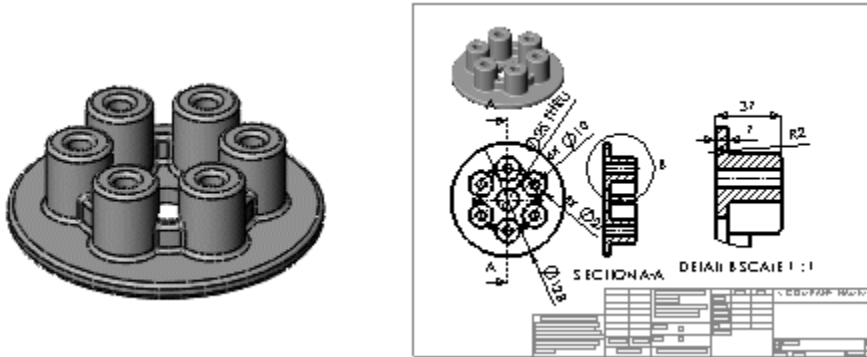


Introduction to SolidWorks

The Introduction to SolidWorks lesson guides you through the creation of a part and a drawing. You create this part and drawing:



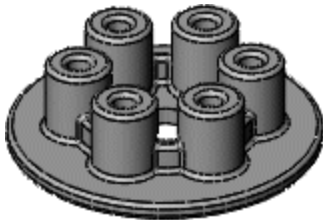
This lesson includes:

- Creating base, boss, and cut features from sketches
- Adding fillets to smooth edges
- Creating a circular pattern
- Adding drawing views
- Adding centerlines, center marks, and dimensions to the drawing

[Next](#) >

My First Part

For your first part, you create the pressure plate shown below. A part is a 3D model made up of features.



You begin this lesson by creating a new part.


1. Click **New**  on the Standard toolbar.

The **New SolidWorks Document** dialog box appears.

2. Click **Part**.
3. Click **OK**.

A new part window appears.



When toolbar buttons have an orange border, for example , you can click them in the tutorial window to flash the corresponding button in the SolidWorks window.

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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.

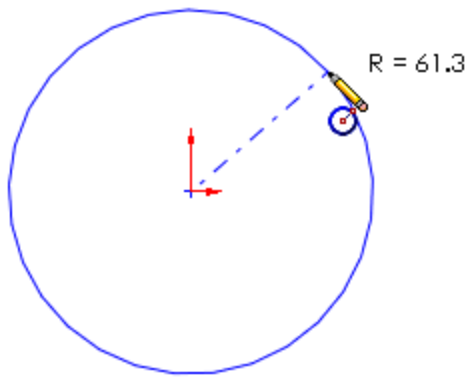
4. Move the pointer over the origin .

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.




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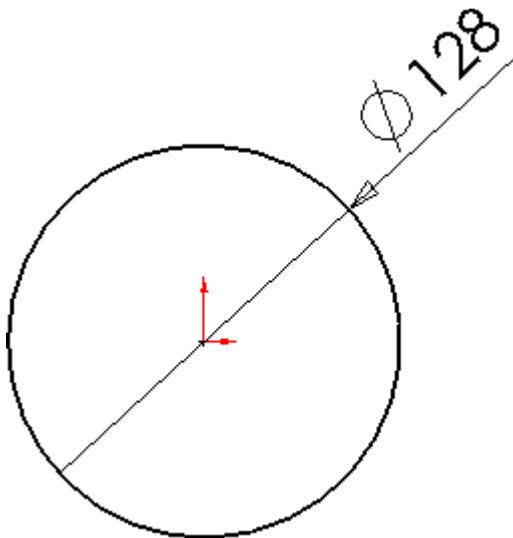
Adding Dimensions

Now add a diameter dimension to the circle.

1. Click **Smart Dimension**  on the Sketch toolbar.
2. Select the circle.

Notice the preview of the diameter dimension.

3. Move the pointer to where you want the dimension and click to add the dimension.
4. In the **Modify** box, type **128**, then click  and click in the graphics area.



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Extruding the Base Feature

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

1. 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.

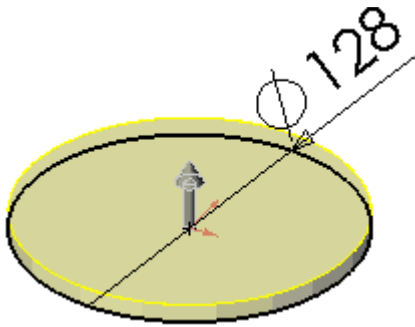
The settings for the extrusion appear in the PropertyManager in the left panel.

2. In the PropertyManager, under **Direction 1**:

- a. Select **Blind** in **End Condition**.

- b. Set **Depth**  to 7.

Notice the shaded preview of the extrusion.




3. Click .

The first feature is complete. **Boss-Extrude1** appears in the FeatureManager design tree in the left panel.

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Saving the Part




1. Click **Save**  on the Standard toolbar.
2. In the dialog box, type **Pressure Plate** for **File name**.
3. Click **Save**.



The extension **.sldprt** is added to the file name, and the file is saved.



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Sketching the Boss

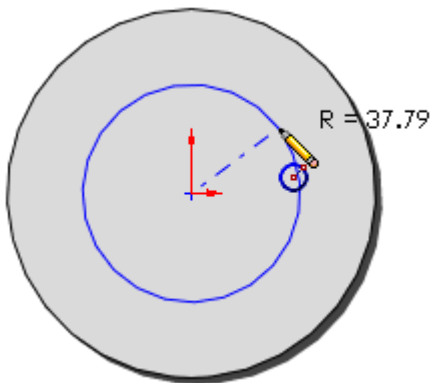
Create a sketch for the ring boss.

1. Click **Extruded Boss/Base**  on the Features toolbar.
2. Select the top face of the part.
3. Click **Top**  on the Standard Views toolbar.
4. Click **Circle**  on the Sketch toolbar.
5. Move the pointer over the origin.

The pointer changes to  .

 The ring boss is concentric with the base cylinder because you started both at the origin. In the completed sketch, the symbol  indicates this relationship.


6. Click to place the center of the circle.
7. Move the pointer to create the circle.
8. Click to finish the circle and click  in the PropertyManager.



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Dimensioning the Boss Sketch



1. Click **Smart Dimension**  on Sketch toolbar.

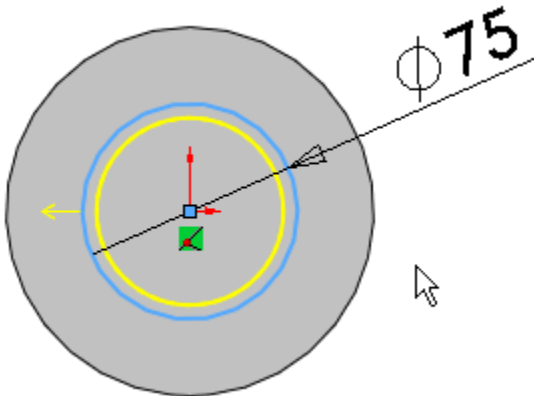
2. Select the circle.
3. Move the pointer and click to place the dimension.
4. In the **Modify** box, type **75**, then click  and click in the graphics area.

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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 .

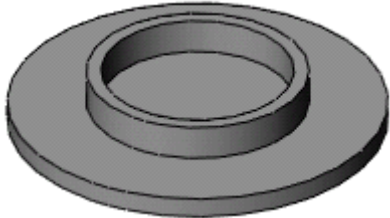
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Extruding the Ring Boss

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 .






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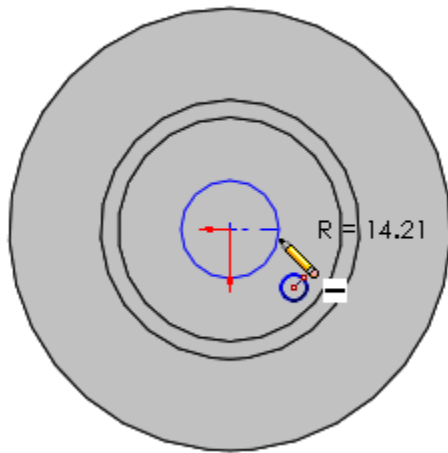
Sketching the Hole

Create a circle for the center hole.

1. Click **Extruded Cut**  on the Features toolbar.
2. Select the top face of the part.
3. Click **Top**  on the Standard Views toolbar.
4. Click **Circle**  on the Sketch toolbar.
5. Move the pointer over the origin.



The pointer changes to  .

6. Click to place the center of the circle.
7. Move the pointer to create the circle.
8. Click to finish the circle and click  in the PropertyManager.



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Dimensioning the Hole Sketch

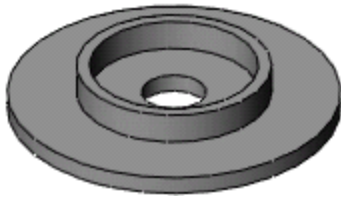
1. Click **Smart Dimension**  on the Sketch toolbar.
2. Select the circle.
3. Move the pointer and click to place the dimension.
4. In the **Modify** box, type **25**, then click  and click in the graphics area.

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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  .





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Creating Holes Using the Hole Wizard

Another tool for creating holes is the Hole Wizard. The following steps give you practice creating Hole Wizard holes. However, because these holes are not part of the model design, you remove them after placement.

1. Click **Top**  (Standard Views toolbar).
2. Click **Hole Wizard**  (Features toolbar).
3. In the PropertyManager, on the **Type** tab, under **Hole Type**, select:

- **Countersink**  for the hole type.
- **Ansi Metric** in **Standard**.
- **Flat Head Screw - ANSI B18.6.7M** in **Type**.

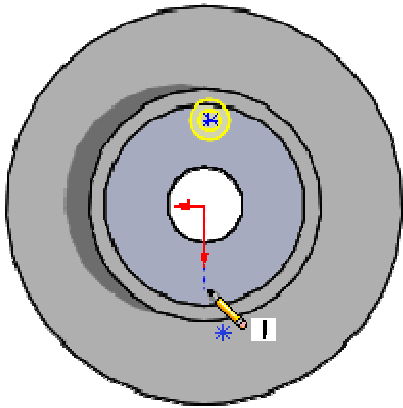
4. Under **Hole Specifications**, select **M6** in **Size**.
5. Under **Options**:
 - a. Select **Head clearance**.
 - b. Set **Head Clearance**  to **1**.
 - c. Select **Added C'Bore** in **Head Clearance Mode** to apply additional head clearance as a 1mm counterbore.
6. Select the **Positions** tab.
7. In the graphics area, click the base cylinder inside the ring boss above the center hole to place a hole.



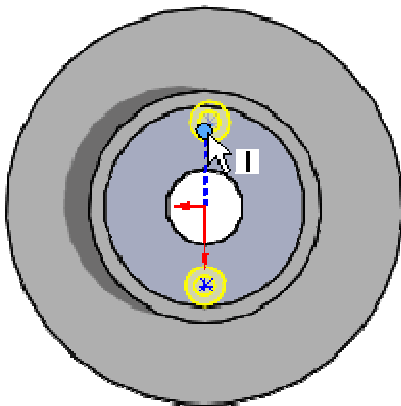
The hole position does not need to be exact. You later move the hole to the centerline.


8. Move the pointer under the center hole so the inference centerline appears, then click to place

another hole.



9. Press **Esc** to stop inserting holes.
10. Drag the first hole onto the inference centerline.





11. Click  to close the **Point** PropertyManager.

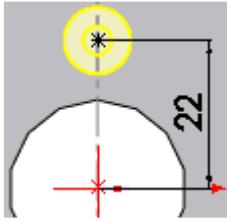
[Next](#) 



Modifying and Dimensioning Hole Wizard Holes

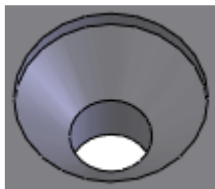
The holes are too large, so you resize them.

1. On the **Type** tab, select **M4** in **Size**.
2. Click **Zoom to Area**  (View toolbar) and zoom in on the two holes.
3. Select the **Positions** tab.
4. Click **Smart Dimension**  (Sketch toolbar).

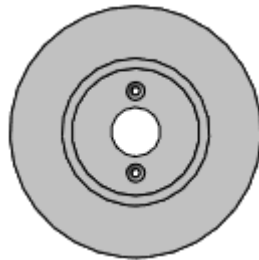
5. Select the center of the top hole and the origin, then click to place the dimension.




6. In the **Modify** dialog box, set the value to **22**, then click .
7. Repeat steps 5 and 6 for the bottom hole.
8. Click  in the **Dimension** and **Hole Position** PropertyManagers.



Single Hole
Wizard hole
close-up






Entire model

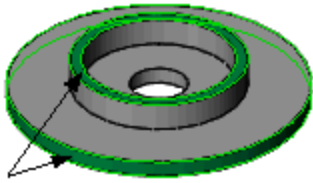
9. In the FeatureManager design tree, right-click **CSK for M4 Flat Head Machine Screw1**  and click **Delete** because it is not part of the model design. Click **Yes** to confirm the deletion.
10. Press **F** to zoom out.

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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 **Items To Fillet**, set **Radius**  to **2**.
4. Select the top face of the ring boss and the outside face of the base.




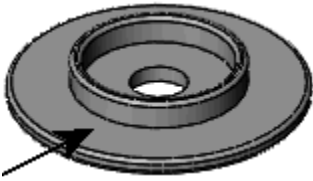
5. Click .



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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.



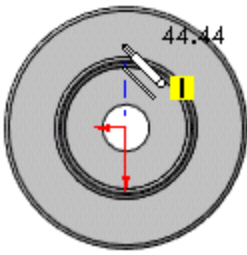
3. Click **Top**  on the Standard Views toolbar.
4. Expand the **Line** flyout menu on the **Sketch** CommandManager and click **Centerline** .

 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. Make the line about 45mm long.
8. Click **Centerline** again to end the line.




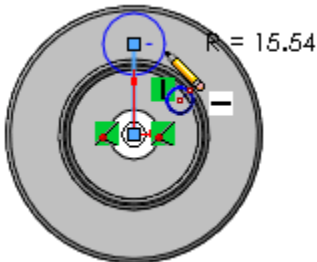
[Next](#) >

Sketching the Tall Cylinder Extrusion (continued)

9. Click **Circle**  on the Sketch toolbar.
10. Move the pointer over the endpoint of the line (not the endpoint by the origin).

The pointer changes to .


11. Click to start the circle.
12. Move the pointer to define the circle and click to finish.
13. Click  in the PropertyManager.





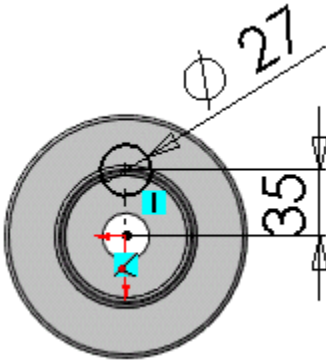
[Next](#) >

Dimensioning the Tall Cylinder Sketch

Add dimensions to the sketch.

1. Click **Smart Dimension**  on the Sketch 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.

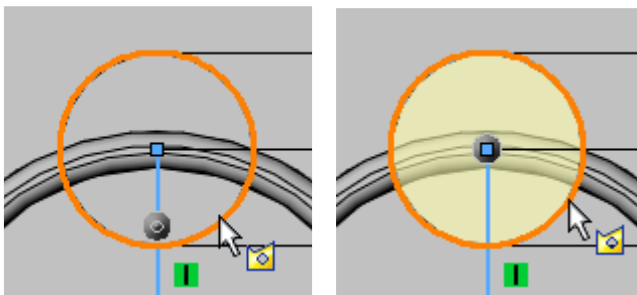


[Next](#) 

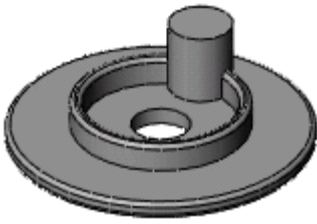
Adding the Tall Cylinder Extrusion

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. 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.

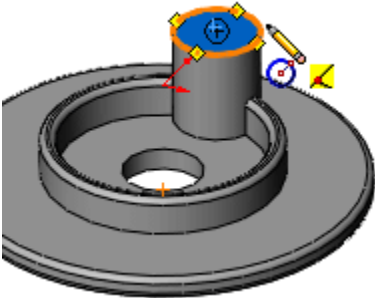



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Sketching the Tall Cylinder Hole

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 .

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Dimensioning the Tall Cylinder Hole Sketch


Add a dimension to control the diameter of the circle.

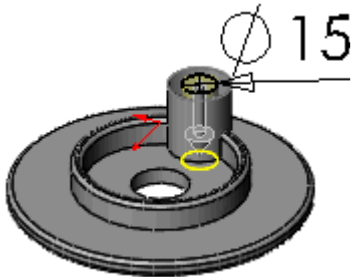
1. Click **Smart Dimension**  on the Sketch toolbar.
2. Select the circle.
3. Move the pointer and click to place the dimension.
4. In the **Modify** box, type **15**, click , and click in the graphics area.

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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 .

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Adding Fillets to the Tall Cylinder

1. Click **Hidden Lines Visible**  on the View toolbar.

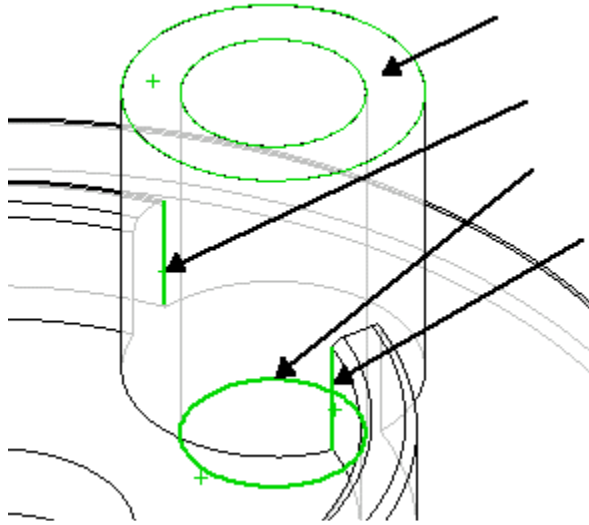
This shows the edges needed for the fillet.

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.

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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.

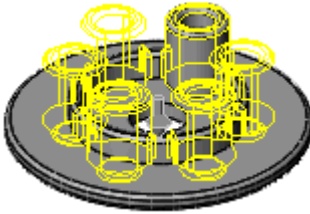
2. On the Features toolbar, expand the **Linear Pattern** flyout toolbar and click **Circular Pattern**



3. In the PropertyManager, under **Parameters**:

- Select the temporary axis in the center of the part for **Pattern Axis**.
- 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**).




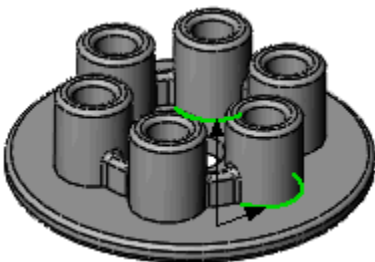
6. Click .



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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.



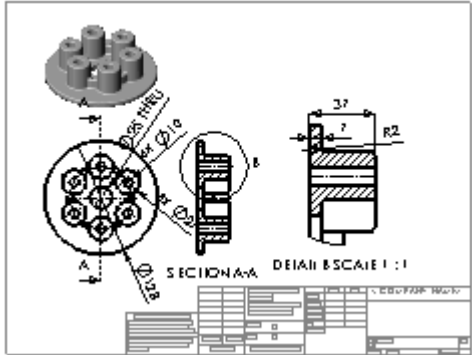
4. Click  to add a 2mm fillet.
5. Click **Save**  on the Standard toolbar.



The part is complete.

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My First Drawing

For your first drawing, you create the drawing shown below. The drawing contains many views, centerlines, center marks, and dimensions.





 If **Pressure Plate.sldprt** is not open from the My First Part lesson, [click here](#)  to open it (or browse to <install_dir>\samples\tutorial\IntroToSW\pressure_plate.sldprt).

[Next](#) 

Creating a New Drawing

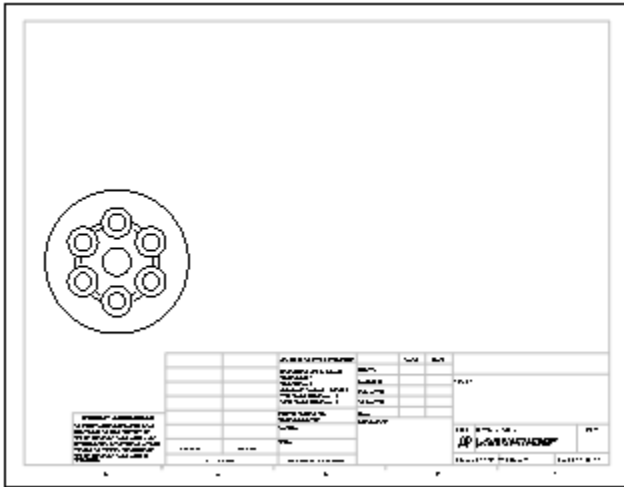
Create a drawing for the Pressure Plate.

1. Click **Options**  on the Standard toolbar.
2. On the **System Options** tab, select **Drawings, Display Style**.
3. Under **Tangent edges in new views**, select **Removed** to hide transition edges between rounded or filleted faces, then click **OK**.
4. On the Menu Bar toolbar, expand the **New** flyout toolbar, and click **Make Drawing from Part/Assembly** .

SolidWorks creates a drawing and begins the process to place a model view.

5. On the **View Palette** tab of the Task Pane:
 - Clear **Auto-start projected view**.
 - Drag the **Top** view into the drawing area.
6. In the PropertyManager:


- Under **Display Style**, click **Hidden Lines Removed** .
- Click .

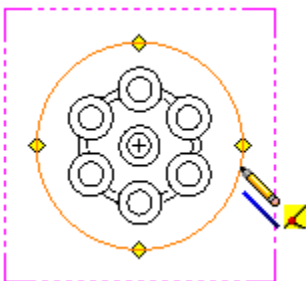


[Next](#) 

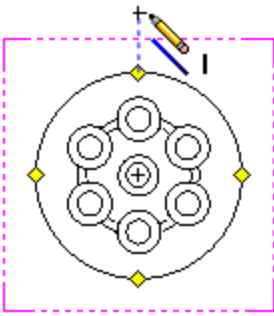
Creating a Section View


Create a section view through the center of the part.

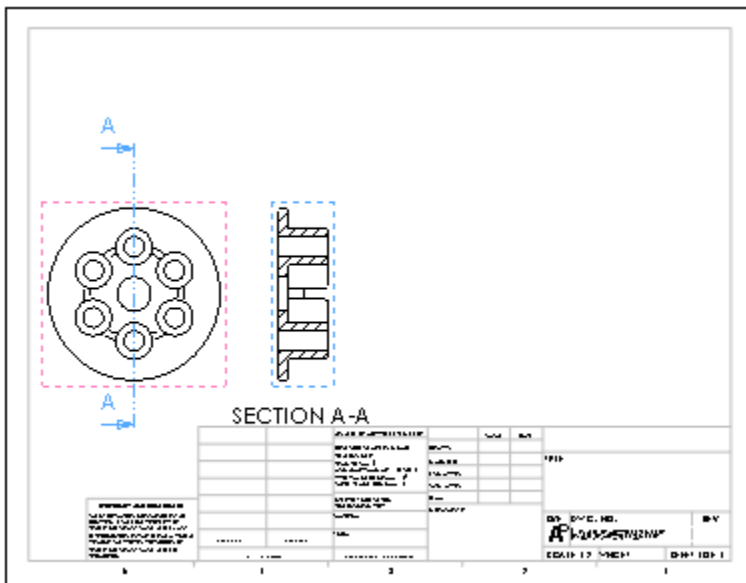
1. Click **Section View**  on the Drawing toolbar.
2. Move the pointer over the outside edge of the pressure plate until the center point appears.



3. Move the pointer above the center point of the plate.



4. Click to start the section line.
5. Move the pointer straight down below the plate.
6. Click to end the section line.
7. Move the pointer to the right to place the view and click to finish.
8. Under **Section Line**, click **Flip direction** to reverse the direction of the section view.
9. Click .



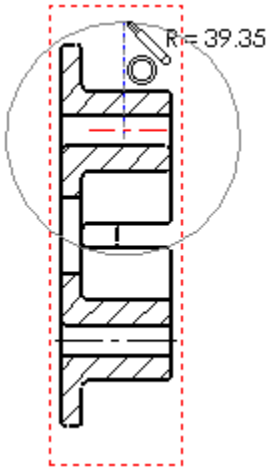
[Next](#) 

Creating a Detail View

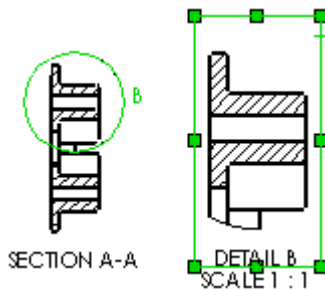
Now make a detail view from the section view.


1. Click **Detail View**  on the Drawing toolbar.

2. Move the pointer over the section view and click to place the center of the detail circle.
3. Move the pointer to define the detail circle and click to finish.



4. Move the pointer to place the detail view and click to add the view.








5. Click **Save**  on the Standard toolbar.
6. Accept the default file name and click **Save**.
7. Click **Yes** if prompted to save referenced models to also save the part.



[Next](#) 

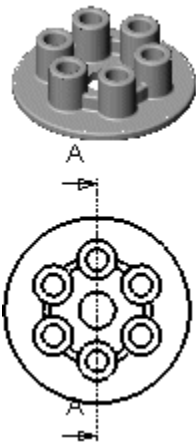
Creating an Isometric View

Create a shaded isometric view.

1. Click **Model View**  on the Drawing toolbar.
2. In the PropertyManager, under **Part/Assembly to Insert**, click **Pressure Plate**.
3. Click **Next** .

4. Under **Orientation**, click ***Isometric** .
5. Under **Display Style**, click **Shaded** .
6. Move the pointer to place the view.
7. Click .


 If the view is not in the correct location on the drawing sheet, you can drag the view. Move your pointer over the view until the pointer includes , then drag the view.

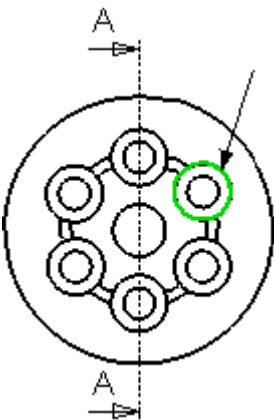




[Next](#) 

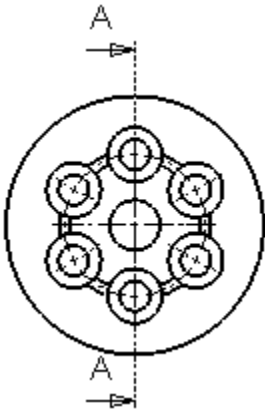
Adding Center Marks

Add center marks to the top view of the model.

1. Click **Center Mark**  on the Annotation toolbar.
2. In the Top view, click the outside edge of one of the tall bosses in the pattern as shown.




3. Click  in the graphics area to propagate the center marks to all the other holes in the pattern.
4. Click .

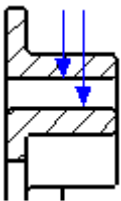


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Adding Centerlines

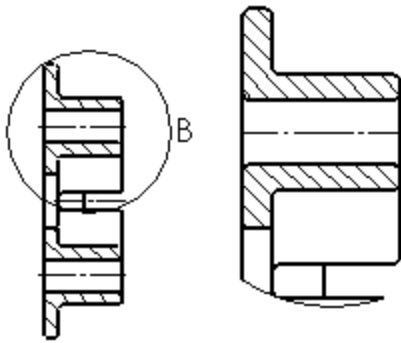
Add centerlines to the section view and detail view.

1. Click **Centerline**  on the Annotation toolbar.
2. In the section view, select the two edges of the top hole.



A centerline appears between the two edges.



3. Repeat for other holes in the section and detail views to add three more centerlines.
4. Click .



[Next](#) 

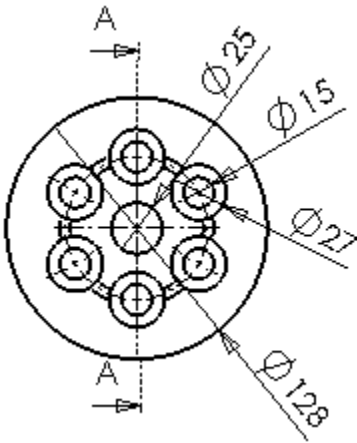
Placing Dimensions

Place some dimensions on the drawing.

1. Click **Options**  on the Standard toolbar.
2. On the **Document Properties** tab, select **Dimensions**.
3. Clear **Add parentheses by default** to display reference dimensions in drawings without parentheses, then click **OK**.
4. Click **Smart Dimension**  on the Annotation toolbar.
5. Move the pointer to the outside edge of the top view and click.
6. Move the pointer to place the dimension and click.

The diameter dimension **128** appears.

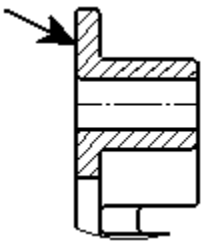
7. Add the three other diameter dimensions:
 - a. Select the edge of the circle.
 - b. Move the pointer to place the dimensions and click.



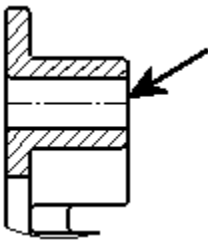
[Next](#) >

Placing Detail Dimensions (continued)

8. Move to the detail view and select the left edge of the plate.



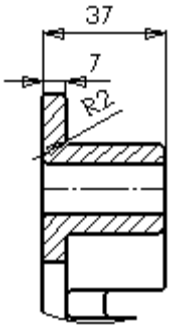
9. Select the right edge of the plate.



10. Move the pointer to place the dimension and click.


The length dimension **37** appears.

11. Place the two other dimensions on the detail view.




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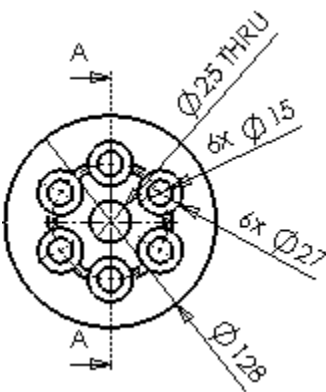
Modifying the Text of Dimensions

1. Select the cylinder boss diameter (27) dimension.
2. In the PropertyManager, under **Dimension Text**, click in front of <MOD-DIAM>.
3. Type **6x**, then click .

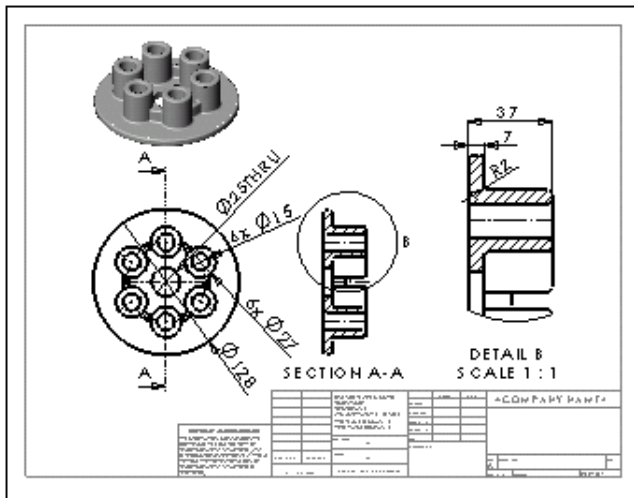
The dimension now indicates that there are six cylinders of the same size in the drawing.

4. Repeat steps 1 through 3 for the cylinder boss hole diameter (15) dimension.
5. Select the center hole diameter (25) dimension.
6. In the PropertyManager, under **Dimension Text**, click after <DIM>.
7. Type **THRU**, then click .

The dimension now indicates a through all cut.



This completes the drawing.

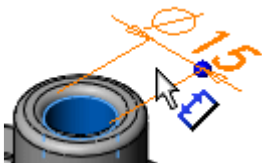


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Modifying the Value of Dimensions

The drawing and part are linked. If one file changes, so does the other.

1. Click **Window, Tile Horizontally** to display the part and drawing windows simultaneously.
2. In the FeatureManager design tree of the part window, click the second **Cut-Extrude** feature, the tall cylinder hole.
3. Move the pointer over the diameter dimensions until it includes the Smart Dimension indicator.



4. Click to edit the dimension.




5. Type **10**, then click in the graphics area.

The original hole and the patterned instances update to reflect the new size.

6. Click in the drawing window.

Notice that all four drawing views and the dimensions update to reflect the new size.

7. In the drawing, click **Save**  to save your work.
8. Click **Yes** when prompted to save referenced models to also save the part.

Congratulations! You have completed this lesson.