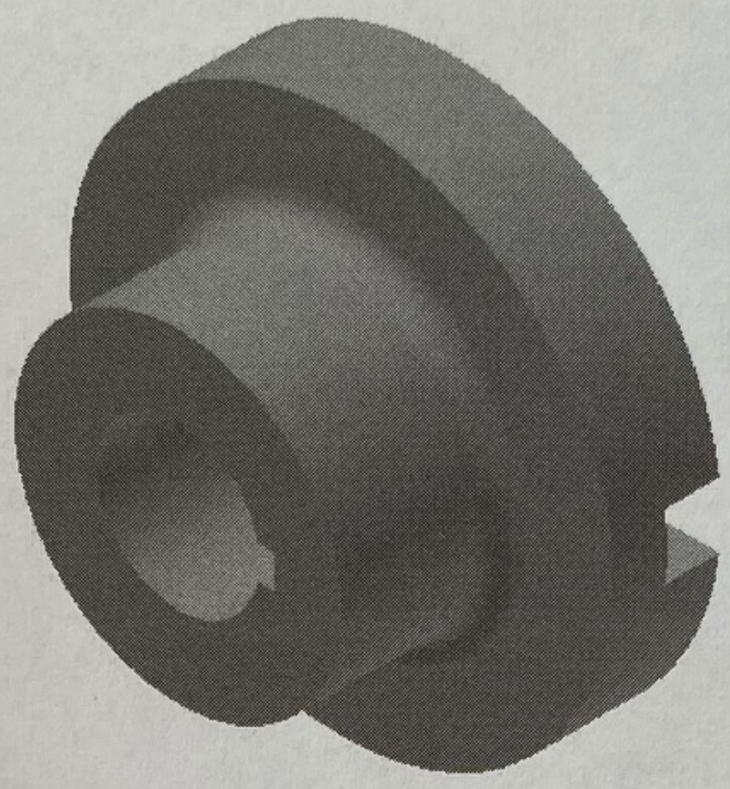
TUTORIAL 2

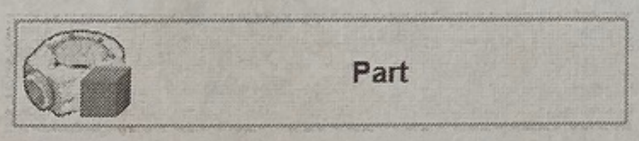
In this tutorial, you create a flange by performing the following:

- Creating a revolved feature
- Creating cut features
- Adding fillets



Starting a New Part File

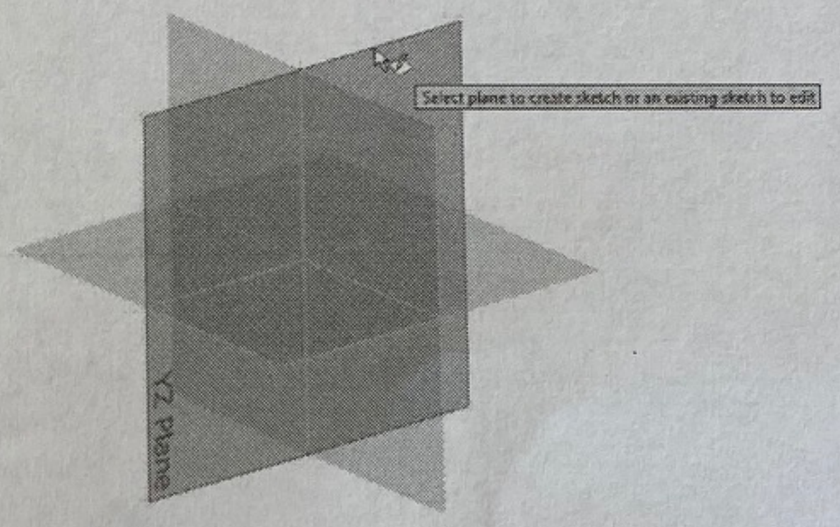
1. To start a new part file, click the **Part** icon on the Home screen.



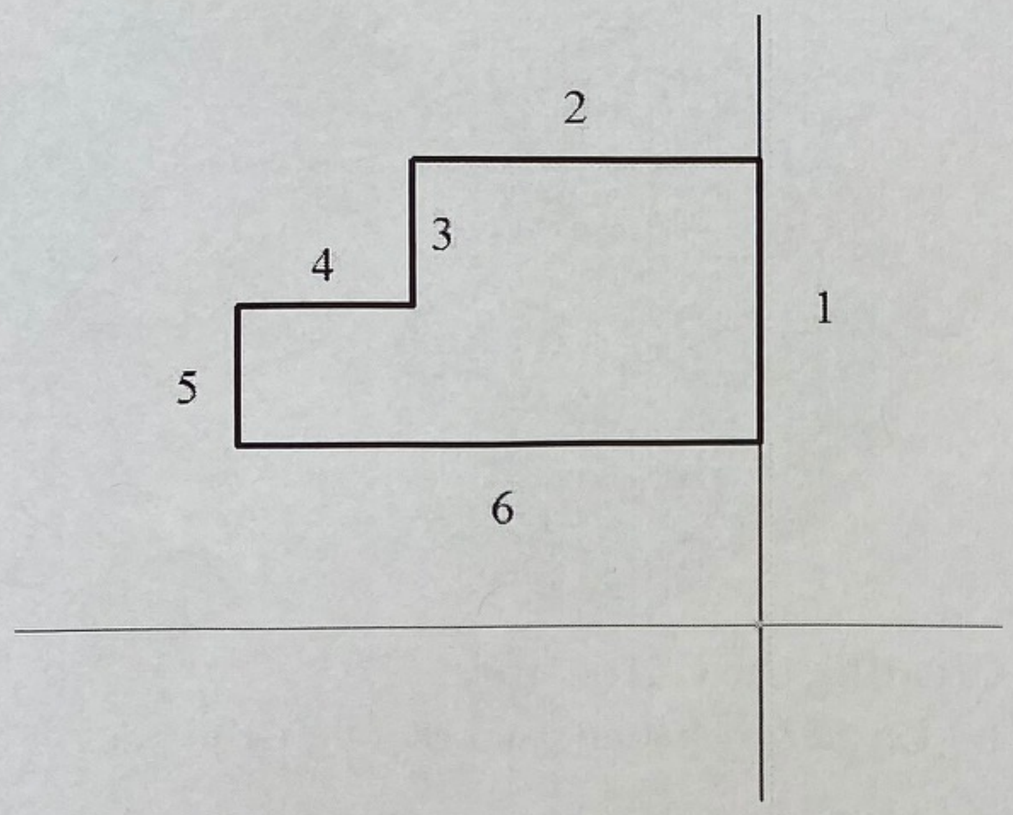
Sketching a Revolve Profile

You create the base feature of the flange by revolving a profile around a centerline.

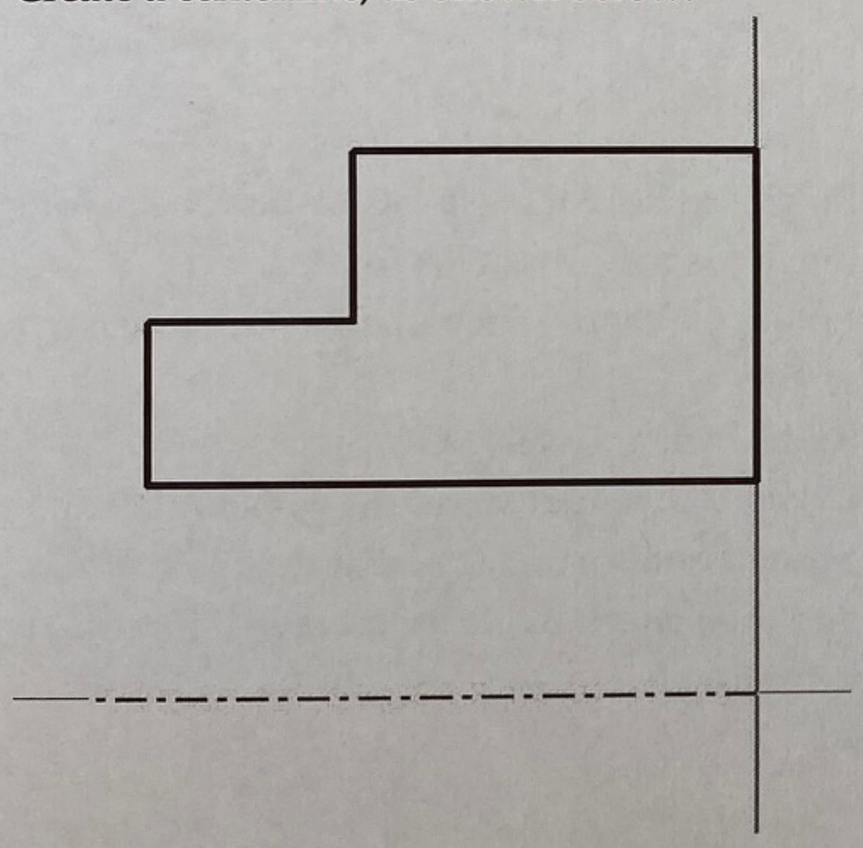
1. Click **3D Model > Sketch > Start 2D Sketch** on the ribbon.
2. Select the YZ plane.



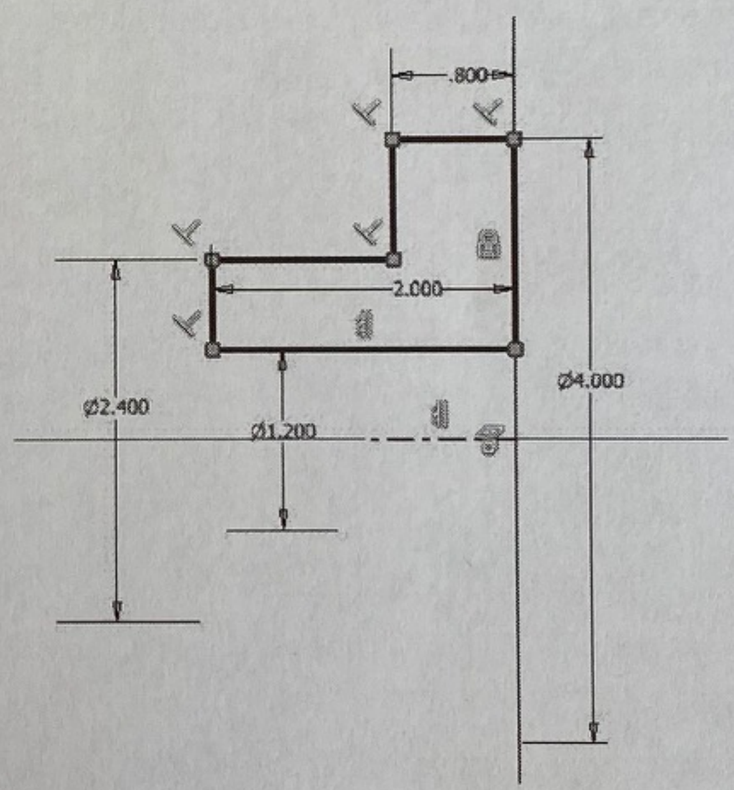
3. Click **Line** on the **Create** panel.
4. Create a sketch similar to that shown in the figure.



5. On the ribbon, click **Sketch > Format > Centerline** .
6. Click **Line** on the **Create** panel.
7. Create a centerline, as shown below.

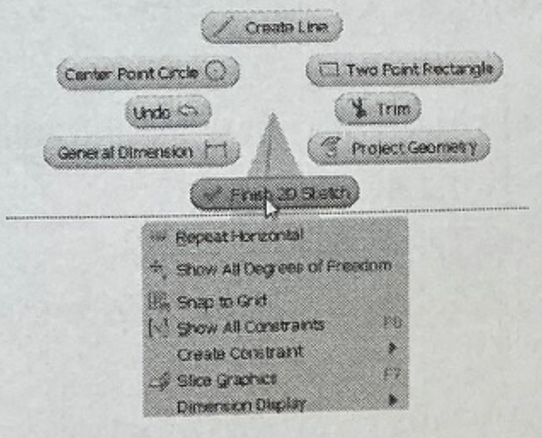


8. Click **Fix** on the **Constrain** panel.
9. Select the Line 1.
10. Click **Dimension** on the **Constrain** panel.
11. Select the centerline and Line 2; a dimension appears.
12. Move the pointer horizontally toward the right and click to place the dimension.
13. Place the dimension and enter **4** in the **Edit Dimension** box.
14. Click the green check on the **Edit Dimension** dialog.
15. Select the centerline and Line 4; a dimension appears.
16. Move the pointer horizontally toward left and click to place the dimension.
17. Enter **2.4** in the **Edit Dimension** box.
18. Click the green check on the **Edit Dimension** dialog.
19. Select the centerline and Line 6; a dimension appears.
20. Move the pointer horizontally toward left and click to place the dimension.
21. Enter **1.2** in the **Edit Dimension** box.
22. Click the green check on the **Edit Dimension** dialog.
23. Create a dimension between Line 1 and Line 3.
24. Set the dimension value to 0.8 inches.
25. Create a dimension between Line 1 and Line 5.
26. Set the dimension value to 2 inches.



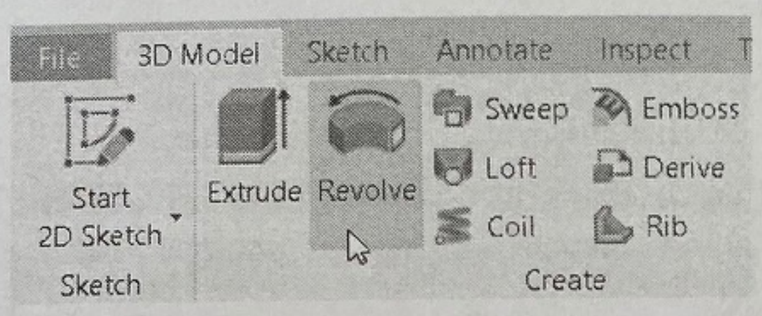
*You can display all the constraints by right-clicking and selecting **Show All Constraints** option. You can hide all the constraints by right-clicking and selecting the **Hide All Constraints** option.*

27. Right-click and select **Finish 2D Sketch**.

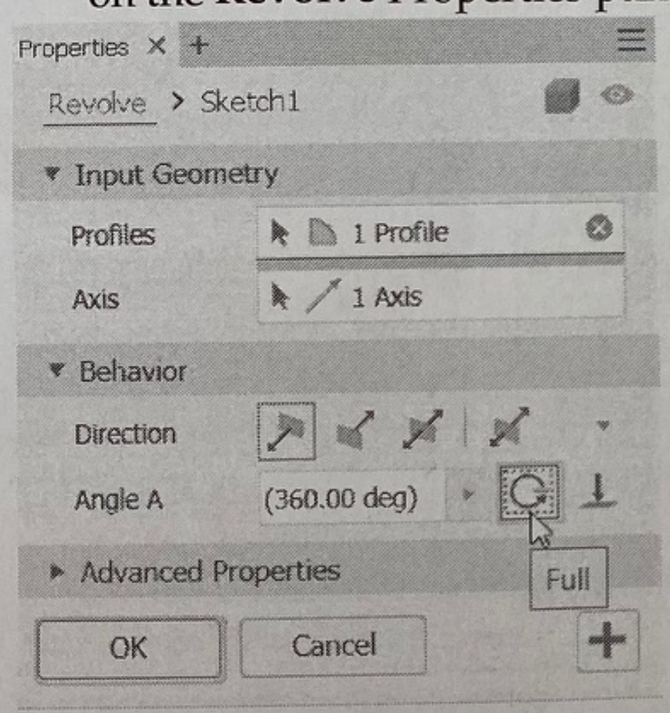


Creating the Revolved Feature

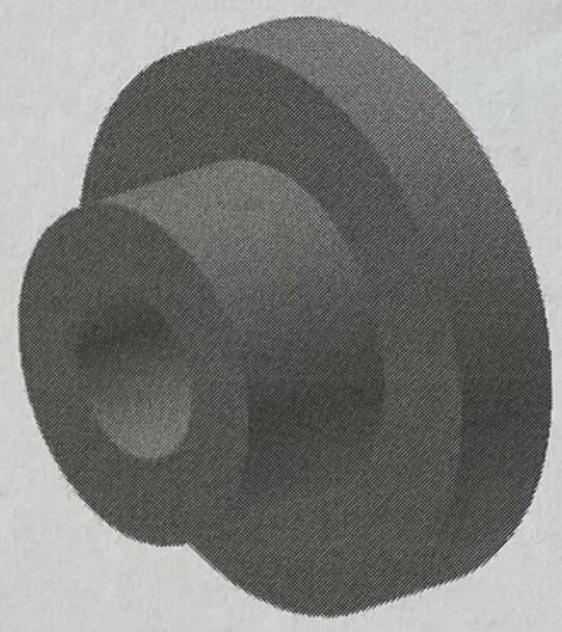
1. On the ribbon, click **3D Model > Create > Revolve** (or) right-click and select **Revolve** from the Marking menu.



2. Click the **Full** button under the **Behavior** section on the **Revolve** Properties panel.

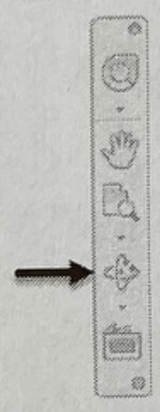


3. Click **OK** to create the revolved feature.

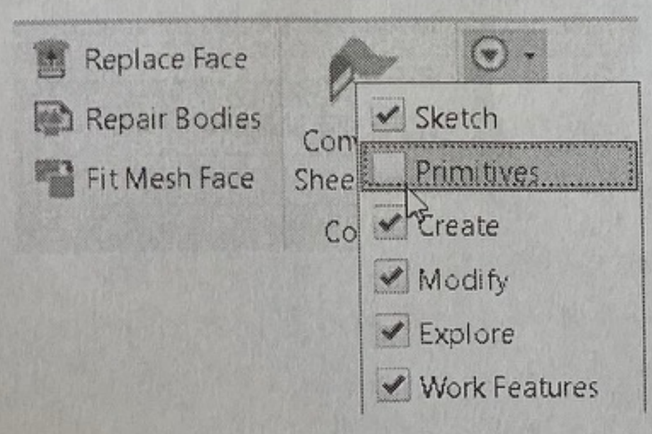


Creating the Cut feature

1. On the **Navigation** pane, click the **Orbit** icon.

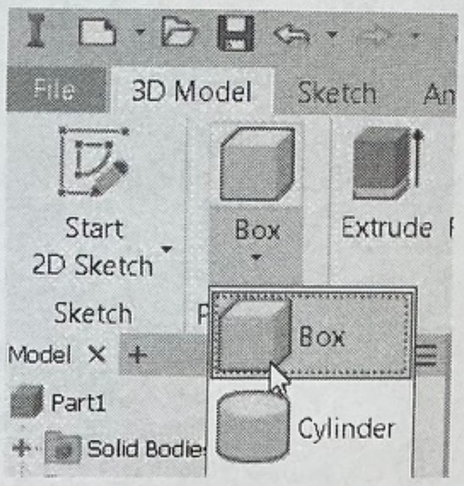


2. Press and hold the left mouse button and drag the mouse; the model is rotated.
3. Rotate the model such that its back face is visible.
4. Right click and select **OK**.
5. On the **3D Model** tab of the ribbon, click the **Show Panels** icon located at the right corner, and then check the **Primitives** option from the drop-down.

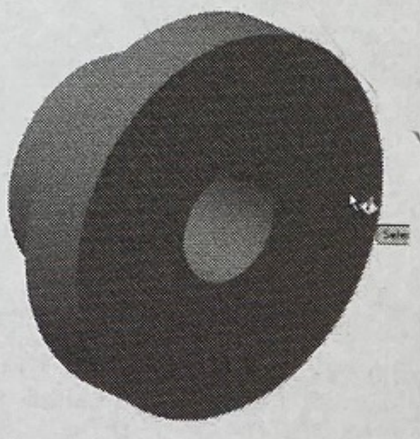


The **Primitives** panel is added to the ribbon.

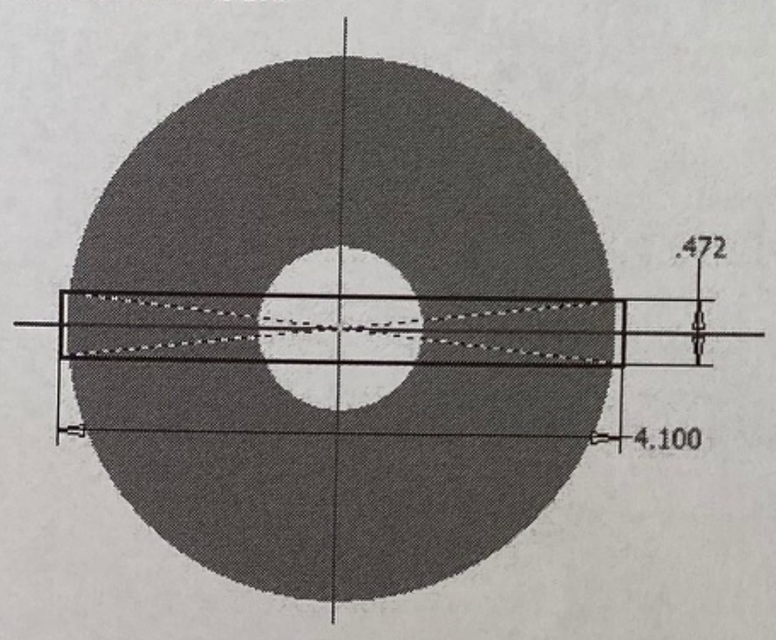
6. On the ribbon, click **3D Model > Primitives > Primitive drop-down > Box** on the **Primitives** panel.



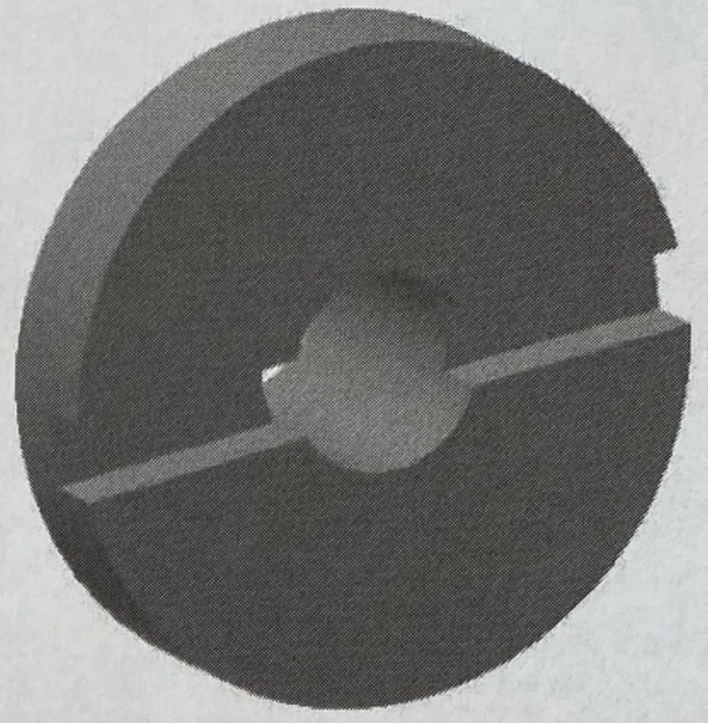
7. Click the back face of the part; the sketch starts.



8. Select the origin to define the center point.
9. Move the cursor diagonally toward the right.
10. Enter 4.1 in the horizontal dimension box.
11. Press Tab key and enter 0.472 in the vertical dimension box.

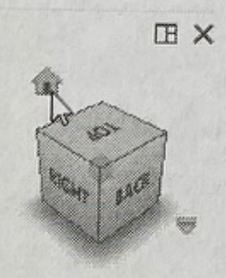


12. Press the Enter key; the **Extrude** Properties panel appears.
13. Expand the **Output** section on the **Extrude** Properties panel by clicking the **Output**.
14. Click the **Cut** icon under the **Output** section on the **Extrude** Properties panel.
15. Enter 0.4 in the **Distance A** box.
16. Click the **Direction > Flipped** icon under the **Behavior** section to reverse the direction.
17. Click **OK** to create the cut feature.



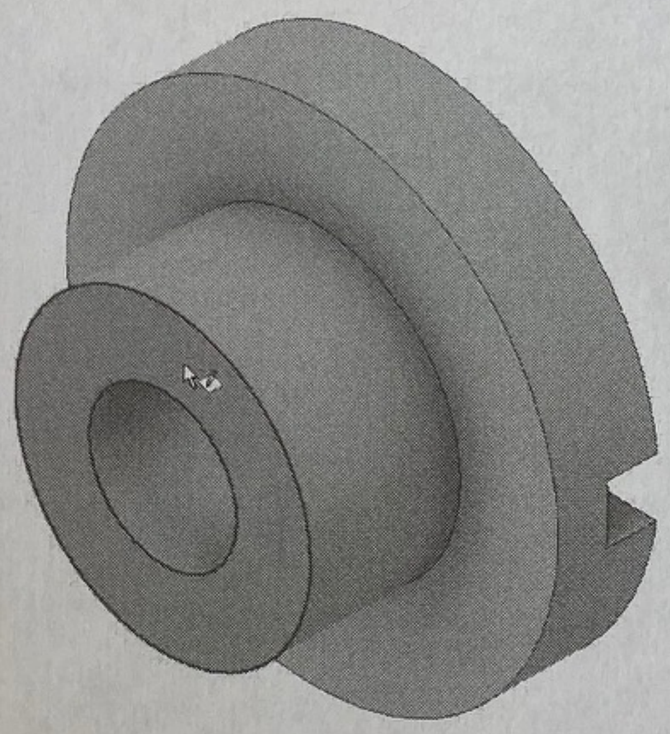
Creating another Cut feature

1. Click the **Home** icon located at the top left corner of the **ViewCube**.

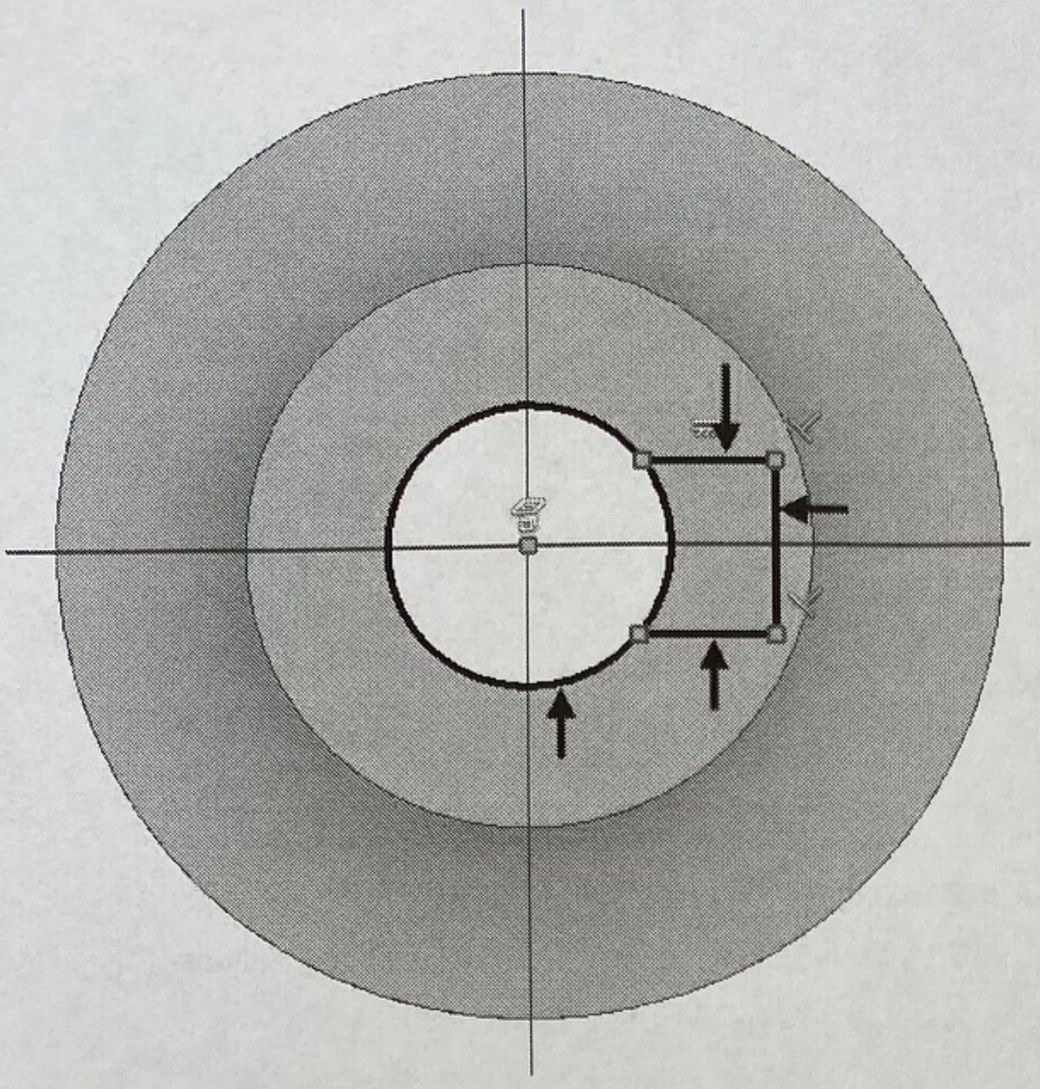


2. Create a sketch on the front face of the base feature.

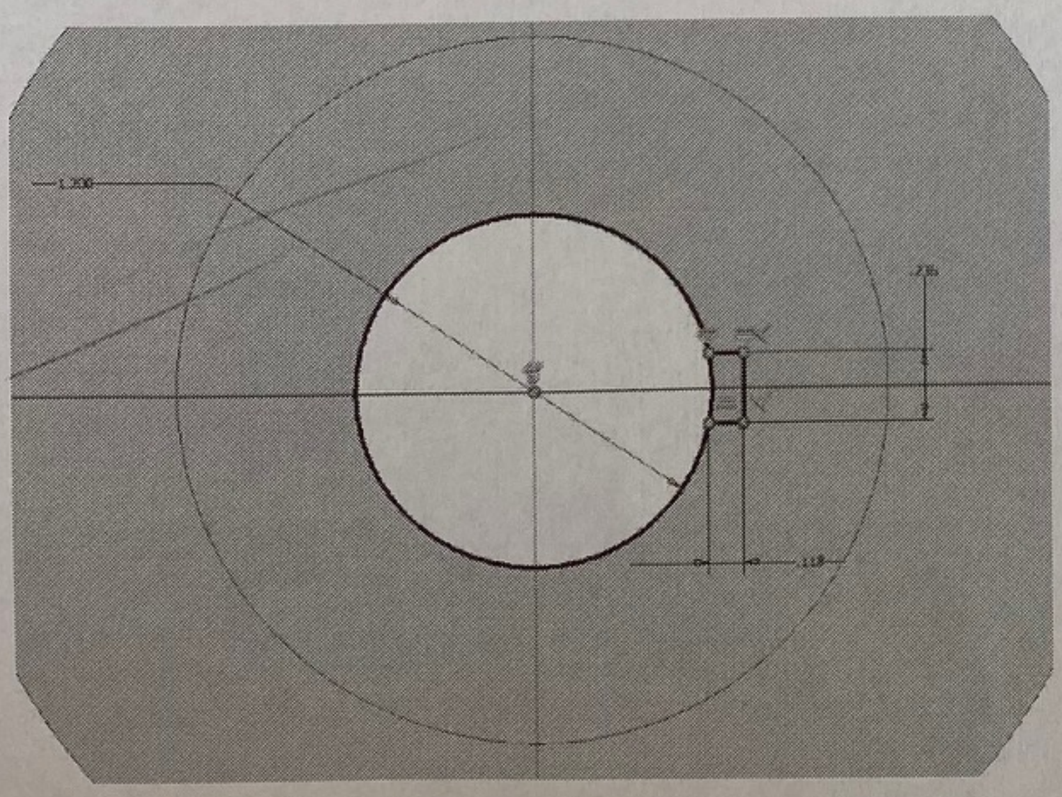
- On the ribbon, click **3D Model > Sketch > Start 2D Sketch**.
- Select the front face of the model.



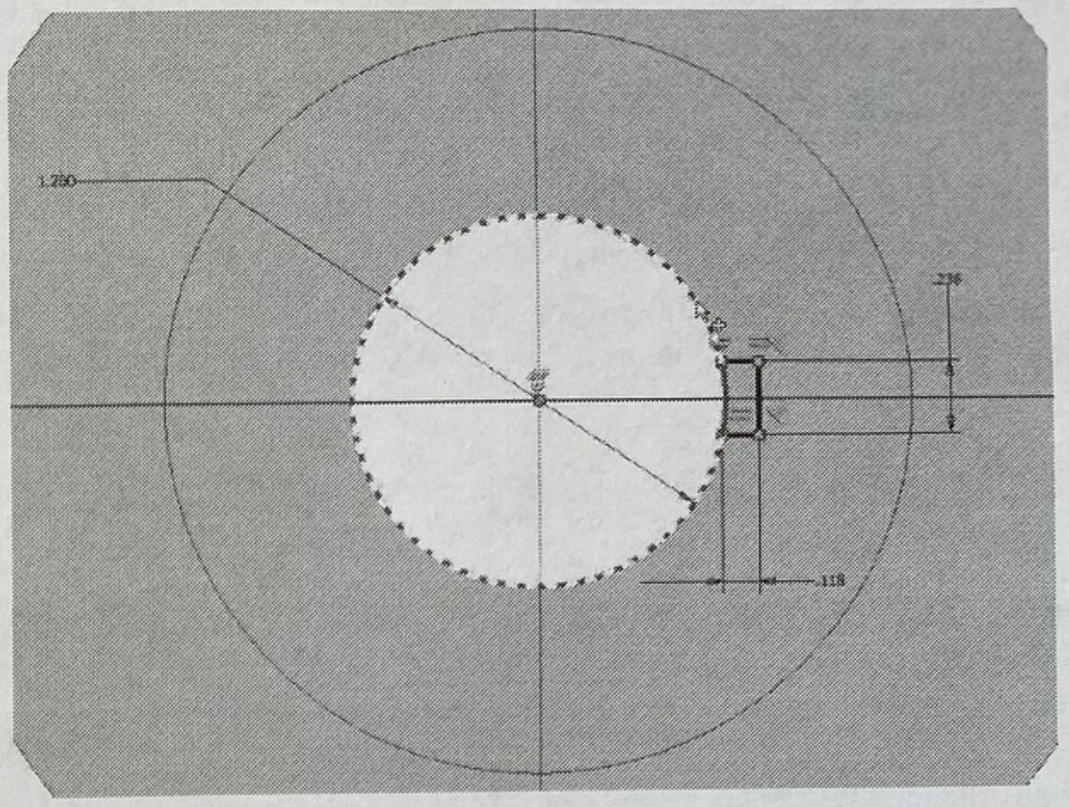
- Draw three lines and the circle, as shown in the figure.



- Apply the **Horizontal** constraint to the horizontal lines, if not applied already.
- Apply the **Equal** constraint between the horizontal lines.
- Ensure that the endpoints of the horizontal line coincide with the circle.
- Apply dimension of 0.236 to the vertical line.
- Apply dimension of 0.118 to the horizontal line.
- Apply the dimension of the 1.2-inch diameter to the circle.

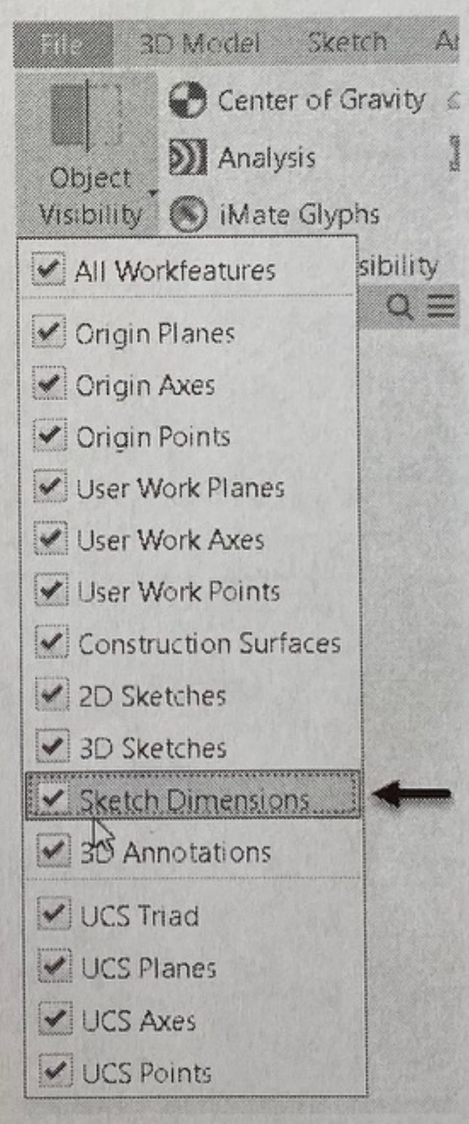


- On the ribbon, click **Sketch > Modify > Trim**.
- Click on the circle to trim it.



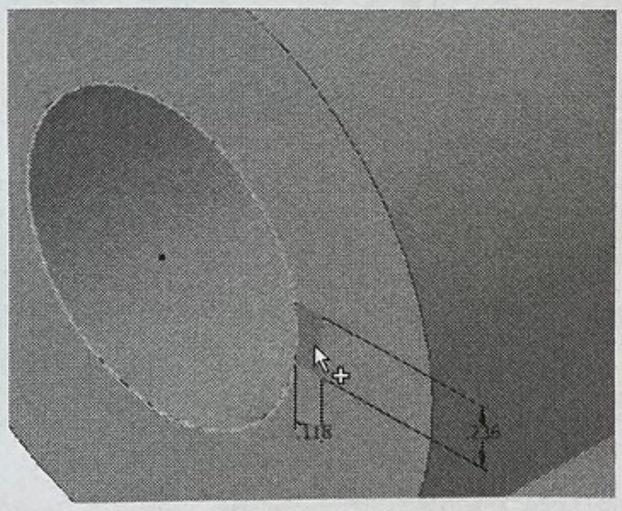
3. Finish the sketch.

You can hide or display the sketch dimensions. To do this, go to ***View > Visibility > Object Visibility*** *and check the* ***Sketch Dimensions*** *option.*

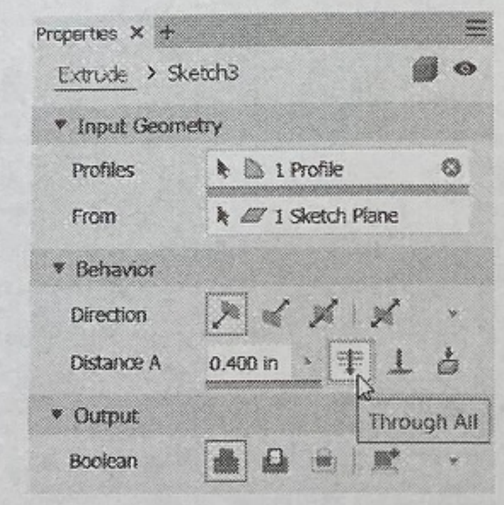


4. Click **Extrude** on the **Create** panel of the **3D Model**.

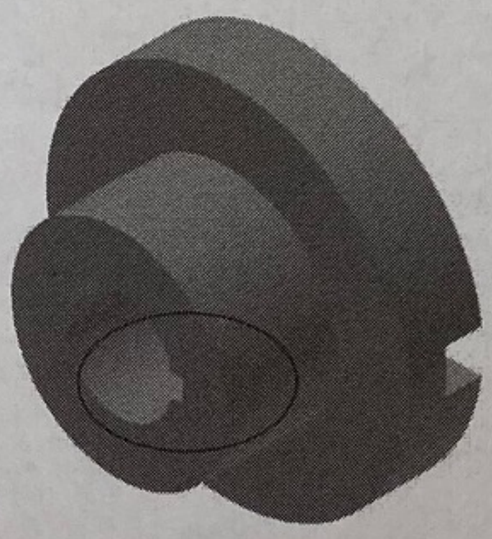
5. Click on the region enclosed by the three lines and the arc.



6. Click the **Through All** icon under the **Behavior** section.

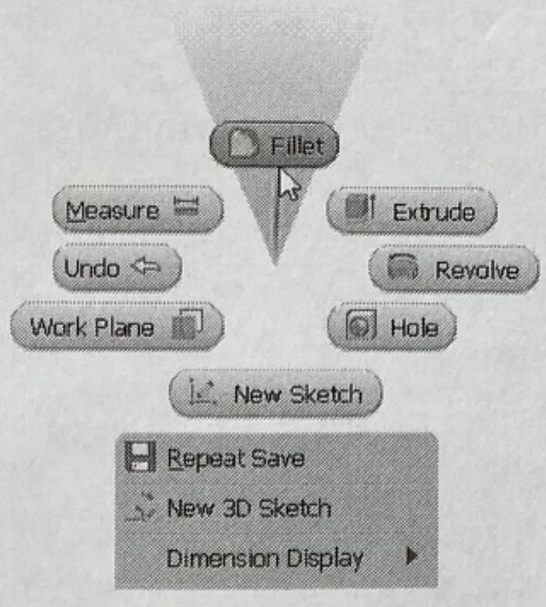


7. Click the **Cut** icon under the **Output** section on the **Extrude** Properties panel.
8. Click **OK** to create the cut feature.

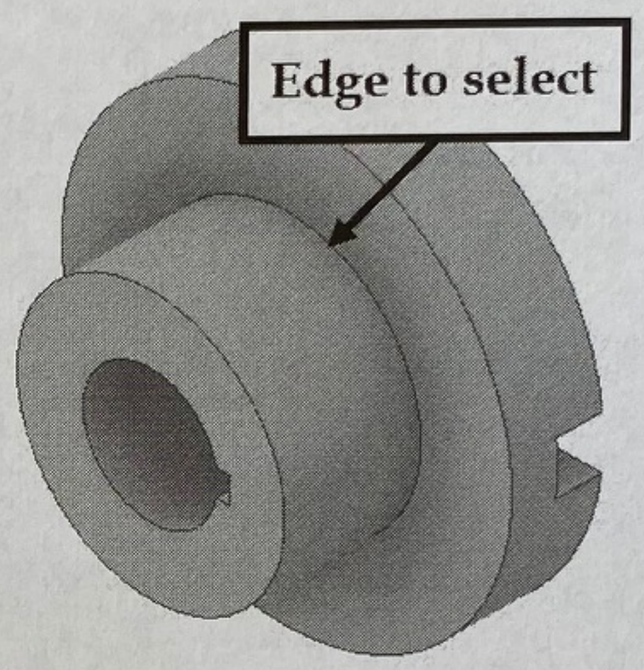


Adding a Fillet

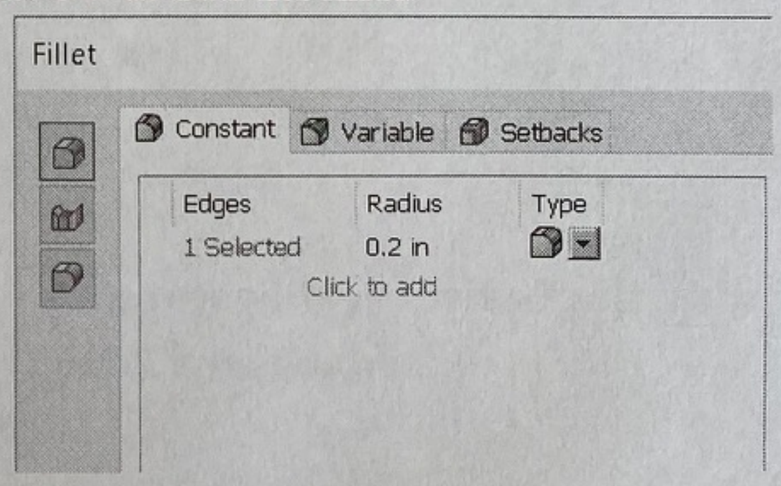
1. On the ribbon, click **3D Model > Modify > Fillet** (or) right-click and select **Fillet** from the Marking menu.



2. Click on the inner circular edge.



3. On the **Fillet** dialog, click the **Constant** tab, and then type 0.2 in the **Radius** box.
4. Click **OK** to add the fillet.



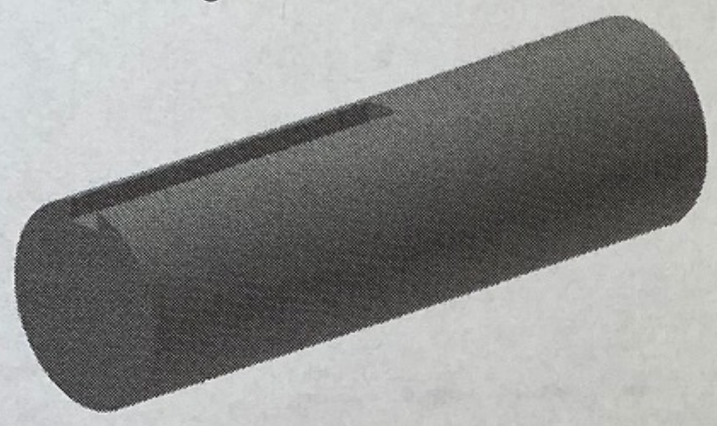
Saving the Part

1. Click **Save** on the **Quick Access Toolbar**.
2. On the **Save As** dialog, type-in **Flange** in the **File name** box.
3. Click **Save** to save the file.
4. Click **File Menu > Close**.

TUTORIAL 3

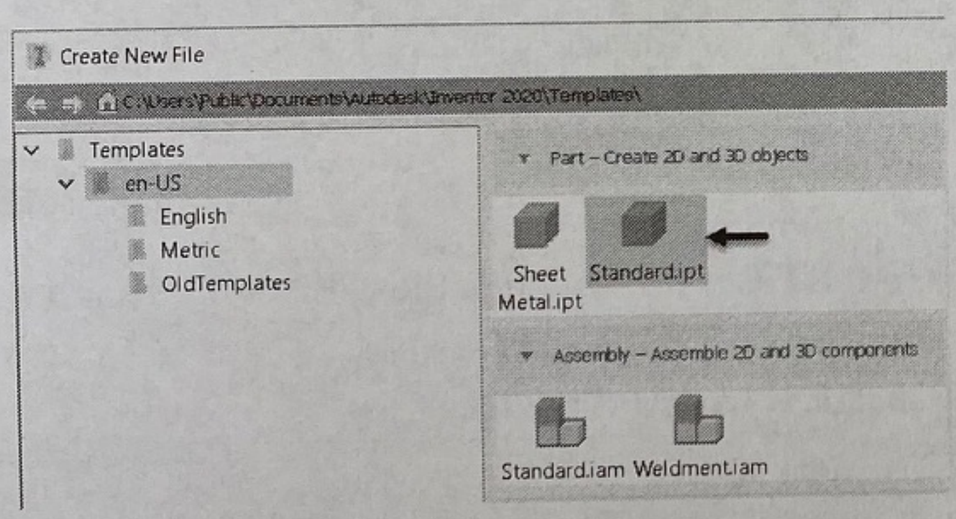
In this tutorial, you create the Shaft by performing the following:

- Creating a cylindrical feature
- Creating a cut feature



Starting a New Part File

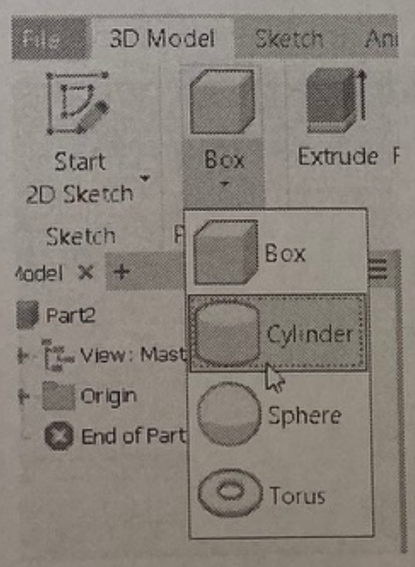
1. On the ribbon, click **Get Started > Launch > New**.
2. On the **Create New File** dialog, select **Standard.ipt**.



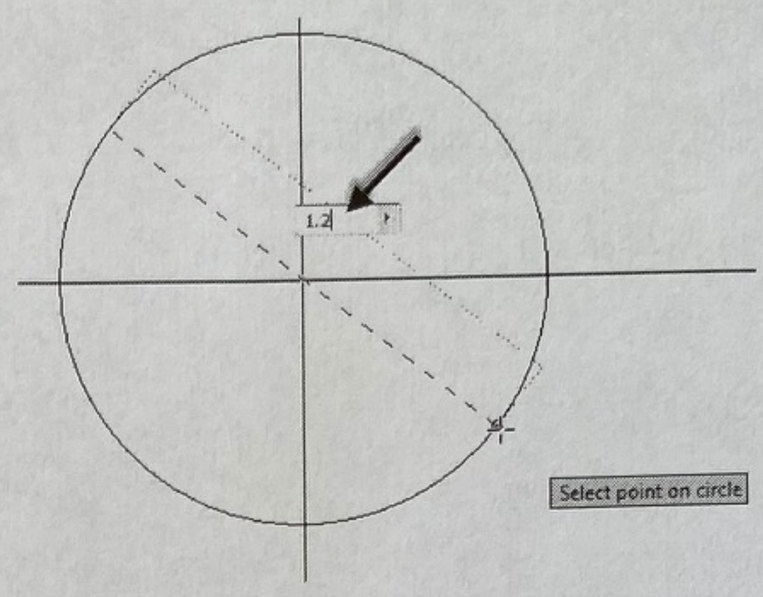
3. Click **Create**.

Creating the Cylindrical Feature

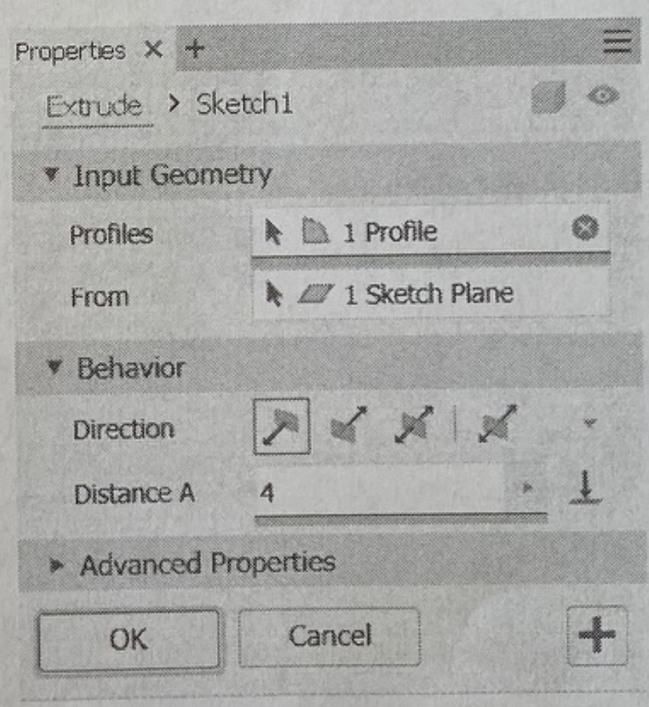
1. On the ribbon, click **Primitives > Primitive** drop-down **> Cylinder**.



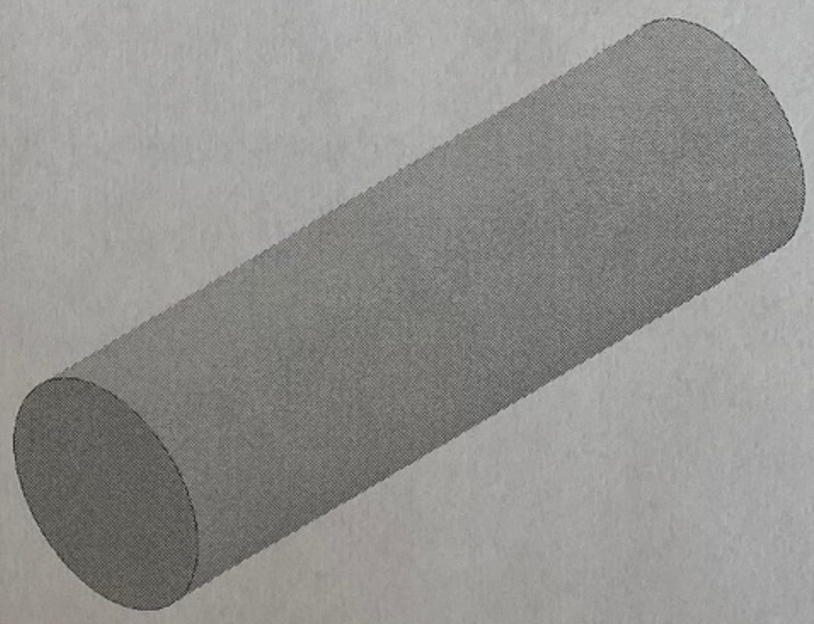
2. Click on the **XY** plane to select it; the sketch starts.
3. Click at the origin and move the cursor outward.
4. Enter 1.2 in the box attached to the circle.



5. Press Enter key; the **Extrude** Properties panel appears.
6. Enter **4** in the **Distance A** box.

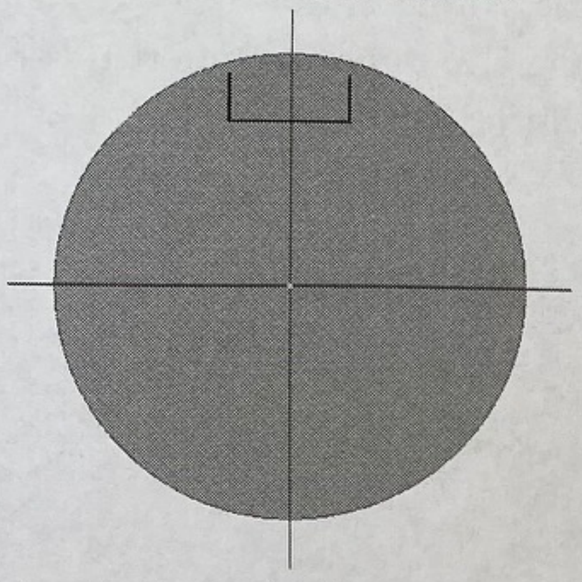


7. Click **OK** to create the cylinder.

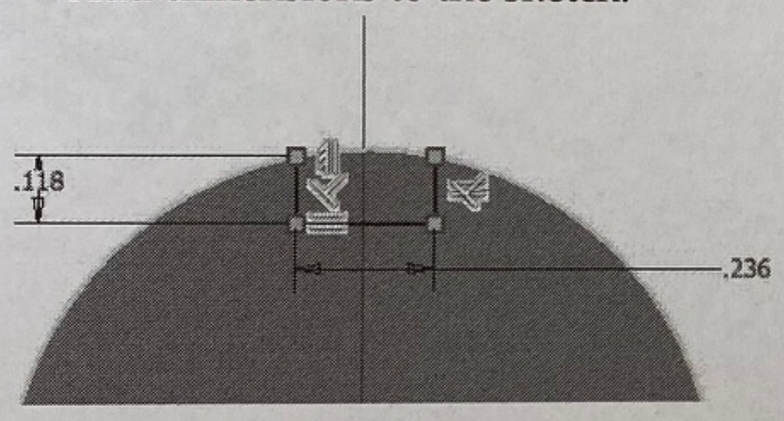


Creating Cut feature

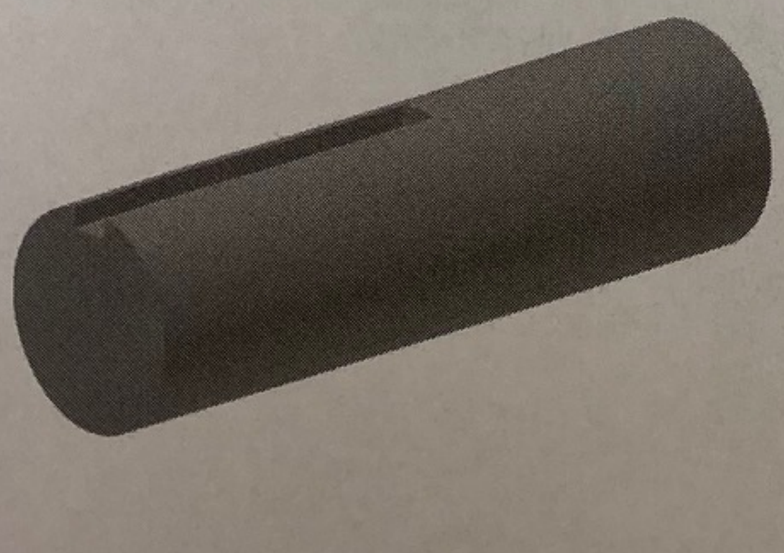
1. Create a sketch on the front face of the base feature.
   - On the ribbon, click **3D Model > Sketch > Start 2D Sketch**.
   - Select the front face of the cylinder.
   - On the ribbon, click **Sketch > Create > Line**.
   - Draw three lines, as shown.



   - Apply the **Coincident** constraint between the endpoints of the vertical lines and the circular edge.
   - Add dimensions to the sketch.



2. Finish the sketch.
3. Click **Extrude** on the **Create** panel.
4. Click on the region enclosed by the sketch.
5. Click the **Cut** icon on the **Extrude** Properties panel.
6. Set **Distance A** to **2.165**.
7. Click **OK** to create the cut feature.



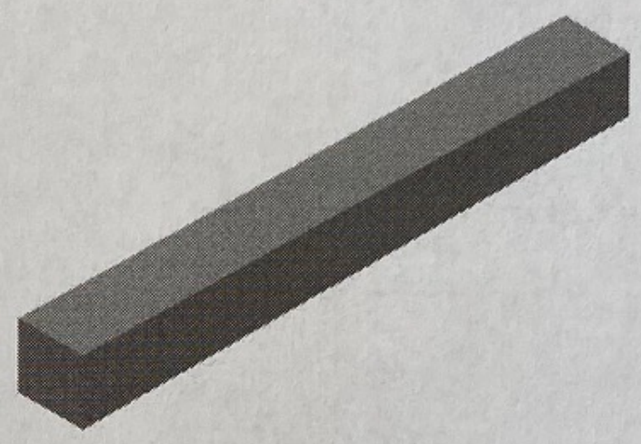
Saving the Part

1. Click **Save** on the **Quick Access Toolbar**; the **Save As** dialog appears.
2. Type-in **Shaft** in the **File name** box.
3. Click **Save** to save the file.
4. Click **File Menu > Close**.

TUTORIAL 4

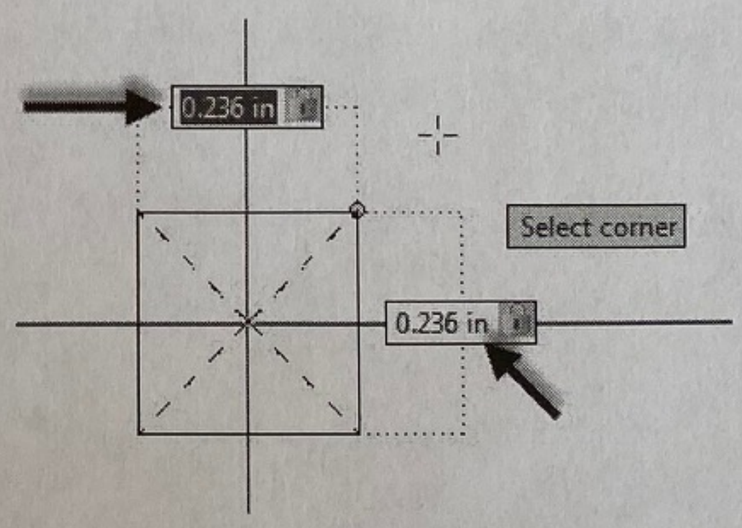
In this tutorial, you create a Key by performing the following:

- Creating an Extruded feature
- Applying draft

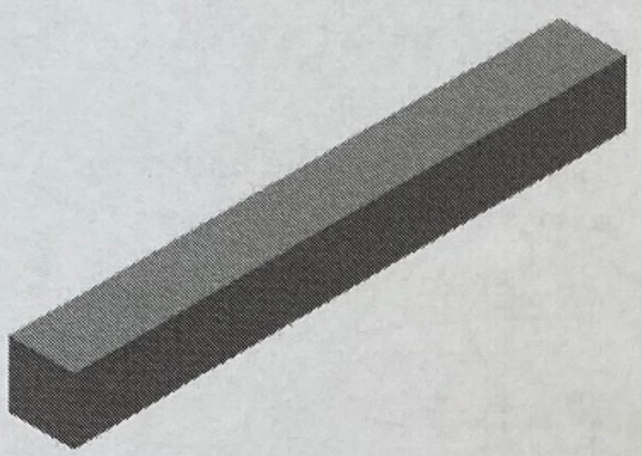


Start Extruded feature

1. Start a new part file using the **Standard.ipt** template.
2. On the ribbon, **Primitives > Primitive** drop-down > **Box**.
3. Select the XY plane.
4. Create the sketch, as shown in the figure.

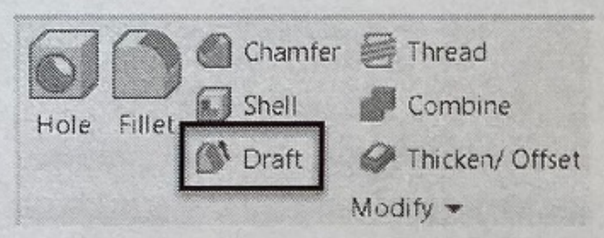


5. Press ENTER.
6. Enter 2 in the **Distance A** box.
7. Click **OK** to create the extrusion.

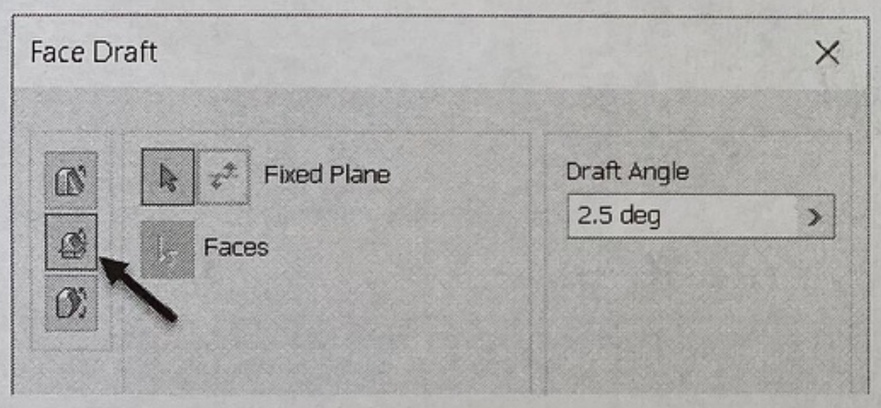


Applying Draft

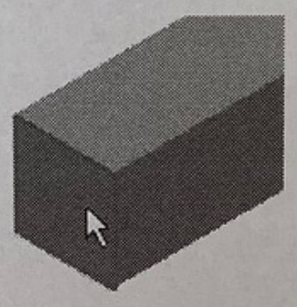
1. On the ribbon, click **3D Model > Modify > Draft**.



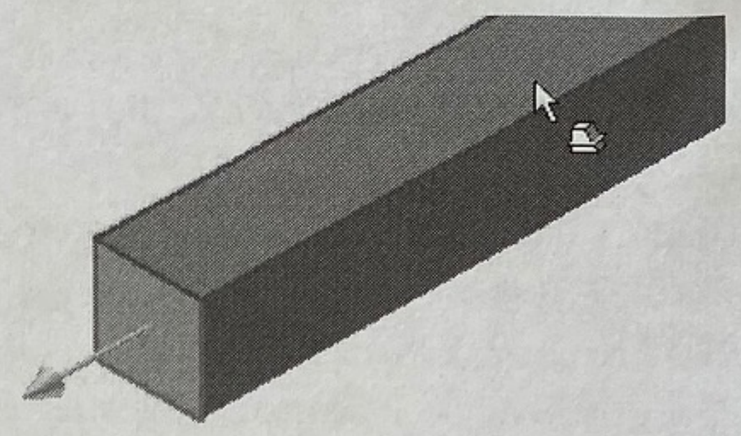
2. Select the **Fixed Plane** option.



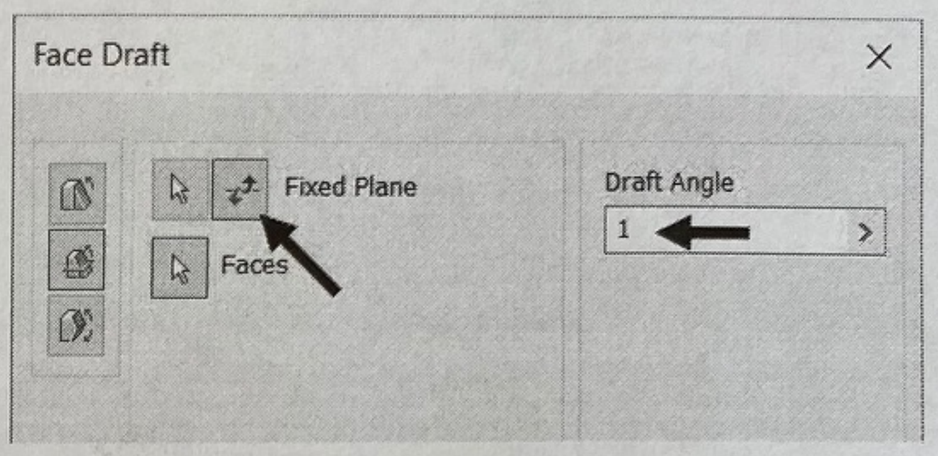
3. Select the front face as the fixed face.

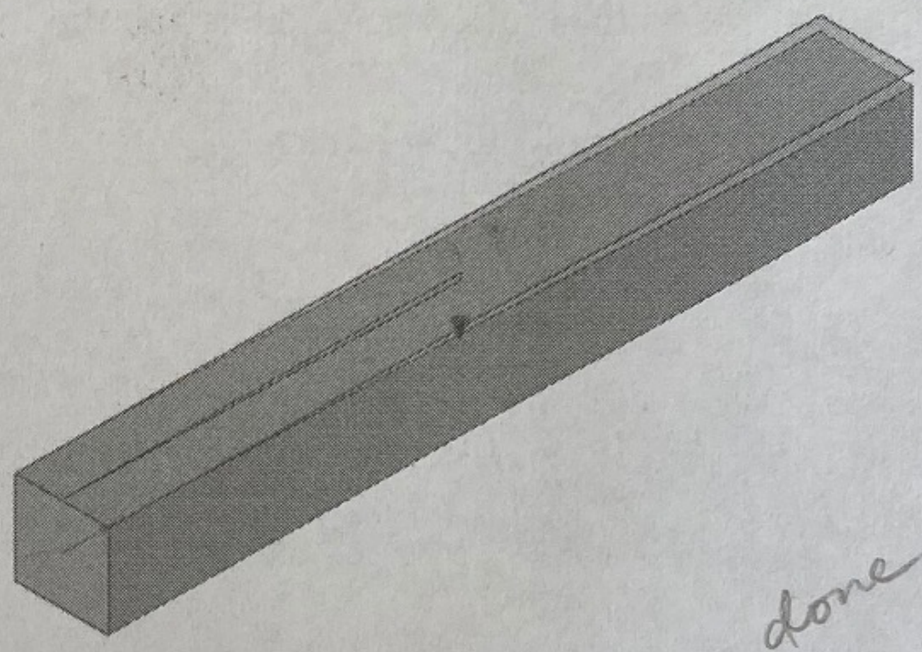


4. Select the top face as the face to be draft.



5. Set **Draft Angle** to **1**.
6. Click the **Flip pull direction** button on the **Face Draft** dialog.





7. Click **OK** to create the draft.

Saving the Part

1. Click **Save** on the **Quick Access Toolbar**; the **Save As** dialog appears.
2. Type-in **Key** in the **File name** box.
3. Click **Save** to save the file.
4. Click **File Menu > Close**.