

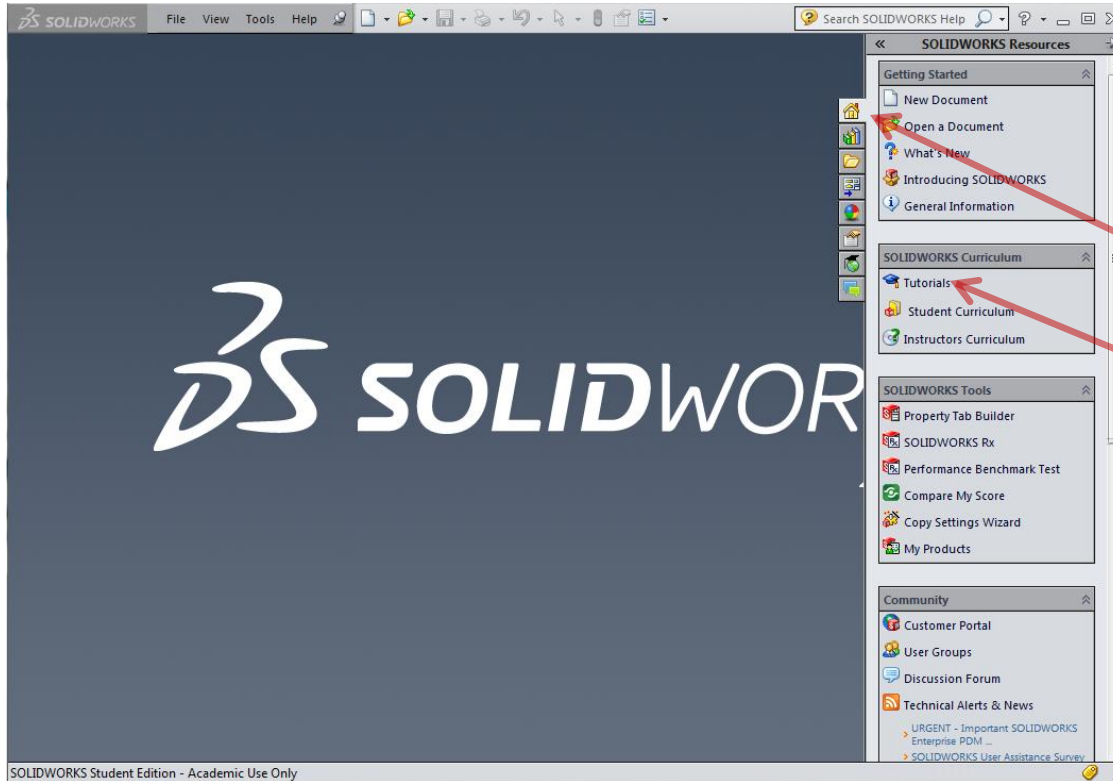
SolidWorks Part 1

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Introductory Medical Device Prototyping

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<http://saliterman.umn.edu/>

Starting The Tutorials



- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

Select Getting Started Exercises

SOLIDWORKS Tutorials: Getting Started

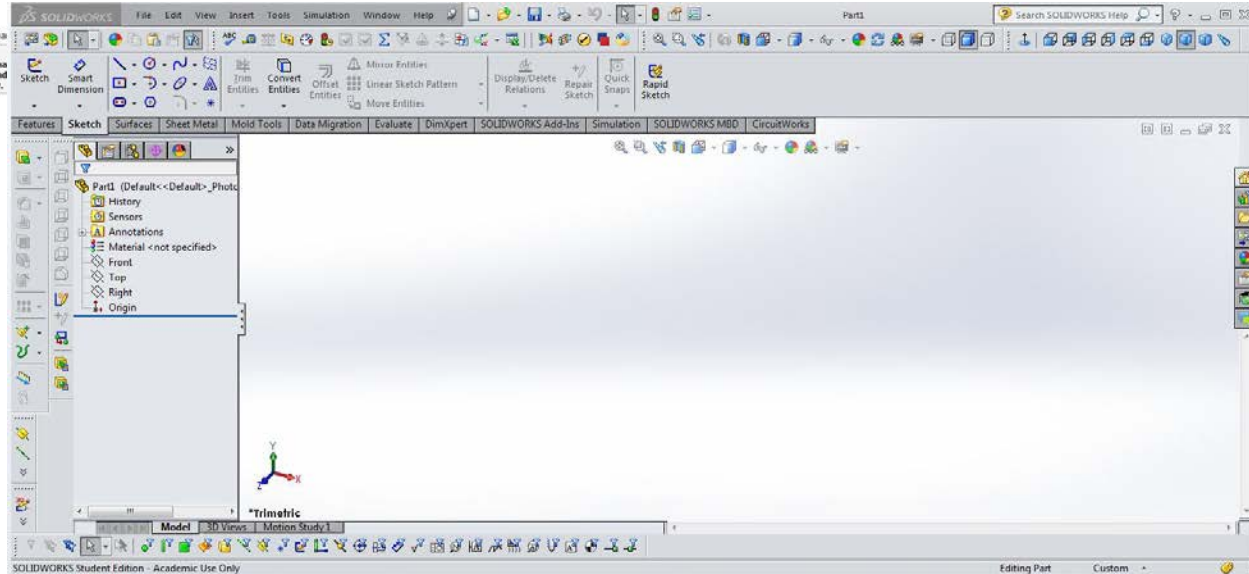
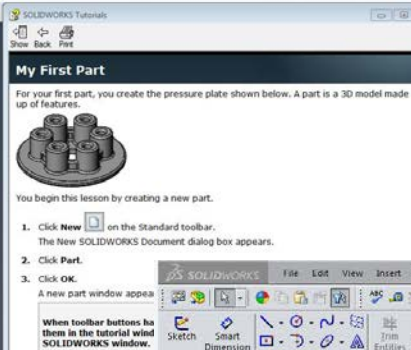
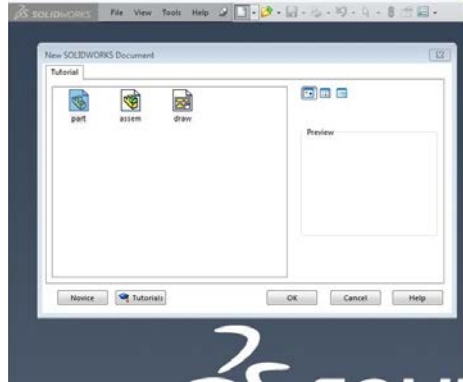
Getting Started	Basic Techniques	Advanced Techniques
Productivity Tools	Design Evaluation	CSWP/CSWA Preparation
What's New Examples	All SOLIDWORKS Tutorials	Go to SOLIDWORKS Simulation Tutorials

These tutorials present SOLIDWORKS functionality in an example-based learning format. For details about typographical conventions and how to navigate through these tutorials, see [Conventions](#).

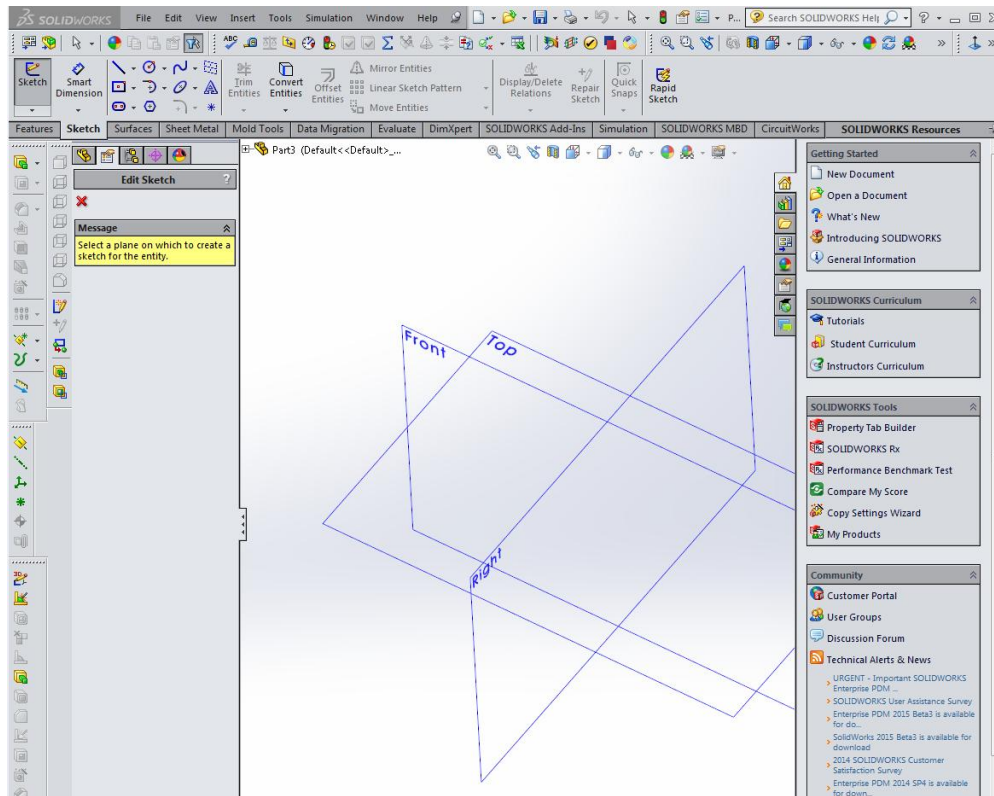
If you are new to the SOLIDWORKS software, familiarize yourself with the tutorials in **Getting Started** first. For examples of What's New in SOLIDWORKS for this release, see **What's New Examples**. All other tutorials can be completed in any order.

Introduction to SOLIDWORKS 	AutoCAD and SOLIDWORKS 	Lesson 1: Parts
Lesson 2: Assemblies 	Lesson 3: Drawings 	SOLIDWORKS Workgroup PDM

Choose “Intro to Solid Works”






Select the Top Plane for Sketch

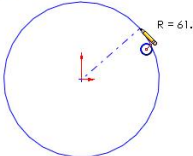


Sketching the Circle

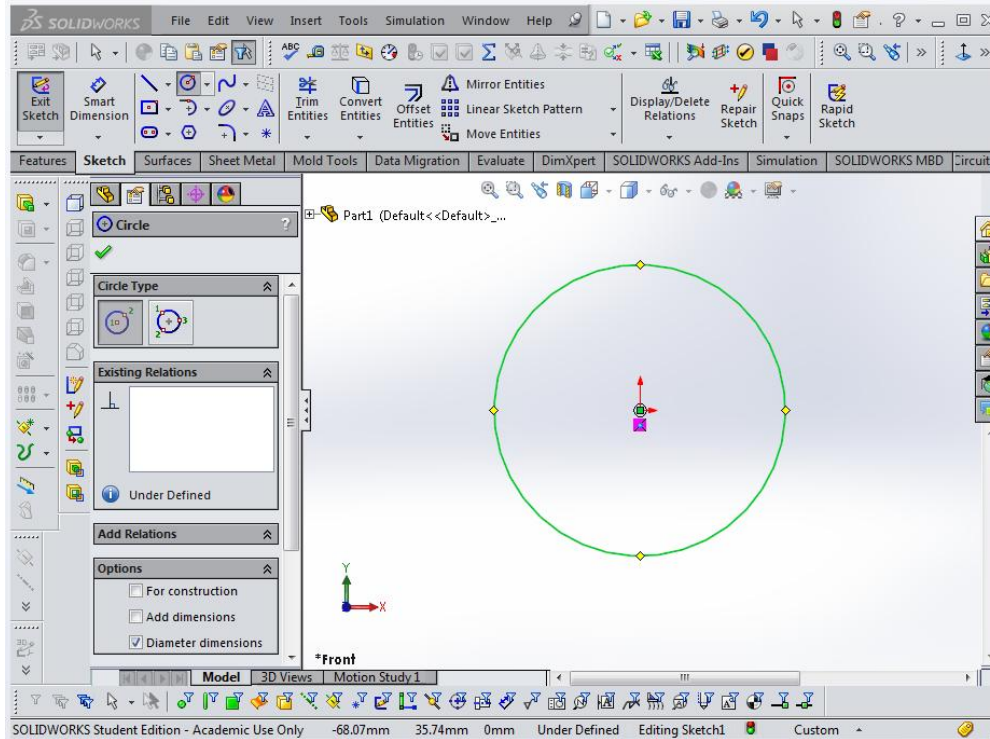
The first feature in the part is a cylinder extruded from a sketched circular profile.

1. Click **Extruded Boss/Base** on the Features toolbar.
The **Front**, **Top**, and **Right** planes appear in the graphics area.
2. Move the pointer over the **Top** plane to highlight it, then click to select it.
The display changes so that the **Top** plane is facing you. A sketch opens on the **Top** plane.
3. Click **Circle** on the Sketch toolbar.
The Circle PropertyManager opens in the left pane.
4. Move the pointer over the origin .
The pointer changes to . This indicates a coincident relation between the center of the circle and the origin.
5. Click to place the center point on the origin.
6. Move the mouse and notice a preview of the circle dynamically follows the pointer.
7. Click to finish the circle and click  in the PropertyManager.

The size of your sketch entities does not need to be exact. For example, this circle does not need to be 61.3mm. You later add dimensions to specify the size of sketch entities.







Sketch a Circle at the origin




Sketching the Circle

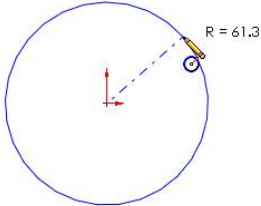
The first feature in the part is a cylinder extruded from a sketched circular profile.

1. Click **Extruded Boss/Base**  on the Features toolbar.
The **Front**, **Top**, and **Right** planes appear in the graphics area.
2. Move the pointer over the **Top** plane to highlight it, then click to select it.
The display changes so that the **Top** plane is facing you. A sketch opens on the **Top** plane.
3. Click **Circle**  on the Sketch toolbar.
The Circle PropertyManager opens in the left pane.
4. Move the pointer over the origin .

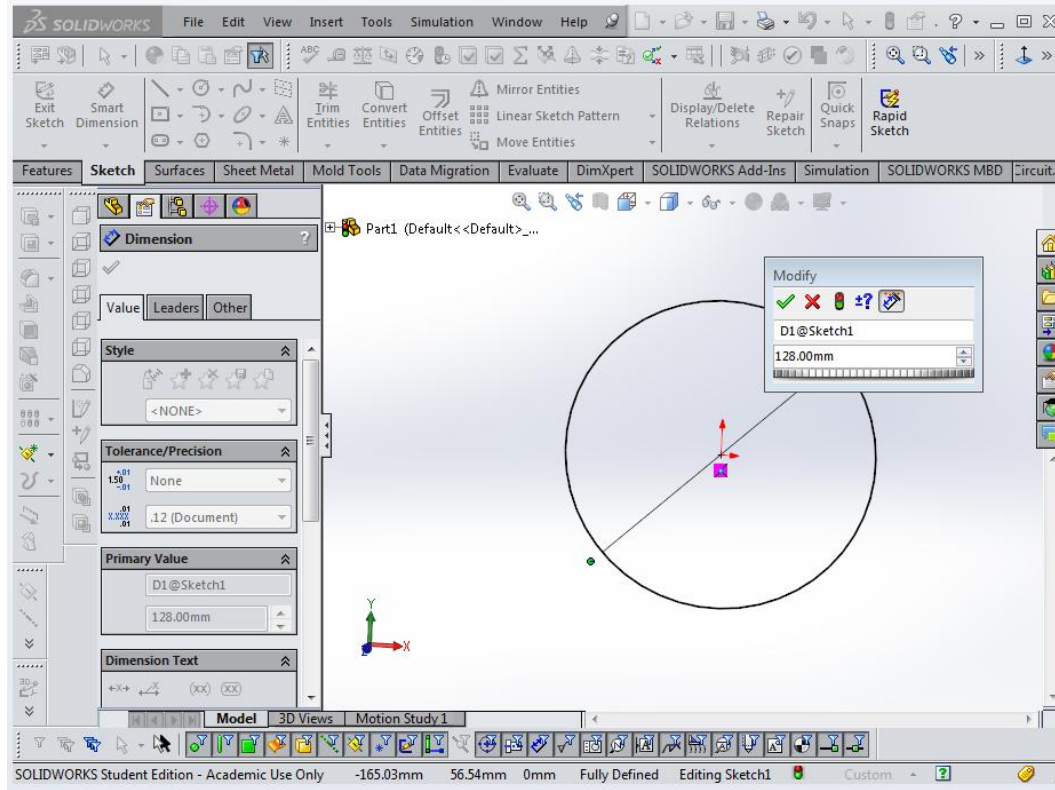
The pointer changes to . This indicates a coincident relation between the center of the circle and the origin.

5. Click to place the center point on the origin.
6. Move the mouse and notice a preview of the circle dynamically follows the pointer.
7. Click to finish the circle and click  in the PropertyManager.

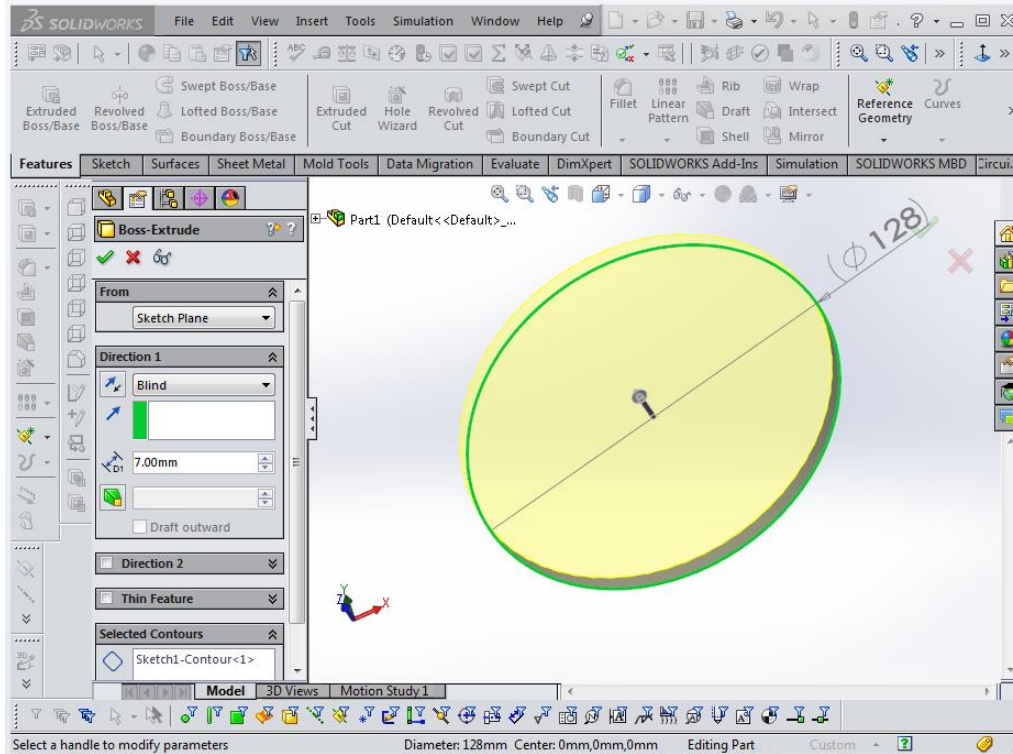
The size of your sketch entities does not need to be exact. For example, this circle does not need to be 61.3mm. You later add dimensions to specify the size of sketch entities.



Specify Dimension



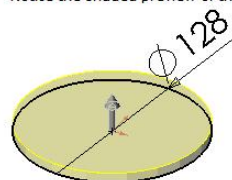
Boss-Extrude Feature



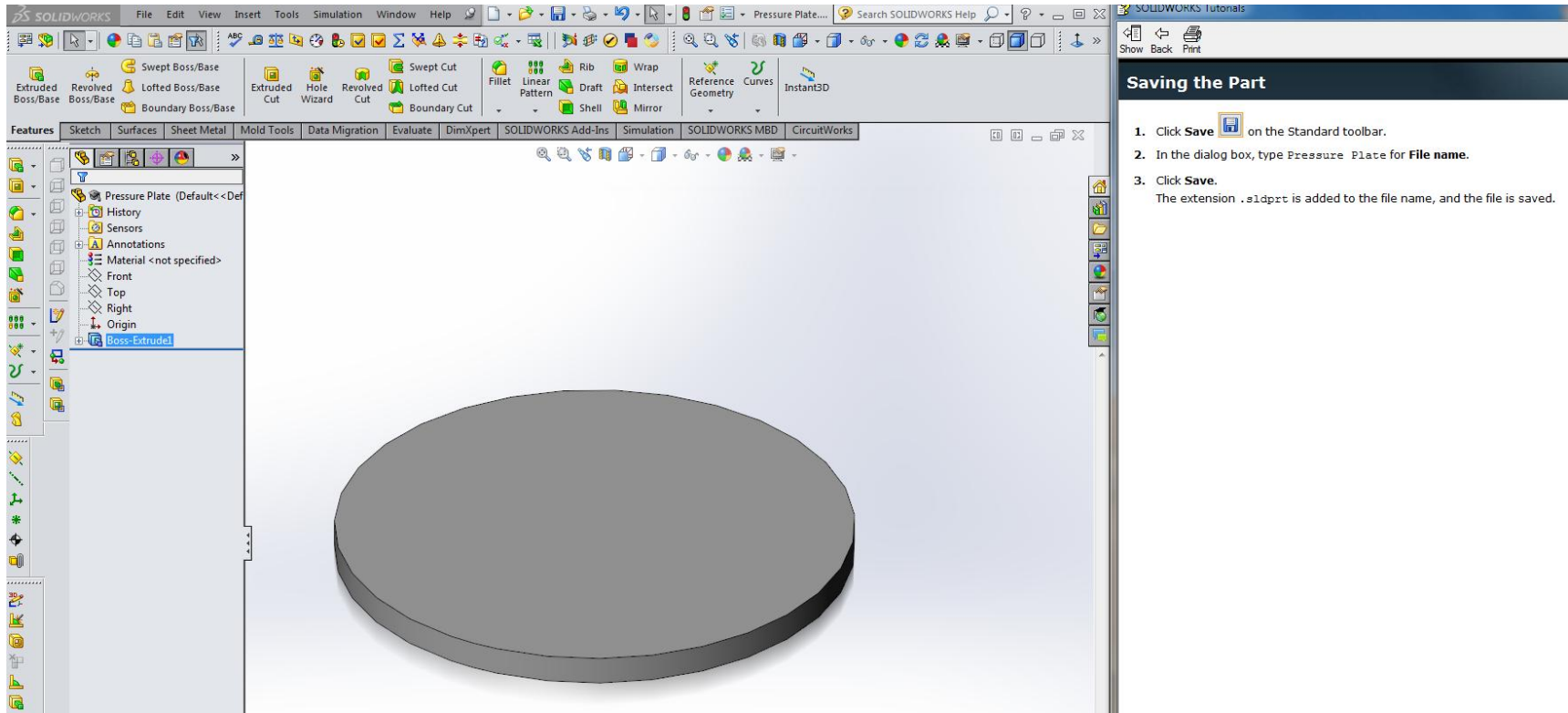
Extruding the Base Feature

Extrude the 2D sketch to create the 3D cylinder.

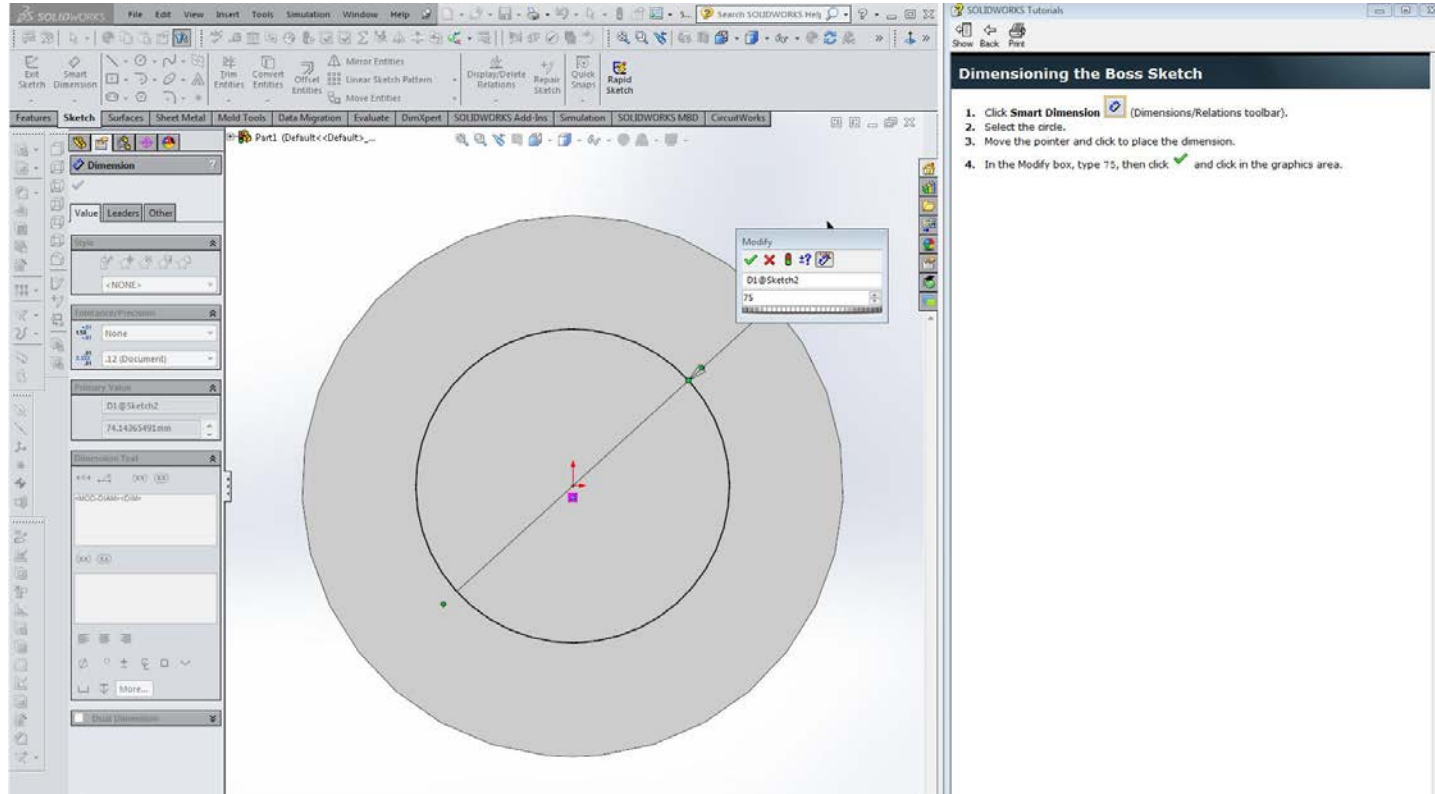
- Click **Exit Sketch** on the Sketch toolbar.
You exit the sketch when you are done with the 2D profile and are ready to create the 3D cylinder.
The settings for the extrusion appear in the PropertyManager in the left panel.
- In the PropertyManager, under **Direction 1**:
 - Select **Blind** in **End Condition**.
 - Set **Depth** to 7.
Notice the shaded preview of the extrusion.
- Click **✓**.
The first feature is complete. Boss-Extrude1 appears in the FeatureManager design tree in the left panel.



Extruded Plate Feature



Select Top and Sketch Circle



Offset Entities from Circle

SOLIDWORKS File Edit View Insert Tools Simulation Window Help

Part4 (Default<<Default>>....

Offset Entities

Parameters

7.00mm

Add dimensions

Reverse

Select chain

Bi-directional

Make base construction

Cap ends

Arcs

Lines

SOLIDWORKS Tutorials

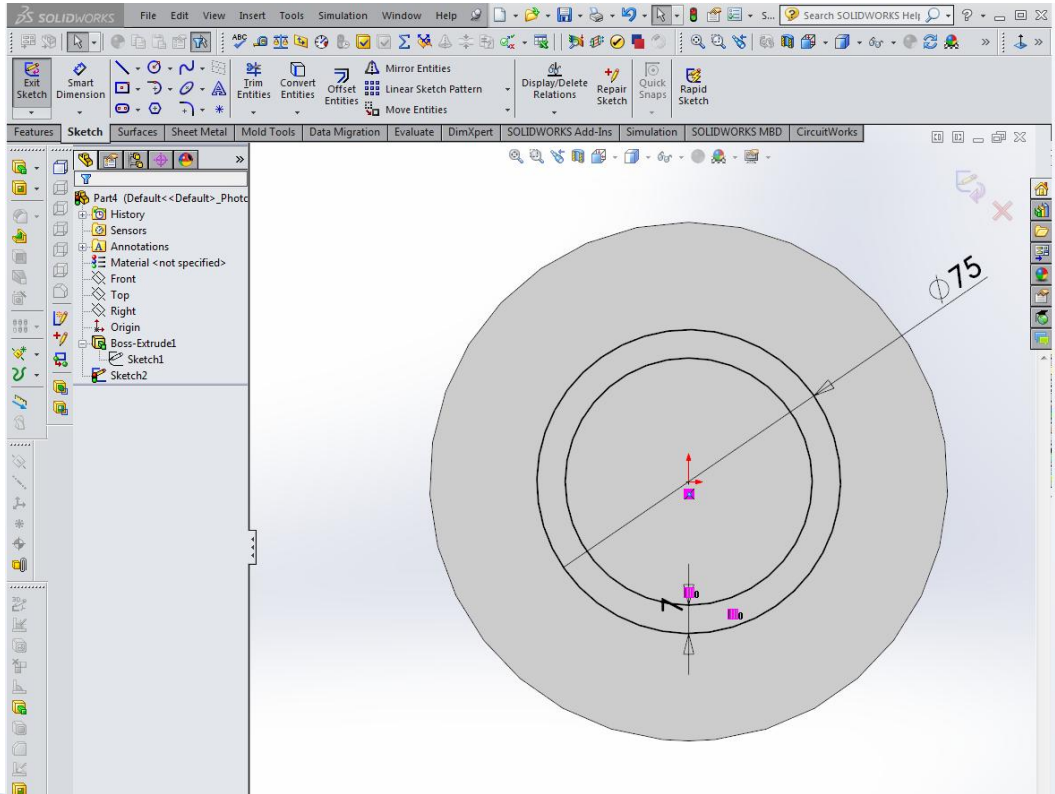
Show Back Print

Offsetting Entities

The sketched circle represents the outside of the ring. Next create the inside of the ring using the Offset Entities tool.





1. Click **Offset Entities** on the Sketch toolbar.
2. In the PropertyManager, under **Parameters**:
 - a. Set **Offset Distance** to 5.
 - b. Select **Reverse** to offset the circle to the inside.
3. Select the sketched circle.
4. Click .

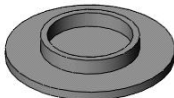
Finished Offset Sketch



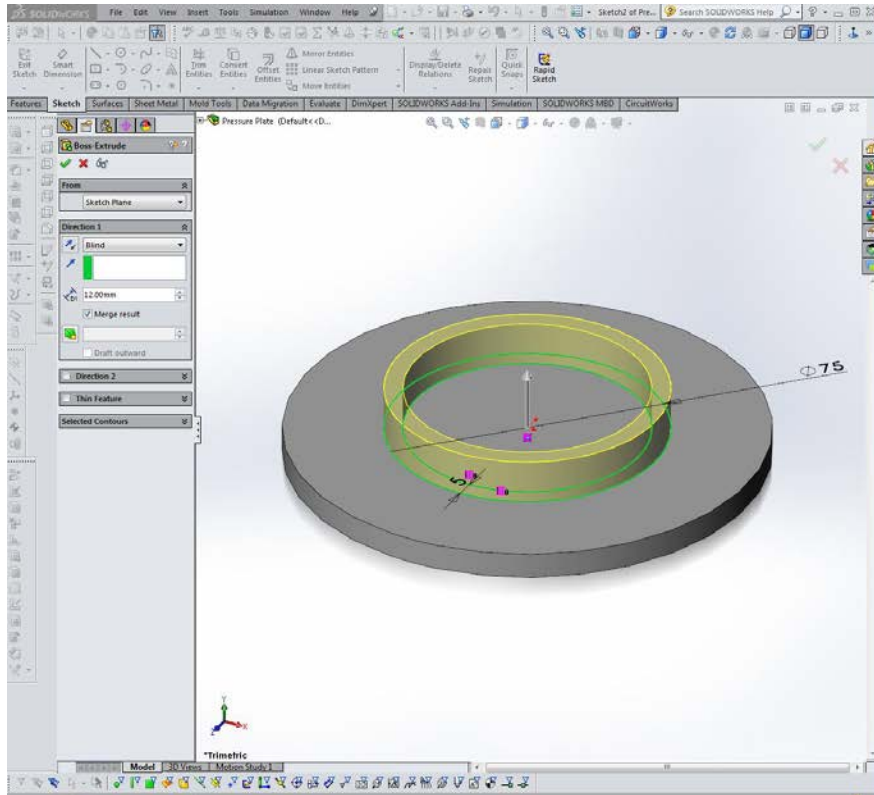
Extruding the Ring Boss

Now that the sketch is complete, extrude the sketch to make the ring boss.

1. Click **Exit Sketch**  on the Sketch toolbar.
2. Click **Trimetric**  on the Standard Views toolbar for a better view of the model.
3. In the PropertyManager, under **Direction 1**, set **Depth**  to 12.
4. Click .




Extruding the Boss Ring



Extruding the Ring Boss

Now that the sketch is complete, extrude the sketch to make the ring boss.

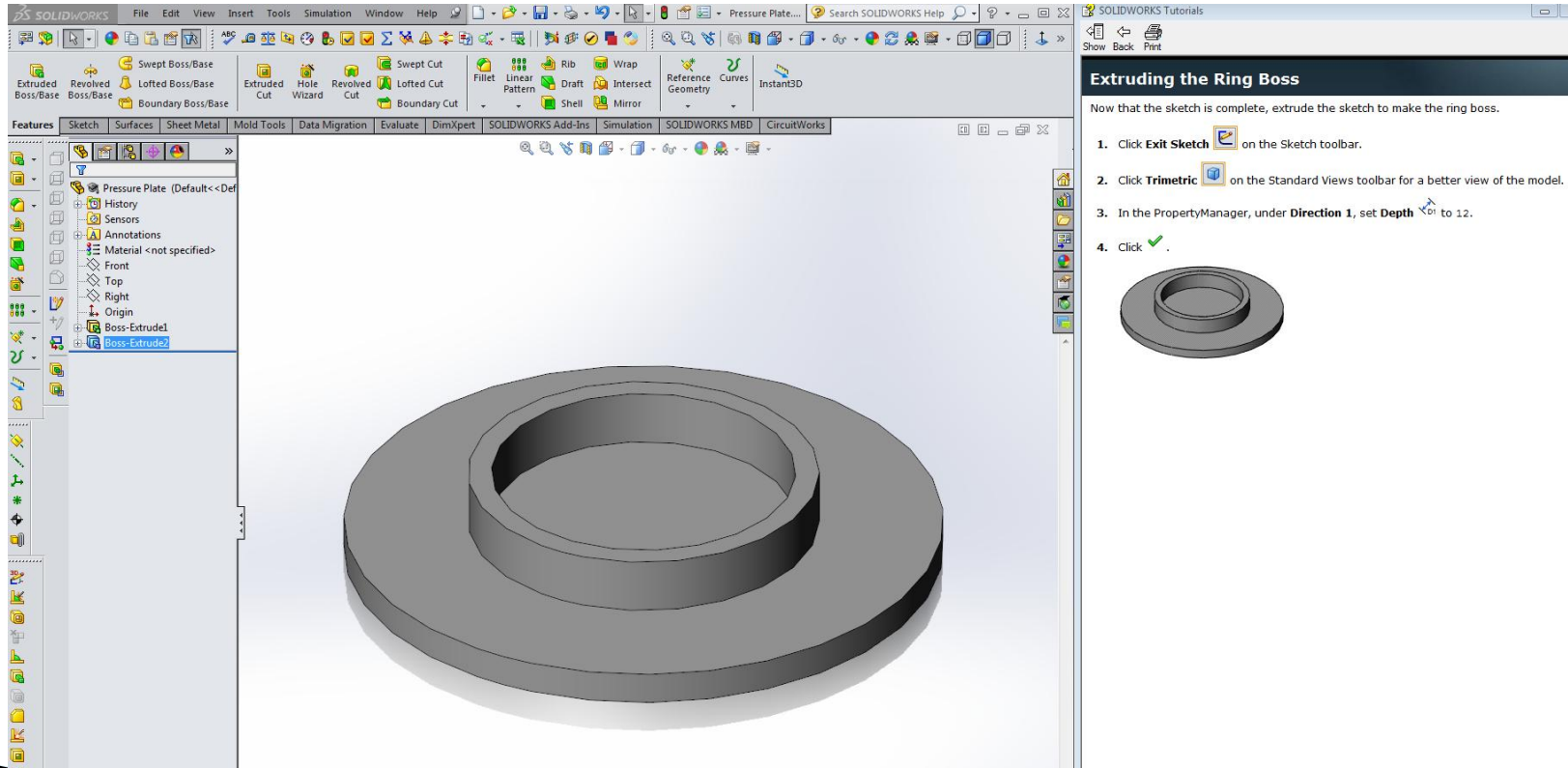
1. Click **Exit Sketch** on the Sketch toolbar.
2. Click **Trimetric** on the Standard Views toolbar for a better view of the model.
3. In the PropertyManager, under **Direction 1**, set **Depth** to 12.
4. Click **✓**.



PREVIOUS TOPIC
Offsetting Entities

NEXT TOPIC
Sketching the Hole

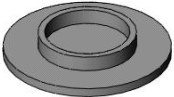
The Extruded Boss Ring



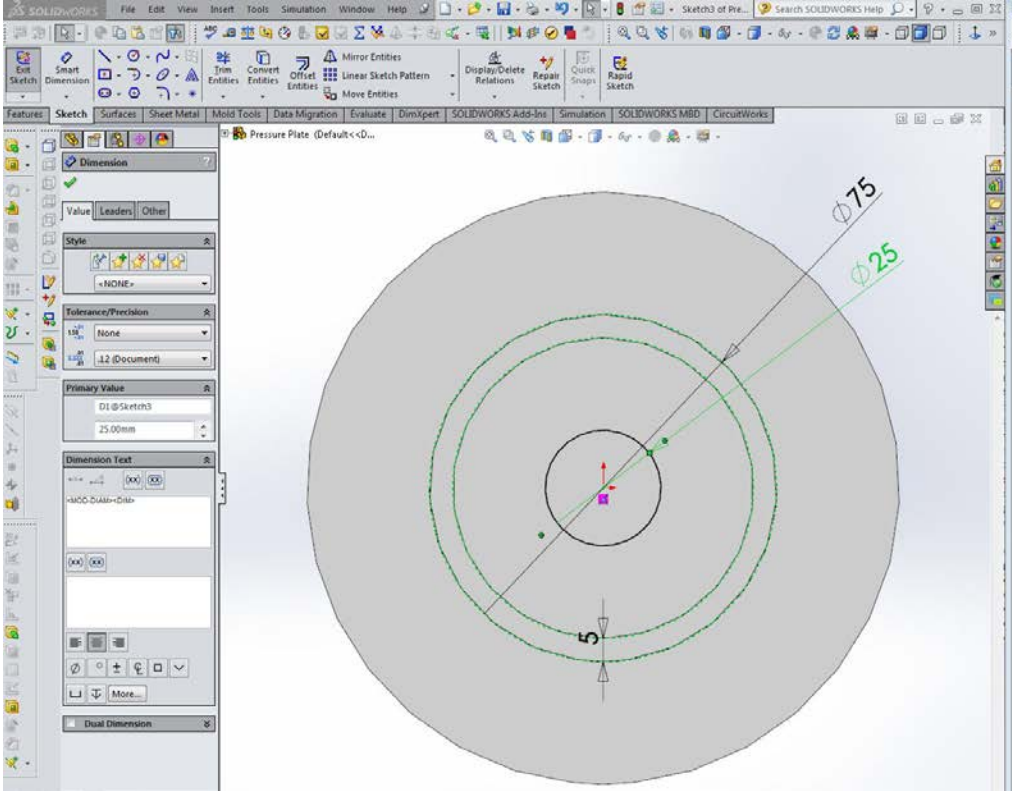
Extruding the Ring Boss

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2. Click **Trimetric** on the Standard Views toolbar for a better view of the model.
3. In the PropertyManager, under **Direction 1**, set **Depth** to 12.
4. Click **✓**.



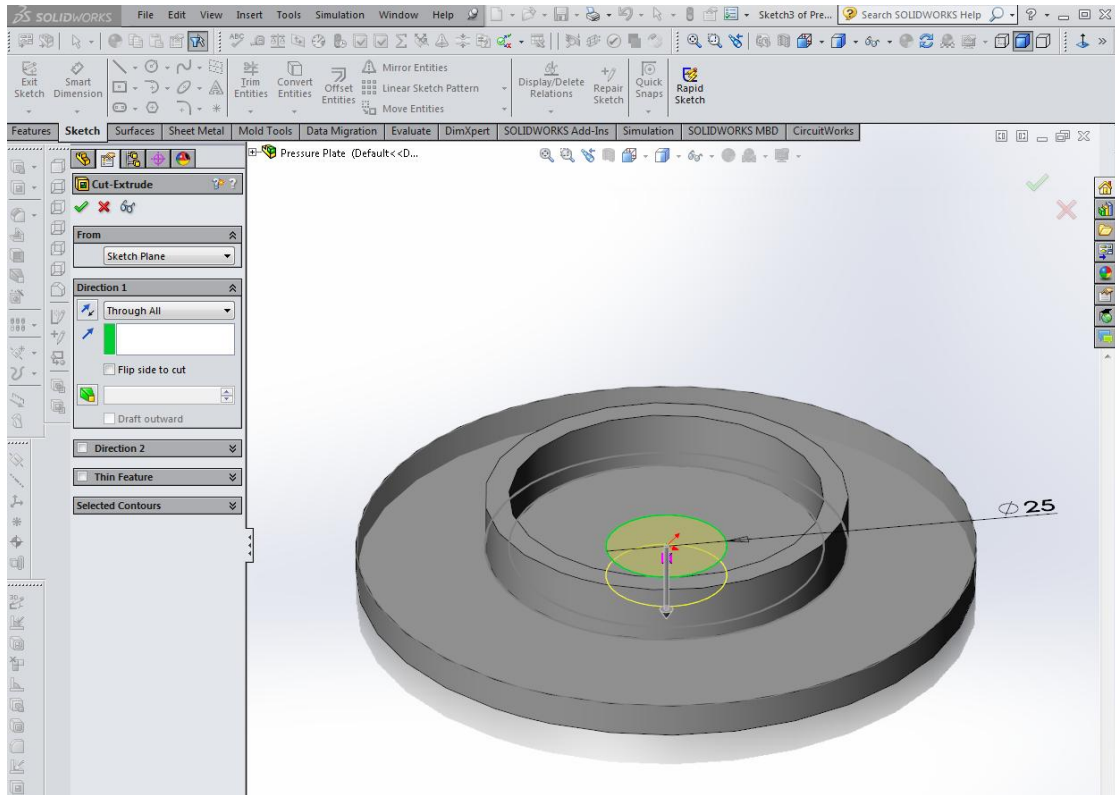
Dimensioning the Hole Sketch



The screenshot displays the SolidWorks CAD environment. The central workspace shows a circular sketch on a grey 'Pressure Plate' background. A central hole is defined with a diameter dimension of 5. A dashed green circle is dimensioned with a diameter of 25. A larger outer circle is dimensioned with a diameter of 75. The left-hand side of the interface shows the 'Dimension' property manager with various settings like 'Style', 'Tolerance/Precision', and 'Dimension Text'. The top toolbar includes the 'Smart Dimension' icon. On the right, a 'SOLIDWORKS Tutorials' window is open, displaying the title 'Dimensioning the Hole Sketch' and a list of four steps.

1. Click **Smart Dimension** (Dimensions/Relations toolbar).
2. Select the circle.
3. Move the pointer and click to place the dimension.
4. In the Modfly box, type 25, then click and click in the graphics area.


Creating the Hole



Creating a Hole

Cut a hole through the center of the part.

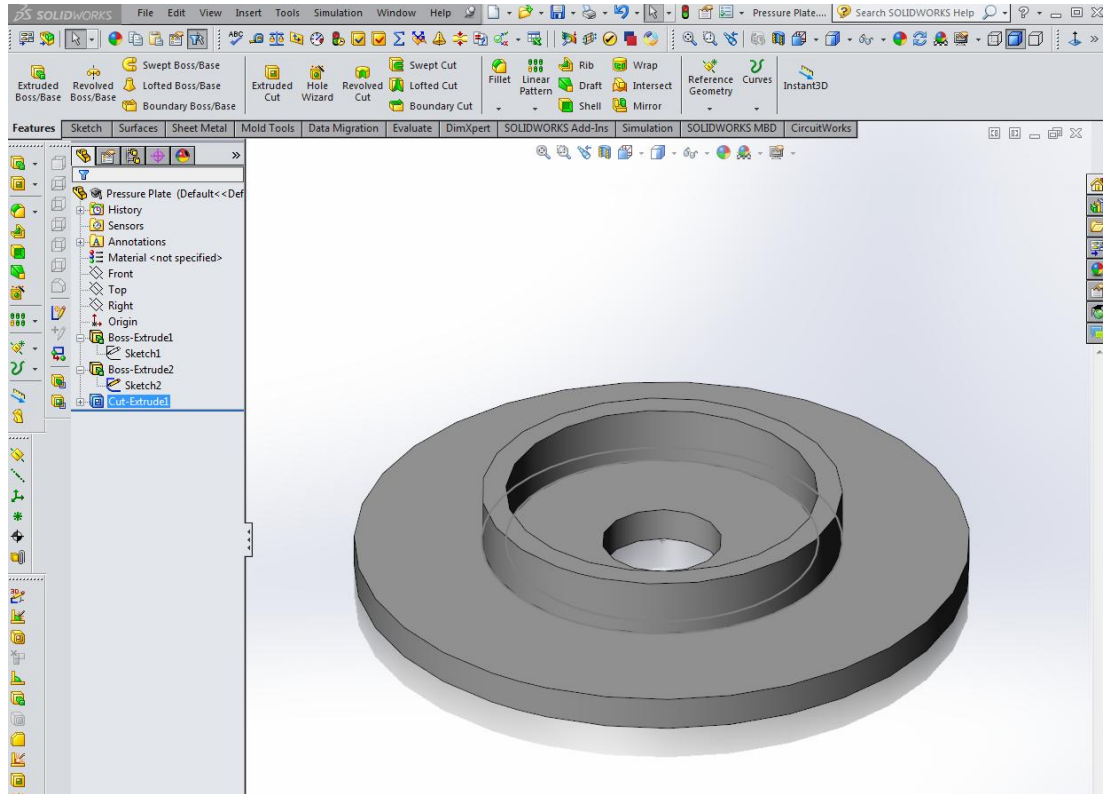
1. Click **Exit Sketch** (Sketch toolbar).
2. Click **Trimetric** (Standard Views toolbar).
3. In the PropertyManager, under **Direction 1**, select **Through All** for End Condition.
4. Click **✓**.



You can use the Hole Wizard to add customized holes to the model.

- [Learn how to create Hole Wizard holes.](#)
- [Skip the Hole Wizard lesson and work on fillets.](#)

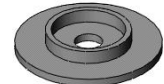
The Hole



Creating a Hole

Cut a hole through the center of the part.

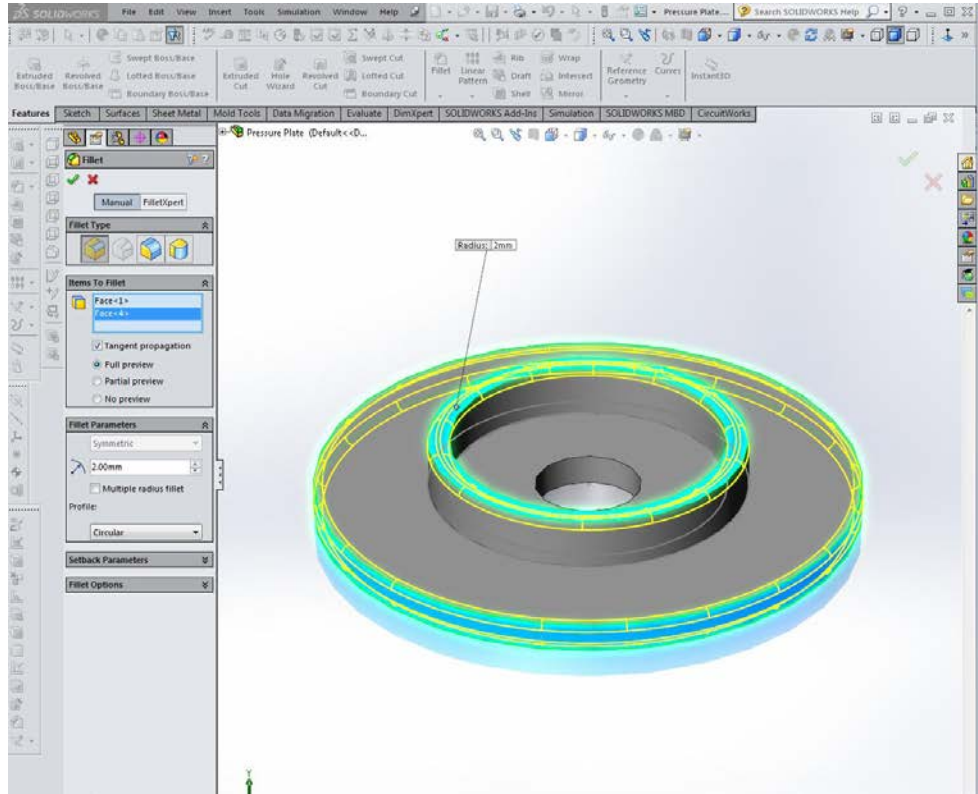
1. Click **Exit Sketch** (Sketch toolbar).
2. Click **Trimetric** (Standard Views toolbar).
3. In the PropertyManager, under **Direction 1**, select **Through All for End Condition**.
4. Click **✓**.



You can use the Hole Wizard to add customized holes to the model.

- Learn how to create Hole Wizard holes.
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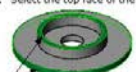
Selecting Fillets



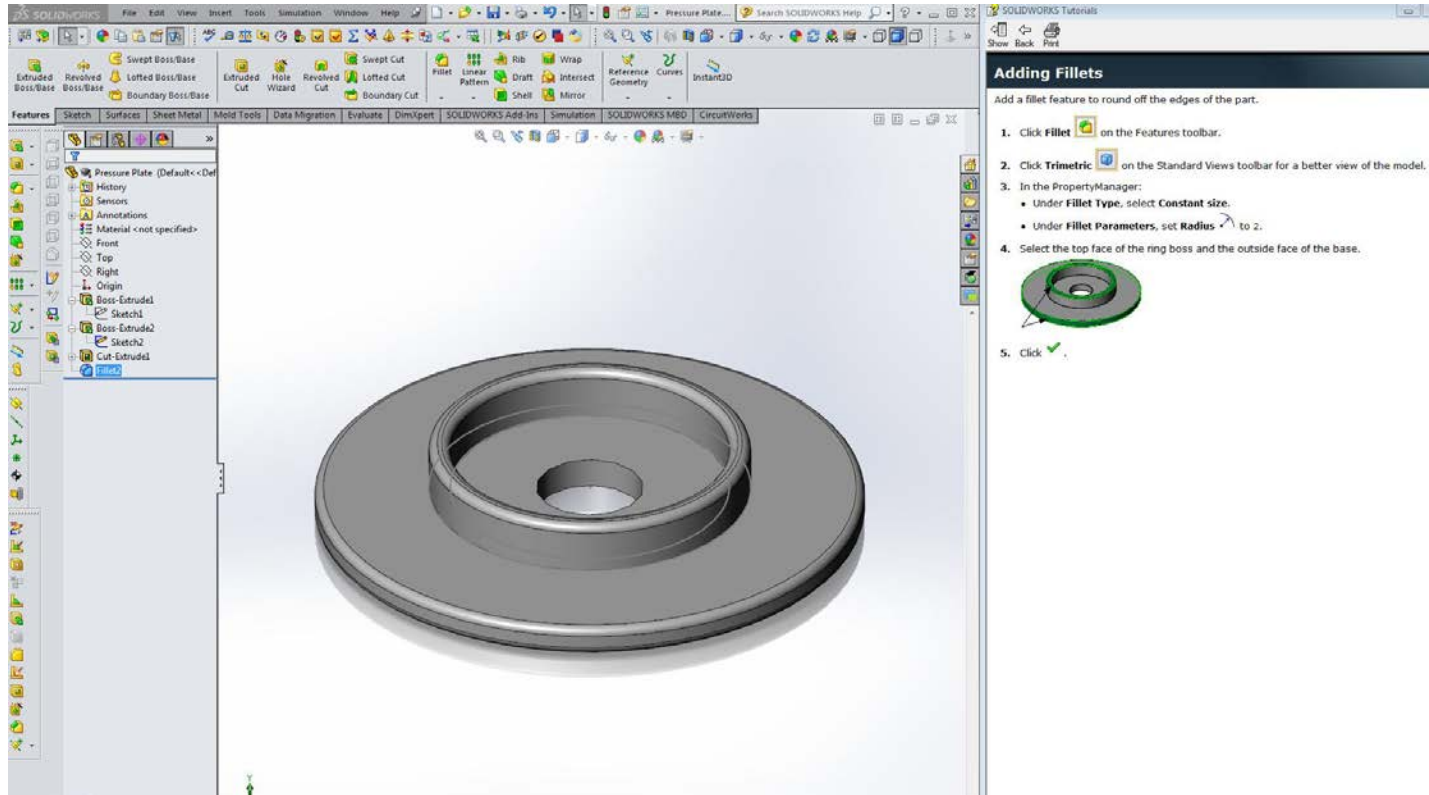
Adding Fillets

Add a fillet feature to round off the edges of the part.

1. Click **Fillet** on the Features toolbar.
2. Click **Trimetric** on the Standard Views toolbar for a better view of the model.
3. In the PropertyManager:
 - Under **Fillet Type**, select **Constant size**.
 - Under **Fillet Parameters**, set **Radius** to 2.
4. Select the top face of the ring boss and the outside face of the base.
5. Click **✓**.




Fillets Shown

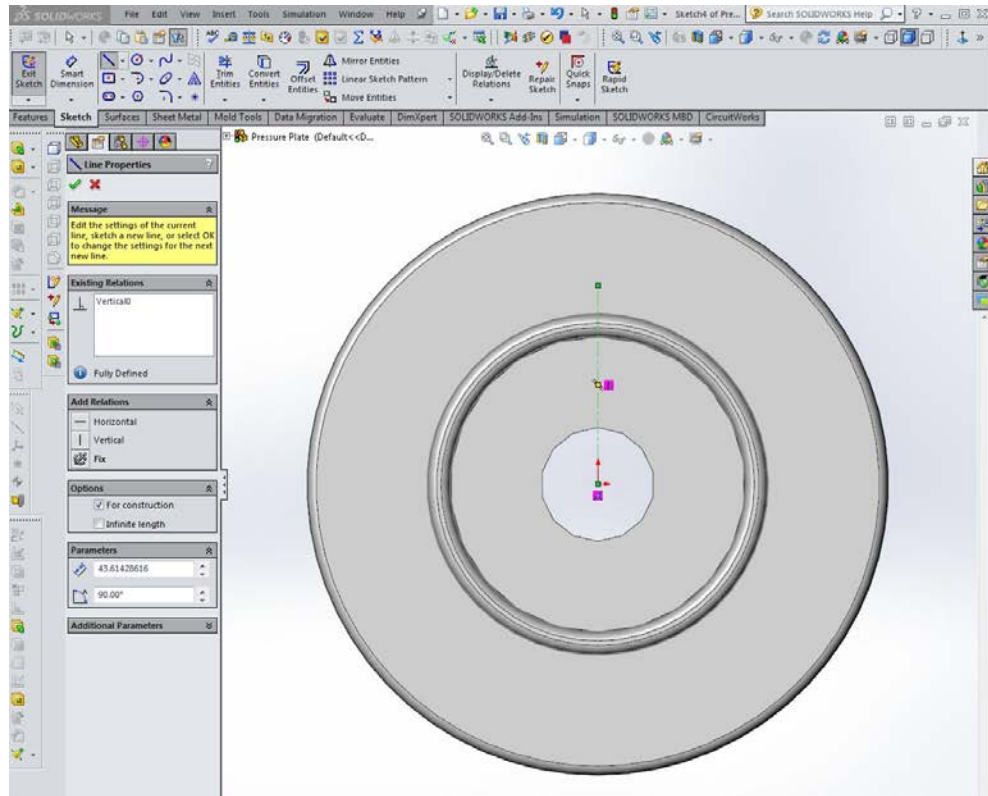


Adding Fillets

Add a fillet feature to round off the edges of the part.

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3. In the PropertyManager:
 - Under **Fillet Type**, select **Constant size**.
 - Under **Fillet Parameters**, set **Radius** to 2.
4. Select the top face of the ring boss and the outside face of the base.
5. Click .

Draw a Center Line




SOLIDWORKS Tutorials


Sketching the Tall Cylinder Extrusion

Sketch a circle for the tall cylinder extrusion.

1. Click **Extruded Boss/Base** on the Features toolbar.
2. Select the top face of the base cylinder.
3. Click **Top** on the Standard Views toolbar.
4. Expand the **Line** flyout menu on the Sketch CommandManager and click **Centerline**.

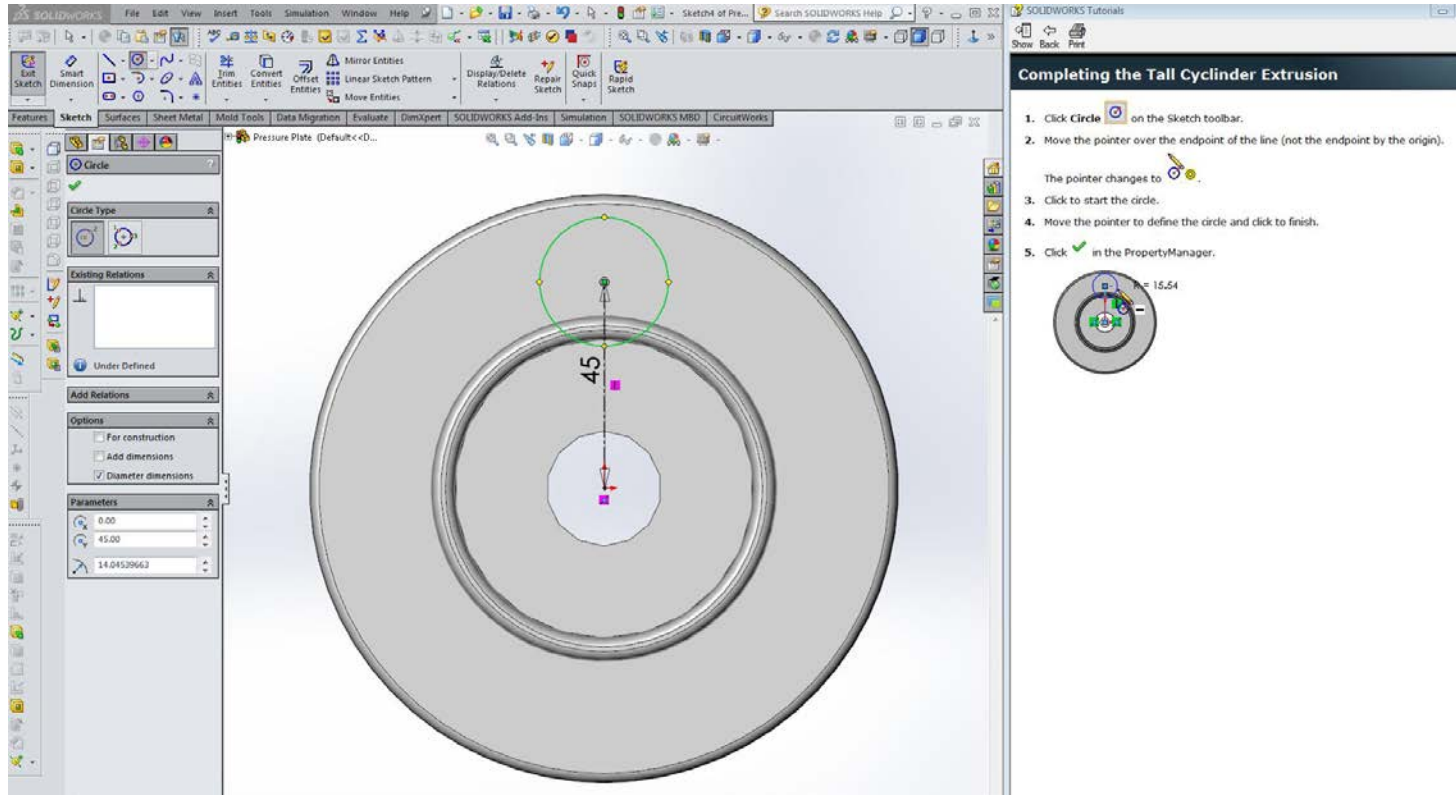
The centerline acts as a construction line for the next circle. It keeps the center of the circle vertical with respect to the origin.

5. Move the pointer over the origin until the pointer changes to  and click to start the centerline.
6. Move the mouse above the start of the centerline.

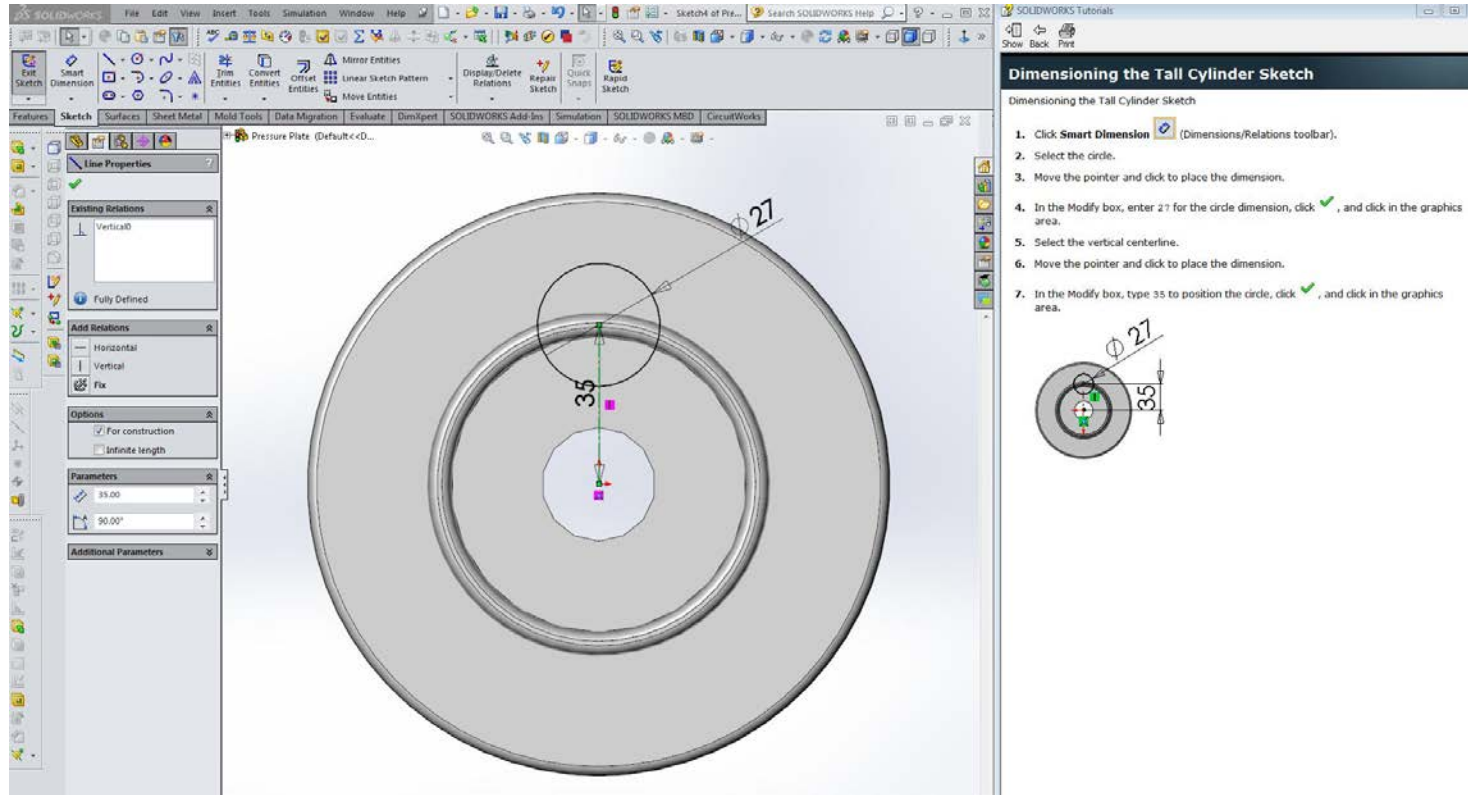
The pointer changes to  to indicate the centerline is vertical.

7. Click again to end the line. Make the line about 45mm long.

Draw a Circle at End of Center Line



Re-dimension the Center Line

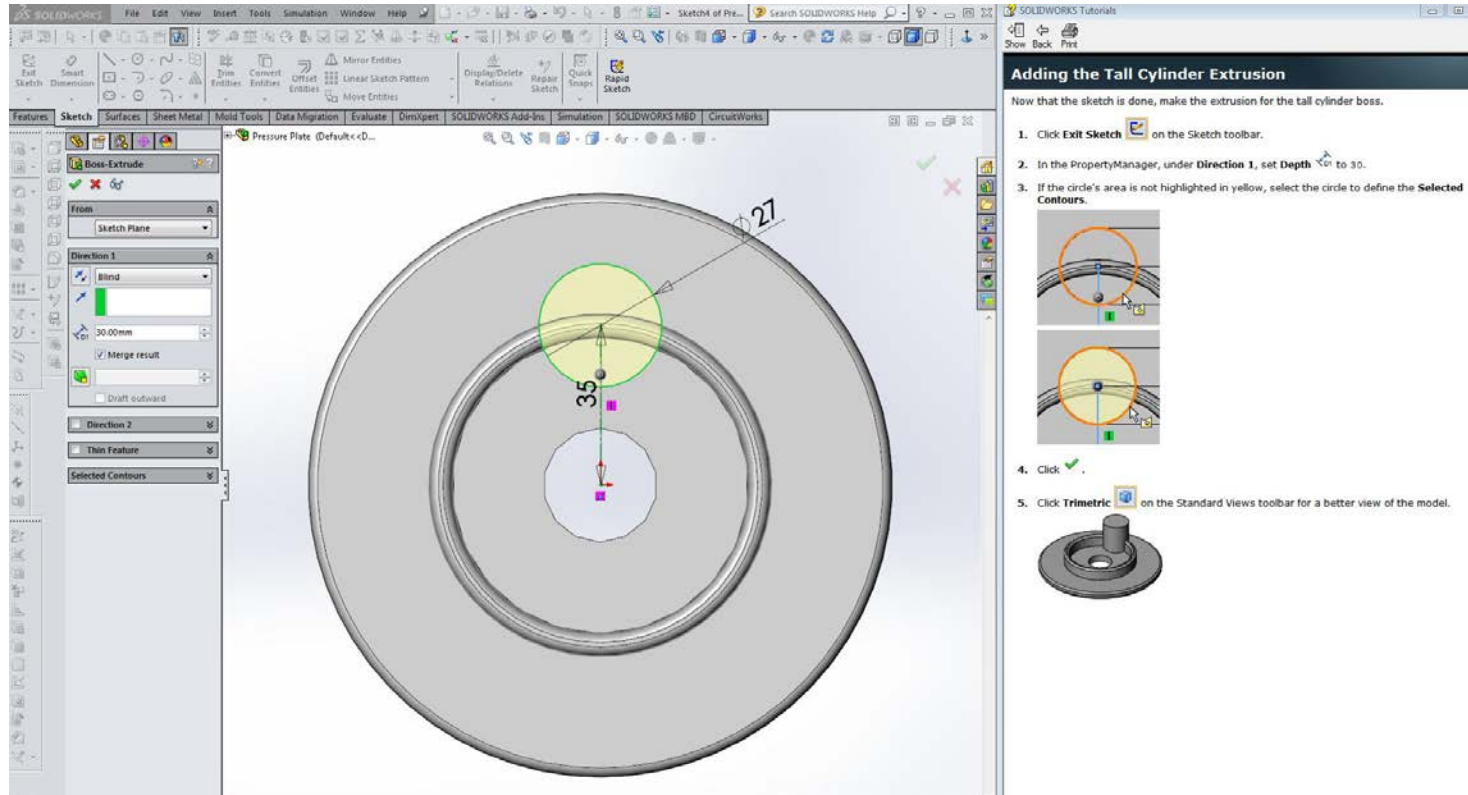


Dimensioning the Tall Cylinder Sketch

Dimensioning the Tall Cylinder Sketch


1. Click **Smart Dimension** (Dimensions/Relations toolbar).
2. Select the circle.
3. Move the pointer and click to place the dimension.
4. In the Modify box, enter 27 for the circle dimension, click , and click in the graphics area.
5. Select the vertical centerline.
6. Move the pointer and click to place the dimension.
7. In the Modify box, type 35 to position the circle, click , and click in the graphics area.


Add the Tall Cylinder Extrusion



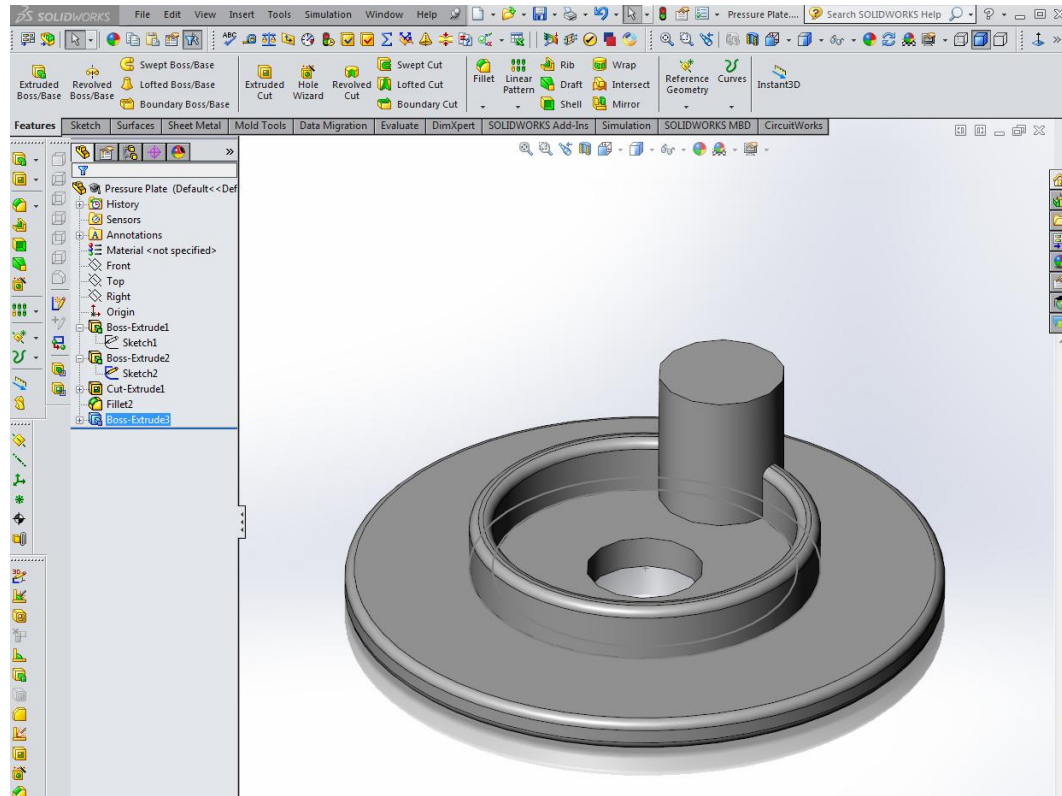
Adding the Tall Cylinder Extrusion

Now that the sketch is done, make the extrusion for the tall cylinder boss.

1. Click **Exit Sketch** on the Sketch toolbar.
2. In the PropertyManager, under **Direction 1**, set **Depth** to 30.
3. If the circle's area is not highlighted in yellow, select the circle to define the **Selected Contours**.
4. Click .
5. Click **Trimetric** on the Standard Views toolbar for a better view of the model.



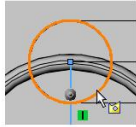
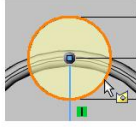


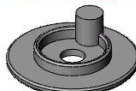


Cylinder Boss Trimetric View

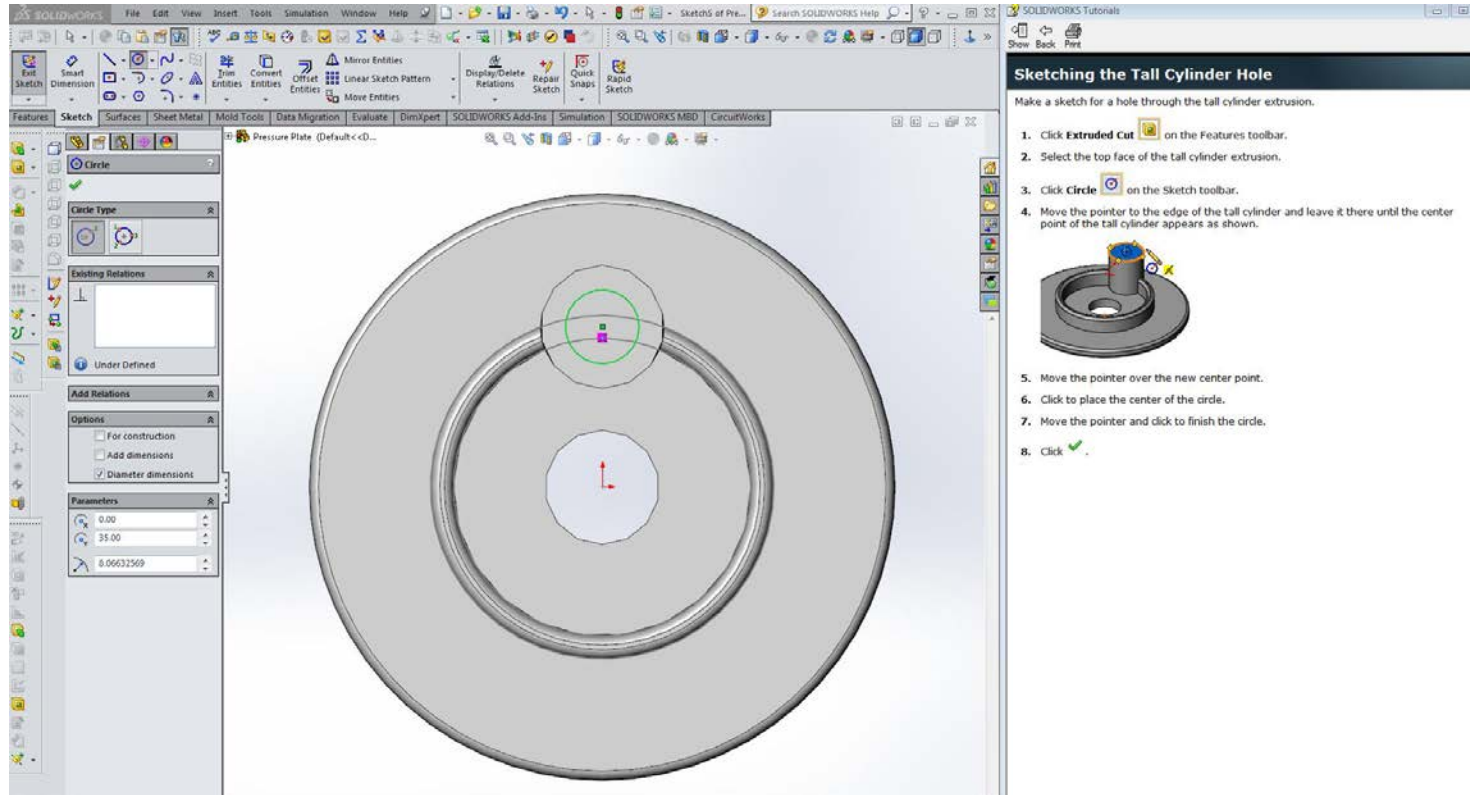


Adding the Tall Cylinder Extrusion

Now that the sketch is done, make the extrusion for the tall cylinder boss.

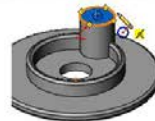
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Sketching the Extended Cut



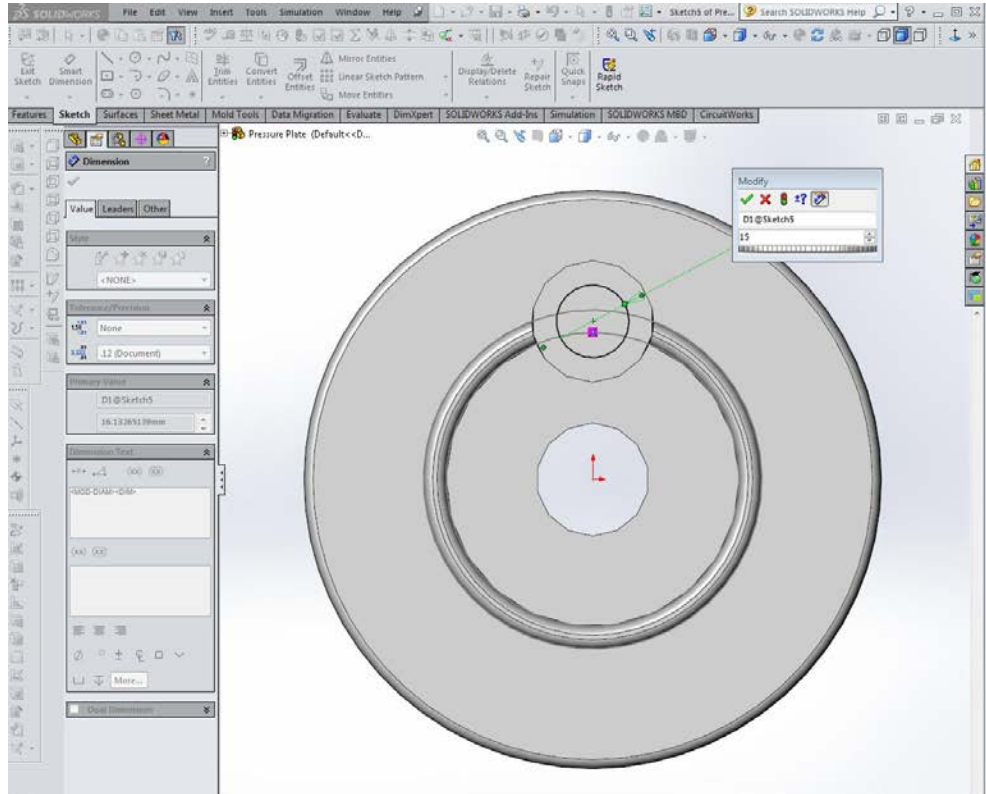
Sketching the Tall Cylinder Hole
Make a sketch for a hole through the tall cylinder extrusion.

1. Click **Extruded Cut** on the Features toolbar.
2. Select the top face of the tall cylinder extrusion.
3. Click **Circle** on the Sketch toolbar.
4. Move the pointer to the edge of the tall cylinder and leave it there until the center point of the tall cylinder appears as shown.



5. Move the pointer over the new center point.
6. Click to place the center of the circle.
7. Move the pointer and click to finish the circle.
8. Click .

Dimensioning the Cut

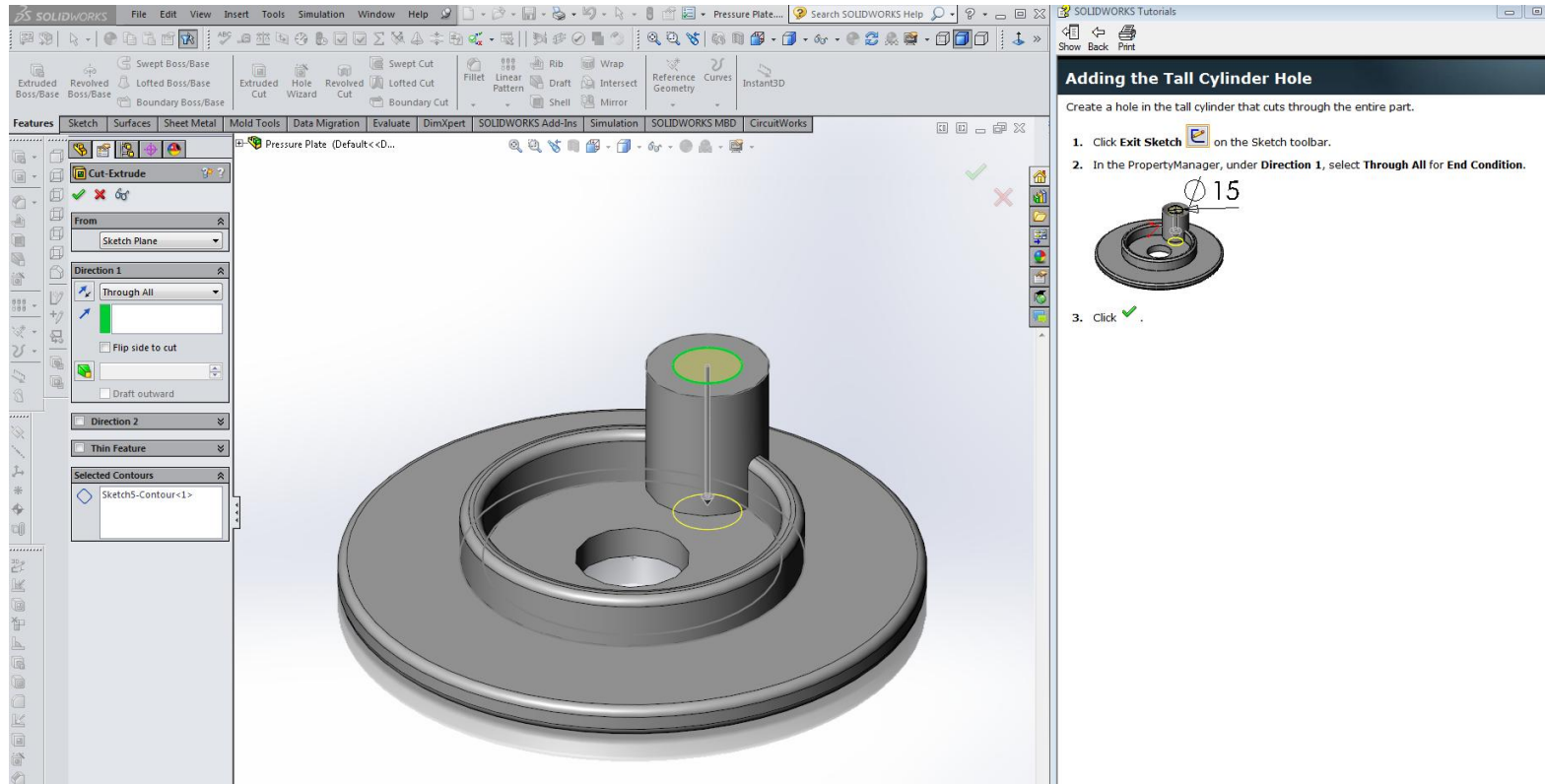


Dimensioning the Tall Cylinder Hole Sketch

Add a dimension to control the diameter of the circle.

1. Click **Smart Dimension** (Dimensions/Relations toolbar).
2. Select the circle.
3. Move the pointer and click to place the dimension.
4. In the Modify box, type 15, click , and click in the graphics area.


Extruding the Cylinder Hole

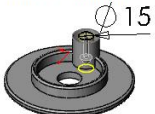



SOLIDWORKS Tutorials

Adding the Tall Cylinder Hole

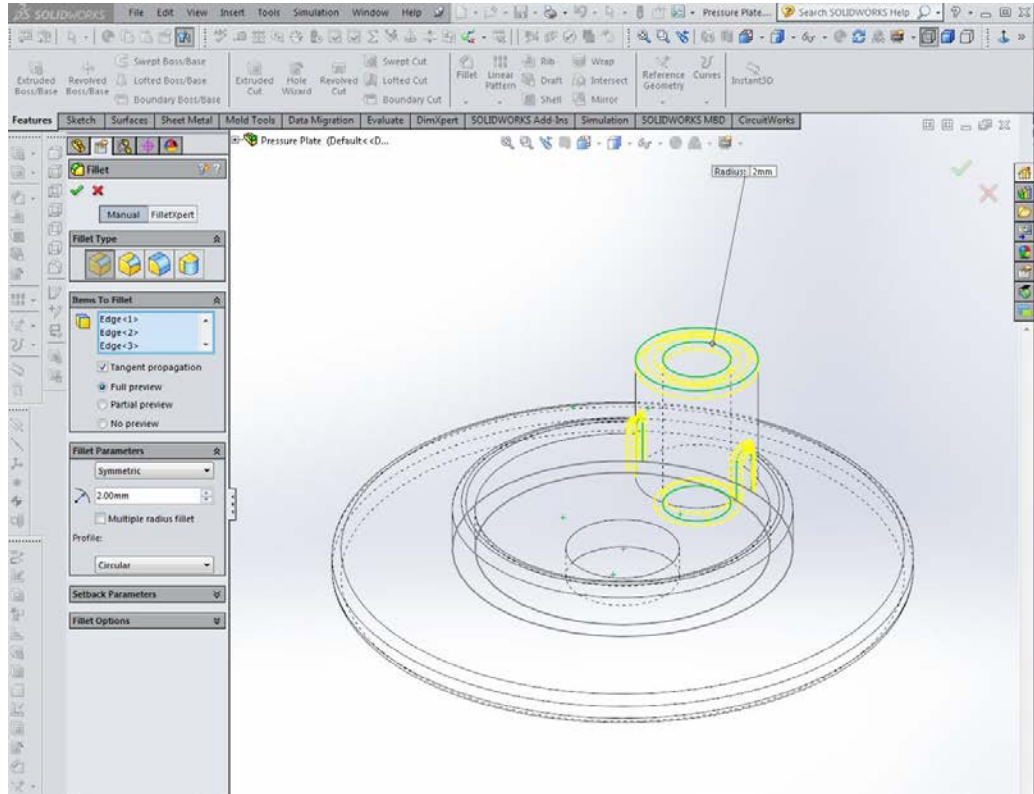
Create a hole in the tall cylinder that cuts through the entire part.

1. Click **Exit Sketch**  on the Sketch toolbar.
2. In the PropertyManager, under **Direction 1**, select **Through All** for **End Condition**.







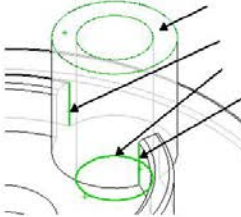
3. Click .

Adding Fillets to the Cylinder

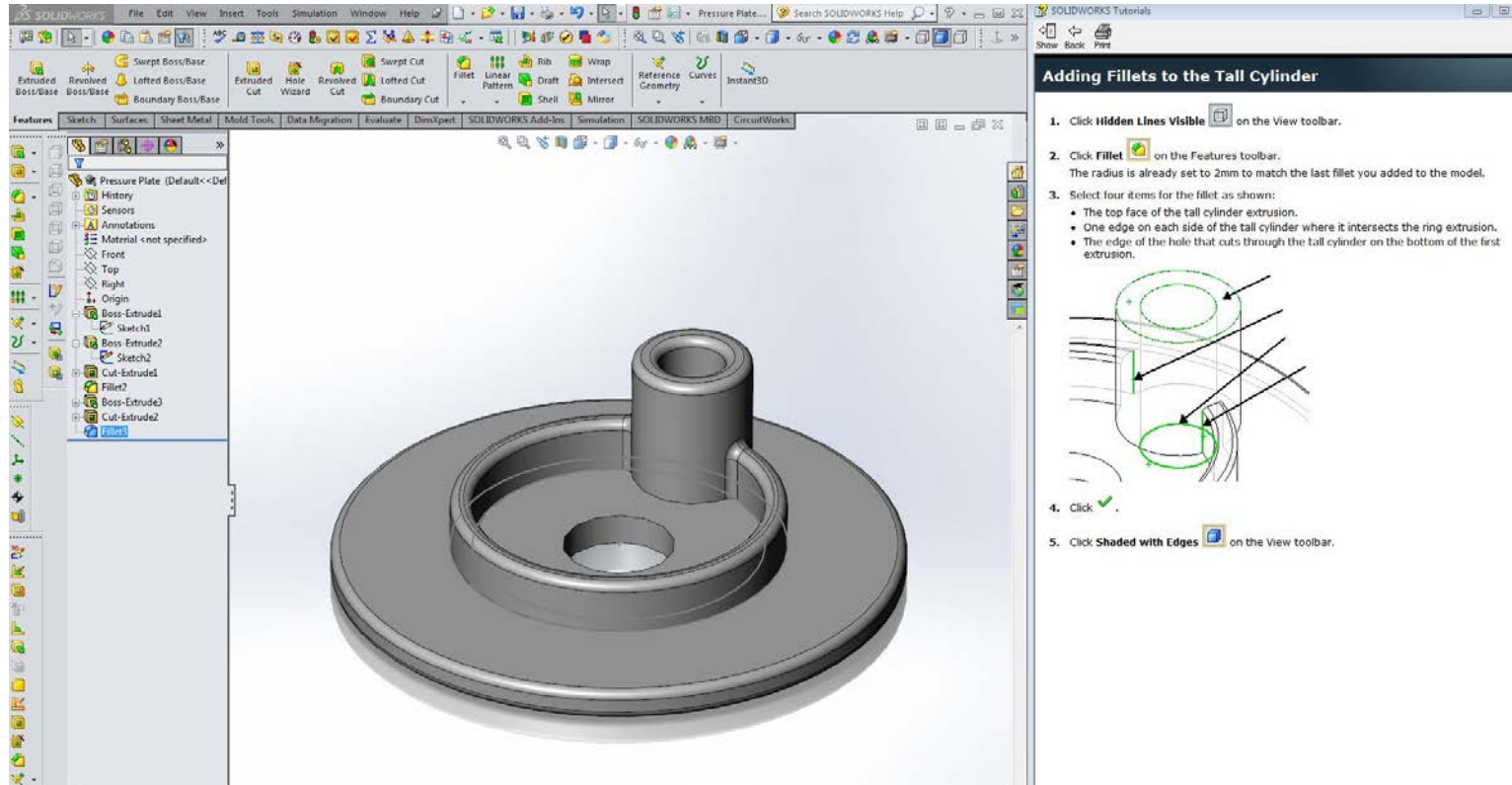


Adding Fillets to the Tall Cylinder





1. Click **Hidden Lines Visible**  on the View toolbar.
2. Click **Fillet**  on the Features toolbar.
The radius is already set to 2mm to match the last fillet you added to the model.
3. Select four items for the fillet as shown:
 - The top face of the tall cylinder extrusion.
 - One edge on each side of the tall cylinder where it intersects the ring extrusion.
 - The edge of the hole that cuts through the tall cylinder on the bottom of the first extrusion.
4. Click .
5. Click **Shaded with Edges**  on the View toolbar.

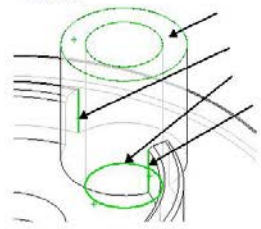


Shaded View of Fillets



Adding Fillets to the Tall Cylinder

1. Click **Hidden Lines Visible**  on the View toolbar.
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The radius is already set to 2mm to match the last fillet you added to the model.
3. Select four items for the fillet as shown:
 - The top face of the tall cylinder extension.
 - One edge on each side of the tall cylinder where it intersects the ring extension.
 - The edge of the hole that cuts through the tall cylinder on the bottom of the first extension.
4. Click .
5. Click **Shaded with Edges**  on the View toolbar.



Circular Pattern

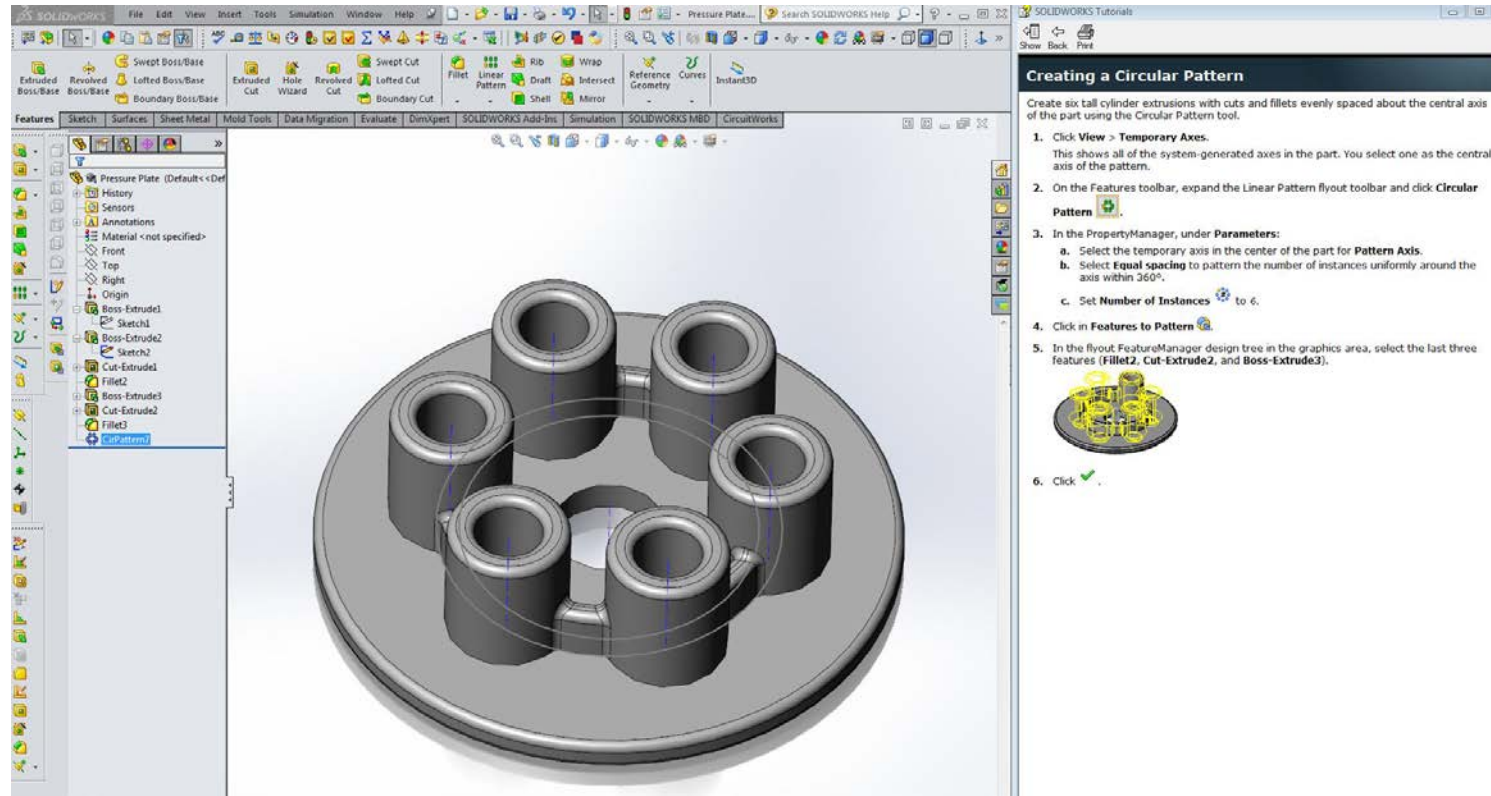
Creating a Circular Pattern

Create six tall cylinder extrusions with cuts and fillets evenly spaced about the central axis of the part using the Circular Pattern tool.

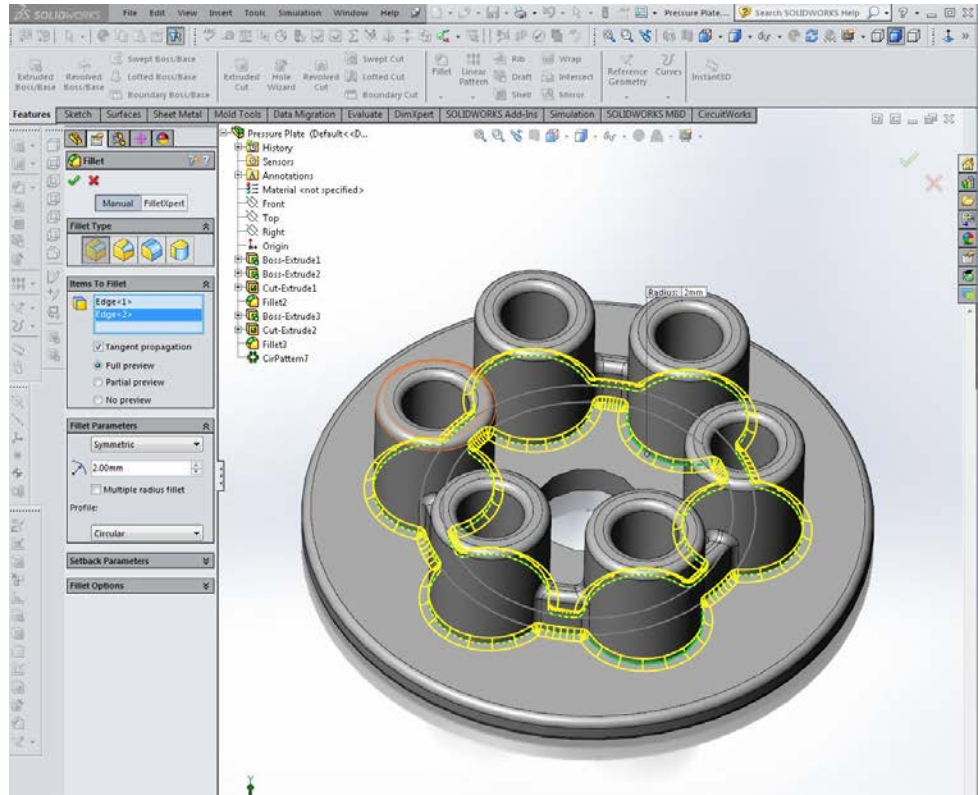
1. Click **View > Temporary Axes**.
This shows all of the system-generated axes in the part. You select one as the central axis of the pattern.
2. On the Features toolbar, expand the Linear Pattern flyout toolbar and click **Circular Pattern**.
3. In the PropertyManager, under **Parameters**:
 - a. Select the temporary axis in the center of the part for **Pattern Axis**.
 - b. Select **Equal spacing** to pattern the number of instances uniformly around the axis within 360°.
 - c. Set **Number of Instances** to 6.
4. Click in **Features to Pattern**.
5. In the flyout FeatureManager design tree in the graphics area, select the last three features (**Fillet2**, **Cut-Extrude2**, and **Boss-Extrude3**).
6. Click **OK**.

This should be Fillet3 and not Fillet2.

Finished Circular Pattern



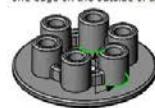
Adding Last Fillets



Adding the Last Fillet

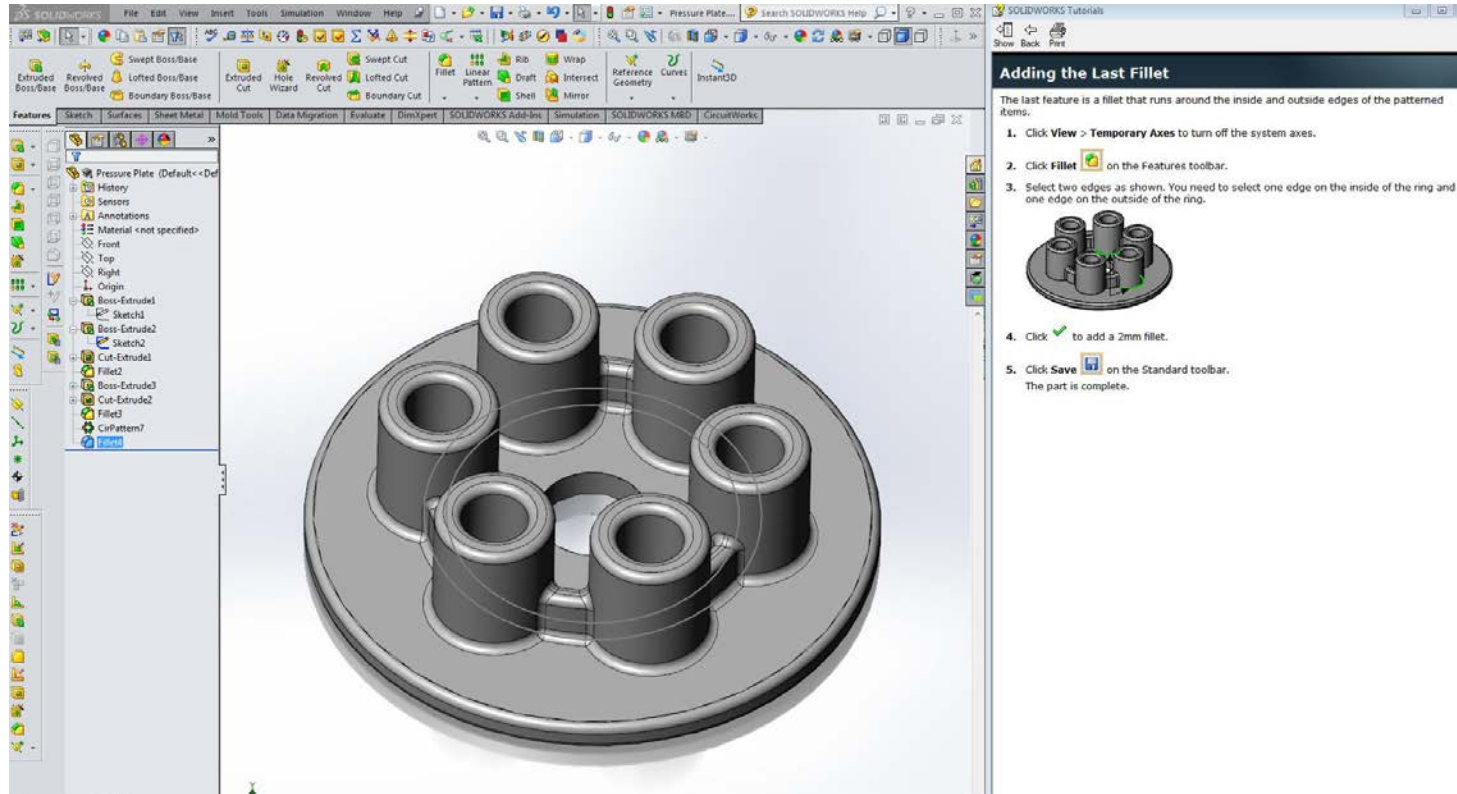
The last feature is a fillet that runs around the inside and outside edges of the patterned items.

1. Click **View** > **Temporary Axes** to turn off the system axes.
2. Click **Fillet** on the Features toolbar.
3. Select two edges as shown. You need to select one edge on the inside of the ring and one edge on the outside of the ring.



4. Click to add a 2mm fillet.
5. Click **Save** on the Standard toolbar.
The part is complete.


Finished part



Adding the Last Fillet

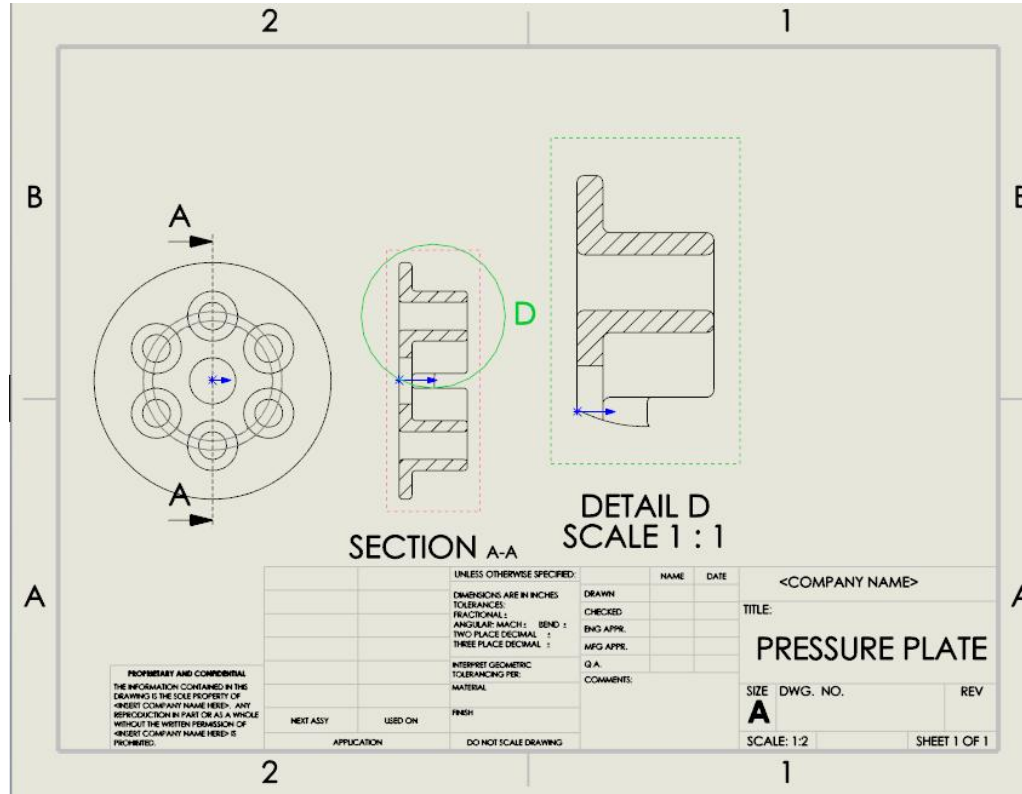
The last feature is a fillet that runs around the inside and outside edges of the patterned items.

1. Click **View** > **Temporary Axes** to turn off the system axes.
2. Click **Fillet** on the Features toolbar.
3. Select two edges as shown. You need to select one edge on the inside of the ring and one edge on the outside of the ring.

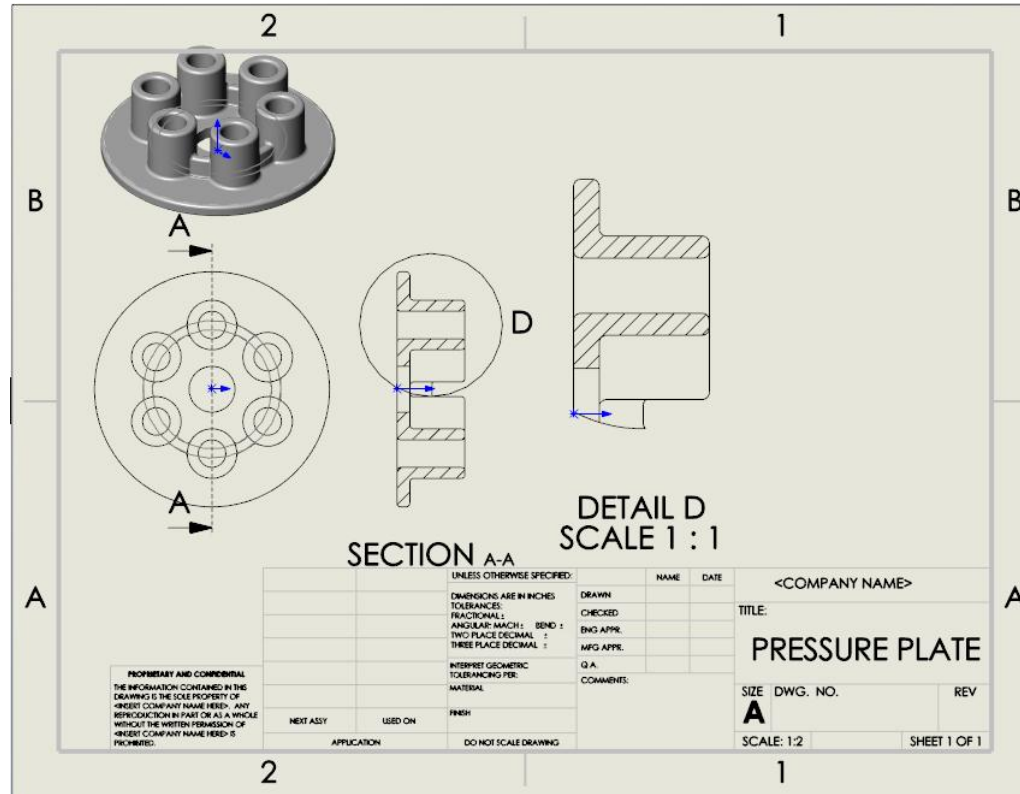


4. Click to add a 2mm fillet.
5. Click **Save** on the Standard toolbar.
The part is complete.

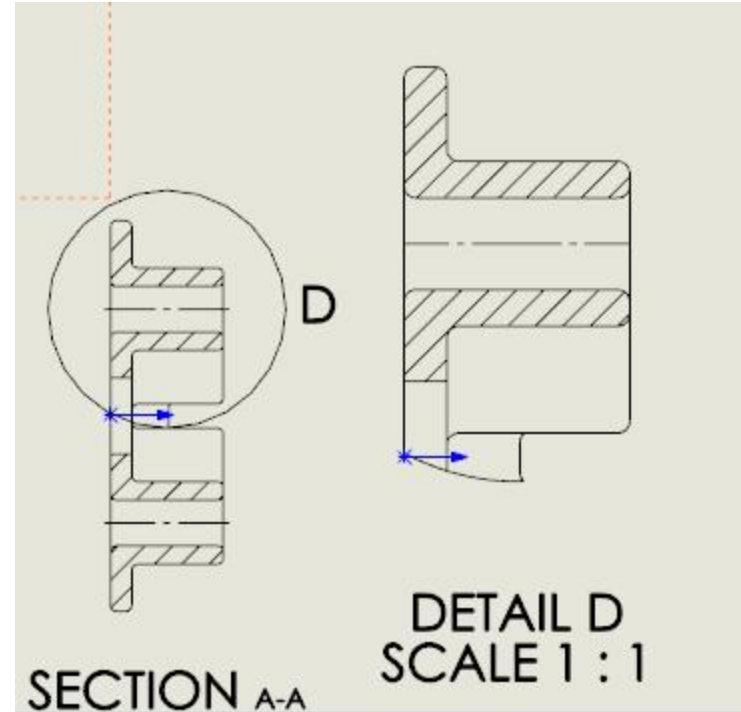
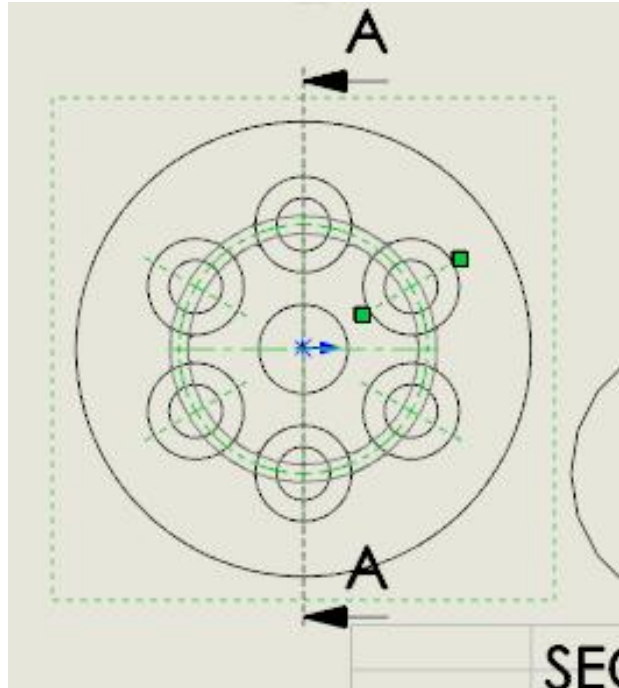
Drawing with Section and Detail View



Adding Isometric View



Adding Center Marks & Center Lines



Document Properties - Units & Dimensions

Document Properties - Units

System Options | Document Properties | Search Options

Drafting Standard

- Annotations
- Dimensions
- Centerlines/Center Marks
- DimXpert
- Tables
- Views
- Virtual Sharps

Detailing

- Drawing Sheets
- Grid/Snap
- Units
- Line Font
- Line Style
- Line Thickness
- Image Quality
- Sheet Metal
- Weldments

Unit system

- MKS (meter, kilogram, second)
- CGS (centimeter, gram, second)
- MMGS (millimeter, gram, second)
- IPS (inch, pound, second)
- Custom

Type	Unit	Decimals	Fractions	More
Basic Units				
Length	millimeters	.12		...
Dual Dimension Length	millimeters	None	2	...
Angle	degrees	.12		
Mass/Section Properties				
Length	millimeters	.12		
Mass	grams			
Per Unit Volume	millimeters^3			
Motion Units				
Time	second	.12		
Force	newton	.12		
Power	watt	.12		
Energy	joule	.12		

Decimal rounding

- Round half away from zero
- Round half towards zero
- Round half to even
- Truncate without rounding

Only apply rounding method to dimensions

OK Cancel Help

Document Properties - Dimensions

System Options | Document Properties | Search Options

Drafting Standard

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Overall drafting standard

ISO

Text

Font... Century Gothic

Dual dimensions

Dual dimensions display Show units for dual display

Top Bottom Right Left

Primary precision Dual precision

15.00 Same as nominal 15.00 Same as nominal

Link precisions with model Link precisions with model

Fractional display

Style: Stack size: 100%

Show double prime mark ("):

Include leading zero for values less than 1"

Bent leaders

Leader length: 6.35mm Extend to text

Leading zeros: Standard

Trailing zeros: Smart

Show units of dimensions

Add parentheses by default

Center between extension lines

Include prefix inside basic tolerance box

Display dual basic dimension in one box

Show dimensions as broken in broken views

Arrows

1.02mm

3.3mm

6.35mm

Scale with dimension height

Style:

Offset distances

Annotation view layout

6mm

10mm

Break dimension extension/leader lines

Gap: 1.52mm

Break only around dimension arrows

Extension lines

Gap: 1mm

Beyond dimension line: 1mm

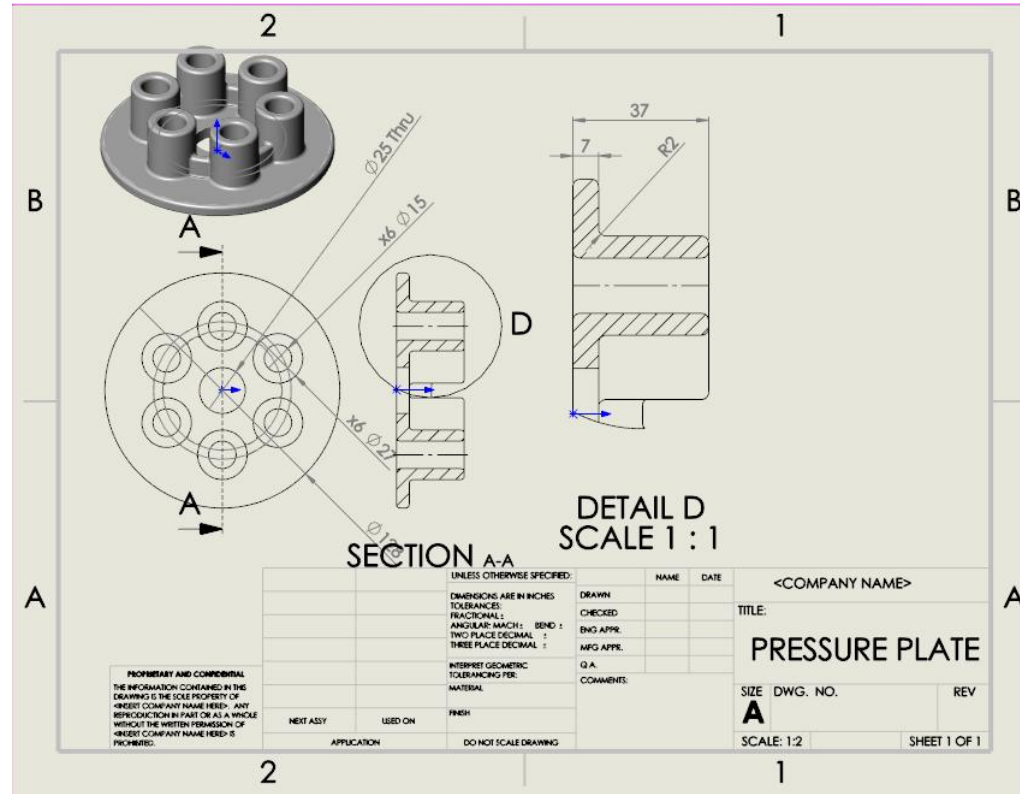
Radial/Diameter leader snap angle: 15deg

Tolerance...

Apply updated rules

OK Cancel Help

Adding Dimensions & Modifying Text



Summary

- ▶ Topics Covered in this exercise:
 - Parts:
 - Sketches such as circles, lines and offsets.
 - Selecting a plane or surface for sketches.
 - Dimensioning.
 - Features such as extruded-boss and extruded cut.
 - Various views (perspectives), fill, and wire frame models.
 - Fillets.
 - Temporary axis.
 - Circular patterns.
 - Drawings:
 - Making drawing, placing view and hidden line removal.
 - Section and detail views.
 - Adding isometric view with filled surfaces.
 - Center lines and center marks.
 - Dimensioning and adding text.