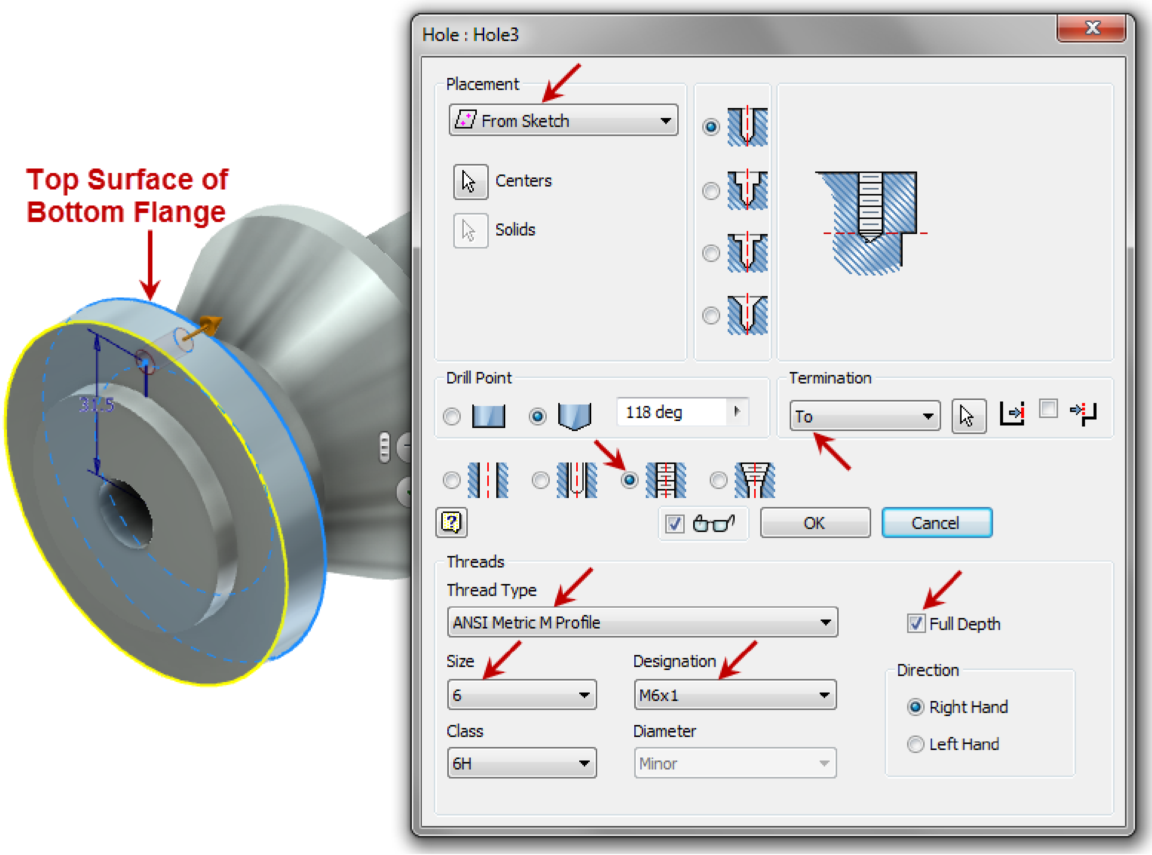
- Start **Hole** Tool
  Place a **From Sketch** Hole picking the line endpoint in the sketch with following options:
  Type: **Tapped Hole**
  Thread Type: **ANSI Metric M Profile**
  Size: **6**
  Designation: **M6x1**

Termination: **To**

Pick the top surface of the bottom flange area for the “To” termination surface

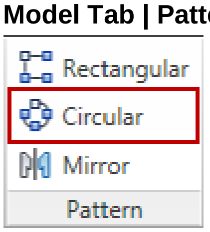
Full Depth

Click **OK**

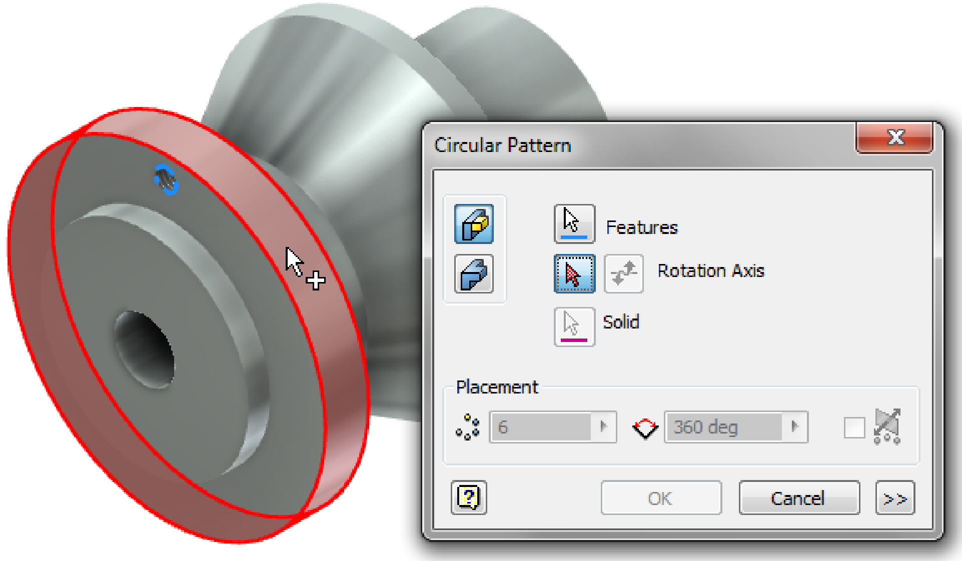


- Start the **Circular Pattern** tool

  Model Tab | Pattern Panel | Circular

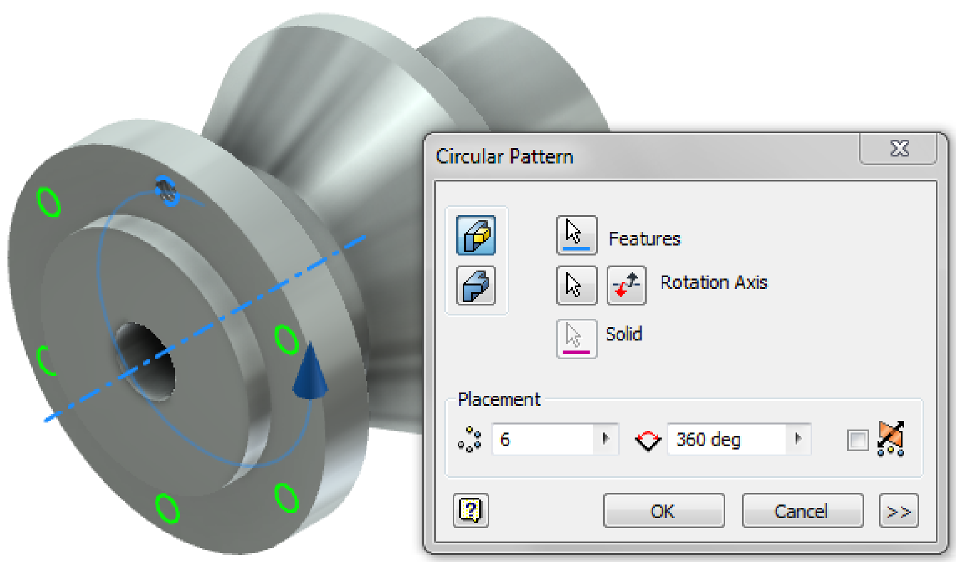


- Select the M6 tapped hole feature
  Select the outside diameter surface of the flange for the Rotation Axis

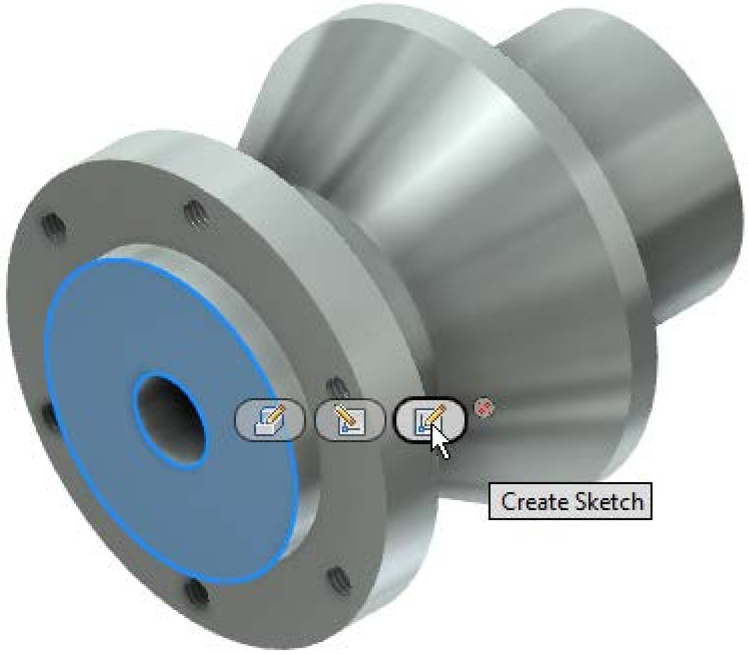


- Enter Pattern Placement: **6**
  Angle: **360 deg**

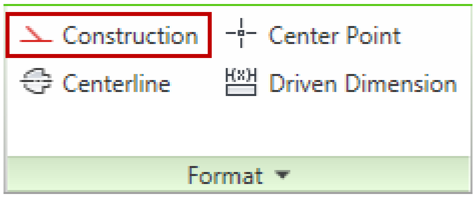
  Click **OK**



11: Create rotational location holes
Create a new sketch on the bottom surface

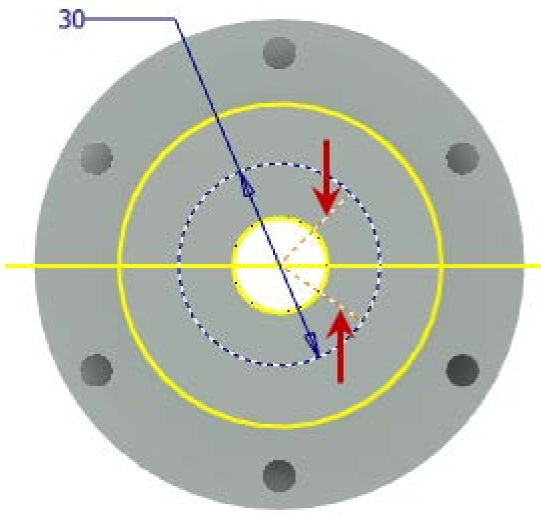


- Change the sketch line format to construction by selecting the **Construction** format from the ribbon.
  Sketch Tab | Format Panel | Construction

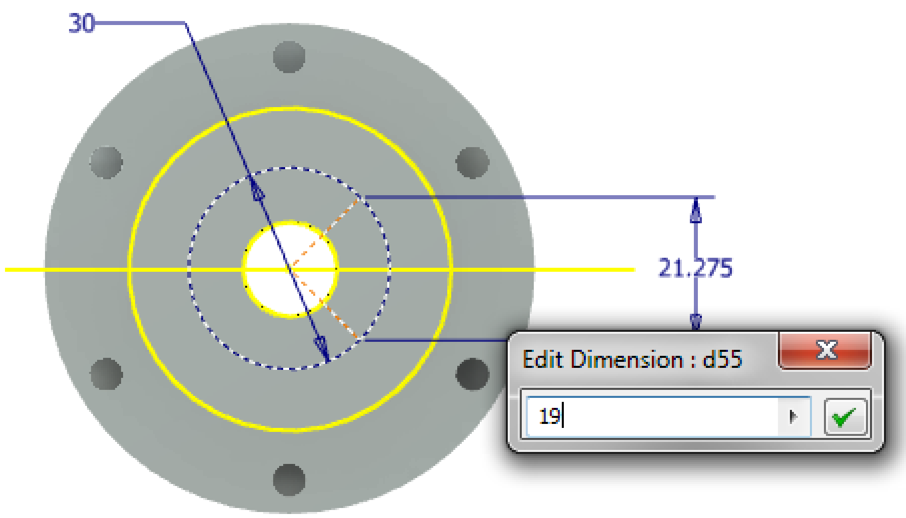


- Start the **Center Point Circle** Tool
  Draw a circle **30 mm** in diameter

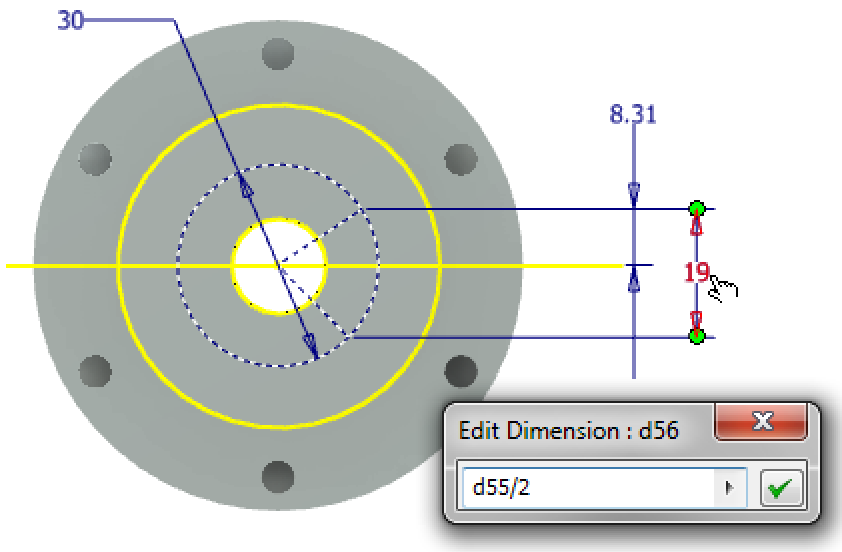
- Sketch two lines as shown.
  Starting from the center point and picking the construction circle for the second point
  Note: Make sure these line are NOT perpendicular to each other.



- Place a dimensional constraint between the line endpoints.
  Enter **19 mm** into the edit dimension field



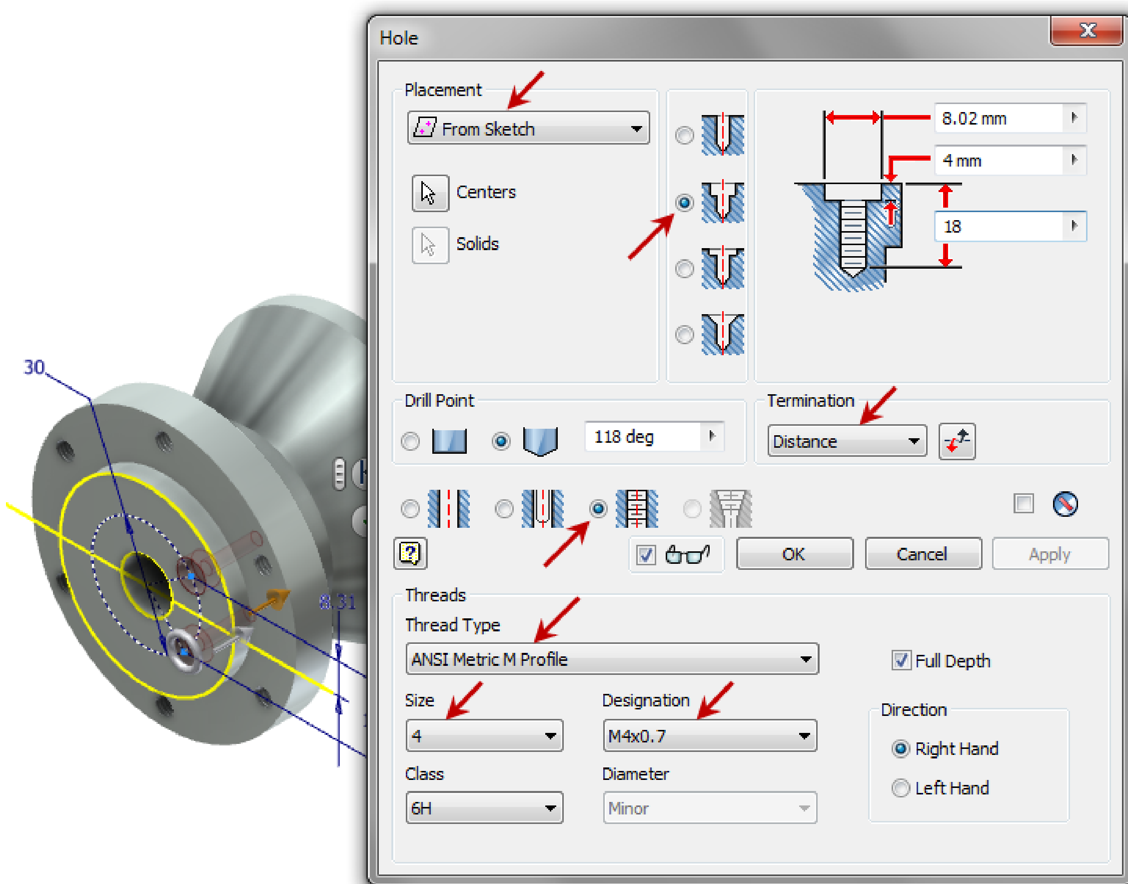
- Place a dimensional constraint between the upper line endpoint and the projected XY Plane.
  - With the Edit Dimension field active, click on the 19 mm dimension.
    The dimension reference will be displayed in the Edit Dimension field (For this example it is shown as *d55*, it will be different for you)
  - Create an equation by adding a forward slash "**/**" (for divide) and a "**2**
  - Click **OK**
  - The value of this dimensional constraint will now be half the distance between the two line endpoints.



- Exit the Sketch

- Start **Hole** Tool
  Place a **From Sketch** hole picking both line endpoints in the sketch with following options:
  Style: **Counterbore**
  Type: **Tapped Hole**
  Termination: **Distance**
  Thread Type: **ANSI Metric M Profile**
  Size: **4**
  Designation: **M4x0.7**
  Counterbore Diameter: **8.02 mm**
  Counterbore Depth: **4 mm**
  Overall Depth: **18 mm**

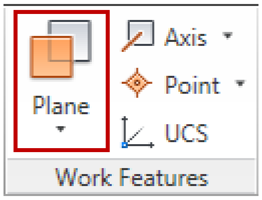
  Click **OK**



12: Reposition the view as shown

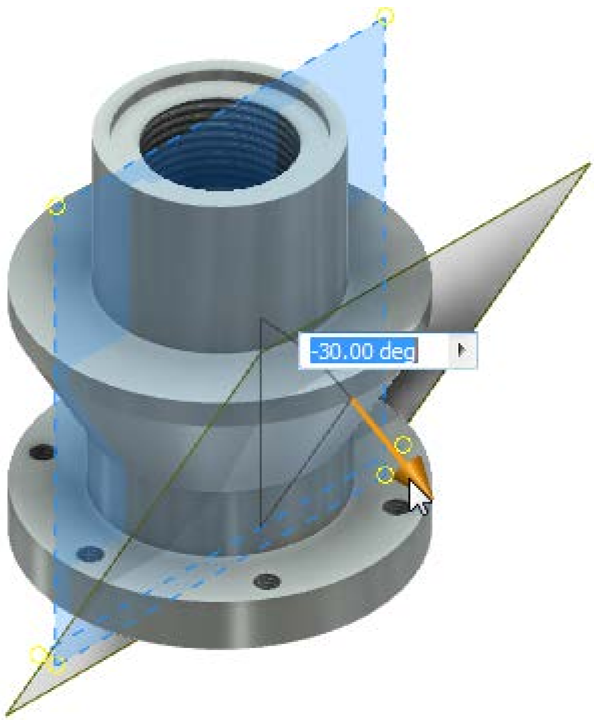
13: Create a Work Plane

- Start the **Work Plane** tool
  Model Tab | Work Features Panel | Plane



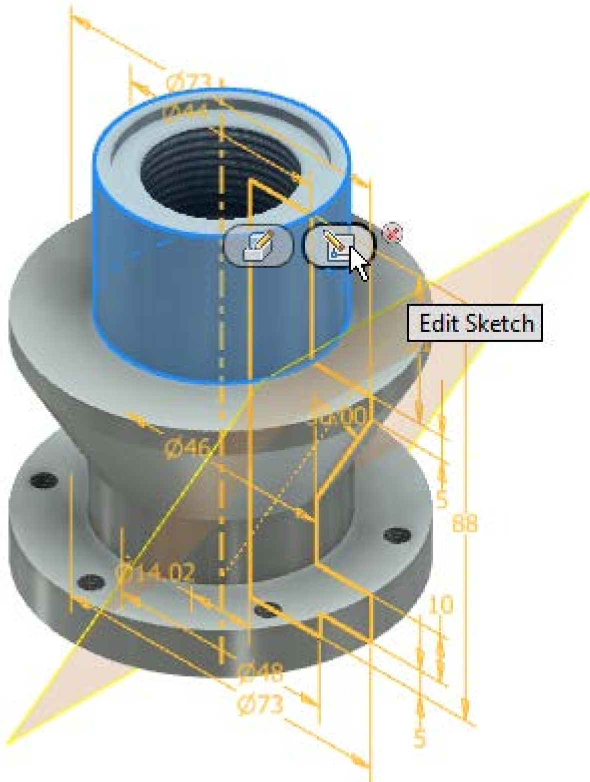
- In the browser click the **Z Axis** for the first reference
  Pick the **YZ Plane** for the second reference

- Drag the direct manipulation arrow back until the work plane is shown at **-30.00 deg**
  Click **OK**



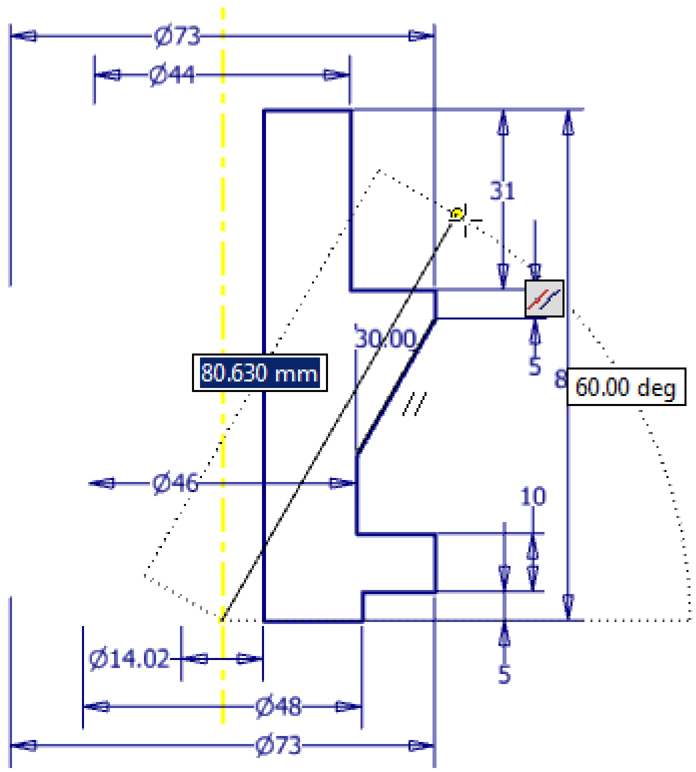
14: Create an Offset Work Plane

- First you need to get a measurement for the offset.
  Click the model and click **Edit Sketch** to activate the first profile sketch you created.

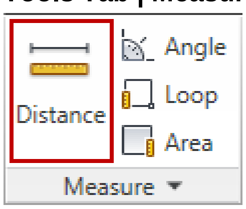


Ø73
Ø44
Edit Sketch
Ø46
5
88
Ø14.02
10
Ø48
Ø73
5

- Draw a construction line parallel to the angled line in the sketch

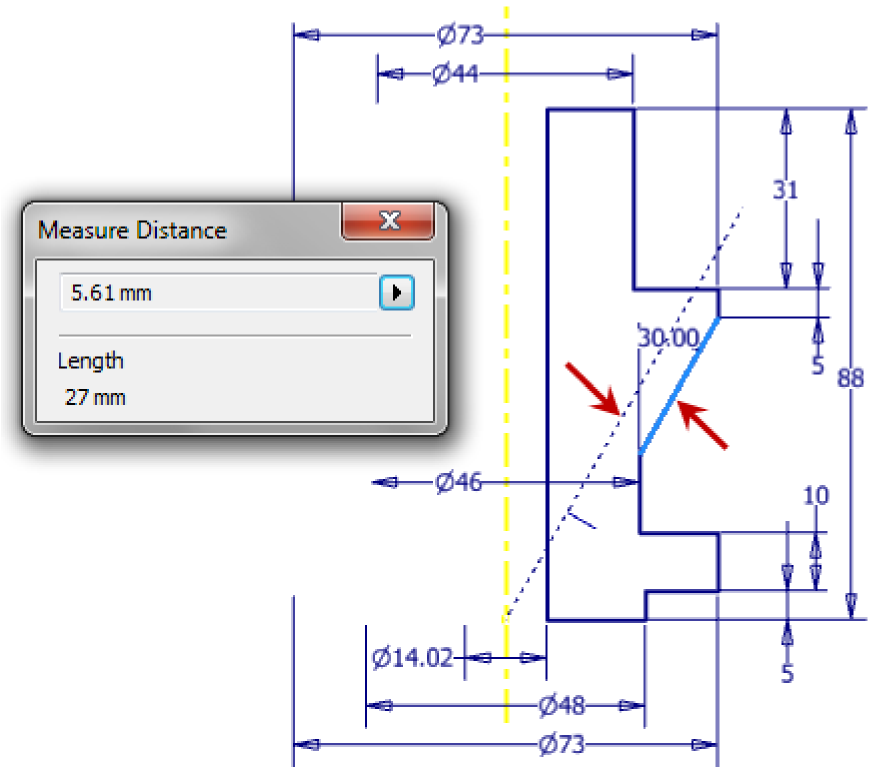


- Start the Measure Distance tool
  Tools Tab | Measure Panel | Distance



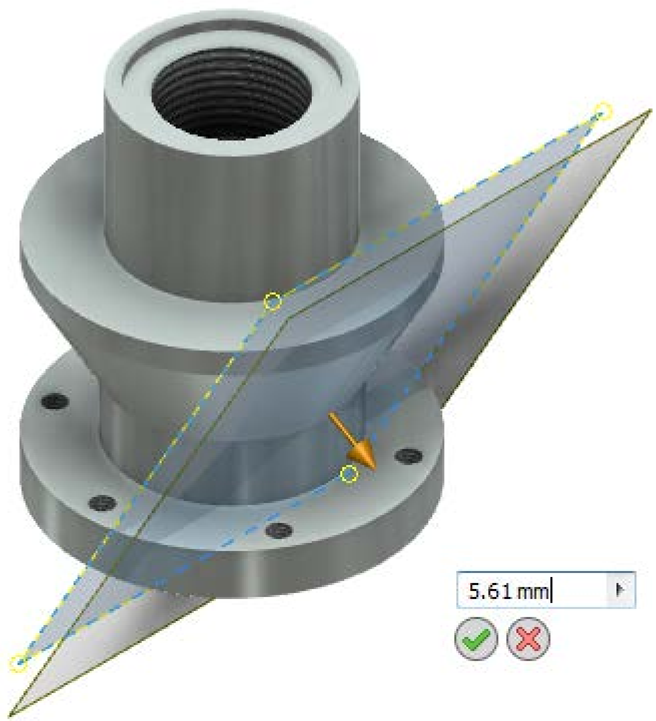
Click on the original angled line and the new construction line to get the measurement between them**.** A distance of *5.61 mm* is shown in the **Measure Distance** dialog box.
Copy this dimension for use in the next step.

Exit the sketch.



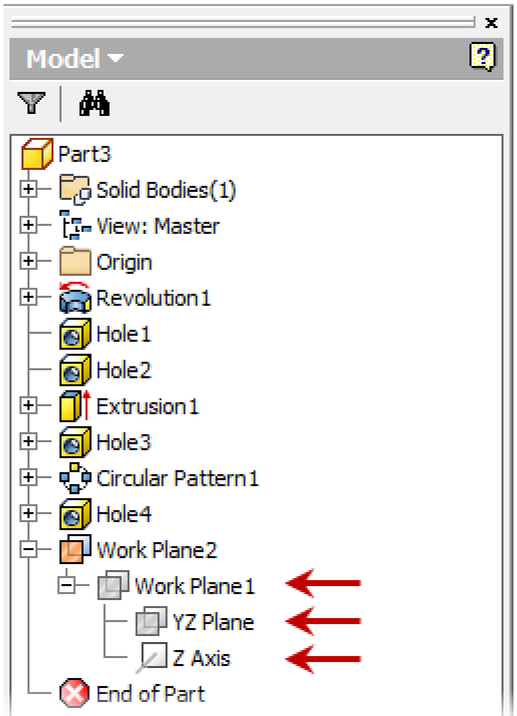
- Start the **Work Plane** tool
  Click-Hold on the previous Work Plane and Drag away from it.
  Enter **5.61 mm** into the direct entry field

  Click **OK**



- Right click on Work Plane1 and click **Visibility** on the Marking Menu
  This will turn the visibility of this work plane OFF

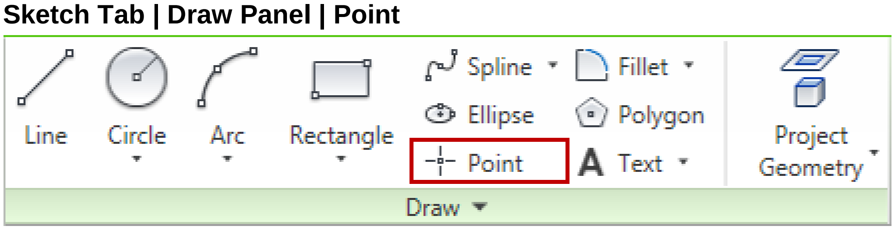
- Notice in the browser that all work features required to create *Work Plane2* are located as children under *Work Plane2*. They are shown as gray icons because their visibility has been turned OFF



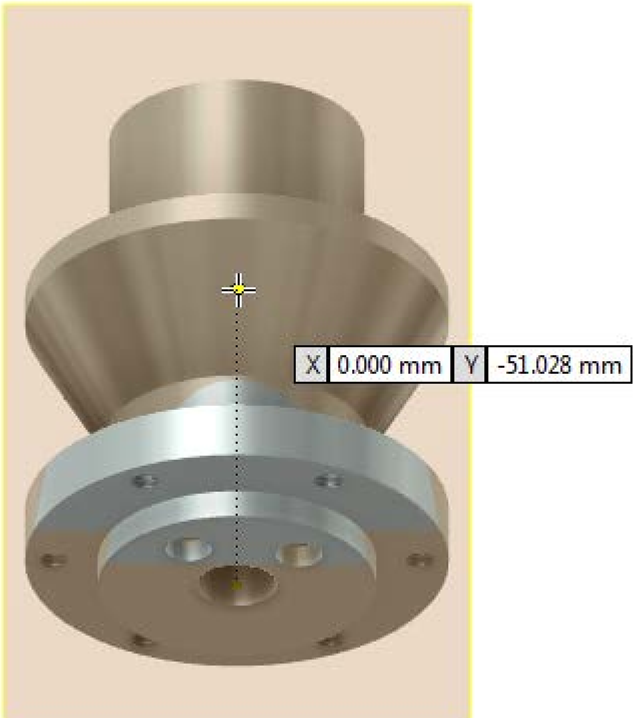
15: Create Manifold Hole Pattern On Round Angled Surface

- Create a new sketch on *Work Plane2*

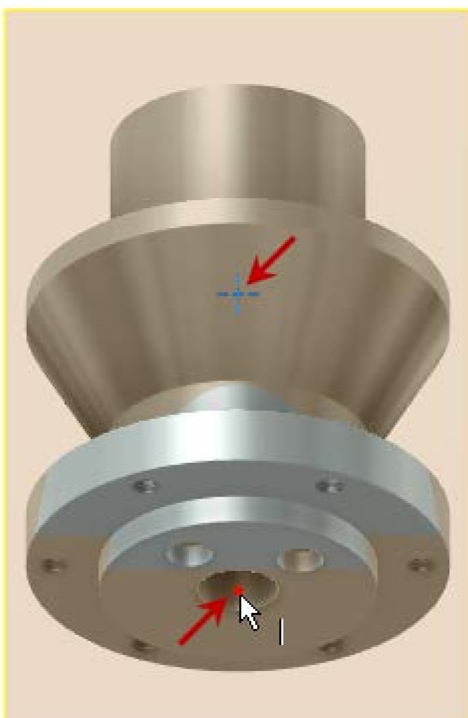
- Start the **Point** tool
  Sketch Tab | Draw Panel | Point



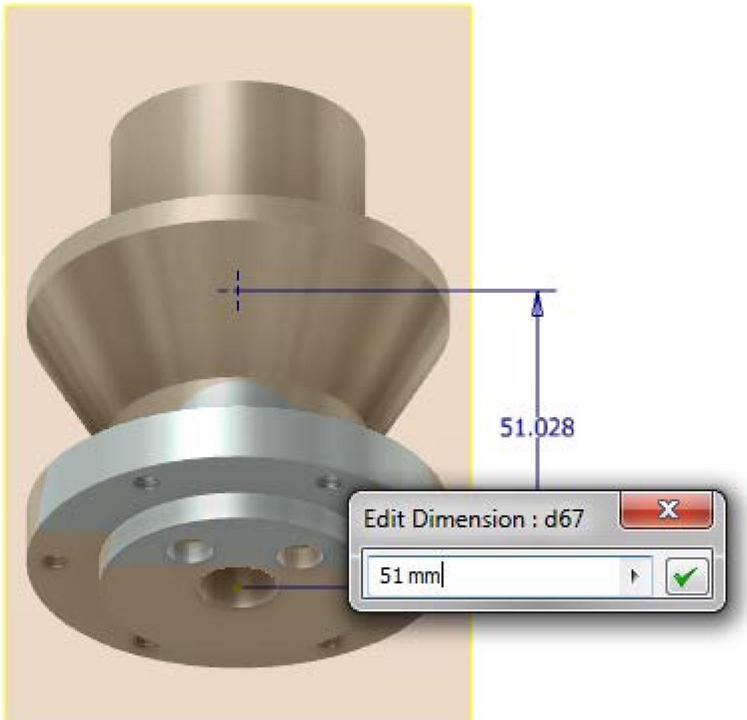
Place a Point approximately **-51 mm** directly above the project Origin point



- Place a Vertical constraint between the drawn point and the projected Origin point

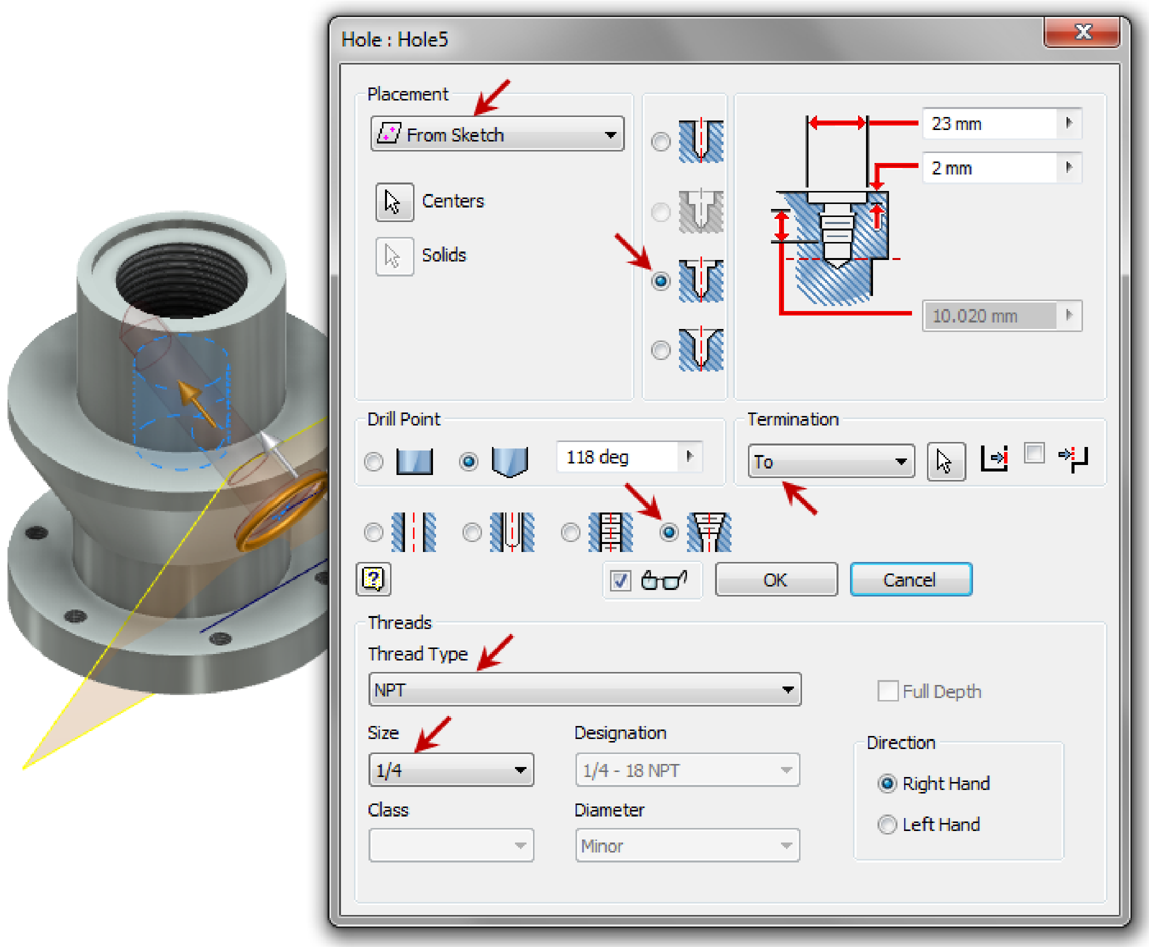


- Place a dimensional constraint of **51 mm** between the two points

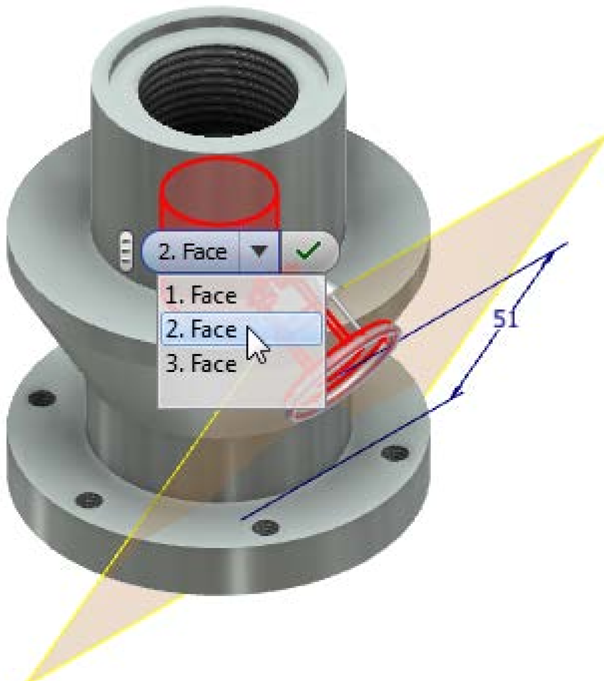


- Exit Sketch

- Start **Hole** Tool
  Place a **From Sketch** hole picking both line endpoints in the sketch with following options:
  Style: **Countersink**
  Type: **Taper Tapped Hole**
  Termination: **To**
  Thread Type: **NPT**
  Size: **1/4**
  Countersink Diameter: **23 mm**
  Countersink Depth: **2 mm**

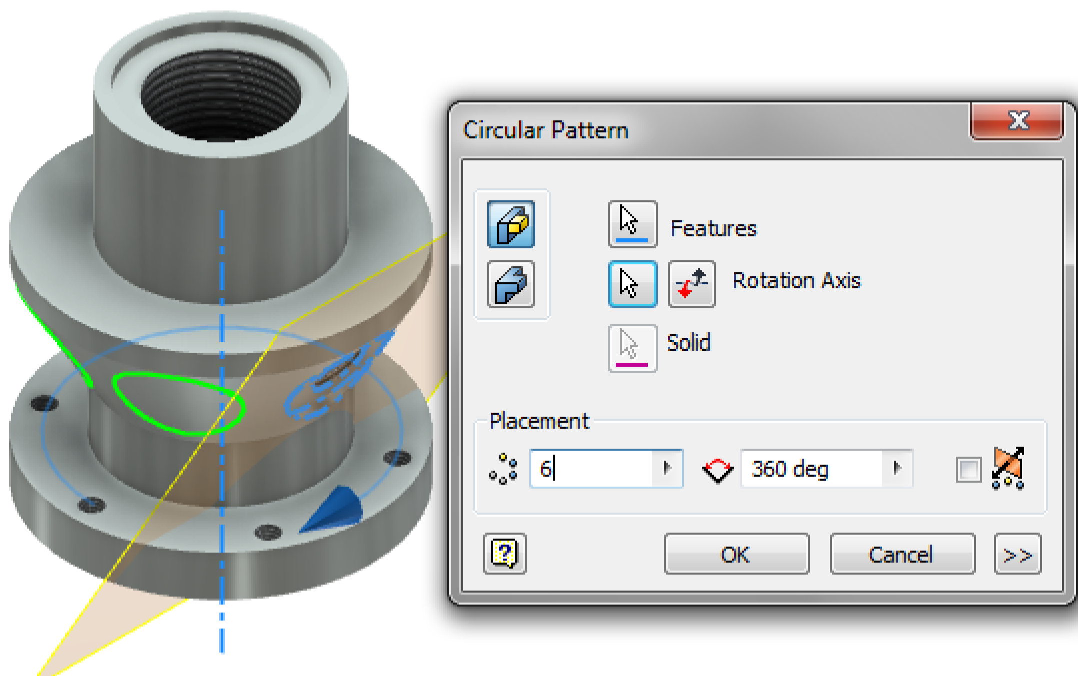


For “**To**” Termination hover cursor over the model until the **Select Other** drop down list appears
Select the inside diameter face



- Click **OK**

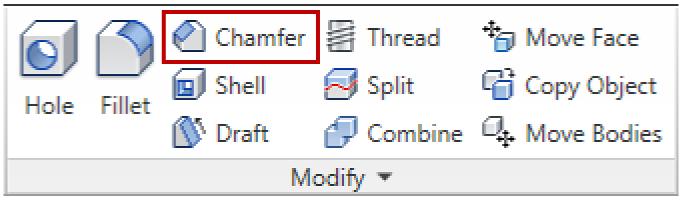
- Start the **Circular Pattern** tool
  Select the 1/4 NPT tapped hole feature
  Select the outside diameter surface of the flange for the Rotation Axis
  Enter Pattern Placement: **6**
  Angle: **360 deg**



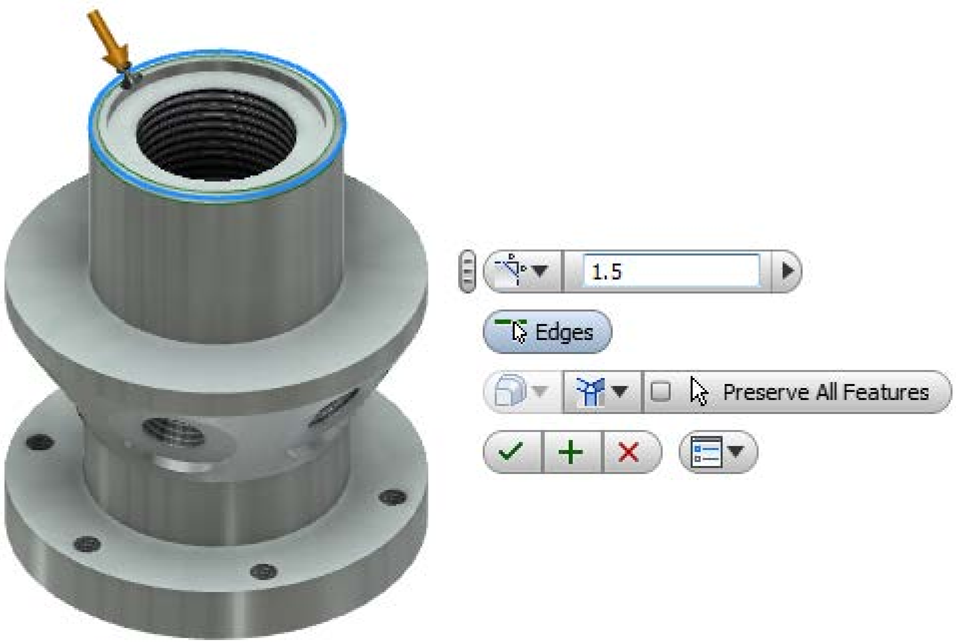
Click **OK**

16: Apply Chamfers

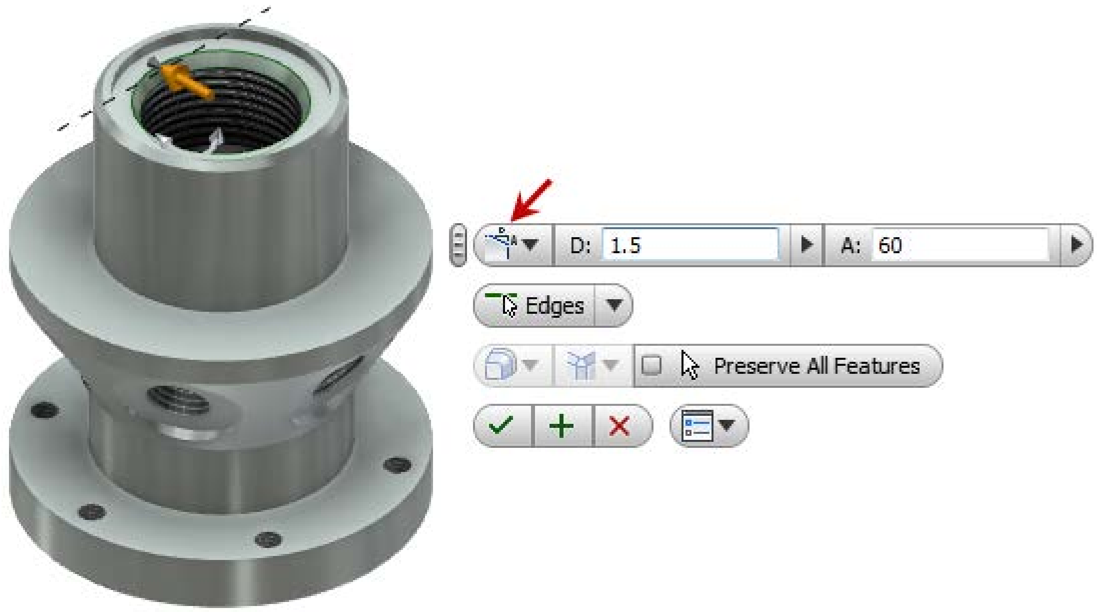
- Start the **Chamfer** tool
  Model Tab | Modify Panel | Chamfer



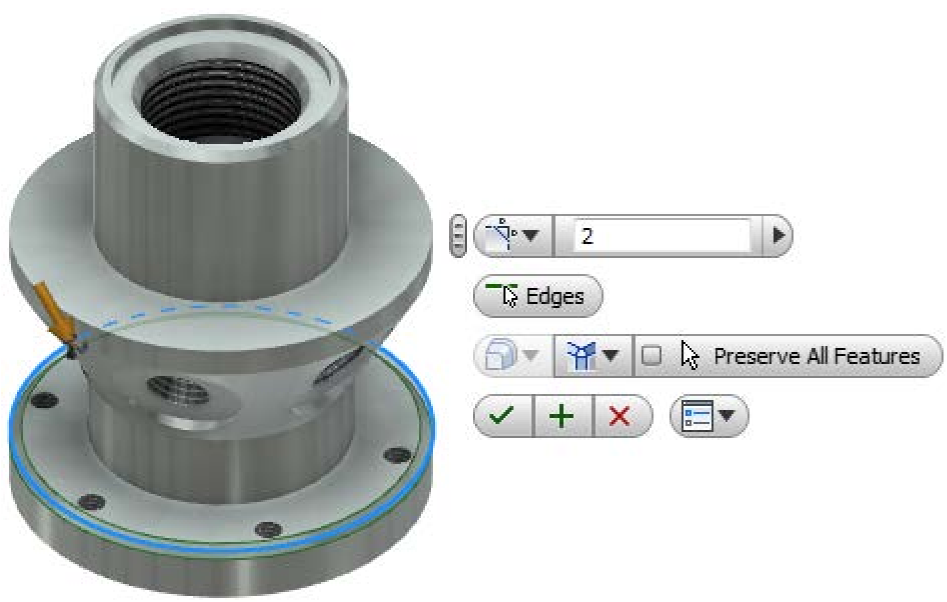
- Select the top edge
  Enter Distance: **1.5 mm** into the Mini-Tool Bar
  Click **Apply** 



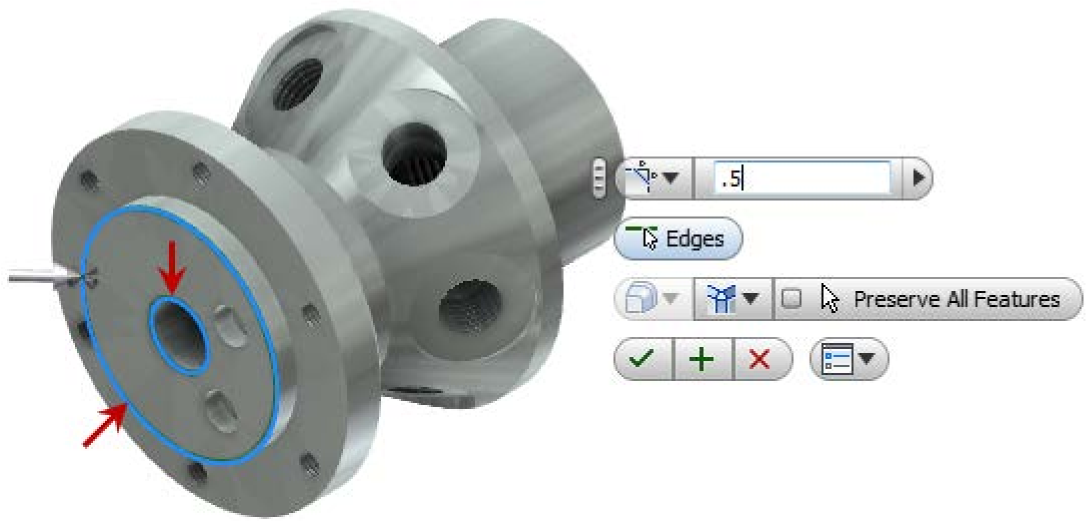
- Change Chamfer style to **Distance and Angle**
  Select top surface adjacent to the tapped hole and top edge of the tapped hole
  Enter Distance: **1.5 mm** and Angle: **60 deg**
  Click **Apply**



- Change Chamfer style to **Distance**
  Select top surface of bottom flange
  Enter Distance: **2 mm**
  Click **Apply**

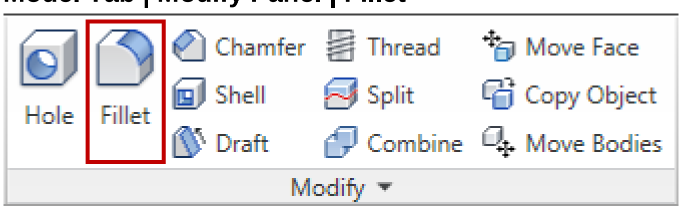


- Select the two bottom edges shown
  Enter Distance: **0.5 mm**
  Click **OK** to accept and exit the **Chamfer** tool

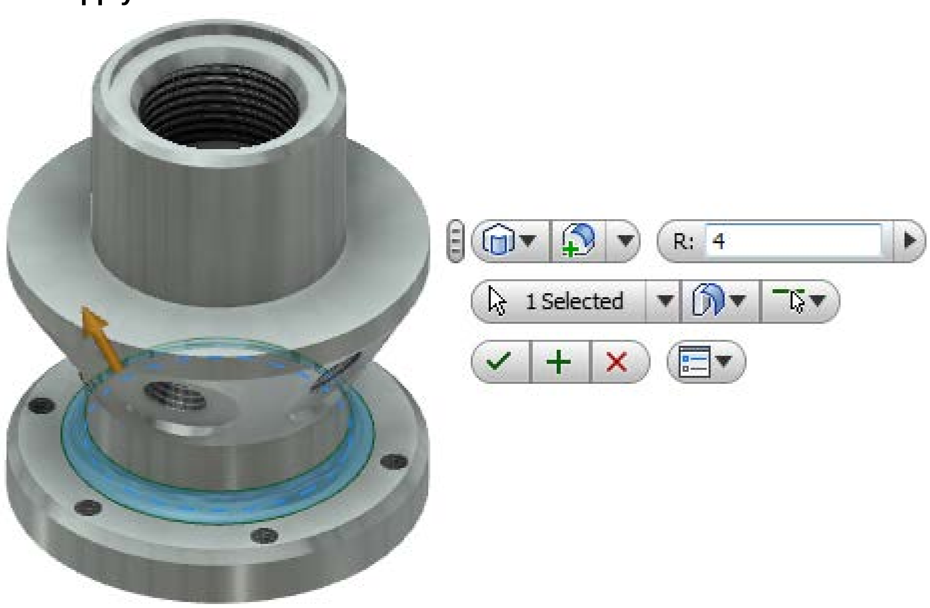


17: Apply Fillets

- Start the **Fillet** tool
  Model Tab | Modify Panel | Fillet



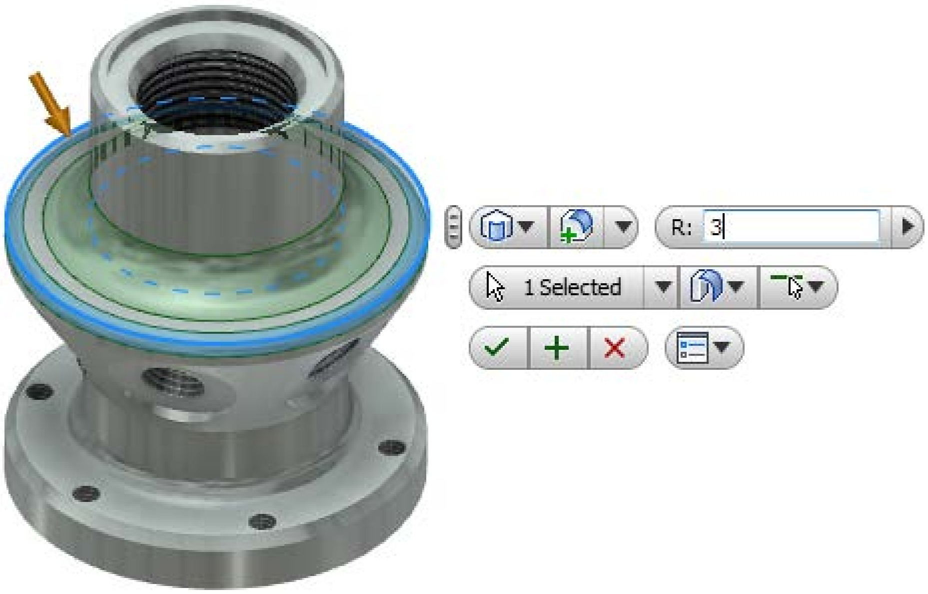
- Select edge as shown
  Enter Radius: **4 mm** into the Mini-Tool Bar
  Click **Apply**



- Add Fillet Set
  - Select edge as shown
    Enter Radius: **8 mm** into the Mini-Tool Bar

  - Click **Add Constant Fillet Radius Set** 

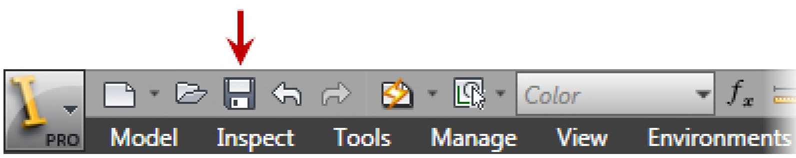
  - Select edge as shown
    Enter Radius: **3 mm** into the Mini-Tool Bar



  - Notice that both the 8mm and 3mm radiuses are created as one Fillet feature in the browser.

18: Save Part

- On the **Quick Access** toolbar, click **Save**.



- In the **Save As** dialog box, enter file name *FlangeManifold.ipt*

- Click **Save**