

SOLIDWORKS 2016

Learn by doing-Part 1



Tutorial Books

SOLIDWORKS 2016 Learn by doing Tutorial Books

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INTRODUCTION

SOLIDWORKS is a package adding a great value to enterprises. It offers a set of tools, which are easy-to-use to design, document and simulate 3D models. Using this software, you can speed up the design process and reduce the product development costs.

This book provides a systematic approach for users to learn SOLIDWORKS. It is aimed at new SOLIDWORKS users. However, users of previous versions of SOLIDWORKS may also find this book useful for them to learn the new enhancements. This book guides the user from starting a SOLIDWORKS 2016 session to constructing parts, assemblies, and drawings. Each chapter has components explained with the help of real world models.

Scope of this book

If you are a student or engineer who is interested to learn SOLIDWORKS 2016 for mechanical design, then this book helps you to do it easily and quickly. You will also enjoy the learning process by creating real world models. In addition, if you are a graphical learner, then you will find this book very interesting.

The topics in this book include Getting Started with SOLIDWORKS 2016, Basic Part Modeling, Creating Assemblies, Creating Drawings, Additional Modeling Tools, and Sheet Metal Modeling.

Chapter 1 introduces SOLIDWORKS. The user interface and terminology are discussed in this chapter.

Chapter 2 takes you through the creation of your first SOLIDWORKS model. You create simple parts.

Chapter 3 teaches you to create assemblies. It explains the Top-down and Bottom-up approaches for designing an assembly. You create an assembly using the Bottom-up approach.

Chapter 4 teaches you to create drawings of the models created in the earlier

chapters. You will also learn to place exploded views, and part list of an assembly.

Chapter 5: In this chapter, you will learn sketch tools.

Chapter 6: In this chapter, you will learn additional modeling tools to create complex models.

Chapter 7: introduces you to Sheet Metal modeling. You will create a sheet metal part using the tools available in the Sheet Metal environment.

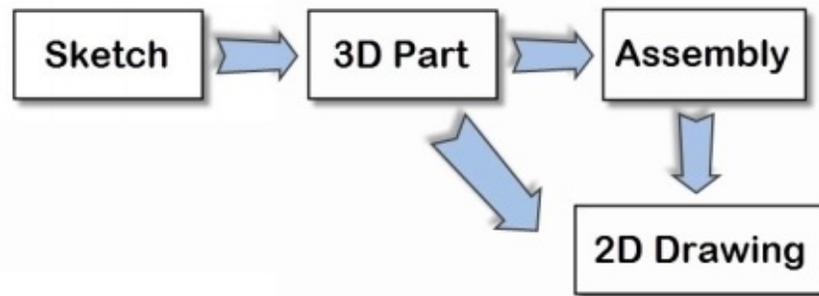
Chapter 8: In this chapter, you will learn to create assemblies using Top-down approach and apply some advanced mates.

Chapter 9: In this chapter, you will learn to add annotations to drawings.

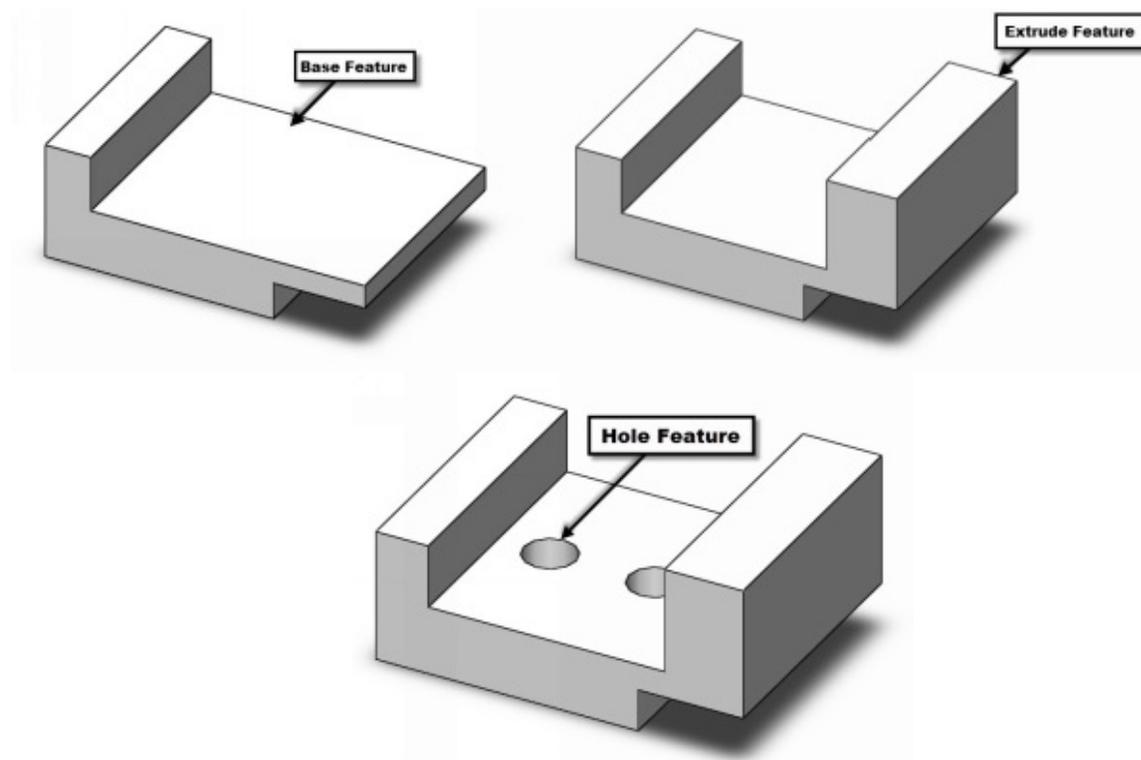
Chapter 1: Getting Started

In this chapter, you will learn some of the most commonly used features of SOLIDWORKS. In addition, you will learn about the user interface.

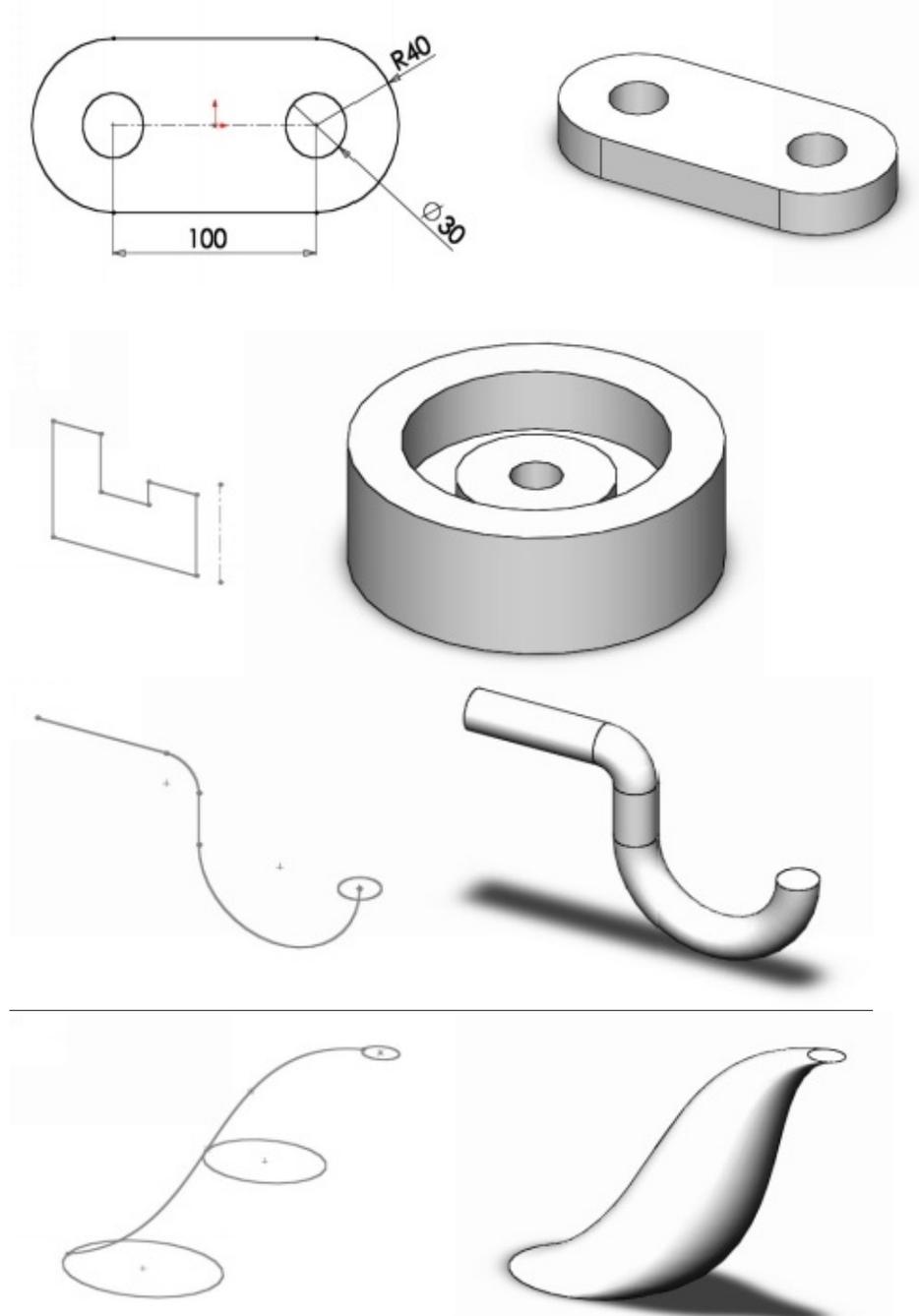
SOLIDWORKS is a parametric and feature-based system that allows you to create 3D parts, assemblies, and 2D drawings. The following figure shows the design process in SOLIDWORKS.



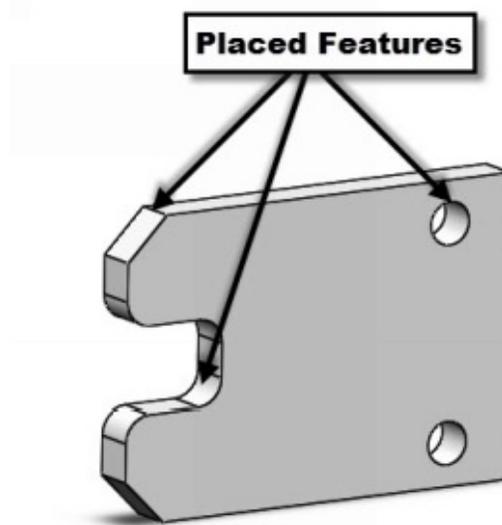
In SOLIDWORKS, you create parts by combining features.



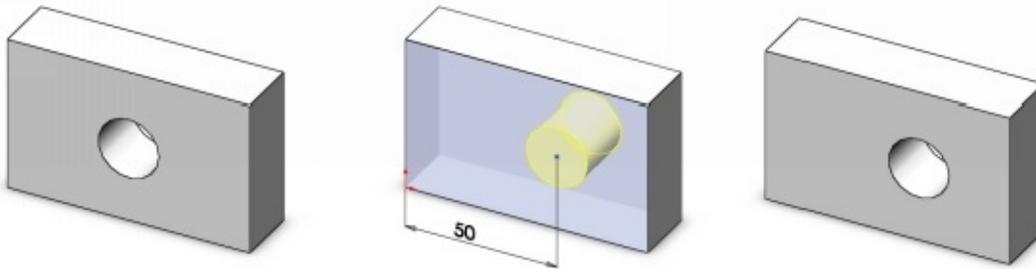
Features are of two types: sketch-based and placed. You create a sketch based feature by drawing a sketch and extruding, revolving, sweeping along a path, or lofting it.



You create placed features by simply placing them onto an existing feature. Examples of placed features are fillets, chamfers, drafts, shells, holes and so on.

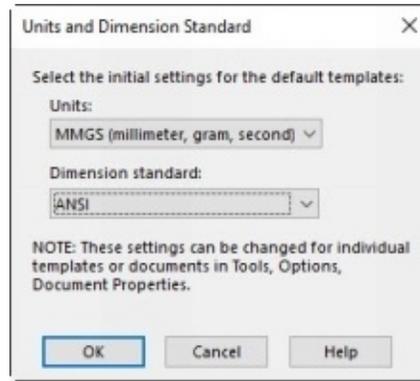


In SOLIDWORKS, parameters, dimensions, or relations control everything. For example, if you want to change the position of the hole shown in figure, you need to change the dimension or relation that controls its position.

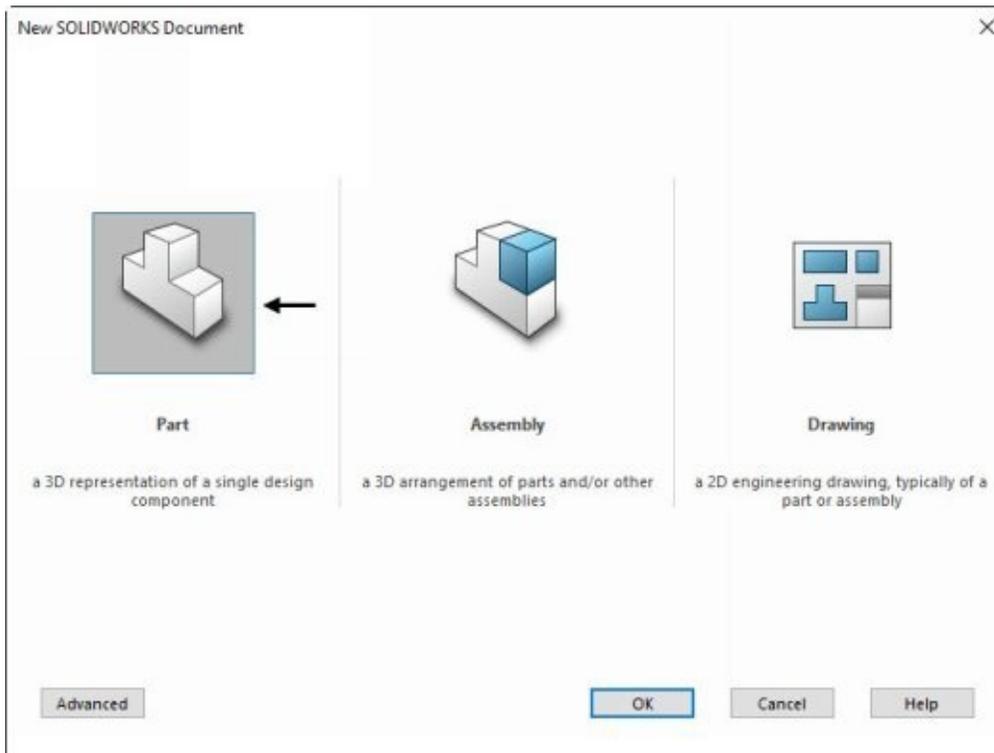


Starting SOLIDWORKS

- Click the **Start** button on the Windows taskbar.
- Click **All apps**.
- Click **SOLIDWORKS 2016 > SOLIDWORKS 2016**.
- Click the **New** button; the **Units and Dimension Standard** dialog appears.
- On the **Units and Dimension Standard** dialog, select **MMGS (millimeter, gram, second)** and **ANSI** from the **Units and Dimension Standard** drop-downs, respectively.

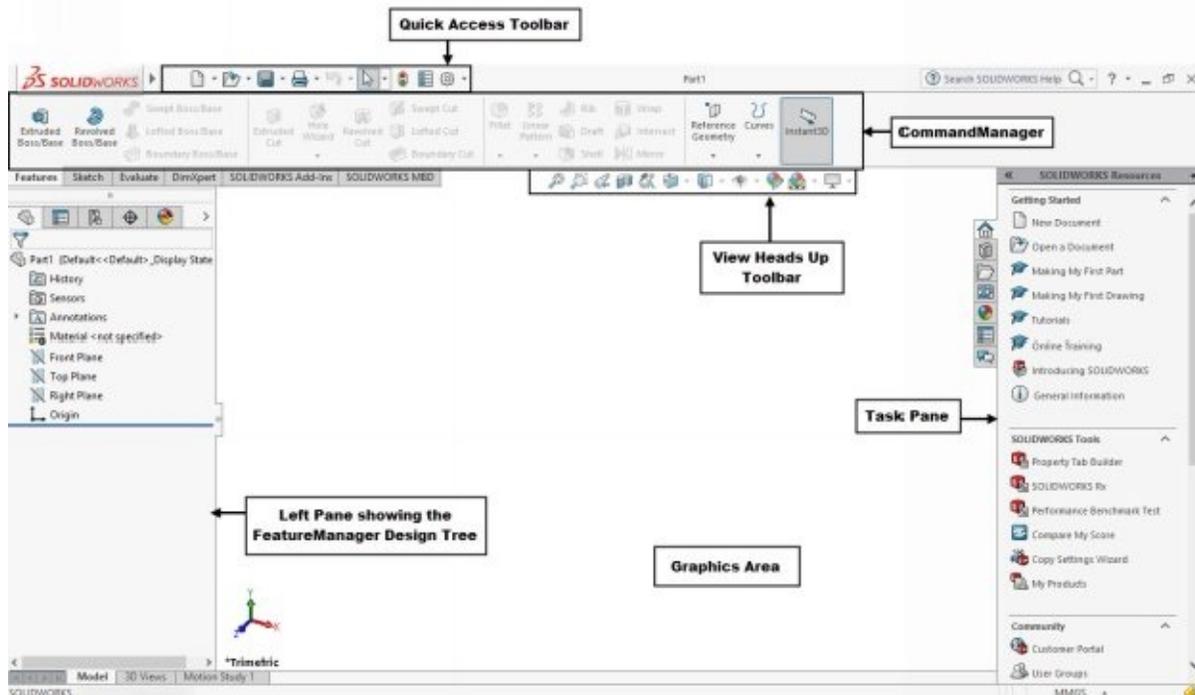


- Click **OK** on the **Units and Dimension Standard** drop-down
- On the **New SOLIDWORKS Document** dialog, click **Part**.



- Click **OK** button

Notice these important features of the SOLIDWORKS window.



User Interface

The following sections explain various components of the user interface.

Quick Access Toolbar

This is available at the top of the window. It contains the tools such as **New**, **Save**, **Open** and so on.



Menu

Menu is located at the top of the window. You can display it by placing the pointer on the arrow located at the top left corner. It has various options (menu titles). When you click on a menu title, a drop-down appears. Select any option from this drop-down.



CommandManager

CommandManager is located below the menu. It consists of various tabs. When you click on a tab, a set of tools appear. You can select the required tool from the tab. The following sections explain the CommandManager tabs available in SOLIDWORKS.

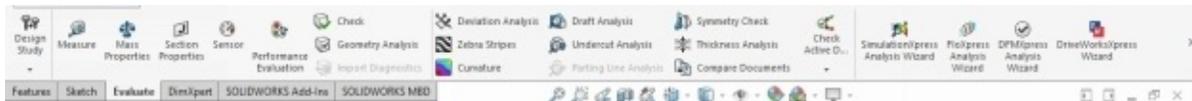
Features tab

This CommandManager tab contains the tools to create 3D features.



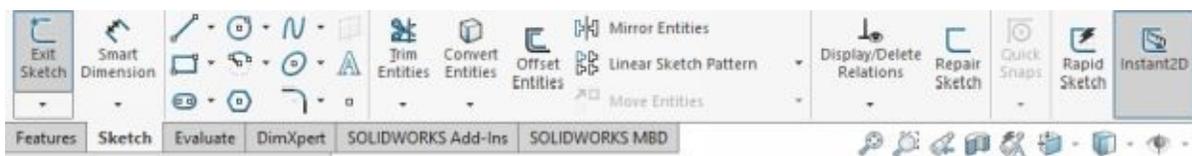
Evaluate tab

This CommandManager tab has the tools to measure the objects. It also has analysis tools to analyze the draft, curvature, surface and so on.



Sketch tab

This tab contains all the sketch tools.



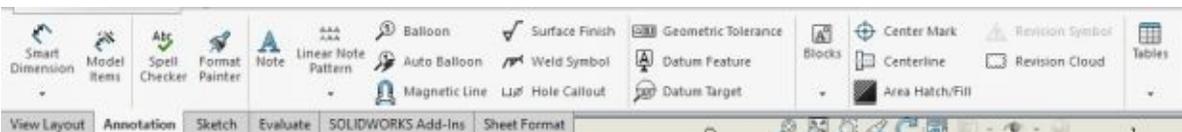
Assemble tab

This CommandManager tab contains the tools to create an assembly. This tab is available in an assembly document.



Drawing Environment tabs

In the Drawing Environment, you can create orthographic views of the 3D model. The CommandManager tabs in this environment contain tools to create 2D drawings.



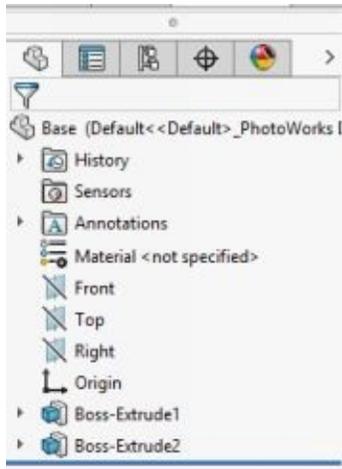
Sheet Metal tab

Use the tools in this tab to create sheet metal components. Note that this tab is not displayed by default. You can display it by right-clicking on any of the CommandManager tabs and selecting Sheet Metal from the menu.



FeatureManager Design Tree

This is located on the left pane of the application window. It contains the list of operations carried in a SOLIDWORKS document.



Status bar

This is available below the graphics window. It displays status of the SOLIDWORKS document. For example, if you are drawing a sketch, it displays **Editing sketch**.

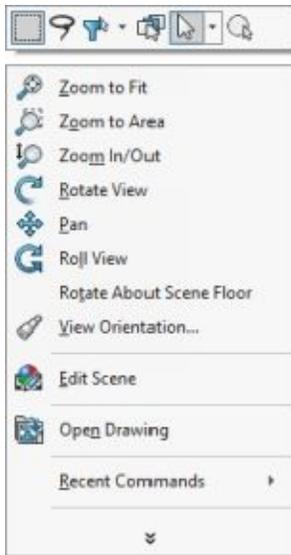
View (Heads Up) Toolbar

This is located at the top of the graphics window. It contains the tools to modify the display of the model.



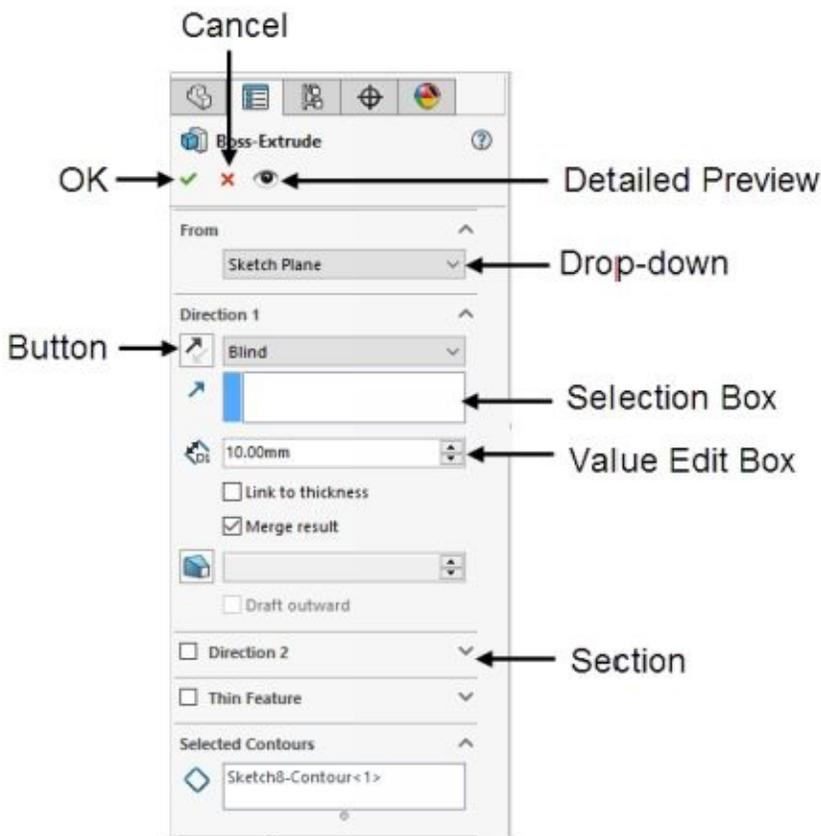
Shortcut Menus

When you click the right mouse button, a shortcut menu appears. A shortcut menu contains a list of some important options that can accessed very quickly.



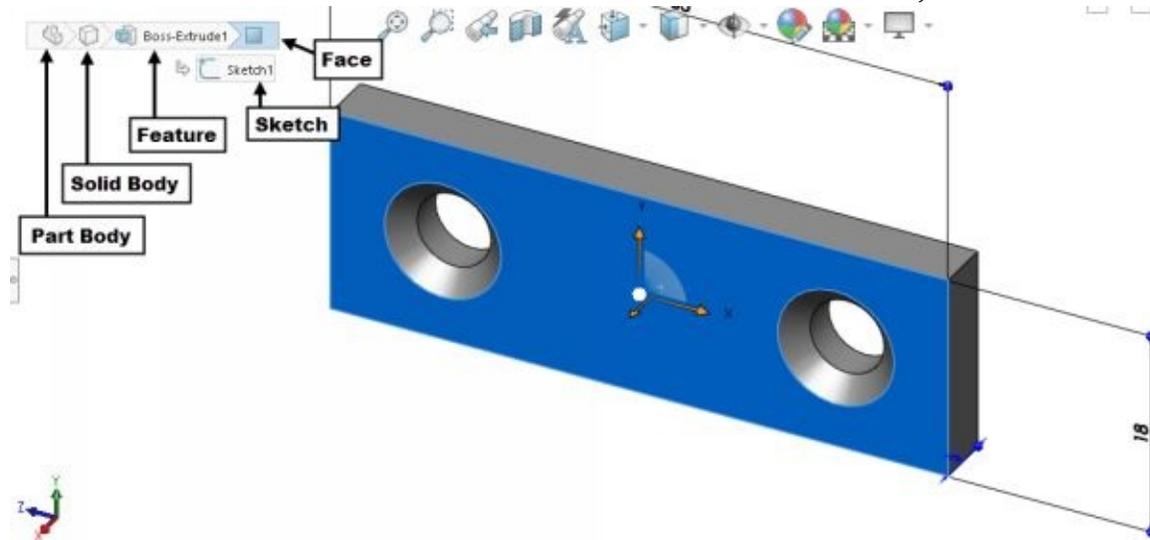
PropertyManager

When you activate any tool in SOLIDWORKS, the PropertyManager related to it appears. It consists of various options that are part of the selected tool. The following figure shows the components of the PropertyManager.



Breadcrumbs

Breadcrumbs is a new functionality added to **SOLIDWORKS 2016**. Whenever you click on an item or a feature of the model geometry, the **Breadcrumbs** toolbar appears at the top left corner of the graphics window. For example, click on a face of the part geometry, as shown; the Selection Breadcrumbs toolbar appears, as shown. On the toolbar, you can select the entire part, solid body, feature or sketch related to the feature, or a face.



Select the feature from the Breadcrumbs toolbar and notice the options to edit the feature, suppress, hide, zoom, and so on. You can hide the left pane and use the Breadcrumbs toolbar to edit model geometry.



Mouse Functions

Various functions of the mouse buttons are:

Left Mouse button (MB1)

You can select an object, tool, or option using clicking the left mouse button. When you double-click the left mouse button (MB1) on an object, the

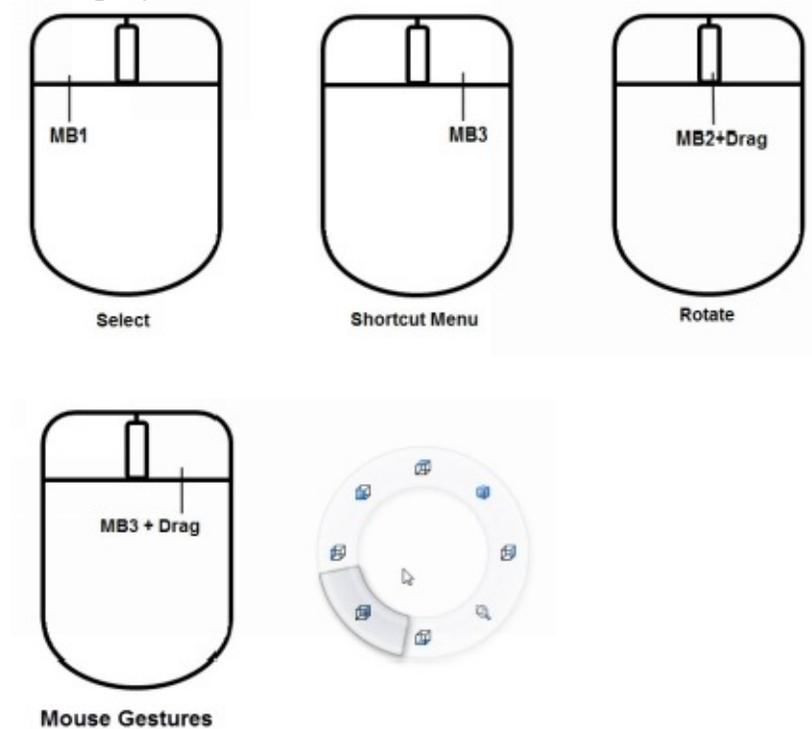
dimensions related to the object appear. You can then edit the dimensions of the objects.

Middle Mouse button (MB2)

Click and drag this button to rotate the model view.

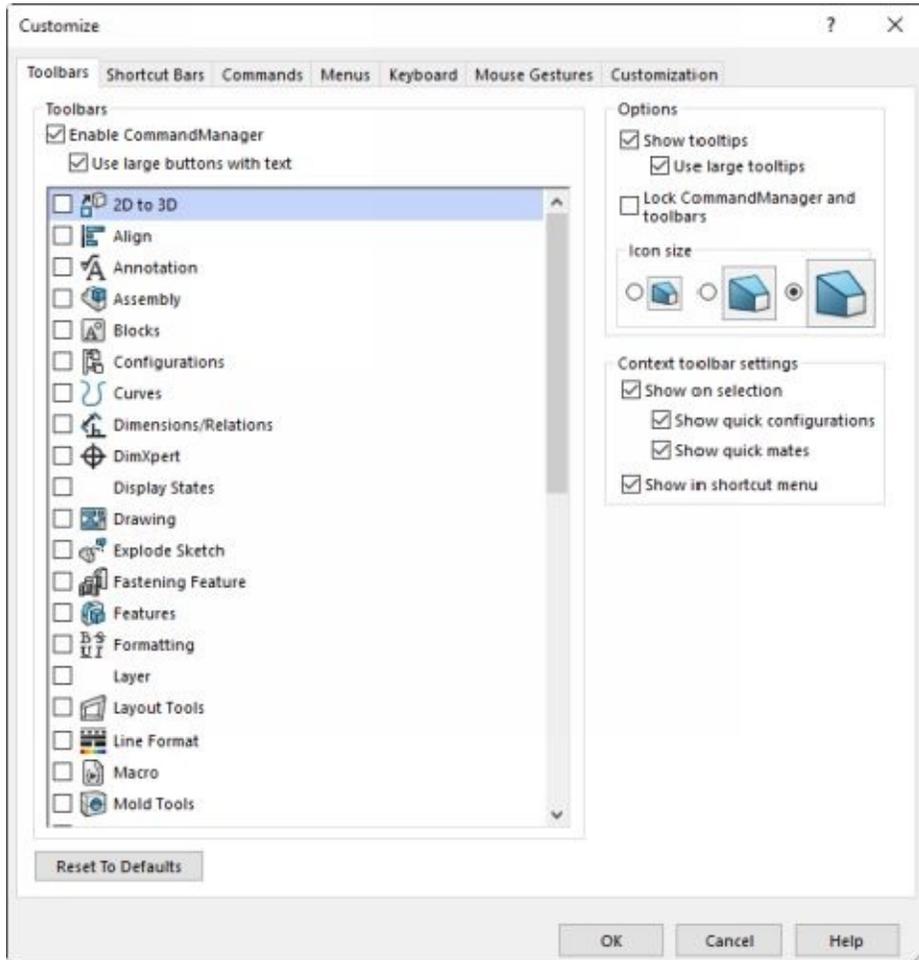
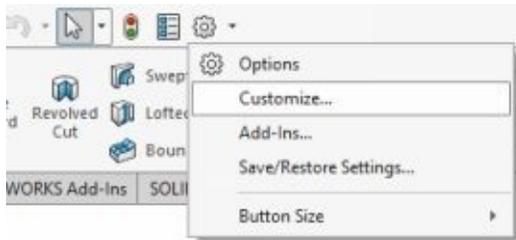
Right Mouse button (MB3)

Click this button to open a shortcut menu. Click and drag the right mouse button to display the



Customizing the CommandManager, Shortcut Bars, and Menus

To customize the CommandManager, shortcut keys, or menus, click **Options > Customize** on the **Quick Access Toolbar**. On the **Customize** dialog, use the tabs to customize the CommandManager or menu, or shortcut bars.



Chapter 2: Modeling Basics

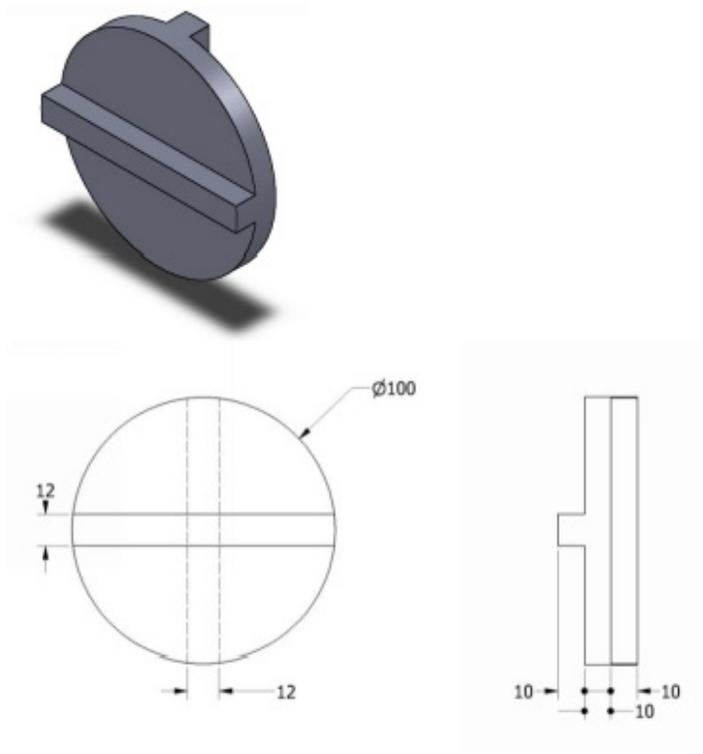
This chapter teaches you to construct your first SOLIDWORKS model. You will construct parts of an Oldham Coupling:

In this chapter, you will:

- Construct Sketches
- Construct a base feature
- Add another feature to it
- Construct revolved features
- Construct cylindrical features and
- Apply draft

TUTORIAL 1

This tutorial takes you through the creation of your first SOLIDWORKS model. You construct the Disc of the Oldham coupling assembly:

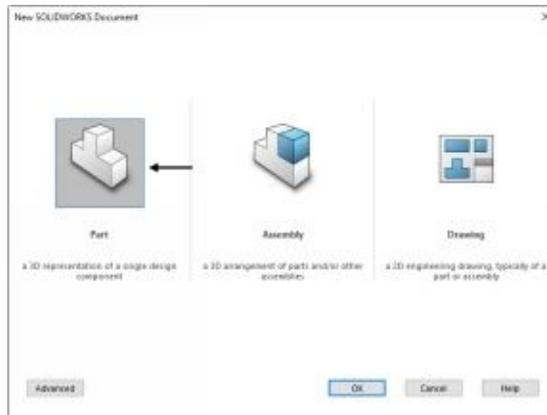


Opening a New Part File

1. To open a new part, click the **New** button on the **Quick Access Toolbar**.



2. On the **New SOLIDWORKS Document** dialog, click **Part**, and then click **OK**.

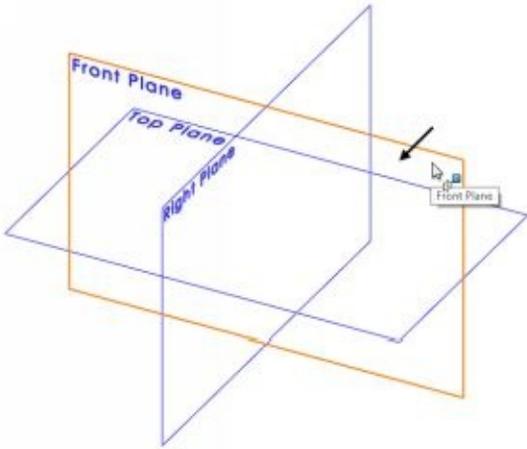


Starting a Sketch

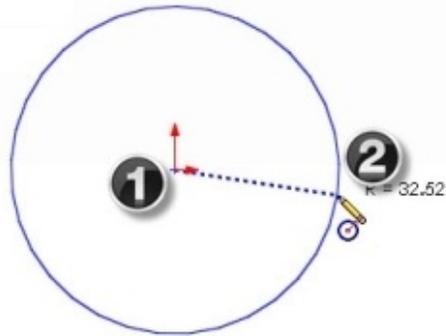
1. To start a new sketch, click **Sketch** > **Sketch** on the CommandManager.



2. Left click on the front plane.



3. Click **Sketch** > **Circle**  on the CommandManager to start creating a circle.
4. Select the origin point of the coordinate system.
5. Move the pointer away from the origin and click to draw the circle.



6. Press **ESC** to deactivate the tool.

Adding Dimensions

In this section, you will specify the size of the sketched circle by adding dimensions.

As you add dimensions, the sketch can attain any one of the following three

states:

Fully Constrained sketch: In a fully constrained sketch, the positions of all the entities are fully described by dimensions, relations, or both. In a fully constrained sketch, all the entities are black color.

Under Constrained sketch: To define the geometry completely, you need additional dimensions, relations, or both. In this state, you can drag the sketch elements to modify the sketch. An under constrained sketch element is in blue color.

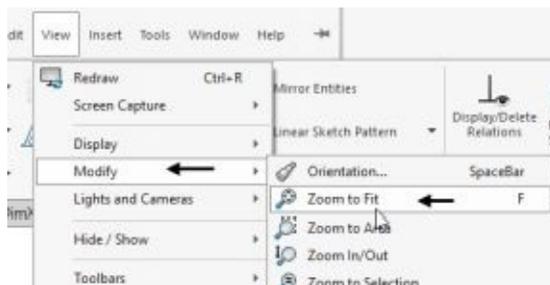
Over Constrained sketch: In this state, an object has conflicting dimensions, relations, or both. An over constraining dimension is in yellow color.

1. Click **Sketch > Smart Dimension**  on the CommandManager to start dimensioning the sketch.
2. Select the circle and click; the **Modify** box appears.
3. Type-in **100** in the box and press **Enter** on the keyboard.
4. Press **Esc** to deactivate the **Smart Dimension** tool.
5. To display the entire circle at full size and to center it in the graphics area, use one of the following methods:

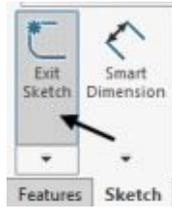
- On the **View (Heads Up)** toolbar, click **Zoom to Fit**.



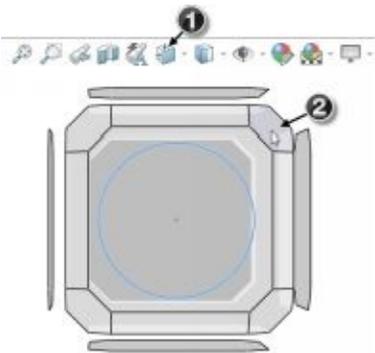
- On the Menu bar, click **View > Modify > Zoom to Fit**.



6. On the CommandManager, click **Sketch > Exit Sketch**.



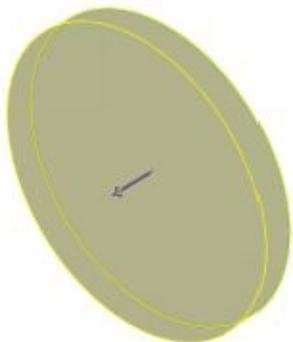
7. On the **View (Heads Up)** toolbar, click **View Orientation**.
8. Click on the top-right corner of the **View Selector** cube.



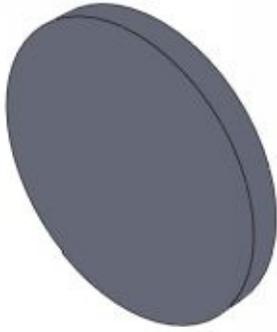
Constructing the Base Feature

You can call the first feature of any part as a base feature. You now construct this feature by extruding the sketched circle.

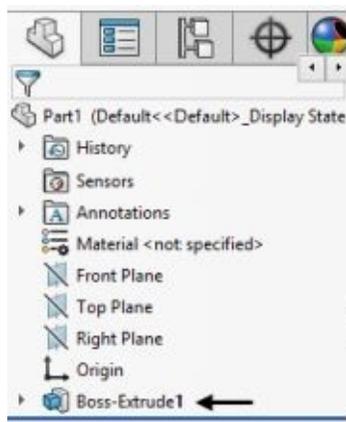
1. On the CommandManager, click **Features** > **Extruded Boss/Base** .
2. Click on the sketched circle.



3. On the **Boss-Extrude** PropertyManager, type-in **10** in the **Depth** box, and then click **OK** .



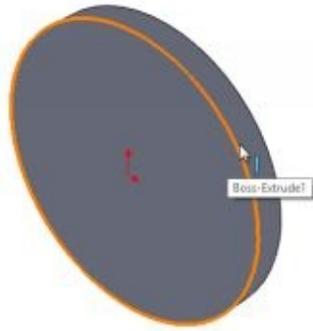
Notice the new feature, **Boss-Extrude1**, in the **FeatureManager Design Tree**.



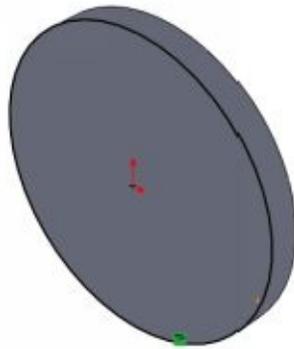
Adding an Extruded Feature

To construct additional features on the part, you must draw sketches on the model faces or planes, and then convert them into features.

1. On the CommandManager, click **Features > Extruded Boss/Base** .
2. Click on the front face of the model.
3. On the CommandManager, click **Sketch > Convert Entities** .
4. Click on the circular edge of the model geometry.

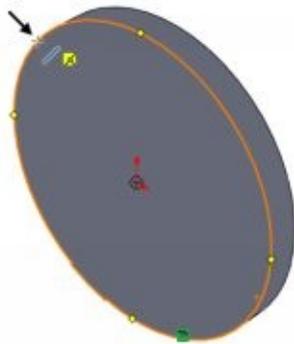


5. On the **Convert Entities** PropertyManager, click **OK** ✓.

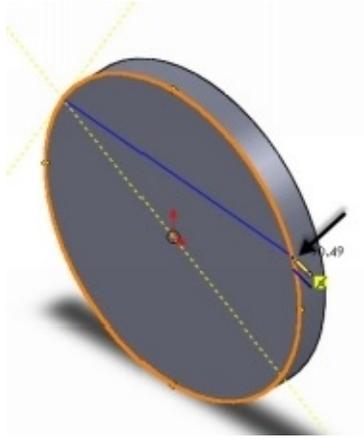


6. On the CommandManager, click **Sketch** > **Line** .

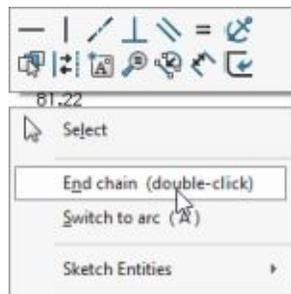
7. Click on the circular edge to specify the first point of the line.



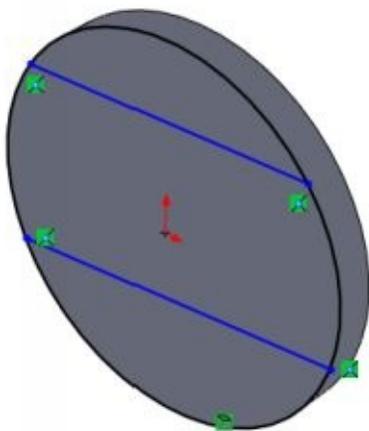
8. Move the pointer towards right.
9. Click on the other side of the circular edge; a line is drawn. In addition, another line is attached to the pointer.



10. Right-click and select **End chain** to end the line.



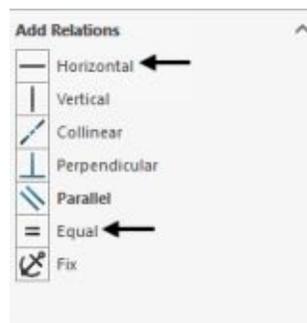
11. Draw another line below the previous line.



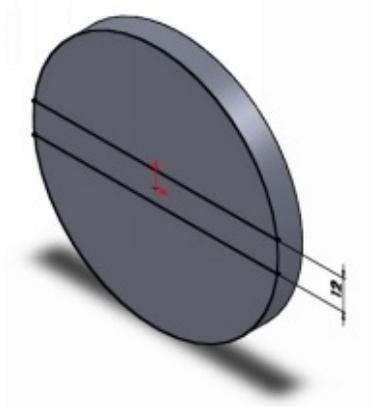
9. Press **Esc** to deactivate the **Line** tool.
10. On the CommandManager, click **Sketch > Display/Delete Relations > Add Relation**.



12. Select the two lines.
13. On the **Add Relations** PropertyManager, under the **Add Relations** section, click **Horizontal** to make the two lines horizontal.
14. Under the **Add Relations** section, click **Equal** to make the lines equal in length.



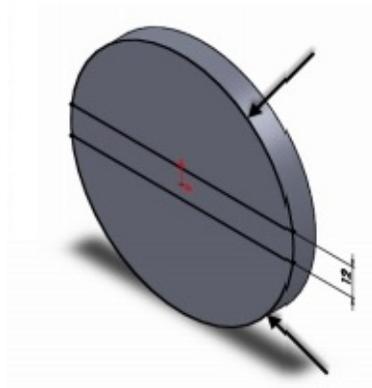
15. Click **OK**  on the PropertyManager.
16. On the CommandManager, click **Sketch > Smart Dimension** .
17. Select the two horizontal lines.
18. Move the pointer toward right and click to locate the dimension.
19. Type-in **12** in the Modify box and click **OK**.



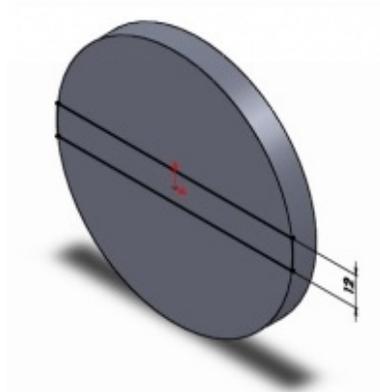
20. On the CommandManager, click **Sketch** > **Trim Entities** .
21. Click the **Trim to Closest** icon on the PropertyManager.



22. Click on the arc portions of the projected entity.



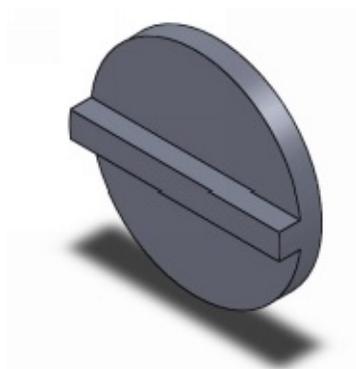
The projected entities are trimmed.



23. Click **OK**  on the PropertyManager.
24. On the top right corner of the graphics window, select **Exit Sketch**.

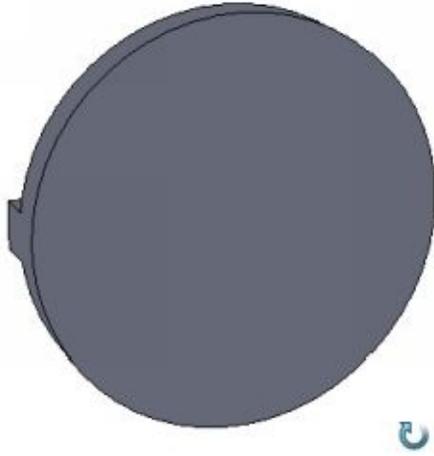


25. On the **Boss-Extrude** PropertyManager, type-in **10** in the **Depth** box, and then click **OK** .

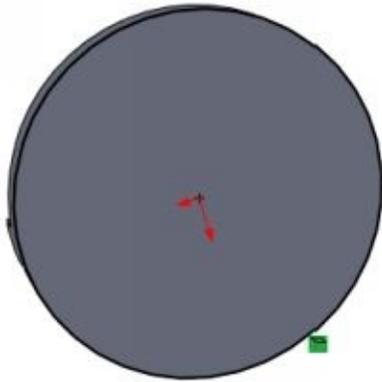


Adding another Extruded Feature

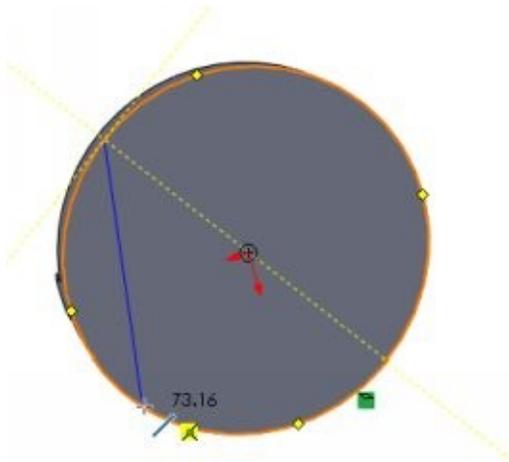
1. Press and hold the Middle Mouse Button and drag the pointer to rotate the part such that the back face of the part is displayed.

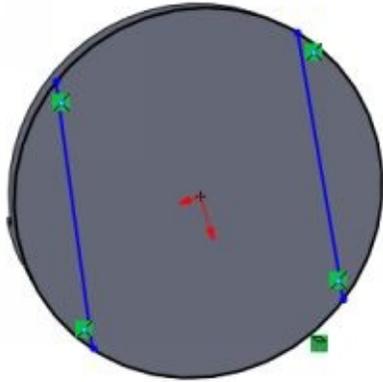


2. On the CommandManager, click **Sketch** > **Sketch** .
3. Click on the back face of the part.
4. Activate the **Convert Entities**  tool and convert the circular edge into a sketch entity.

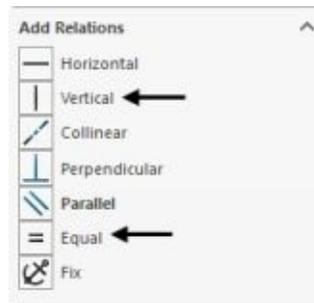


5. Draw two lines, as shown below.

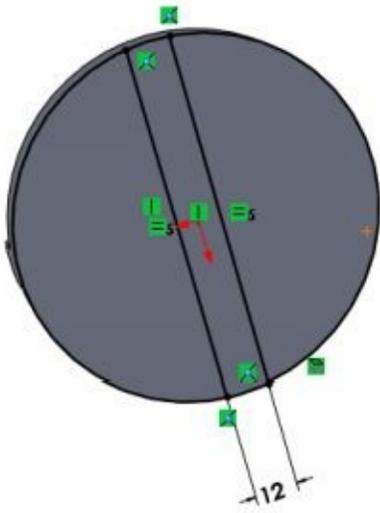




6. Press Esc to deactivate the **Line** tool.
7. Press the Ctrl key and click on the lines.
8. On the PropertyManager, under the **Add Relations** section, click **Vertical**.
9. Click **Equal**, and then click **OK** on the PropertyManager.



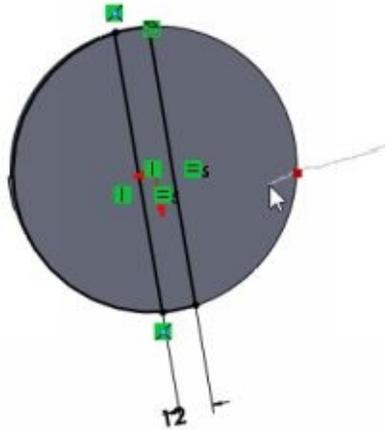
10. Activate the **Smart Dimension**  tool and apply a dimension of 12 mm distance between the vertical lines.



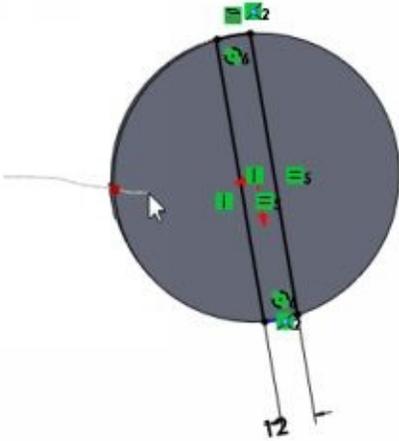
11. Activate the **Trim Entities**  tool.
12. Click the **Power Trim** icon on the PropertyManager.



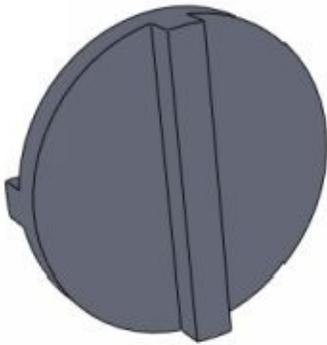
13. Click the left mouse button and drag the pointer across the right side arc; the arc is trimmed.



14. Release the left mouse button.
15. Likewise, trim the left side arc by dragging the mouse across it.

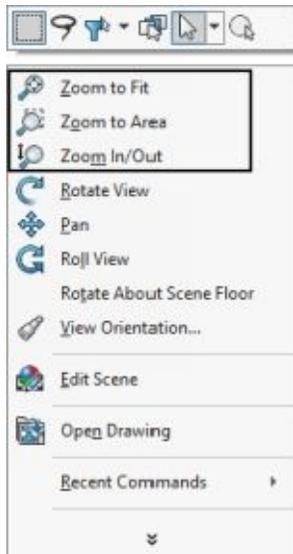
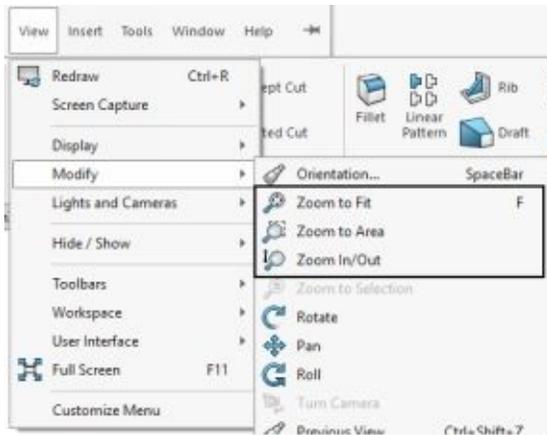


16. Click **Exit Sketch**.
17. Extrude the sketch up to 10 mm depth.



To magnify a model in the graphics area, you can use the zoom tools available on the **View (Heads Up)** toolbar or **View > Modify** menu or on the Shortcut menu.





Click **Zoom to Fit**  to display the part full size in the current window.

Click **Zoom to Area** , and then drag the pointer to construct a rectangle; the area in the rectangle zooms to fill the window.

Click **Zoom In/Out** , and then drag the pointer. Dragging up zooms in; dragging down zooms out.

Click on a face, edge, vertex or any object, and then click **Zoom to Selection**  on the Context Toolbar. You will zoom to the selection.

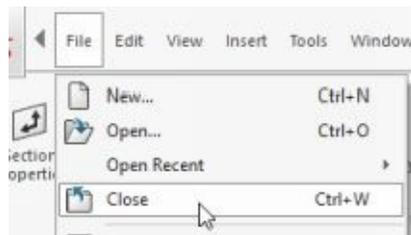
To move the part view, click the right mouse button and select **Pan** .

Drag the part to move it around in the graphics area.

Select **Rotate About Scene Floor** from the right-click shortcut menu helps you to rotate the view about axis perpendicular to the scene floor. After enabling this option, you can use the **Roll View**  option to rotate the view.

Saving the Part

1. Click the **Save**  button on the **Quick Access Toolbar**.
2. Enter **Disc** in the **File name** field.
3. Browse to **C:\Users\Username\Documents** folder, and then create a new folder with the name Oldham Coupling.
4. Click **Save** to save the file.
5. Click **File > Close**.

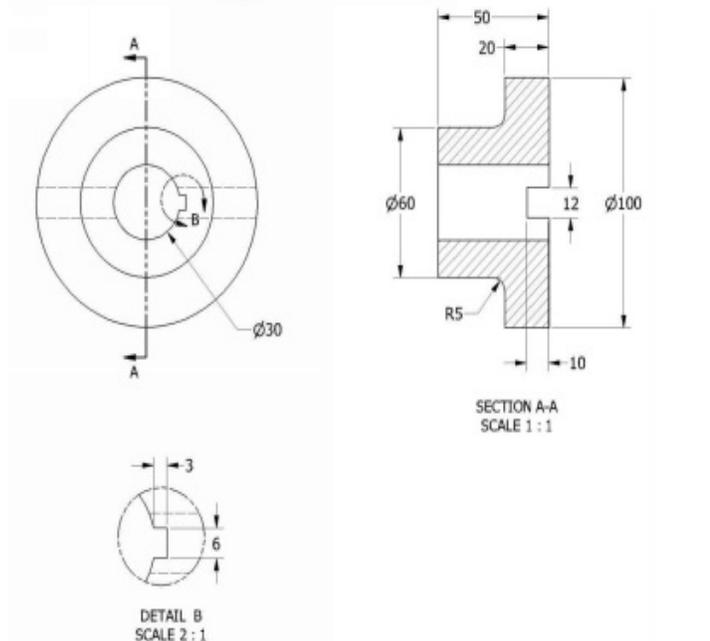
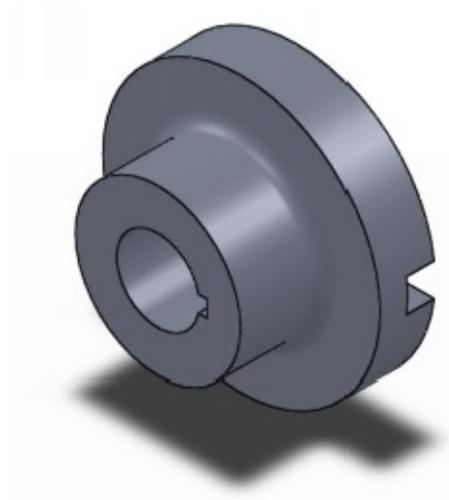


Note: *.prt* or *.sldprt* are the file extensions for the files that are created in the Part environment of SOLIDWORKS.

TUTORIAL 2

In this tutorial, you will construct a flange by performing the following operations:

- Constructing revolved feature
- Constructing cut features
- Adding Rounds



Opening a New Part File

1. To open a new part, click the **New**  button on the **Quick Access Toolbar**.
2. On the **New SOLIDWORKS Document** dialog, click **Part**, and then click **OK**.

Sketching a Revolve Profile

Now, you will construct a sketch for the revolved feature.

1. On the CommandManager, click **Sketch** > **Sketch** .
2. Click on the right plane.

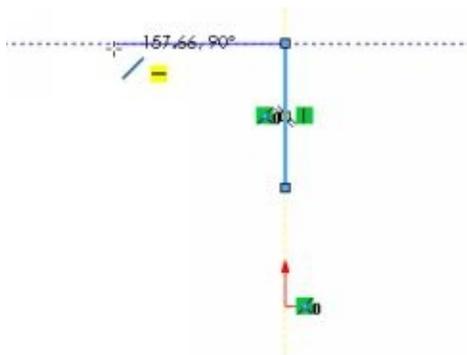
3. Activate the **Line** tool and click somewhere above the vertical axis of the origin.



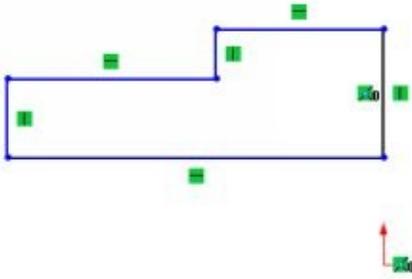
4. Move the pointer upwards. You will notice that the Vertical relation symbol appears.
5. Click to draw a vertical line.



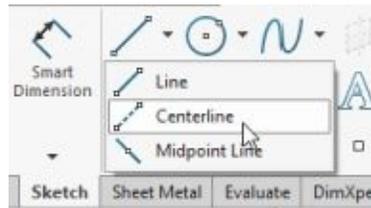
6. Move the pointer toward left. You will notice that the Horizontal relation symbol appears.
7. Click to draw a horizontal line.



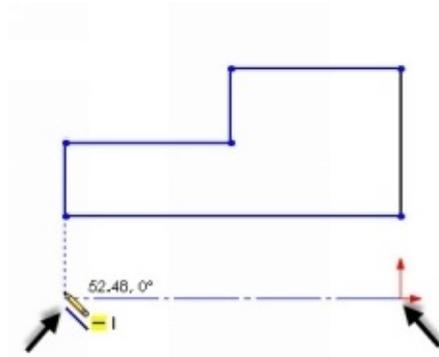
8. Complete the sketch by constructing the other lines.



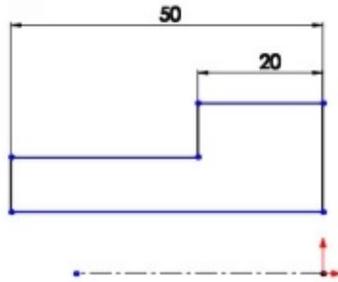
9. On the CommandManager, click **Sketch** > **Line** > **Centerline**.



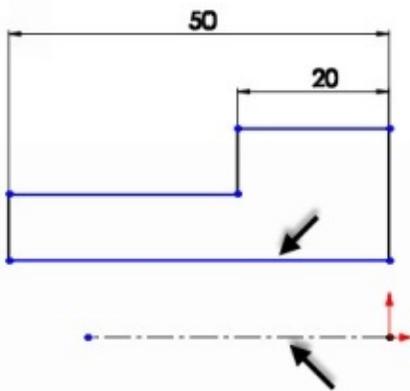
10. Draw a horizontal line passing through the horizontal axis of the coordinate system.



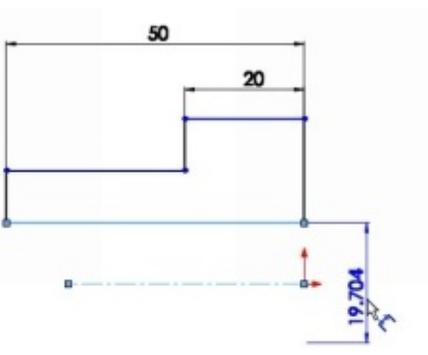
6. Activate the **Smart Dimension**  tool and apply to two horizontal dimensions.



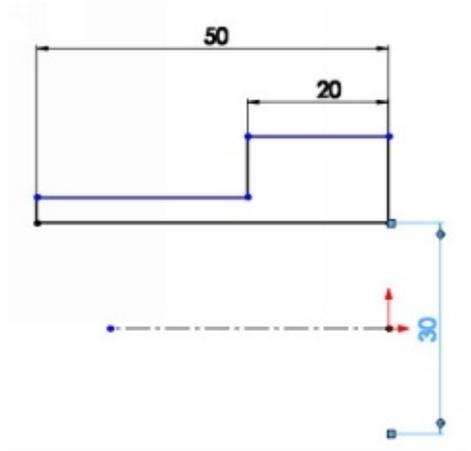
7. Click on the centerline and bottom horizontal line.



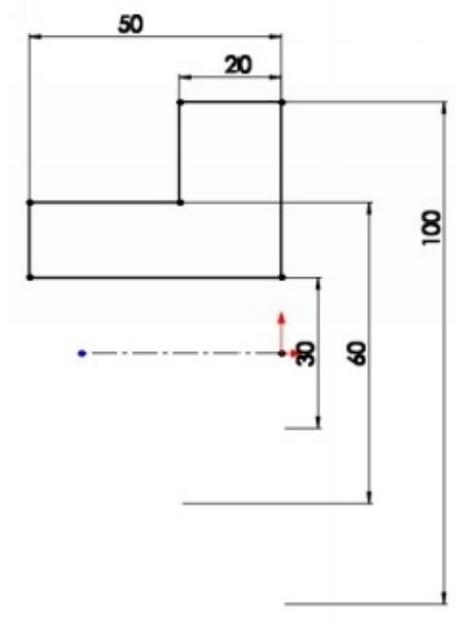
7. Move the pointer downwards right. Click to position the dimension.



8. Type-in 30 in the **Modify** box and click **OK**.



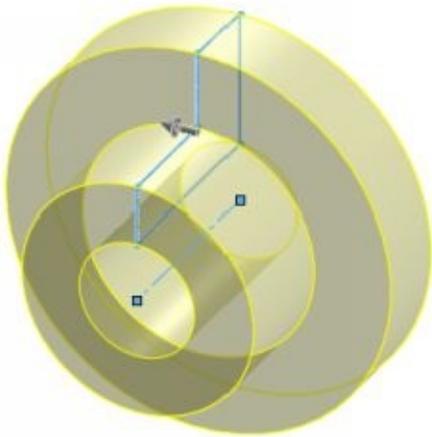
9. Create other symmetric dimensions, as shown.



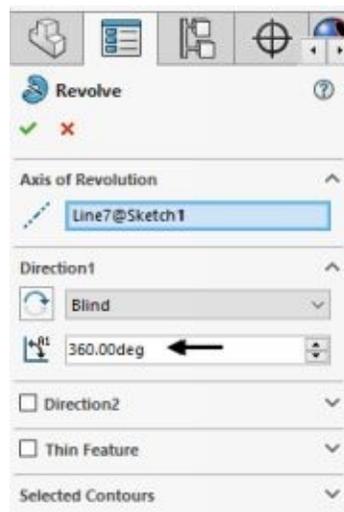
10. Click **Exit Sketch** .

Constructing the Revolved Boss

1. On the CommandManager, click **Features** > **Revolved Boss/Base** .



2. On the **Revolve** PropertyManager, set the **Direction1 Angle** value to **360**.

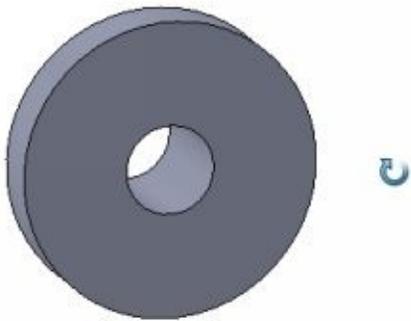


3. Click **OK** ✓.

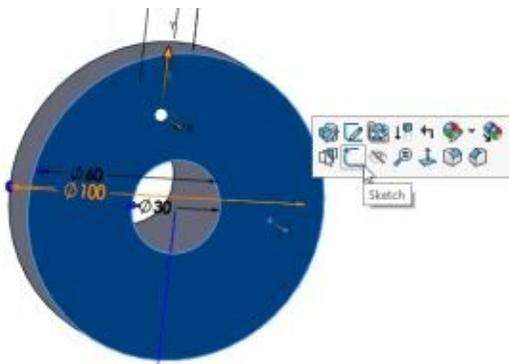


Constructing the Cut feature

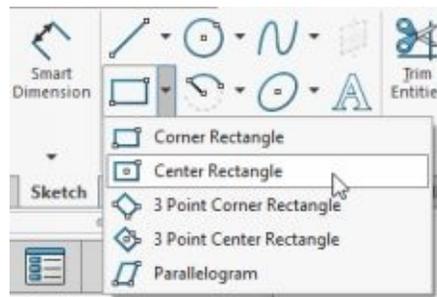
1. Press and hold the Middle Mouse Button and drag the pointer to rotate the part.



2. Click on the back face and click **Sketch**.

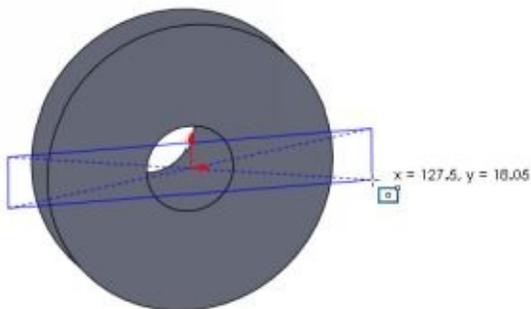


3. On the CommandManager, click **Sketch** > **Rectangle** drop-down > **Center Rectangle**.

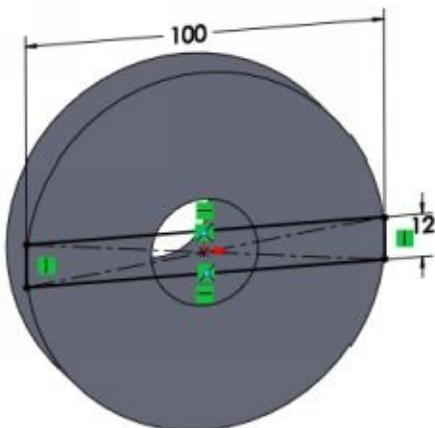


4. Select the origin to define the center point.

11. Move the pointer diagonally toward right and click to create the rectangle.

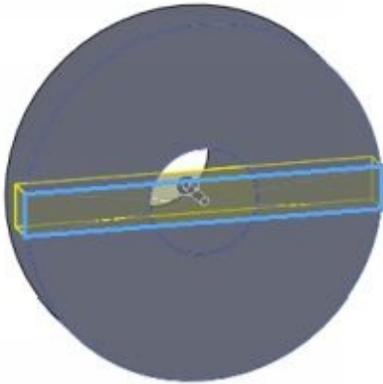


5. Add dimensions to the sketch.

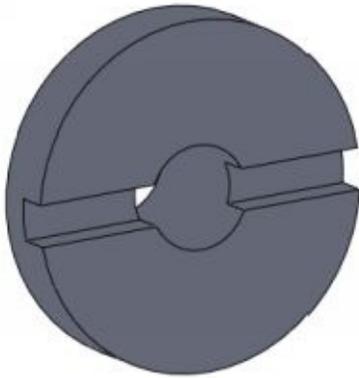


6. Click **Exit Sketch**.

7. On the CommandManager, click **Features** > **Extruded Cut** .

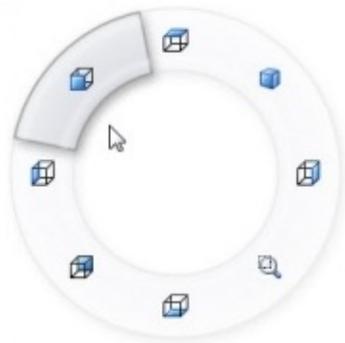


8. On the **Cut-Extrude** PropertyManager, set the **Depth** value to 10 and click **OK** ✓.



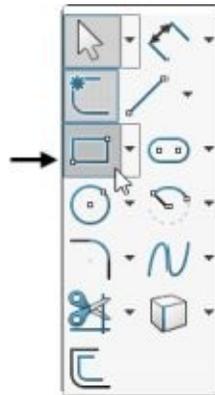
Constructing another Cut feature

1. Press the right mouse button and drag the mouse to display the Mouse gesture.
2. On the Mouse gesture, select Front.

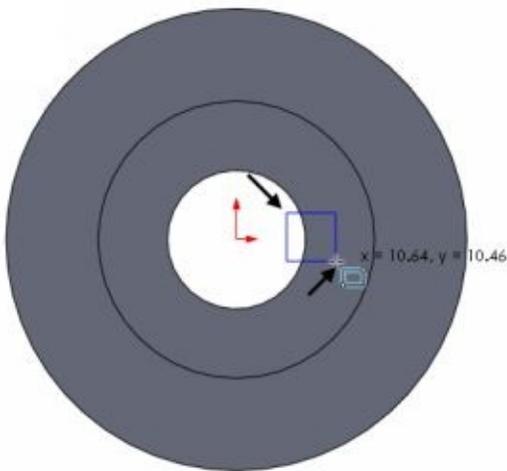


3. On the CommandManager, click **Sketch** > **Sketch**  and click on the front face.

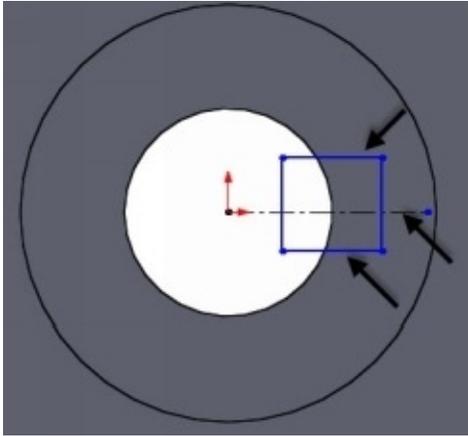
4. Press **S** on the keyboard.
5. On the Flyout Toolbar, click **Rectangle**.



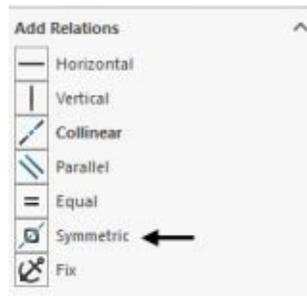
6. On the PropertyManager, select **Corner Rectangle** .
7. Specify the corners of the rectangle.



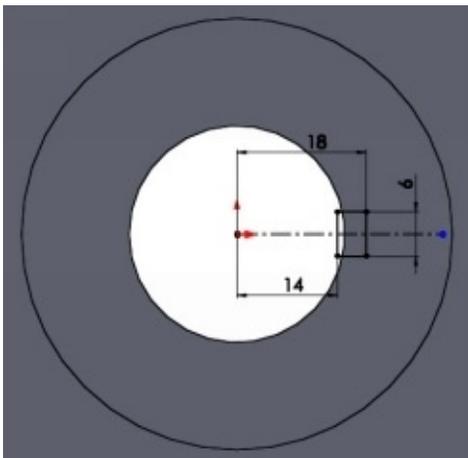
8. On the CommandManager, click **Sketch** > **Line** > **Centerline** .
9. Draw a horizontal centerline passing through the origin.
10. Press the Ctrl key and click on the horizontal lines of the sketch.



11. On the PropertyManager, under the **Add Relations** section, click **Symmetric**.

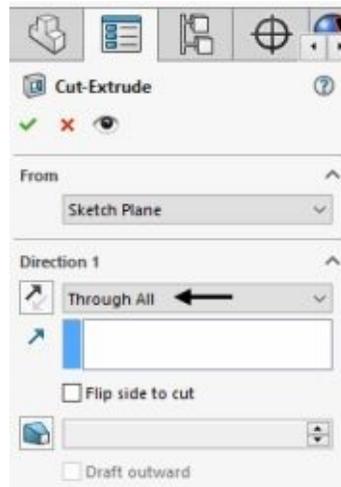


12. Click **OK** ✓.
13. Add dimensions to the sketch.



14. Click **Exit Sketch**.

15. On the CommandManager, click **Features** > **Extruded Cut** .
16. Click on the sketch.
17. On the **Cut-Extrude** PropertyManager, under the **Direction 1** section, select **End Condition** > **Through All**.

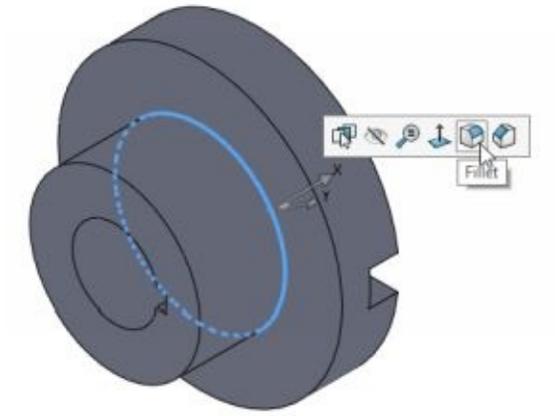


18. Click **OK**  to construct the cutout.
19. Press Ctrl+7 on your keyboard to change the view to isometric.

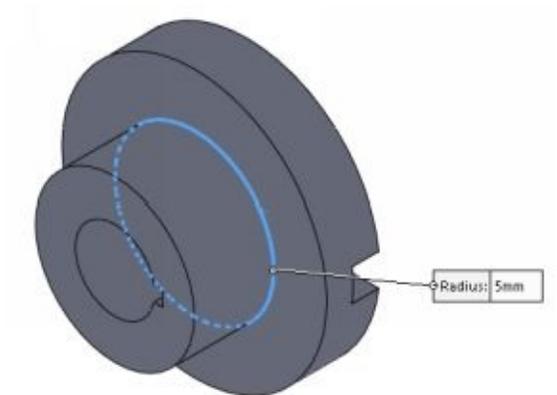


Adding a Fillet

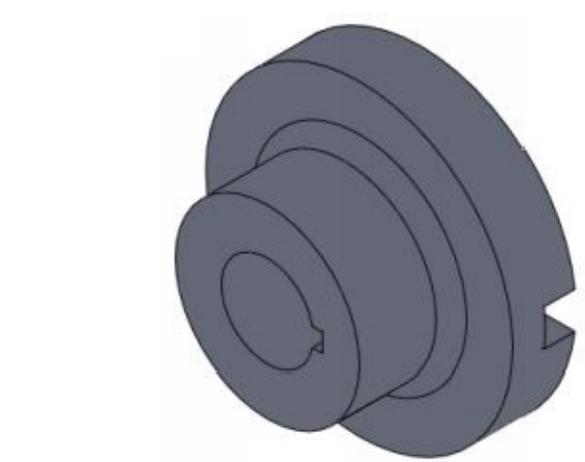
1. Click on the circular edge of the model geometry and select **Fillet**.



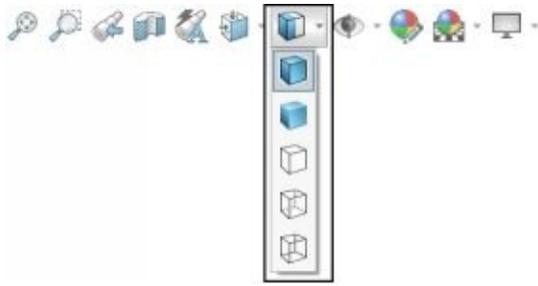
2. Enter **5** in the callout attached to the selected edge.



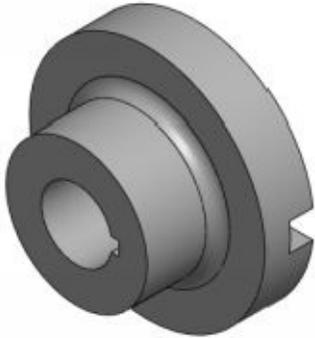
3. Click **OK**  to add the fillet.



To display the part in different modes, use the options on the **Display Style** menu on the **View (Heads Up)** toolbar.



Shaded With Edges



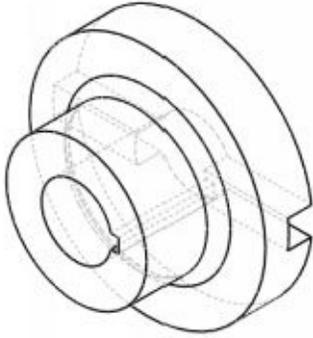
Shaded



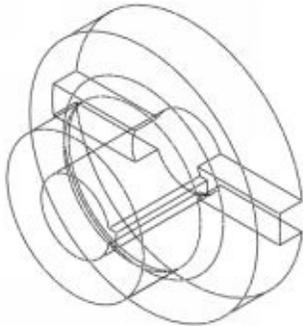
Hidden Lines Removed



Hidden Lines Vissible



Wireframe



The default display mode for parts and assemblies is **Shaded With Edges**. You may change the display mode whenever you want.

Saving the Part

1. Click the **Save**  button on the **Quick Access Toolbar**.
2. Enter **Flange** in the **File name** field.
3. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
4. Click **Save** to save the file.
5. Click **File > Close**.

TUTORIAL 3

In this tutorial, you will construct a Shaft by performing the following operations:

- Constructing a cylindrical feature
- Constructing a cut feature

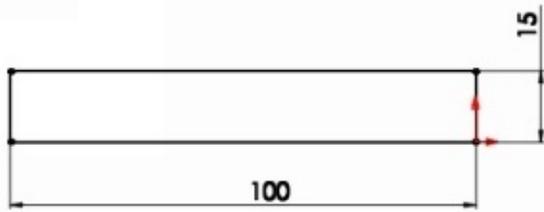
Opening a New Part File

1. To open a new part, click **File > New** on the menu bar.

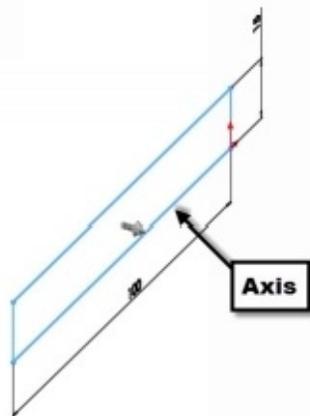
2. On the **New SOLIDWORKS Document** dialog, click the **Part** button.
3. Click **OK**.

Constructing the Cylindrical Feature

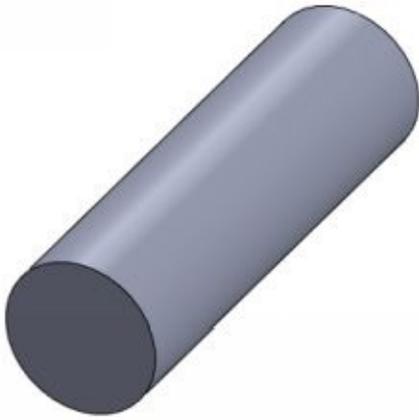
1. Activate the **Revolved Boss/Base**  tool and click the right plane.
2. Construct the sketch, as shown in figure.



3. Click **Exit Sketch** .
4. Click on the bottom horizontal line of the sketch to define the axis of rotation.

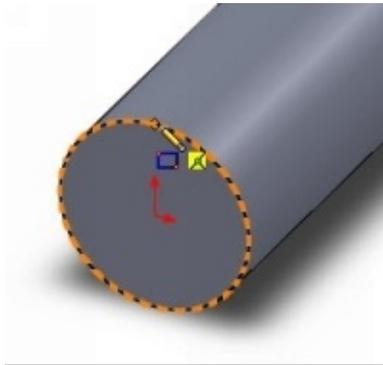


5. Click **OK** .

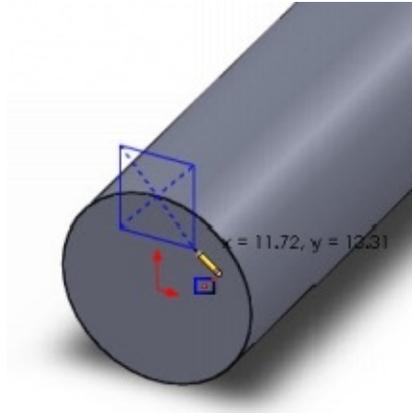


Constructing Cut feature

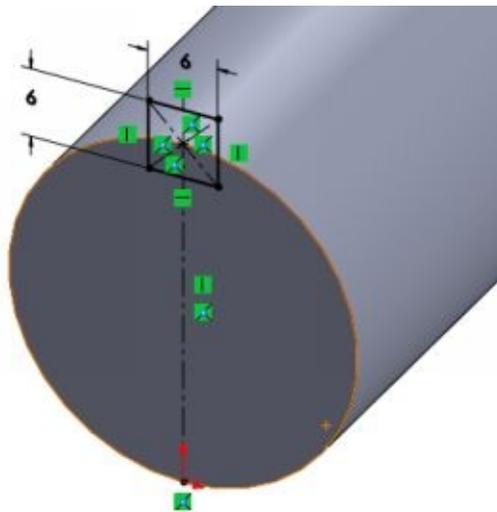
1. Activate the **Extruded Cut**  tool and click on the front face of the model geometry.
2. On the CommandManager, click **Sketch** > **Center Rectangle** .
3. Click on the top quadrant point of the circular edge.



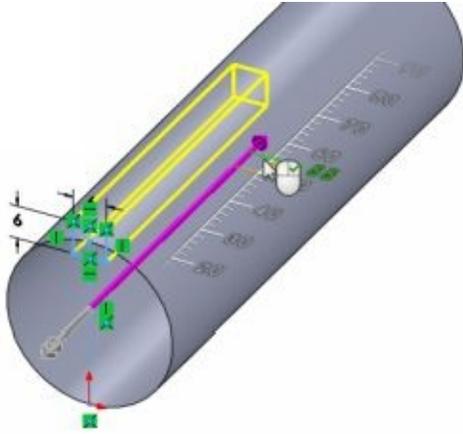
4. Move the pointer away and click to construct the rectangle.



5. Add dimensions and relations to fully constrain the sketch.



6. Click **Exit Sketch**.
7. Click on the extrude handle and drag. A scale appears on the model.
8. Release the pointer when the depth is set to 55.



9. Click **OK** ✓.



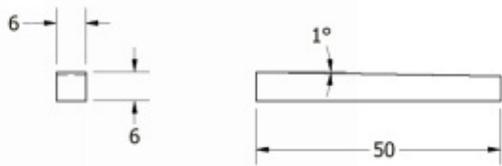
Saving the Part

1. Click the **Save**  button on the **Quick Access Toolbar**.
2. Enter **Shaft** in the **File name** field.
3. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
4. Click **Save** to save the file.
5. Click **File > Close**.

TUTORIAL 4

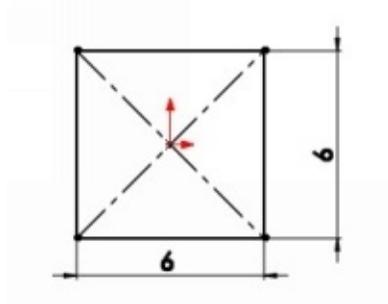
In this tutorial, you construct a Key by performing the following operations:

- Constructing an Extruded Boss
- Applying draft

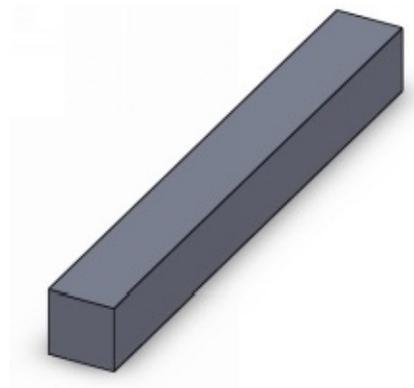


Constructing Extruded Boss

1. To open a new part, click **File** > **New** on the menu bar.
2. On the **New SOLIDWORKS Document** dialog, click the **Part** button.
3. Click **OK**.
4. On the CommandManager, click **Features** > **Extruded Boss/Base** .
5. Click on the front plane.
6. Construct the sketch, as shown in figure.

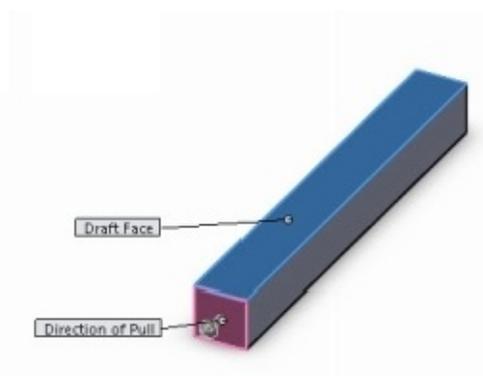


7. Click **Exit Sketch**.
8. On the **Boss-Extrude** PropertyManager, under the **Direction 1** section, set **End Condition** to **Mid Plane**.
9. Set the **Depth** value to 50.
10. Click **OK** to construct the extruded boss.



Applying Draft

1. On the CommandManager, click **Features** > **Draft** .
2. Click on the front face of the model geometry to define the neutral plane
3. Click on the top face to define the draft face.



4. Type-in 1 in the **Draft Angle** box and click the **Reverse Direction**  button.

5. Click **OK**.



Saving the Part

1. Click the **Save** button on the **Quick Access Toolbar**.
2. Enter **Key** in the **File name** field.
3. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
4. Click **Save** to save the file.
5. Click **File > Close**.

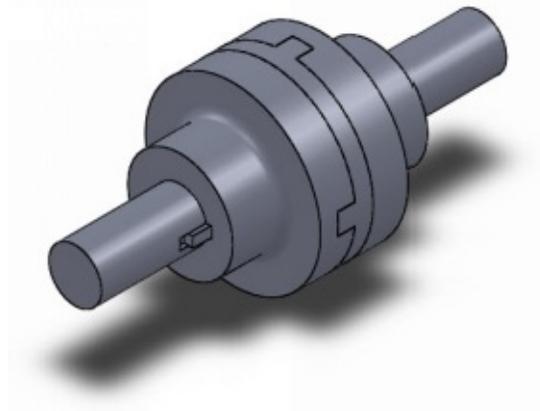
Chapter 3: Assembly Basics

In this chapter, you will:

- Add Components to the assembly
- Apply constraints between components
- Create the exploded view of the assembly

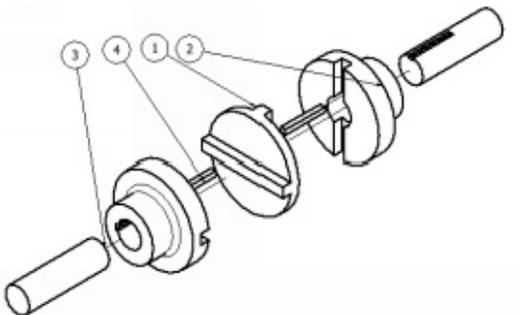
TUTORIAL 1

This tutorial takes you through the creation of your first assembly. You create



the Oldham coupling assembly:

PARTS LIST		
ITEM	PART NUMBER	QTY
1	Disc	1
2	Flange	2
3	Shaft	2
4	Key	2



There are two ways to create an assembly model.

- Top-Down Approach
- Bottom-Up Approach

Top-Down Approach

First, you will start the assembly document, and then construct components in it.

Bottom-Up Approach

First, you will create the components, and then add them to the assembly document. In this tutorial, you will create an assembly using this approach.

Opening a New Assembly File

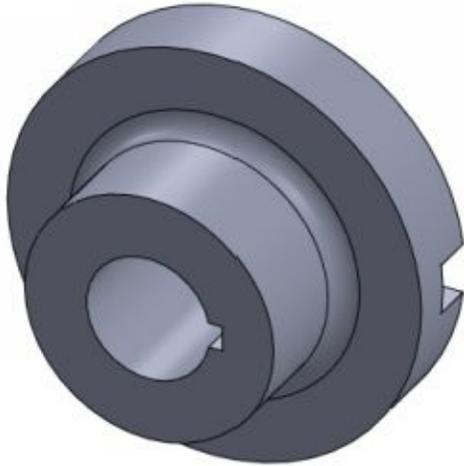
1. To open a new assembly, click **New**  on the **Quick Access Toolbar**.
2. On the **New SOLIDWORKS Document** dialog, click **Assembly**, and then click **OK**.



The Assembly window appears along with the **Begin Assembly** PropertyManager located on the left side.

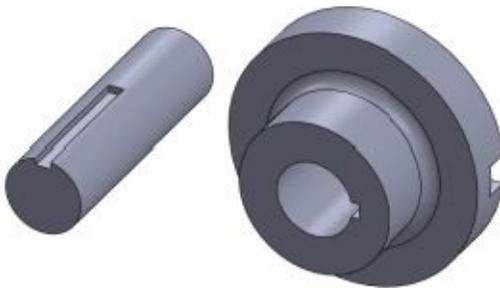
Inserting the Base Component

1. On the **Begin Assembly** PropertyManager, under the **Part/Assembly to Insert** section, click the **Browse** button.
2. Browse to the location: **C:\Users\Username\Documents\Oldham Coupling**.
3. Double-click on **Flange**. This attaches the component to the pointer.
4. Click inside the graphics window to position the component.



Adding the second component

1. To insert the second component, click **Assembly > Insert Component**  on the CommandManager.
2. On the **Insert Component** PropertyManager, click the **Browse** button.
3. Browse to the location **C:\Users\Username\Documents\Oldham Coupling**, and then double-click on **Shaft**.
4. Click inside the graphics window to position the shaft.



In SOLIDWORKS 2016, you can view a component in separate preview window. This helps you to the rotate or move component independently, and apply mates. To view component in the preview window, click on it and select **Component Preview Window** .

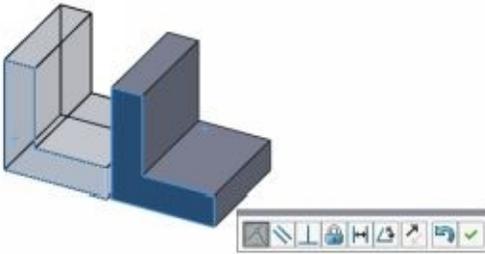
Applying Mates

After adding the components to the assembly window, you have to apply mates between them. By applying mates, you establish real contact and motion between components.

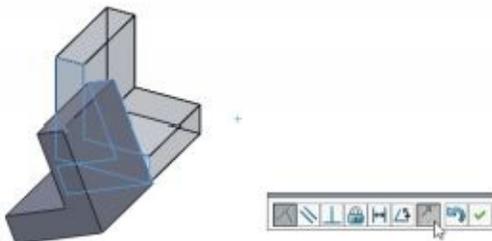
You can apply mates between parts using the **Mate**  tool.

The following sections explain some of the assembly mates that you can apply using the **Mate** tool.

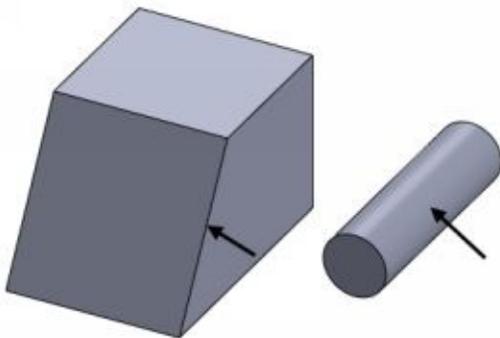
 **Coincident:** Using this mate, you can align two faces such that they will point in the same direction.

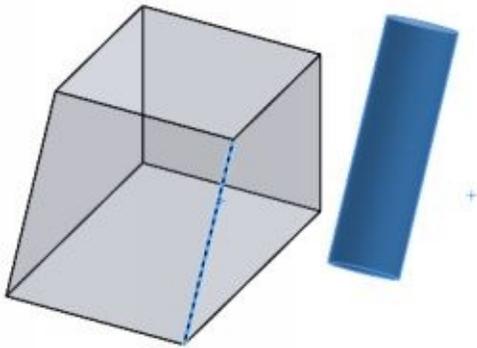


On the context toolbar, click **Flip Mate Alignment** to flip the alignment direction.

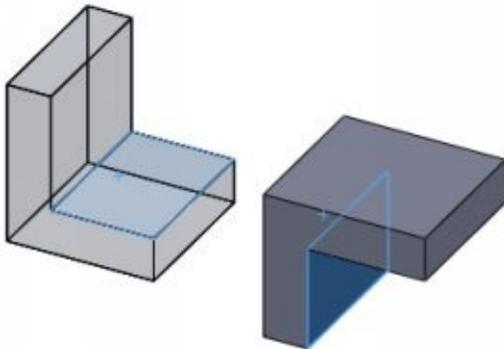
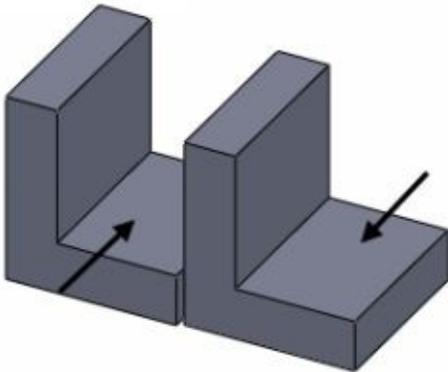


 **Parallel:** Using this mate, you can make an axis, face, or edge of one part parallel to that of another part.

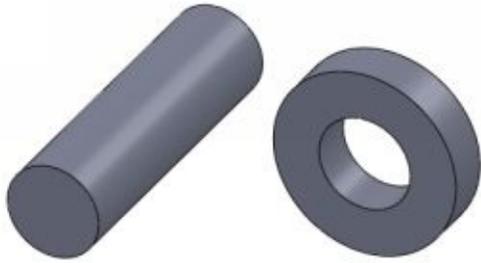




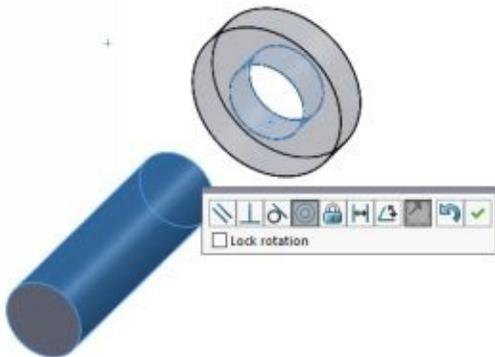
⊥ Perpendicular: Using this mate, you can make an axis, face or edge of one part perpendicular to that of another part.



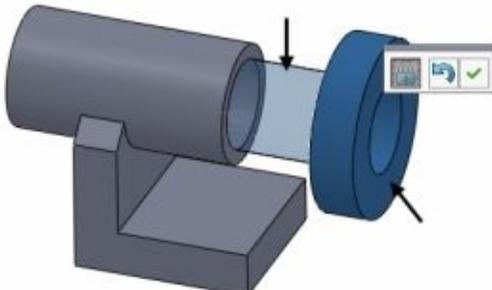
☯ Concentric: Using this mate, you can align the centerlines of the cylindrical faces.

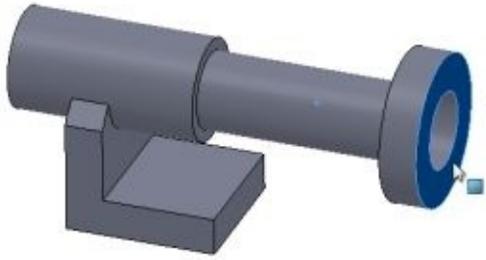


Click **Flip Mate Alignment** to flip the alignment direction. Check the **Lock Rotation** option to lock the rotation between components.

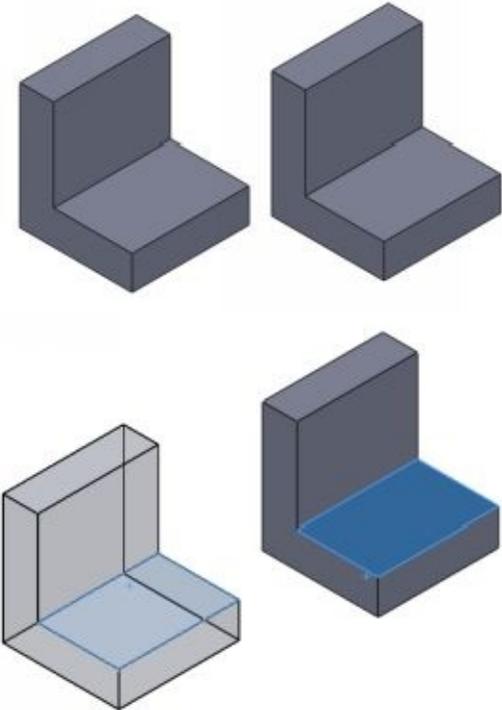


Lock: Makes parts to form a rigid set. As you move a single part in a rigid set, all the other parts will also be moved.

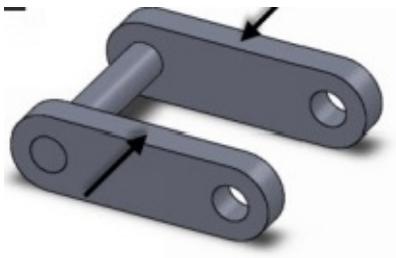


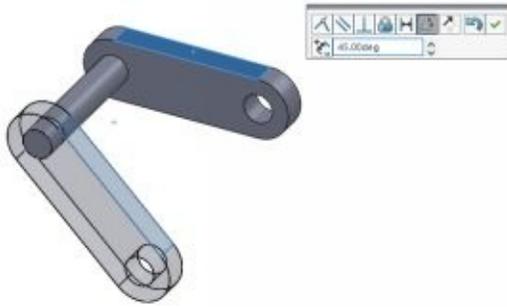


 **Distance:** Applies the distance mate between two components.



 **Angle:** Applies the angle mate between two components.

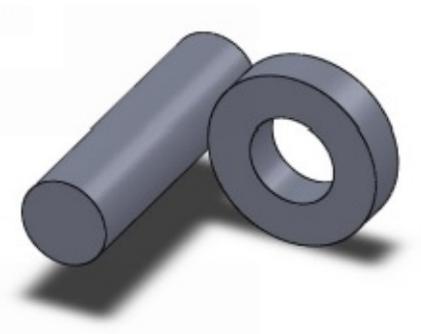
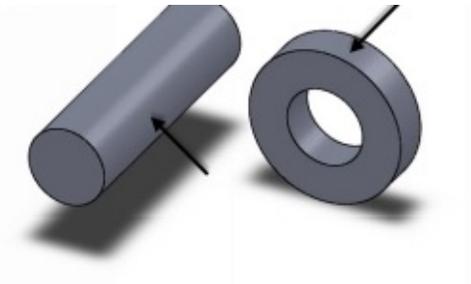




Click **Flip Dimension** to reverse the angle direction.



 **Tangent:** This mate makes two faces tangent to each other.



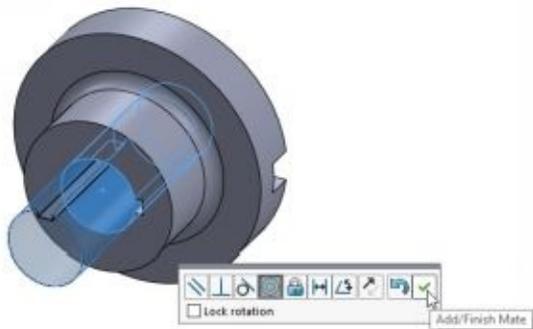
1. On the CommandManager, click **Assembly > Mate** .
2. On the **Mate PropertyManager**, click **Concentric** .

3. Click on the cylindrical face of the Shaft.
4. Click on the inner cylindrical face of the Flange



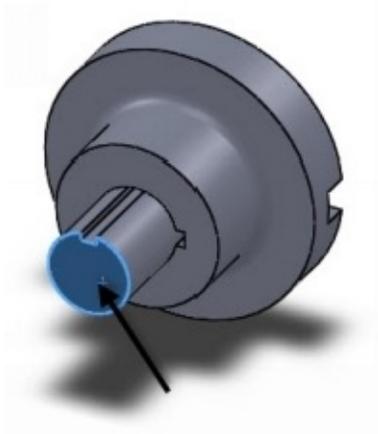
 In SOLIDWORKS 2016, the first selection is made transparent while applying mates. You can turn OFF the transparency by unchecking the **Make first selection transparent** option in the **Options** section of the **Mate** PropertyManager.

5. Click **Add/Finish Mate**.

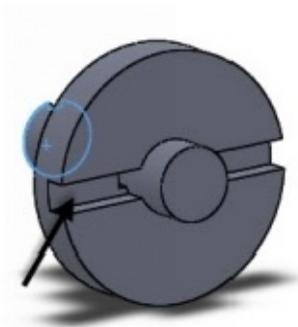


This aligns the Shaft and the Flange axially.

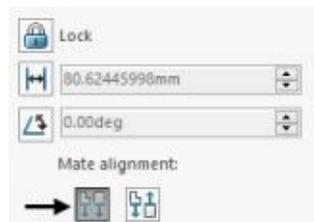
6. On the **Mate** PropertyManager, click **Coincident** .
7. Click on the front face of the shaft.



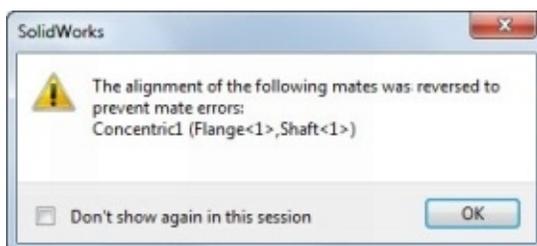
8. Rotate the model.
9. Click on the slot face of the flange, as shown in figure.



10. On the **Mate** PropertyManager, under the **Standard Mates** section, click the **Aligned** button.

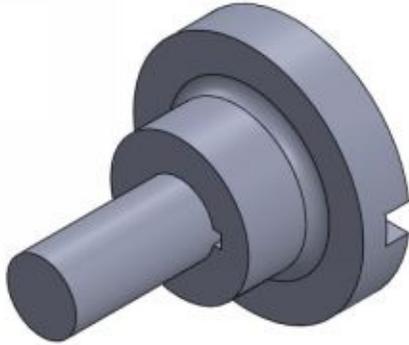


11. Click **OK** on the **SOLIDWORKS** message.



12. Click **Add/Finish Mate**.

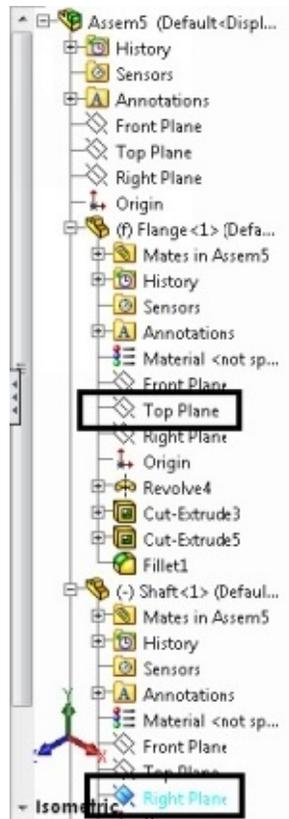
This aligns front face of the Shaft and the slot face of the Flange.



13. On the **Mate** PropertyManager, click **Coincident** .

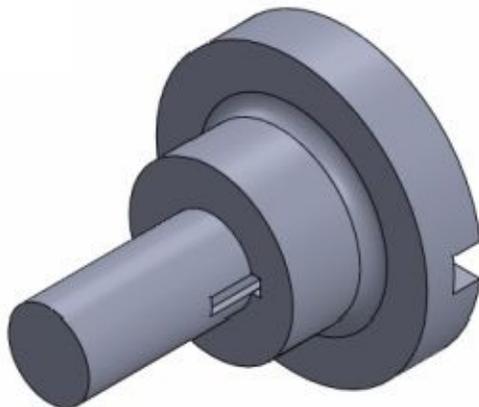
14. In the **FeatureManager tree**, expand **Flange** and select **Top Plane**.

15. Expand Shaft and select **Right Plane**.



- 16. Click **Add/Finish Mate**.
- 17. Click **OK** on the **PropertyManager**.

This assembles the two components and fully constrains the Shaft.



Adding the Third Component

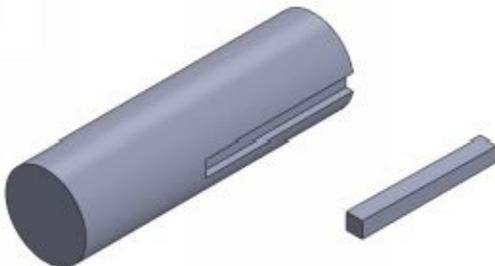
In order to assemble the third component, you need to hide the Flange.

1. In the **FeatureManager Design Tree**, click the right mouse button on the Flange and select **Hide Components** to hide the Flange. Now, it is easy to assemble the third component.



NEW SOLIDWORKS 2016 allows to hide a body placing the cursor on it and pressing the Tab key. You can unhide the body by placing the cursor at its position and pressing Shift+Tab.

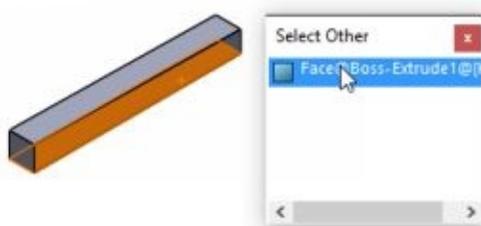
2. To insert the third component, click **Assembly > Insert Component** .
3. On the PropertyManager, click the **Browse** button.
4. Browse to the folder: **C:\Users\Username\Documents\Oldham Coupling**, and then double-click on the Key.
5. Click inside the graphics window to position the key.



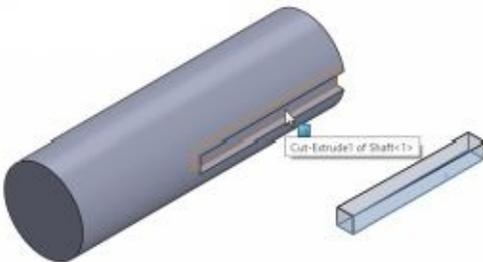
6. Activate the **Mate**  tool.
7. On the **Mate** PropertyManager, click **Coincident** .
8. Right-click on the side face of the key and select **Select Other**.



9. From the **Select Other** box, select the bottom face of the Key.

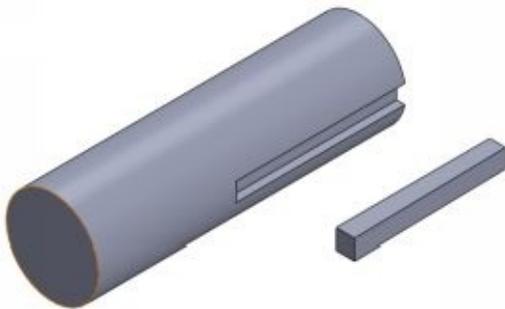


10. Select the bottom face of the slot.

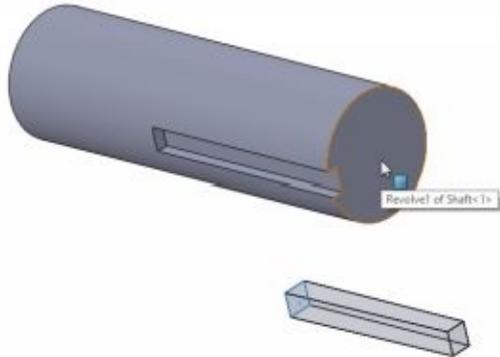


11. Click **Add/Finish Mate** .

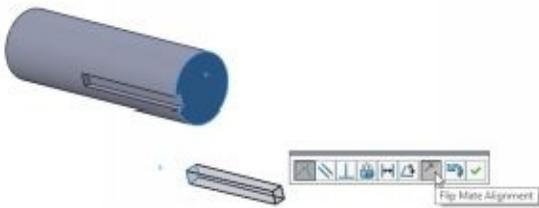
12. This aligns the bottom face of the key with the bottom face of the slot.



13. Select the front face of the Key and back face of the Shaft.

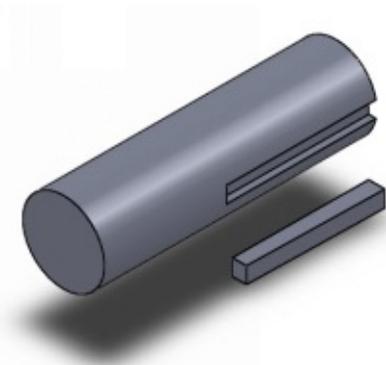


14. Click **Flip Alignment**.

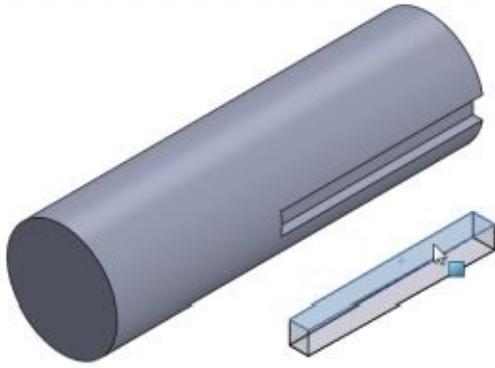


15. Click **Add/Finish Mate** .

16. This applies the mate between the selected faces.



17. Click on the top face of the key.



18. Rotate the shaft and select the side face of the slot.



19. Click **Add/Finish Mate**.

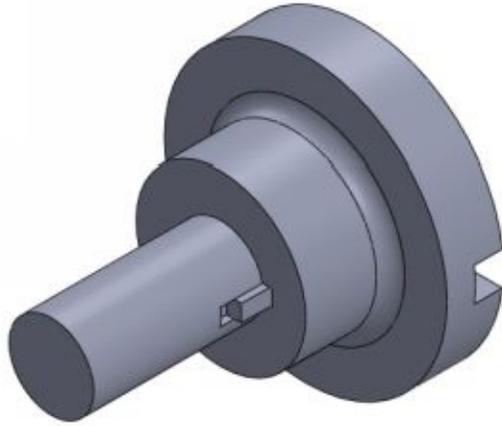


20. Click **OK**  on the PropertyManager.

21. Now, you have to turn-on the display of the Flange.

22. Click the right mouse button on the Flange in the **FeatureManager** tree, and then select **Show Components** to display the Flange.





Checking the Interference

1. On the CommandManager, click **Evaluate > Interference Detection** .
2. On the PropertyManager, click **Calculate**.

The **Results** section shows **No Interferences**.

3. Click **OK** .

Saving the Assembly

1. Click **Save**  on the **Quick Access Toolbar**.
2. Enter **Flange_subassembly** in the **File name** field.
3. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
4. Click **Save** to save the file.
5. Click **File > Close**.

Starting the Main assembly

1. Click the **New**  button on the **Quick Access Toolbar**.
2. On the **New SOLIDWORKS Document** dialog, click **Assembly**, and then click **OK**.

Adding Disc to the Assembly

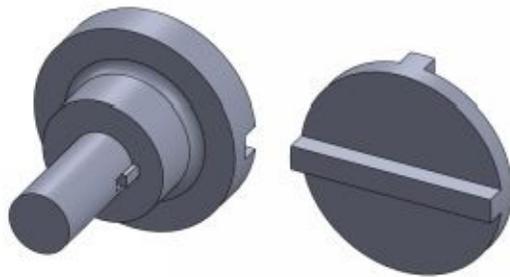
1. On the **Begin Assembly** PropertyManager, click **Browse**.
2. Browse to the location **C:\Users\Username\Documents\Oldham**

Coupling and double-click on **Disc.ipt**.

3. Click inside the graphics window to place the component at the origin.

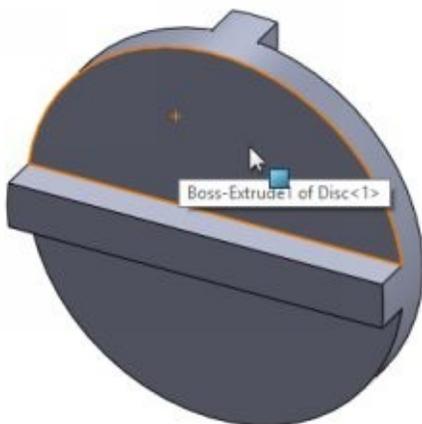
Placing the Subassembly

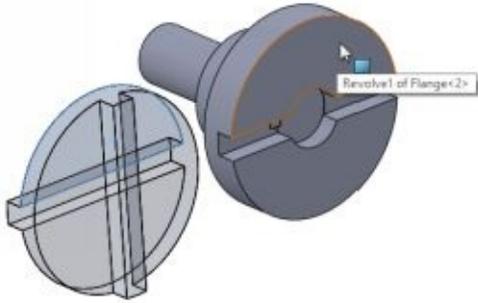
1. To insert the subassembly, click **Assembly > Insert Components**  on the CommandManager.
2. Click **Browse** on the PropertyManager.
3. Browse to the location **C:\Users\Username\Documents\Oldham Coupling** and double-click on **Flange_subassembly.asm**.
4. Click in the window to place the flange sub assembly.



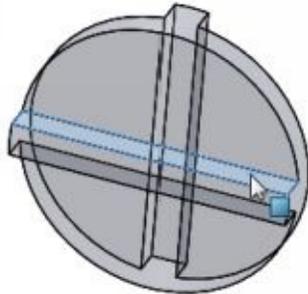
Adding Constraints

1. Activate the **Mate**  tool.
2. Click on the front face of the disc and back face of the flange.

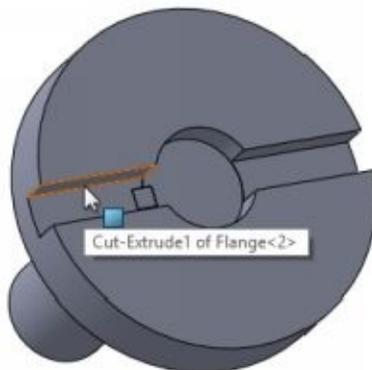




3. Click **Add/Finish Mate**.
4. Click on top flat face of the disc.

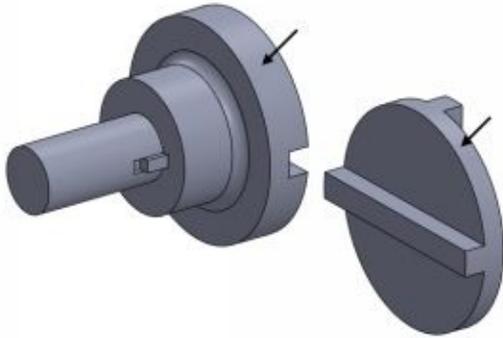


5. Rotate the model geometry and click on the slot face of the flange, as shown in figure.



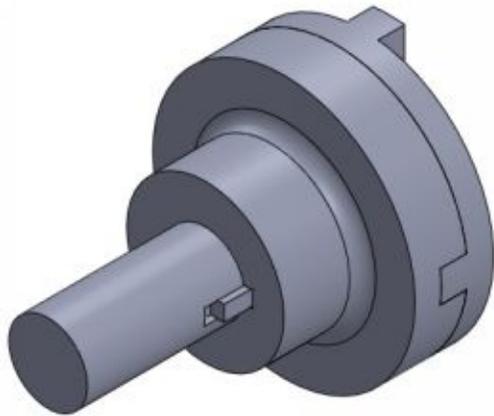
6. Click **Add/Finish Mate**.

7. Click on the cylindrical faces of the disc and flange.



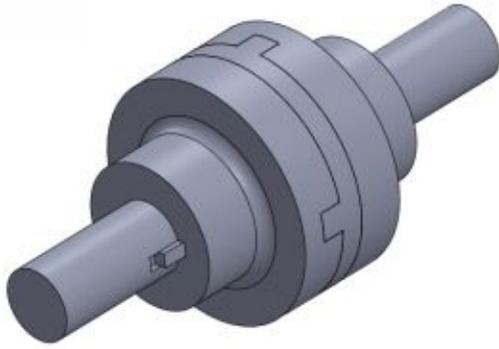
8. Click **Add/Finish Mate**.

9. Click **OK**.



Placing the second instance of the Subassembly

1. In the FeatureManager Design Tree, click on the Flange assembly.
2. Press Ctrl+C on your keyboard to copy the assembly.
3. Press Ctrl+V to paste the assembly. Expand the copied flange assembly in the FeatureManager Design Tree and notice that it is copied along with mates.
4. Mate the subassembly copy with the disc using the **Coincident**  and **Concentric**  relations.

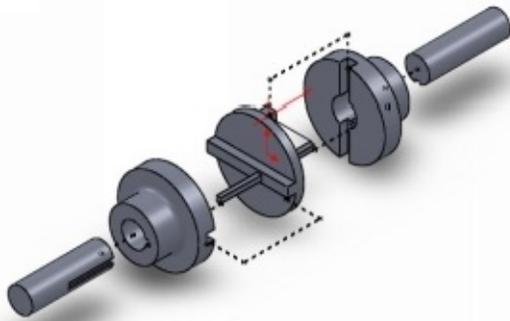


Saving the Assembly

1. Click **Save** on the **Quick Access Toolbar**.
2. Specify **Oldham_coupling** as **File name**.
3. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
4. Click **Save** to save the file.

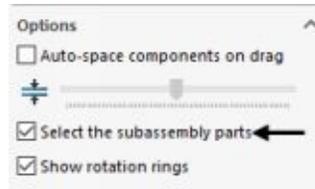
TUTORIAL 2

In this tutorial, you create the exploded view of the assembly:

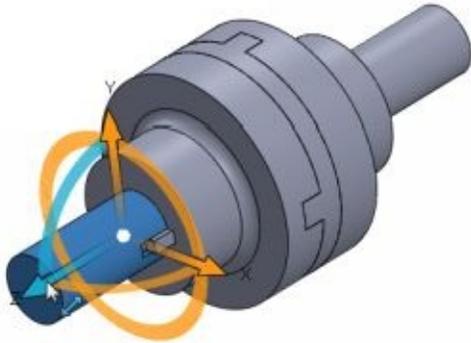


Creating the Exploded View

1. To create the exploded view, click **Assembly > Exploded View**  on the CommandManager.
2. On the **Explode** PropertyManager, under the **Explode Step type** section, click **Regular Step** .
3. On the **Explode** PropertyManager, under the **Options** section, check **Select the subassembly parts**.



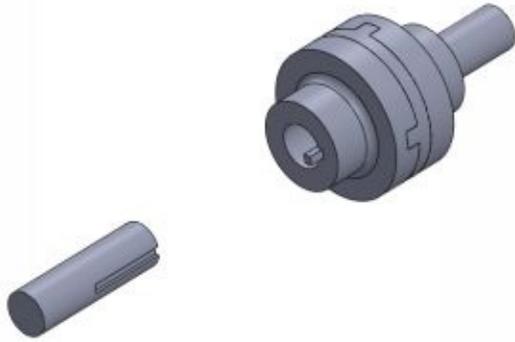
4. Click on the shaft, and then click on the Z-axis.



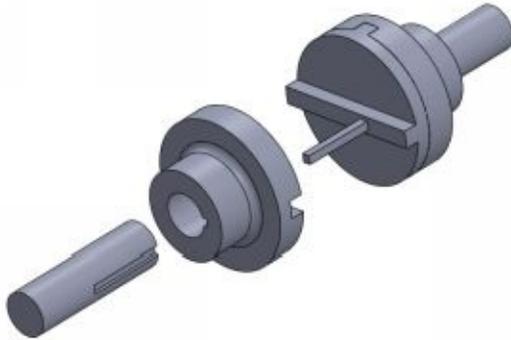
 SOLIDWORKS 2016 has an improved triad, which helps you to rotate or translate parts easily.

5. On the PropertyManager, under the **Settings** section, set the **Explode Distance** value to **180**.
6. Click **Apply**, and then click **Done**.





7. Click on the flange and set the **Explosion Distance** to 100.
8. Click **Apply**, and then click **Done**.



9. Click on the other shaft and set the **Explosion Distance** to -180.



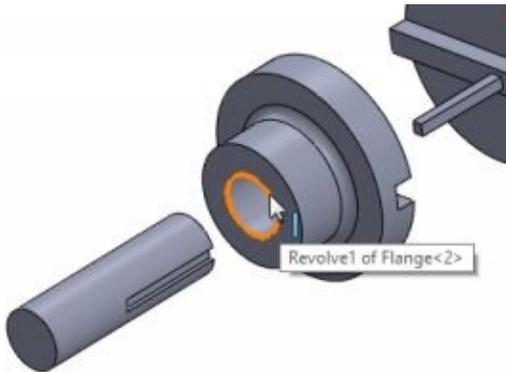
10. Likewise, explode the other flange to -100 mm distance.



11. Click **OK** on the PropertyManager.

Creating Explode lines

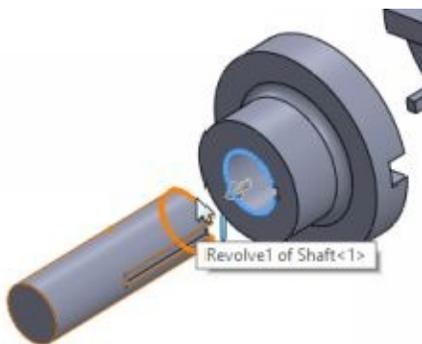
1. To create explode lines; click **Assembly > Explode Line Sketch** .
2. Click on the circular edge of the flange.



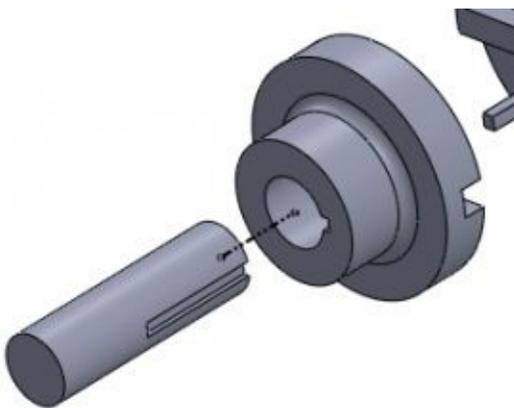
3. On the PropertyManager, under the **Options** section, check the **Reverse** option.



4. Click on the circular edge of the shaft.

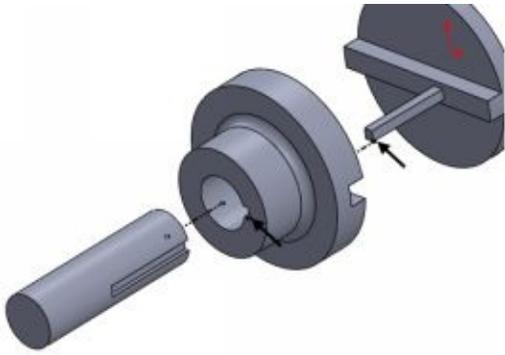


5. Click **OK**.

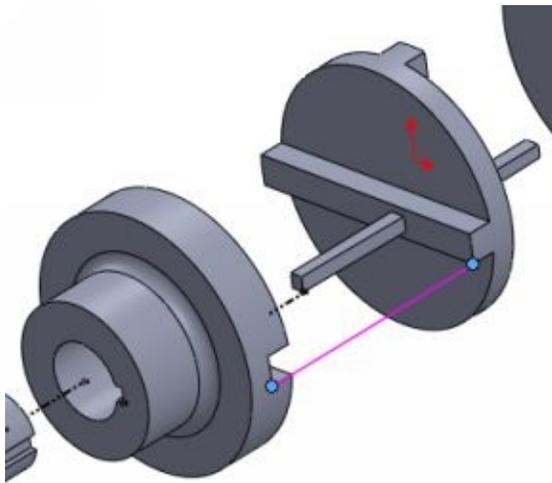


6. Click on the vertices of the flange and key, as shown in figure.

7. Click **OK**.



8. Click on the vertices of the flange and disc, as shown in figure.

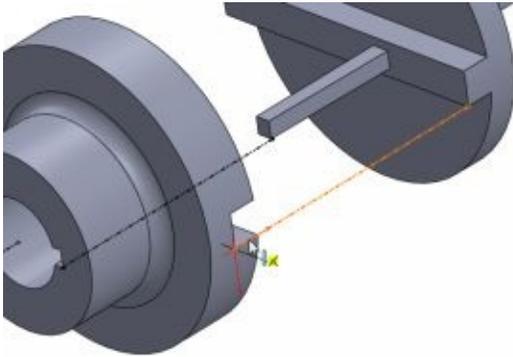


9. Click **OK**.

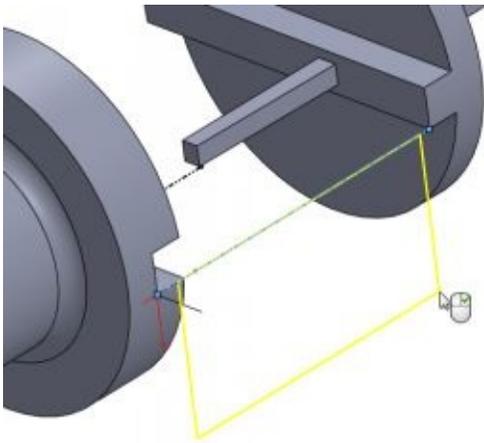
10. Click the **Jog Line** button located next to the PropertyManager.



11. Click on the explode line, as shown in figure.

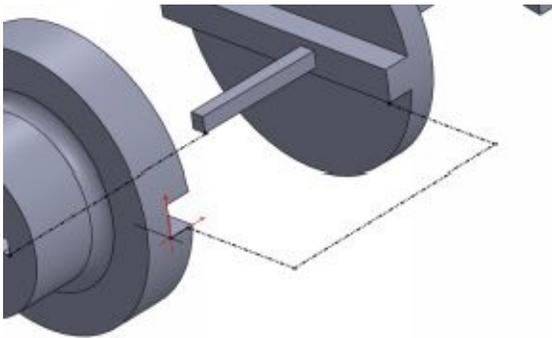
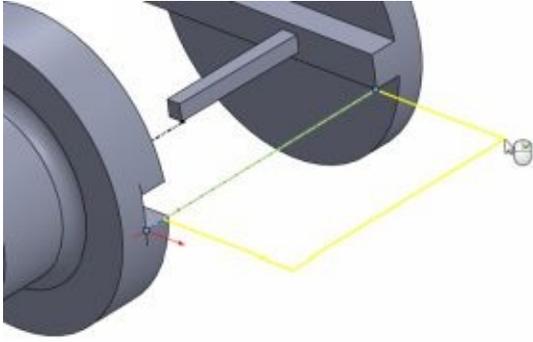


12. Drag the pointer to draw a rectangle.



13. Press the Tab key to change the orientation of the sketch line.

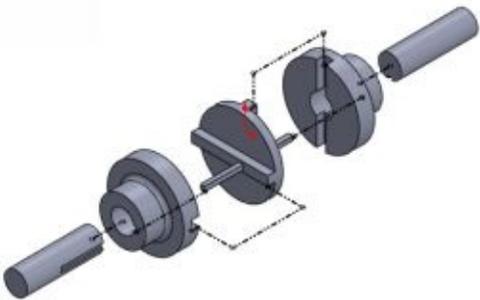
14. Click to draw the jogged line.



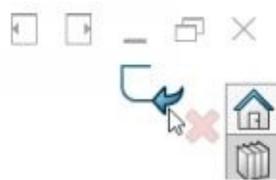
15. Click the **Route Line** icon located next to the PropertyManager.



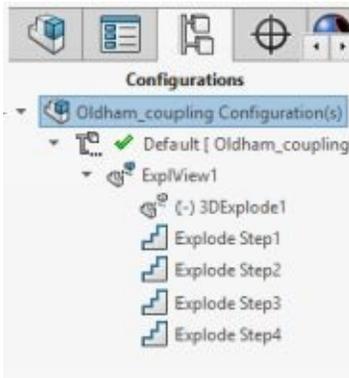
16. Draw the other explode lines. Click **OK** on the PropertyManager.



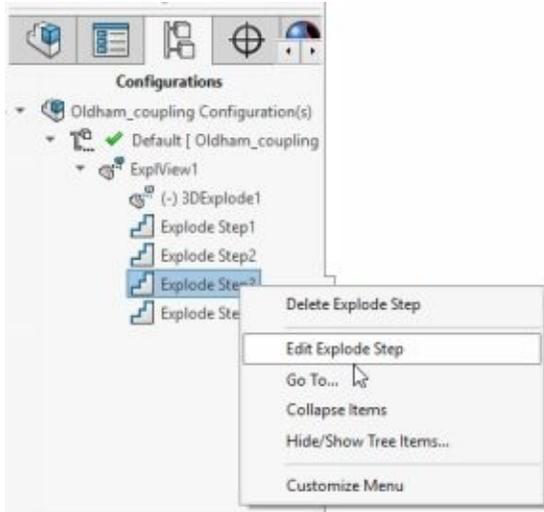
17. Click **Exit Sketch**.



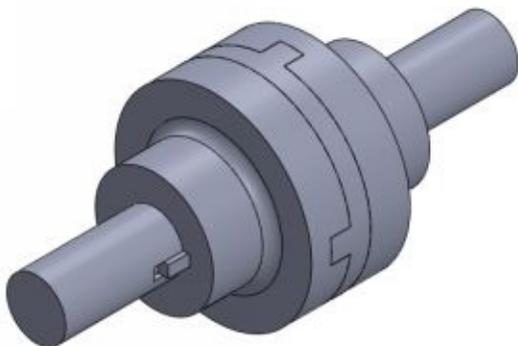
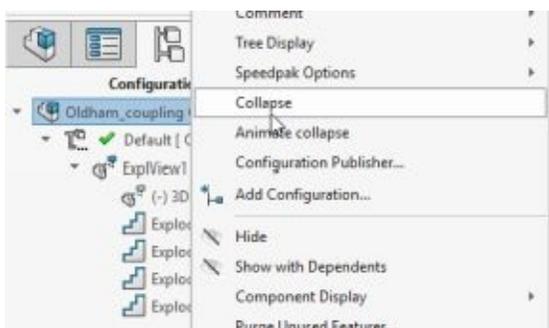
The **ConfigurationManager** lists all the explode steps and exploded lines.



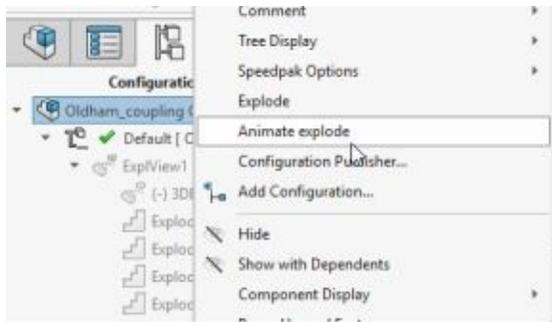
To edit an explode step, click the right mouse button on it and select **Edit Feature**.



To collapse the explosion, click the right mouse button on the exploded view and select **Collapse**.



To animate the explosion, click the right mouse button on the exploded view and select **Animate Explode**.



18. Save and close the assembly file.

Chapter 4: Creating Drawings

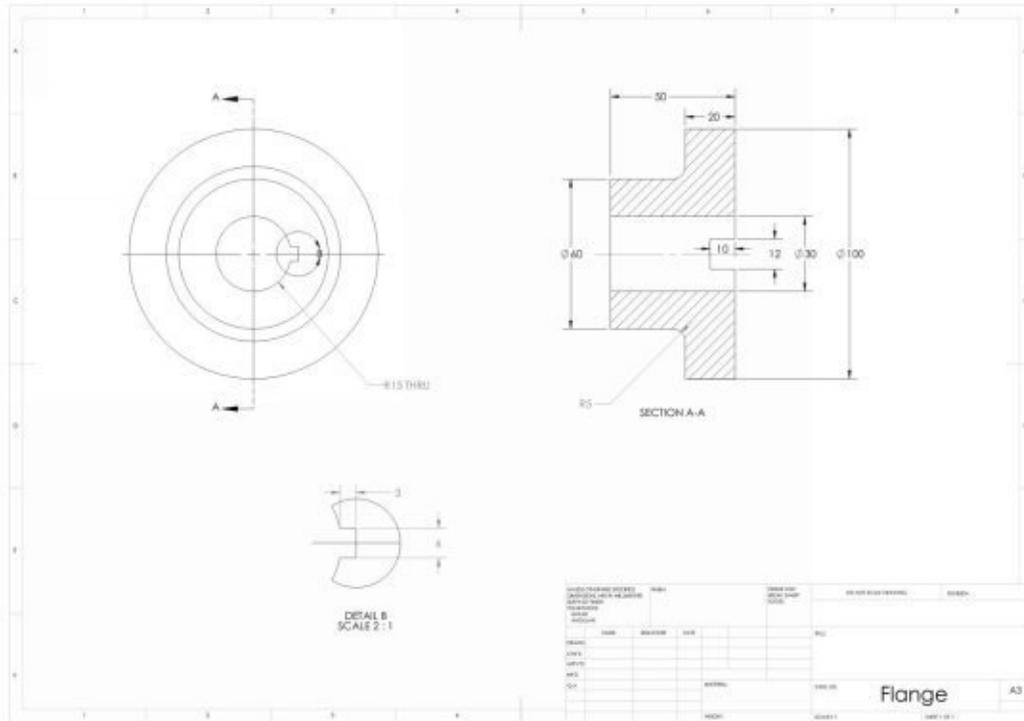
In this chapter, you create drawings of the parts and assembly from the previous chapters.

In this chapter, you will:

- Create model views
- Create section views
- Create detail views
- Add dimensions and annotations
- Create Custom templates
- Insert exploded view of the assembly
- Insert a bill of materials of the assembly
- Apply balloons to the assembly drawing

TUTORIAL 1

In this tutorial, you will create the drawing of the Flange.sldprt file created in the second chapter.

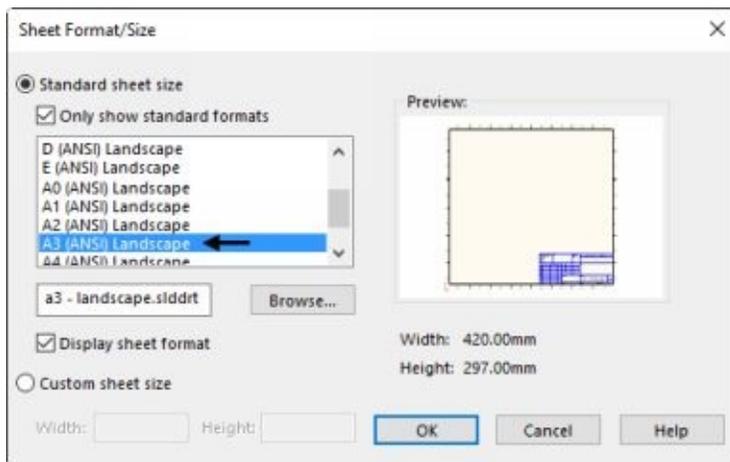


Opening a New Drawing File

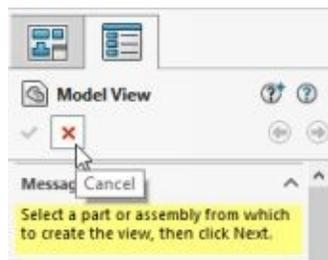
1. To open a new drawing, click the **New**  button on the **Quick Access Toolbar**.
2. On the **New SOLIDWORKS Document** dialog, select **Drawing** and click **OK**.



3. Select **A3 (ANSI) Landscape** from the list box in the **Sheet Format/Size** dialog.



4. Click **OK**.
5. Click **Cancel** on the **Model View** PropertyManager.



6. On the **FeatureManager Design Tree**, click the right mouse button on **Sheet1** and select **Properties**.

7. On the **Sheet Properties** dialog, set the **Type of projection** to **Third angle**.
8. Click **OK**.

Generating the Base View

1. On the CommandManager, click **View Layout > Model View** .
2. To generate the base view, click the **Browse** button on the **Model View** PropertyManager.



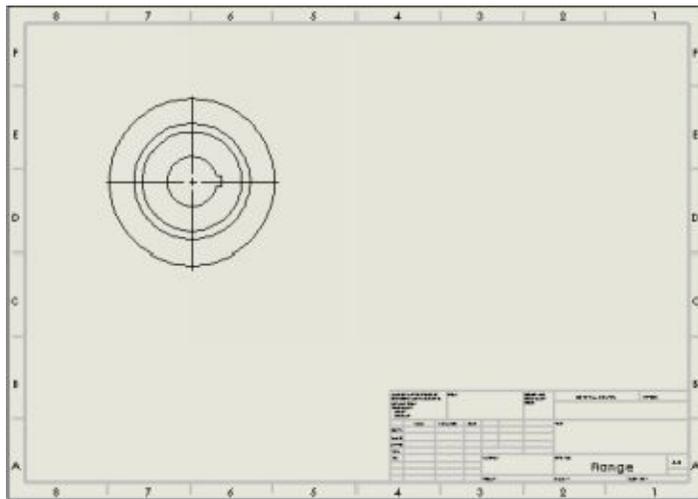
3. Browse to the location

C: UserDocument/Oldham_Coupling and double-click on **Flange.sldprt**.

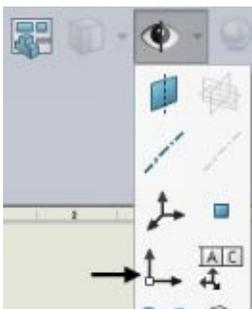
4. On the **Model View** PropertyManager, under the **Orientation** section, click **Front**.



5. Click on the sheet to position the view. The **Projected View** tool gets active.

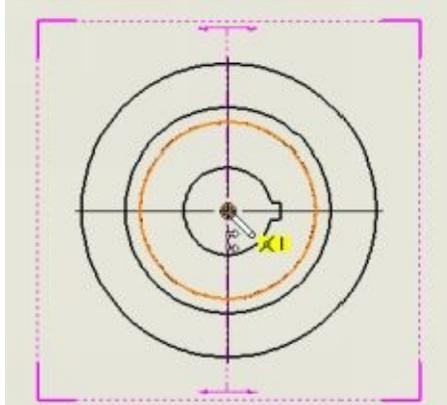


6. Click **OK** ✓ on the **PropertyManager**.
7. On the **View Heads-up** toolbar, click the **Hide** flyout and make sure that the **View Origin** icon is deactivated.

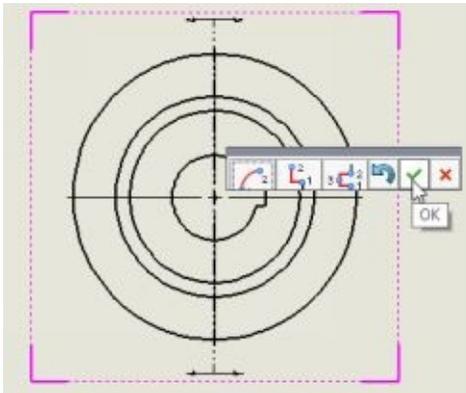


Creating the Section View

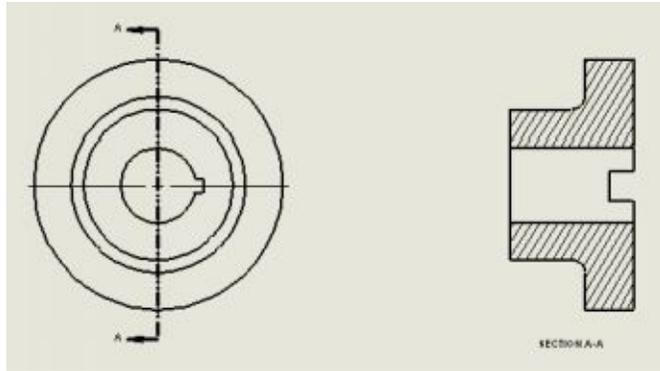
1. To create the section view, click **View Layout > Section View**  on the CommandManager.
2. Snap to the center point of the base view and click to position the section line.



3. Click **OK**.



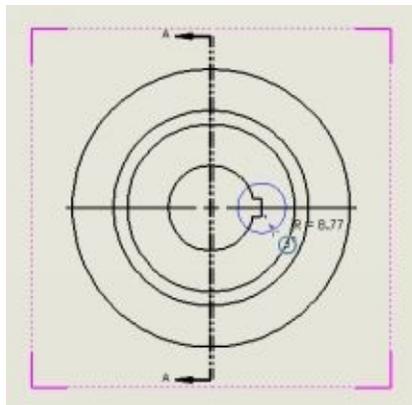
4. Move the pointer toward right and click to place the section view.
5. Click the green check on the PropertyManager.



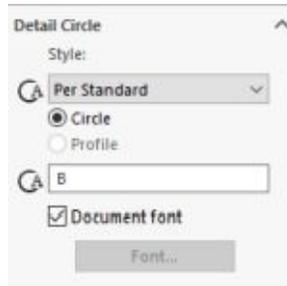
Creating the Detailed View

Now, you need to create the detailed view of the keyway that appears on the front view.

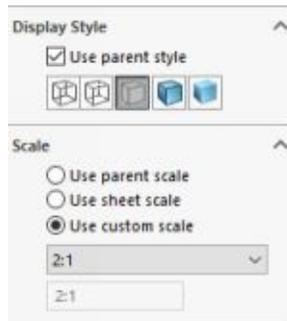
1. To create the detailed view, click **View Layout > Detail View**  on the CommandManager.
2. Specify the center point and boundary point of the detail view, as shown in figure.



3. On the PropertyManager, under the **Detail Circle** section, set the options shown below.



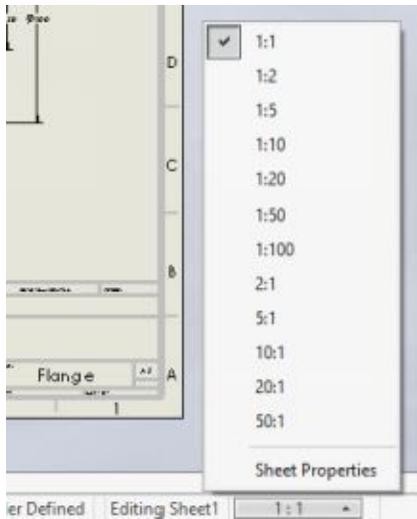
4. Set the **Display Style** and **Scale** options, as shown below.



5. Place the detail view below the base view. Click **OK** on the PropertyManager.

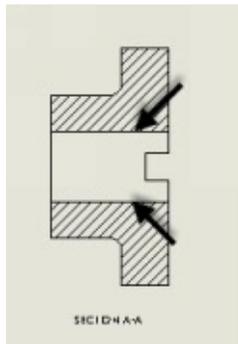


 In SOLIDWORKS 2016, you can change the scale of the drawing views by using the drop-down available at the bottom right. Note that the scale of the detail view remains unchanged.

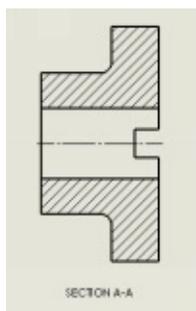


Creating the Centerline

1. On the CommandManager, click **Annotation > Centerline** .
2. Click on the inner horizontal edges of the section view, as shown in figure.



3. Click **OK**  on the PropertyManager.

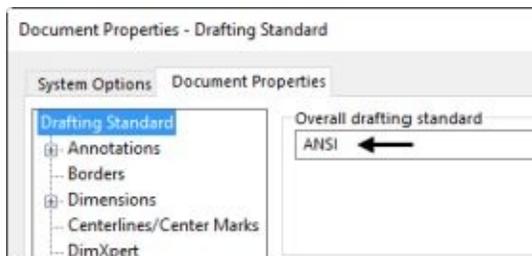


Specifying Dimension Settings

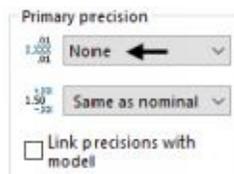
1. To specify the dimension settings, click **Options** on the **Quick Access Toolbar**.



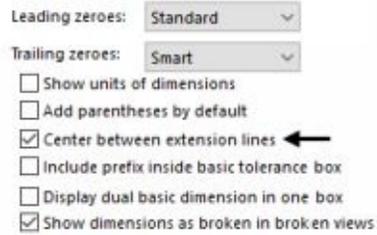
2. On the dialog, under the **Document Properties** tab, click **Drafting Standard**.
3. Select **Overall drafting standard > ANSI**.



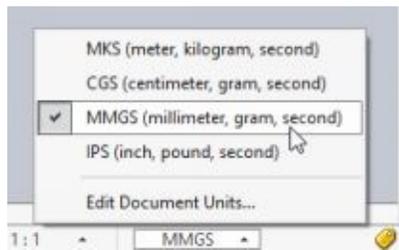
4. Under **Drafting Standard**, click **Dimensions**.
5. Under **Primary Precision**, select **Unit Precision > None**.



6. At the bottom of the dialog, check the **Center between extension lines**.



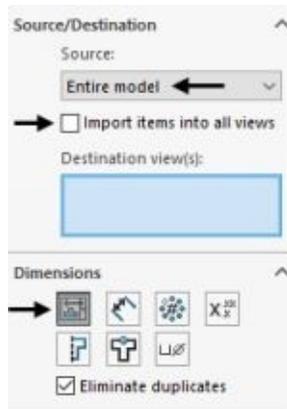
7. Click **OK**.
8. At the bottom of the window, set the **Unit System** to **MMGS**.



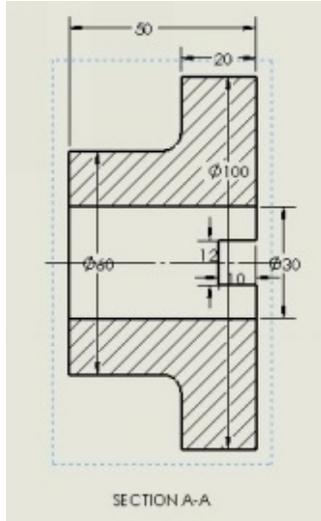
Retrieving Dimensions

Now, you will retrieve the dimensions that were applied to the model while creating it.

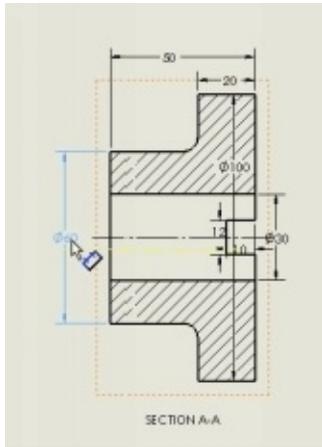
1. To retrieve dimensions, click **Annotation > Model Items**  on the CommandManager.
2. On the PropertyManager, select **Source > Entire model**.
3. Uncheck the **Import Items into all views** option.
4. Under the **Dimensions** section, select **Marked for drawing**.

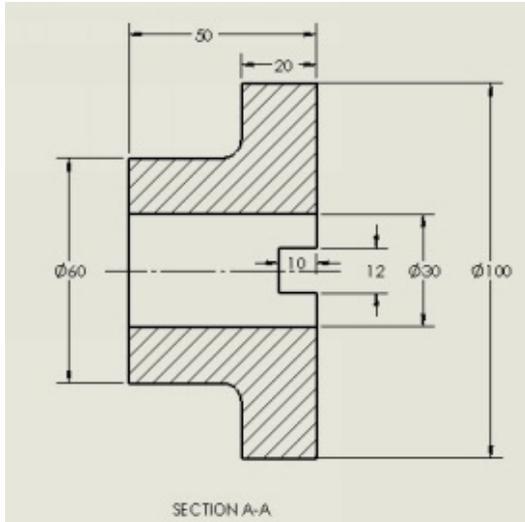


5. Click on the view boundary of the section view and click **OK**  on the PropertyManager.



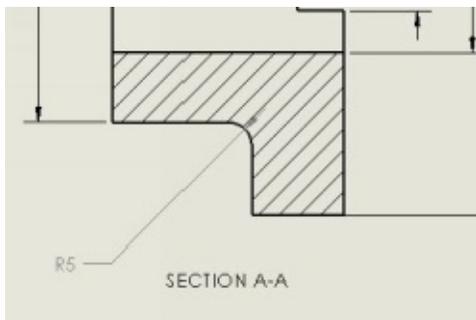
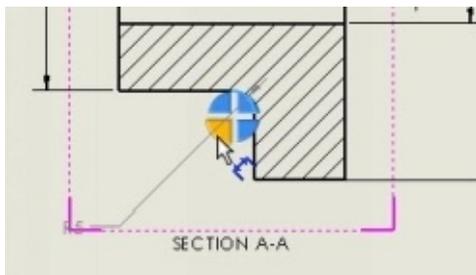
6. Drag and arrange the dimensions.



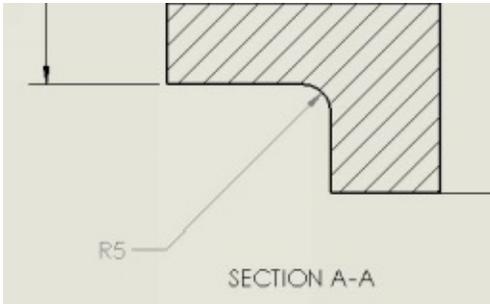
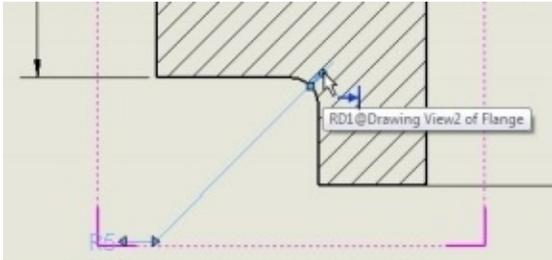


Adding additional dimensions

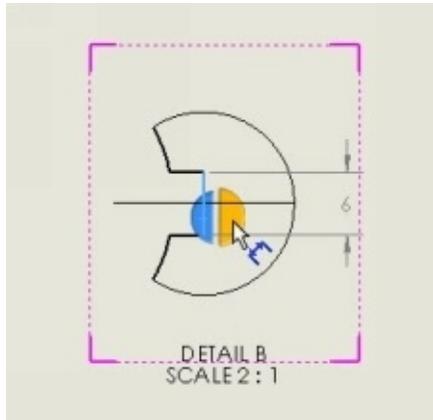
1. To add dimensions, click **Annotation > Smart Dimension**  on the CommandManager.
2. On the section view, click on the fillet edge and position the radial dimension.



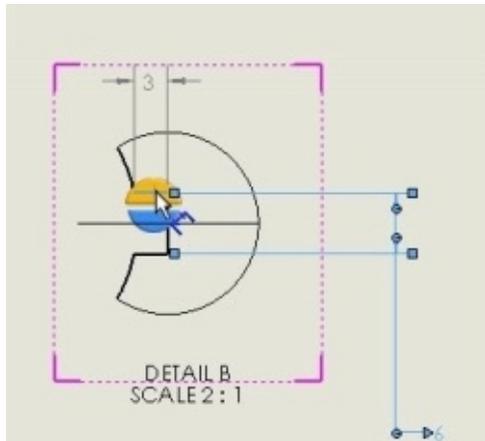
3. Select the radial dimension and click on the arrow to change its direction.



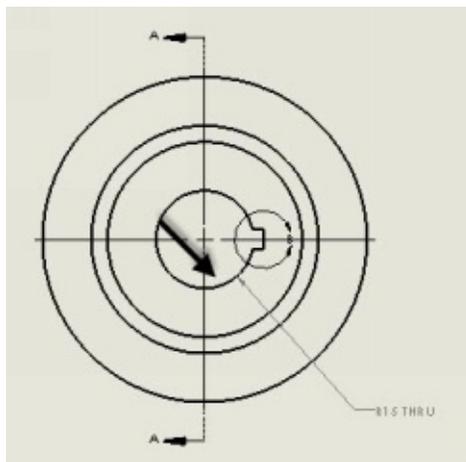
4. On the detail view, click on the vertical edge of the slot and position the dimension.



5. Likewise, add dimension to the horizontal edge of the slot.



6. On the CommandManager, click **Annotation > Hole Callout**.
7. Click on the innermost circular edge of the front view and position the hole callout.
8. Click **Close Dialog** on the PropertyManager



Saving the Drawing

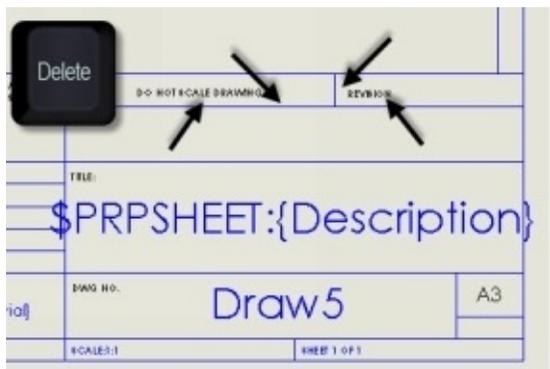
1. Click **Save** on the **Quick Access Toolbar**.
2. Specify **Flange** as **Name**.
3. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
4. Click **Save** to save the file.
5. Click **File > Close**.

TUTORIAL 2

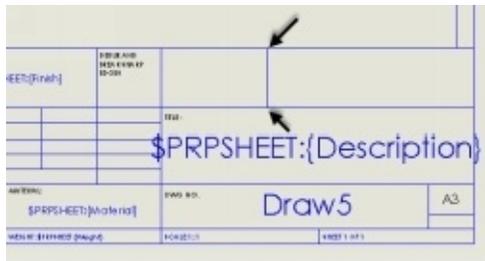
In this tutorial, you will create the drawing of Disc.sldprt file created in the second chapter.

Creating a Custom Sheet Format

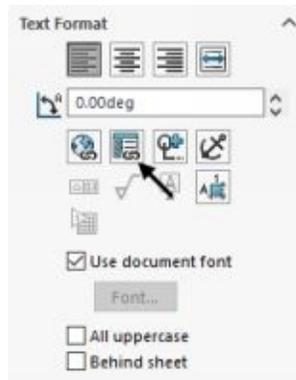
1. On Quick Access Toolbar, click the **New** button.
2. On the **New SOLIDWORKS Document** dialog, select **Drawing** and click **OK**.
6. Select **A3 (ANSI) Landscape** from the list box in the **Sheet Format/Size** dialog.
7. Click **OK**.
8. Close the **Model View** PropertyManager.
9. On the **FeatureManager Design Tree**, click the right mouse button on **Sheet1** and select **Properties**.
10. On the **Sheet Properties** dialog, set the **Type of projection** to **Third angle**. Click **OK**.
11. On the CommandManager, click **Sheet Format** > **Edit Sheet Format** .
12. Zoom to the title block, and then select the lines and notes shown in figure. Press **Delete** on your keyboard.



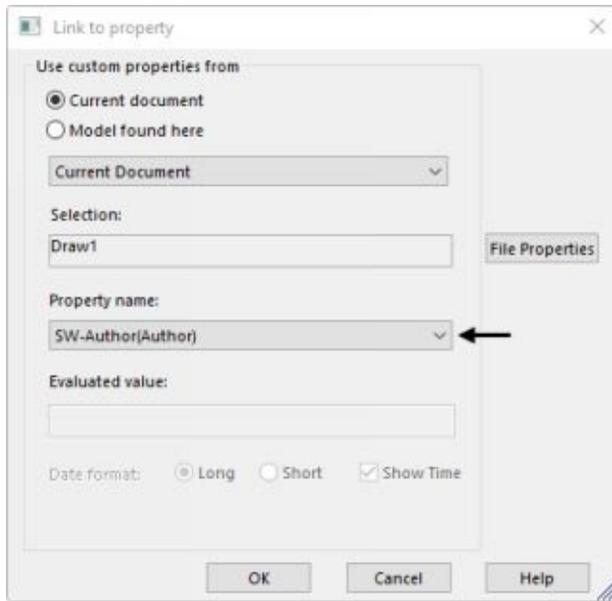
3. On the CommandManager, click **Sketch** > **Line**  and draw a line on the title block.



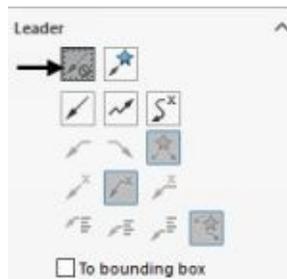
4. On the CommandManager, click **Annotation > Note**.
5. On the PropertyManager, under the **Text Format** section, select **Link to Property**.



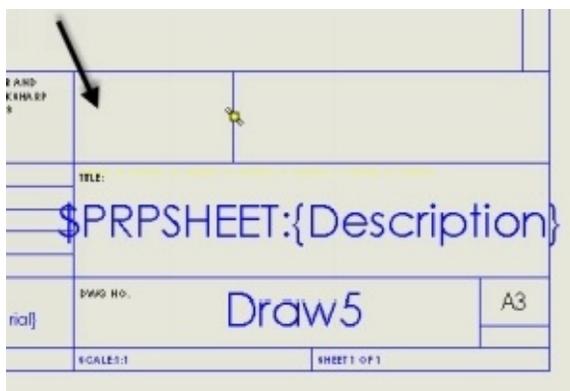
6. On the **Link to Property** dialog, select **Current document**.
7. Select **SW-Author (Author)** from the **Property name** drop-down, and then click **OK**.



8. On the PropertyManager, under the **Leader** section, select **No Leader**.



9. Click inside the box, as shown in figure.



10. On the **Formatting** toolbar, set the font size to 8 and type **Author**.

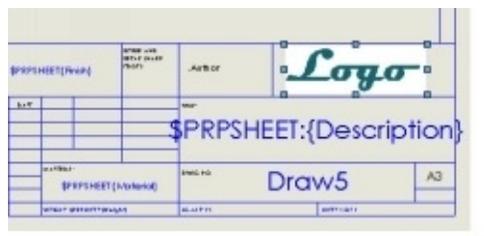
11. Click **OK**  on the PropertyManager.



12. On the Menu bar, click **Insert > Picture**.

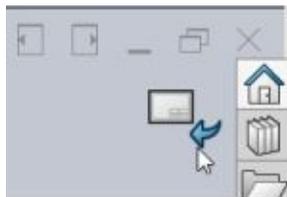
13. Select your company logo and click **Open**.

14. Resize and position the image on the title block.



15. Click **OK**  on the PropertyManager.

16. Click **Exit Sheet Format**.



17. On the **Quick Access Toolbar**, click **Options** .

18. On the dialog, under the **Document Properties** tab, click **Drafting Standard**.

19. Select **Overall drafting standard > ANSI**.

20. Click **Units** and set the **Unit System** to **MMGS**.

21. Click **OK**.

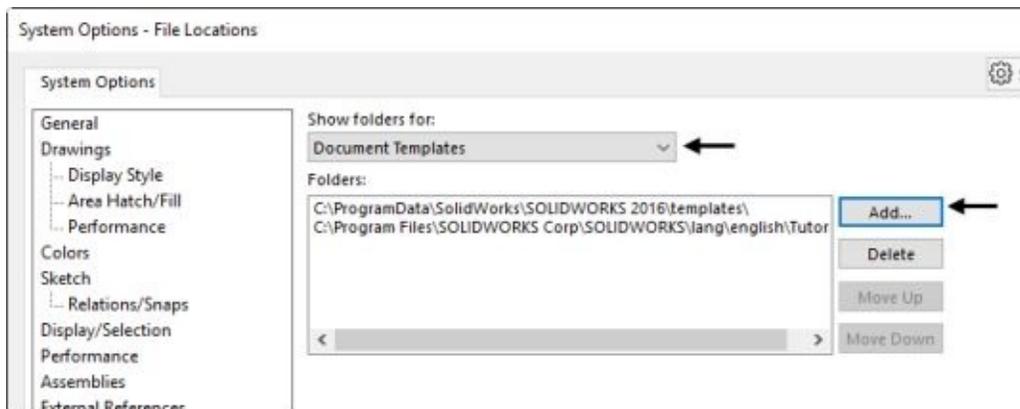
22. Click **File > Save**.

23. On the **Save As** dialog, select **Save as type > Drawing Templates**.

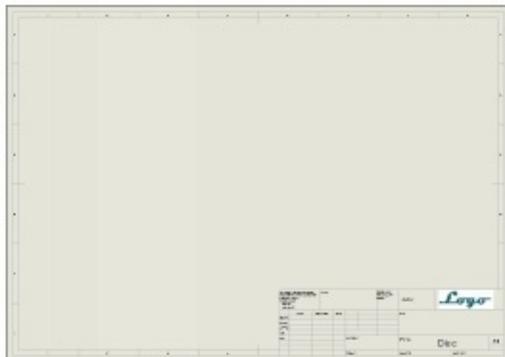
24. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder.
25. Type-in **Sample** in the **File name** box and click **Save**.
26. On the Menu bar, click **File > Close**.

Starting a New drawing

1. On the Quick Access Toolbar, click the **Options**  button.
2. On the **System Options** dialog, click **File Locations**.
3. Select **Show folders for > Document Templates** and click the **Add** button.



4. Browse to **C:\Users\Username\Documents\Oldham Coupling** folder. Click **OK**.
5. Click **OK** on the **System Options** dialog, and then click **Yes**.
6. On the Quick Access Toolbar, click the **New** button.
7. On the **New SOLIDWORKS Document** dialog, click the **Advanced** button.
8. Under the **Oldham Coupling** tab, select **Sample** and click **OK**.

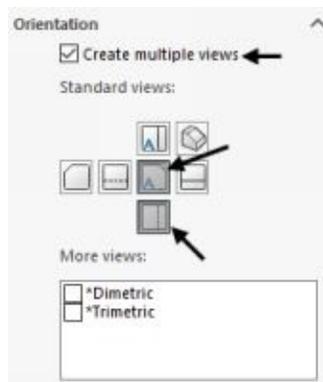


Generating the Drawing Views

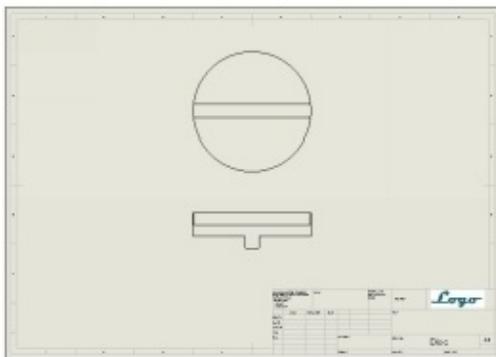
1. On the **Model View** PropertyManager, click the **Browse** button and browse to the location:

C: UserDocument/Oldham_Coupling

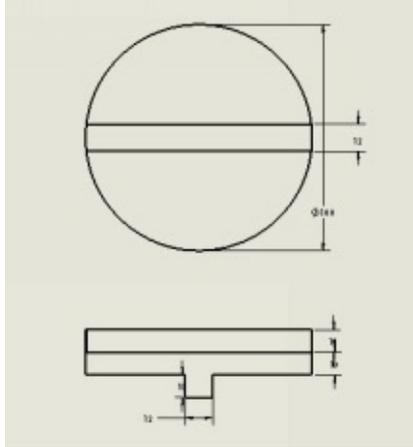
2. Double-click on **Disc.sldprt**.
3. On the PropertyManager, under the **Orientation** section, check the **Create multiple views** option.
4. Select the **Front** and **Bottom** views.



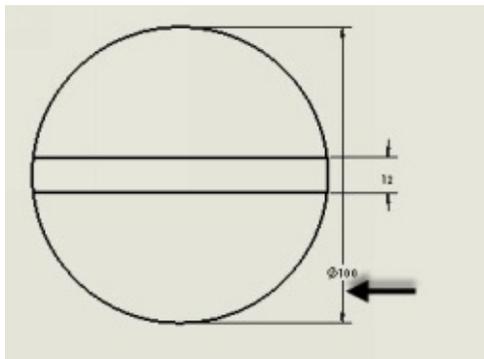
5. Click **OK** on the PropertyManager.



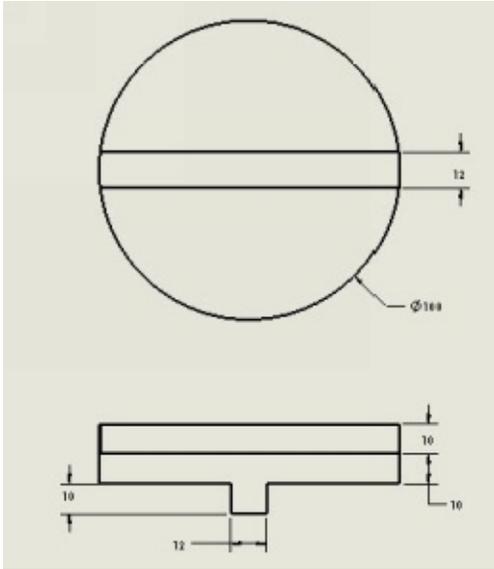
6. On the CommandManager, click **Annotation > Model Items** .
7. On the PropertyManager, select **Source > Entire model**.
8. Click on the two views and click **OK** .



9. Click the right mouse button on the 100-diameter dimension on the front view and select **Display Options > Display as Diameter**.



10. Arrange the dimensions by dragging them.



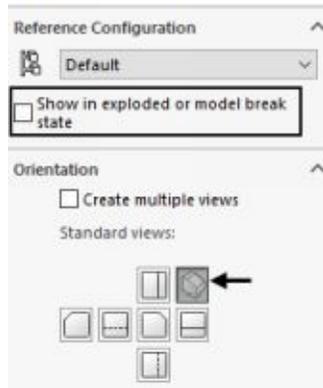
11. Save and close the drawing file.

TUTORIAL 3

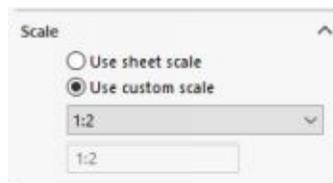
In this tutorial, you will create the drawing of Oldham coupling assembly created in the previous chapter.

Creating the Isometric View

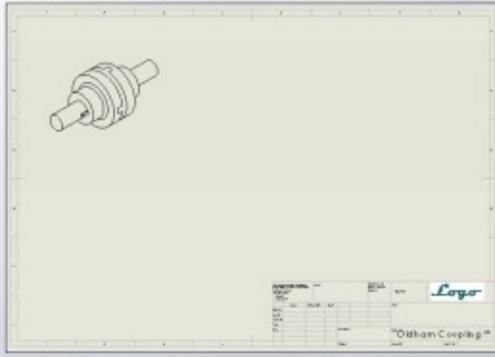
1. Start a new drawing file using the **Sample** template.
2. On the **Model View** PropertyManager, click the **Browse** button.
3. Browse to the location *C: UserDocument/Oldham_Coupling* folder and double-click on **Oldham_Coupling.sldasm**.
4. On the **Model View** PropertyManager, under the **Reference Configuration** section, uncheck the **Show in exploded or model break state** option.
5. Under the **Orientation** section, select **Isometric**.



6. Under the **Scale** section, select **Use custom scale** and set the scale factor to **1:2**.



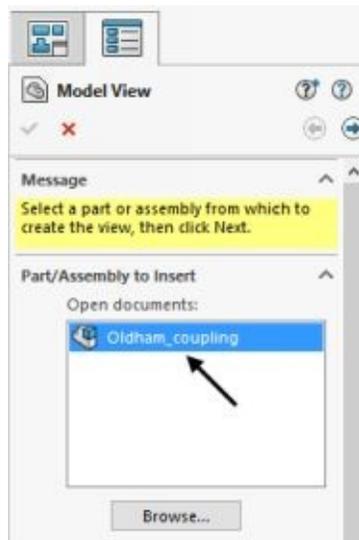
7. Click on the sheet to position the view.



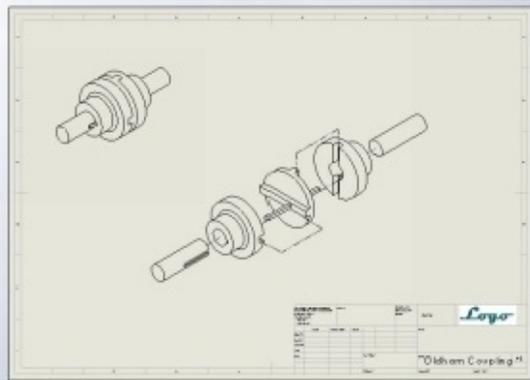
8. Click the green check on the PropertyManager.

Generating the Exploded View

1. On the CommandManager, click **View Layout > Model View** .
2. On the PropertyManager, under the **Part/Assembly to Insert** section, double-click on **Oldham Coupling**.

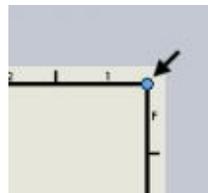


3. Under the **Reference Configuration** section, enable **Shown in exploded or model break state**.
4. Under the **Orientation** section, select **Isometric** .
5. Under the **Scale** section, select **Use custom scale** and set the scale factor to **1:2**.
6. Click on the sheet to position the exploded view.

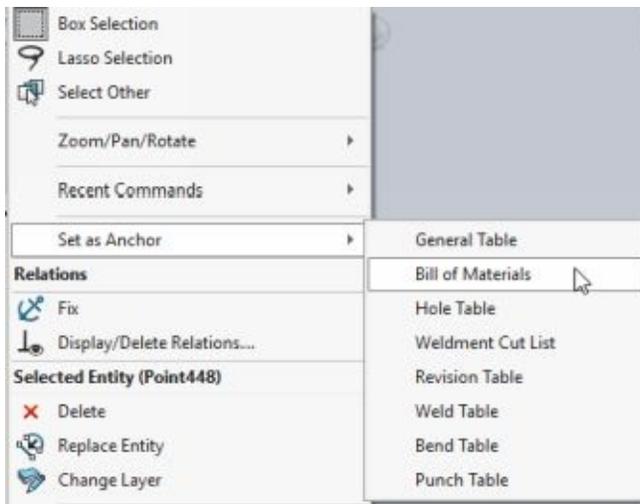


Creating the Bill of Materials

1. Click the right mouse button on the sheet and select **Edit Sheet Format**.
2. Click on the top right corner of the sheet border.

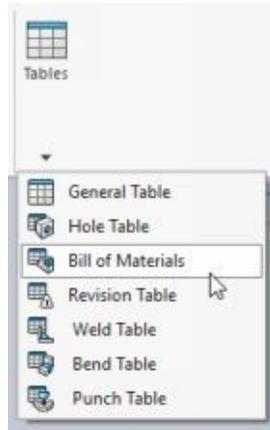


3. Click the right mouse button and select **Set Anchor as > Bill of Materials**.

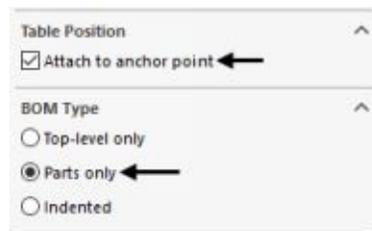


4. Click **Close Dialog** ✓ on the **Point** PropertyManager.
5. Click the right mouse button on the sheet and select **Edit Sheet**.

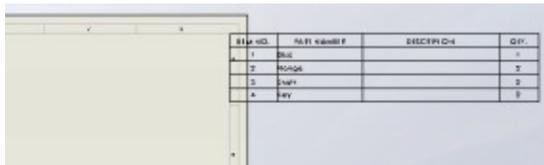
- To create a parts list, click **Annotation** > **Tables** > **Bill of Materials** on the CommandManager.



- Click on the exploded view.
- On the PropertyManager, under the **Table Position** section, check the **Attach to anchor point** option.
- Set the **BOM Type** to **Parts only**.



- Click **OK**. You will notice that the BOM appears outside the sheet.

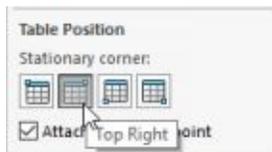
A screenshot showing a BOM table floating outside the main sheet area. The table has four columns: 'ITEM NO.', 'PART NUMBER', 'DESCRIPTION', and 'QTY.'. It contains four rows of data.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	BLU		1
2	HOLE		2
3	WEL		2
4	BEN		2

- Click on the anchor point of the BOM.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Disc		1
2	Flange		2
2	Shaft		2
2	Key		2

- On the PropertyManager, under the **Table Position** section, select **Top Right**.

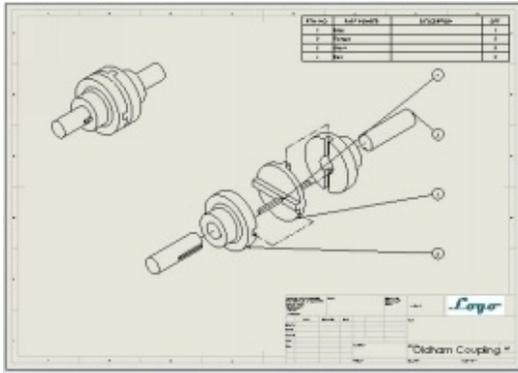


ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Disc		1
2	Flange		2
2	Shaft		2
2	Key		2

- Click the green check on the PropertyManager.

Creating Balloons

- To create balloons, click **Annotation** > **Auto Balloon** on the CommandManager.
- On the PropertyManager, under **Balloon Layout** section, select **Layout Balloons to Right** .
- Click on the exploded view and click **OK** .



4. Save and close the drawing file.

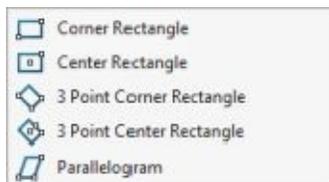
Chapter 5: Sketching

In this chapter, you will learn the sketching tools. You will learn to create:

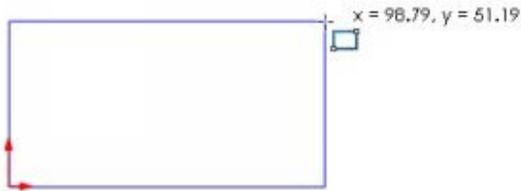
- Rectangles
- Polygons
- Resolve Sketch
- Relations
- Splines
- Ellipses
- Move, rotate, scale, copy, and stretch entities
- Circles
- Arcs
- Circular pattern
- Trim Entities
- Fillets and Chamfers

Creating Rectangles

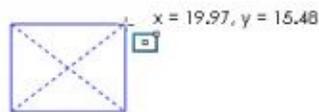
A rectangle is a four sided object. You can create a rectangle by just specifying its two diagonal corners. However, there are various tools to create a rectangle. You can access these tools from the **Rectangle** drop-down available on the **Sketch** CommandManager. These tools are explained next.



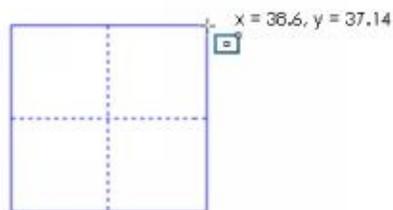
1. On the CommandManager, click **Sketch** > **Sketch**  and select the Front plane.
2. On the CommandManager, click **Sketch** > **Rectangle Drop-down** > **Corner Rectangle** .
3. Select the origin point to define the first corner.
4. Move the pointer diagonally and click to define the second corner.



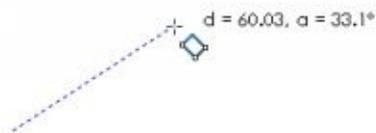
5. On the PropertyManager, click the **Center Rectangle**  icon under the **Rectangle Type** section.
6. Click in the graphics window to define the center point.
7. Move the pointer and click to define the corner.



8. On the PropertyManager, select **From Midpoints** under the **Rectangle Type** section.
9. Select the center point and corner point of the rectangle. Two construction lines are created from the midpoints of the rectangles.



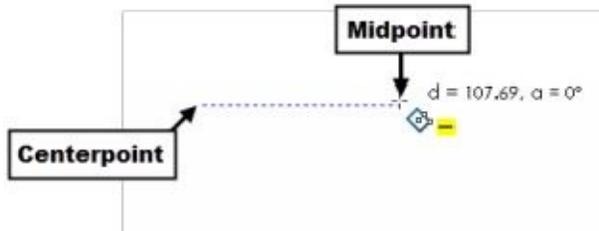
10. On the PropertyManager, click the **3 Point Corner Rectangle**  icon under the **Rectangle type** section. This option creates a slanted rectangle. You can also check the **Add construction lines** option to add construction lines between the corners or midpoints.
11. Select two points to define the width and inclination angle of the rectangle.



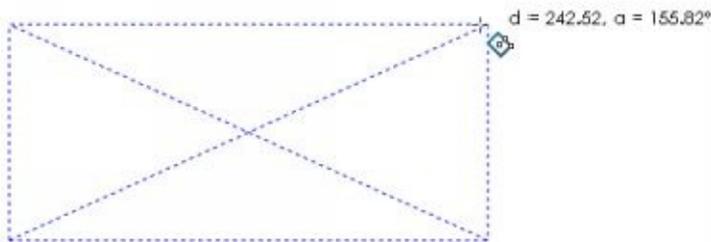
12. Select the third point to define its height.



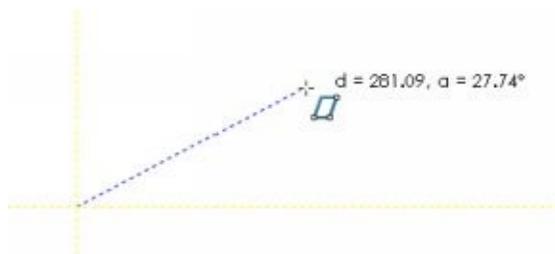
13. On the PropertyManager, click the **3 Point Center Rectangle** icon under the **Rectangle Type** section.
14. Click to define the center point of the rectangle.
15. Move the pointer and click to define the midpoint of one side.



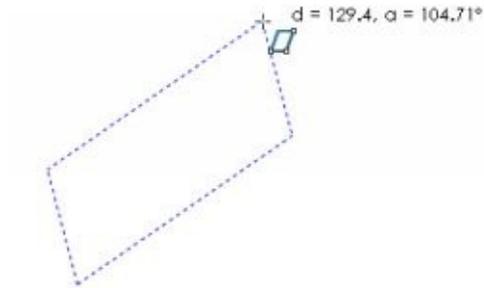
16. Move the pointer and click to define the corner point.



17. On the PropertyManager, click the **Parallelogram** icon under the **Rectangle Type** section. This option creates a parallelogram by using three points that you specify.
18. Select two points to define the width of the parallelogram.



19. Drag the pointer and click to define the height of parallelogram.

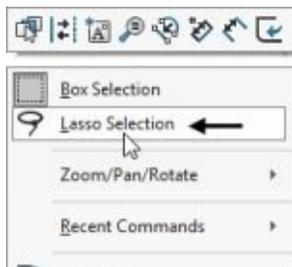


20. Click **Close Dialog** on the PropertyManager.

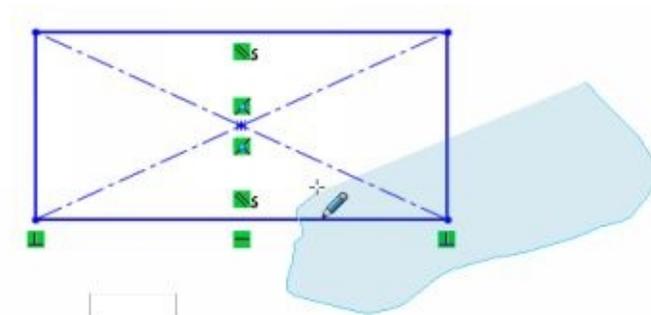
Lasso Selection

The **Lasso Selection**  tool helps you to select multiple elements by dragging the pointer over them.

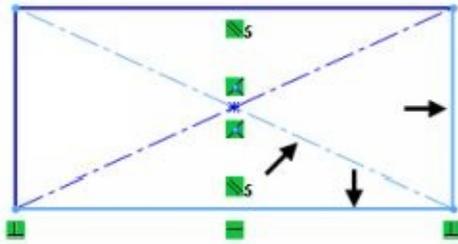
1. Click the right mouse button and select **Lasso Selection**.



2. Click the left mouse button and drag the pointer covering the three elements of the rectangle, as shown.



The three elements will be selected.



3. Press **Delete** to erase the elements of the rectangle.
4. Press Ctrl+A to select all the sketch elements.
5. Press **Delete** to erase the elements.

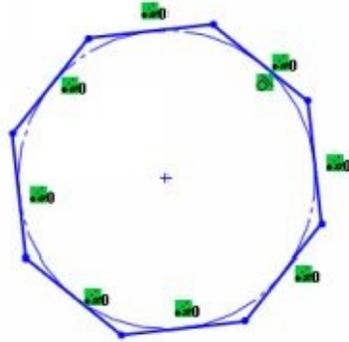
Creating Polygons

A Polygon is a shape having many sides ranging from 3 to 1024. In SOLIDWORKS, you can create regular polygons having sides with equal length. Follow the steps given next to create a polygon.

1. Activate the **Sketch** mode.
2. On the CommandManager, click **Sketch > Polygon** .
3. On the PropertyManager, type **8** in the **Number of Sides** box under the **Parameters** section.
4. Select **Inscribed circle**. This option creates a polygon with its sides touching a dotted circle. You can also select the **Circumscribed circle** option to create a polygon with its vertices touching a dotted circle.
5. Click to define the center of the polygon.
6. Move the pointer and click to define the size and angle of the polygon.

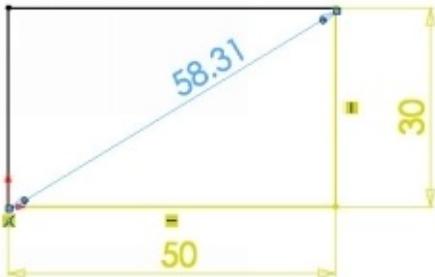
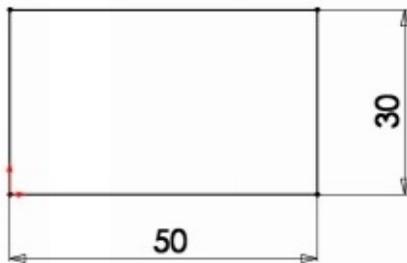


7. Press Esc to deactivate the tool.

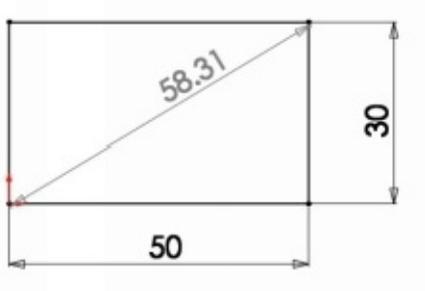


Resolve Sketch

When creating sketches for a part, SOLIDWORKS will not allow you to over-constrain the geometry. The term ‘over-constrain’ means adding more dimensions than required. The following figure shows a fully constrained sketch. If you add another dimension to this sketch (e.g. diagonal dimension), the **Make Dimension Driven** dialog appears.



Now, you have to convert the new dimension into driven. Select **Make this dimension driven** and click **OK**.



Now, if you change the value of the width, the reference dimension along the diagonal updates, automatically.

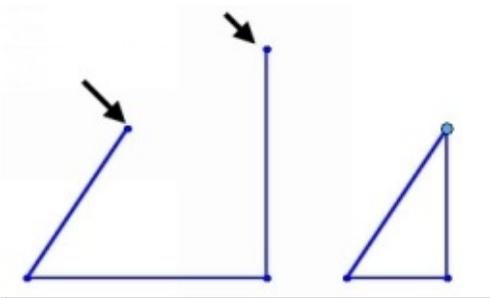
Relations

Relations are used to control the shape of a sketch by establishing relationships between the sketch elements. You can add relations using the **Add Relation** tool or pop-up toolbar or by selecting sketch elements.

Merge

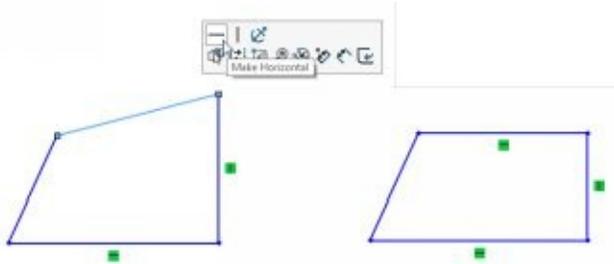
This relation connects a point to another point.

1. On the **Sketch** tab of the CommandManager, click **Delete/Display Relations > Add Relation** .
2. Select two points.
3. Click **Merge**  on the PropertyManager. The selected points will be merged together.

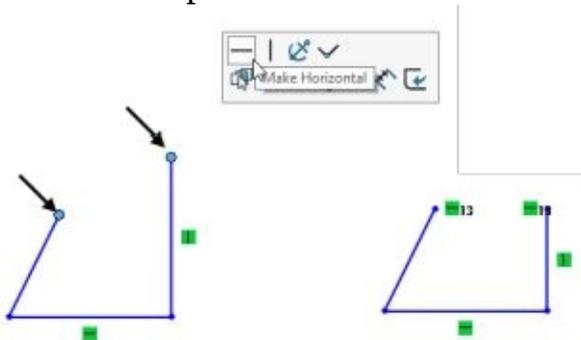


Horizontal

To apply the **Horizontal** relation, click on a line and click the **Horizontal** icon on the context toolbar.



You can also align two points or vertices horizontally. Press the Ctrl key and select the two points. Click the **Horizontal** icon on the PropertyManager.

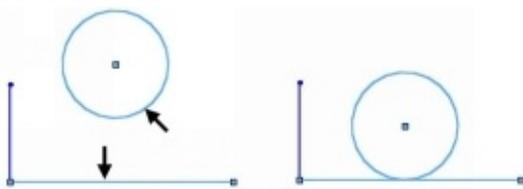


Vertical

Use the **Vertical** relation to make a line vertical. You can also align two vertices vertically by using this relation.

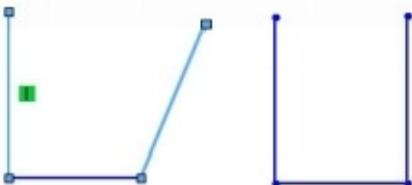
Tangent

This relation makes an arc, circle, or line tangent to another arc or circle. Select a circle, arc, or line. Press the Ctrl key and select another circle, or arc. Click the **Tangent** icon on the PropertyManager. The two elements will be tangent to each other.



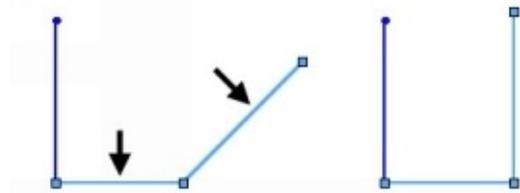
Parallel

Use the **Parallel** relation to make two lines parallel to each other.



Perpendicular

Use the **Perpendicular** relation to make two entities perpendicular to each other.



Midpoint

Use the **Midpoint** relation to make a point coincide with the midpoint of a line or arc.

1. Press the Ctrl key and select a line/arc and a point.
2. Click the **Midpoint** icon on the PropertyManager.



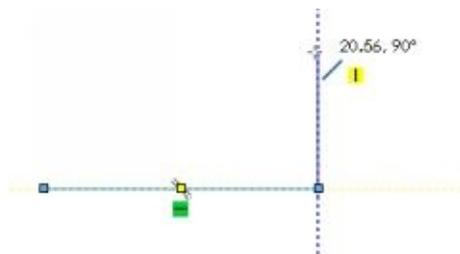
Automatic Relations

SOLIDWORKS automatically adds relations when you create sketch elements.

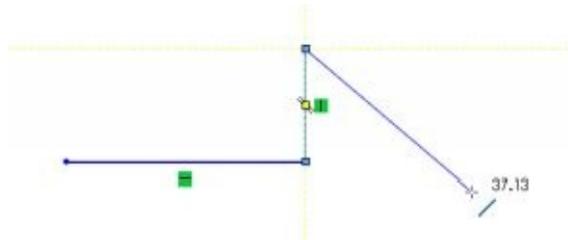
1. Start a new sketch and activate the **Line**  tool from the **Sketch** CommandManager.
2. Click to specify the start point of the line.
3. Move the pointer in the horizontal direction and notice the **Horizontal** relation flag.
4. Click to create a line with the Horizontal relation.



5. Likewise, create a line with the **Vertical** relation.



6. Create an inclined line as shown.

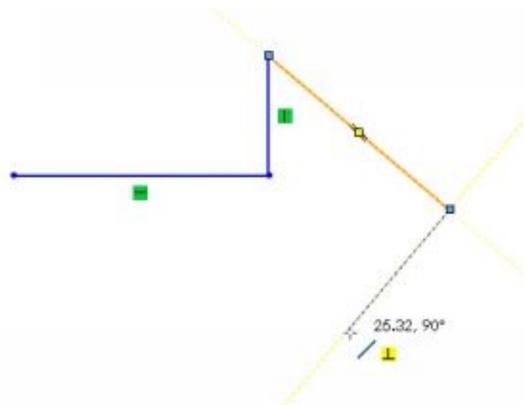


7. Move the pointer along the inclined line and notice the **Collinear** relation flag.



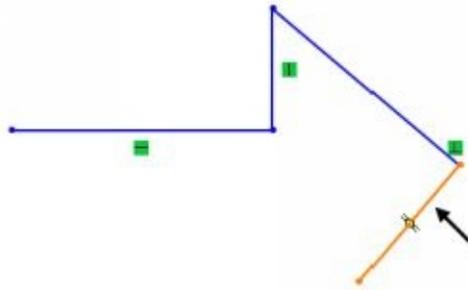
8. Move the pointer in the direction perpendicular to inclined line and notice the Perpendicular relation flag.

9. Click to create the line with the Perpendicular relation.

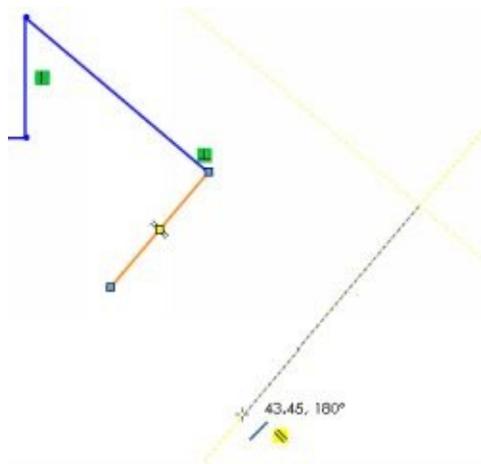


10. Press Esc to deactivate the **Line** tool.

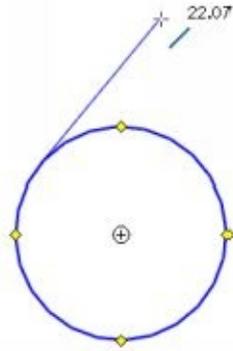
11. Select the perpendicular line.



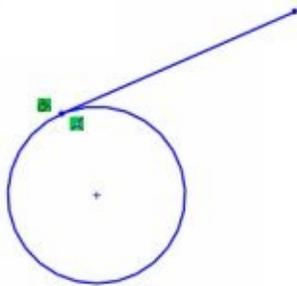
12. Right click and select **Sketch Entities** > **Line**.
13. Click in the graphics window to specify the start point of the line. Notice the yellow inferred lines with reference to the selected line.
14. Move along the yellow inferred line notice the **Parallel** relation flag.



15. Click to create a line parallel to the selected line.
16. Press Esc to deactivate the **Line** tool.
17. On the CommandManager, click **Sketch** > **Circle** , and then create circle.
18. Activate the **Line** tool and click on the circle.
19. Move the pointer around the circle and notice that the line maintains the **Tangent** relation with the circle.

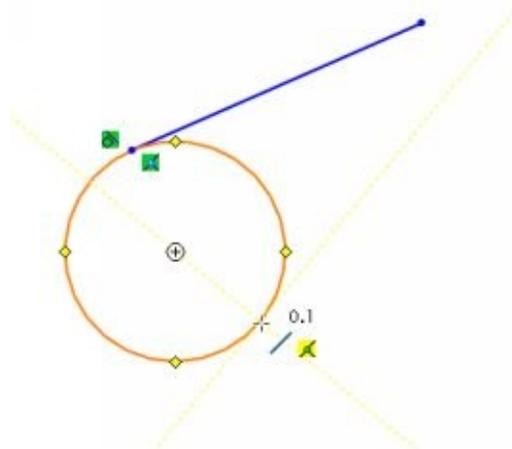


20. Click to create a line which is tangent and coincident to the circle. Right click and select **End chain**.



21. Select a point on the circle and move the pointer.
 22. Move the pointer back to the start point of the line.

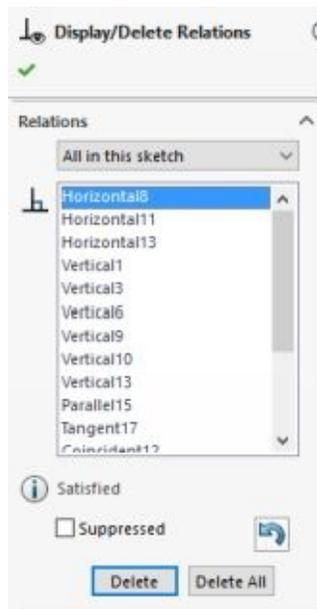
Notice the **Coincident** relation flag and the yellow inferred lines. You can create a line which is tangent or normal to the circle using the inferred lines.



Deleting Relations

You can delete relations by using the following methods

- Select the relation and press **Delete**.
- Select the sketch entity to view all its relations in the PropertyManager. Select the relations from the **Existing Relations** section of the PropertyManager and press **Delete**.
- On the CommandManager, click **Sketch > Display/Delete Relations** . On the PropertyManager, select the relations from the Relations section and click **Delete**. You can also use the **Delete All** button to delete all the relations.



Hiding Relations

To hide sketch relations by clicking the **Hide/Show Items**  drop-down on the **View Heads-Up** Toolbar, and then deactivating the **View Sketch Relations**  icon.

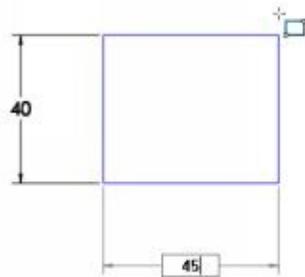
Sketch Numeric Input

The **Sketch Numeric Input** tool automatically adds dimensions to the sketch elements when you create them.

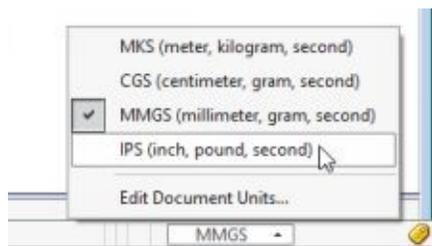
1. Click the Right Mouse button and select **Sketch Numeric Input** from the shortcut menu.



2. Activate a sketching tool and check the **Add Dimensions** option on the PropertyManager. In case of a circle, you can select the **Diameter dimensions** option to add the diameter dimension to the circle.
3. Start creating the sketch elements. Notice that you can enter the dimension values of the sketch elements. Use the Tab key to walk through the dimension values.



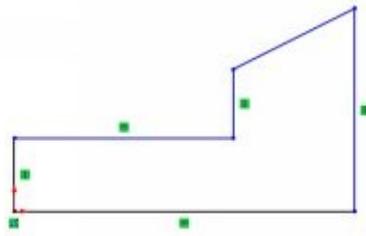
You can change the dimension units from flyout located at the bottom right corner of the window. The dimension units are reflected in the sketch.



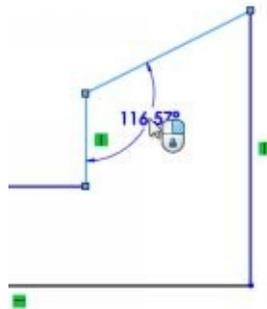
Replace Entity

The **Replace Entity** tool replaces a sketch entity with another one. You can use this tool instead of deleting a sketch entity and creating another one. This tool transfers all the associated features of the replaced entity to the replacement entity. This will avoid any parent-child errors when you modify the sketch. The following example shows the significance of this tool.

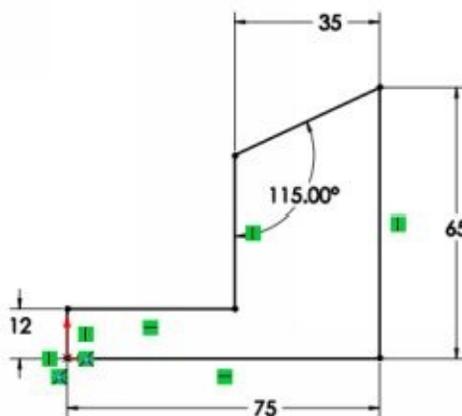
1. Start a new sketch on the Front plane and create the sketch, as shown.



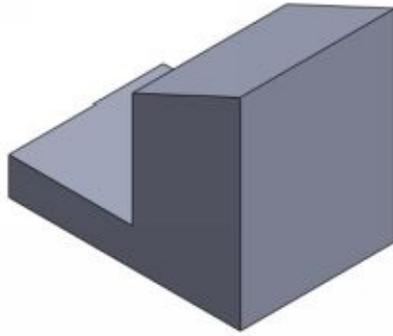
2. On the **Sketch** CommandManager, click the **Smart Dimension**  tool.
3. Select the inclined line and the vertical line connected to it.



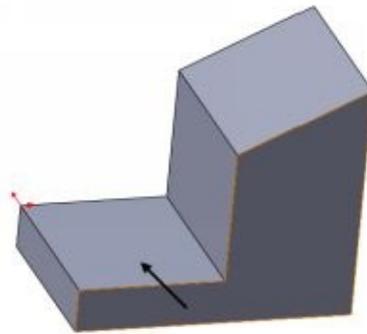
4. Place the dimension, type 115 in the **Modify** box, and then click **OK**.
5. Select the inclined line and move the pointer. Notice that the orientation of the dimension changes as per the movement of the pointer.
6. Move the pointer vertically upward and right click to lock the orientation of the dimension. Now, the orientation of the dimension does not change with the cursor movement.
7. Place the dimension the specify its value as 35.
8. Likewise, add other dimensions to the sketch.



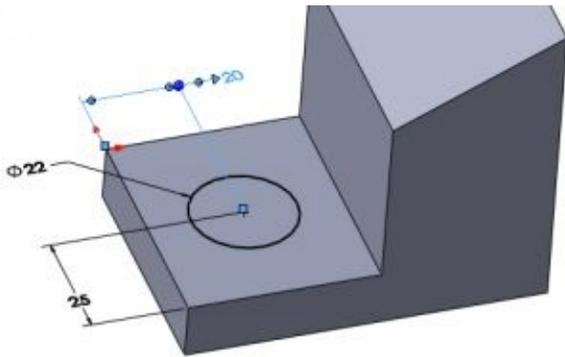
9. Exit the sketch, and then extrude it up to 50 mm distance.



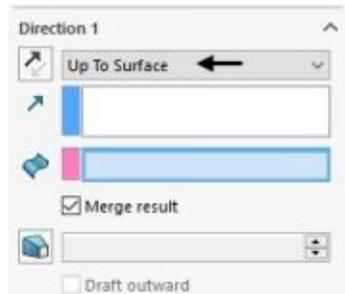
10. Activate the **Extrude Boss/Base** tool and click on the horizontal face of the model.



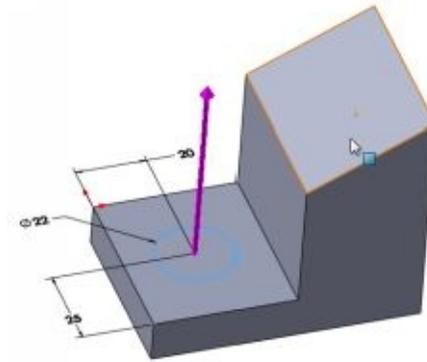
11. Create the sketch as shown. Exit the sketch.



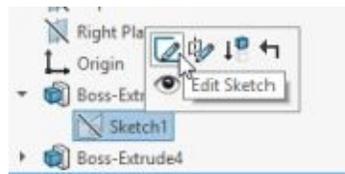
12. On the PropertyManager, select **Up To Surface** from the **Direction 1** drop-down.



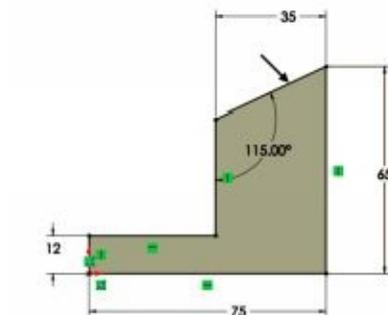
13. Select the inclined face of the model, and then click **OK**.



14. In the FeatureManager Design Tree, expand the Boss-Extrude1, click on the Sketch1, and then select the **Edit Sketch**.

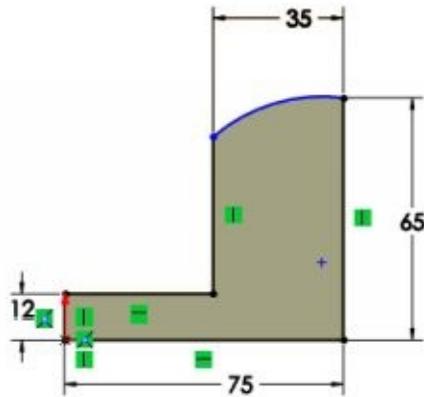


15. Select the inclined line and press **Delete**.

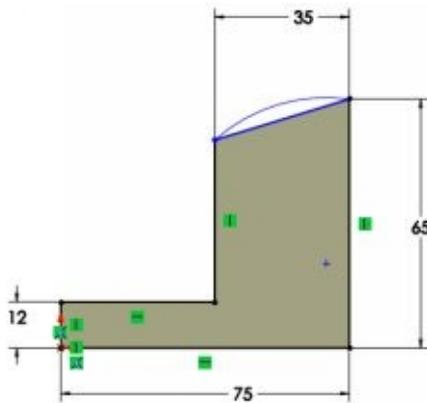


16. Click **Yes**.

17. Create a 3-point arc closing the sketch loop, as shown.



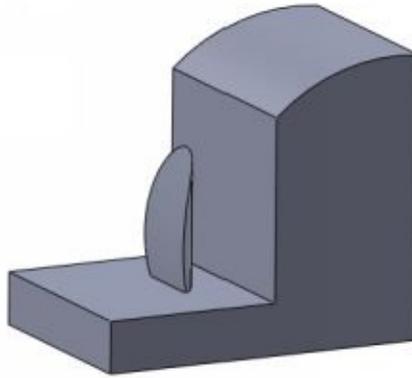
18. Click **Exit Sketch**. The **What's Wrong** dialog pops up on the screen. Its shows that the end face cannot terminate the extruded feature.
19. Press Ctrl+Z thrice to undo the changes.
20. Again, click on the **Sketch1** in the FeatureManager Design Tree and select **Edit Sketch**.
21. Create an arc, as shown.



22. Right click and select **Replace Entity** from the shortcut menu.



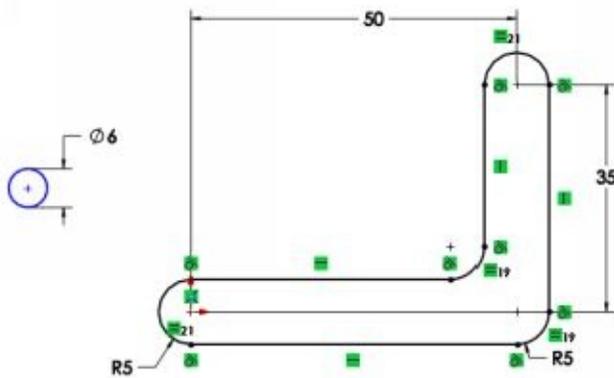
23. Select the inclined line as the entity to be replaced.
24. Select the arc as the replacing entity.
25. Check the **Make construction** option on the PropertyManager, and the click **OK**.
26. Exit the sketch and notice that end face of the extruded feature is replaced.



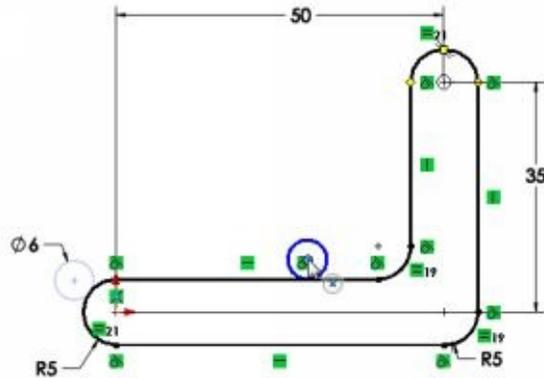
Make Path

The Make Path tool creates path combining all the sketch entities. When you apply a relation between anyone of the entity of the path and other individual entity, the relation will be persistent through all the entities of the path.

1. Create a closed sketch and a circle, as shown.



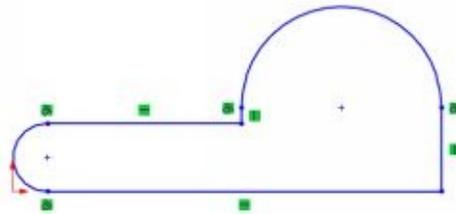
2. On the Menu bar, click **Tools > Sketch Tools > Make Path**.
3. Select all the entities of the closed loop, and click **OK** on the PropertyManager.
4. Apply the **Tangent** relation between the circle and anyone of the entity of the closed loop.
5. Click and drag the circle notice that it maintains the **Tangent** relation with all the entities of the path.



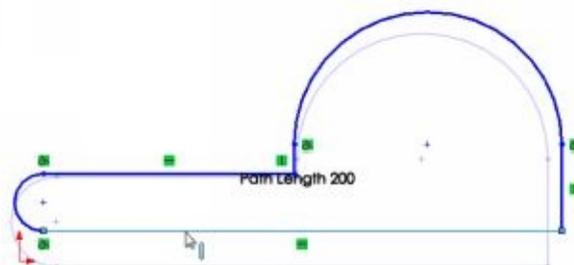
Path Length Dimension

The **Path Length Dimension** tool defines a fixed length for all the sketch entities combined together.

1. Create the sketch as shown.



2. On the **Sketch** CommandManager, click **Smart Dimension** > **Path Length Dimension**.
3. Drag a selection box across all the sketch entities to select them.
4. Click OK on the PropertyManager.
5. Double click on the Path Length dimension, type 200, and press Enter.
6. Click and drag the bottom horizontal line of the sketch. Notice that the path length is fixed and the sketch shape changes.

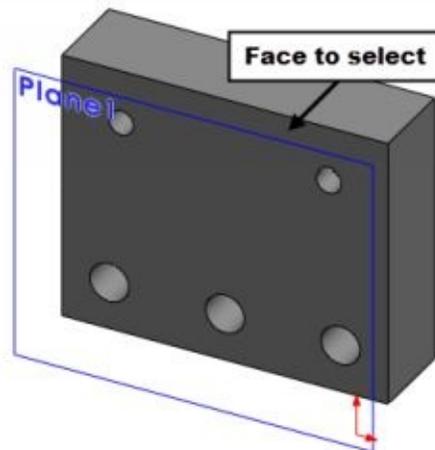


Convert Entities

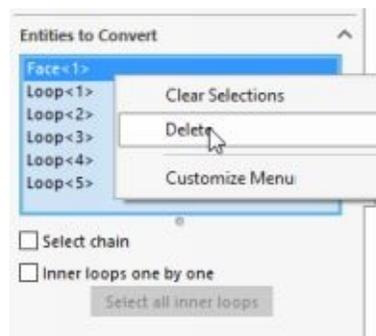
The **Convert Entities** tool helps you to use the edges of the part geometry to

create sketch elements.

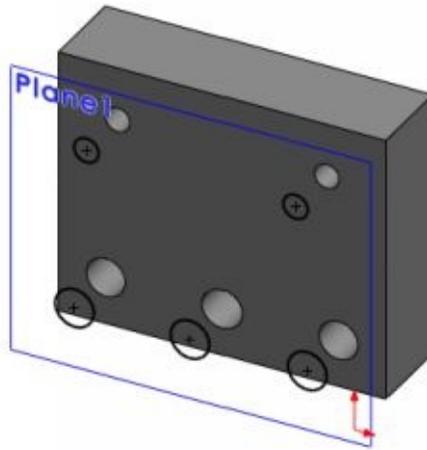
1. Start a sketch on the plane offset to the model face, as shown.
2. On the **Sketch** CommandManager, click the **Convert Entities**  button.
3. Select the face of the model, as shown.



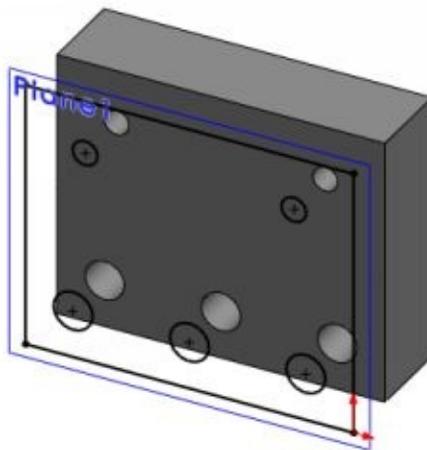
4. On the PropertyManager, click the **Select all inner loops** button to select the inner loops of the face.
5. Click on **Face<1>** in the **Entities to convert** selection box, and then select **Delete**.



6. Click **OK**  on the PropertyManager. All the internal loops of the selected face are projected onto the sketch plane.



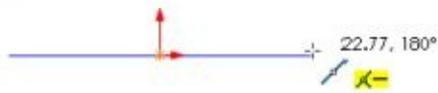
7. Select the previously selected face, and then click the **Convert Entities** button on the **Sketch** CommandManager. The outer loop of the face is converted into sketch elements.



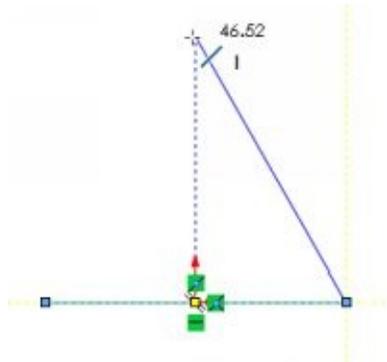
Midpoint line, Perimeter Circle, and SketchXpert

In this tutorial, you will create a sketch using the **Midpoint line** and **Perimeter Circle** tools.

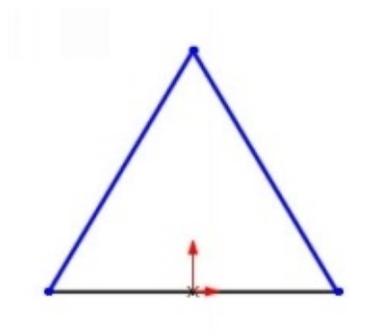
1. Activate the **Sketch** mode.
2. On the CommandManager, click **Sketch** > **Line drop-down** > **Midpoint Line** .
3. Click on the origin point to define the midpoint of the line.
4. Move the pointer horizontally and click.



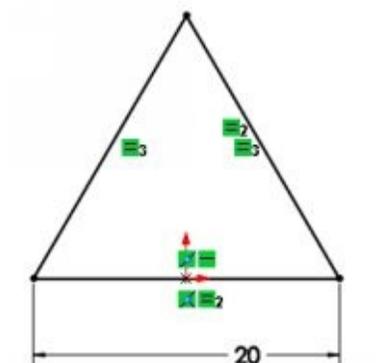
5. Move the pointer upward and click when a dotted line appear from the midpoint of the horizontal line.



6. Create another line closing the sketch.
7. Press Esc to deactivate the **Line** tool.



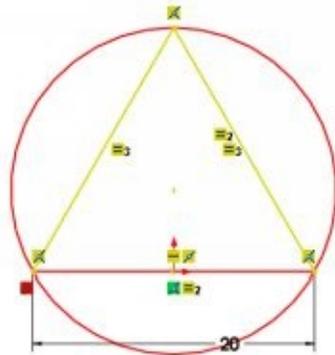
8. Apply the **Equal** relation between the three the line segments and add a dimension to one segment.



9. On the CommandManager, click **Sketch** > **Circle drop-down** > **Perimeter**

Circle

- Click on the endpoints of three lines. The circle is created passing through the endpoints. Make sure that you click the endpoints when the **Coincident**  symbol appears.



Notice that the sketch is over-constrained. You can solve an over-constrained sketch using the SketchXpert.

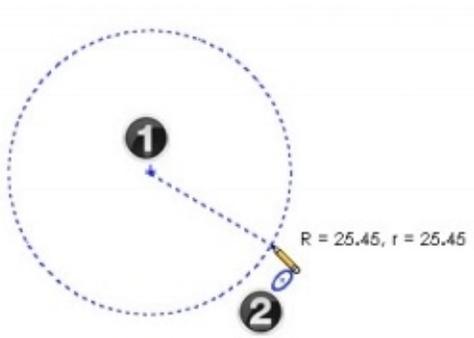
- On the Menu Bar, click **Tools > Sketch Tools > SketchXpert**.
- On the **SketchXpert** PropertyManager, click the **Diagnose** button.
- Under the **Results** section, click the **Next**  button to go through the different results.
- Click **Accept** when the 1 of 3 result is displayed. It deletes the **Tangent** relation from the sketch.
- Click **OK** on the PropertyManager.
- Click the right mouse button and select **Recent Commands > Perimeter Circle**.
- Select the three lines to create a circle tangent to them.



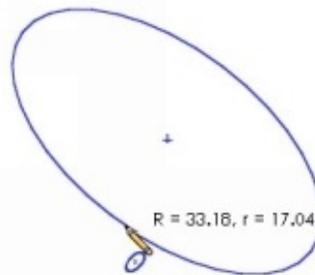
Ellipses

Ellipses are also non-uniform curves, but they have a regular shape. They are actually splines created in regular closed shapes.

1. Activate the **Sketch** mode.
2. On the CommandManager, click **Sketch** > **Ellipse** drop-down > **Ellipse** .
3. Pick a point in the graphics window to define the center of the ellipse.
4. Move the pointer and click to define the radius and orientation of the first axis.

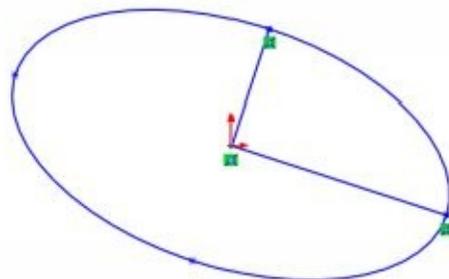


5. Move the pointer and click to define the radius of the second axis.

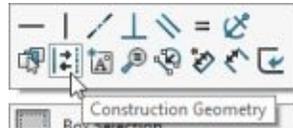


Click and drag the ellipse the notice that it is under-defined. You need to add dimensions and relations to fully define the ellipse.

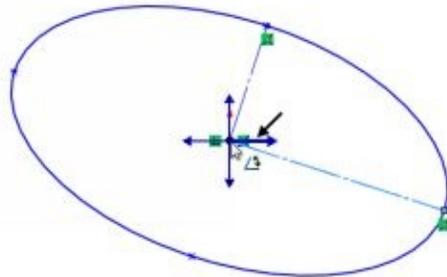
6. Activate the **Line** tool and select the top quadrant point of the ellipse.
7. Move the pointer downward and select the centerpoint.
8. Move the pointer toward right and select the quadrant point.



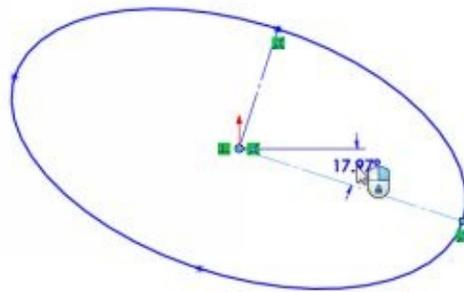
9. Press the Ctrl key and select the two lines.
10. Select **Perpendicular**  from the PropertyManager.
11. Right click and select **Construction Geometry**.



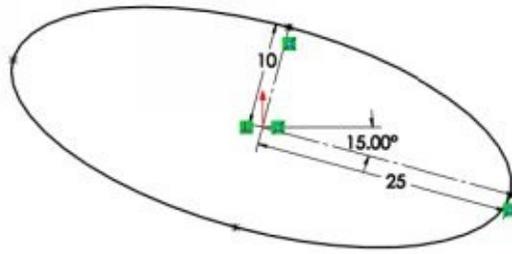
12. On the **Sketch** CommandManager, click the **Smart Dimension**  tool.
13. Select the major axis line of the ellipse.
14. Select the origin point of the sketch. Notice the horizontal and vertical vectors of the sketch.
15. Select the horizontal vector of the sketch. The angled dimension is attached to the pointer.



16. Move the pointer and place the angled dimension.

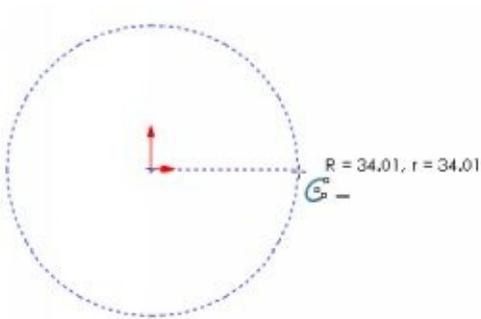


17. Type 15 and click **OK**.
18. Add the major and minor axes dimensions to the sketch. This fully-defines the sketch.

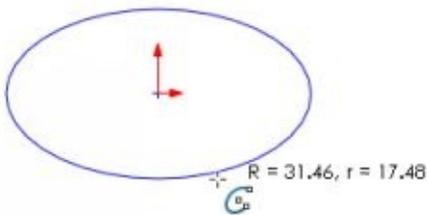


Partial Ellipse

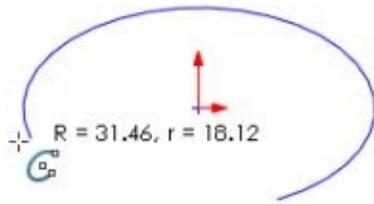
1. On the CommandManager, click **Sketch** > **Ellipse drop-down** > **Partial Ellipse**.
2. Click to define the center of the partial ellipse.
3. Move the pointer and click to define the radius and orientation of the first axis.



4. Move the pointer and click to define the radius of the second axis and start point.



5. Rotate the pointer and click to define end point.

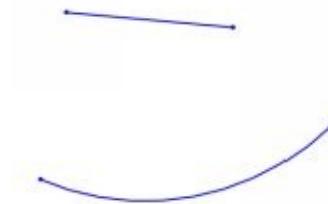


6. Click **Close Dialog** on the PropertyManager.

Conic

The **Conic** tool is used to create conic curve. You can use a conic curve to connect two open sketch entities.

1. Create a line and arc, as shown.



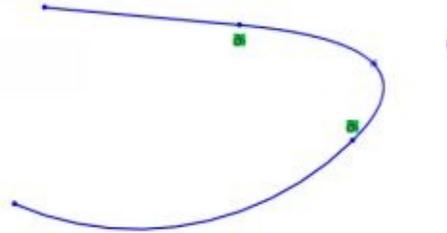
2. On the **Sketch** CommandManager, click **Ellipse** drop-down > **Conic**.
3. Select the endpoints of the line and arc.
4. Move the pointer and notice the dotted lines. These lines are tangent to the line and arc.
5. Select the intersection point of the dotted tangent lines. Move the pointer and notice the rho value displayed. The rho value defines the type of conic created.



6. Click to create the conic curve. Now, you have to specify the Rho value of the conic curve.
7. Expand the Parameters section of the PropertyManager, and then type 0.5

in the ρ box.

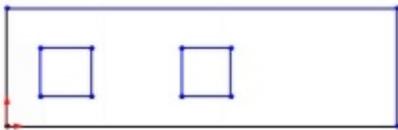
8. Click **Close Dialog**  on the PropertyManager.



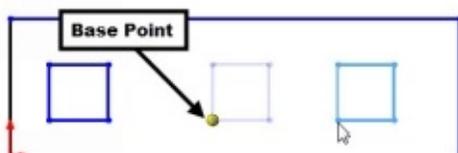
Move and Rotate Entities

The **Move Entities** tool moves a selected object(s) from one location to a new location without changing its orientation. To move objects, you must activate this tool and select the objects from the graphics window. After selecting objects, you must define the 'base point' and the 'destination point'.

1. Create the sketch, as shown below.



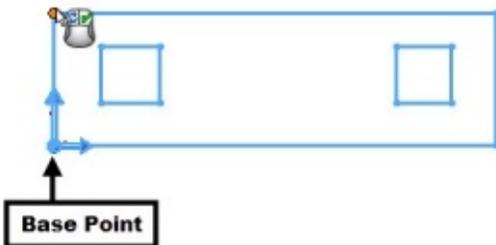
2. Click **Sketch > Move Entities**  on the CommandManager.
3. Select the square located at the right-side by dragging a selection box, and then right-click to accept the selection.
4. On the PropertyManager, under the **Parameter** section, select **From/To**.
5. Select the lower corner point as the base point.
6. Move the pointer toward right and pick a point as shown below. This moves the square to the new location.



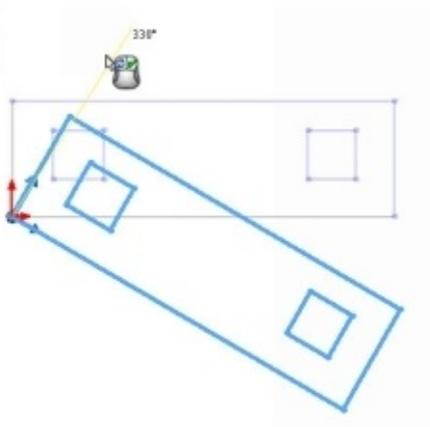
Rotate Entities

The **Rotate Entities** tool rotates an object or a group of objects about a base point.

1. On the CommandManager, click **Sketch** > **Move Entities** > **Rotate Entities** .
2. Drag a selection box covering the sketch elements, and click the right mouse button.
3. Click to define the base point of the rotation.



4. Click again and rotate the sketch entities. You can also type-in the rotation angle on the PropertyManager.

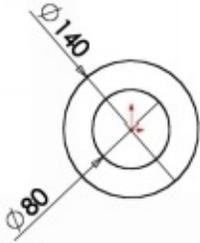


5. Click **OK** on the PropertyManager.

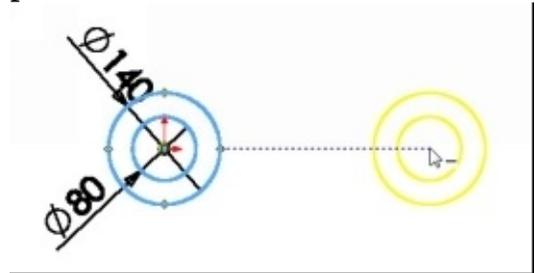
Copy Entities

The **Copy Entities** tool is used to copy objects and place them at a required location. This tool is similar to the **Move Entities** tool, except that object will remain at its original position and a copy of it will be placed at the new location.

1. Draw two circles of 80 mm and 140 mm diameter, respectively.

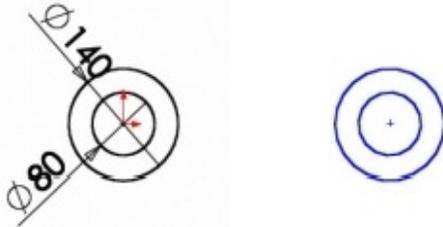


2. Click **Sketch** > **Move Entities** > **Copy Entities**  on the CommandManager.
3. Select the two circles, and then right-click to accept the selection.
4. Select the center of the circle as the base point.



5. Move the pointer toward right and click.

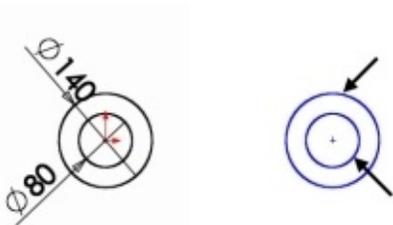
This creates a copy of the circles at the new location.



Scale Entities

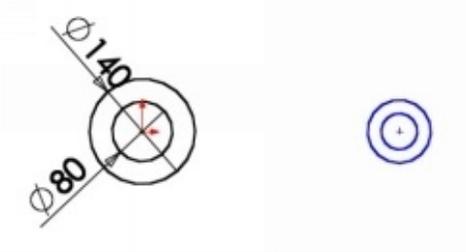
The **Scale Entities** tool changes the size of objects. It reduces or enlarges the size without changing the shape of an object.

1. Click **Sketch** > **Move Entities** > **Scale Entities**  on the CommandManager.
2. Select the circles shown below and right-click to accept the selection.

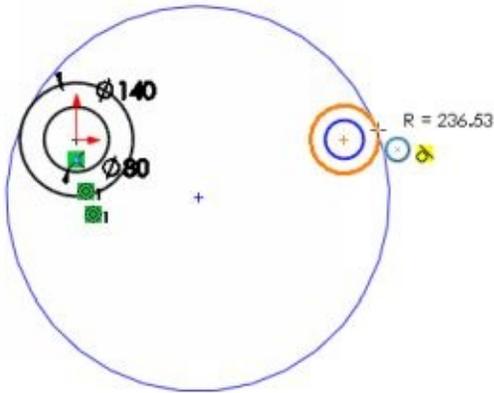


3. Select the center point of the selected circles as the base point.

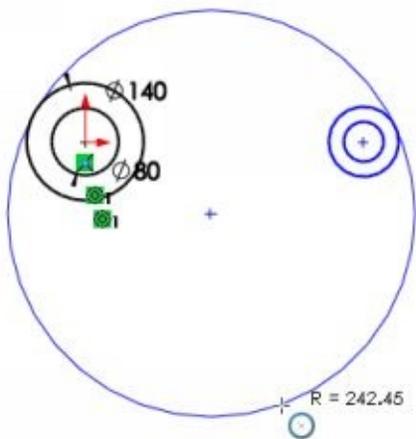
4. Type 0.6 in the **Scale Factor** box under the **Parameters** section of the PropertyManager.
5. Uncheck the **Copy** option on the PropertyManager and click **OK**.



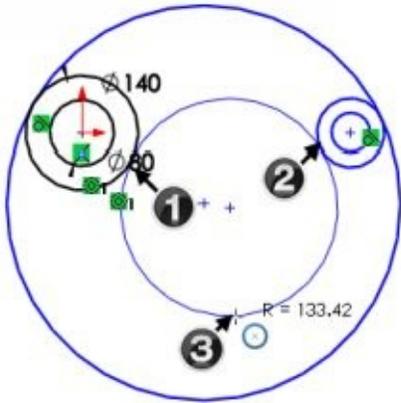
6. Click **Sketch** > **Circle drop-down** > **Perimeter Circle**  on the CommandManager.
7. Select the two circles shown below to define the tangent points.



8. Move the pointer down and click to create the circle.



9. Likewise, create another circle by selecting the points as shown.

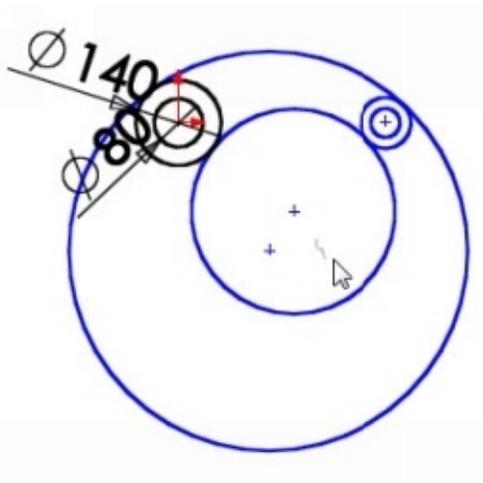


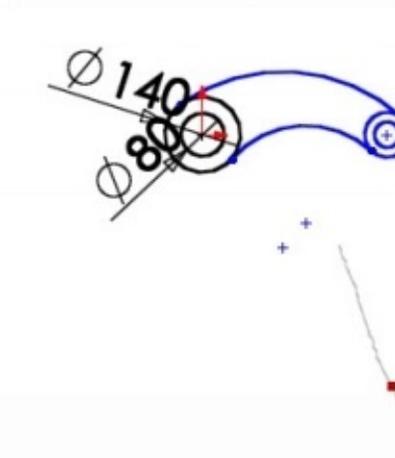
10. Click **Close Dialog** on the PropertyManager.

Trim Entities

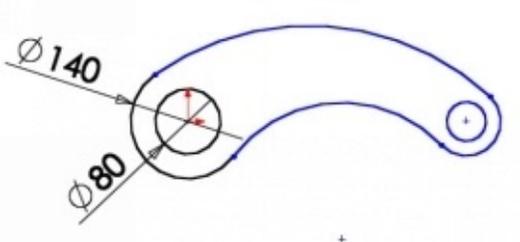
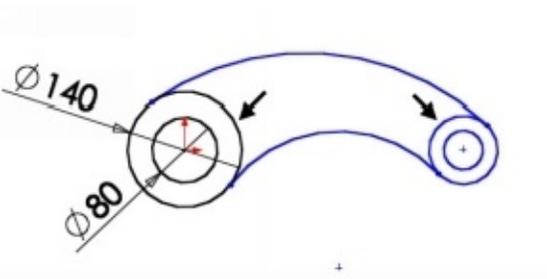
When an object intersects with another object, you can remove its unwanted portion by using the **Trim Entities** tool.

1. Click **Sketch > Trim Entities**  on the CommandManager.
2. On the PropertyManager, click the **Power trim**  icon.
3. Press and hold the left mouse button and drag the pointer across the unwanted portions of the circles as shown.

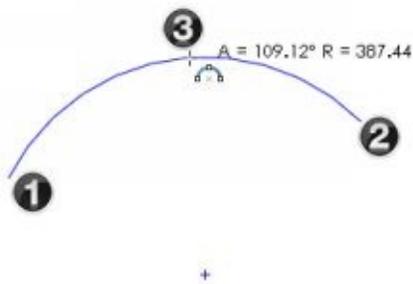




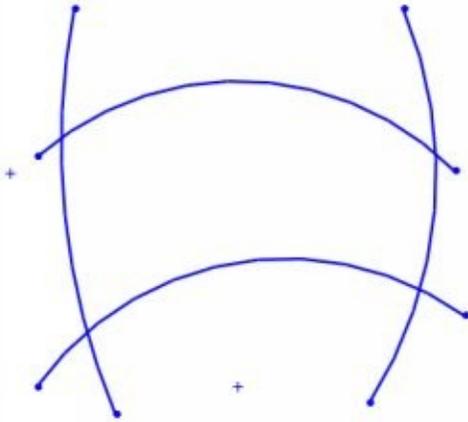
4. On the PropertyManager, click the **Trim Closest**  icon.
5. Click on the inner portions of the circles, as shown.



6. Click **Close Dialog**  on the PropertyManager.
7. On the CommandManager, click **Sketch > Arc drop-down > 3 Point Arc** .
8. Select three points in the graphics window, as shown.



9. Create other arcs using the **3 Point Arc** tool, as shown.

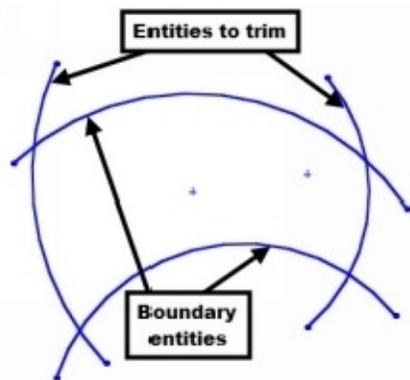


10. On the CommandManager, click **Sketch > Trim Entities** .

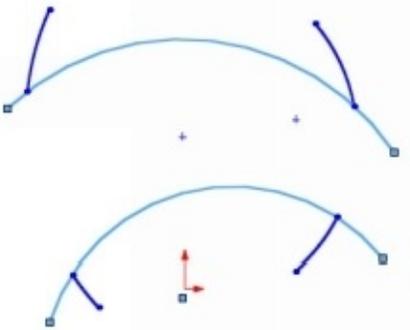
11. On the PropertyManager, click the **Trim away inside**  icon.

12. Click on the boundary entities, as shown.

13. Click on the entities to trim, as shown.



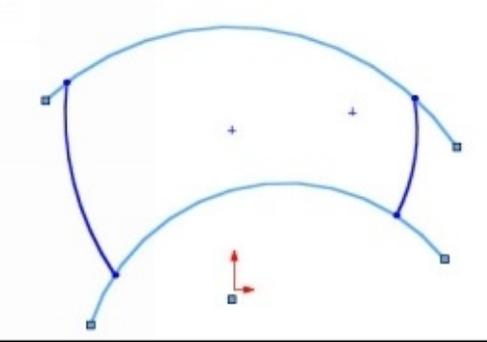
The inner portions of the entities are trimmed.



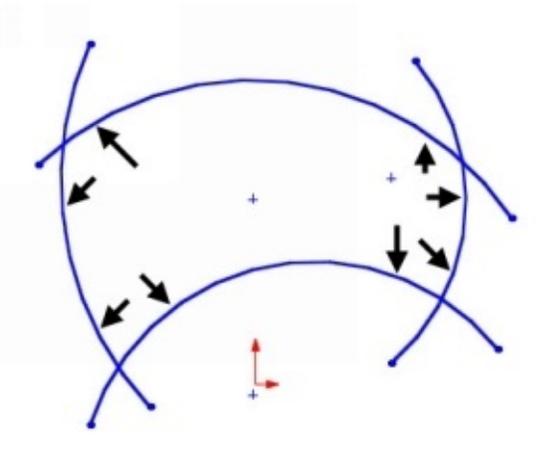
14. Press **Ctrl+Z** twice to undo the trim operations (or) click **Undo**  twice on

the Quick Access Toolbar.

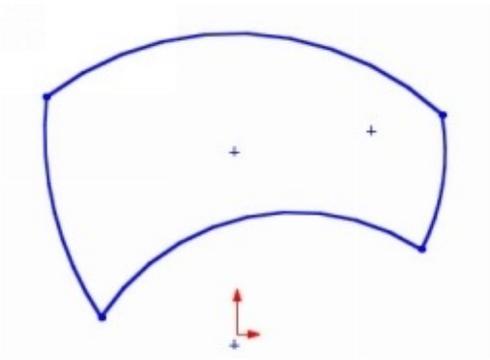
15. Activate the **Trim Entities**  tool and click the **Trim away outside**  icon on the PropertyManager.
16. Click on the boundary and trim entities. The outer portions are trimmed, as shown.



17. Undo the trim operations.
18. Activate the **Trim Entities**  tool and click the **Corner**  icon on the PropertyManager.
19. Click on the intersecting entities, as shown.



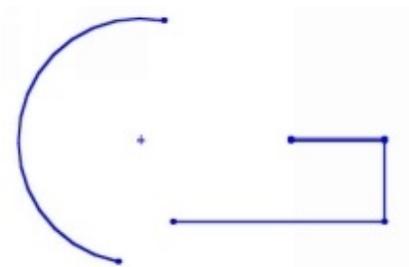
The entities are trimmed to form a corner.



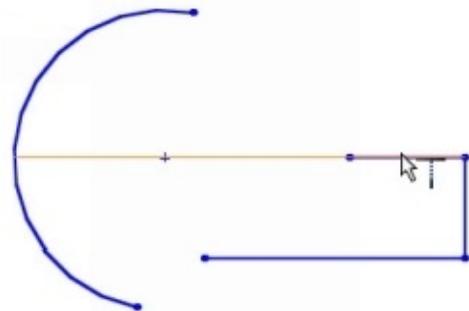
Extend Entities

The **Extend Entities** tool is similar to the **Trim Entities** tool but its use is opposite of it. This tool is used to extend lines, arcs and other open entities to connect to other objects.

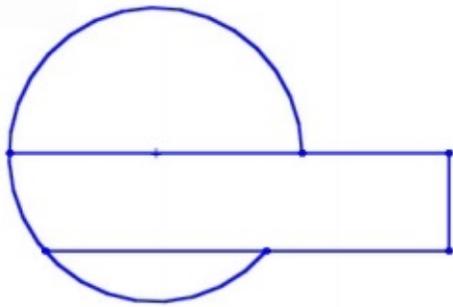
1. Create a sketch as shown below.



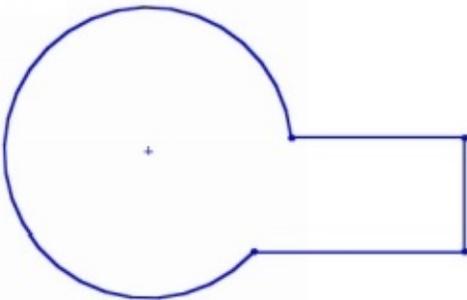
2. Click **Sketch** > **Trim Entities** > **Extend Entities**  on the CommandManager.
3. Select the horizontal open line. This will extend the line up to arc.



Likewise, extend the other elements, as shown.



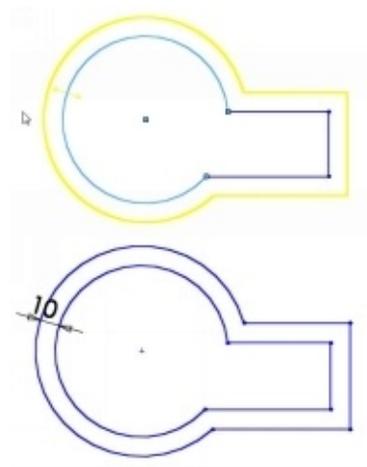
4. Trim the unwanted portions.



Offset Entities

The **Offset Entities** tool creates parallel copies of lines, circles, arcs and so on.

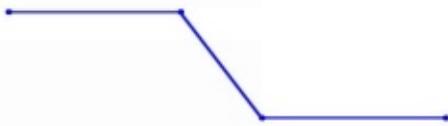
1. On the CommandManager, click **Sketch** tab > **Offset Entities** .
2. Check the **Select Chain** option on the PropertyManager to select connected elements.
3. Select an entity and notice that all the connected entities are selected.
4. Type-in the offset distance on the PropertyManager.
5. Click inside or outside the sketch to define the offset side.



You can also reverse the offset direction by double-clicking on the offset dimension and selecting the **Reverse the sense of direction** button.



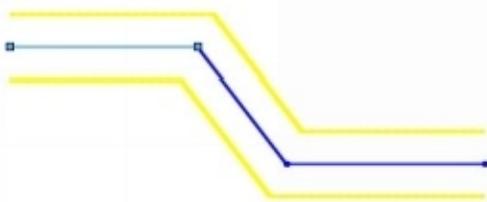
6. Create a line chain as shown.



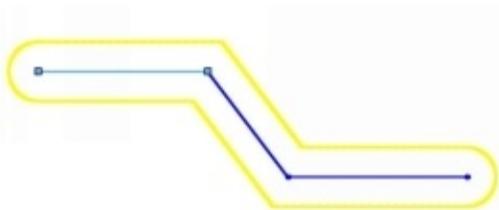
7. Activate the **Offset Entities** tool.

8. Select a line segment and notice that entire chain is selected.

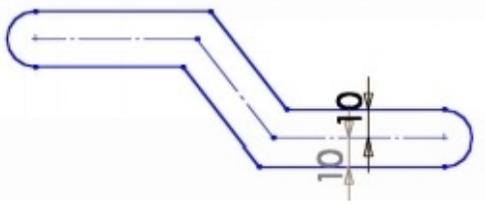
9. Check the **Bi-directional** option. Notice that the offset preview appears on both sides.



10. Check the **Cap ends** option and select **Arcs**. Notice that the ends are capped with arcs.



11. Check the **Base geometry** option under the **Construction geometry** section and click **OK**. The line chain is offset on both sides, and then converted into construction entity.



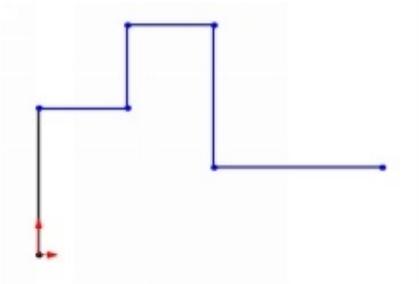
TUTORIAL 1

This tutorial teaches you to use **Sketch Fillet**, **Sketch Chamfer**, **Mirror Entities**, and **Tangent Arc** tools.

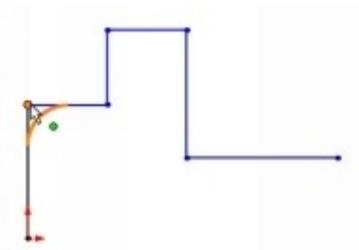
Sketch Fillet

The **Sketch Fillet** tool converts the sharp corners into round corners.

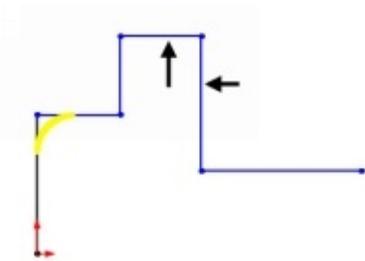
1. Draw the lines as shown below.



2. Click **Sketch > Sketch Fillet**  on the CommandManager.
3. On the PropertyManager, type-in a value in the **Radius** box.
4. Click on the corner, as shown.



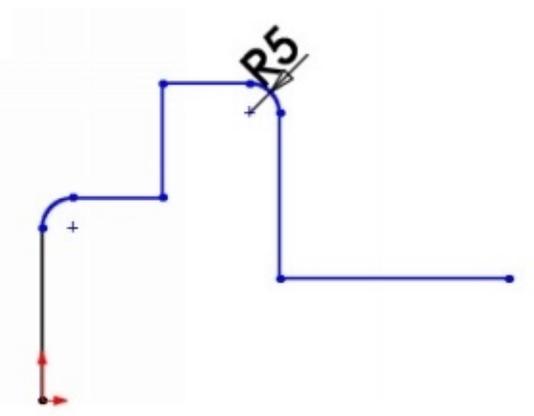
5. Select the vertical and horizontal lines, as shown.



The **Keep constrained corners** option retains all the existing constraints.

The **Dimension each fillet** option dimensions all the fillets created in single instance. If you uncheck this option, only a single fillet is dimensioned and the **Equal** relation is applied between all other fillets.

6. Click **OK** on the PropertyManager. The fillets are created. The **Keep visible** icon on the PropertyManager will keep it open. You can select other corners or click the **Close** icon.



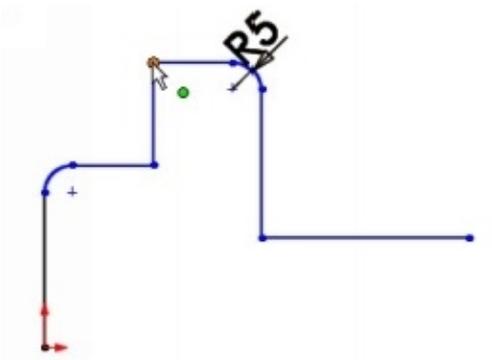
Sketch Chamfer

The **Sketch Chamfer** tool replaces the sharp corners with an angled line. This tool is similar to the **Sketch Fillet** tool, except that an angled line is placed at the corners instead of rounds.

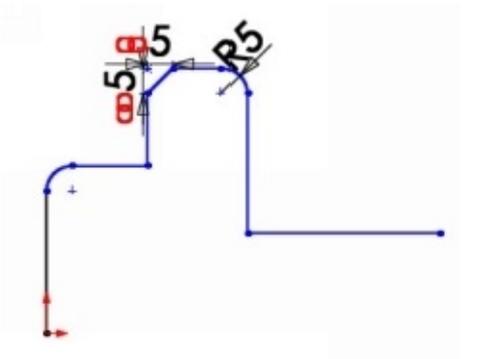
1. Click **Sketch** > **Sketch Fillet** > **Sketch Chamfer** on the CommandManager.
2. On the PropertyManager, select the **Distance-distance** option and check the **Equal distance** option.

3. Type-in a value in the **Distance** box.

4. Click on the corner as shown.



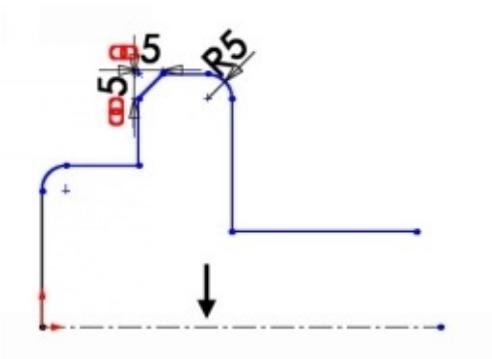
5. Click **OK** on the PropertyManager.



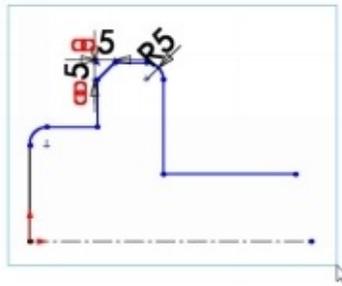
Mirror Entities

The **Mirror Entities** tool creates a mirror image of objects. You can create symmetrical sketches using this tool.

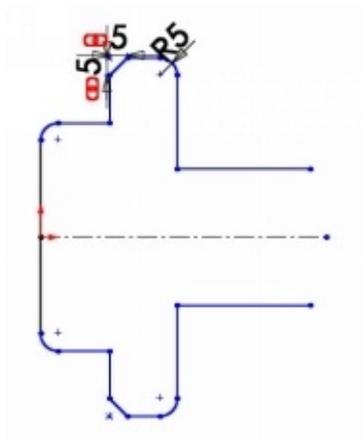
1. Create a horizontal centerline passing through the origin.



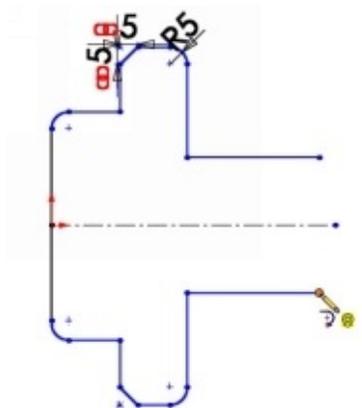
2. On the Command Manager, click **Sketch > Mirror Entities** .
3. Drag a selection box covering all the sketch entities.



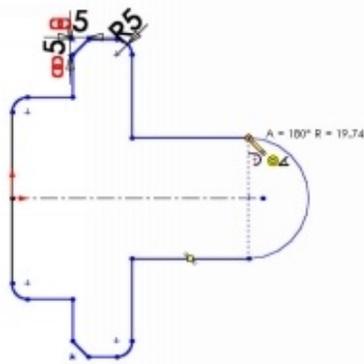
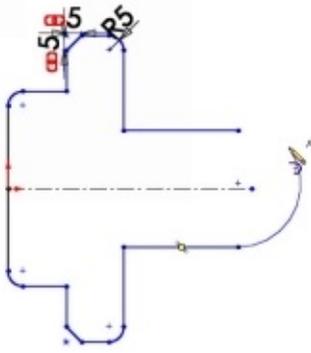
4. On the PropertyManager, click in the **Mirror about** box and select the centerline passing through the origin.
5. Click **OK** to mirror the selected entities.



6. On the CommandManager, click **Sketch > Arc drop-down > Tangent Arc** .
7. Select the start point of the arc as shown below.



8. Move the pointer and select the end point of the arc, as shown below.

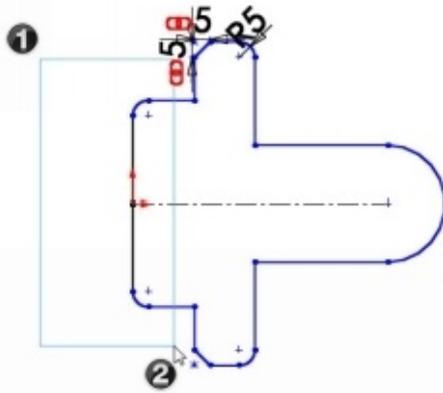


9. Click **OK** on the PropertyManager.

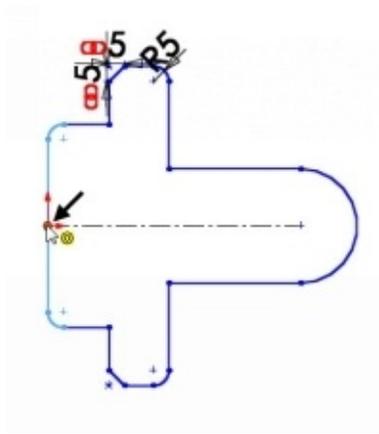
Stretch Entities

The **Stretch Entities** tool lengthens or shortens drawings or parts of sketches. Note that you cannot stretch circles using this tool. In addition, you must select the portion of the drawing to be stretched by dragging a selection box.

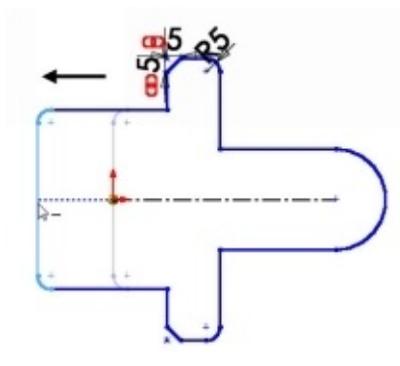
1. On the CommandManager, click **Sketch** > **Move Entities** > **Stretch Entities** .
2. Drag a crossing window to select the objects of the sketch, as shown. Click the right mouse button.



- Select the base point as shown below.



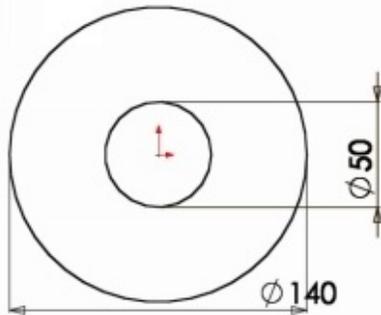
- Move the pointer toward left and click to stretch the sketch.



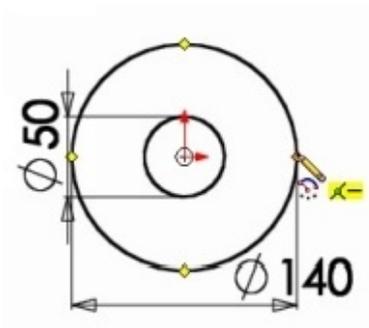
Circular Sketch Pattern

The **Circular Sketch Pattern** tool creates an arrangement of objects around a point in circular form.

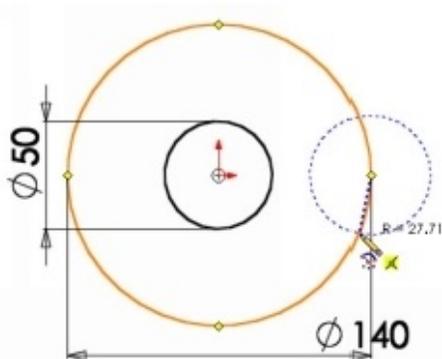
1. Create two concentric circles of 140 and 50 diameters.



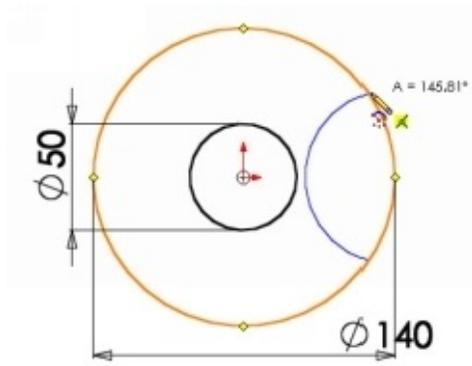
2. On the CommandManager, click **Sketch** > **Arc drop-down** > **Centerpoint Arc** .
3. Select the quadrant point of the circle as shown below.



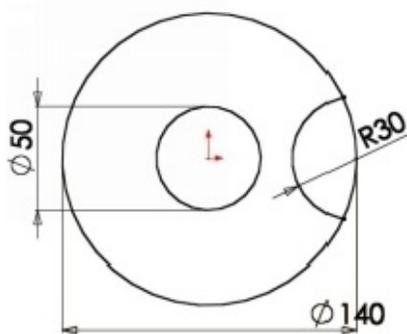
4. Move the pointer down and click on the circle when the **Coincident** symbol appears.



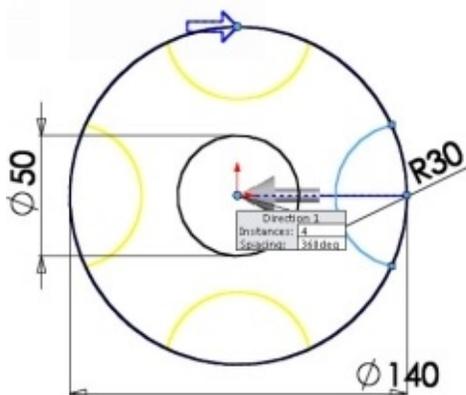
5. Move the pointer leftwards and up.
6. Click on the circle when the Coincident symbol appears.



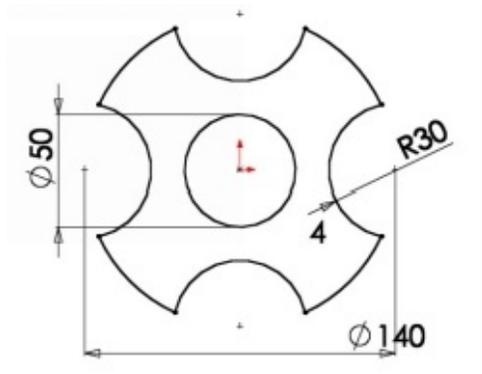
7. Click **Close Dialog** on the PropertyManager.
8. Dimension the arc with radius 30.



9. On the CommandManager, click **Sketch** > **Linear Sketch Pattern** > **Circular Sketch Pattern**.
10. Select the arc. The preview of the circular pattern appears as the center of the pattern is selected, automatically.



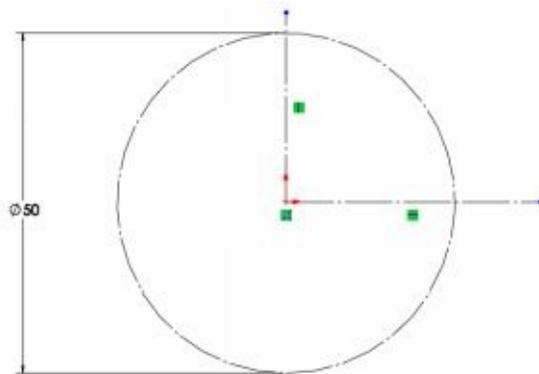
11. Leave the default options on the PropertyManager and click **OK**.
12. Trim the unwanted portions as shown below.



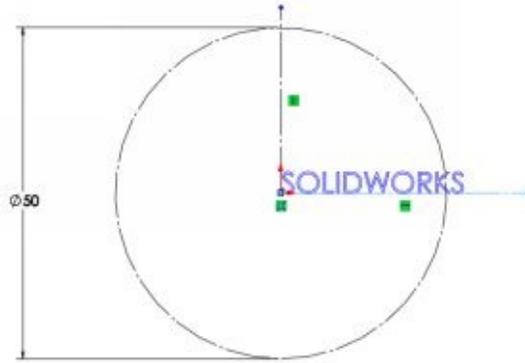
Text

Texts are used to provide information on the models such as company name, model specifications, and so on.

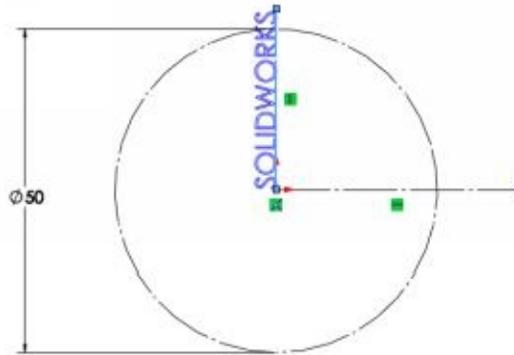
1. Start a new sketch.
2. Create the circle and two lines perpendicular to each other, as shown.



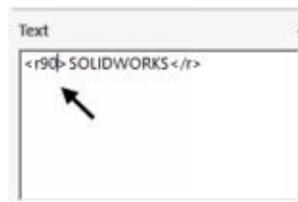
3. On the **Sketch** CommandManager, click the **Text**  tool.
4. Select the horizontal line as the supporting curve.
5. Click in the Text box in the PropertyManager and type SOLIDWORKS. On the PropertyManager, use the **Alignment** options (**Left Align** , **Center Align** , **Right Align** , and **Full Justify** ) to change the alignment of the text. Use the Flip options (**Flip Vertical** and **Flip Horizontal**) to flip the text.



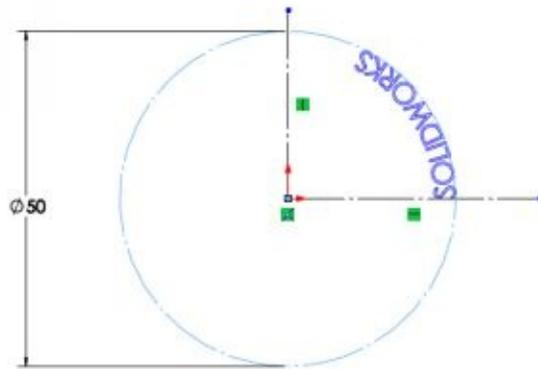
6. Click right mouse button in the **Curve** box and select the vertical construction line.



7. Select the text from **Text** box and click **Rotate** .
8. Change the rotation angle value in the **Text** box, as shown.



9. Right click in the **Curve** box and select Delete.
10. Select the circle as the alignment curve.
11. Select the rotation angle value text and press.



12. Click the **OK** on the PropertyManager.

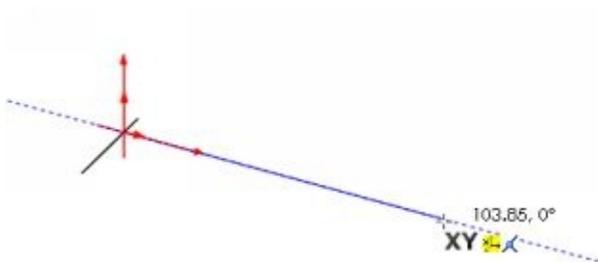
3D Sketch

3D sketches are created without selecting any plane and sketch elements spread over the 3D space instead of a single plane. However, you can also use planes to create 3D sketch.

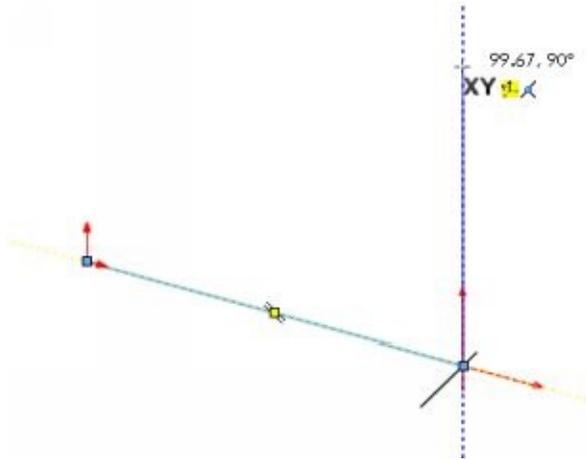
1. On the CommandManager, click **Sketch** > **Sketch drop-down** > **3D Sketch** .
2. On the CommandManager, click **Sketch** > **Line** .
3. Select the origin point to define the start point.



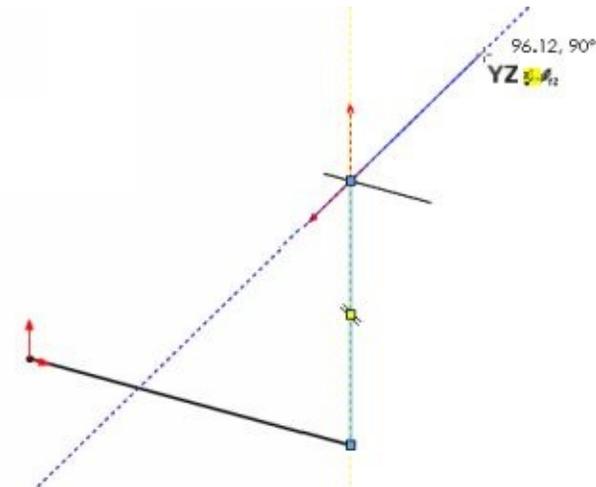
4. Move the pointer along the X-axis and click to create a line.



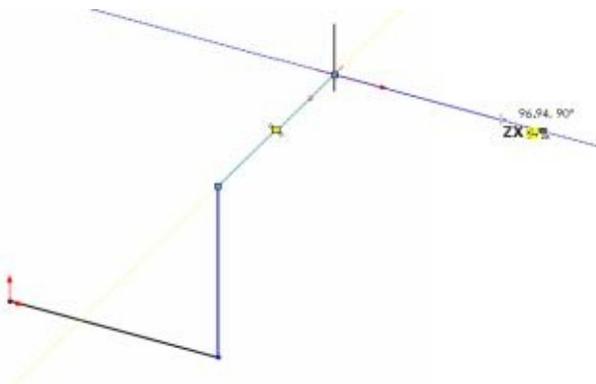
5. Move the pointer along the Y-axis and click.



6. Press the Tab key to change the plane.
7. Move the pointer in the Z-direction and click.

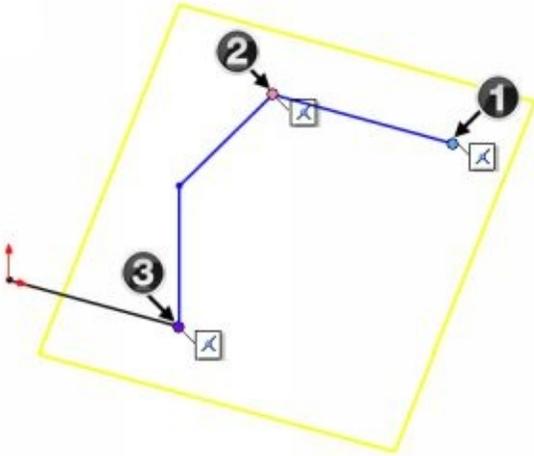


8. Press the Tab key to change the plane.
9. Move the pointer in the X-direction and click.



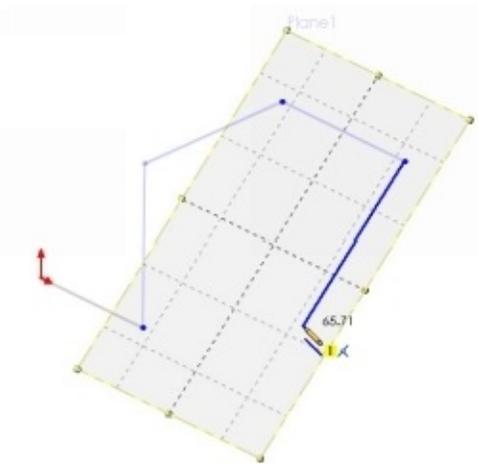
10. Click the right mouse button and select **Select**.
11. On the CommandManager, click **Sketch** > **Plane** .
12. Press the Ctrl key and select the end points as shown.

13. Click **OK** on the PropertyManager.

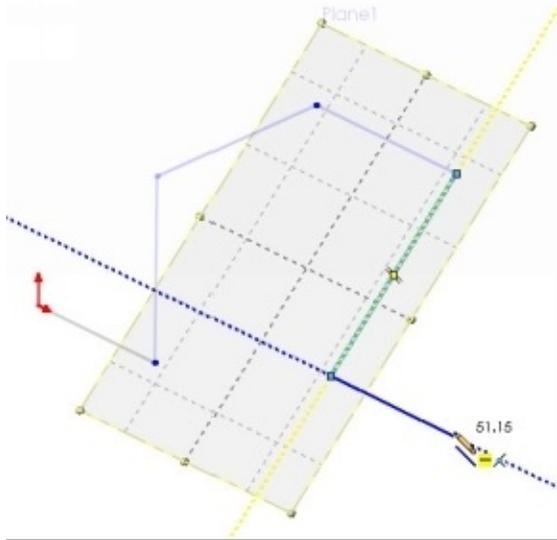


14. Activate the **Line** tool and click the end of the previous line.

15. Move the pointer and click when the **Vertical** symbol appears next to the pointer.



16. Move the pointer and click when the **Horizontal** symbol appears.



17. Click the right mouse button and select **Select**.
18. Add dimensions and relations, and then click **Exit Sketch**.



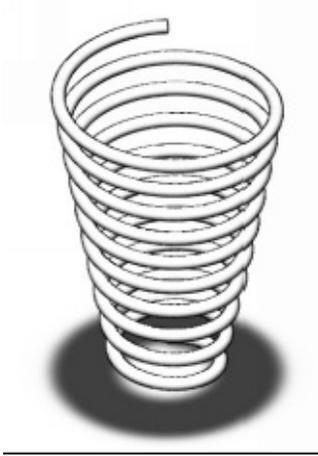
Chapter 6: Additional Modeling Tools

In this chapter, you construct models using additional modeling tools. You will learn to:

- Construct Helixes
- Add Reference Planes
- Construct Sweep features
- Construct Revolved Cuts
- Mirror Features
- Construct Loft features
- Create multibody parts
- Add Indent Features
- Shell Models
- Create Helical threads
- Construct rib features
- Insert Holes
- Create Patterns

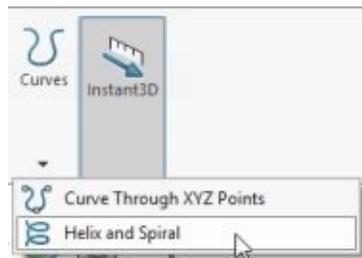
TUTORIAL 1

In this tutorial, you will construct a helical spring using the **Helix** and **Sweep Boss/Base** tools.

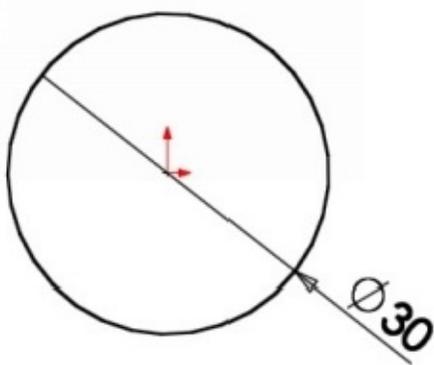


Constructing the Helix

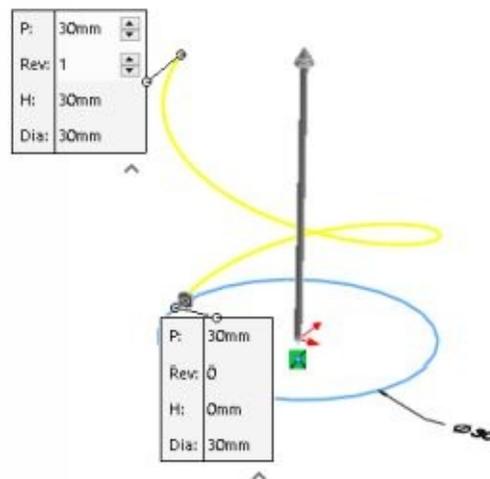
1. Open a new SOLIDWORKS file using the **Part** template.
2. To construct a helix, click **Features** > **Curves** > **Helix and Spiral** on the CommandManager.



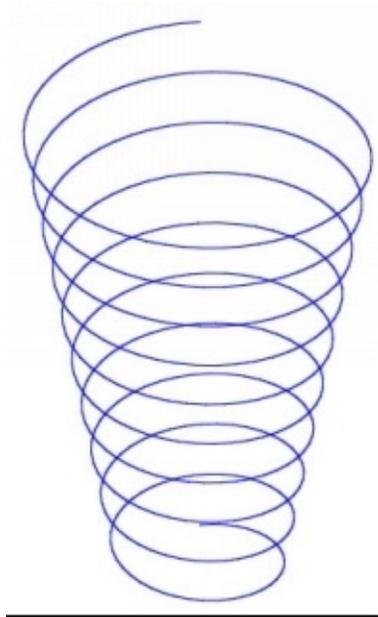
3. Click on the top plane.
4. Draw a circle of diameter 30.



5. Click **Exit Sketch** and change the view to Isometric.

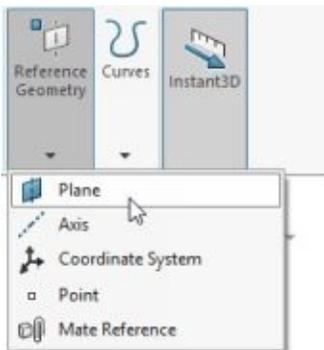


6. On the PropertyManager, set **Pitch** to 10 mm.
7. Set **Revolutions** to 10.
8. Select **Clockwise**.
9. Check the **Taper Helix** option and set **Taper Angle** to 10.
10. Click **OK**.

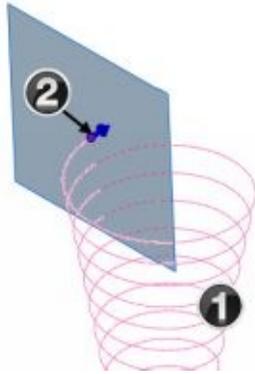


Adding the Reference Plane

1. To add a reference plane, click **Features** > **Reference Geometry** > **Plane** on the CommandManager.



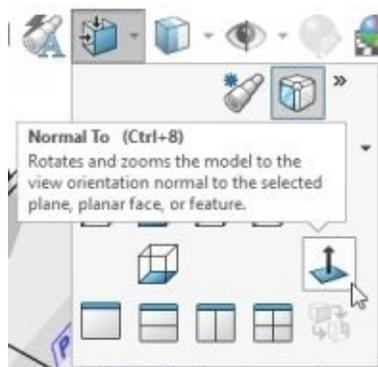
2. Select the helix from the graphics window.
3. Select the end of the helix.



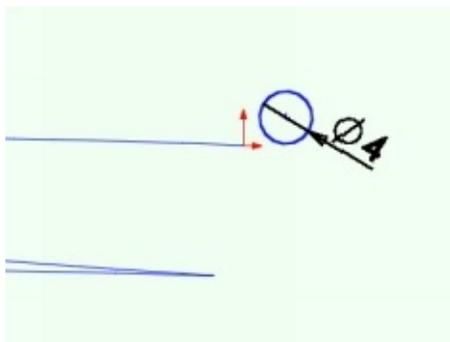
4. Leave the default values and click **OK** ✓.

Constructing the Sweep feature

1. On the CommandManager, click **Sketch** > **Sketch** .
2. Select the plane created normal to helix.
3. On the **View (Heads Up)** Toolbar, click **View Orientation** > **Normal To**.



4. Draw circle of 4 mm diameter.



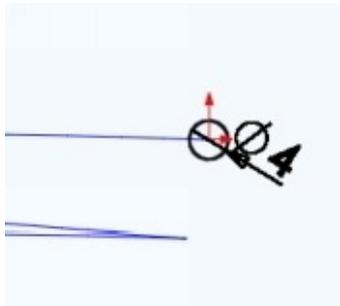
5. On the CommandManager, click **Sketch** > **Display/Delete Relations** >

Add Relations .

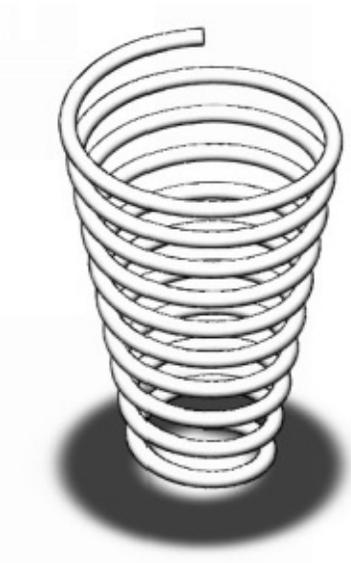
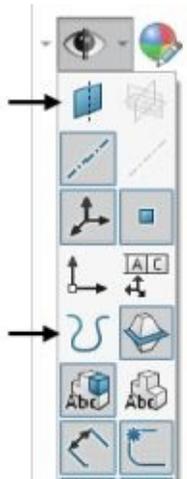
- Click on the center point of the circle and helix.
- On the PropertyManager, under the **Add Relations** section, select **Pierce**.



- Click **OK** .

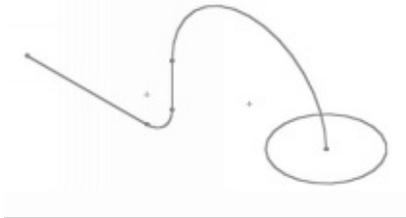


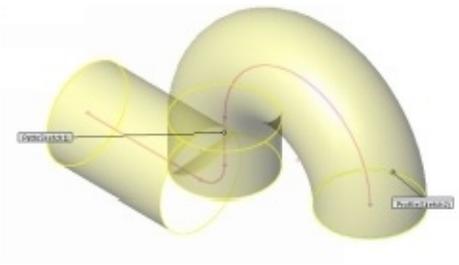
- Click **Exit Sketch** and change the view orientation to Isometric.
- To construct a sweep feature, click **Features > Sweep Boss/Base**  on the CommandManager.
- Select the circle to define the profile.
- On the PropertyManager, click in the **Path**  selection box and select the helix to define the path.
- Leave the default settings and click **OK** to construct the sweep feature.
- On the **View Heads-Up** toolbar, click the **Visibility** drop-down and turn off the **Plane** and **Curves** icons.



15. Save and close the file.

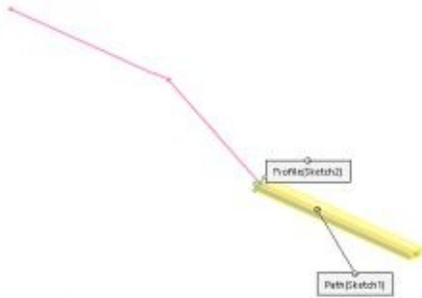
You need to make sure that the size of the cross section is not larger than the curves on the path, the resulting geometry will intersect and the sweep will fail.



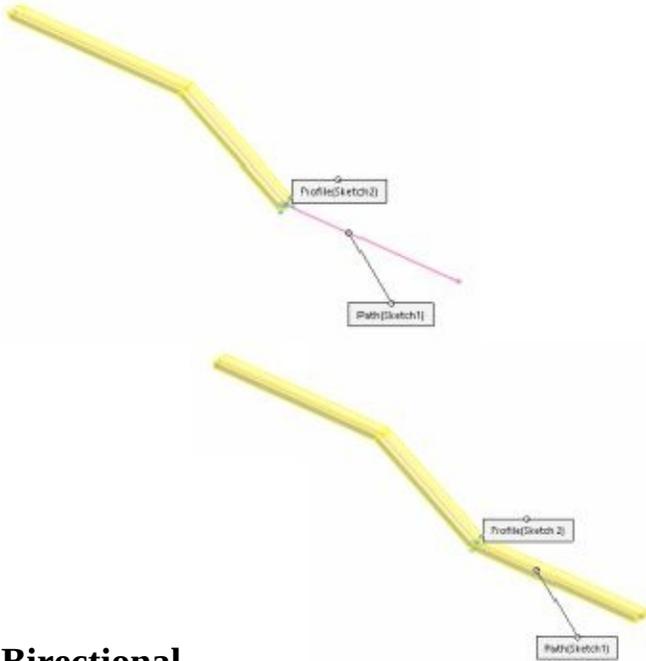


In SOLIDWORKS 2016, you can create a profile anywhere along the path. For example, create a profile at the middle of the path, as shown. Activate the Sweep tool and select the profile and path. On the PropertyManager, use the **Direction 1** , **Birectional** , and **Direction 2**  icons to sweep the profile.

Direction 1



Direction 2



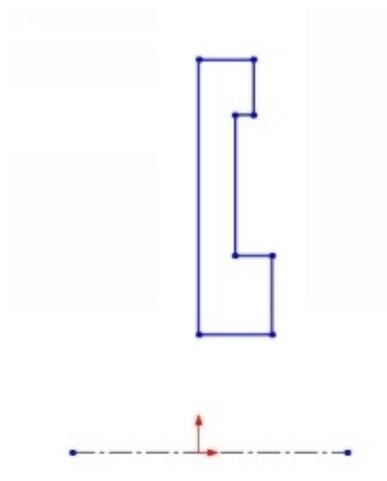
Birectional

TUTORIAL 2

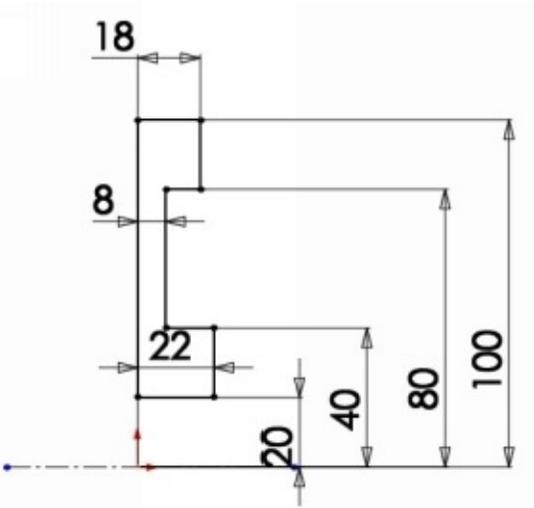
In this tutorial, you construct a pulley wheel using the **Revolve Boss/Base** and **Revolved Cut** tools.



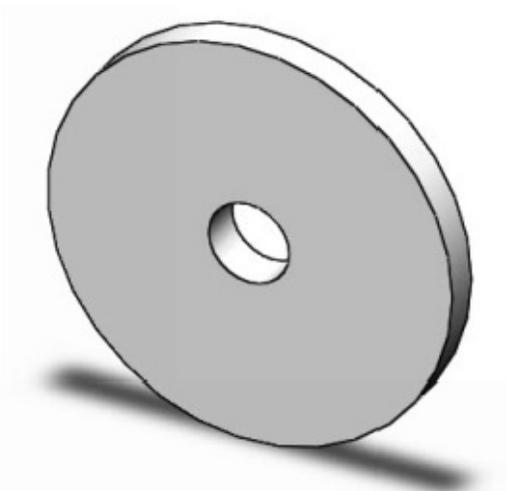
1. Open a file in the **Part** Environment.
2. On the CommandManager, click **Sketch** > **Sketch**.
3. Click on the right plane.
4. On the CommandManager, click **Sketch** > **Line** > **Centerline**.
5. Construct a horizontal centerline passing through origin.
6. Construct the sketch, as shown in figure.



7. Add dimensions to the sketch.
8. Add the **Coincident** relation between the origin and the vertical line at the center.

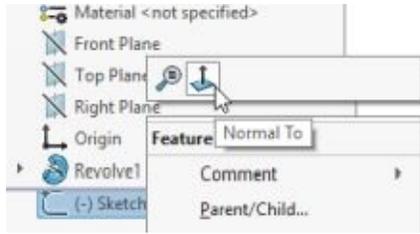


9. Construct the revolved boss.

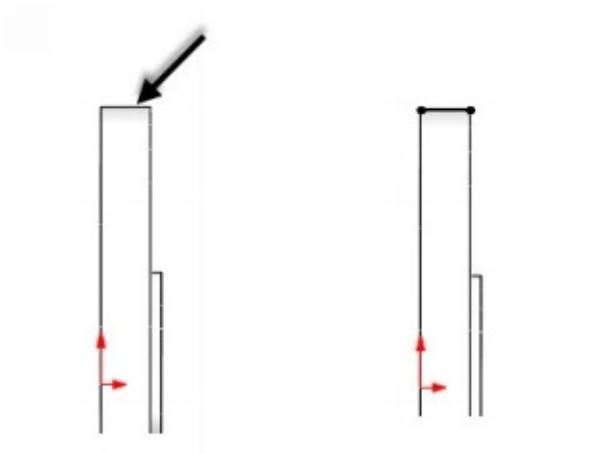


Constructing a Revolved Cut

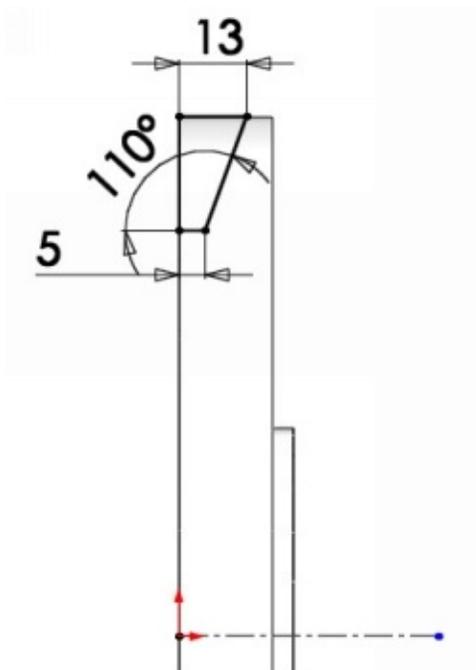
1. To construct a revolved cut, click **Features** > **Revolved cut**  on the CommandManager.
2. Expand the **FeatureManager Design Tree** and select the right plane.
3. Under the **FeatureManager Design Tree**, click the right mouse button on the sketch and select **Normal To**.



4. On the CommandManager, click **Sketch** > **Convert Entities** .
5. Click on the horizontal edge of the model. Click **OK**.



6. Draw the sketch entities and add dimensions. Trim the unwanted entities.

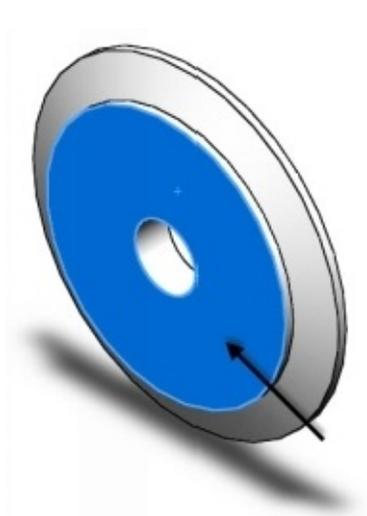


7. Click **Exit Sketch** and click **OK**.



Constructing a Mirror Feature

1. To construct the mirror feature, click **Features** > **Mirror**  on the CommandManager.
2. Click on the front flat face to define the mirror face.



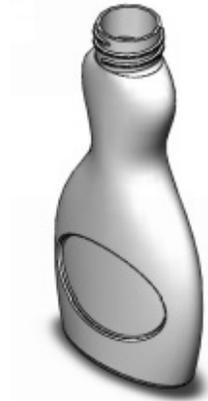
3. On the PropertyManager, expand the **Bodies to Mirror** section and click on the model geometry.
4. Click **OK**.



5. Save and close the model.

TUTORIAL 3

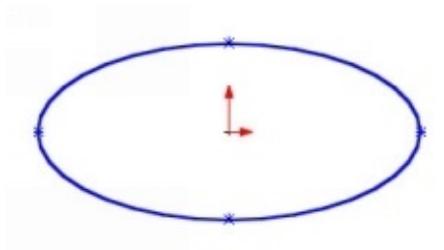
In this tutorial, you construct a shampoo bottle using the **Loft**, **Extrude**, **Shell**, **Indent**, and **Sweep** tools.



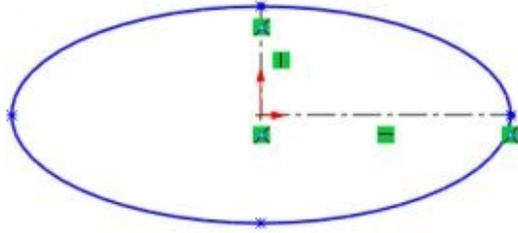
Creating Sections and Guide curves

To construct a lofted boss, you need to create sections and guide curves.

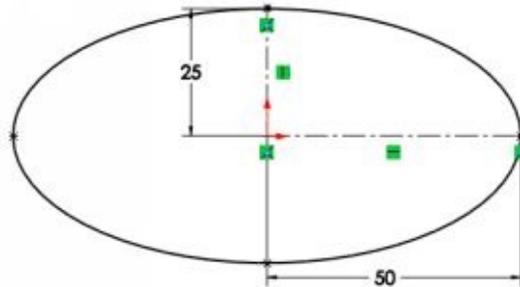
1. Open a new file in the **Part** Environment.
2. On the CommandManager, click **Sketch** > **Sketch**.
3. Select the top plane.
4. On the CommandManager, click **Sketch** > **Ellipse** .
5. Select the origin point of the sketch.
6. Move the pointer horizontally and left click.
7. Move the pointer vertically and left click.



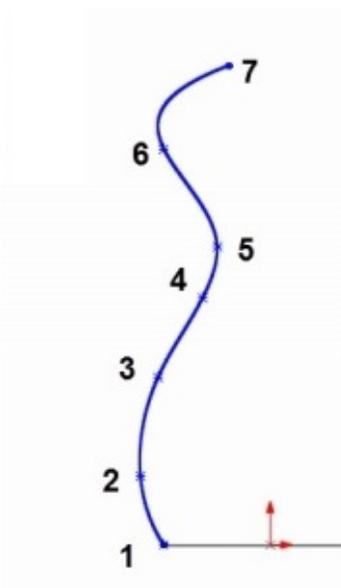
8. Create two centerlines passing to the origin and perpendicular to each other, as shown.



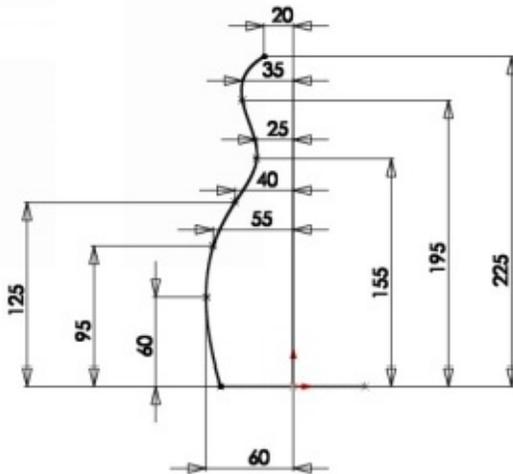
9. Activate **Smart Dimension** tool and add dimensions to the centerlines, as shown.



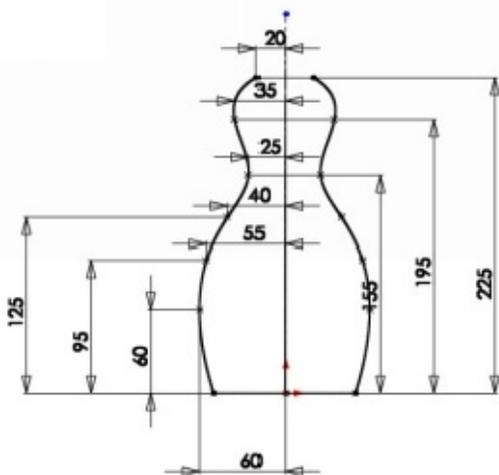
10. Click **Exit Sketch**.
11. Change the orientation to Isometric.
12. On the CommandManager, click **Sketch > Sketch**.
13. Select the front plane.
14. On the CommandManager, click **Sketch > Spline** [~].
15. Draw a spline similar to the one shown in figure.



16. Press the Ctrl key and click on the first point of the spline and the previous sketch.
17. On the PropertyManager, under the **Add Relations** section, click **Pierce**.
18. Apply dimensions to the spline, as shown in figure. First, create the horizontal dimensions, and then create the vertical dimensions in the descending order.



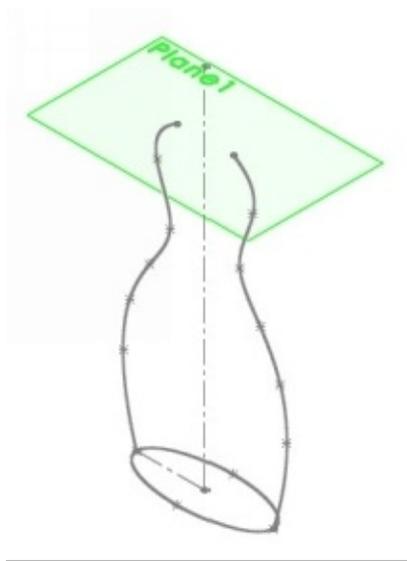
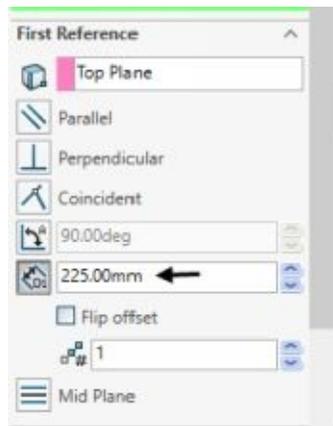
19. Draw a vertical centerline passing through the sketch origin.
20. On the CommandManager, click **Sketch** > **Mirror Entities** .
21. Select the spline.
22. On the PropertyManager, click **Mirror about**, and then select the vertical centerline.
23. Click **OK**.



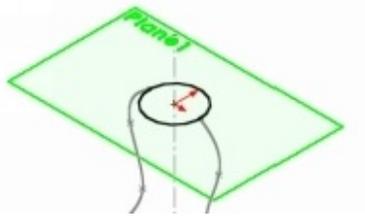
24. Click **Exit Sketch**.
25. Change the view orientation to Isometric.

Creating another section

1. On the CommandManager, click **Features** > **Reference Geometry** > **Plane** .
2. Select the top plane from FeatureManager Design Tree.
3. On the PropertyManager, type-in **225** in the **Offset Distance** box.



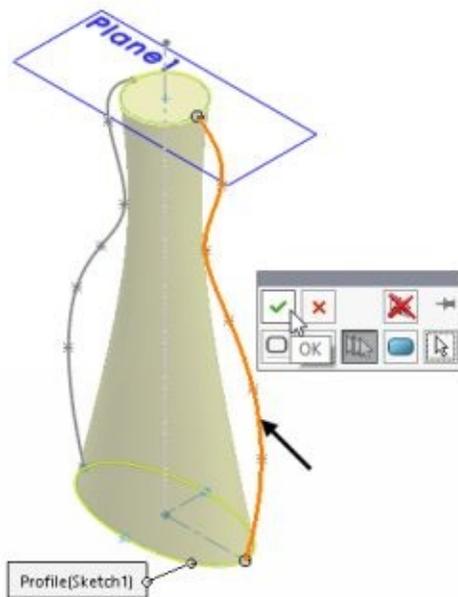
4. Click **OK**.
5. Start a sketch on the new reference plane.
6. Draw a circle of 40 mm diameter.



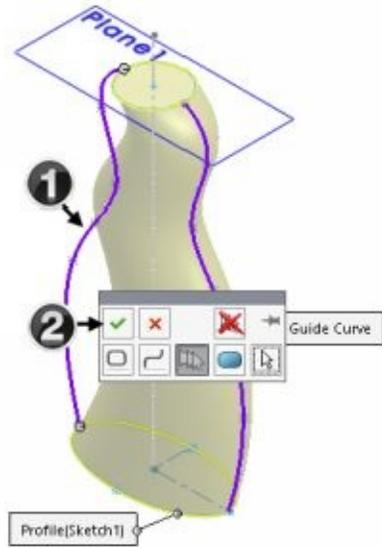
7. Click **Exit Sketch**.
8. Change the view to Isometric.

Constructing the swept feature

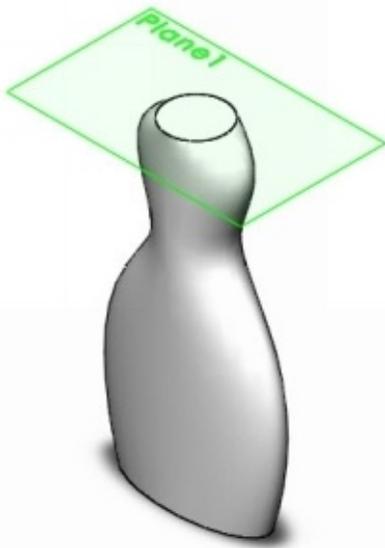
1. On the CommandManager, click **Features** > **Lofted Boss/Base** .
2. Select the circle and ellipse.
3. On the PropertyManager, click in the **Guide Curves** selection box and click **OK** on the box.



4. Select the second guide curve and click **OK** on the box.

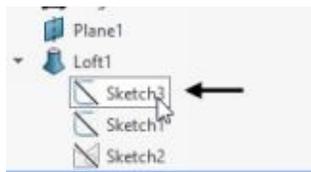


5. Click **OK**.

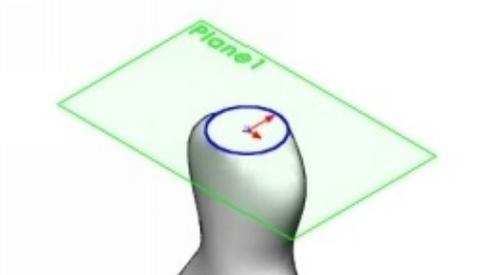
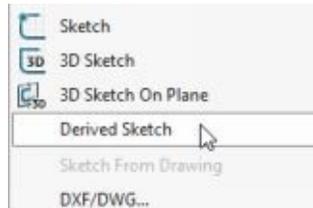


Constructing the Extruded feature

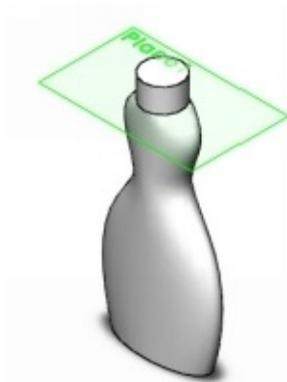
1. In the **FeatureManager Design Tree**, expand the Loft feature and select the Sketch 3.



2. Press the Ctrl key and select the reference plane located at the top of the lofted feature.
3. On the Menu bar, click **Insert > Derived Sketch**.

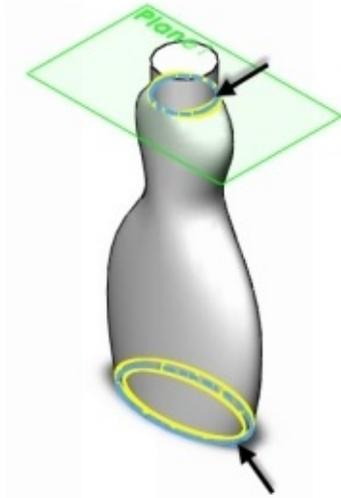


4. Click **Exit Sketch**.
5. Activate the **Extrude Boss/Base**  tool and extrude the sketch up to 25 mm depth.



Adding Fillets

1. On the CommandManager, click **Features > Fillet** .
2. On the PropertyManager, click the **FilletXpert** tab.
3. Click on the bottom and top edges of the loft feature.



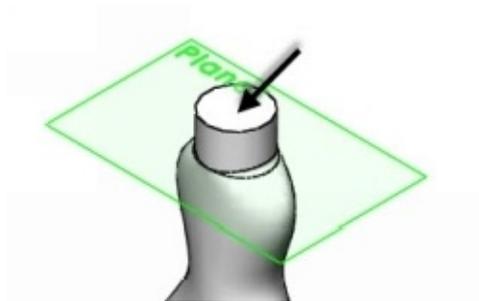
4. Set **Radius** to 5 mm.



5. Click **OK** to add the fillet.

Shelling the Model

1. On the CommandManager, click **Features** > **Shell** .
2. On the PropertyManager, set **Thickness** to 2 mm.
3. Select the top face of the cylindrical feature.

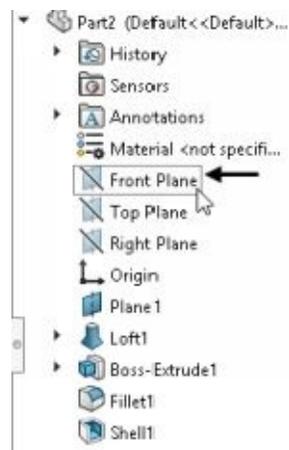


4. Click **OK** to shell the geometry.

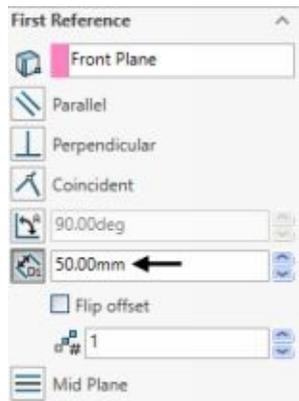


Adding the Indent feature

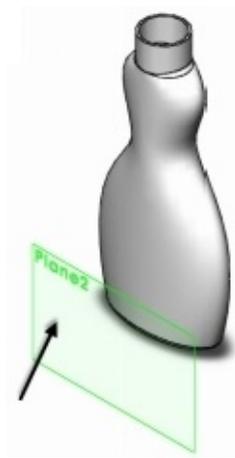
1. On the CommandManager, click **Features > Reference Geometry > Plane**.
2. In the **FeatureManager Design Tree**, select the Front plane.



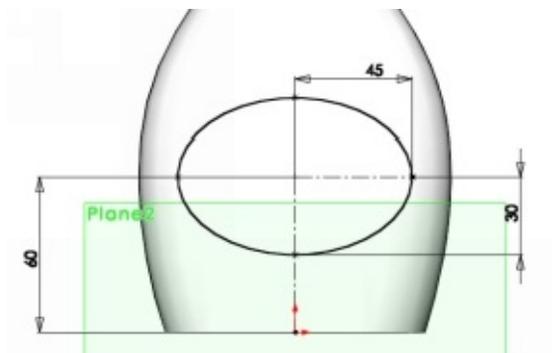
3. On the PropertyManager, type-in **50** in the **Distance** box.



4. Click **OK**.

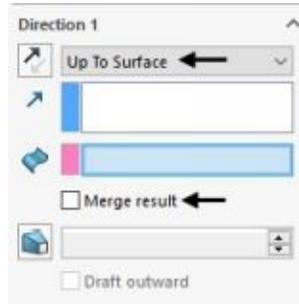


5. Create a sketch on the plane, as shown in figure.

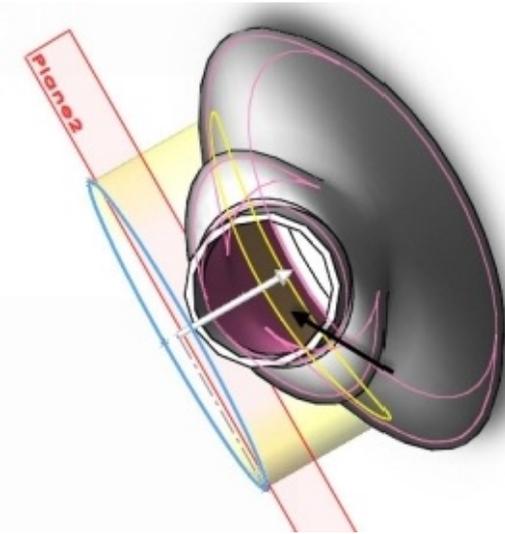


6. Click **Exit Sketch**.

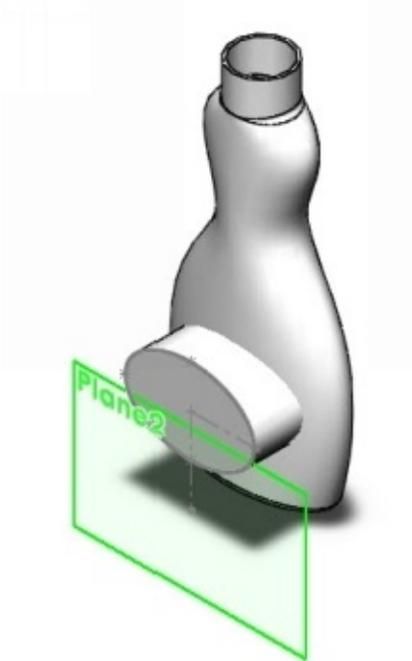
7. On the CommandManager, click **Features** > **Extruded Boss/Base** .
8. Select the sketch.
9. On the PropertyManager, under the **Direction1** section, select **End Condition** > **Up To Surface**.
10. On the **PropertyManager**, under **Direction 1**, uncheck **Merge result**.



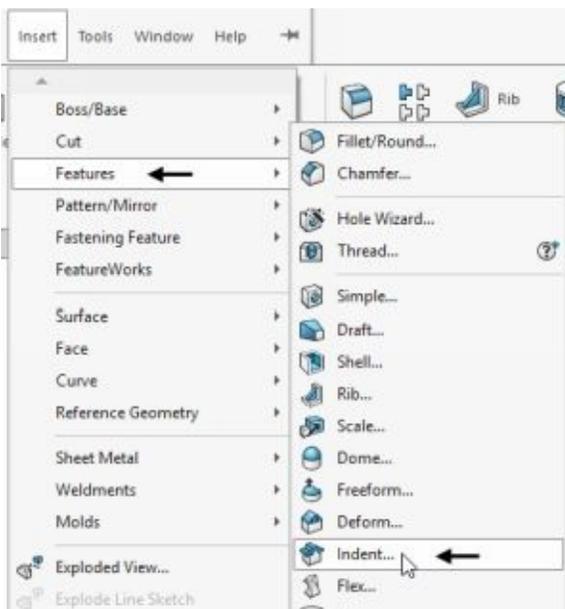
11. Rotate the model and select the inner face of the model geometry.



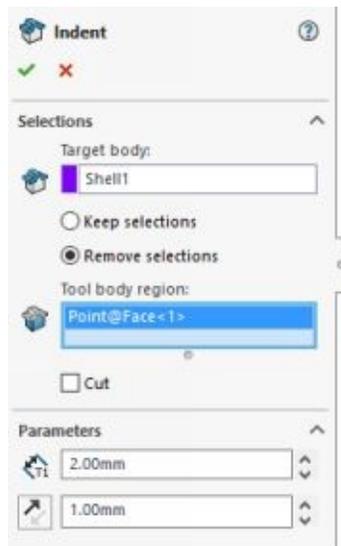
12. Click **OK** to create separate extruded body.



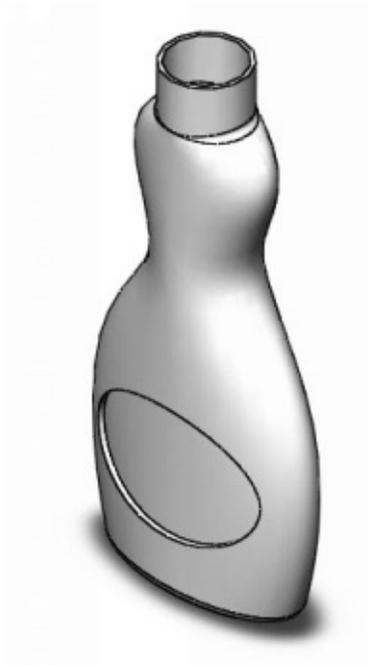
13. On the Menu bar, click **Insert** > **Features** > **Indent**.



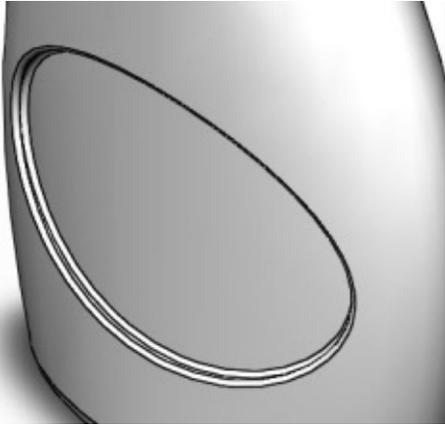
14. Click on the loft feature to define the target body.
15. Click on the extruded body to define the tool body.
16. On the PropertyManager, select **Remove Selections**.
17. Under the **Parameters** section, set the **Thickness** value to 2 and **Clearance** value to 1.



18. Click **OK** to indent the extruded body onto the lofted body.
19. Hide the extruded body, sketch, and plane.

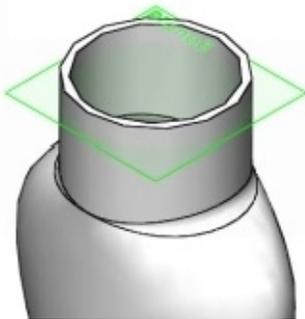


20. Add 1 mm fillet to the inner and outer edges of the indent.

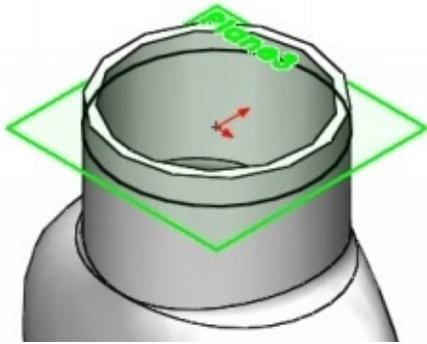


Adding Threads

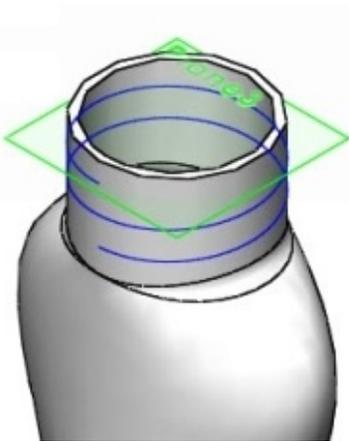
1. On the CommandManager, click **Features > Reference Geometry > Plane** .
2. Click on the top face of the model geometry.
3. Set the **Distance** value to 5 and check the **Flip offset** option.
4. Click **OK**.



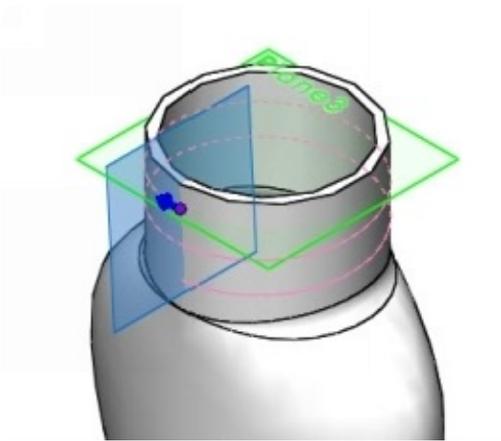
5. Start sketch on the new reference plane.
6. On the CommandManager, click **Sketch > Convert Entities** .
7. Click on the outer edge of the cylindrical feature and click **OK**.



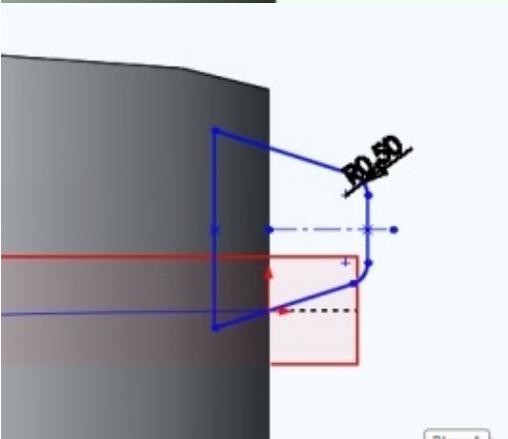
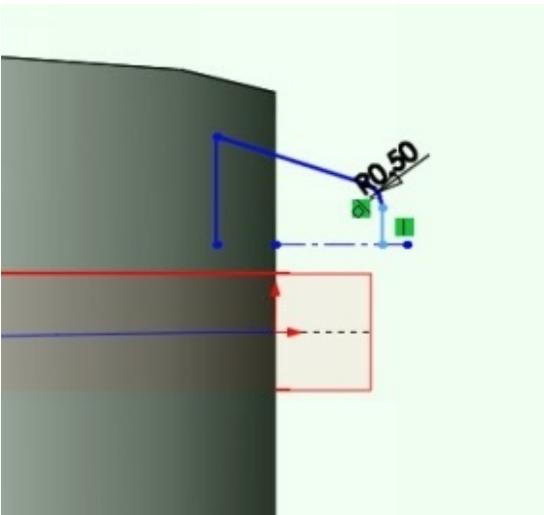
8. Click **Exit Sketch**.
9. On the CommandManager, click **Features > Curves > Helix and Spiral** 
10. Select the circular sketch.
11. On the PropertyManager, select **Constant pitch**.
12. Type-in 7 in the **Pitch** box and check the **Reverse Direction** option.
13. Type-in 2 in the **Revolutions** box.
14. Set the **Start Angle** value to **0**.
15. Uncheck the **Taper Helix** option.
16. Click **OK**.

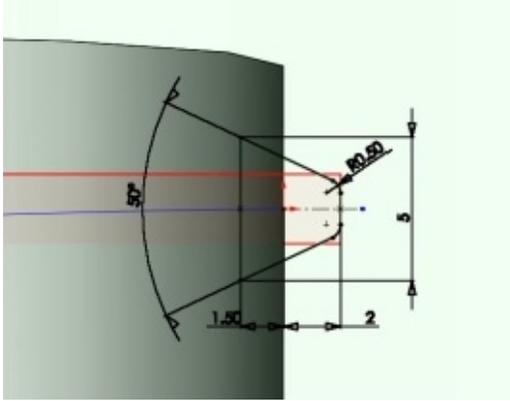


17. On the CommandManager, click **Features > Reference Geometry > Plane**.
18. Click on the helix and its start point.



19. Click **OK**.
20. Start a sketch on the new reference plane.
21. Draw the thread profile.

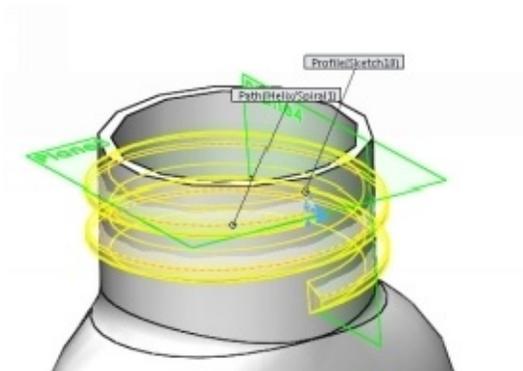




22. Click **Exit Sketch**.

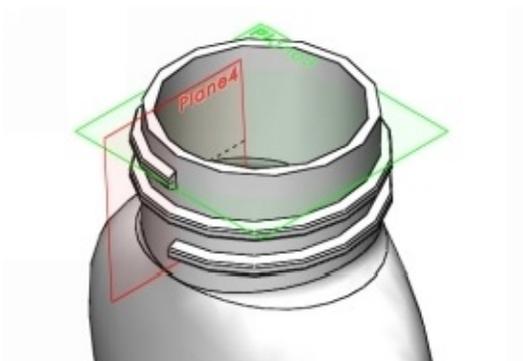
23. On the CommandManager, click **Features** > **Swept Boss/Base** .

24. Click on the thread profile and the helix.

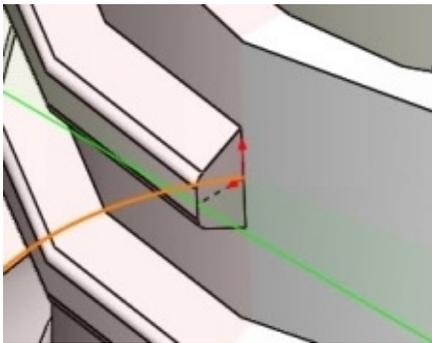


25. On the PropertyManager, expand the **Options** section and check **Merge result**.

26. Click **OK**.



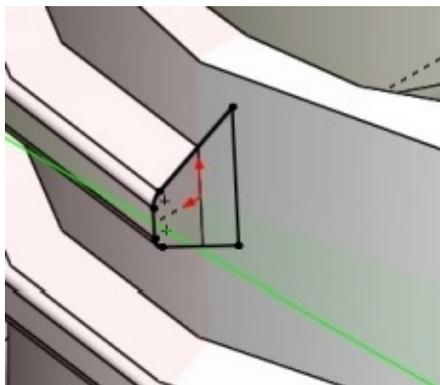
27. Start a sketch on the end face of the thread.



28. On the CommandManager, click **Sketch** > **Covert Entities**  and select the thread profile.



29. Click **OK**.

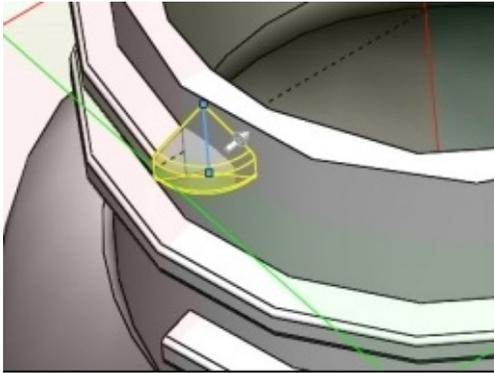


30. Click **Exit Sketch**.

31. Activate the **Revolved Boss/Base**  tool and click on the vertical line of the sketch.

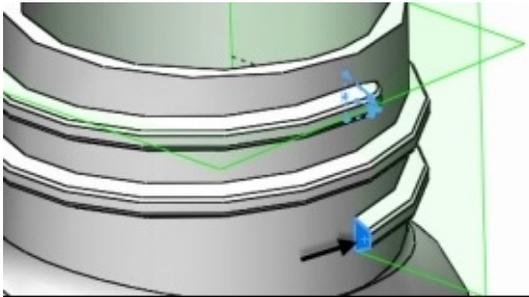
32. On the PropertyManager, set **Direction 1 Angle** to **100**.

33. Check the **Merge result** option.



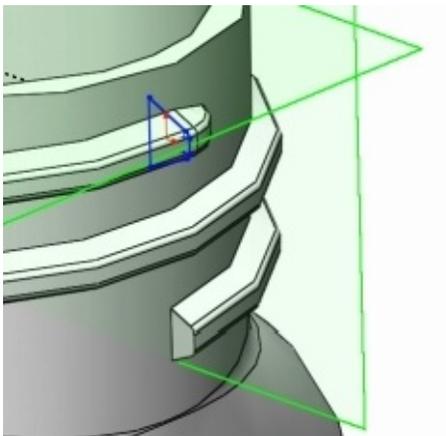
34. Click **OK**.

35. Rotate the model and click on the other end of the thread.



36. Press the Ctrl key and select the sketch used for the previous revolved feature.

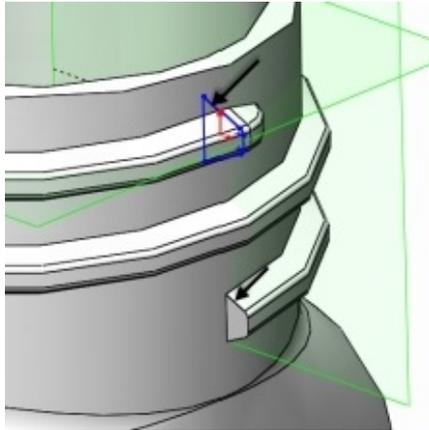
37. On the Menu bar, click **Insert > Derived Sketch**.



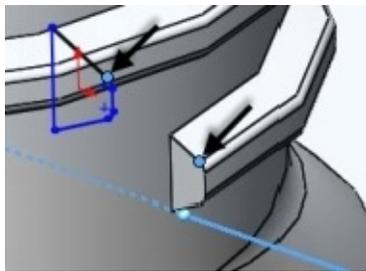
38. On the CommandManager, click **Display/Delete Relations > Add**

Relations.

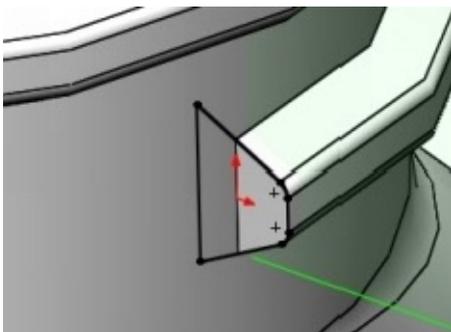
39. Click on the inclined line of the sketch and the inclined edge of the end face.



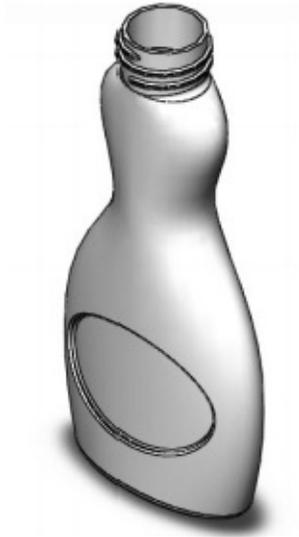
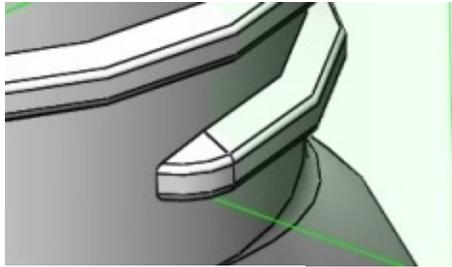
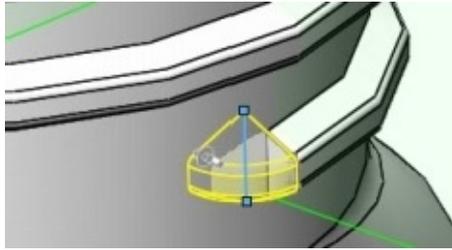
40. On the PropertyManager, click **Collinear**.
41. Click on the endpoints, as shown in figure.



42. On the PropertyManager, click **Coincident**.



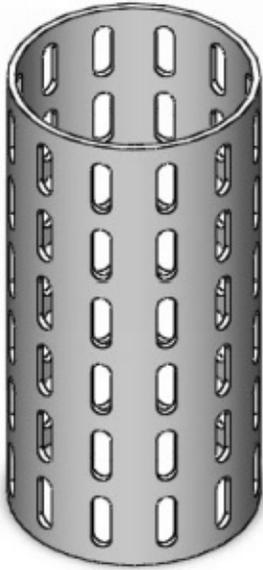
43. Create a revolved boss using the sketch.



44. Save the model and close it.

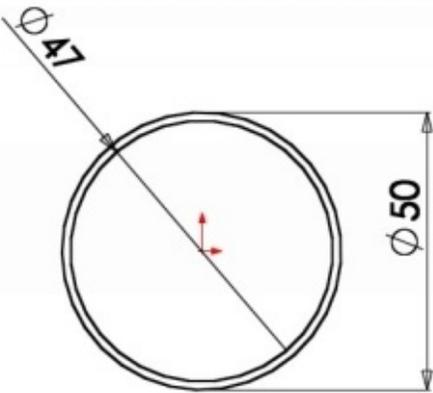
TUTORIAL 4

In this tutorial, you construct a patterned cylindrical shell.

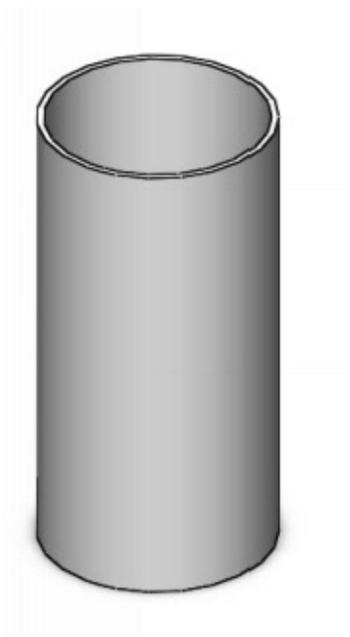


Constructing a cylindrical shell

1. Start a new file using the **Part** template.
2. Create a sketch on the top plane.

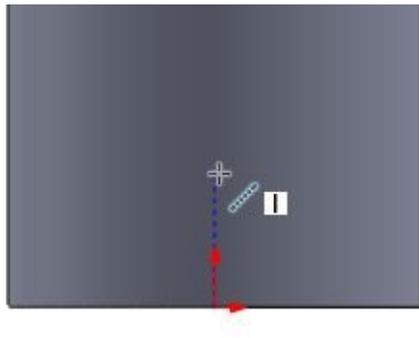


3. Extrude the sketch up to 100 mm depth.

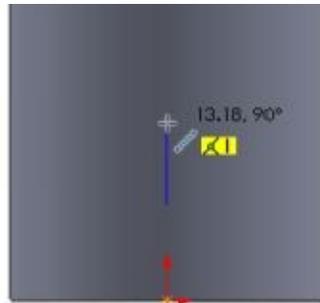


Adding a Slot

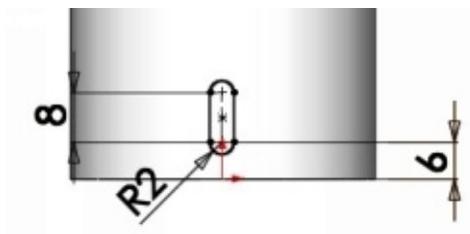
1. Activate the **Extruded Cut**  tool.
2. Click on the front plane.
3. On the CommandManager, click **Sketch** > **Straight Slot** .
4. Click to define the first point of the slot.



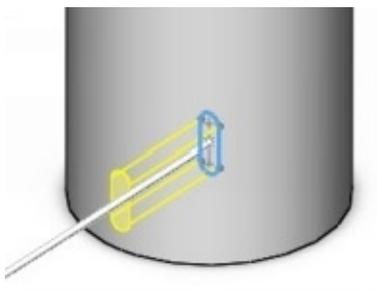
5. Move the pointer up and click to define the second point.



6. Move the pointer and click to create a slot.
7. Add dimensions to the slot.



8. Click **Exit Sketch**.
9. On the PropertyManager, select **End Condition** > **Through All**.
10. Click the **Reverse Direction** button next to the **End Condition** dropdown.

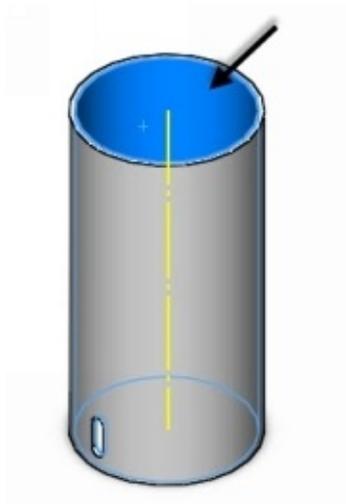


11. Click **OK**.



Constructing the Axis

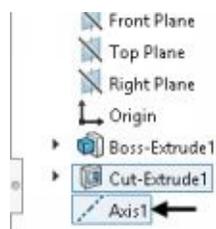
1. On the CommandManager, click **Features** > **Reference Geometry** > **Axis** .
2. Click on the inner cylindrical face of the model geometry.



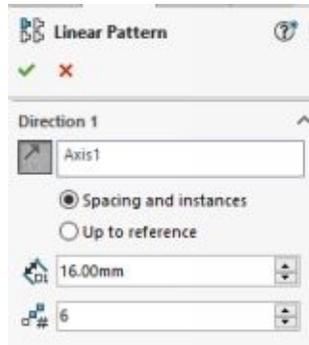
3. Click **OK** to construct the axis.

Constructing the Linear pattern

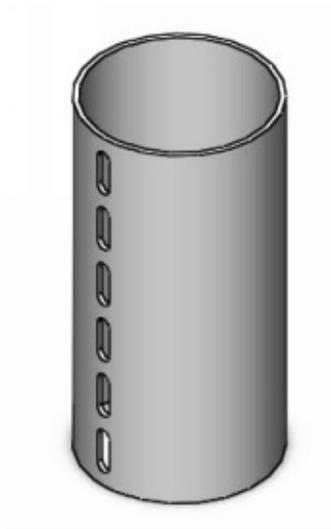
1. On the CommandManager, click **Features** > **Linear Pattern** .
2. On the PropertyManager, under the **Features and Faces** section, click in the **Features to Pattern** box, and then select the slot.
3. In the FeatureManager Design Tree, select the newly created axis. You can also select the Top Plane from the FeatureManager Design Tree.



4. On the PropertyManager, under the **Direction1** section, set the **Spacing** value to 16.
5. Set **Number of Instances** to 6.

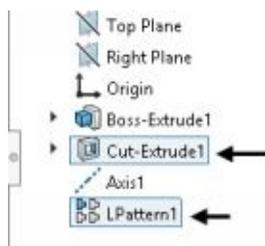


6. Click **OK**.

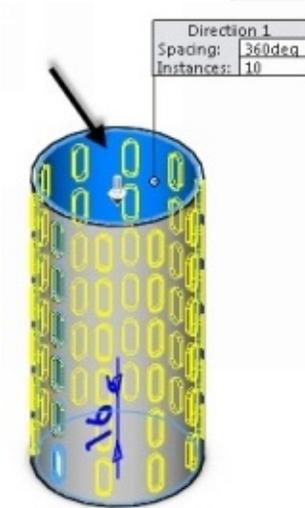
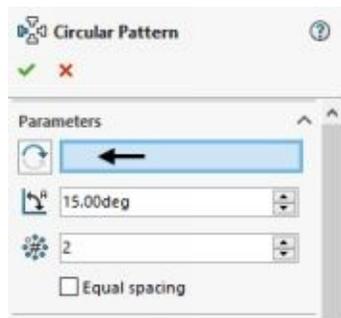


Constructing the Circular pattern

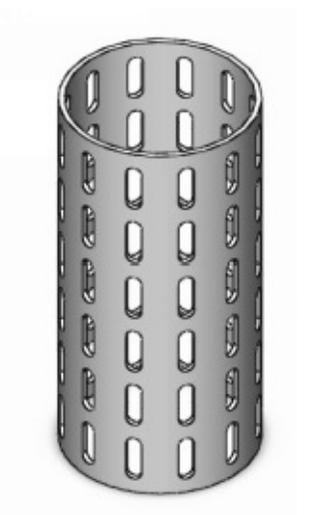
1. On the CommandManager, click **Features** > **Linear Pattern** > **Circular Pattern**.
2. In the FeatureManager Design Tree, select the linear pattern and the cut-extrude feature.



3. On the PropertyManager, click in the **Pattern Axis** selection box and select the cylindrical face of the model.



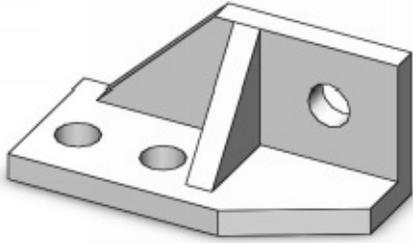
4. Type-in **12** in the **Number of Instances** box.
5. Check the **Equal Spacing** option.
6. Click **OK** to make the circular pattern.



11. Save and close the model.

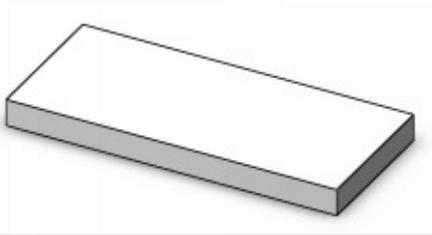
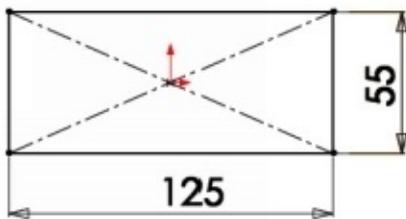
TUTORIAL 5

In this tutorial, you construct the model shown in figure.



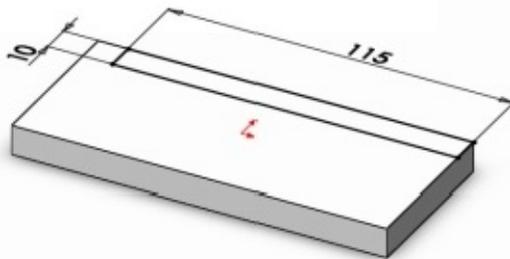
Constructing the first feature

1. Open a new part file.
2. Construct the first feature on the top plane (extrude the sketch up to a distance of 10 mm).

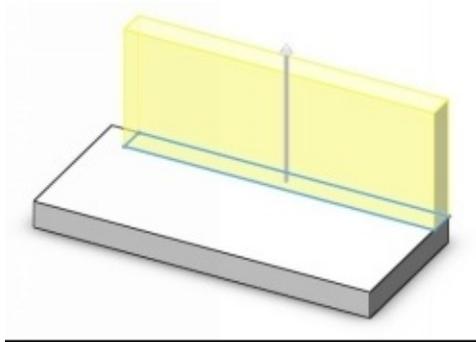


Constructing the Second Feature

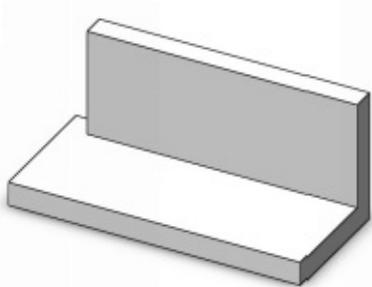
1. Draw the sketch on the top face of the first feature.



2. Click **Exit Sketch**.
3. On the CommandManager, click **Features** > **Extruded Boss/Base** .
4. Type-in **45** in the **Depth** box.

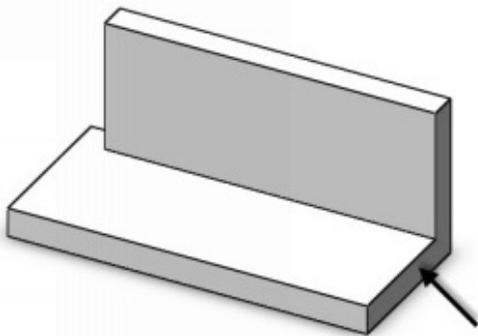


5. Click **OK**.

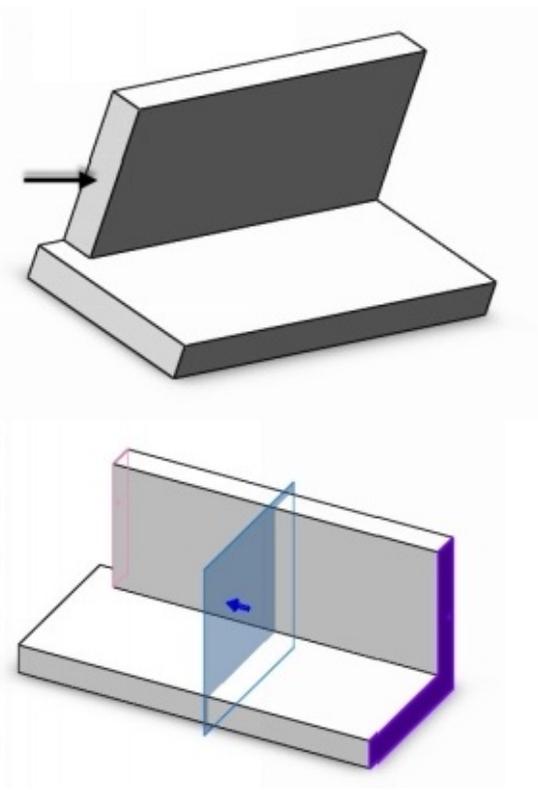


Constructing the third feature

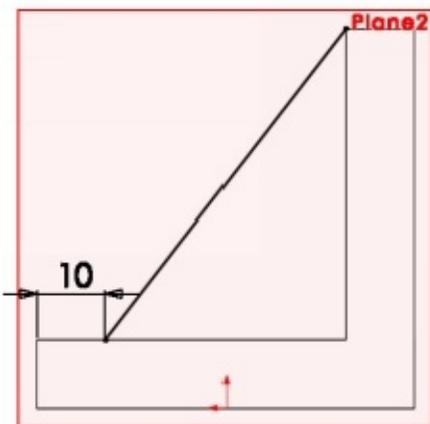
1. On the CommandManager, click **Features** > **Reference Geometry** > **Plane**.
2. Click on the right-side face of the model geometry.



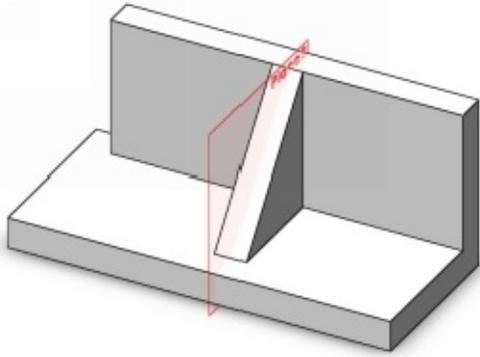
3. Click on the left-side face of the second feature.



4. Click **OK**.
5. Draw the sketch on the new reference plane.

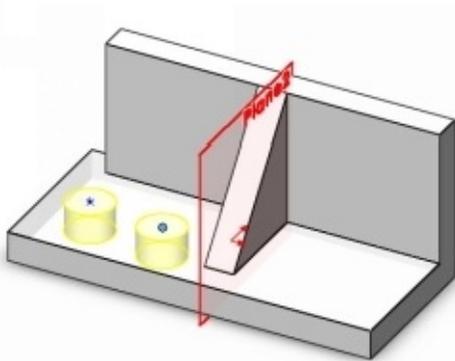


6. Click **Exit Sketch**.
7. On the CommandManager, click **Features** > **Rib** .
8. On the PropertyManager, click **Thickness** > **Both Sides** .
9. Set **Rib Thickness** to 10.
10. Select **Extrusion direction** > **Parallel to Section** .
11. Check the **Flip material side** option.
12. Click **OK**.

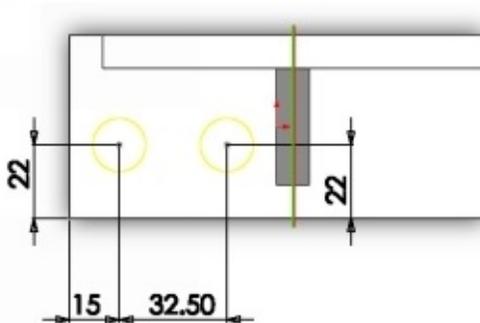


Inserting Holes

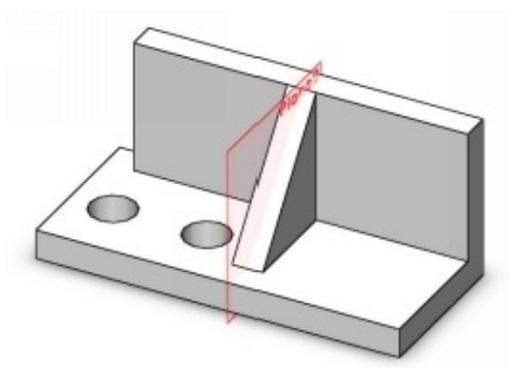
1. To insert holes, click **Features** > **Hole Wizard**  on the CommandManager.
2. On the **Hole Specification** PropertyManager, select the **Hole**  icon.
3. Select **Standard** > **ANSI Metric**.
4. Select **Type** > **Drill Sizes**.
5. Under the **Hole Specifications** section, select **Size** > **16**.
6. Select **End Condition** > **Through All**.
7. Click the **Positions** tab and click on the top face of the base feature.
8. Place two holes on the top face.



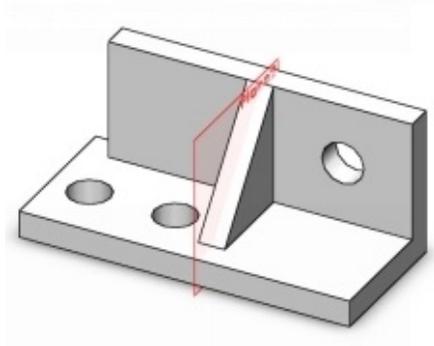
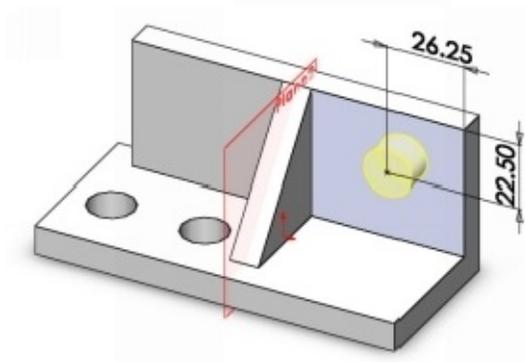
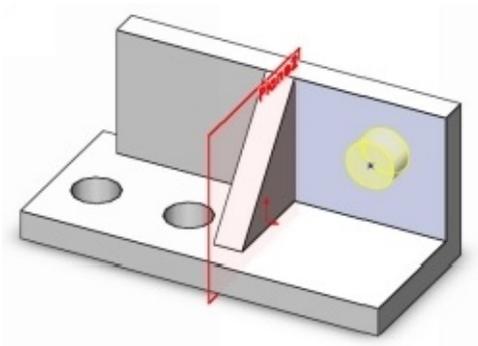
9. Add dimensions to define the hole location.



10. Click **OK**.

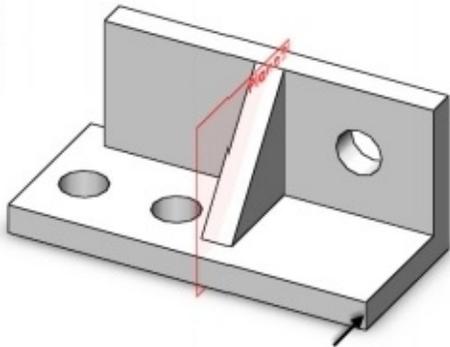


11. Insert another hole on the front face of the second feature.

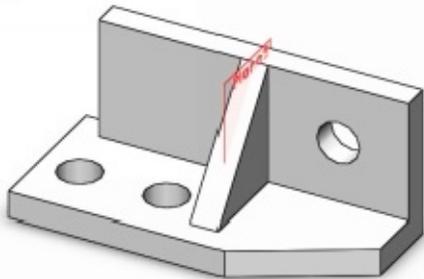


Adding Chamfers

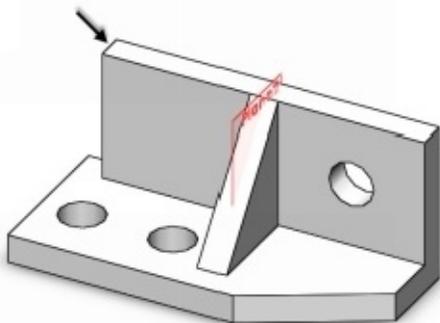
1. To add a chamfer, click **Features > Fillet Chamfer**  on the CommandManager.
2. On the PropertyManager, select **Distance distance**.
3. Type-in **45** and **25** in the **Distance 1** and **Distance 2** boxes.
4. Click on the corner edge of the first feature.



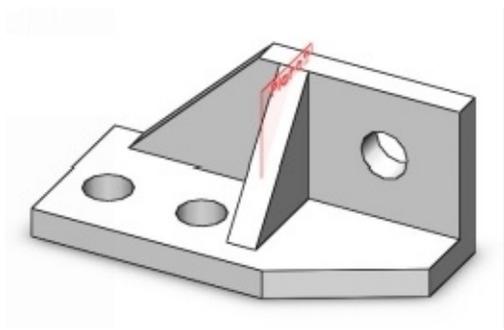
5. Click **OK** to add the chamfer.



6. Activate the **Chamfer** tool.
7. On the PropertyManager, select **Angle distance**.
8. Type-in **45** in the **Distance** and **Angle** boxes.
9. Click on the corner edge of the second feature.

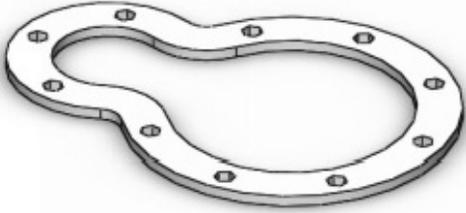


10. Click **OK**.



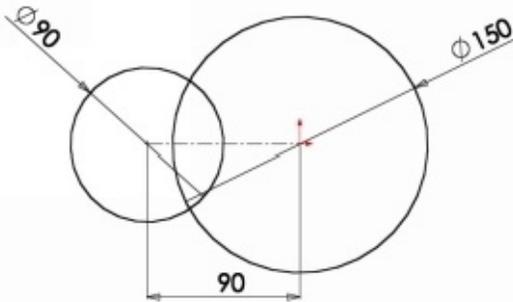
TUTORIAL 6

In this tutorial, you create the model shown in figure.

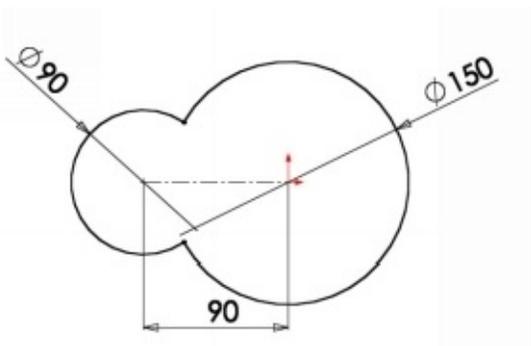


Constructing the first feature

1. Open a new part file.
2. On the CommandManager, click **Features** > **Extruded Boss/Base** .
3. Click on the top plane.
4. Construct two circles and add dimensions to them.

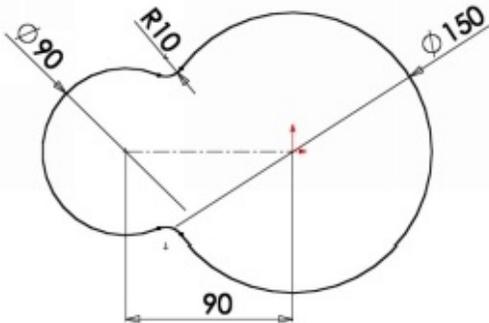


5. On the CommandManager, click **Sketch** > **Trim Entities**  and trim the intersecting entities.

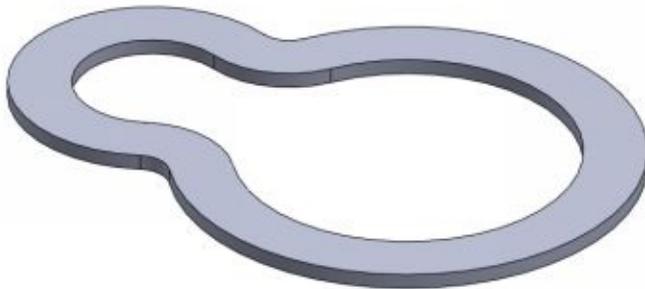


6. On the CommandManager, click **Sketch** > **Sketch Fillet**  and set the **Fillet Radius** value to 10.

7. Select the intersecting corners of the circles, and then click **OK**.

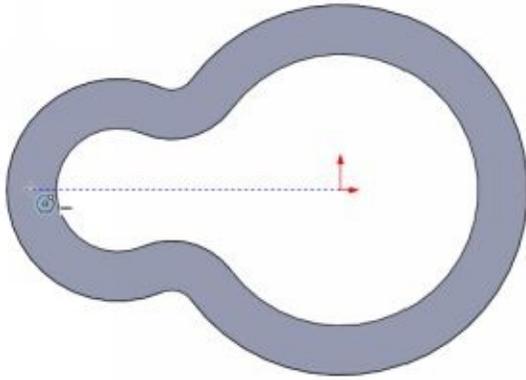


8. Click **Exit Sketch**.
9. On the PropertyManager, type 5 in the **Depth** box.
10. Check the Thin Feature option and select Type > One Direction.
11. Type 20 in the Thickness box.
12. Use the **Reverse Direction**  button next to the **Type** drop-down to make sure that the extruded feature is thickened from inside.
13. Click OK.

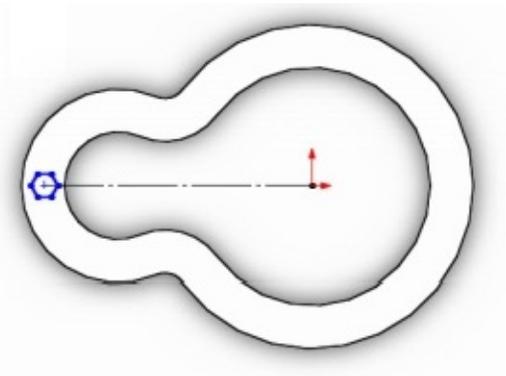


Constructing the Extruded cut

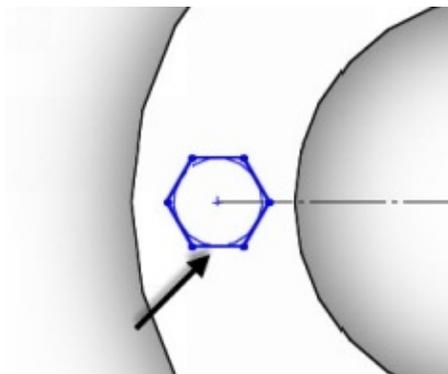
1. On the CommandManager, click **Features > Extruded Cut** .
2. Click on the top face of the model geometry.
3. On the CommandManager, click **Sketch > Polygon** .
4. On the PropertyManager, type-in 6 in the **Number of Sides** box.
5. Select the **Inscribed circle** option.
6. Click to define the center point of the polygon.



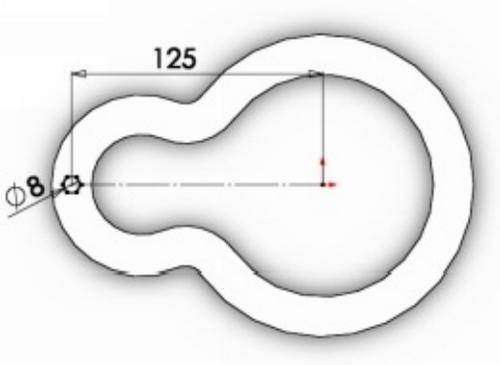
7. Move the pointer and click to construct the polygon.
8. Draw a horizontal centerline connecting the center point of the polygon and sketch origin.



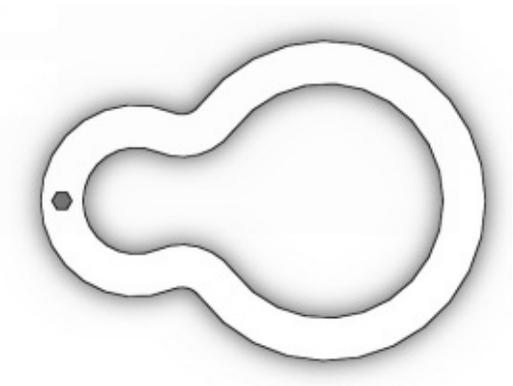
9. On the CommandManager, click **Sketch** > **Display/Delete Relations** > **Add Relation**.
10. Click on the horizontal line of the polygon.



11. On the PropertyManager, click **Horizontal**, and then click **OK**.
12. Add dimensions to the sketch.

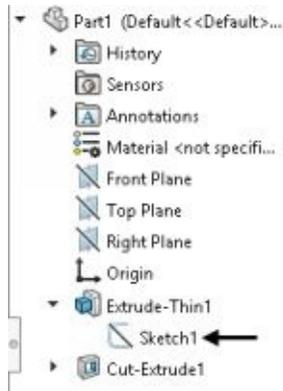


13. Click **Exit Sketch**.
14. Create the cut throughout the body.

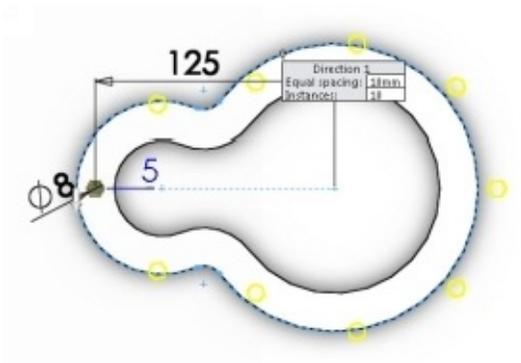


Making the Curve Driven Pattern

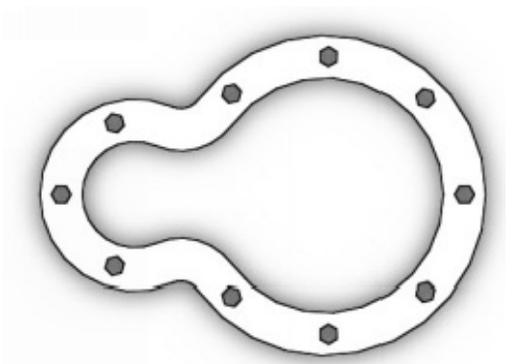
1. On the CommandManager, click **Features** > **Linear Pattern** > **Curve Driven Pattern**  .
2. In the FeatureManager Design Tree, select the sketch used to create Extrude-Thin1.



3. Select the polygonal cut to define the feature to pattern.
4. Type-in **10** in the **Number of Instances** box.
5. Check the **Equal Spacing** option.



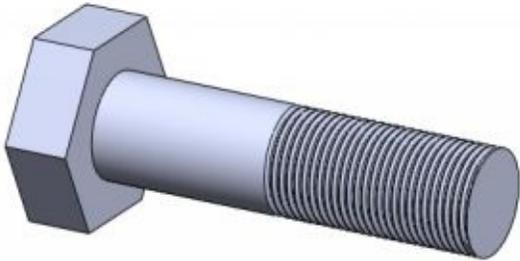
6. Set **Curve method** to **Offset Curve**.
7. Set **Alignment method** to **Tangent to curve**.
8. Click **OK**.



9. Save and close the file.

TUTORIAL 7

In this tutorial, you create a bolt.



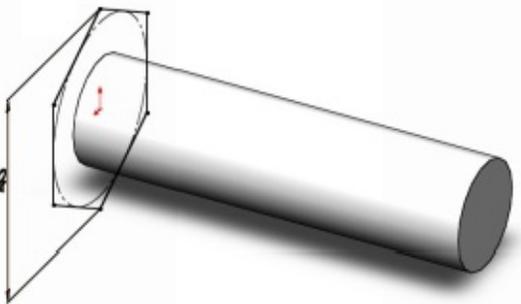
Start a new part file

1. Start a new part file using the **Part** template.
2. Construct a cylindrical feature of 20 mm diameter and 75 mm length.

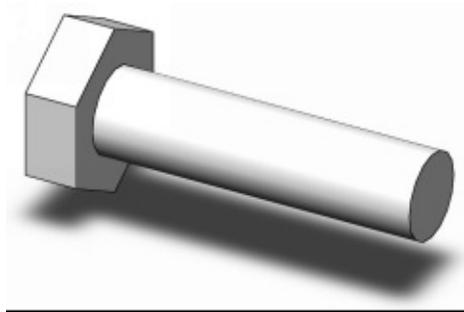


Creating the second feature

1. Start a sketch on the left face of the cylinder.
2. On the CommandManager, click **Sketch > Polygon**.
3. Create a hexagon and dimension it as shown.

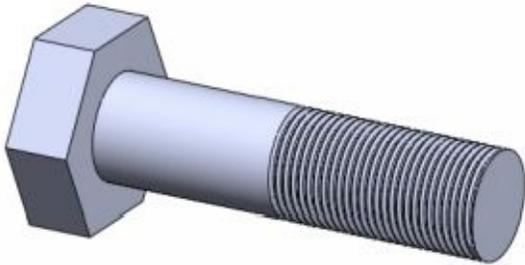


4. Exit the sketch.
5. Extrude the sketch up to 12 mm depth.



Adding Cosmetic Threads

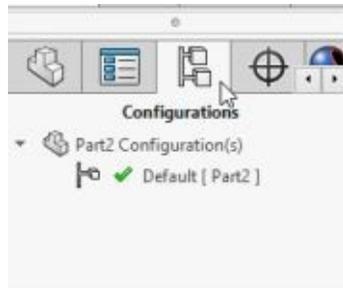
1. On the CommandManager, click **Features > Hole > Thread** .
2. Click on the right circular edge of the model geometry.
3. Select **End Condition > Blind** and type-in 40 in the **Distance** box.
4. On the PropertyManager, select **Type > Metric Die**.
5. Select **Size > M20x1.5**.
6. Click the **Override Diameter**  icon below the **Size** drop-down. This will enable you to override the thread diameter.
7. Click **OK** to add the thread.
8. Save the file as Tutorial7.sldprt.



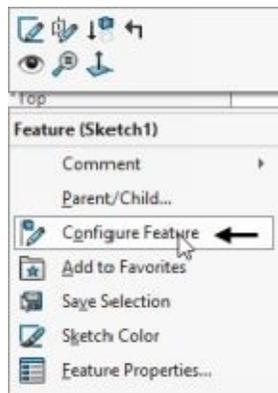
Creating Configurations Table

Configurations Table allows you to design a part with different variations, sizes, materials and other attributes. Now, you will create different variations of the bolt created in the previous section.

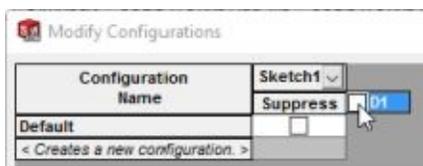
1. Click the **Configurations** tab on the left-side of the window. Notice that there is only one configuration available in the tree.



2. Click the **FeatureManager Design Tree**  tab.
3. In the FeatureManager Design Tree, expand **Boss-Extrude1** and click the right mouse button on **Sketch1**.
4. Select **Configure Feature** from the shortcut menu.

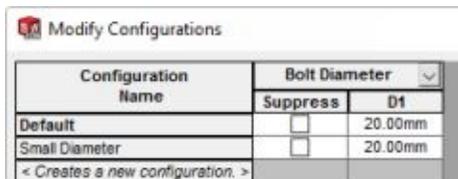


5. On the **Modify Configurations** dialog, click the down-arrow next to **Sketch1** and check **D1**. Click on the dialog to display the Diameter dimension.

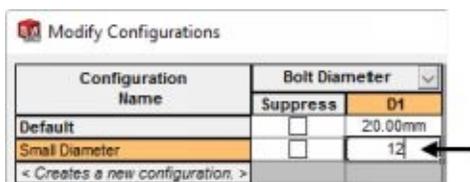


6. On the **Modify Configurations** dialog, click the right mouse button on **Sketch1** and select **Rename**.
7. Type-in **Bolt Diameter** and right-click on the **Modify Configurations** dialog.
8. On the **Modify Configurations** dialog, click in the *< Creates a new configuration. >* cell and type-in **Small Diameter**. Click on the table to

create a new configuration.



9. Type **12** in the **D1** cell of Small Diameter configuration.



10. In the FeatureManager Design Tree, double-click on **Thread1**, **Boss-Extrude2**, and **Sketch2** to add them to the table.
11. On the **Modify Configurations** dialog, click the right mouse button on **Boss-Extrude2** column and select Delete. The column is removed from the table.
12. Add the dimensions of **Thread1** and **Sketch2** by clicking the down-arrows next to columns and checking the dimensions.
13. Modify the D6 value of Thread1 for Small Diameter configuration to 12.
14. Modify the D1 value of the Sketch2 for Small Diameter configuration to 20.

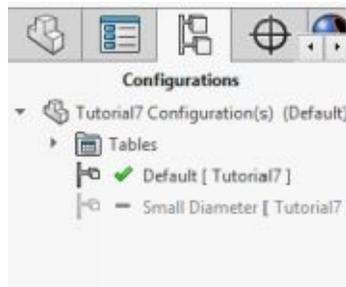
, D1=20. Black arrows point to the D6 and D1 cells."/>

Thread2					Sketch2	
D1	D4	D6	D7	D8	Suppress	D1
0.00deg	40.00mm	20.00mm	1.50mm	0.00deg	<input type="checkbox"/>	40.00mm
0.00deg	40.00mm	12.00mm	1.50mm	0.00deg	<input type="checkbox"/>	20

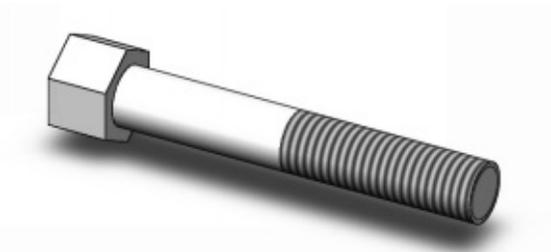
15. On the **Modify Configurations** dialog, type-in **Custom Table** in the box located at the bottom, and then click the **Save table view** button next to it.



16. Click **OK** to close the dialog.
17. Click the **Configurations** tab on the left-side pane and notice two configurations along with a configurations table.



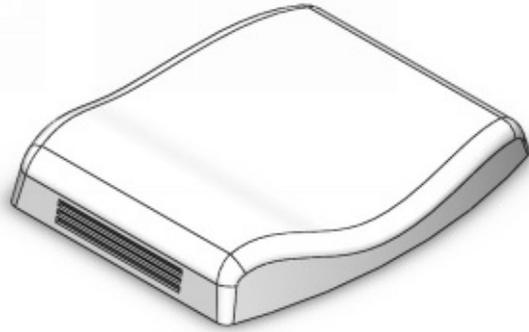
18. Click the right mouse button on the **Small Diameter** configuration and select **Show Configuration**.



19. To edit the configuration table, click the right mouse button on **Custom Table** and select **Show Table**. The **Modify Configurations** dialog appears. Change the values or create new configurations, and then click the **Save the table view**  icon.
20. Save as **Configurations Table.sldprt** and close the file.

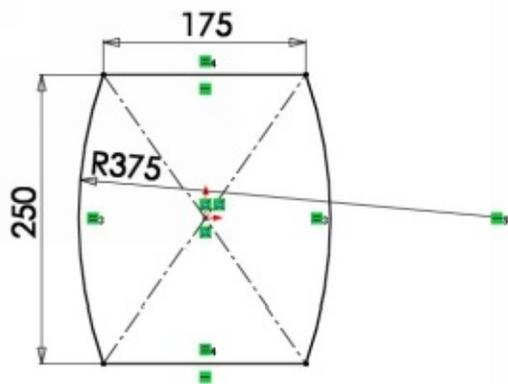
TUTORIAL 8

In this tutorial, you create a plastic casing.

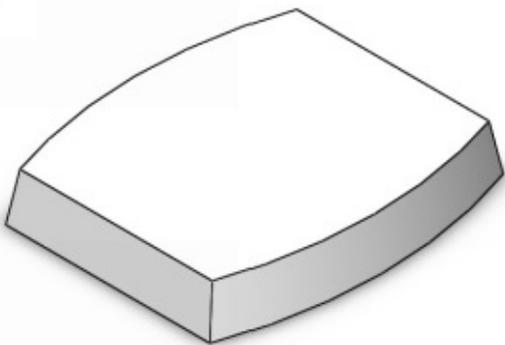


Creating the First Feature

1. Open a new Part file.
2. Create a sketch on the Top Plane, as shown in figure.

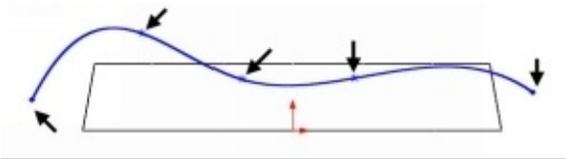


3. Click **Exit Sketch**.
4. Click the **Features > Extruded Boss/Base**  on the CommandManager.
5. Set the **Distance** to 40.
6. Click the **Draft on/off**  icon and set the **Draft Angle** to 10.
7. Click **OK**.

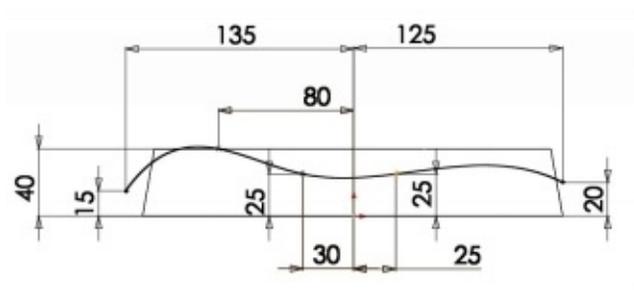


Creating the Extruded surface

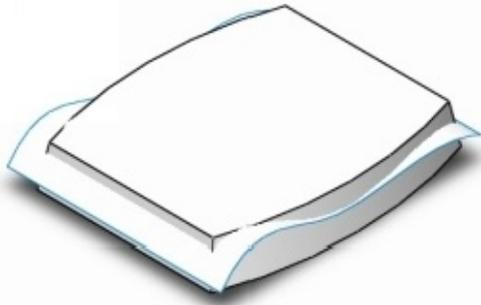
1. On the CommandManager, click the right mouse on any one of the tabs and select **Surfaces**.
2. Click **Surfaces** > **Extruded Surface**  on the CommandManager and select the Right Plane.
3. On the View Heads up toolbar, click **View Orientation** > **Normal To** .
4. Click **Sketch** > **Spline**  on the CommandManager.
5. Create a spline, as shown in figure.



6. Apply dimensions to the spline, as shown below.



7. Click **Exit Sketch**.
8. Set the **End Condition** type to **Mid Plane**.
9. Extrude the sketch up to 215 mm depth.



Replacing the top face of the model with the surface

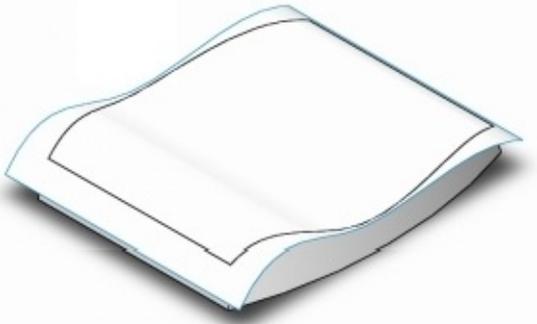
1. On the CommandManager, click **Surfaces > Replace Face** .

Now, you need to select the face to be replaced.

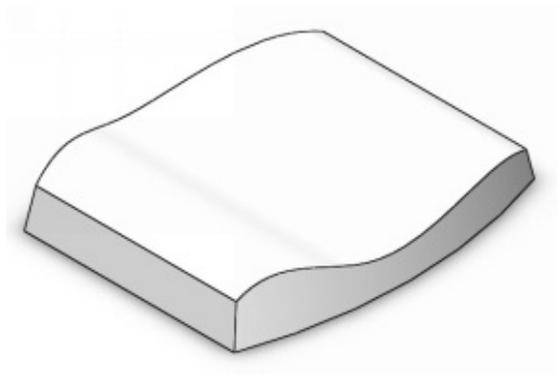
2. Select the top face of the model.

Next, you need to select the replacement face or surface.

3. Click in the **Replacement Face(s)**  box on the PropertyManager and select the extruded surface.
4. Click **OK** to replace the top face with a surface.

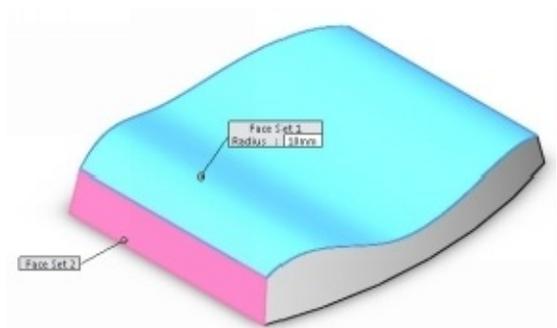


5. Hide the extruded surface by clicking on it and selecting **Hide** .

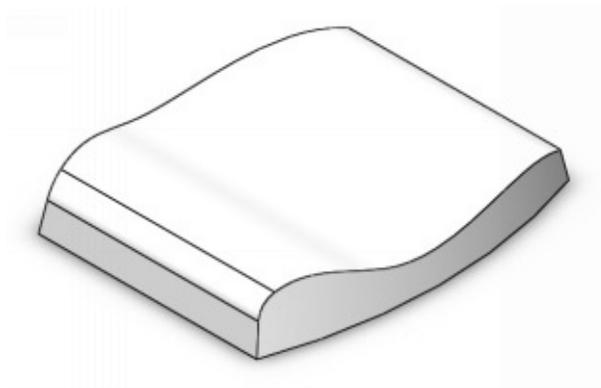


Creating a Face fillet

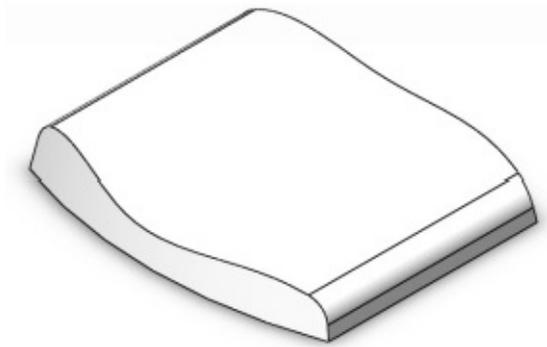
1. On the CommandManager, click the **Fillet**  icon on the **Features** tab.
2. On the PropertyManager, click the **Manual** tab.
3. Click the **Face Fillet**  button on the **Fillet** PropertyManager.
4. Select the top surface as the first face.
5. On the PropertyManager, click in the **Face Set 2** box and select the inclined front face as the second face.



6. On the PropertyManager, under the **Fillet Parameters** section, select **Symmetric** from the drop-down and type 20 in the **Radius** box. Click **OK** to create the face fillet.

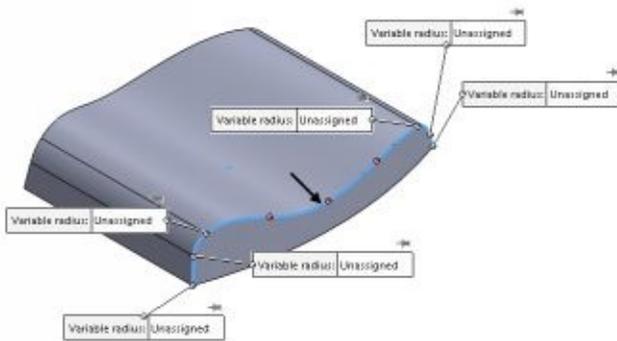


- Likewise, apply a face fillet of 20 mm radius between top surface and the back inclined face of the model.

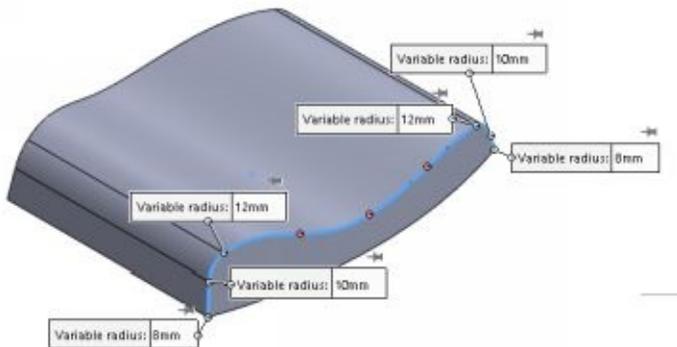


Creating a Variable Radius fillet

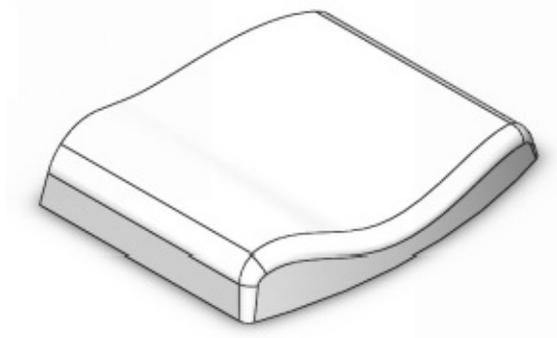
- Click the **Fillet** icon on the **Features** tab.
- Click the **Variable Size Fillet**  icon on the **Fillet** PropertyManager.
- Click the right mouse button on the curved edge and select **Select Tangency**.



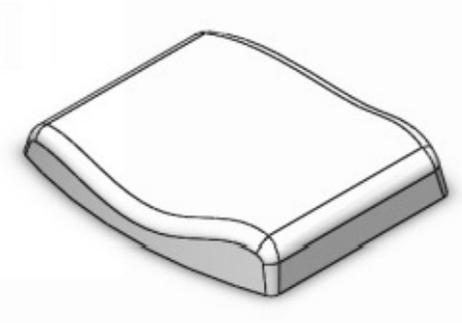
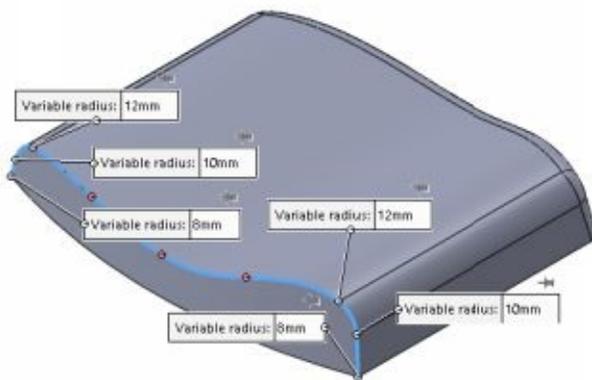
- Modify the radii values in callouts as shown.



6. Make sure the **Smooth transition** option is checked under the **Profile** drop-down.
7. Click **OK** to create the variable fillet.



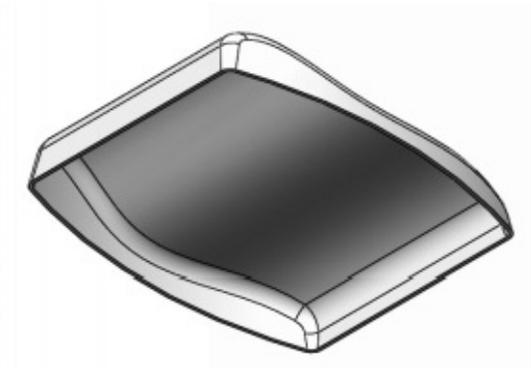
8. Create a variable fillet on other curved edge as shown.



Shelling the Model

1. Click the **Shell**  icon on the **Features** tab.
2. On the PropertyManager, type 2.5 mm in the **Thickness** box.

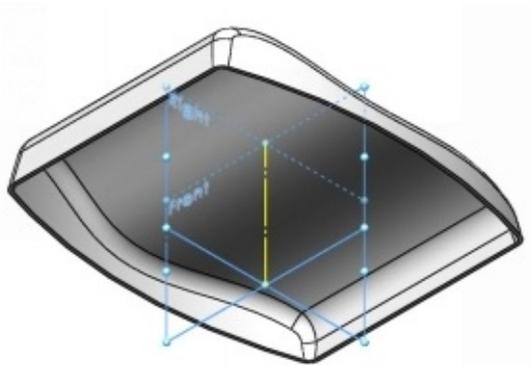
3. Change the orientation of the model and select the bottom face.



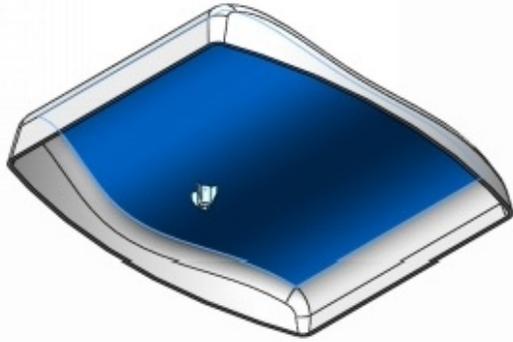
4. Click **OK**.

Creating the Mounting Bosses

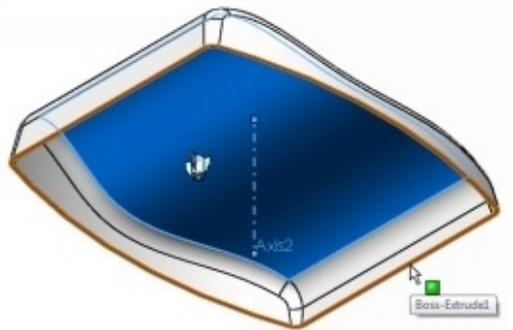
1. On the CommandManager, click **Features** > **Reference Geometry** > **Axis** .
2. Press the Ctrl key and select the Right and Front planes from the FeatureManager Design Tree.



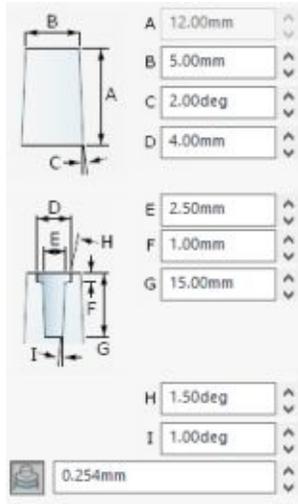
3. Click **OK** on the PropertyManager.
4. Click **Insert** > **Fastening Feature** > **Mounting Boss**  on the Menu bar and select the inside of top surface.



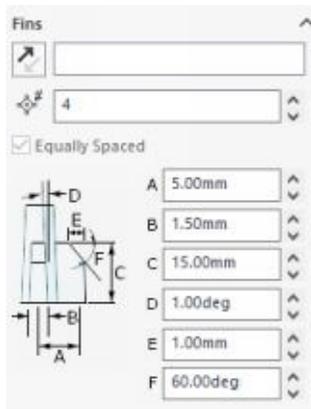
5. Click in the **Select Direction** box on the PropertyManager and select the **Axis** from the FeatureManager Design Tree.
6. Click the **Reverse Direction**  icon next to the **Select Direction** box, if the mounted boss appears upwards.
7. Set the **Boss Type** to **Hardware Boss**.
8. Select the **Thread**  icon.
9. In the **Boss** section, select **Select mating face**.
10. Select the bottom face of the model geometry.



11. Specify the parameters in the **Boss** section as shown.



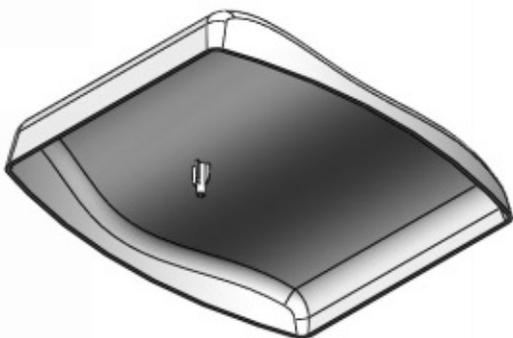
12. Specify the parameters in the **Fins** section, as shown.



13. In the **Favorite** section, click the **Add or update a favorite**  icon.

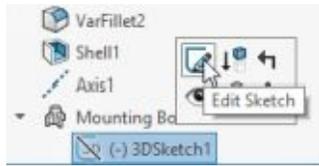
14. Type **Favorite1** in the **Add or Update a Favorite** dialog and click **OK**.

15. Click **OK** on the PropertyManager to create the boss.

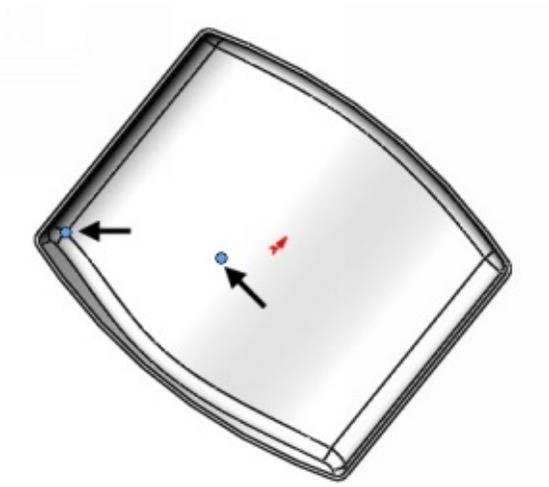


16. In the FeatureManager Design Tree, expand **Mounting Boss1** and notice that a 3D sketch is created.

17. Select **3DSketch1** and click **Edit Sketch**.

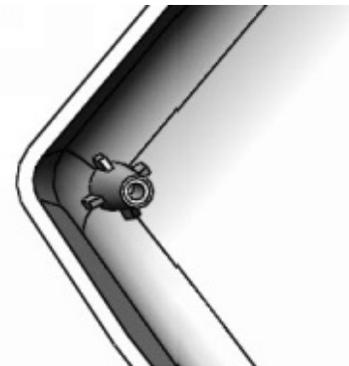


18. Press the Ctrl key and select the points as shown.



19. Click the **Coincident** icon on the PropertyManager, and click **OK**.

20. Click **Exit Sketch**. The position of the mounting boss is changed.

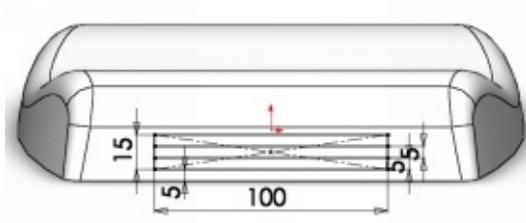


21. Likewise, create other mounting bosses as shown.

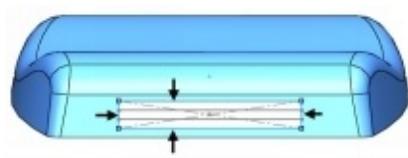


Creating the Vent Feature

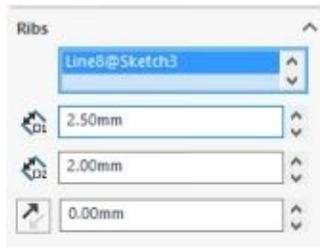
1. Create a sketch on the front inclined face.



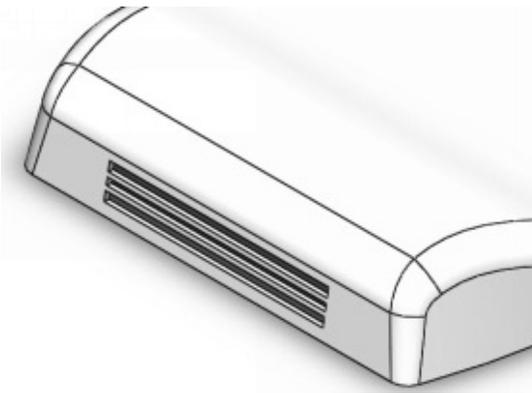
2. Click **Exit Sketch**.
3. On the Menu bar, click **Insert > Fastening Feature > Vent** .
4. Select the four lines of the rectangle to define the boundary.



5. Click in the **Ribs** selection box and select the two horizontal lines.
6. Set the rib parameters, as shown below.



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



8. Save the model as Plastic cover.sldprt and close it.

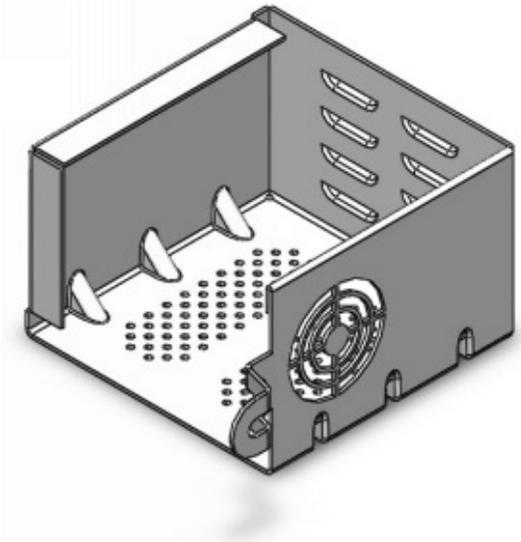
Chapter 7: Sheet metal Modeling

This chapter will show you to:

- Construct Base Flange
- Construct Edge Flange
- Construct Miter Flange
- Close Corners
- Apply Corner Treatments
- Add Tab features
- Add Jogs
- Unfold and refold the bends
- Cut across bends
- Perform Forming Operations
- Add Hems
- Add Vents
- Add Gussets
- Make Fill Patterns
- View Flat Pattern

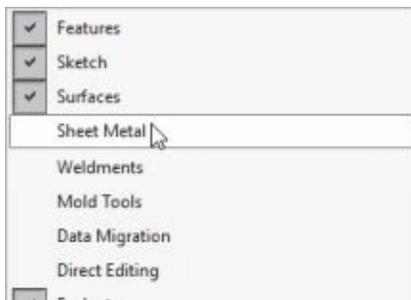
TUTORIAL 1

In this tutorial, you construct the sheet metal model shown in figure.



Opening a New File

1. Open a new part file.
2. On the CommandManager, click the right mouse button on anyone of the tabs and select **Sheet Metal**, if the **Sheet Metal** tab is displayed.

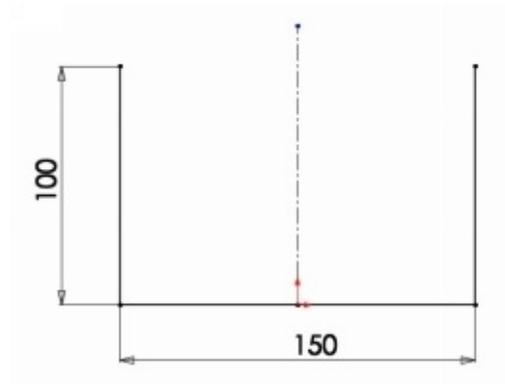


The **Sheet Metal** tab appears.

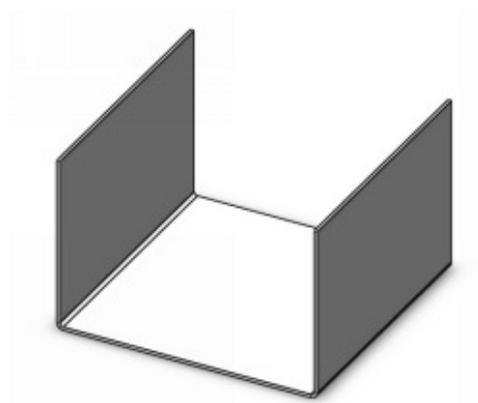
Constructing the Base Flange

1. To construct a base flange, click **Sheet Metal** > **Base Flange/Tab**  on

- the CommandManager.
2. Select the front plane.
 3. Construct the sketch, as shown.

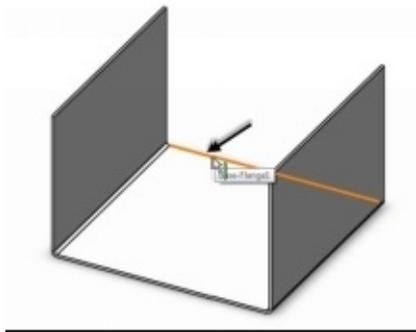


4. Click **Exit Sketch**.
5. On the PropertyManager, under **Direction 1** section, type-in 150 in the **Depth** box.
6. Check the **Use gauge table** option.
7. Select **SAMPLE TABLE –ALUMINIUM-METRIC UNITS**.
8. On the PropertyManager, under the **Sheet Metal Parameters** section, select **Gauge 14**.
9. Set the **Bend Radius** value to 2.
10. Leave the **Auto Relief** section to default settings.
11. Click **OK** to construct the base flange.

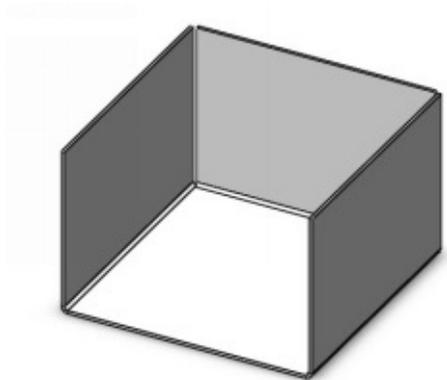


Adding an Edge flange

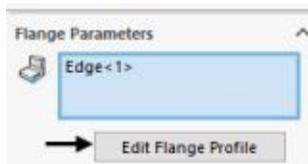
1. To add the flange, click **Sheet Metal > Edge Flange**  on the CommandManager.
2. Select the edge on the top face.

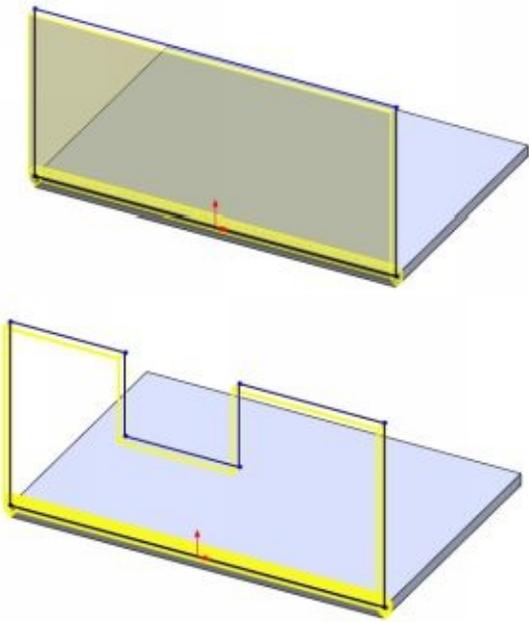


3. Move the pointer upward and click.
4. On the PropertyManager, under the **Flange Length** section, set **Length** to 100.
5. Click on the **Inner Virtual Sharp**  icon in the **Flange Length** section. This measures the flange length from the outer face.
6. Under the **Flange Position** section, click the **Material Outside**  icon.
7. Click **OK** to add the flange.



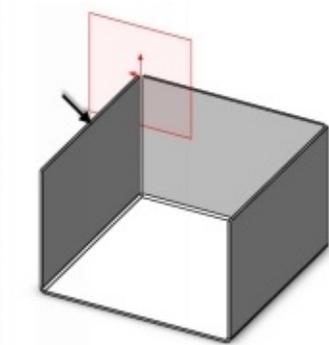
In SOLIDWORKS 2016, you can use the **Edit Flange Profile** button to change the shape of the flange. The sketch mode is activated when you click this button. Use the sketch tools to change the flange shape.



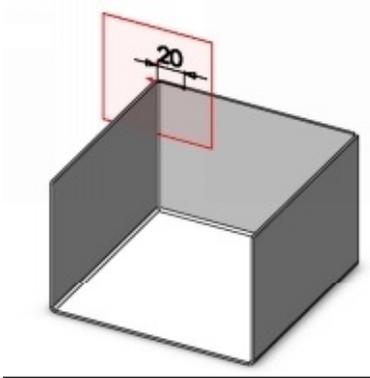


Constructing the Miter Flange

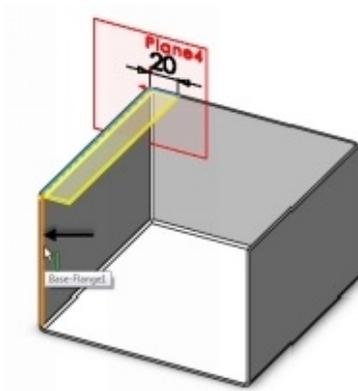
1. To construct the miter flange, click **Sheet Metal > Miter Flange**  on the CommandManager.
2. Click the top edge left flange.



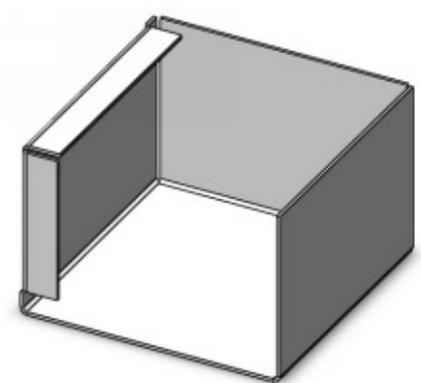
3. Draw the sketch, as shown.



4. Click **Exit Sketch**.
5. Click on the vertical edge of the left flange.

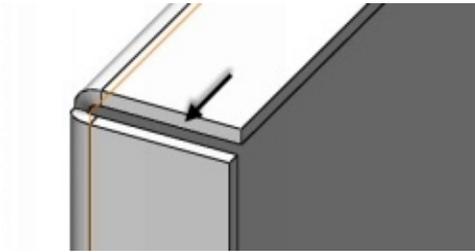


6. On the **Miter Flange** PropertyManager, set the **Flange Position** to **Material Inside**  and **Gap distance** to 1.25.
7. Under the **Start/End Offset** section, set the **Start Offset Distance** and **End Offset Distance** to **10**.
8. Click **OK** to construct the miter flange.

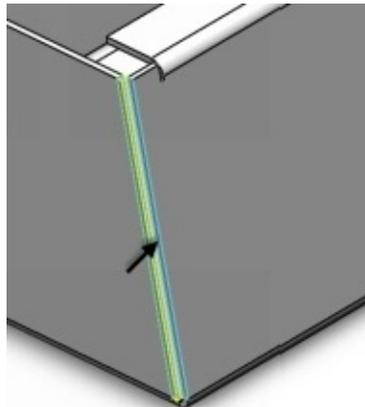


Closing the Corners

1. To close the corners, click **Sheet Metal > Corners > Closed Corner** .
2. Select the corner face of the miter flange.



3. On the PropertyManager, set the **Corner type** to **Butt** .
4. Rotate the model and select the other corner faces.

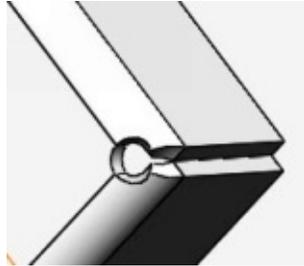


5. Click **OK**  to close the corners.

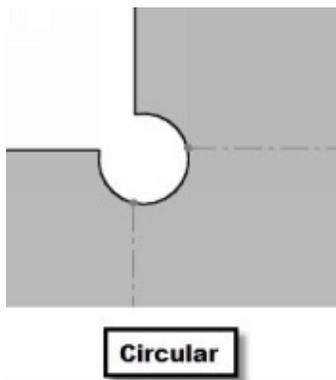
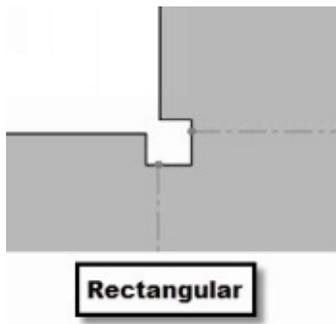
Adding Corner Reliefs

1. To add corner reliefs, click **Sheet Metal > Corners > Corner Relief** . This selects the entire part.
2. On the PropertyManager, under the **Corners** section, click **Collect all corners**.
3. Under the **Relief Options** section, set the **Relief type** **Circular** .

4. Type **2** in the **Slot Width** box.
5. Click **OK** to add the corner relief.

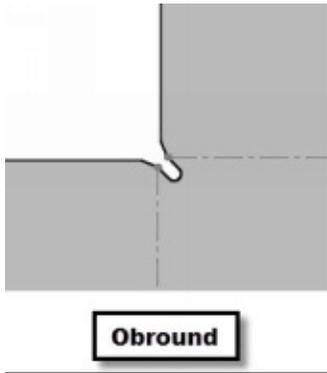


The different types of the reliefs are:

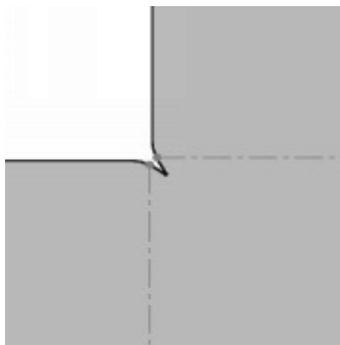




Tear



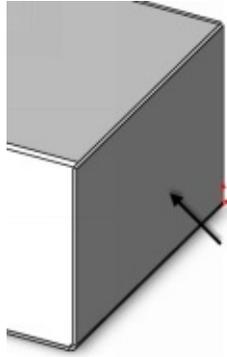
Obround



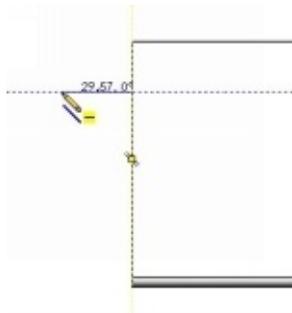
Constant Width

Adding a Tab feature

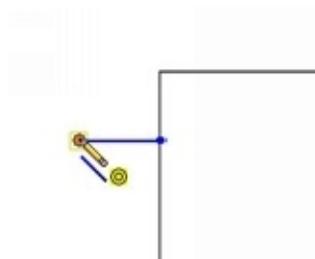
1. To add a tab, click **Sheet Metal** > **Base Flange/Tab**  on the CommandManager.
2. Click on the right side of the base flange.



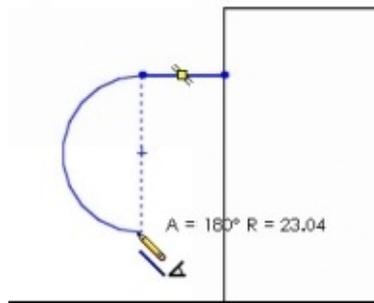
3. On the CommandManager, click **Sketch** > **Line**.
4. Click on the vertical edge of the face and move the pointer toward left.
5. Again, click to draw a horizontal line.



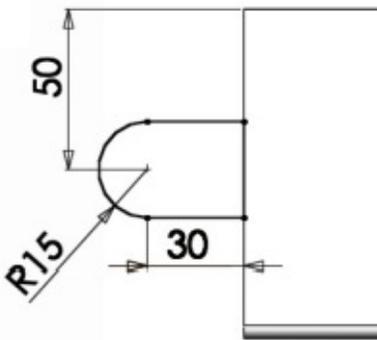
6. Move the pointer toward endpoint of the line. The arc mode gets active.



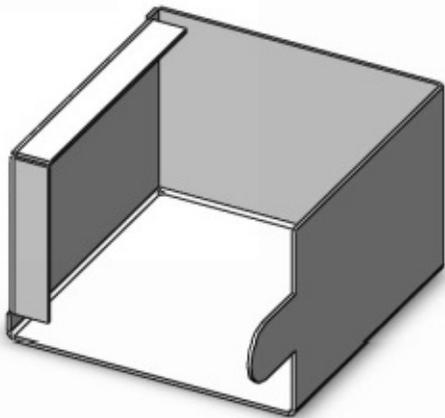
7. Move the pointer toward left, and then downwards.
8. Click to create an arc.



9. Close the sketch and add dimensions.

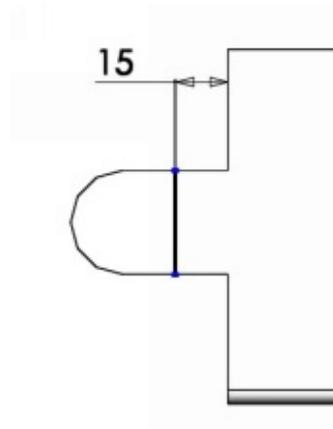


10. Exit the sketch and click **OK** on the PropertyManager.

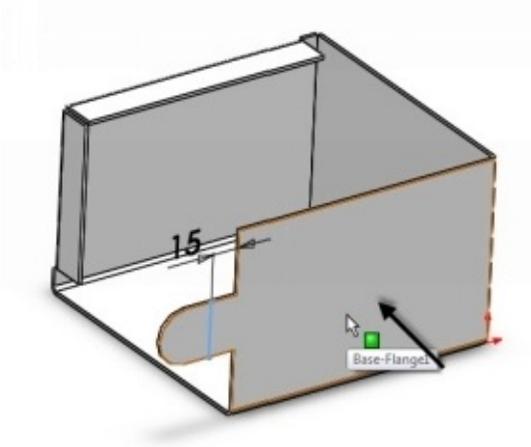


Adding a Jog

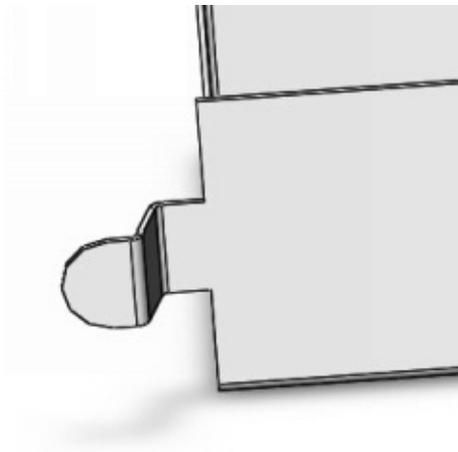
1. To add a jog, click **Sheet Metal > Jog**  on the CommandManager.
2. Click on the tab face and draw a line.



3. Click **Exit Sketch**.
4. Click on the base flange to define the fixed face.

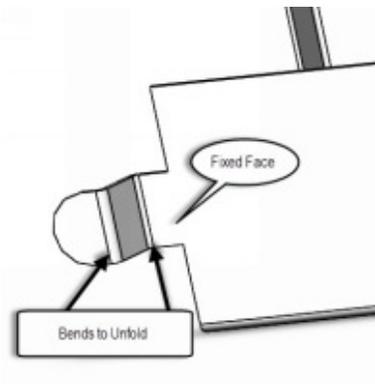


5. On the PropertyManager, under the **Jog Offset** section, set the **Offset Distance** to 20.
6. Set the **Dimension position** to **Outside Offset** .
7. Set the **Jog Position** to **Bend Centerline** .
8. Set the **Jog Angle** to 75.
9. Click **OK**.

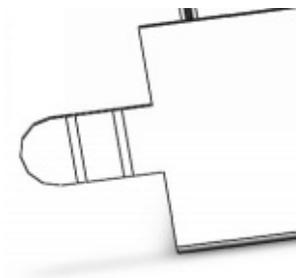


Unfolding the bends

1. To unfold a bend, click **Sheet Metal > Unfold** .
2. Click on the tab face to define the fixed face.
3. Click on the bends of the jog feature.



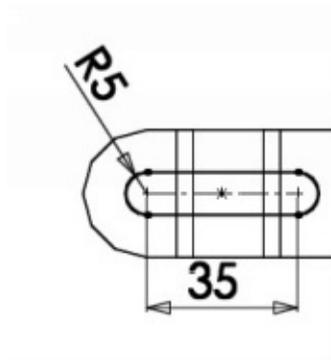
4. Click **OK**.



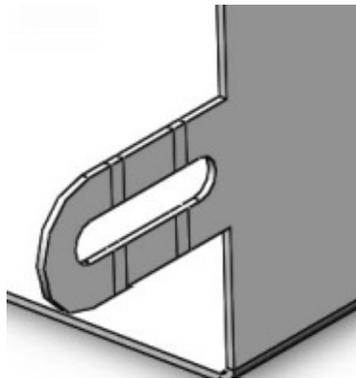
Creating Cuts across bends

1. To create cuts across bends, click **Sheet Metal > Extruded Cut**  on the CommandManager.

2. Click on the tab and draw the sketch, as shown.

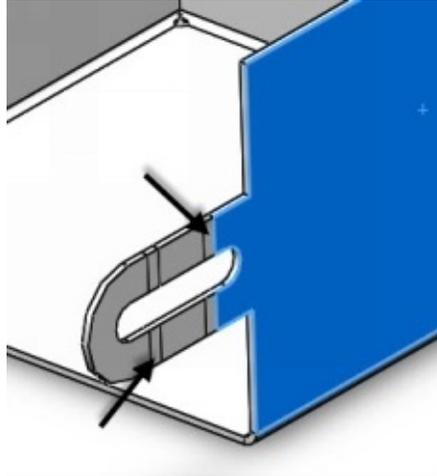


3. Click **Exit Sketch**.
4. Set the **End Condition** to **Up To Next**.
5. Click **OK** on the PropertyManager.

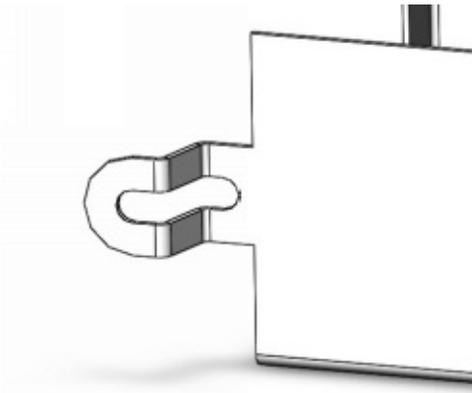


Folding the Unfolded bends

1. To fold the unfolded bends, click **Sheet Metal > Fold**  on the CommandManager.
2. Click on the jog bends.

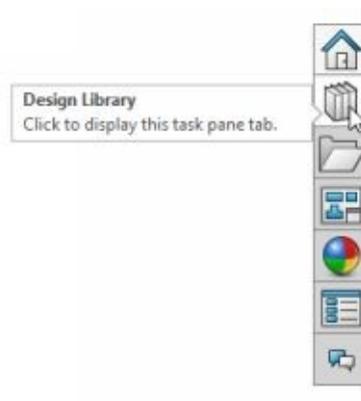


3. Click **OK**.



Performing the Forming Operations

1. To perform forming operations, click **Design Library** on the Task pane.

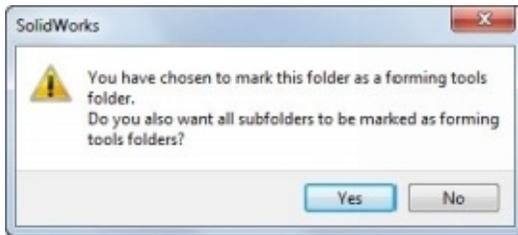


2. If the Design Library is not displayed, click **Add File Location** .

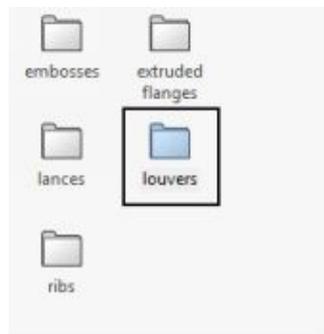
3. Browse to the location: **C:\ProgramData\SolidWorks\SOLIDWORKS**

2016\design library

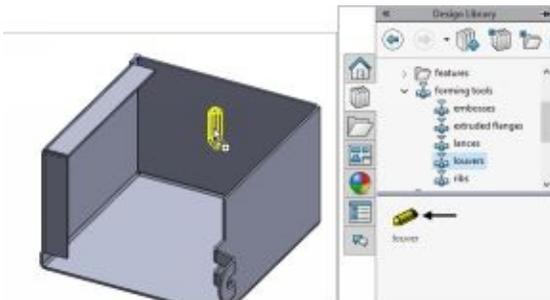
4. Click **OK**.
5. In the **Design Library**, click the right mouse button on the **forming tools** folder and select **Forming Tools Folder**.
6. Click **Yes** on the **SOLIDWORKS** message box.



7. Double-click on the **Louvers** folder.

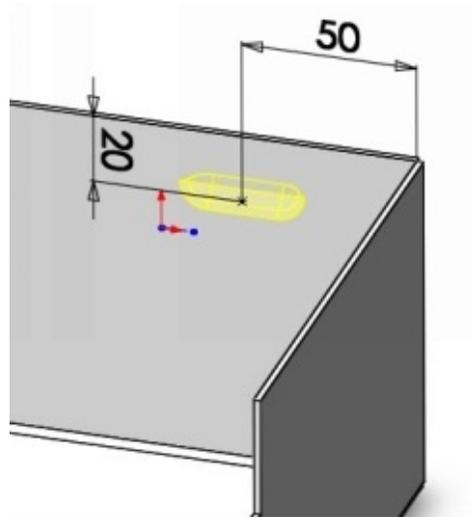


8. Click and drag the Louver onto the flange face.



9. On the PropertyManager, type-in 90 in the **Rotation Angle** box.
10. Under the **Link** section, uncheck the **Link to form tool** option.
11. Under the **Flat Pattern Visibility** section, check the **Override document settings** option.
12. On the PropertyManager, click the **Position** tab and add dimensions to

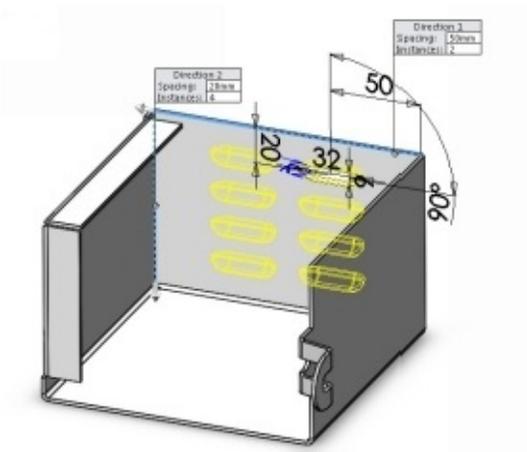
position the louver.



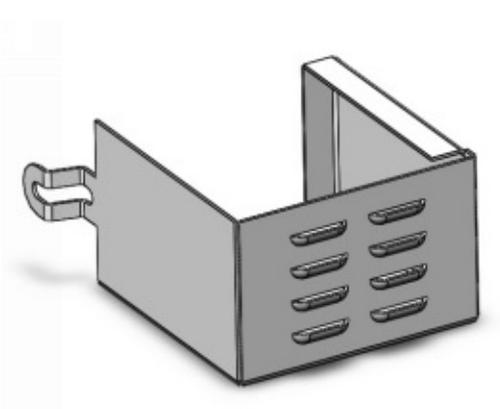
13. Click **OK**.

Making the Linear Pattern

1. On the CommandManager, click **Features** > **Linear Pattern** .
2. Click on the horizontal edge of the flange to define the first direction.
3. On the PropertyManager, under the **Direction 1** section, type-in 50 in the **Spacing** box and 2 in the **Number of Instances** box.
4. Click the **Reverse Direction**  button.
5. Click on the vertical edge of the flange feature to define the second direction.
6. Under the **Direction 2** section, type-in 20 in the **Spacing** box and 4 in the **Number of Instances** box.
7. Select the louver feature.

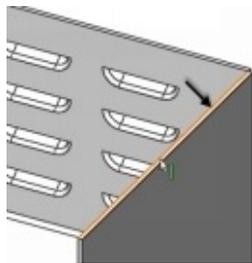


8. Click **OK** to construct the pattern.

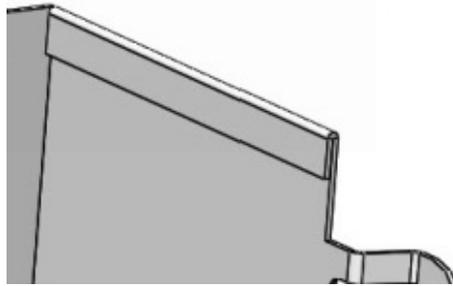


Adding a Hem

1. To add a hem, click **Sheet Metal > Hem**  on the CommandManager.
2. Click on the inner edge of the right side flange.

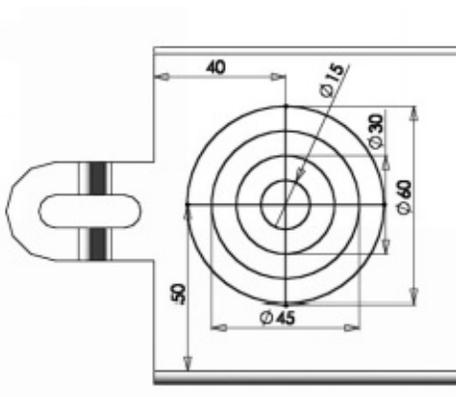


3. On the PropertyManager, under the **Type and Size** section, click the **Closed**  icon.
4. Type-in **15** in the **Length** box.
5. Click **OK** to add the hem.

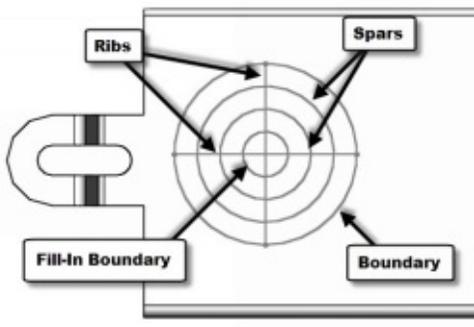


Adding a vent

1. Construct a sketch on the right side face of the base flange.

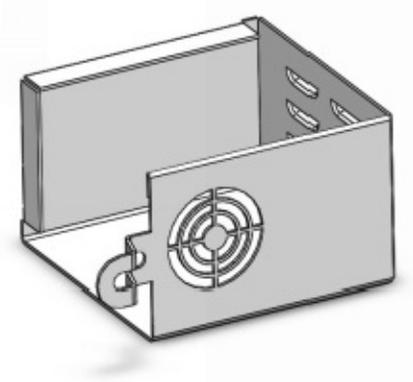


2. Click **Exit Sketch**.
3. On the CommandManager, click **Sheet Metal** > **Vent** .
4. Click on the outer most circle of the sketch to define the boundary of the vent.



5. On the PropertyManager, click in **Ribs** selection box and select the line segments from the sketch.
6. Type-in 2 in the **Width** box.
7. Click in the **Spars** selection box and select the two middle circles.

- Type-in 2 in the **Width** box.
- Click in the **Fill-In Boundary** selection box and select the inner most circle.
- Click **OK**.

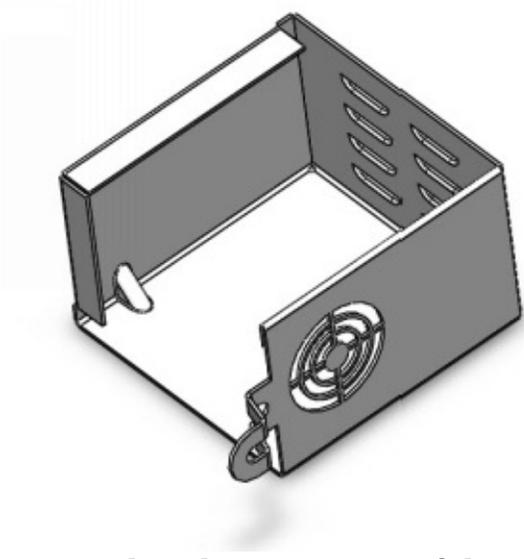


Adding Sheet Metal Gussets

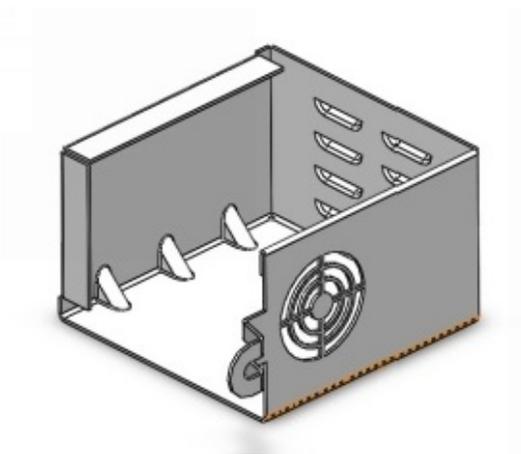
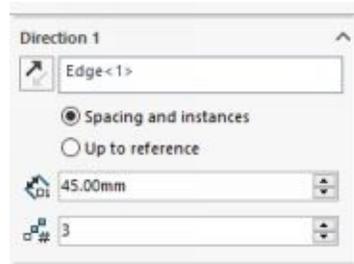
- To add gussets, click **Sheet Metal > Sheet Metal Gusset**  on the CommandManager.
- Click on the left side bend face of the base flange.
- On the PropertyManager, under the **Position** section, set the **Offset** value to 30.
- Under the **Profile** section, set the **Indent depth** to 12.
- Click the **Rounded gusset**  icon.
- Under the **Dimensions** section, set the **Indent Width** value to 10.
- Set **Indent Thickness** to 2.
- Set **Side face draft** to 2.
- Set **Inner corner fillet** and **Outer corner fillet** to 1 and 2, respectively.



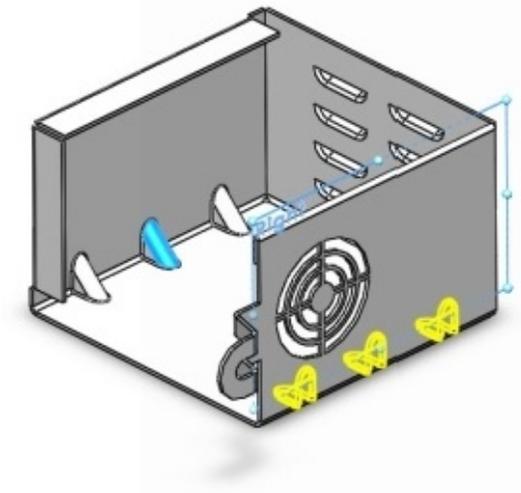
- Click **OK**.



11. Make a linear pattern of the gusset.

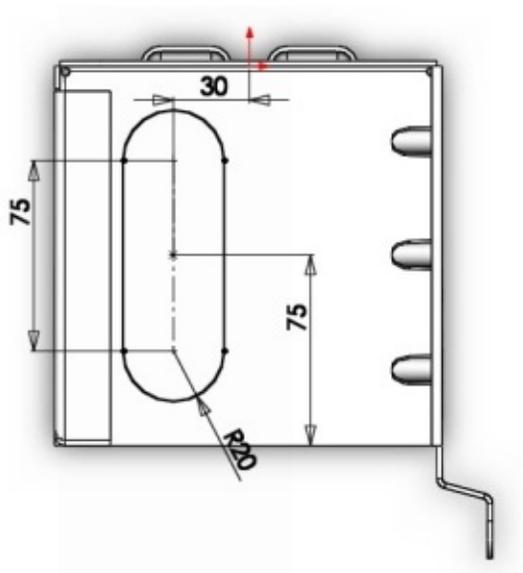


12. Mirror the gusset and the linear pattern about the right plane.

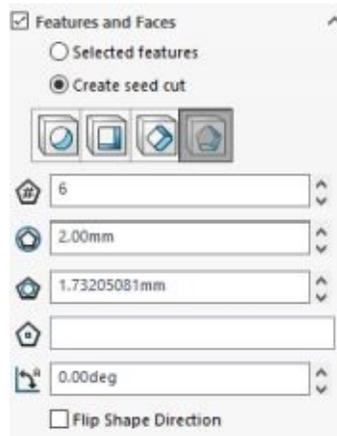


Making a Fill Pattern

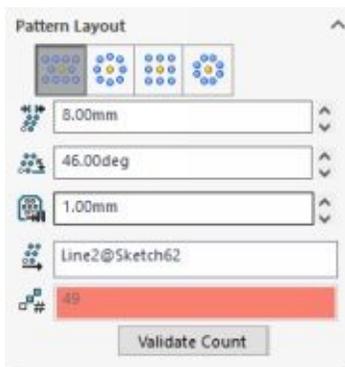
1. Draw a sketch on the top face of the base flange.



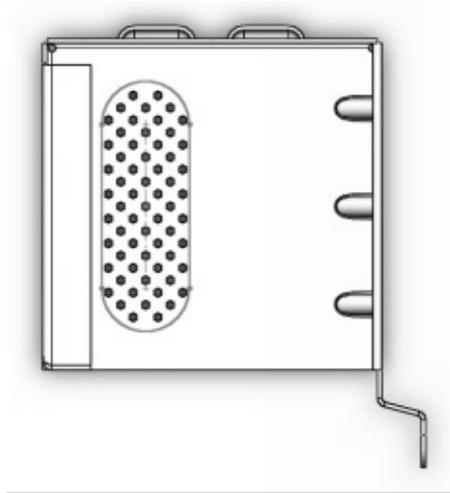
2. Click **Exit Sketch**.
3. To make a fill pattern, click **Features > Linear Pattern > Fill Pattern**  on the CommandManager.
4. Select the sketch to define the fill boundary.
5. On the PropertyManager, under the **Features and Faces** section, select **Create seed cut** and select the **Polygon** icon.
6. Set the options in the **Features and Faces** section, as shown.



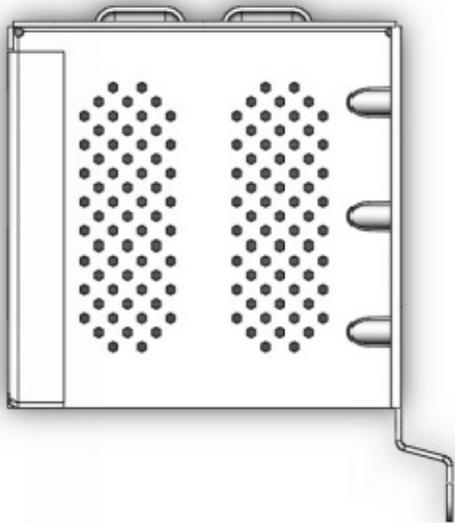
7. Under the **Pattern Layout** section, select the **Perforation** button.
8. Type-in 8 in the **Instance Spacing** box.
9. Set **Stagger Angle** to 46.
10. Set **Margins** to 1.



11. Click **OK** to make the fill pattern.

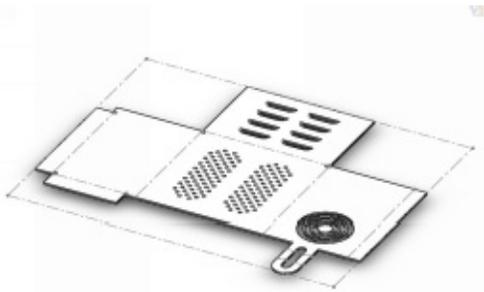


12. Mirror the fill pattern about the right plane.

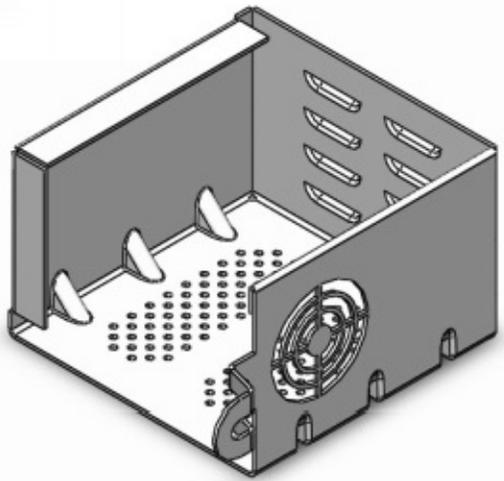


View the Flat Pattern

1. To view the flat pattern, click **Sheet Metal** > **Flatten**  on the CommandManager.



2. Again, click the **Flatten** button to switch back to the model view.



3. Save the sheet metal part.

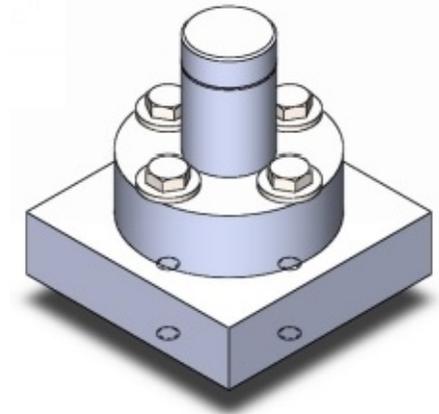
Chapter 8: Top-Down Assembly

In this chapter, you will learn to

- Create a top-down assembly
- Insert fasteners using Toolbox
- Create advanced mates

TUTORIAL 1

In this tutorial, you will create the model shown in figure. You use top-down assembly approach to create this model.



Creating a New Assembly File

1. Click the **New**  icon on the Quick Access Toolbar, select the **Assembly** icon, and click **OK**.



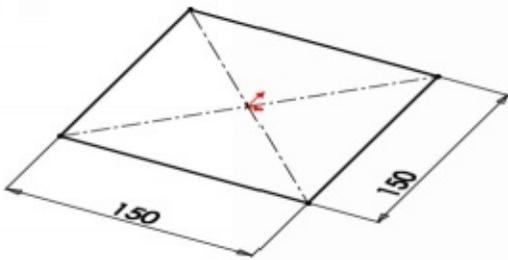
2. Click **Cancel**  on the **Begin Assembly** PropertyManager.

Creating a component in the Assembly

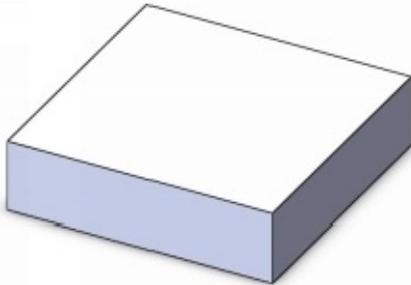
In a top-down assembly approach, you create components of an assembly

directly in the assembly by using the **New Part** tool.

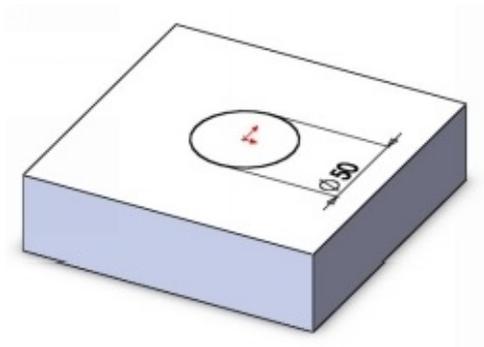
1. On the CommandManager, click **Assembly > Insert Components > New Part** .
2. In the FeatureManager Design Tree, click the Front plane. A new part is created and the **Edit Component** mode is activated.
3. Click **Exit Sketch**.
4. Click **Sketch > Sketch**  on the CommandManager and select the Top plane from the FeatureManager Design Tree.
5. Create the sketch as shown below.



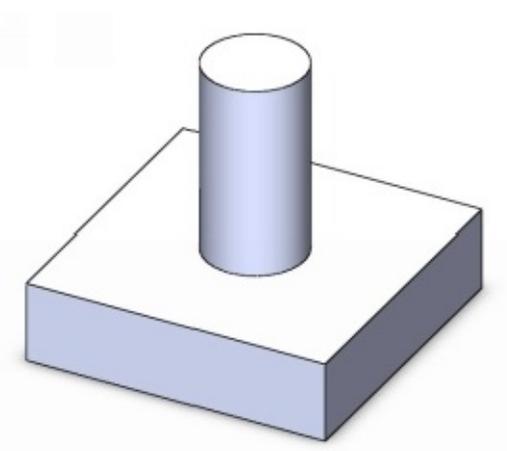
6. Click **Exit Sketch**.
7. Click **Features > Extruded Boss/Base**  on the CommandManager and extrude the sketch up to 40 mm.



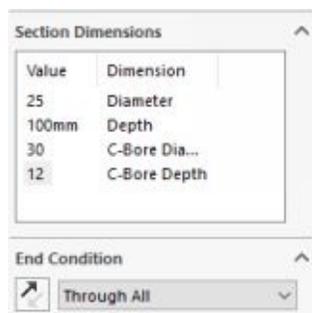
8. Start a sketch on the top face and draw a circle of **50 mm** diameter.



9. Click **Exit Sketch**
10. Extrude the sketch up to 95 mm distance.

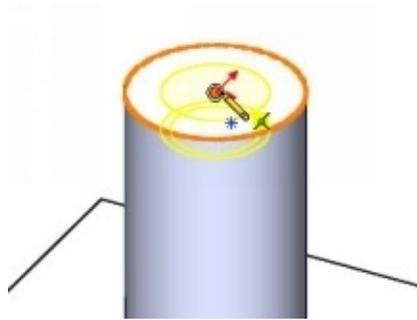


11. On the CommandManager, click **Features > Hole Wizard** .
12. On the PropertyManager, click the **Legacy Hole**  icon.
13. On the PropertyManager, select **Type > Counterbored**.
14. In the **Section Dimensions** section, modify the dimensions as shown.

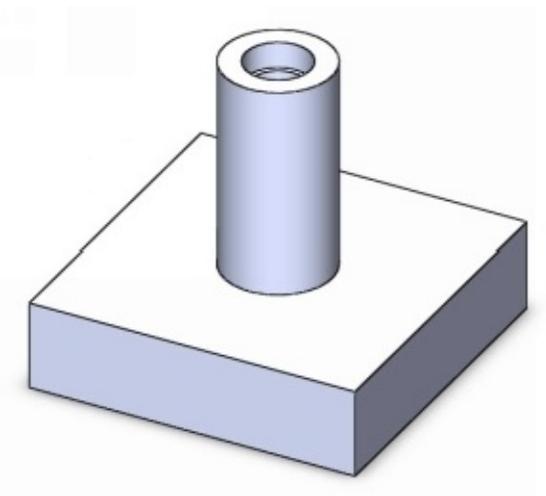


15. Set the **End Condition** to **Through All**.
16. Click the **Positions** tab and click on the top face of the second feature.

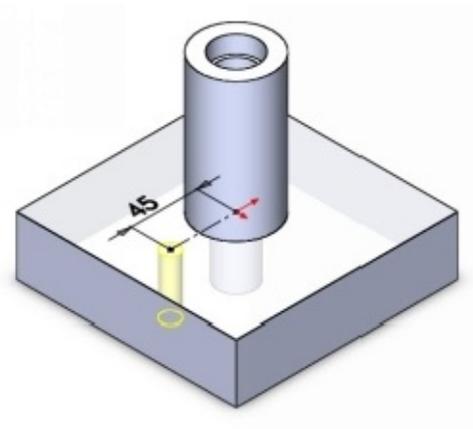
17. Select the origin point of the 3D sketch.

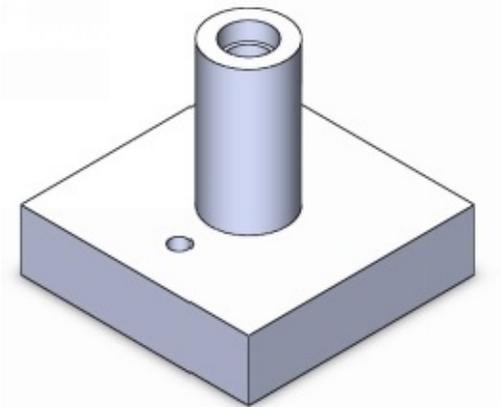


18. Click **OK**.

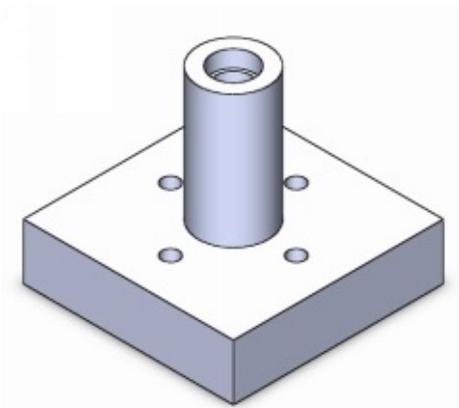


19. Create a through hole of 12 mm diameter on the first feature.





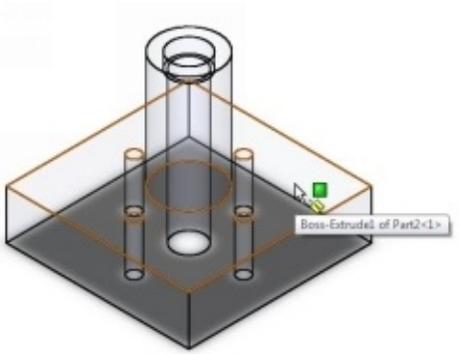
20. Create a circular pattern of the hole.



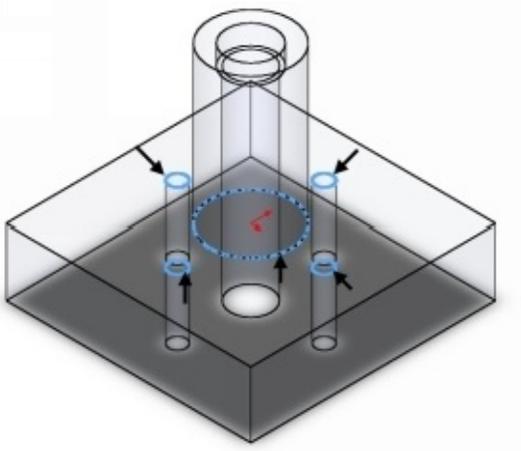
21. Deactivate the **Edit Component**  icon on the CommandManager.

Creating the Second Component of the Assembly

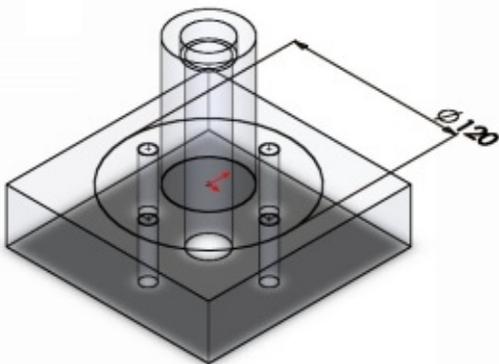
1. On the CommandManager, click **Assembly > Insert Components > New Part** .
2. Select the Front plane of the assembly from the FeatureManager Design Tree.
3. Click **Exit Sketch** on the CommandManager.
4. Click **Sketch > Sketch**  on the CommandManager.
5. Select top face of the Base.



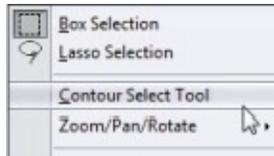
6. On the ribbon, click **Sketch** > **Convert Entities**  and select the circular edges of the Base.



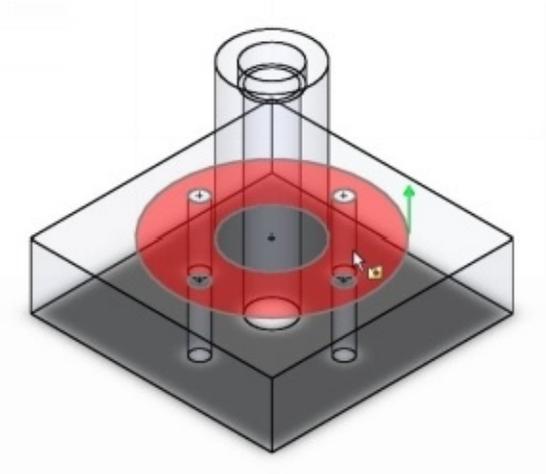
7. Click **OK** on the PropertyManager.
8. Draw a circle of 120 mm diameter.



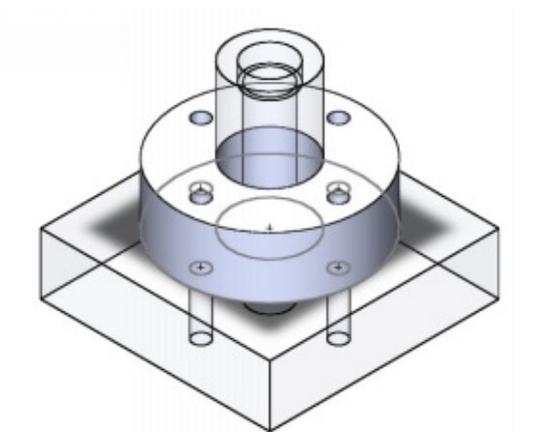
9. Click **Exit Sketch**.
10. Click the outer circle, right-click, click the down-arrows located at the bottom of the shortcut menu, and select **Contour Select Tool**.



11. Click in the region enclosed by the sketch.



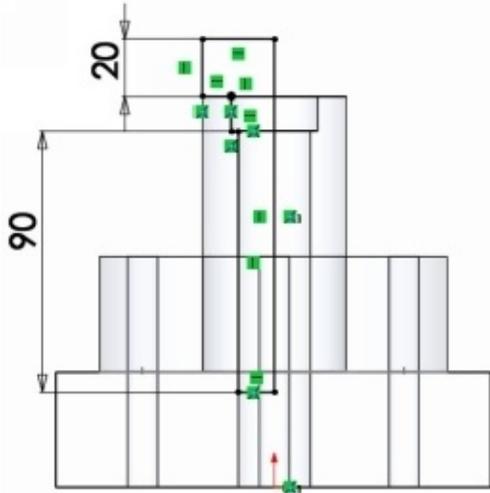
12. Activate the **Extruded Boss/Base**  tool and extrude the sketch up to 40.



13. Exit the **Edit Component**  Mode.

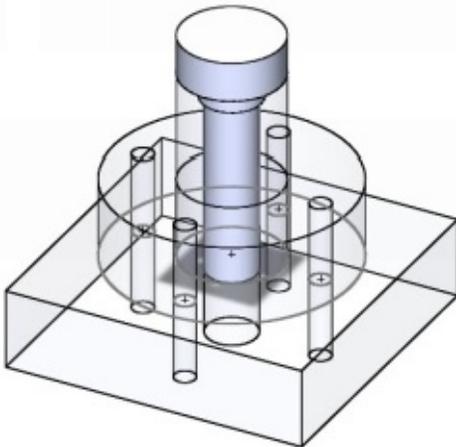
Creating the third Component of the Assembly

1. On the CommandManager, click **Assembly > Insert Components > New Part** .
2. Select the Front plane of the assembly from the FeatureManager Design Tree.
3. Draw a sketch, as shown in figure.

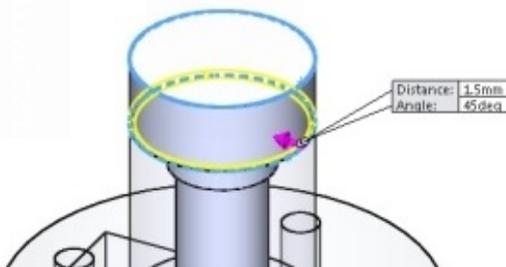


4. Click **Exit Sketch**.

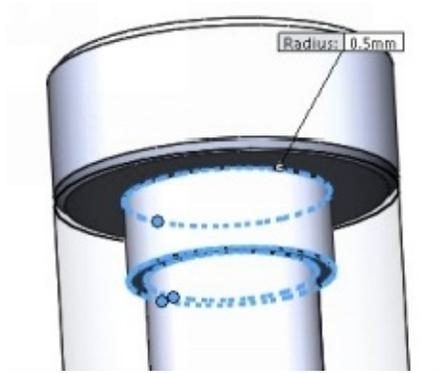
5. Activate the **Revolve**  tool and revolve the sketch.



6. Activate the **Chamfer**  tool and chamfer the edges, as shown in figure.



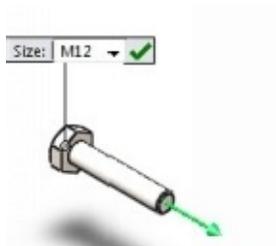
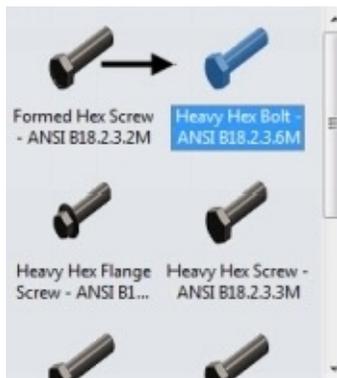
7. Activate the **Fillet**  tool and round the edges, as shown in figure.



8. Click **Edit Component**  on the CommandManager.

Adding Bolt Connections to the assembly

1. Click the **Design Library**  tab on the left side of the graphics window.
2. On the **Design Library** tab, select **Toolbox** and click **Add in now**.
3. On the **Design Library** tab, click **Toolbox > ANSI Metric**.
4. Under the **ANSI Metric** folder, click **Bolts and Screws > Hex Head**.
5. Click **Heavy Hex Bolt-ANSI B18.2.3.6M** and drag it into the graphics window.



6. On the **Configure Component** PropertyManager, set the bolt properties as shown.

Properties ^

Size:
M12 v

Length:
100 v

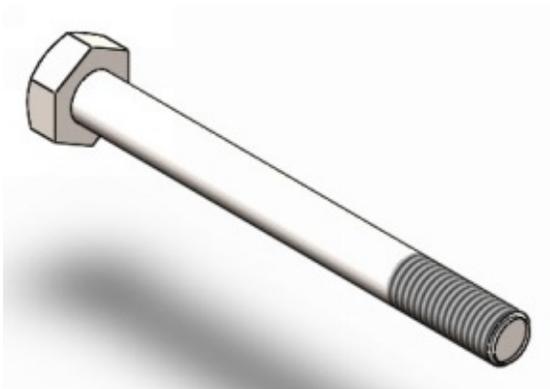
Thread Length:
30 v

Thread Display:
Cosmetic v

Comment:

Configuration Name:
B18.2.3.6M - Heavy hex bolt M12 x 1.75

7. Click the **Add** button in the **Part Number** section and click **OK**.
8. Click **OK** on the PropertyManager. Another instance of the bolt is attached to the pointer.



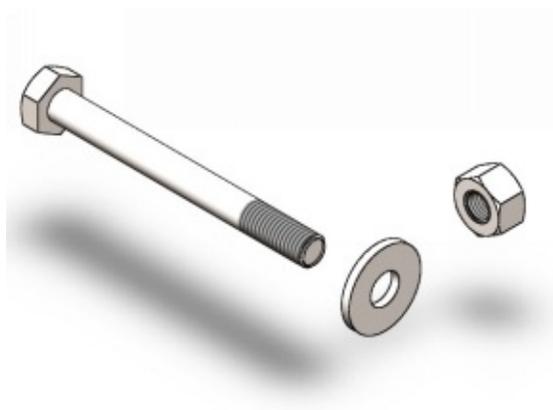
9. Click **Cancel**  to stop inserting more bolts.
10. On the **Design Library** tab, click **Washer** > **Plain Washers**.
11. The **Regular Flat Washer** and drag it into the graphics window.
12. Set the washer size to **M12** and click the **Add** button under the **Part Numbers** section of the PropertyManager.
13. Click **OK** on the **Part Number** dialog.
14. Click **OK** and **Cancel**.



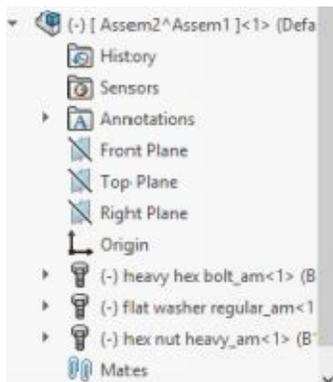
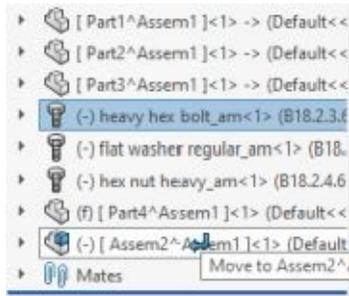
15. On the **Design Library** tab, under **ANSI Metric**, click **Nuts > Hex Nuts**.
16. Click and drag **Hex Heavy Nut** into the graphics window.
17. On the **Configure Component** PropertyManager, set the nut properties as shown.



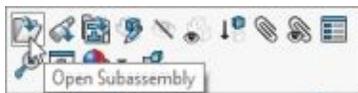
18. Click the **Add** button under the **Part Numbers** section of the PropertyManager.
19. Click **OK** on the **Part Number** dialog.
20. Click **OK** and **Cancel**.



21. On the CommandManager, click **Assembly > Insert Components > New Assembly** . A new subassembly is inserted into the assembly.
22. In the FeatureManager Design Tree, click and drag the bolt, nut and washer into the subassembly.



23. Click on the subassembly and select **Open Subassembly**. The Subassembly is opened in another window.



24. On the CommandManager, click **Assembly > Mate** .
25. Click the **Multiple mate mode**  icon on the PropertyManager.
26. Click on the cylindrical face of the bolt, and then click the cylindrical faces of the washer and nut. The bolt face is selected as a common mate entity.



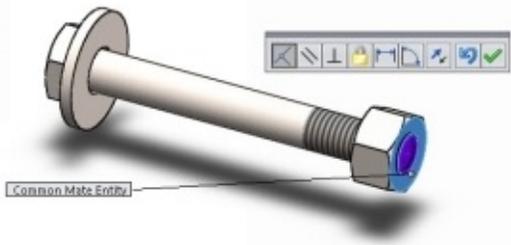
27. Click **Add/Finish Mate** .
28. Click in the **Entities to mate**.
29. Select the flat face of the bolt.
30. Select the flat face of washer.



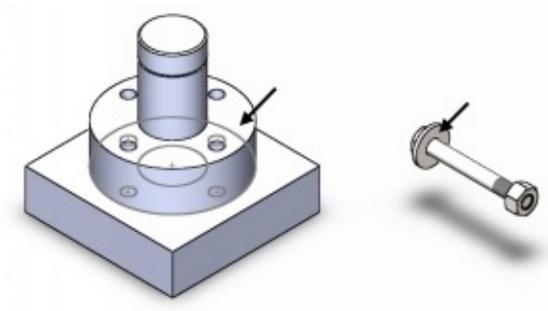
31. Click **Flip Mate Alignment** , and then click **OK** on the **SOLIDWORKS** dialog.
32. Click **Add/Finish Mate** .



33. Likewise, align the end faces of the bolt and nut, and click **OK** on the PropertyManager.

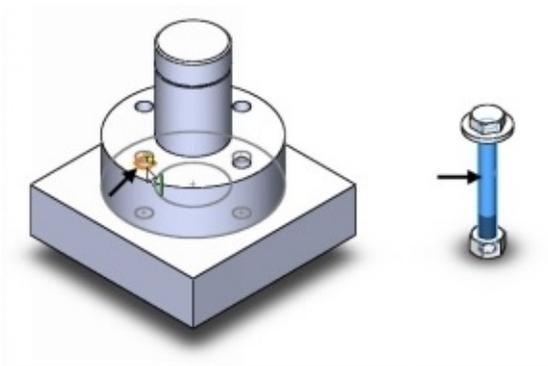


34. Save and close the subassembly.
35. On the CommandManager, click **Assembly > Mate** .
36. Click on the flat face for the washer and the part with holes.



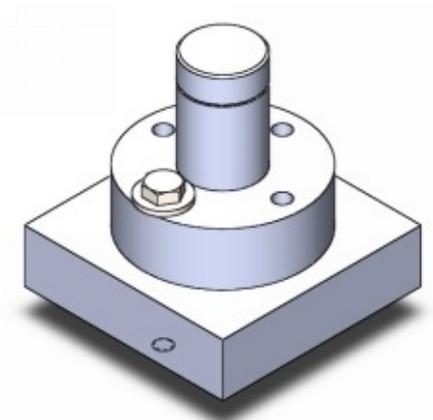
37. Click the **Coincident**  icon, and click **Add/Finish Mate** .

38. Click the Cylindrical face of the bolt and circular edge of the hole as shown.



39. Click the **Concentric**  icon, check the **Lock Rotation** option, and click **Add/Finish Mate** .

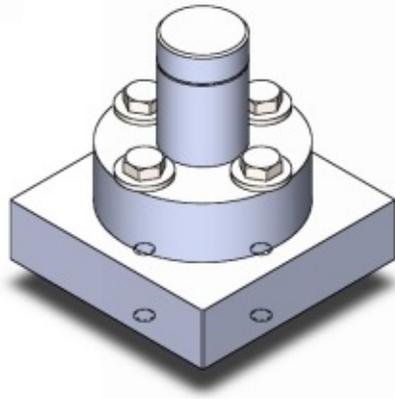
40. Click **OK** on the PropertyManager.



Patterning components in an assembly

1. On the CommandManager, click **Assembly > Linear Pattern > Pattern Driven Component Pattern** .
2. Select the subassembly from the FeatureManager Design Tree.

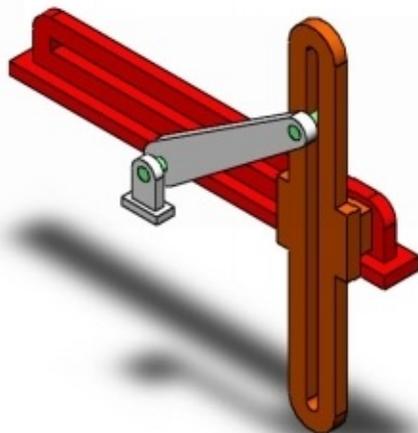
3. On the **Pattern Driven** PropertyManager, click in the **Driving Feature or Component** box.
4. Select the CircularPattern of the first component from the FeatureManager Design Tree.
5. Click **OK**  on the PropertyManager.



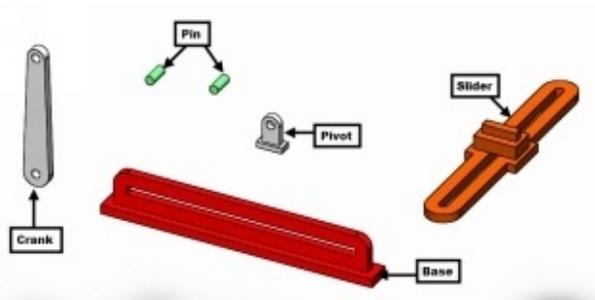
6. Click the **Save**  icon on the Quick Access Toolbar.
7. Click **Save All** on the **Save Modified Documents** dialog.
8. Type Ch8_tutorial1 in the **File name** box and click **Save**.
9. Select **Save internally(inside the assembly)**, and then click **OK**.

TUTORIAL 2

In this tutorial, you create a slider crank mechanism by applying advanced mates.

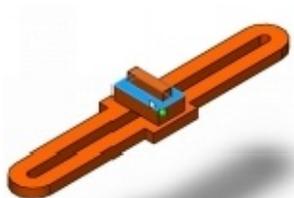


1. Create **Slider Crank Assembly** folder.
2. Download the part files of the assembly from the companion website. Next, save the files in the **Slider Crank Assembly** folder.
3. Start a new assembly file.
4. Click the pin  icon on the **Begin Assembly** PropertyManager. This keeps the PropertyManager active.
5. Click the **Browse** button on the PropertyManager.
6. Browse to the **Slider Crank Assembly** folder, select all the components, and click **Open**.
7. On the PropertyManager, in the **Part/Assembly to Insert** section, click **Base**.
8. Click in the assembly window to position the Base component.
9. Likewise, position the other components in the assembly window.
10. Click **OK** on the PropertyManager.

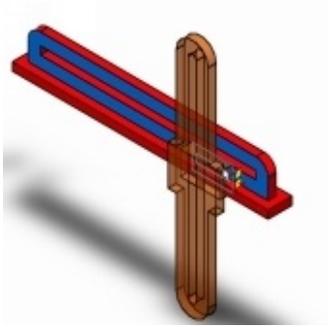


Creating Smart Mates

1. Press the Alt key and click on the face of the slider, as shown.



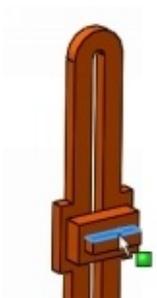
2. Keep the Alt key pressed and drag the component onto the face of the Base as shown.



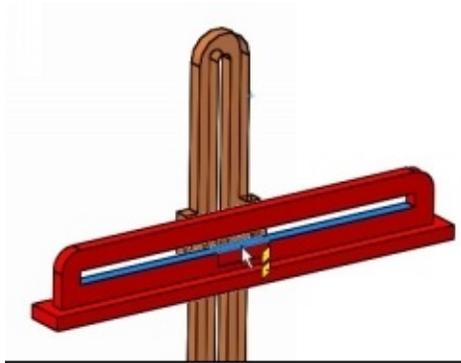
3. Release the left mouse button and notice that the Mates toolbar appears with the **Coincident**  icon selected.
4. Click **Add/Finish Mate** to create the **Coincident** mate between the two faces.
5. Drag the slider up and rotate the view as shown.



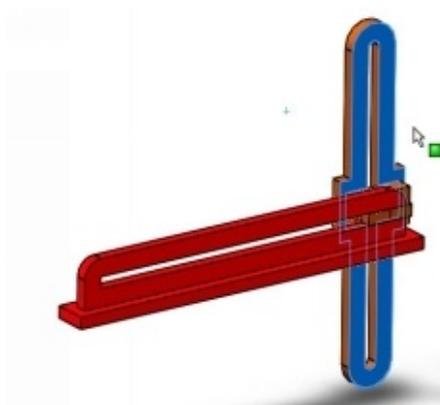
6. Click the face of the slider.



7. Press the Alt key and drag the slider onto the face of the Base as shown.

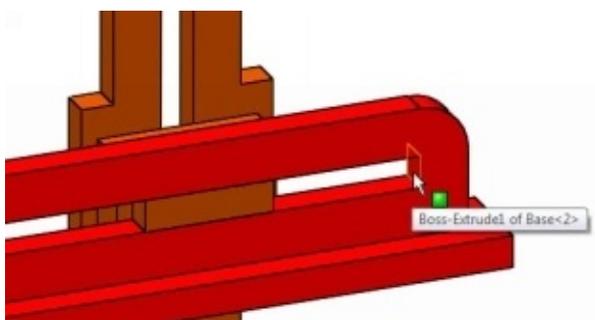


8. Click the **Flip Mate Alignment** icon.
9. Click **Add/Finish Mate**.
10. Click and drag the slider and notice that it can intersect with the base.

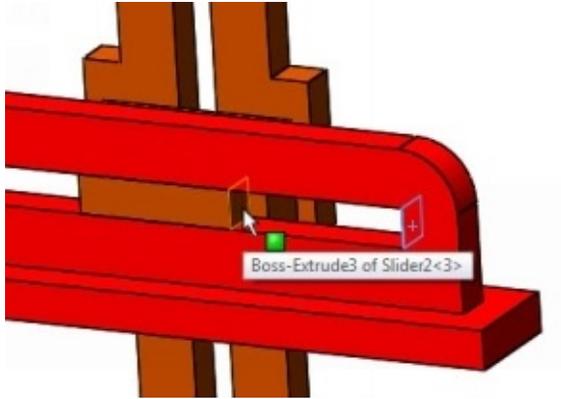


Creating the Limit Mate

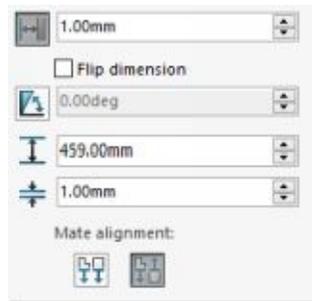
1. On the CommandManager, click **Assembly > Mate** .
2. On the PropertyManager, expand the **Advanced Mates** section and click the **Distance**  icon.
3. Select the end face of the Base as shown.



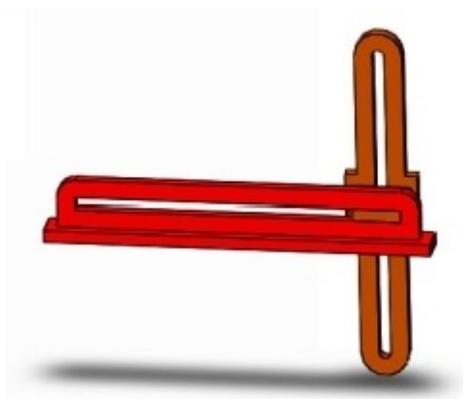
4. Rotate the view and select the end face of the slider as shown.

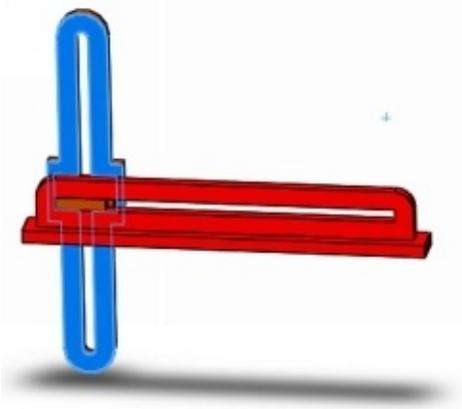


5. Set the **Distance** value to 1 mm.
6. Set the **Maximum Value** and **Minimum Value** to 459 and 1, respectively.



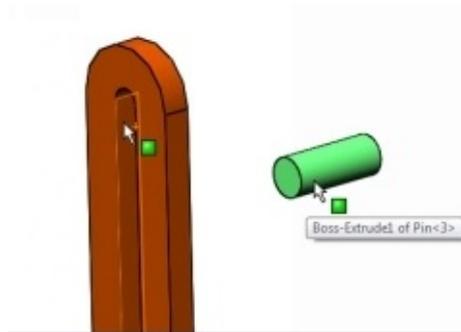
7. Click **OK** twice on the PropertyManager.
8. Click and drag the slider and notice that it is constrained between the end faces of the slot.



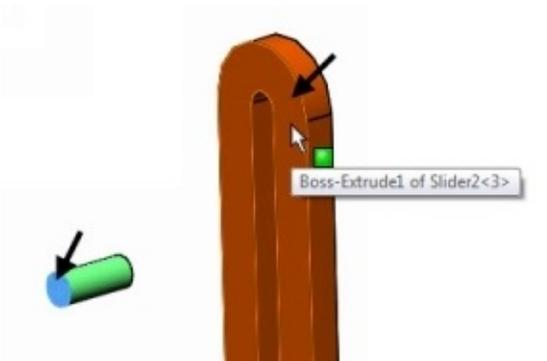


Creating the Slot Mate

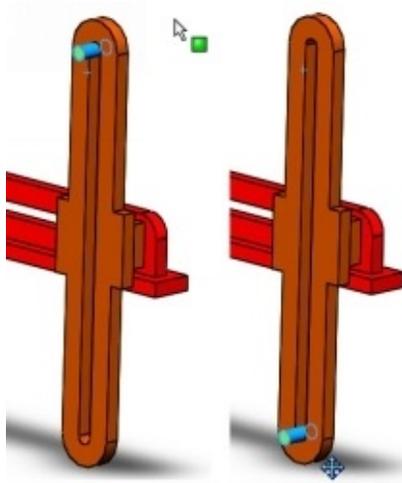
1. On the CommandManager, click **Assembly** > **Mate** .
2. On the PropertyManager, expand the **Mechanical Mates** section and click the **Slot**  icon.
3. Set the **Constraint** to **Free**.
4. Select the cylindrical face of the pin and flat face of the slider slot as shown.



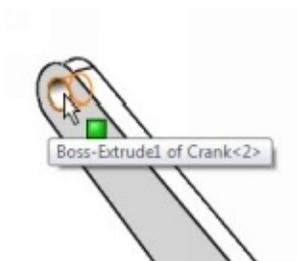
5. Click **OK** on the PropertyManager.
6. Expand the **Standard Mates** section and click the **Coincident**  icon.
7. Select the flat faces of the pin and slider as shown.



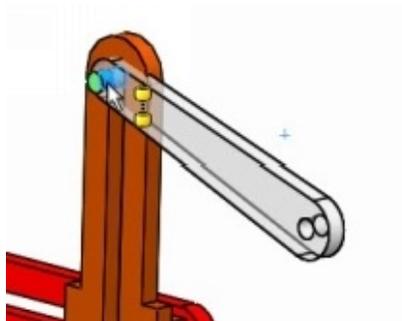
8. Click **Add/Finish Mate**, and then click **OK**.
9. Click and drag the pin and notice that it is constrained along the length of the slot.



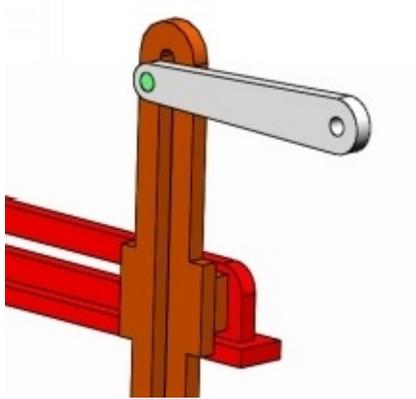
10. Press the **Alt** key, click and drag the upper hole of the crank.



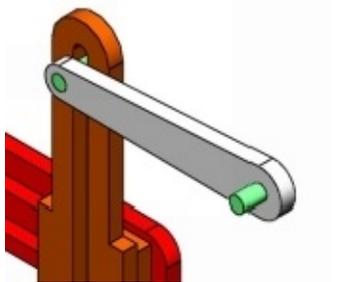
11. Release the pointer onto the cylindrical of the pin.



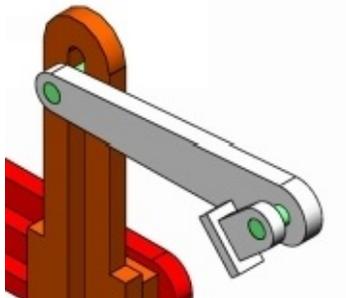
12. Click **Add/Finish Mate** to add the **Concentric** mate between the hole and pin.
13. Use the **Mate** command to add the **Coincident** mate between the end face of the pin and flat face of the crank.



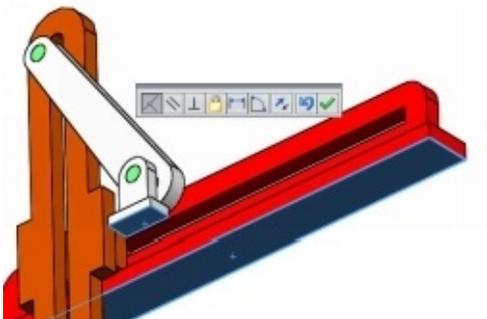
14. Assemble the other instance of the pin with the crank.



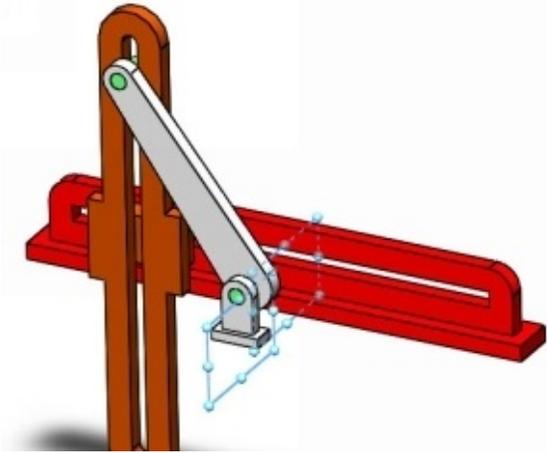
15. Assemble the pivot with the pin.



16. Make the bottom faces of the pivot and base coincident to each other.



17. Make the Right planes of the pivot and base coincident.



18. Click and drag the crank and notice that the slider moves along the slot.
19. Save and close the assembly and its parts.

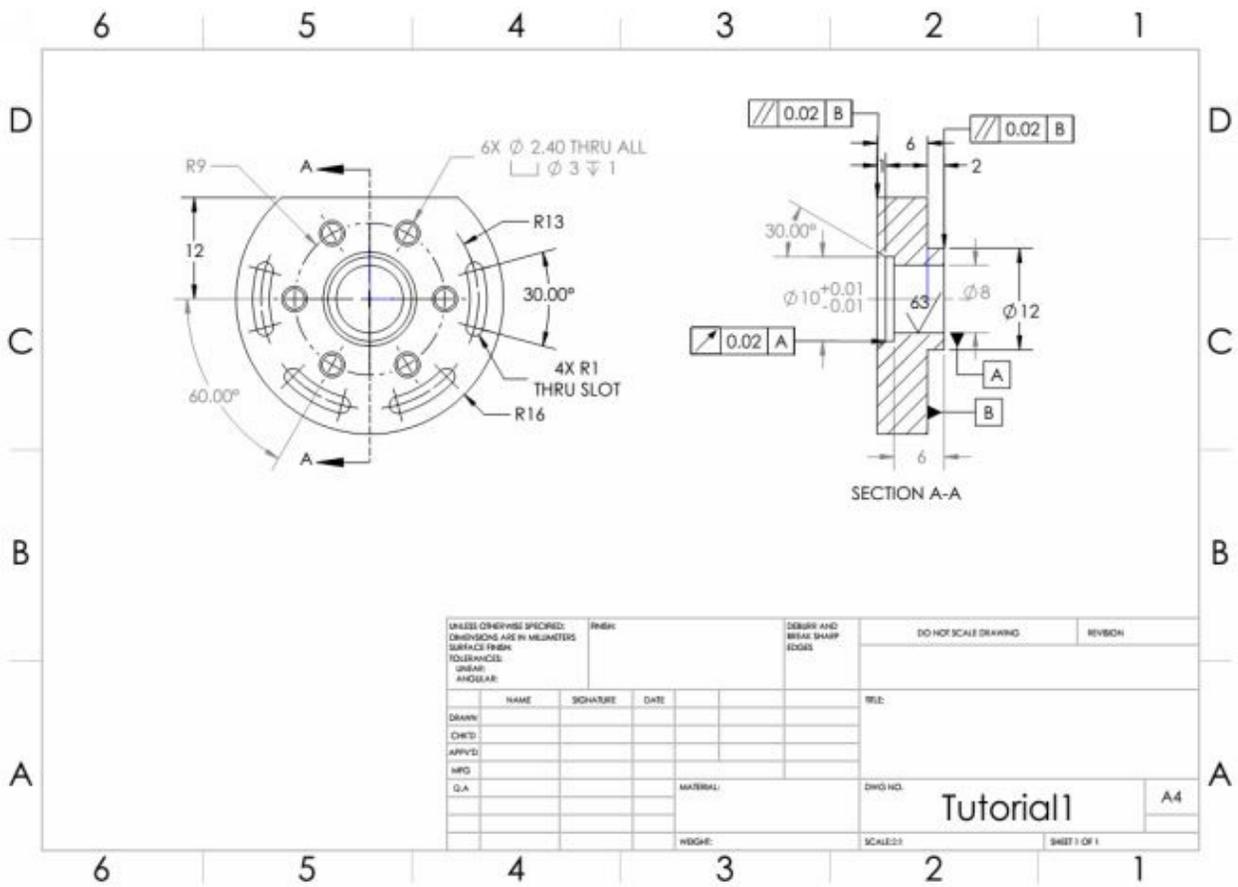
Chapter 9: Dimensions and Annotations

In this chapter, you will learn to

- Create Centerlines and Center Marks
- Edit Hatch Pattern
- Apply Dimensions
- Place Hole callouts
- Place Datum Feature
- Place Feature control frame
- Place Surface symbol

TUTORIAL 1

In this tutorial, you create the drawing shown below.



UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:				FINISH:	DEBURR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION
DRAWN:	NAME	SIGNATURE	DATE			TITL:	
CHECKED:							
APPROVED:							
MFG:							
Q.A.					MATERIAL:	DWG NO.	A4
						Tutorial1	
					NOTE:	SCALE:1:1	SHEET 1 OF 1

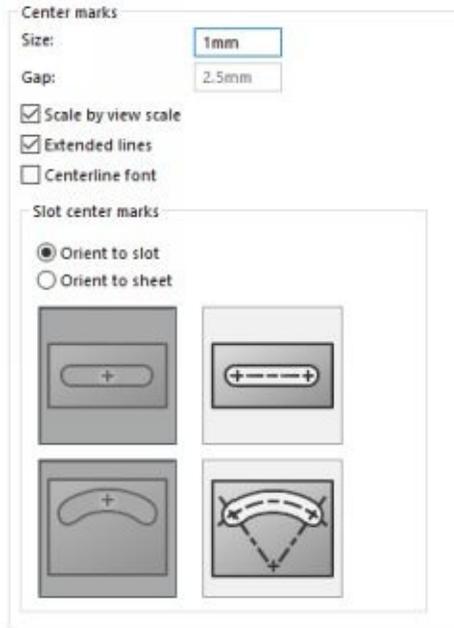
1. Download the Adapter Plate file from the Companion website.
2. Open a new drawing file using the **Sample** template.
3. Click **Cancel**  on the **Model View** PropertyManager.
4. In the FeatureManager Design Tree, click the right mouse on Sheet1 and select **Properties**.
5. On the **Sheet Properties** dialog, select **Sheet/Format Size > A4 (ANSI) Landscape**, and then click **OK**.

Creating a Model View with Center Marks

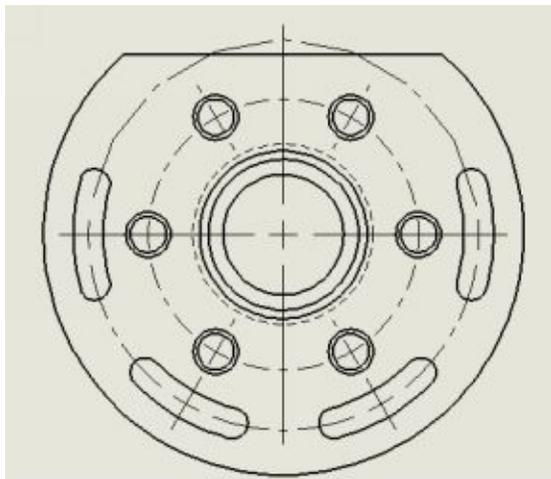
1. On the **Quick Access Toolbar**, click the **Options**  icon.
2. Click the **Document Properties** tab on the dialog and select **Detailing** from the tree.
3. Check the options in the **Auto insert on view creation** section, as shown.



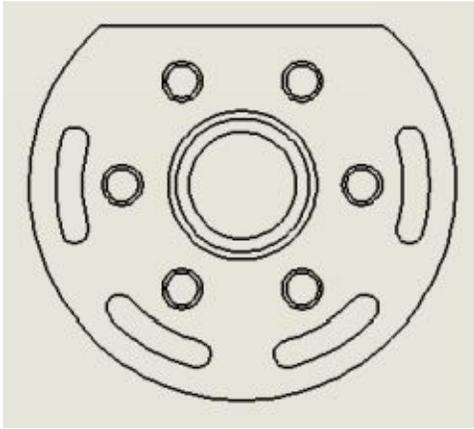
4. Select **Centerlines/Center Marks** in the tree.
5. Under the **Slot center marks** section, select **Orient to slot** and click the icons, as shown.
6. Type-in 1 in the **Size** box.
7. Check the **Scale by view scale** option to change the gap in the centreline with the view scale.



8. Click **OK**.
9. Click **View Layout > Model View**  on the CommandManager.
10. Click **Browse** button on the PropertyManager.
11. Browse to the location of the Adapter Plate and double-click on it.
12. Set the **Scale** to **Use sheet scale**.
13. Set the **Orientation** to **Front** .
14. Set the **Display Style** to **Hidden Line Removed** .
15. Place the front view on the left-side on the drawing sheet.
16. Click **OK** .



17. Select the view and press **Delete**.
18. Click **Yes** on the **Confirm Delete** dialog.
19. On the **Quick Access Toolbar**, click the **Options**  icon.
20. Click the **Document Properties** tab on the dialog and select **Detailing** from the tree.
21. Uncheck the options in the **Auto insert on view creation** section and click **OK**.
22. Create the front view and notice the difference.

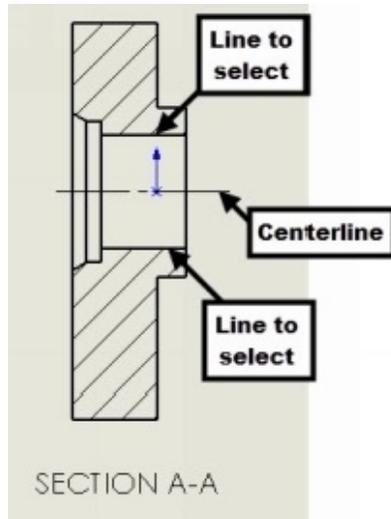


23. Click **View Layout > Section View**  on the CommandManager.
24. Select the center point of the front view.
25. Click **OK**.
26. Place the section view on the left side.

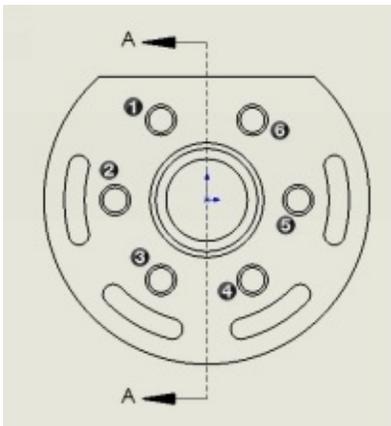


Creating Centerlines and Center Marks

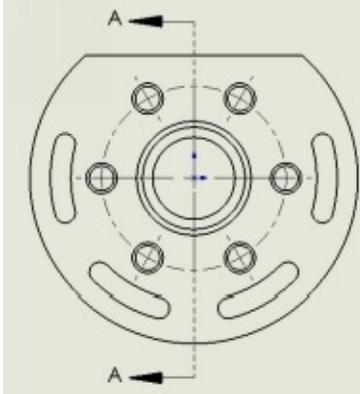
1. Click **Annotation > Centerline**  on the CommandManager.
2. Select the parallel lines on the section view, as shown below. The centerline is created.



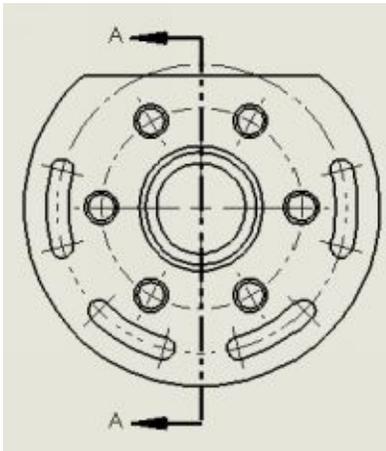
3. Click **OK**  on the PropertyManager.
4. Click **Annotation > Center Mark**  on the CommandManager.
5. On the PropertyManager, under **Manual Insert Options** section, click the **Circular Center Mark**  icon.
6. Select the outer circles of the counterbore holes



7. Click **OK**  on the PropertyManager.

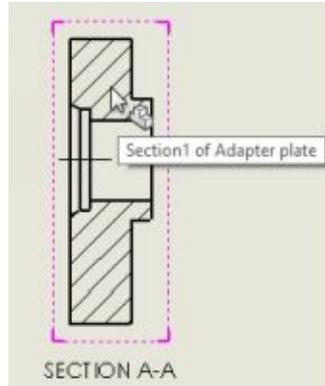


8. Activate the **Center Mark**  command.
9. On the PropertyManager, under the **Auto Insert** section, check **For all slots**.
10. Check the **Circular lines** option and uncheck all other options.
11. Click the **Arc Slot Ends**  icon.
12. Deactivate the **Slot Centers**  icon.
13. Select the front view and click **OK**.



Editing the Hatch Pattern

1. Click on the hatch pattern of the section view. The **Area Hatch/Fill** PropertyManager appears.



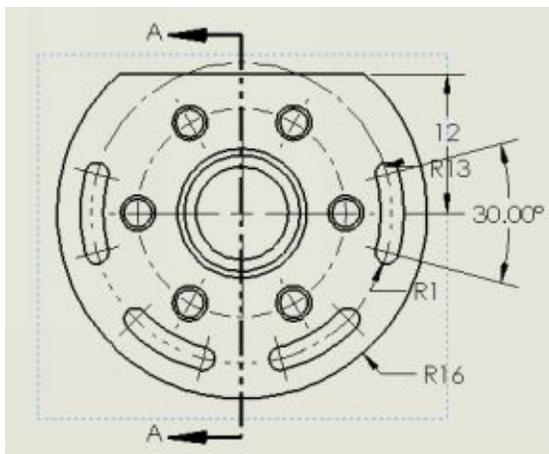
2. Uncheck the **Material crosshatch** option.

You can select the required hatch pattern from the **Hatch Pattern** drop-down. You can adjust the hatch scale and angle.

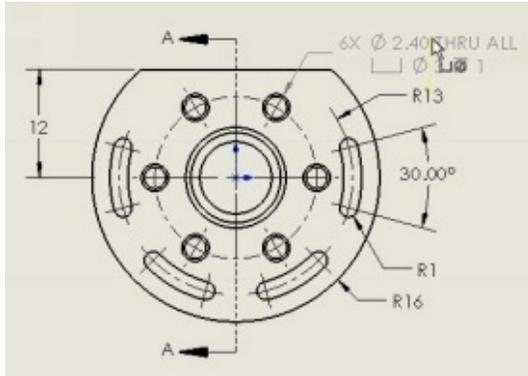
3. Click **OK** ✓.

Applying Dimensions

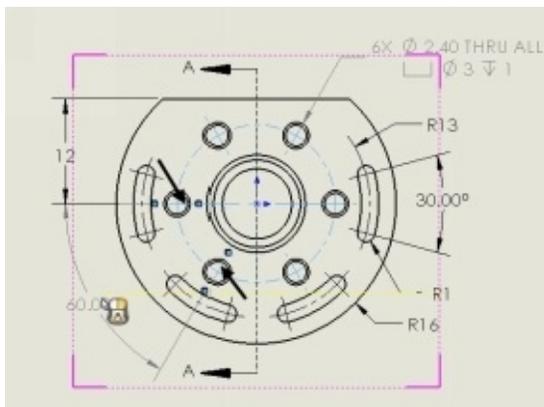
1. Click **Annotation > Model Items**  on the CommandManager.
2. Select **Source > Entire model** and uncheck the **Import items into all views** option on the PropertyManager.
3. Click on the front view.
4. Click **OK**.
5. Drag and arrange the dimensions as shown.



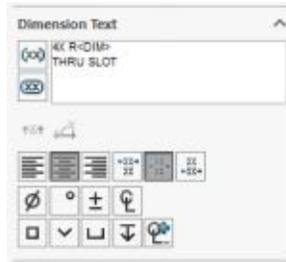
6. On the CommandManager, click **Annotation > Hole Callout** .
7. Select the counterbore hole and position the hole callout as shown.



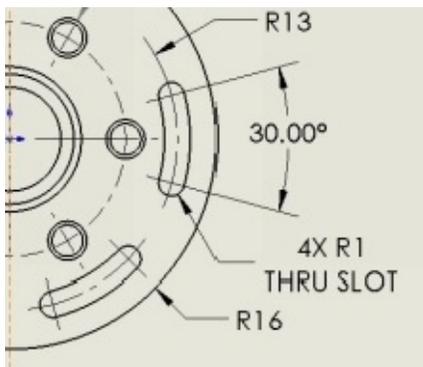
8. On the CommandManager, click **Annotation > Smart Dimension** .
9. Select the center marks of the holes.
10. Move the pointer in the region between the holes and click to position the dimension.



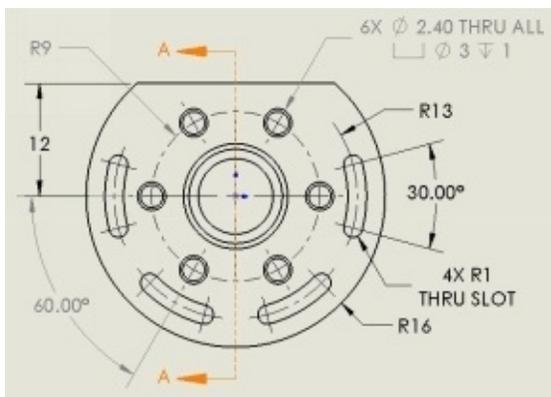
11. Select the R1 dimension and type-in the text in the **Dimension Text** box on the PropertyManager.



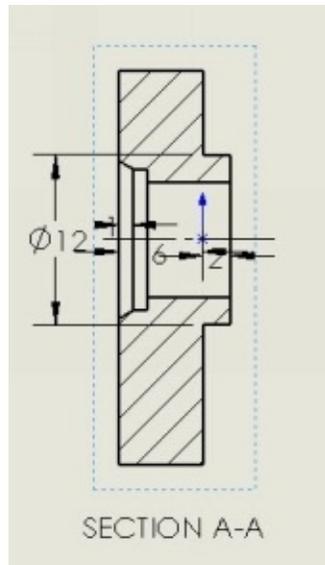
- Click **Close Dialog** on the PropertyManager and position the dimension properly.



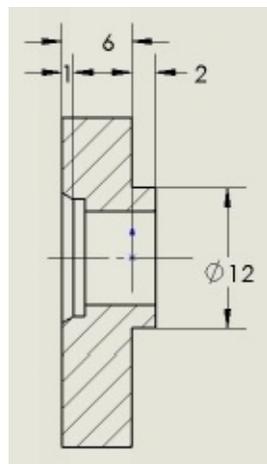
- Dimension the pitch circle diameter of the counterbore holes as shown.



- On the CommandManager, click **Annotation > Model Items**.
- Select **Source > Entire model** and uncheck the **Import items into all views** option on the PropertyManager.
- Click on the section view. Click **OK**.

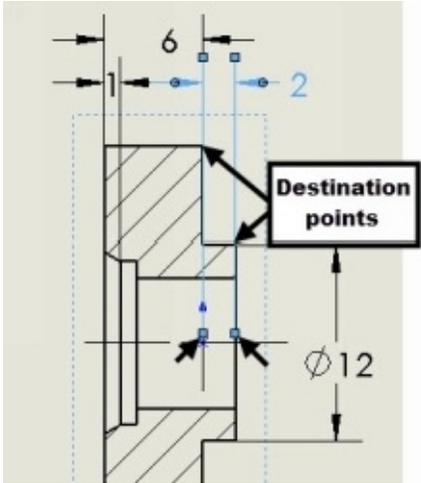


17. Click and drag the dimensions as shown.

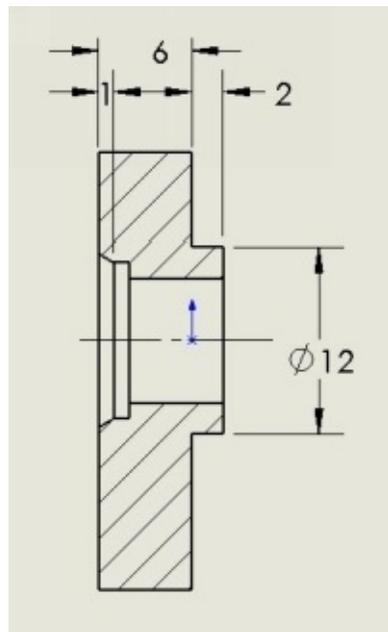


18. Select the 2 mm dimension and notice the highlighted origins.

19. Click and drag the endpoints to the new locations as shown.

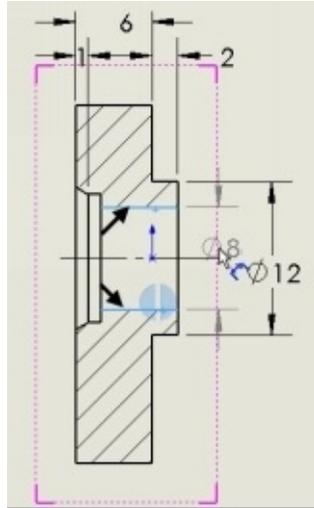


20. Likewise, modify the origins of other dimensions as shown.

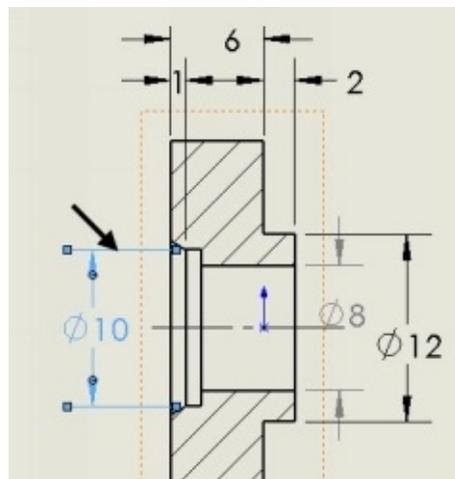


21. Select the lines, as shown.

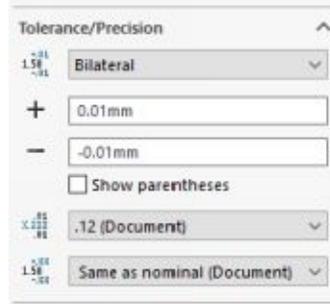
22. Move the pointer toward right and click to place the dimension.



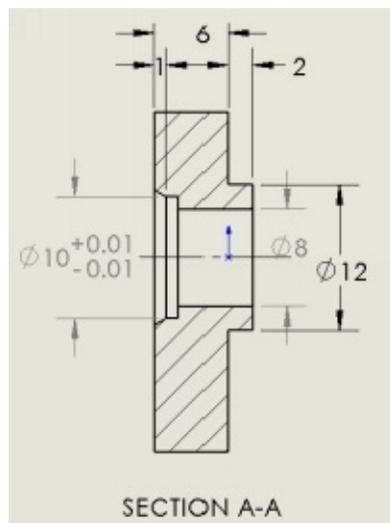
23. Likewise, add the counterbore dimension.



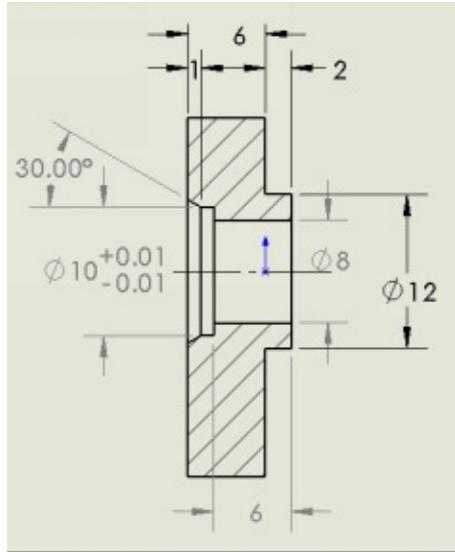
24. Select the counterbore dimension.
25. On the PropertyManager, under the **Tolerance/Precision** section, set the **Tolerance Type** to **Bilateral**.
26. Set the tolerance parameters as shown.



27. Click **Close Dialog**.

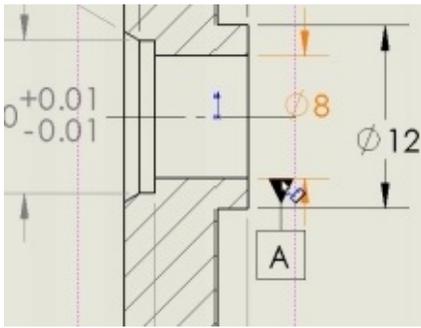


28. Likewise, apply the other dimensions, as shown below.

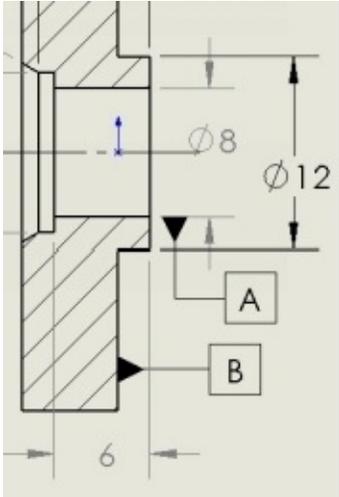


Placing the Datum Feature

1. Click **Annotation > Datum Feature**  on the CommandManager.
2. Select the extension line of the dimension, as shown below.

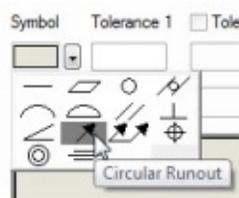


3. On the PropertyManager, click the **Filled Triangle with Shoulder**  icon.
4. Move the cursor downward and click.
5. Likewise, place a datum feature B, as shown below. Click **OK**.

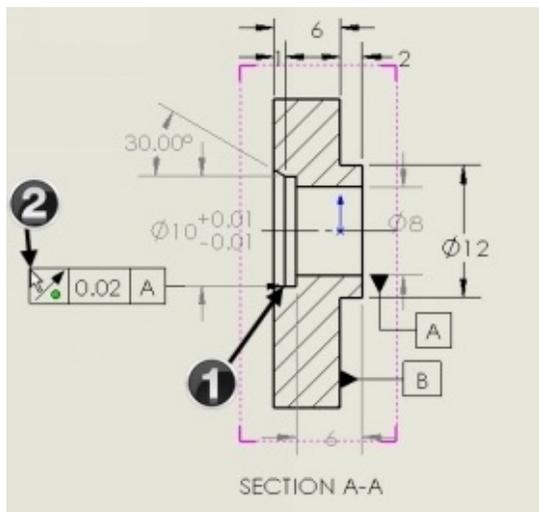


Placing the Feature Control Frame

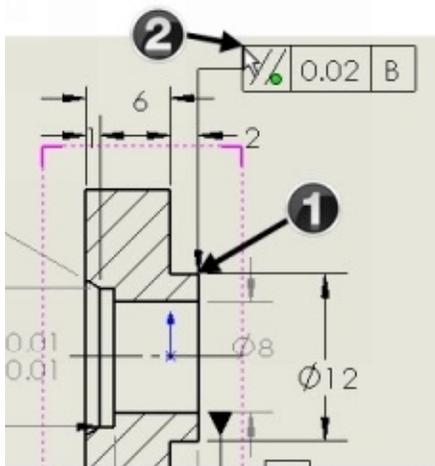
1. Click **Annotation** > **Geometric Tolerance**  on the CommandManager.
2. On the dialog, select **Circular Runout** from the **Symbol** drop-down.



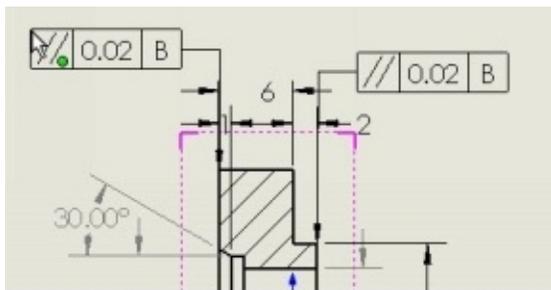
3. Type-in **0.02** and **A** in the **Tolerance 1** and **Primary** boxes.
4. Select a point on the line, as shown.
5. Move the cursor horizontally toward right and click.
6. Click **OK**.



7. Activate the **Geometric Tolerance** command.
8. On the dialog, select **Circular Runout** from the **Symbol** drop-down.
9. Type-in **0.02** and **B** in the **Tolerance 1** and **Primary** boxes.
10. Select the endpoint of the edge as shown.
11. Move the pointer vertically up and click to position the feature control frame.

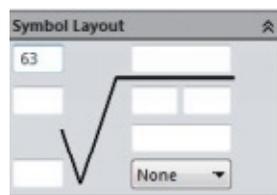


12. Likewise, add another feature control frame and click **OK**.

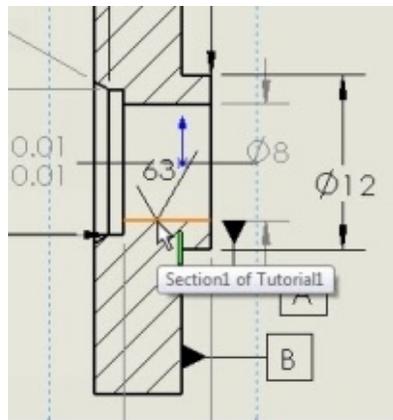


Placing the Surface Texture Symbols

1. Click **Annotation > Surface Symbol**  on the CommandManager.
2. Set the **Maximum Roughness** value to 63 on the PropertyManager.



3. Click on the inner cylindrical face of the hole, as shown below.

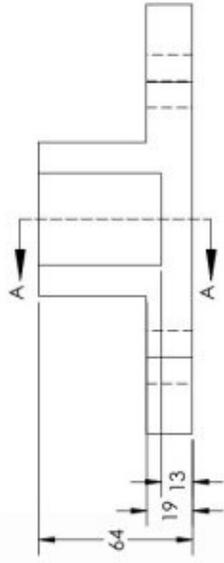
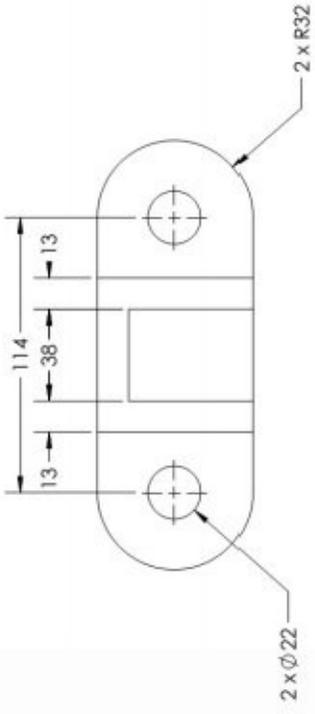
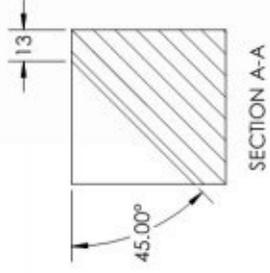


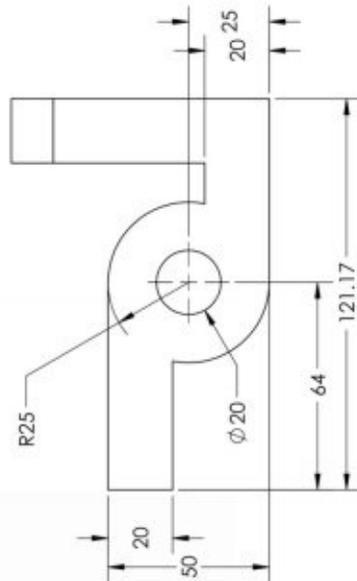
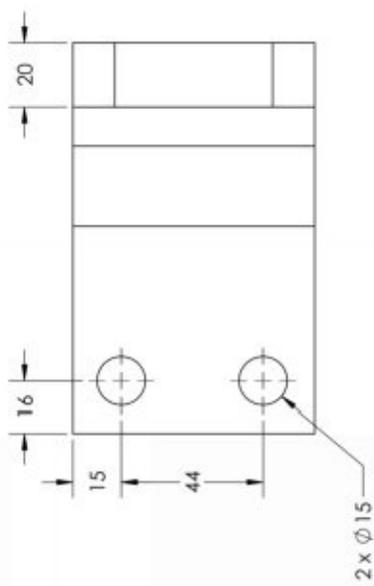
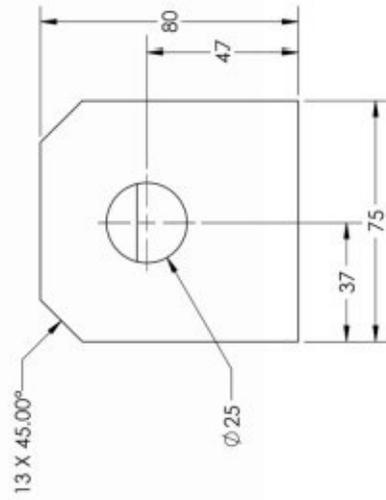
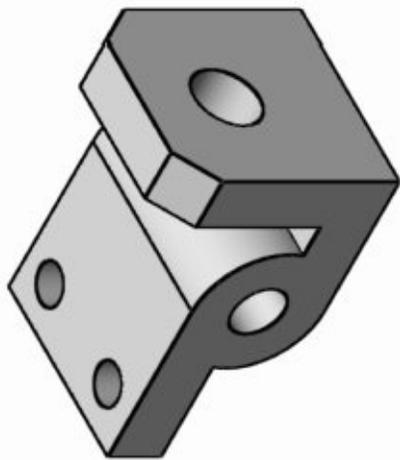
4. Click **OK**.

5. Save and close the file.

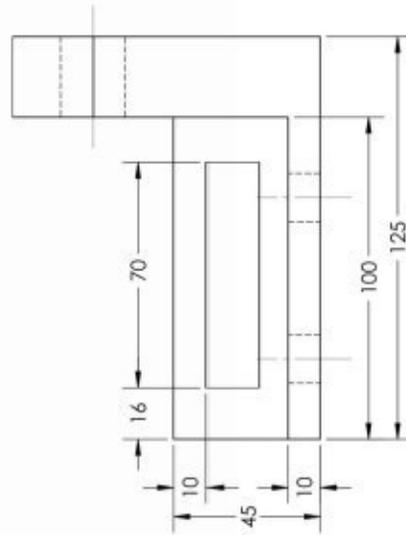
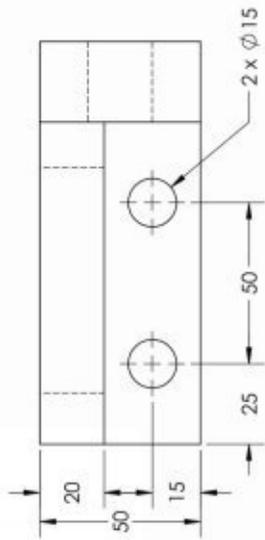
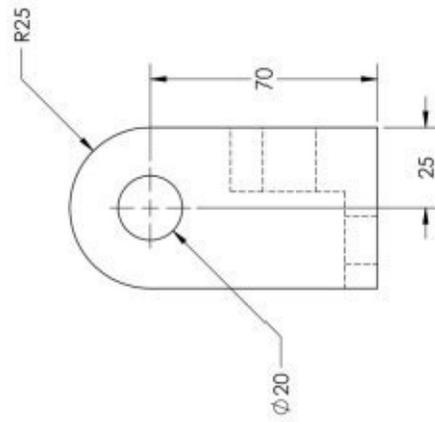
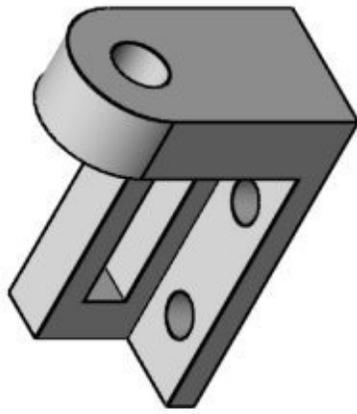
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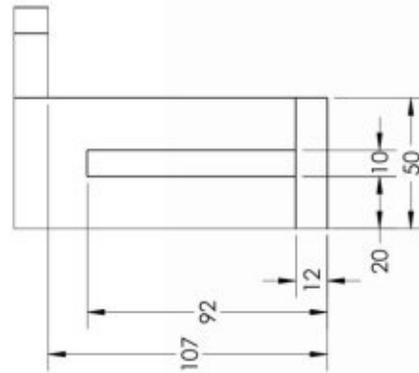
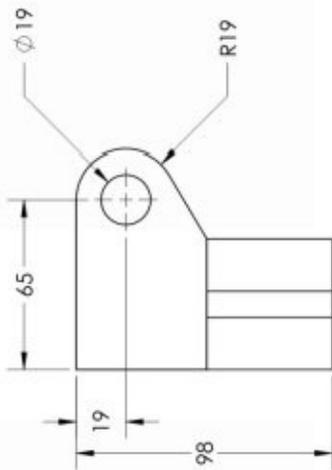
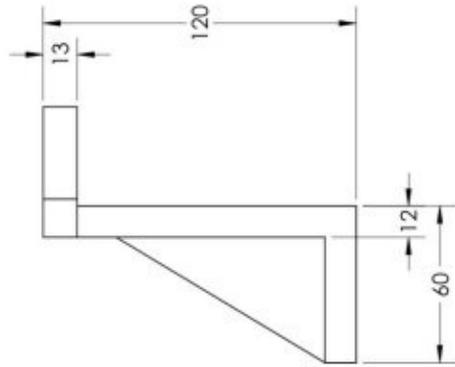
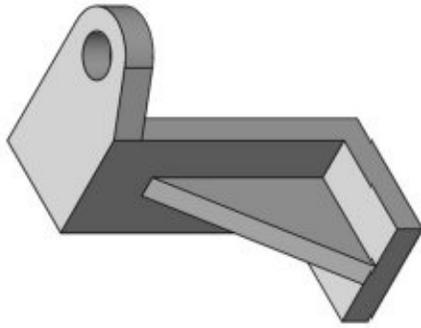


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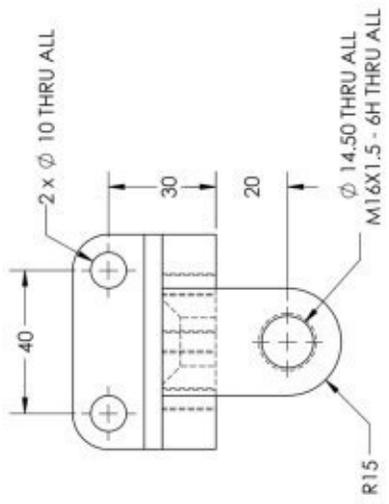
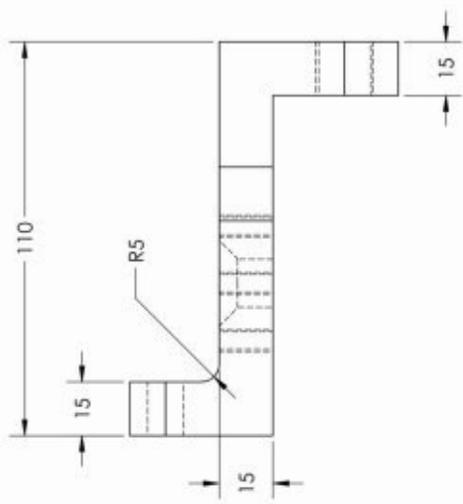
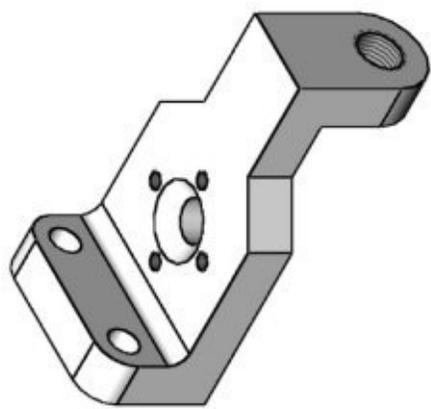
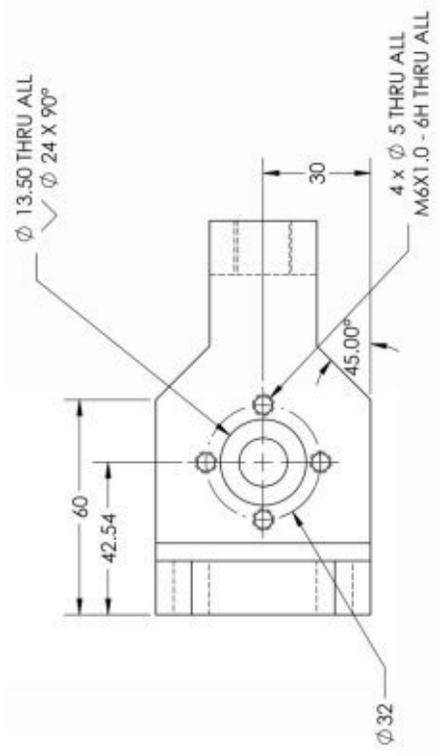


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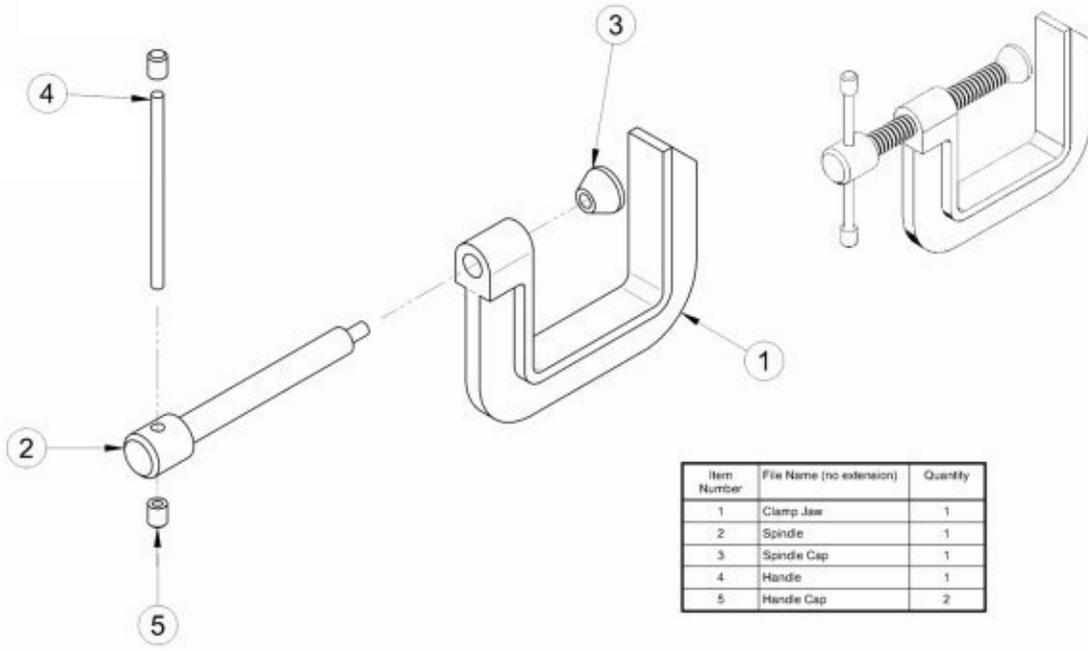
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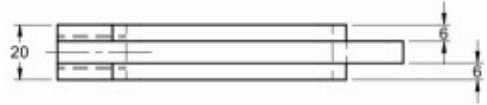
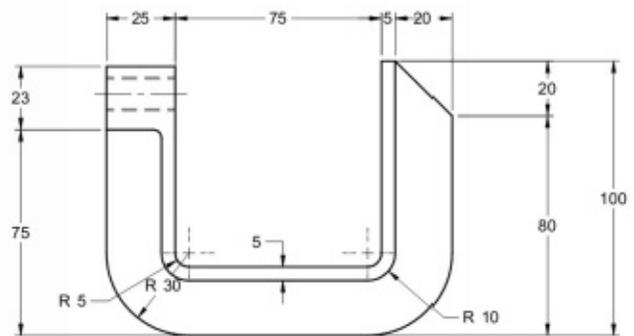
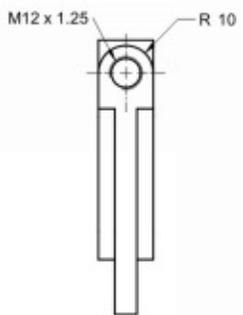
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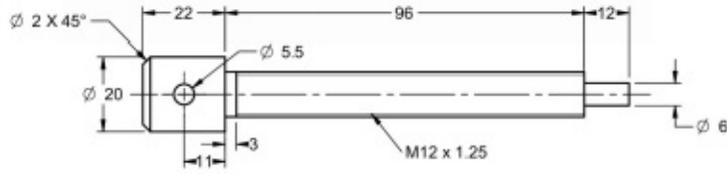
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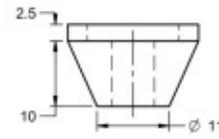
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3	Spindle Cap	1
4	Handle	1
5	Handle Cap	2



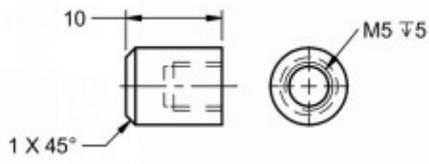
CLAMP JAW



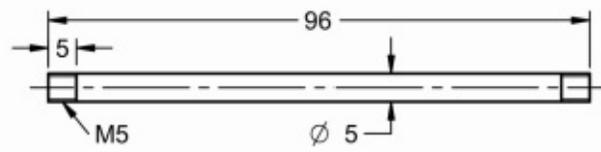
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SPINDLE CAP



HANDLE CAP



HANDLE

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