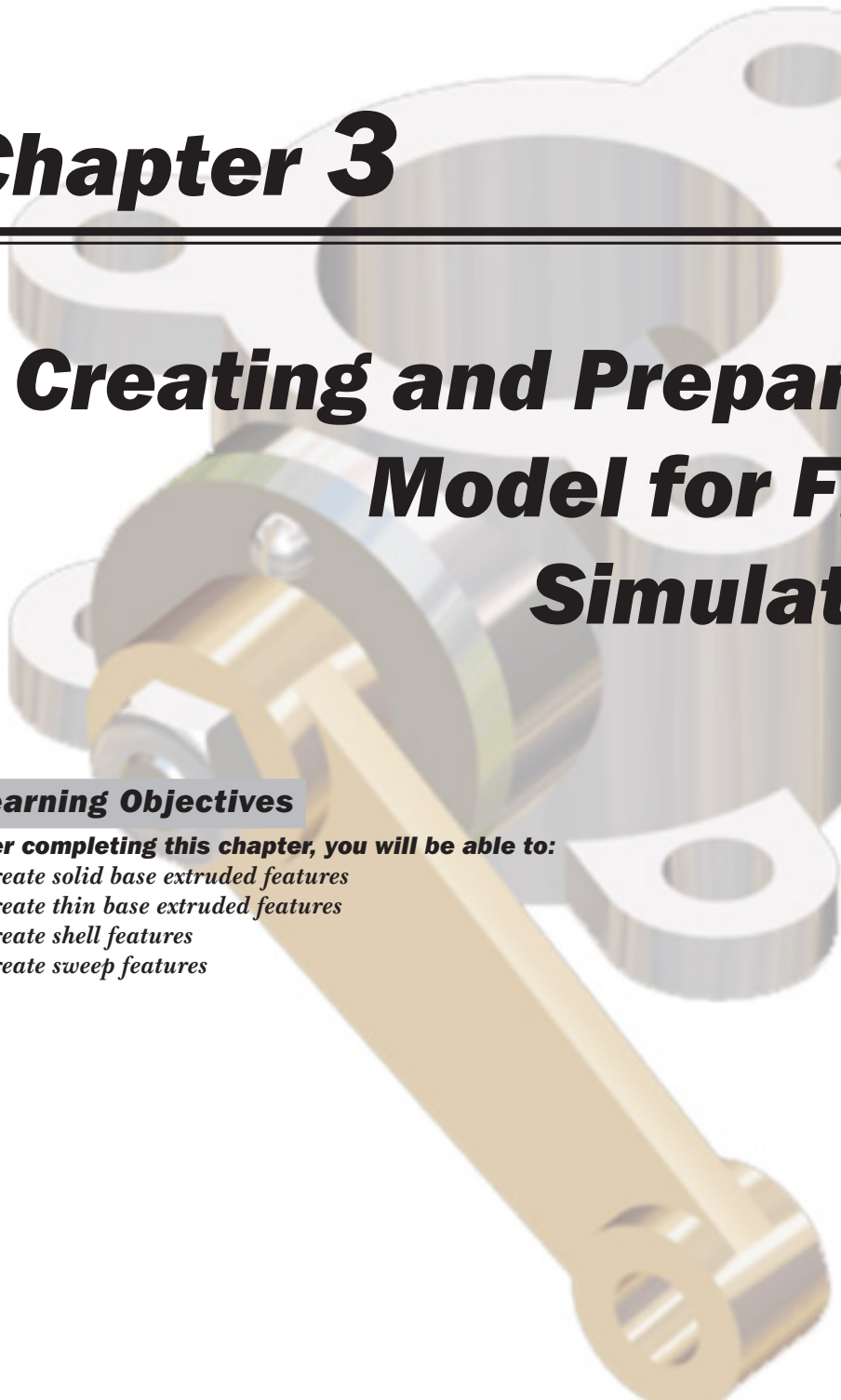
Chapter 3

Creating and Preparing Model for Flow Simulation



Learning Objectives

After completing this chapter, you will be able to:

- *Create solid base extruded features*
- *Create thin base extruded features*
- *Create shell features*
- *Create sweep features*

BASIC TOOLS TO CREATE A MODEL

In SOLIDWORKS, there are various tools that can be used to create a model. Some of the tools which are needed to create a model are discussed in this chapter. For more tools in SOLIDWORKS, you can refer to SOLIDWORKS 2025 for Designers, 22nd Edition authored by Prof. Sham Tickoo, Purdue University Northwest, USA. In this chapter, you will learn about the usage of the **Extrude** tool, which is used to extrude a sketch; the **Shell** tool, which is used to scoop out material from a model to make it hollow; and the **Sweep** tool, which is used to create a swept surface along a closed or open profile.

CREATING BASE FEATURES BY EXTRUDING SKETCHES

CommandManager:	Features > Extruded Boss/Base
SOLIDWORKS Menus:	Insert > Boss/Base > Extrude
Toolbar:	Features > Extruded Boss/Base > Extruded Boss/Base



The sketches that you have drawn can be converted into base features by extruding the sketch using the **Extruded Boss/Base** tool from the **Features CommandManager**. After drawing the sketch, choose the **Features** tab from the **CommandManager** to display the **Features CommandManager**. Next, choose the **Extruded Boss/Base** tool from the **Features CommandManager**; the sketching environment will be closed and the part modeling environment will be invoked. Also, the preview of the feature that is created using the default options will be displayed in the trimetric view. The trimetric view gives a better display of the solid feature.

On the basis of the options and the sketch selected for extruding, the resulting feature can be a solid feature or a thin feature. If the sketch is closed, it can be converted into a solid feature or a thin feature. However, if the sketch is open, it will be converted into a thin feature. The process of creating a solid or thin feature is discussed next.

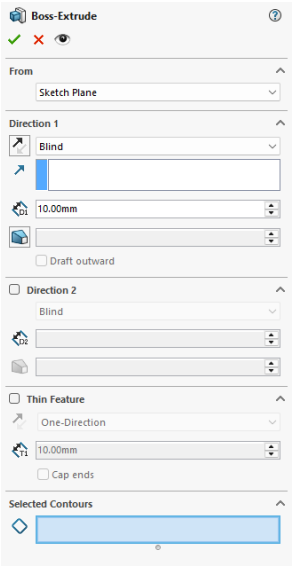


Figure 3-1** The **Boss-Extrude PropertyManager

Creating Solid Extruded Features

After you have completed drawing a closed sketch, dimension it to convert it into a fully defined sketch. Next, choose the **Features** tab from the **CommandManager**; the **Features CommandManager** will be displayed. Choose the **Extruded Boss/Base** tool; the **Boss-Extrude PropertyManager** will be displayed, refer to Figure 3-1. Also, you will notice that the view is automatically changed to the trimetric view.

You will also notice that the preview of the base feature is displayed in temporary graphics. Additionally, an arrow will appear in front of the sketch. Note that if the sketch consists of some closed loops inside the outer loop, they will automatically be subtracted from the outer loop while extruding, as shown in Figure 3-2.

The options in the **Boss-Extrude PropertyManager** are discussed next.

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Direction 1

The **Direction 1** rollout is used to specify the end condition for extruding the sketch in one direction from the sketch plane. The options in the **Direction 1** rollout list are discussed next.

End Condition

The **End Condition** drop-down list provides options to define the termination of the extruded feature. Note that when you create the first feature, some of the options in this drop-down list will not be used. Also, some additional options will be available later in this drop-down list. Some of the options that will be used to define the termination of the base feature are discussed next.

Blind

The **Blind** option is selected by default and is used to define the termination of the extruded base feature by specifying the depth of extrusion. The depth of the extrusion is specified in the **Depth** spinner. This spinner will be displayed in the **Direction 1** rollout on selecting the **Blind** option. To reverse the extrusion direction, choose the **Reverse Direction** button provided on the left of this drop-down list. Figure 3-2 shows the preview of the feature being created by extruding the sketch using this option.

You can also extrude a sketch to a blind depth by dragging the feature dynamically using the mouse. To do so, move the cursor to the arrow displayed in the preview; the move cursor will be displayed and the color of the arrow will also change. Left-click once on the arrow; a scale will be displayed, as shown in Figure 3-3. Now, move the cursor to specify the depth of extrusion; the value of the depth of extrusion will change dynamically on this scale as you move the cursor. Left-click again to specify the termination of the extruded feature; the select cursor will be replaced by the mouse cursor. Right-click and choose **OK** to complete the feature creation or choose the **OK** button from the **Boss-Extrude PropertyManager**.

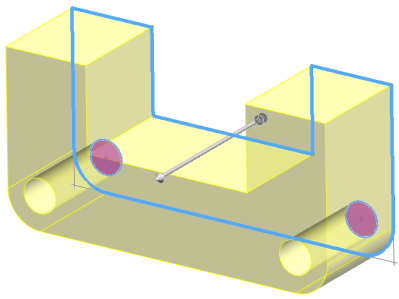


Figure 3-2 *Preview of the feature being extruded using the* ***Blind*** *option*

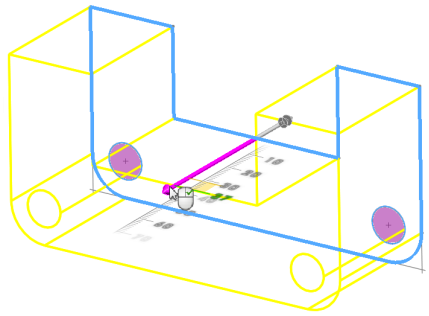


Figure 3-3 *Preview of the feature being extruded by dragging the arrow dynamically*

Mid Plane

The **Mid Plane** option is used to create the base feature by extruding the sketch equally in both the directions of the plane on which the sketch is drawn. For example, if the total depth of the extruded feature is 30 mm, it will be extruded 15 mm toward the front of the sketching plane and 15 mm toward the back. The depth of the feature can be defined in the **Depth** spinner that is displayed below the **Direction of Extrusion** selection box. Figure 3-4 shows preview of the feature being created by extruding the sketch using the **Mid Plane** option.

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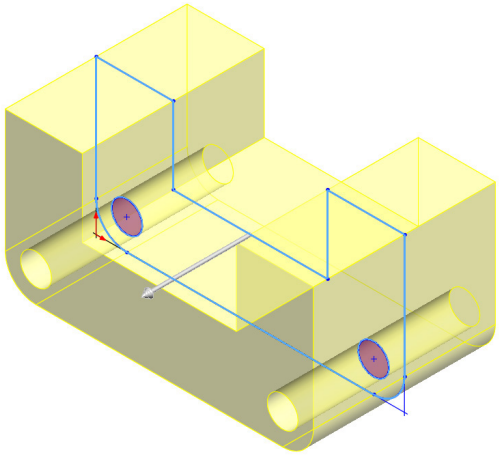


Figure 3-4 *Preview of the feature being extruded using the* ***Mid Plane*** *option*

Draft On/Off

The Draft On/Off button is used to specify a draft angle while extruding a sketch to taper the resulting feature. To add a draft angle to the feature, choose this button; the Draft Angle spinner and the Draft outward check box will be available. You can enter the draft angle for the feature in the Draft Angle spinner. By default, the feature will be tapered inward, as shown in Figure 3-5.

If you want to taper the feature outward, select the Draft outward check box that is displayed below the Draft Angle spinner. The feature created with the outward draft is shown in Figure 3-6.



Note

The ***Direction of Extrusion*** *area will be discussed in the later chapters.*

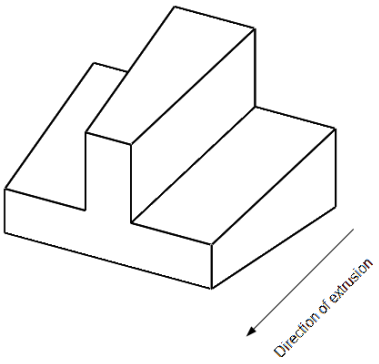


Figure 3-5 *Feature created with inward draft*

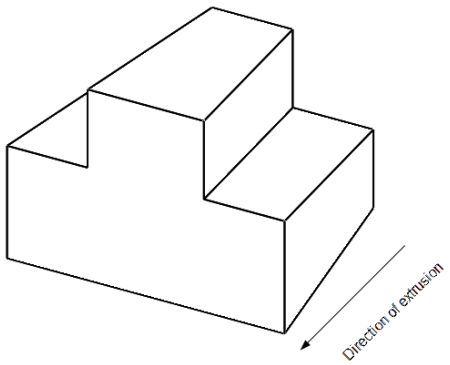


Figure 3-6 *Feature created with outward draft*

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Direction 2

The **Direction 2** rollout is used to extrude a sketch with different values in the second direction of the sketching plane. The **Direction 2** rollout will be activated only when you select the **Direction 2** check box.

The options in this rollout are similar to those in the **Direction 1** rollout. Note that unlike the **Mid Plane** termination option, the depth of extrusion and other parameters in both directions can be different. For example, you can extrude the sketch to a blind depth of 20 mm and an inward draft of 15-degree in front of the sketching plane, and to a blind depth of 15 mm and an outward draft of 0-degree behind the sketching plane, as shown in Figure 3-7.

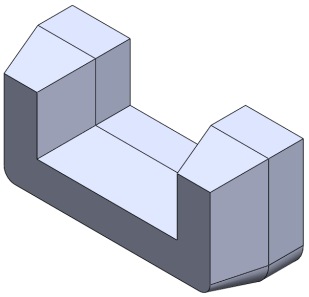


Figure 3-7 *Feature created in two directions with different values*

After setting the values for both directions, choose the **OK** button or choose the **OK** icon from the confirmation corner; the feature will be created with defined values.

Creating Thin Extruded Features

The thin extruded features can be created using a closed or an open sketch. If the sketch is closed, the thickness will be specified inside or outside the sketch to create a cavity inside the feature, as shown in Figure 3-8. To convert a closed sketch into a thin feature, select the check box in the **Thin Feature** rollout title bar; the rollout will expand, as shown in Figure 3-9.

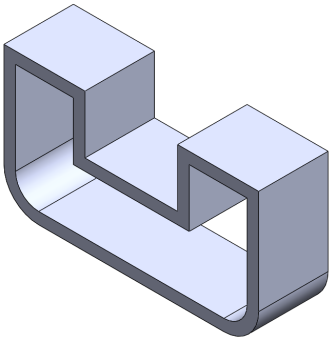


Figure 3-8 *Thin extruded feature created using a closed loop*

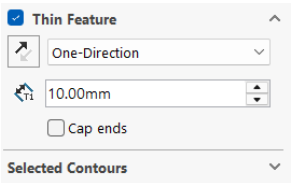


Figure 3-9 *Feature created in two directions with different values*

The options in the **Thin Feature** rollout of the **Boss-Extrude PropertyManager** are discussed next.

Type

The options provided in the **Type** drop-down list are used to select the method of defining the thickness of the thin feature. These options are discussed next.

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One-Direction

The **One-Direction** option is used to add thickness on one side of the sketch. The amount of thickness to be applied can be specified in the **Thickness** spinner provided below the **Type** drop-down list. For the closed sketches, the direction can be inside or outside the sketch. Similarly, for open sketches, the direction can be below or above the sketch. You can reverse the direction of thickness using the **Reverse Direction** button available on the left of the **Type** drop-down list. This button will be available only when you select the **One-Direction** option from this drop-down list.

Mid-Plane

The **Mid-Plane** option is used to add thickness equally on both sides of the sketch. The value of the thickness of the thin feature can be specified in the **Thickness** spinner provided below this drop-down list.

Two-Direction

The **Two-Direction** option is used to create a thin feature by adding different thicknesses on both sides of a sketch. The thickness values in direction 1 and direction 2 can be specified in the **Direction 1 Thickness** spinner and the **Direction 2 Thickness** spinner, respectively. These spinners will automatically be displayed below the **Type** drop-down list when you select the **Two-Direction** option from this drop-down list.

Cap ends

The **Cap ends** check box will be displayed only when you select a closed sketch for converting into a thin feature. This check box is selected to cap the two open faces of the thin extruded feature. Both the open faces will be capped with a face having specified thickness. When you select this check box, the **Cap Thickness** spinner will be displayed below this check box. The thickness of the end caps can be specified by using this spinner.

If the sketch to be extruded is open, as shown in Figure 3-10, the **Thin Feature** rollout will be invoked automatically on invoking the **Boss-Extrude PropertyManager**. The resulting feature is shown in Figure 3-11.

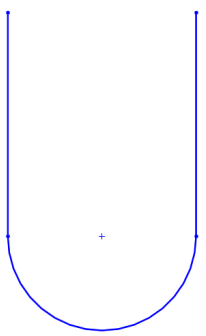


Figure 3-10 *Open sketch to be converted into a thin feature*

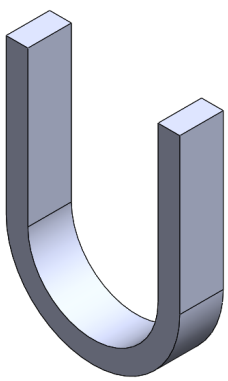


Figure 3-11 *Resulting thin feature*

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Auto-fillet corners

The **Auto-fillet corners** check box will be displayed only when you select an open sketch to convert it into a thin feature. If you select this check box, all sharp vertices in the sketch will automatically be filleted during conversion into a thin feature. As a result, the thin feature will have filleted edges. The radius of the fillet can be specified in the **Fillet Radius** spinner, which will be displayed below the **Auto-fillet corners** check box.

Figure 3-12 shows the thin feature created by extruding an open sketch in both directions. Note that a draft angle is applied to the feature while extruding in the front direction and the **Auto-fillet corners** check box is selected while creating this thin feature.

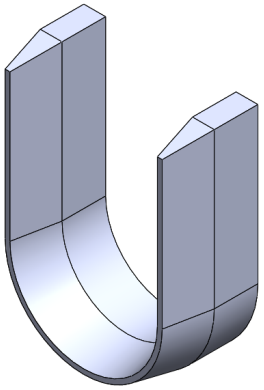


Figure 3-12 *Thin feature created in both directions*

Creating Shell Features

CommandManager: Features > Shell
SOLIDWORKS menus: Insert > Features > Shell
Toolbar: Features > Shell

Shelling is a process in which the material is scooped out from a model. The resulting model will be a hollow model with walls of a specified thickness and a cavity inside. The selected face or the faces of the model are also removed in this operation. If you do not select a face to be removed, a closed hollow model will be created. You can also specify multiple thicknesses to the walls.

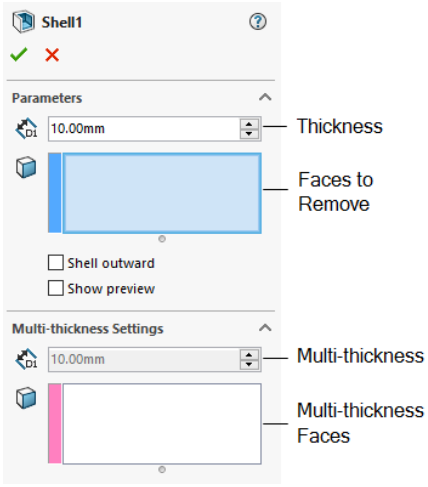


Figure 3-13 *The* ***Shell PropertyManager***

To create a shell feature, invoke the **Shell** tool from the **Features CommandManager**; the **Shell PropertyManager** will be displayed, as shown in Figure 3-13. Also, you will be prompted to select the faces to be removed. Select the face or the faces of the model that you want to remove. The selected faces will be highlighted in blue and their names will be displayed in the **Faces to Remove** selection box. Set the value of the wall thickness in the **Thickness** spinner and choose the **OK** button from the **Shell PropertyManager**. Figure 3-14 shows the face selected to remove and Figure 3-15 shows the resultant shell feature.

If none of the faces are selected to be removed, the resulting model will be hollowed from inside with no face removed. Figure 3-16 shows a model in the **Hidden Lines Visible** mode with a shell feature in which no face is selected to be removed.

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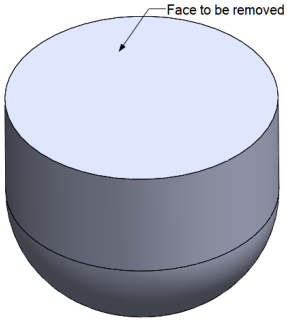


Figure 3-14 *Face to be removed*

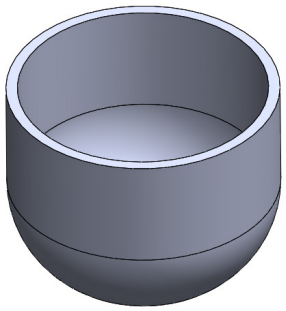


Figure 3-15 *Resultant shell feature*

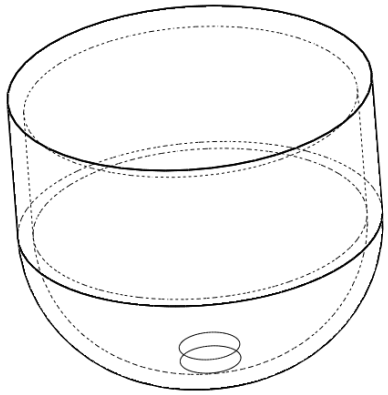


Figure 3-16 *Shell feature with no face selected to be removed*

Based on the geometric conditions, the quantity of the material to be removed from the shell feature will be decided automatically. Figure 3-17 shows the shell feature whose wall thickness is small for uniform shelling of the entire model. Figure 3-18 shows the shell feature whose wall thickness is large. As a result, it cannot accommodate uniform shelling of the entire model. Therefore, the material will not be removed from the area where the material removal is not possible.

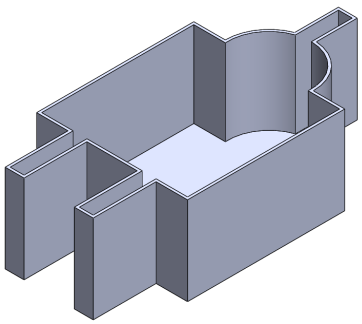


Figure 3-17 *Shell feature with small shell thickness*

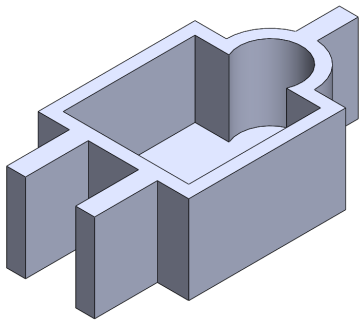


Figure 3-18 *Shell feature with larger shell thickness*

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The **Shell outward** check box is selected to create the shell feature on the outer side of the model. You can also display the preview of the shell feature by selecting the **Show preview** check box from the **Parameters** rollout of the **Shell PropertyManager**.

Creating Multi-thickness Shell

The **Shell** tool can be used to shell the model by applying different thickness values to the selected faces. To do so, invoke the **Shell PropertyManager**, select the faces to be removed and then specify the thickness in the **Thickness** spinner of the **Parameters** rollout. Click once in the **Multi-thickness Faces** selection box to activate the selection mode. Select the faces for which you want to specify different thicknesses. Set the thickness value using the **Multi-thickness(es)** spinner and choose the **OK** button. Note that you can specify different thickness for each face. Figure 3-19 shows the faces selected to create a multi-thickness shell and Figure 3-20 shows the resulting shell feature.

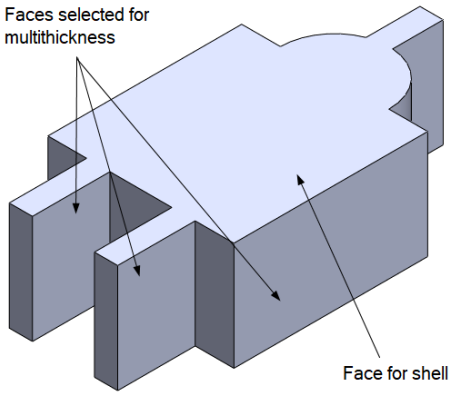


Figure 3-19 *Faces selected to create the multi-thickness shell feature*

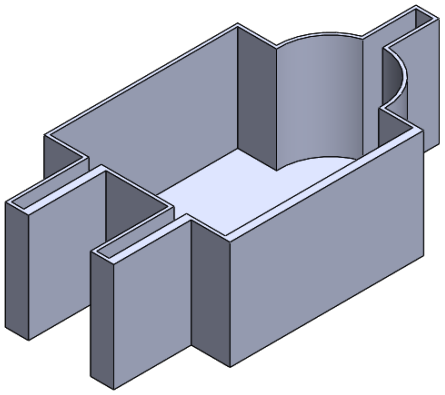


Figure 3-20 *Shell feature with larger shell thickness*

Error Diagnostics

While creating the shell feature, specify all parameters in the **Shell PropertyManager** and choose the **OK** button. If the creation of shell feature fails because of geometric inconsistency, the **Error Diagnostics** rollout will be displayed in the **Shell PropertyManager**, as shown in Figure 3-21. Also, the **Rebuild Errors** dialog box will be displayed informing you about the possible errors that can lead to the failure of feature creation. You will learn more about the **Rebuild Errors** dialog box in the later chapters. In the **Error Diagnostics** rollout, you can figure out the possible reasons behind the failure of the shell feature creation.

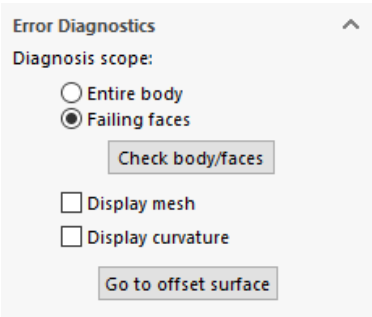


Figure 3-21 *The* ***Error Diagnostics*** *rollout*

The radio buttons in the **Diagnosis scope** area are used to specify whether the diagnosis has to be done on the entire body or only on the faces that have failed while being shelled. The **Check body/faces** button is used to run the diagnostic tool. On choosing this button, the areas of the model that can lead to failure of feature creation will be highlighted using callouts. The

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Display mesh check box is used to display the surface curvature mesh. The **Display curvature** check box is selected to display the surface curvature. The **Go to offset surface** button is used to open **Offset Surface PropertyManager** where you can offset faces with smaller gaps which sometimes lead to errors.

Creating Sweep Features

CommandManager:	Features > Swept Boss/Base
SOLIDWORKS menus:	Insert > Boss/Base > Sweep
Toolbar:	Features > Extruded Boss/Base > Swept Boss/Base



One of the most important advanced modeling tools is the **Swept Boss/Base** tool. This tool is used to extrude a closed profile along an open or a closed path. Therefore, you need a profile and a path to create a sweep feature. A profile is a section for the sweep feature and a path is the course taken by the profile while creating the sweep feature. The profile has to be a sketch, but the path can be a sketch, curve, or an edge. You will learn more about the procedure to create the curves later in this chapter. Figure 3-22 shows a profile and a path for creating a sweep feature.

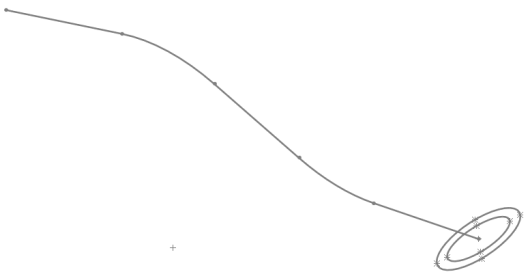


Figure 3-22 *Profile and path to create a sweep feature*

To create a sweep feature, choose the **Swept Boss/Base** button from the **Features** CommandManager; the **Sweep PropertyManager** will be displayed, as shown in Figure 3-23. You can create swept feature using sketch based profile or circular profile. By default, the **Sketch Profile** radio button is selected. As a result, you are prompted to select a sweep profile. Select the sketch drawn as the profile from the drawing area; the sketch will be highlighted and the **Profile** callout will be displayed. Also, you will be prompted to select a path for the sweep feature. Select the sketch or an edge to be used as the path; selection will be highlighted in magenta and the **Path** callout will be displayed. Also, the preview of the sweep feature will be displayed in the drawing area. Choose the **OK** button from the **Sweep PropertyManager** to end the feature creation. Figure 3-24 shows the resulting sweep feature.

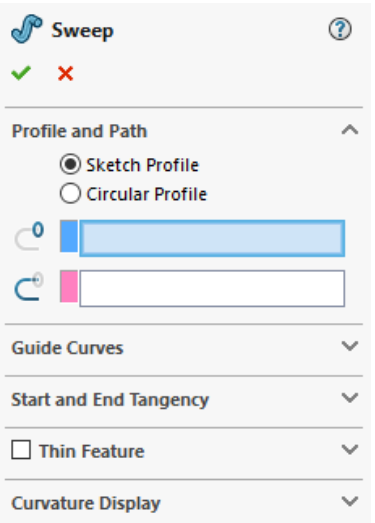


Figure 3-23 *The* ***Sweep PropertyManager***

To create a sweep feature using a circular profile along a sketch line, edge, or curve directly on a model, you need to choose

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the **Circular Profile** button from the **Profile and Path** rollout in the **Sweep PropertyManager**. Next, from the drawing area, select the sketch or edge as a path along which the circular profile is to be created from the drawing area and enter the diameter value in the **Diameter** spinner.

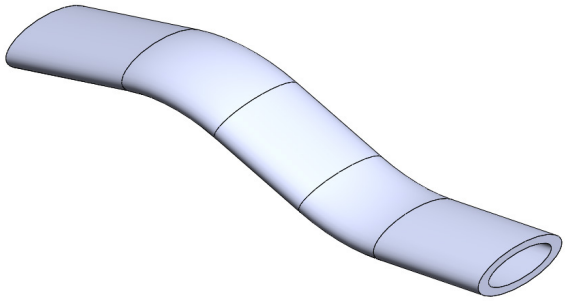


Figure 3-24 Sweep feature

TUTORIALS

Tutorial 1

In this Tutorial, you will create a model and make it ready for flow simulation. The model and its drawing are shown in Figure 3-25 and 3-26. The shell thickness of the model is 10 mm.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

a. Start SOLIDWORKS and then start a new part document.
b. Create the base feature.
c. Create Inlet 1 in the base feature.
d. Create Inlet 2 in the base feature.
e. Create Outlet in the base feature.
f. Shell the feature for fluid flow.
g. Create Lid 1, Lid 2, and Lid 3.

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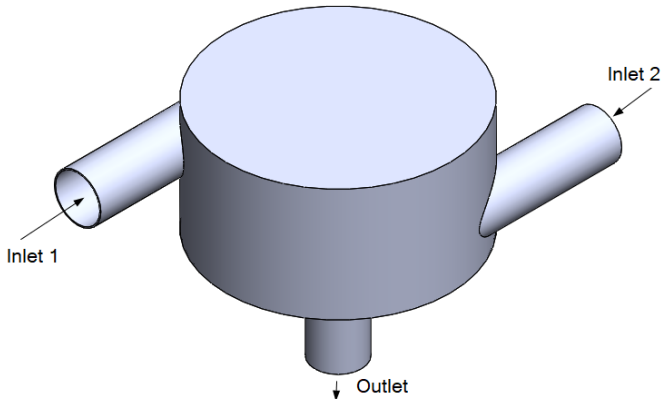


Figure 3-25 *Solid model for Tutorial 1*

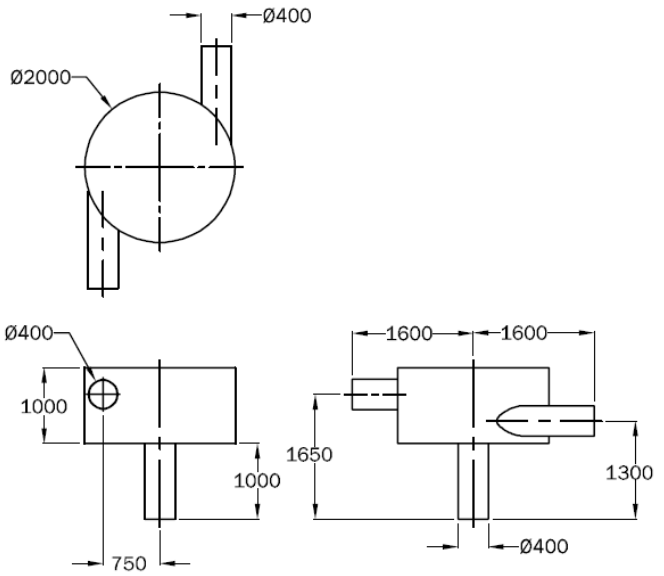


Figure 3-26 *Sketch for Tutorial 1*

Starting SOLIDWORKS and then a New SOLIDWORKS Document

1. Start SOLIDWORKS by double-clicking on the shortcut icon of SOLIDWORKS 2025 on the desktop of your computer; the SOLIDWORKS 2025 window along with the **Welcome-SOLIDWORKS** dialog box is displayed.

2. Choose the **Part** button available in the **New** area of the **Home** tab in the **Welcome - SOLIDWORKS** dialog box. A new **SOLIDWORKS** part document starts. Next, you need to invoke the sketching environment.

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3. Choose the **Sketch** tab from the **CommandManager** if not chosen by default. Next, choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.

4. Select the **Top Plane** from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view. You will notice that red colored arrows are displayed at the center of the screen indicating that you are in the sketching environment. Also, the confirmation corner with the Exit Sketch and Cancel options is displayed on the upper right corner in the graphics area. The screen display in the sketching environment of SOLIDWORKS 2025 is shown in Figure 3-27.

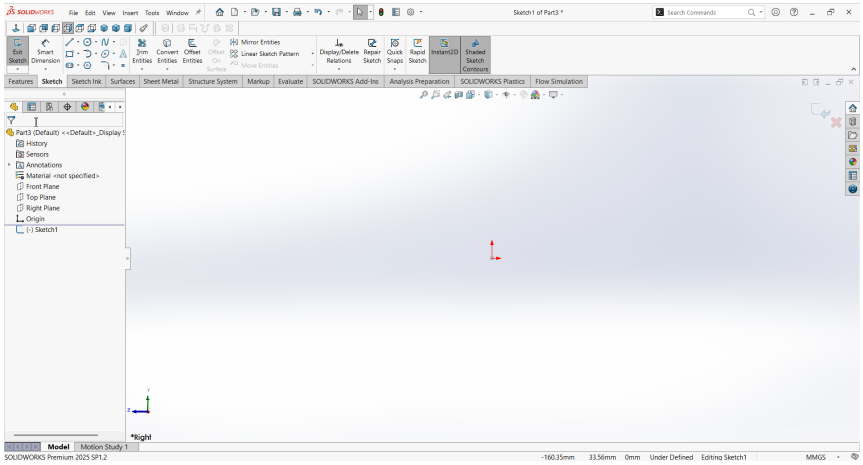


Figure 3-27 *Screen display in the sketching environment*

Drawing the Sketch

Next, you need to draw the sketch using the sketch tools. As it is evident from Figure 3-26, the sketch needs to be drawn using the **Circle** tool.

1. Press and hold the right-mouse button and drag the cursor to the right; the mouse gesture is displayed with sketching tools. Move the cursor over the **Circle** tool; the **Circle** tool is invoked and the **Circle PropertyManager** is displayed.

2. Make sure that the **Circle** button is chosen in the **Circle Type** rollout in the **Circle PropertyManager**. Move the circle cursor to the origin and click when an orange circle is displayed to specify the center point of the circle.

3. Move the cursor horizontally toward the right and draw a circle close to **2000** mm diameter and exit the **Circle** tool.

   Next, add dimension to the sketch.

4. Choose the **Smart Dimension** tool and select the circle; the dimension is attached to the cursor. Place the dimension; the **Modify** dialog box is displayed with the default value. Enter **2000** in the **Modify** dialog box and press Enter, refer to Figure 3-28.

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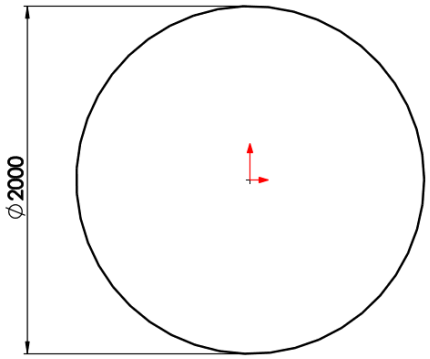


Figure 3-28 *Sketch after applying dimension*

Extruding the Sketch

Next, you need to invoke the **Extruded Boss/Base** tool and extrude the sketch using the parameters given in the tutorial description.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed, as shown in Figure 3-29.

   As you are converting the closed sketch into a feature, only the **Direction 1** rollout is displayed in the **Boss-Extrude PropertyManager**. Also, a preview of the feature is displayed in the temporary shaded graphics with the default values.

2. Set **1000** in the **Depth** spinner as the depth in the **Direction 1** rollout and then choose the **OK** button; the extrude feature is created, as shown in Figure 3-30.

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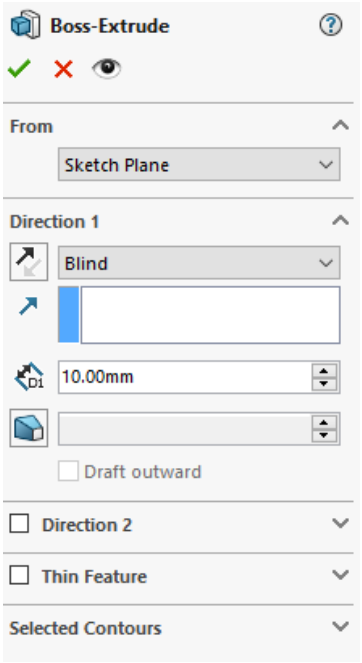


*Figure 3-29 The **Boss Extrude PropertyManager***

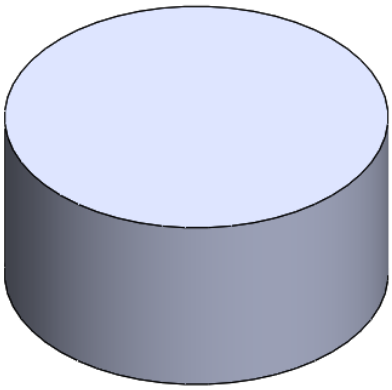


Figure 3-30 Sketch for Tutorial 1

Creating the Inlet Pipe 1

Now, you need to create a cylindrical feature to get the required shape of the base feature. The sketch for this feature is created using a reference plane that is offset from Front plane.

1. Choose the **Plane** tool from the **Reference Geometry** flyout in the **Features CommandManager** to display the **Plane PropertyManager**.

2. Select the Front plane as the first reference and choose the **Offset distance** button from the **First Reference** rollout if not selected by default; the **Offset distance** spinner, and the **Flip offset** check box, are displayed in the **Plane PropertyManager**.

3. Set the value in the **Offset distance** spinner to **1600** and choose the **OK** button from the **Plane PropertyManager**; the required plane is created, as shown in Figure 3-31.

4. Select the reference plane which you just created if not already selected, and invoke the sketching environment. Set the current view normal to the eye view.

5. Draw the sketch of the circle and apply required relations to the sketch, as shown in Figure 3-32.

6. Change the current view to the isometric view and invoke the **Extruded Boss/Base** tool. You will observe in the preview that the direction of the feature creation is opposite to the required direction. Therefore, you need to change the direction of the feature creation.

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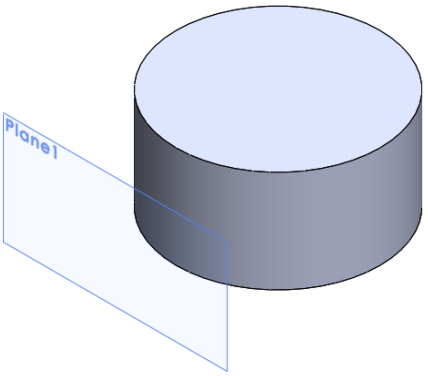


Figure 3-31 *The plane created*

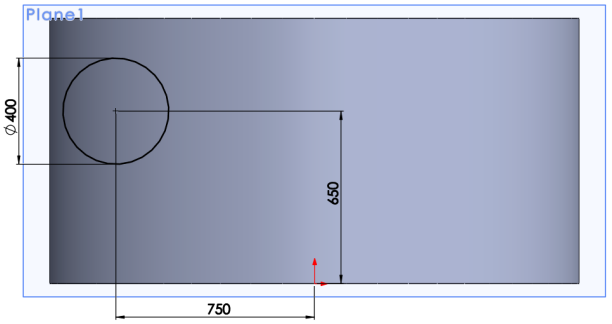


Figure 3-32 *Sketch created on plane*

7. Choose the **Reverse Direction** button on the left of the **End Condition** drop-down list to reverse the direction of feature creation; preview of the feature changes dynamically.

8. Right-click in the drawing area and choose the **Up To Surface** option from the shortcut menu; you are prompted to select a face or a surface to specify the first direction. Also, the **Face/Plane** selection box is displayed below the **Direction of Extrusion** selection box in the **Direction 1** rollout.

9. Select the curved surface of the model using the left mouse button. You will observe in the preview that the feature is extruded up to the selected surface, as shown in Figure 3-33.

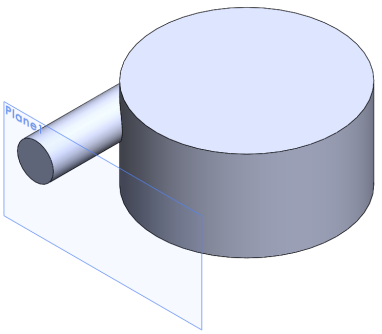


Figure 3-33 *Inlet Pipe 1 created*

10. Choose the **OK** button from the **Boss-Extrude PropertyManager**.

For better display turn off the plane display.

Creating the Inlet Pipe 2

Now, you need to create a cylindrical feature to get required shape of the base feature. The sketch for this feature is created using a reference plane that is offset from Front plane.

1. Choose the **Plane** tool from the **Reference Geometry** flyout in the **Features CommandManager** to display the **Plane PropertyManager**.

2. Select the Front plane as the first reference and choose the **Offset distance** button from the **First Reference** rollout if not selected by default; the **Offset distance** spinner and the **Flip offset** check box are displayed in the **Plane PropertyManager**.

3. Set the value in the **Offset distance** spinner to **1600** and choose the **OK** button from the **Plane PropertyManager**; the required plane is created. Select the **Flip offset** check box to reverse the direction of plane, as shown in Figure 3-34.

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4. Select the reference plane which you just created if not already selected, and invoke the sketching environment. Set the current view normal to the eye view.

5. Draw the sketch of the circle and apply required relations and dimensions to the sketch, as shown in Figure 3-35.

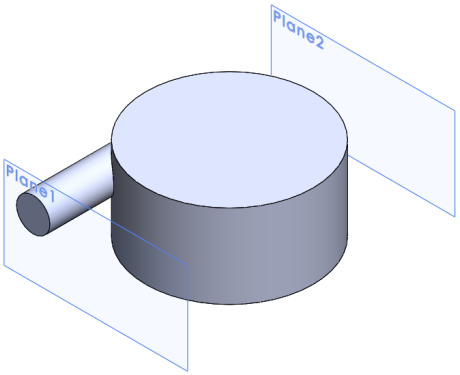


Figure 3-34 *The plane created*

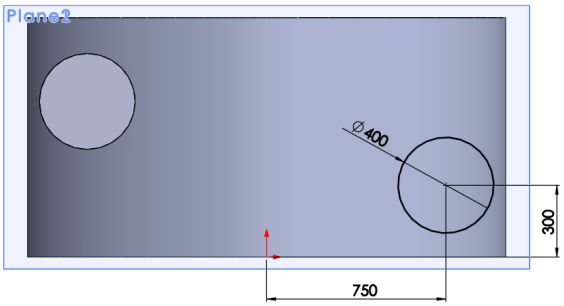


Figure 3-35 *Sketch created on plane*

6. Change the current view to the isometric view and invoke the **Extruded Boss/Base** tool.

7. Right-click in the drawing area and choose the **Up To Surface** option from the shortcut menu; you are prompted to select a face or a surface to specify the first direction. Also, the **Face/Plane** selection box is displayed below the **Direction of Extrusion** selection box in the **Direction 1** rollout.

8. Select the upper curved surface of the model using the left mouse button. You will observe in the preview that the feature is extruded up to the selected surface.

9. Choose the **OK** button from the **Boss-Extrude PropertyManager**, as shown in Figure 3-36.

   For better display of the model turn off the plane display.

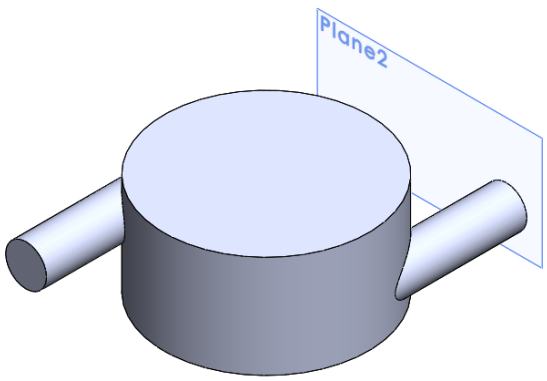


Figure 3-36 *Inlet Pipe 2 created*

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Creating the Inlet Pipe 3

Now, you need to create a cylindrical feature to get the required shape of the base feature. The sketch for this feature is created on the bottom face of the base feature.

1. Select the bottom face of the base feature and invoke the sketching environment. Set the current view normal to the eye view.

2. Draw the sketch of the circle and apply required relations to the sketch, as shown in Figure 3-37.

3. Change the current view to the isometric view and invoke the **Extruded Boss/Base** tool.

4. Set **1000** in the **Depth** spinner and choose the **OK** button from the **Boss-Extrude PropertyManager**; the selected contour is extruded, refer to Figure 3-38.

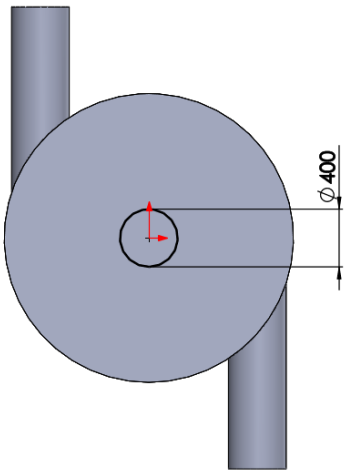


Figure 3-37 *Sketch created on bottom face*

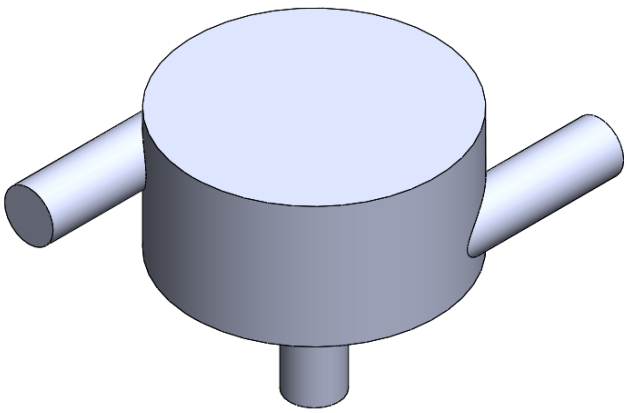


Figure 3-38 *Inlet Pipe 3 created*

Creating the Shell Feature for Fluid Flow

The **Shell** tool is used to scoop out material from the model leaving behind a thin-walled hollow part. This hollow space is used for fluid flow.

1. Choose the **Shell** button from the **Features CommandManager**; the **Shell PropertyManager** is displayed and you are prompted to select the faces to be removed.

2. Rotate the model and select the faces to be removed, as shown in Figure 3-39; the names of the selected faces are displayed in the **Faces to Remove** selection box.

3. Set the value to **10** in the **Thickness** spinner and choose the **OK** button from the **Shell PropertyManager**. The model after creating the shell feature is shown in Figure 3-40.

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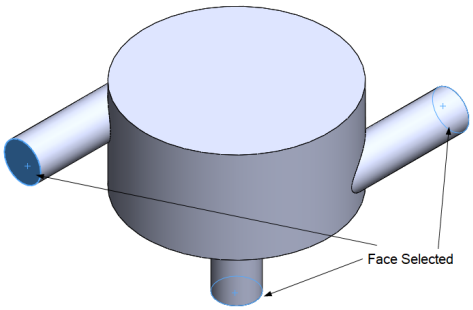


Figure 3-39 Face to be shell

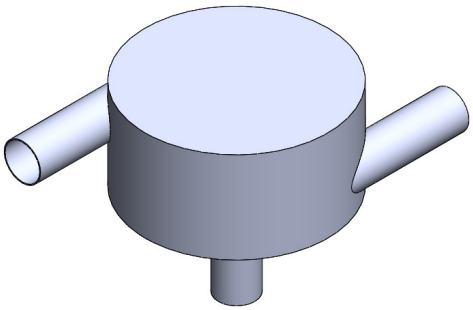


Figure 3-40 Model after shell

Creating the Lids by Using the Extruded Feature

Next, you need to create the lid at the inlet and outlet of the model.

1. Invoke the **Extruded Boss/Base** tool, select the inlet pipe 1 face as the sketching plane, and create the sketch using the standard sketching tools. The sketch consists of a circle having 380 mm diameter.

2. Exit the sketching environment; the **Boss-Extrude PropertyManager** is displayed.

3. Set **50** in the **Depth** spinner and clear the **Merge Result** check box. Choose the **OK** button from the **Boss-Extrude PropertyManager**; the selected contour is extruded, refer to Figure 3-41.

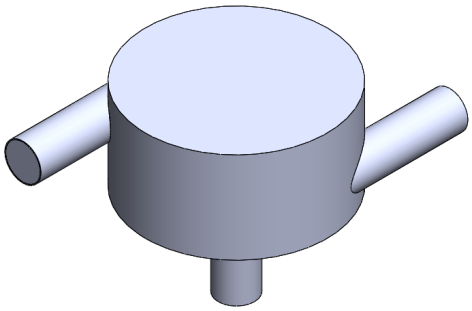


Figure 3-41 Lid added to the model

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4. Similarly, create lid 2 and lid 3 for the inlet 2 and outlet 1.

   Next, rename the tree items to Lid 1, Lid 2, and Lid 3.

5. Right-click on **Boss-Extrude5** and choose the **Rename tree item** option. Type the name **Lid 1**; the name is updated in the design tree. Refer to Figure 3-42 for design tree.

   Similarly, you can rename the other extruded feature as Lid 2 and Lid 3.

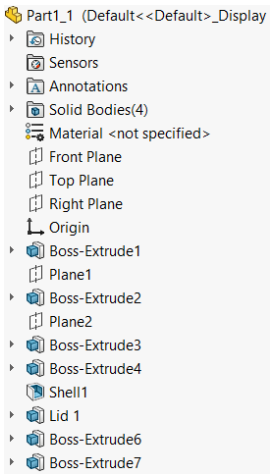


Figure 3-42 *Feature Manager Design Tree*

Saving the Model

1. Save the part document with the name **c03_tut01** at the following location: ***\Documents\SOLIDWORKS\c03***

2. Choose **File > Close** from the SOLIDWORKS menus to close the document.

Tutorial 2

In this tutorial, you will create a model and make it ready for flow simulation. The model and its drawing are shown in Figure 3-43 and 3-44. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Start SOLIDWORKS and then start a new part document.
b. Create the base feature.
c. Create the pipe channel.
d. Connect the base feature and pipe channel for heat transfer.
e. Create Lid 1.
f. Create Lid 2.

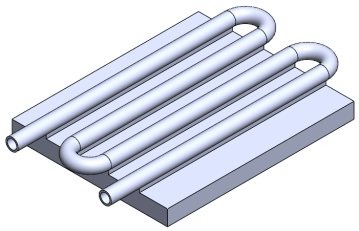


Figure 3-43 *Solid model for Tutorial 2*

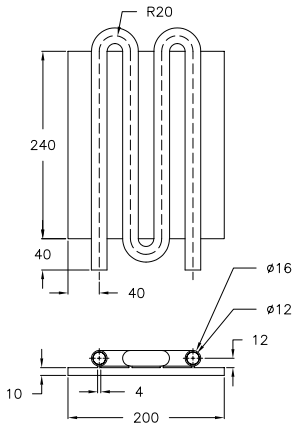


Figure 3-44 *Sketch for Tutorial 2*

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Starting SOLIDWORKS and then a New SOLIDWORKS Document

1. Start SOLIDWORKS by double-clicking on the shortcut icon of SOLIDWORKS 2025 on the desktop of your computer; the SOLIDWORKS 2025 window along with the **WELCOMESOLIDWORKS 2025** dialog box is displayed.

2. Choose the **Part** button available in the **New** area of the **Home** tab in the **Welcome - SOLIDWORKS 2025** dialog box; a new **SOLIDWORKS** part document starts.

   Next, you need to invoke the sketching environment.

3. Choose the **Sketch** tab from the **CommandManager** if not selected by default. Next, choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.

4. Select the **Top Plane** from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view. You will notice that red colored arrows are displayed at the center of the screen indicating that you are in the sketching environment. Also, the confirmation corner with the Exit Sketch and Cancel options is displayed on the upper right corner in the graphics area. The screen display in the sketching environment of SOLIDWORKS 2025 is shown in Figure 3-45.

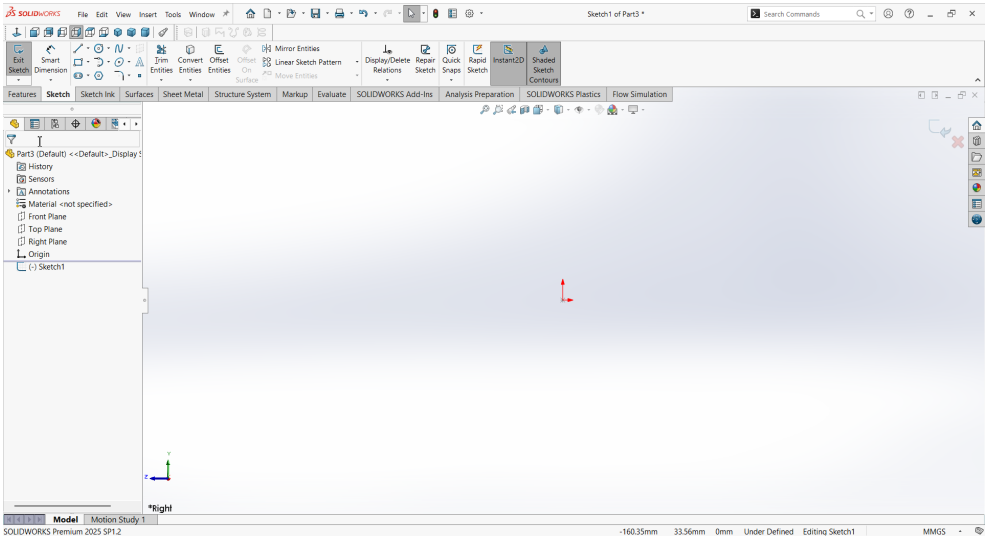


Figure 3-45 *Screen display in the sketching environment*

Drawing the Sketch

Next, you need to draw the sketch using the sketch tools. As it is evident from Figure 3-44, the sketch needs to be drawn using the **Rectangle** tool to create the sketch of base feature.

1. Choose the **Center Rectangle** tool from the **Rectangle** flyout of the **Sketch CommandManager** to draw a rectangle by specifying the center and one of the corners; the **Rectangle PropertyManager** is displayed on the left of the drawing area. Also, the arrow cursor changes to the rectangle cursor.

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2. Move the rectangle cursor to the origin and click when an orange circle is displayed to specify the center of the rectangle.

3. Move the cursor toward the right and click when a dimension is displayed above the rectangle cursor close to the dimension of the base feature.

   Next, add dimension to the sketch.

4. Choose the **Smart Dimension** tool and select the vertical line; the dimension is attached to the cursor. Place the dimension; the **Modify** dialog box is displayed with the default value. Enter **240** in the **Modify** dialog box and press ENTER.

5. Next, select the horizontal line; the dimension is attached to the cursor. Place the dimension; the **Modify** dialog box is displayed with the default value. Enter **200** in the **Modify** dialog box and press ENTER. Refer to Figure 3-46 for the sketch.

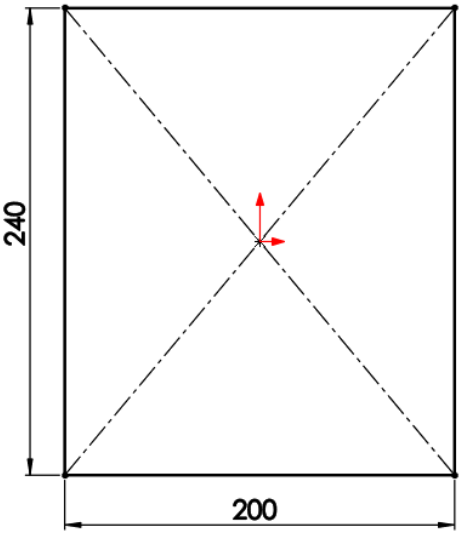


Figure 3-46 *Sketch after applying dimension*

Extruding the Sketch

The next step after creating the sketch is to extrude it. You can extrude the sketch using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

   As you are converting the closed sketch into a feature, only the **Direction 1** rollout is displayed in the **Boss-Extrude PropertyManager**. Also, a preview of the feature is displayed in the temporary shaded graphics with the default values.

2. Set **10** in the **Depth** spinner as the depth in the **Direction 1** rollout and then choose the **OK** button to extrude the sketch. Refer to Figure 3-47 for the model.

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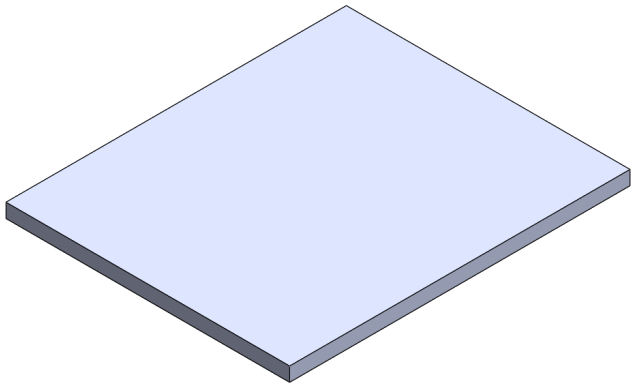


Figure 3-47 *Model created by extruding the sketch*

Creating the Plane

Next, create a plane at an offset of 6mm from the top face of the base feature.

1. Choose the **Plane** tool from the **Reference Geometry** flyout in the **Features CommandManager**; the **Plane PropertyManager** is displayed.

2. Select the top face of the base feature; the selected face is displayed in the **First Reference** selection box.

3. Enter **12** in the **Offset distance** spinner to create the plane at an offset distance. Choose the **OK** button to close the **Plane PropertyManager**, refer to Figure 3-48 for the created plane.

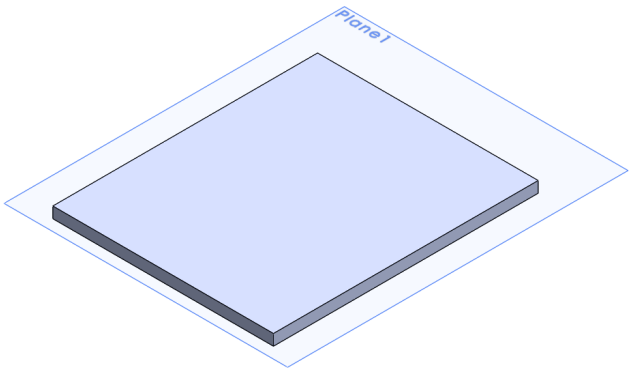


Figure 3-48 *Creating a plane offset from face*

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Creating the Channel

Next, you need to draw the sketch for channel using the sketch tools. As it is evident from Figure 3-44, the sketch needs to be drawn using the **Arc** and **Line** tools. After creating the sketch, you need to sweep the circular profile along that sketch.

1. Choose the **Sketch** button from the **Sketch CommandManager** and select Plane 1 to invoke the sketching environment.

2. Press and hold the right-mouse button and drag the cursor to the right; the mouse gesture is displayed with sketching tools. Move the cursor over the **Line** tool; the **Line** tool is invoked and the **Insert Line PropertyManager** is displayed.

3. Create the sketch of the channel of the model on the offset plane created earlier. Apply the required relations and dimensions to the sketch, as shown in Figure 3-49. Make sure that you do not exit the sketching environment.

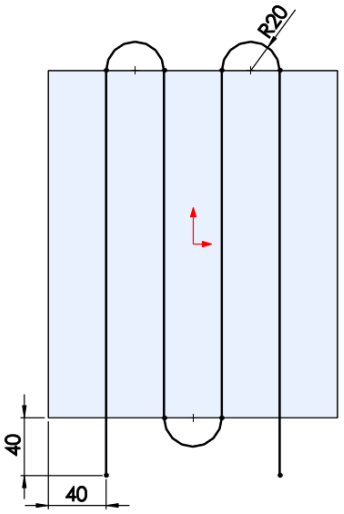


Figure 3-49 *The sketch on a plane*

Next, the sweep feature that you need to create is a thin circular sweep feature. You need to use the **Thin Feature** rollout to specify the parameters of the sweep feature.

4. Choose the **Swept Boss/Base** tool from the **Features CommandManager**; the **Sweep PropertyManager** is displayed.

5. Select the **Circular Profile** radio button from the **Profile and Path** rollout. As a result, the **Path** selection box and the **Diameter** spinner is displayed.

6. Select the sketch from the drawing area; the name of the sketch is displayed in the **Path** selection box. Also, set the value of diameter in the **Diameter** spinner to **16**.

7. Select the check box on the left of the **Thin Feature** rollout; the **Thin Feature** rollout is invoked.

8. Set the value of thickness in the **Thickness** spinner to **2**.

9. Choose the **OK** button from the **Sweep PropertyManager** or choose **OK** from the confirmation corner. Refer to Figure 3-50 for the model.

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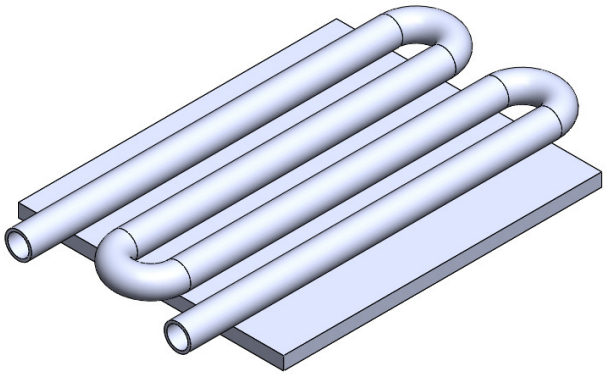


Figure 3-50 *Model created by sweeping*

Connecting Plate and Pipe Channel Through Joint

Next, you need to connect the plate and pipe channel so that heat transfer takes place between these two.

1. Choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.

2. Select the front face of the plate from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view.

3. Create the sketch of joint and dimension it as shown in Figure 3-51. Similarly, create the sketch of other joints and dimension them.

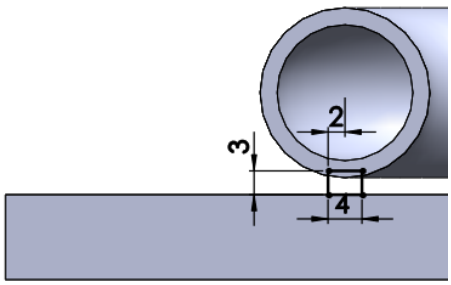


Figure 3-51 *Creating the sketch of joint*

4. Choose the **Exit Sketch** button to exit the sketching environment.

Next, you need to extrude the sketch upto the back face of the plate.

5. Invoke the **Extruded Boss/Base** tool and select the sketch if it is not selected.

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6. Select the **Up To Surface** option from the **End Condition** drop-down in the **Direction 1** rollout. Select the back face of the plate, a preview of the feature is displayed.

7. Choose the **OK** button to extrude the sketch. Refer to Figure 3-52 for the joint created by extruding the sketch.

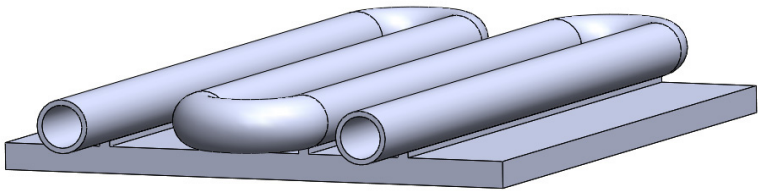


Figure 3-52 *Joint created by extrusion*

Creating the Lids by Using the Extruded Feature

Next, you need to create Lid 1 and Lid 2 which helps you to specify the inlet and outlet conditions in flow simulation environment.

1. Invoke the **Extruded Boss/Base** tool, select the inlet pipe 1 face as the sketching plane, and create the sketch using the standard sketching tools. The sketch consists of a circle having 12 mm diameter.

2. Exit the sketching environment; the **Boss-Extrude PropertyManager** is displayed.

3. Set **1** in the **Depth** spinner and clear the **Merge Result** check box. Choose the **OK** button from the **Boss-Extrude PropertyManager**; the selected contour is extruded, refer to Figure 3-53.

4. Similarly, create Lid 2 for outlet of pipe channel, refer to Figure 3-54.

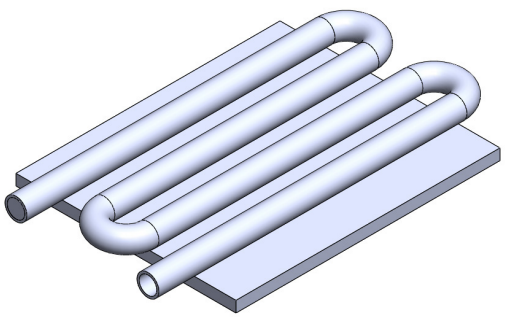


Figure 3-53 *Creating Lid 1 at inlet*

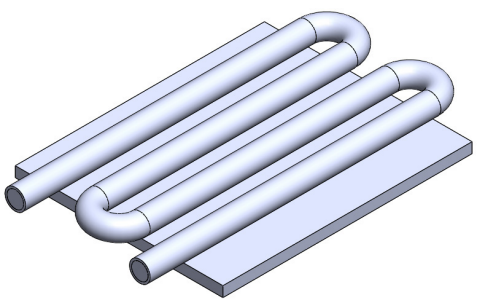


Figure 3-54 *Creating Lid 2 at outlet*

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Next, rename the tree items to Lid 1, and Lid 2.

5. Right-click on **Boss-Extrude5** and select the **Rename tree item** option. Type the name **Lid 1**; the name is updated in the tree.

Similarly, you can rename the other extruded feature as Lid 2.

Saving the Model

1. Save the part document with the name **c03_tut02** at the following location: ***\Documents\SOLIDWORKS\c03***

2. Choose **File > Close** from the SOLIDWORKS menus to close the document.

Tutorial 3

In this tutorial, you will create a Ahmed body model. The model and its drawing are shown in Figure 3-55 and 3-56. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Start SOLIDWORKS and then start a new part document.
b. Create the base feature.
c. Apply fillet to the feature.
d. Create the cut feature.
e. Create legs of body by extrusion.

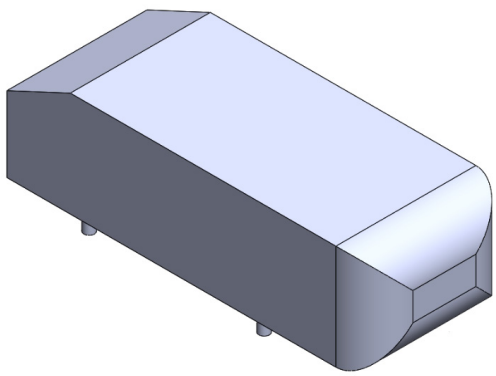


Figure 3-55 *Solid model for Tutorial 3*

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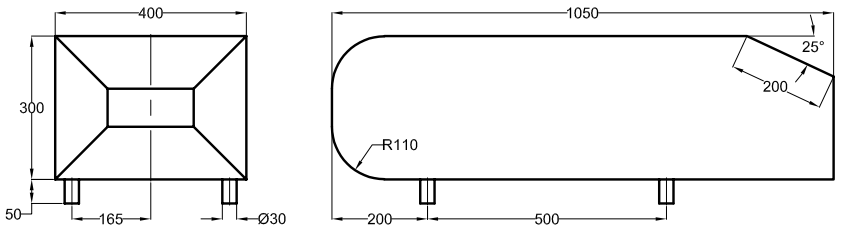


Figure 3-56 *Sketch for Tutorial 3*

Starting SOLIDWORKS and then a New SOLIDWORKS Document

1. Start SOLIDWORKS by double-clicking on the shortcut icon of SOLIDWORKS 2025 on the desktop of your computer; the SOLIDWORKS 2025 window along with the **Welcome-SOLIDWORKS 2025** dialog box is displayed.

2. Choose the **Part** button available in the **New** area of the **Home** tab in the **Welcome - SOLIDWORKS 2025** dialog box; a new **SOLIDWORKS** part document starts.

   Next, you need to invoke the sketching environment.

3. Choose the **Sketch** tab from the **CommandManager** if not selected by default. Next, choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.

4. Select the **Right Plane** from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view. You will notice that the red colored arrows are displayed at the center of the screen indicating that you are in the sketching environment. Also, the confirmation corner with the Exit Sketch and Cancel options is displayed on the upper right corner in the graphics area. The screen display in the sketching environment of SOLIDWORKS 2025 is shown in Figure 3-57.

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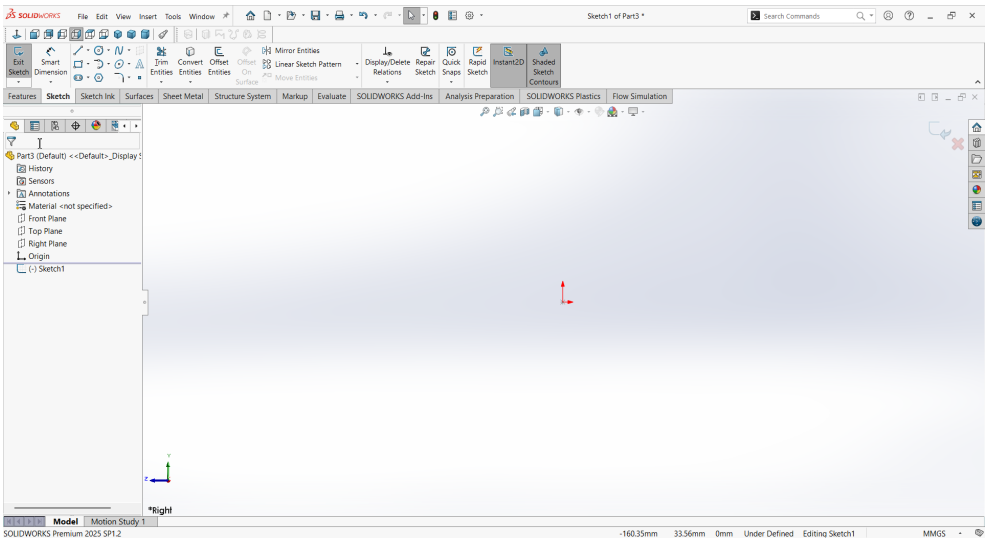


Figure 3-57 Screen display in the sketching environment

Drawing the Sketch

Next, you need to draw the sketch using the sketch tools. As it is evident from Figure 3-56, the sketch needs to be drawn using the **Rectangle** tool to create the sketch of base feature.

1. Choose the **Corner Rectangle** tool from the **Rectangle** flyout of the **Sketch CommandManager** to draw a rectangle by specifying the center and one of the corners; the **Rectangle PropertyManager** is displayed on the left of the drawing area. Also, the arrow cursor changes to the rectangle cursor.

2. Create the rectangle shown in Figure 3-58.

   Next, add dimension to the sketch.

3. Choose the **Smart Dimension** tool and add a dimension to the sketch, refer to Figure 3-58 for the sketch.

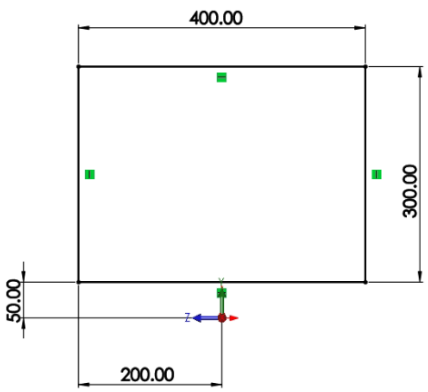


Figure 3-58 Sketch after applying dimension

Extruding the Sketch

The next step after creating the sketch is to extrude it. You can extrude the sketch using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

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As you are converting the closed sketch into a feature, only the **Direction 1** rollout is displayed in the **Boss-Extrude PropertyManager**. Also, a preview of the feature is displayed in the temporary shaded graphics with the default values.

2. Set **1050** in the **Depth** spinner as the depth in the **Direction 1** rollout and then choose the **OK** button to extrude the sketch. Refer to Figure 3-59 for the model.

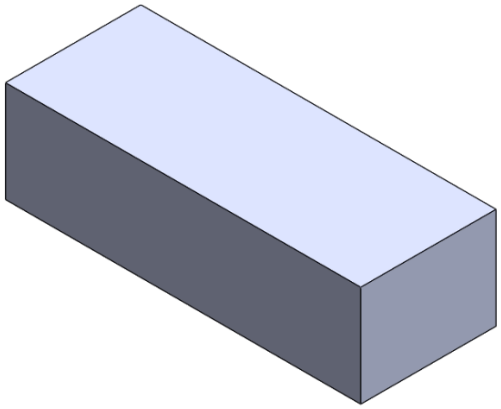


Figure 3-59 *Model created by extruding the sketch*

Creating the Fillet Features

After creating the base feature, you need to add fillets to the model. In this model, you need to add four fillet features.

1. Choose the **Fillet** button from the **Features CommandManager** to invoke the **Fillet PropertyManager**.

2. Select the edges of the model, as shown in Figure 3-60.

On selecting the edges of the model, the preview of the fillet with default values and the radius callout is displayed in the drawing area.

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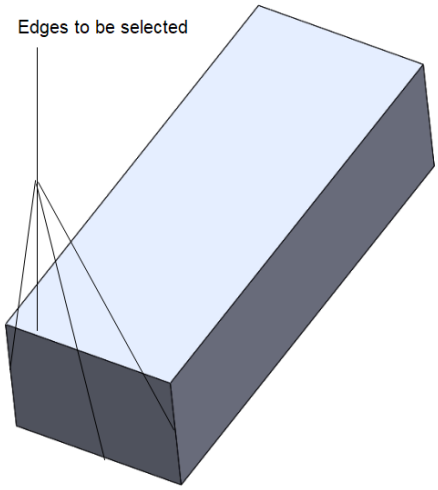


Figure 3-60 *Edges to be selected*

3. Set the value in the **Radius** spinner to **110** and choose the **OK** button from the **Fillet PropertyManager**. Figure 3-61 shows the model after adding the fillet feature.

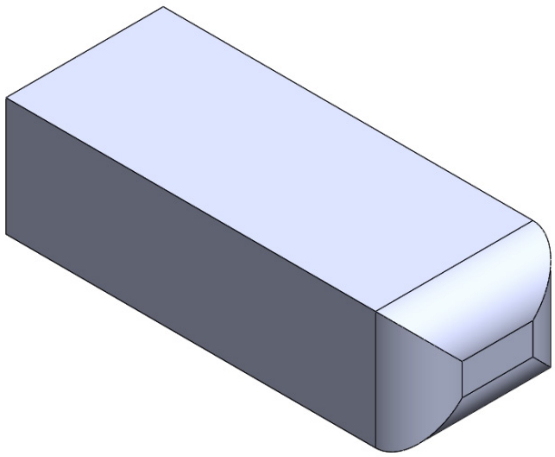


Figure 3-61 *Fillet added to the model*

Creating the Cut Feature

The next feature that you will create is a cut feature.

1. Select the Front plane from the drawing area as the sketching plane and press **S** on the keyboard. Invoke the **Extruded Cut** tool from the Shortcut toolbar.

2. Draw a sketch using the standard sketch tools, refer to Figure 3-62.

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Figure 3-62 *Sketch for the cut feature*

3. Exit the sketching environment and choose the **Through All** option from the **End Condition** drop-down list of the **Direction 1** and **Direction 2** rollouts.

4. Choose the **OK** button from the **Cut-Extrude1 PropertyManager,** refer to Figure 3-63 for the model.

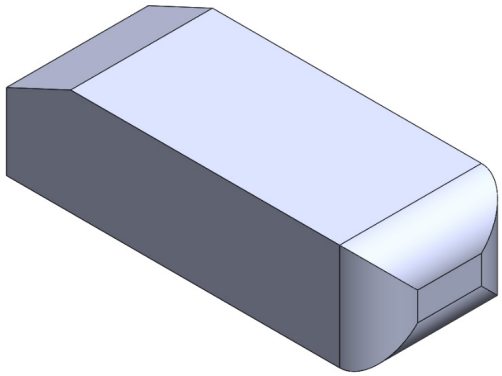


Figure 3-63 *Model after the cut feature*

Creating the Extruded Feature

Next, you need to create legs of the body using the **Extruded Boss/Base** tool.

1. Create the sketch on the bottom face of the body, refer to Figure 3-64.

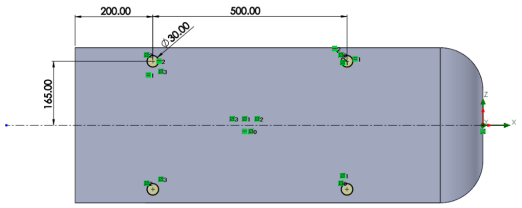


Figure 3-64 *Sketch for the extruded feature*

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2. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

3. Set **50** in the **Depth** spinner as the depth in the **Direction 1** rollout and then choose the **OK** button to extrude the sketch. Refer to Figure 3-65 for the model.

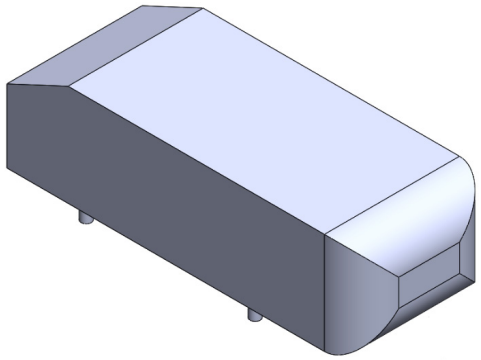


Figure 3-65 *Creating legs at the bottom of the body*

Saving the Model

1. Save the part document with the name **c03_tut03** at the following location: ***\Documents\SOLIDWORKS\c03***

2. Choose **File > Close** from the SOLIDWORKS menus to close the document.

Tutorial 4

In this tutorial, you will create a centrifugal pump model which is a multi body parts. The model and its drawing are shown in Figure 3-66 and 3-67. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Start SOLIDWORKS and then start a new part document.
b. Create body 1.
c. Create body 2.
d. Create the inlet and outlet for body 2.
e. Create the shell feature.

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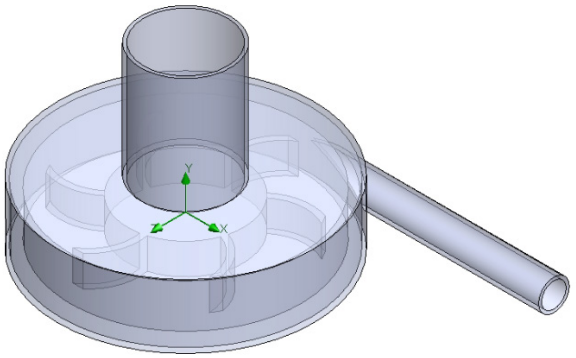


Figure 3-66 *Solid model for Tutorial 4*

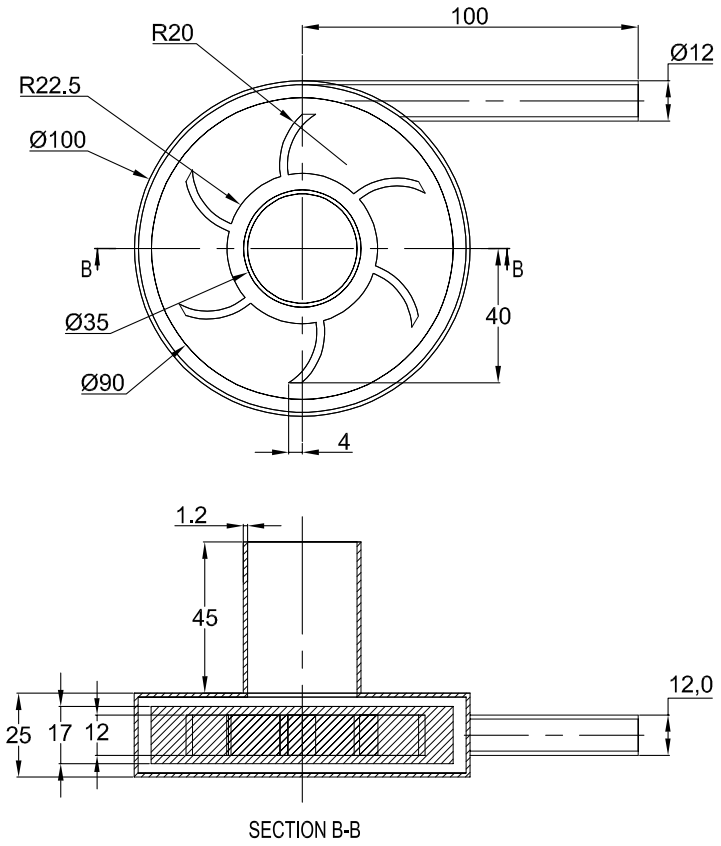


Figure 3-67 *Sketch for Tutorial 4*

Starting SOLIDWORKS and then a New SOLIDWORKS Document

1. Start SOLIDWORKS by double-clicking on the shortcut icon of SOLIDWORKS 2025 on the desktop of your computer; the SOLIDWORKS 2025 window along with the **WELCOME-SOLIDWORKS 2025** dialog box is displayed.

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2. Choose the **Part** button available in the **New** area of the **Home** tab in the **Welcome - SOLIDWORKS 2025** dialog box; a new **SOLIDWORKS** part document starts.

   Next, you need to invoke the sketching environment.

3. Choose the **Sketch** tab from the **CommandManager** if not selected by default. Next, choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.

4. Select the **Top Plane** from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view. You will notice that the red colored arrows are displayed at the center of the screen indicating that you are in the sketching environment. Also, the confirmation corner with the Exit Sketch and Cancel options is displayed on the upper right corner in the graphics area. The screen display in the sketching environment of SOLIDWORKS 2025 is shown in Figure 3-68.

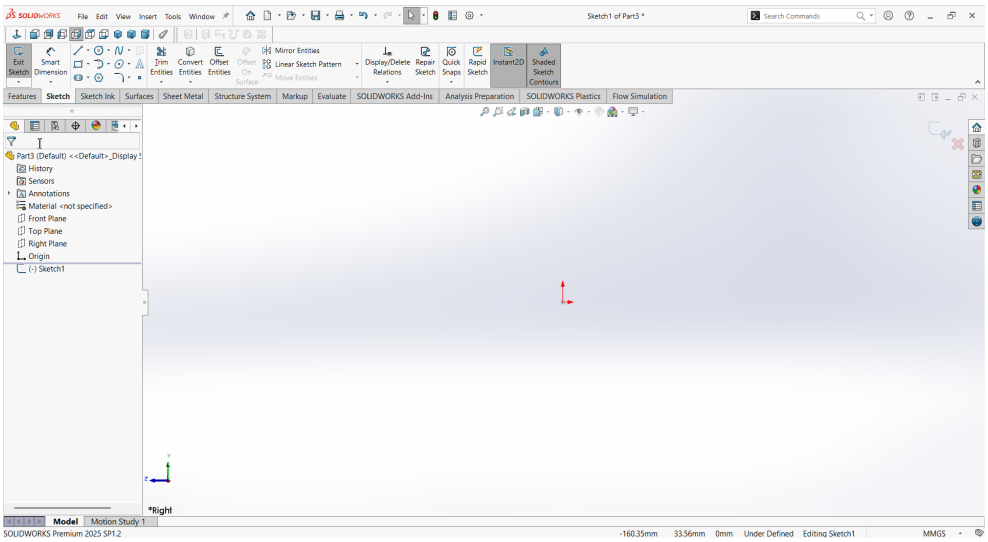


Figure 3-68 *Screen display in the sketching environment*

Drawing the Sketch

Next, you need to draw the sketch using the sketch tools. As it is evident from Figure 3-67, the sketch needs to be drawn using the **Circle** tool to create the sketch of base feature.

1. Create the sketch using the standard sketching tools, as shown in Figure 3-69.

   Next, add dimension to the sketch.

2. Choose the **Smart Dimension** tool and add a dimension to the sketch, refer to Figure 3-69 for the sketch.

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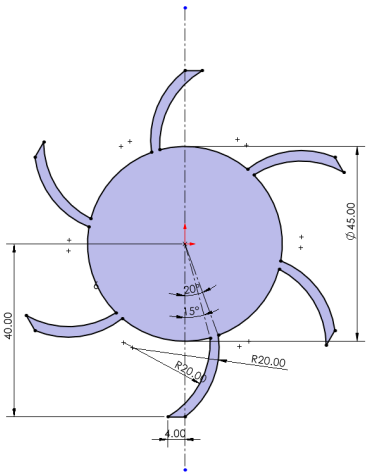


Figure 3-69 *Sketch after applying dimension*

Extruding the Sketch

The next step after creating the sketch is to extrude it. You can extrude the sketch using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

   As you are converting the closed sketch into a feature, only the **Direction 1** rollout is displayed in the **Boss-Extrude PropertyManager**. Also, a preview of the feature is displayed in the temporary shaded graphics with the default values.

2. Set **12** in the **Depth** spinner as the depth and select the **Midplane** option in the **End Condition** drop-down list in the **Direction 1** rollout and then choose the **OK** button to extrude the sketch. Refer to Figure 3-70 for the model.

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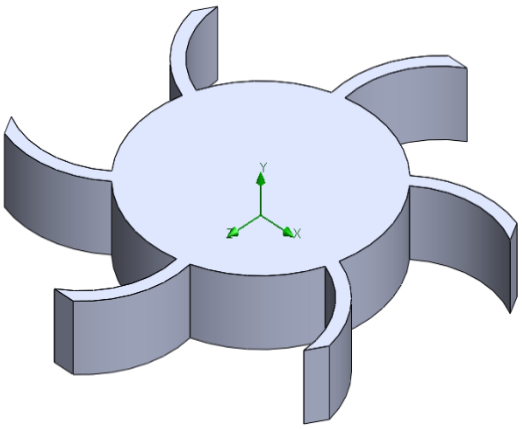


Figure 3-70 *Model created by extruding the sketch*

Drawing the Casing Sketch

Next, you need to draw the sketch using the sketch tools.

1. Create the sketch using the **Circle** tool as shown in Figure 3-71.

   Next, add dimension to the sketch.

2. Choose the **Smart Dimension** tool and add a dimension to the sketch, refer to Figure 3-71 for the sketch.

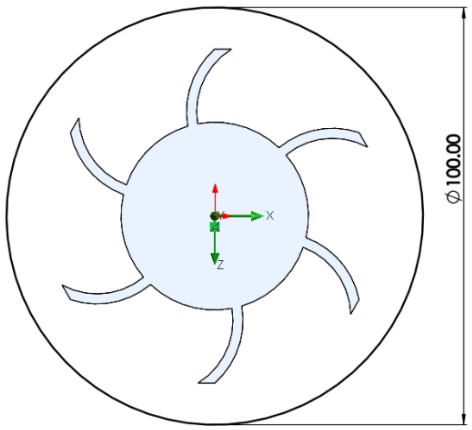


Figure 3-71 *Sketch after applying dimension*

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Extruding the Casing Sketch

The next step after creating the sketch is to extrude it. You can extrude the sketch using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

   As you are converting the closed sketch into a feature, only the **Direction 1** rollout is displayed in the **Boss-Extrude PropertyManager**. Also, a preview of the feature is displayed in the temporary shaded graphics with the default values.

2. Set **25** in the **Depth** spinner as the depth and select the **Midplane** option in the **End Condition** drop-down list in the **Direction 1** rollout and then choose the **OK** button to extrude the sketch. Refer to Figure 3-72 for the model.

   You need to ensure that the **Merge result** check box is not selected and adjust the transparency of the model in order to see inside it.

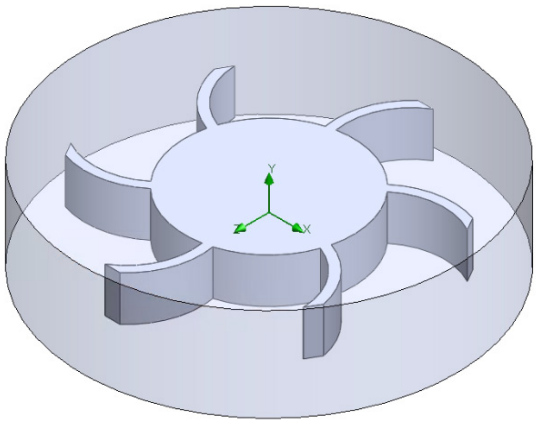


Figure 3-72 *Model created by extruding the sketch*

Drawing the Inlet Sketch

Next, you need to draw the sketch on top face of the casing using the sketch tools.

1. Create the sketch using the **Circle** tool, as shown in Figure 3-73.

   Next, add dimension to the sketch.

2. Choose the **Smart Dimension** tool and add a dimension to the sketch, refer to Figure 3-73 for the sketch.

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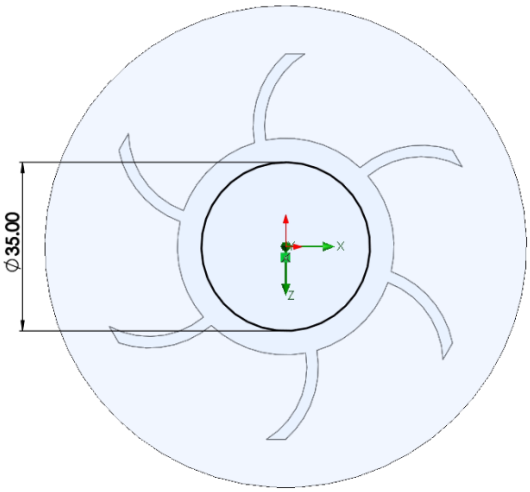


Figure 3-73 *Sketch after applying dimension*

Creating the Inlet Extruded Feature

Next, you need to create legs of the body using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

2. Set **45** in the **Depth** spinner as the depth in the **Direction 1** rollout and then choose the **OK** button to extrude the sketch. Refer to Figure 3-74 for the model. Ensure that the **Merge result** check box is selected.

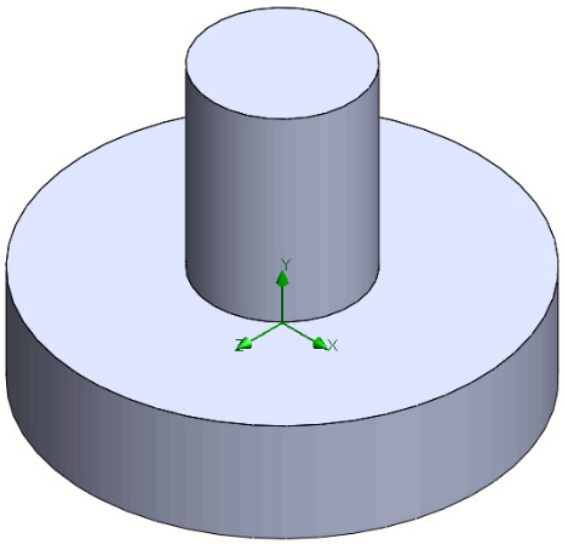


Figure 3-74 *Model created by extruding the sketch*

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Drawing the Outlet Sketch

Next, you need to draw the sketch using the sketch tools. Before creating the sketch, you need to create a plane 100 mm offset from the Right plane.

1. Choose the **Plane** tool from the **Reference Geometry** drop-down; the **Plane PropertyManager** is displayed.

2. Select the **Right Plane** from the drawing area; the name of the plane will be selected in the **First Reference** edit box. Enter **100** in the **Offset distance** edit box. Refer to Figure 3-75 for the plane.

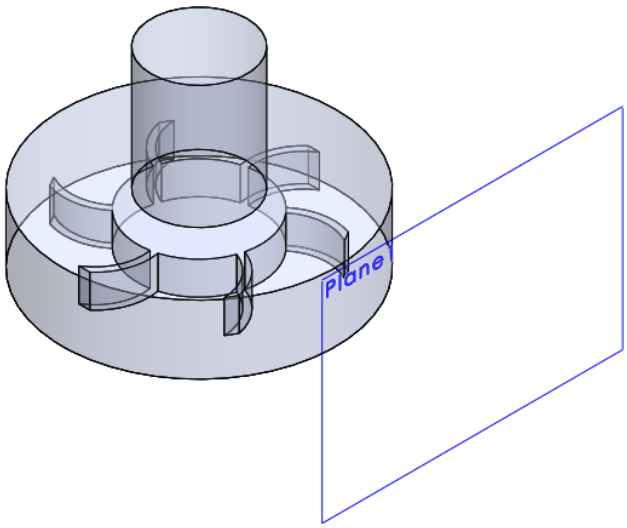


Figure 3-75 *The plane created*

Next, create the sketch on the plane.

3. Create the sketch using the **Circle** tool, as shown in Figure 3-76.

Next, add dimension to the sketch.

4. Choose the **Smart Dimension** tool and add a dimension to the sketch, refer to Figure 3-76 for the sketch.

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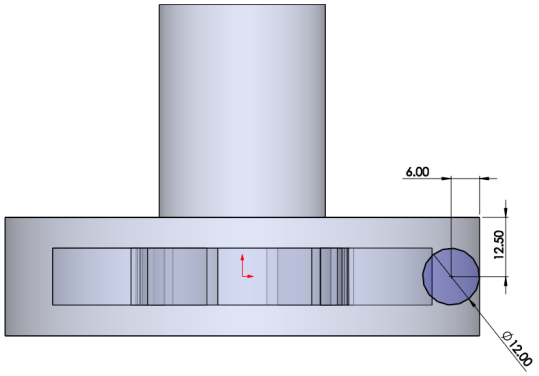


Figure 3-76 *Sketch after applying dimensions*

Creating the Outlet Extruded Feature

Next, you need to create legs of the body using the **Extruded Boss/Base** tool.

1. Choose the **Features** tab from the CommandManager to display the **Features CommandManager**. Then, choose the **Extruded Boss/Base** button; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed.

2. Select **Up To Body** option from the **End Condition** drop-down in the **Direction 1** rollout and then select the casing from the drawing area which will be selected in Solid/Surface Body area in the **Direction 1** rollout. Then, clear the **Auto-Select** check box in the **Feature Scope** area and then select the casing from the drawing area again. Choose the **OK** button to extrude the sketch. Refer to Figure 3-77 for the model. Ensure that the **Merge result** check box is selected.

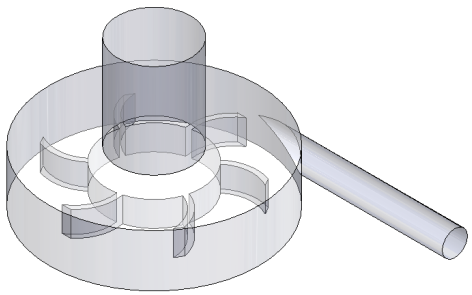


Figure 3-77 *Model created by extruding the sketch*

Creating the Shell Feature

It is evident from Figure 3-66 that a shell feature is required to create a thin-walled structure.

1. Choose the **Shell** button from the **Features CommandManager**; the **Shell PropertyManager** is displayed and you are prompted to select the faces to be removed.

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2. Select the inlet and outlet faces of the model; the selected face is displayed in the **Faces to Remove** selection box, refer to Figure 3-78.

3. Set the value to **1.2** mm in the **Thickness** spinner of the PropertyManager. Choose the **OK** button from the **Shell PropertyManager**.

The model after creating the shell feature is shown in Figure 3-79.

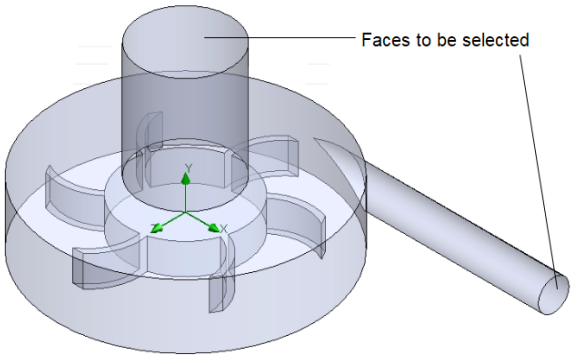


Figure 3-78 *Faces selected to be removed*

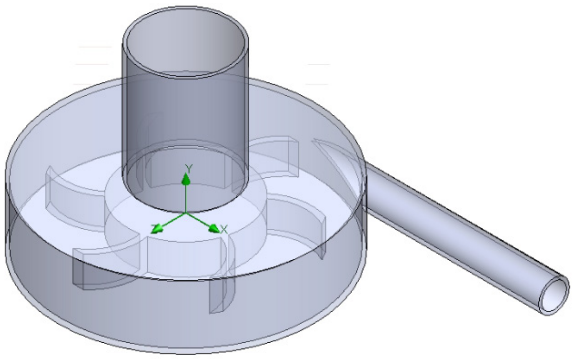


Figure 3-79 *Model after creating the shell feature*

Creating the Rotating Region

Next, you need to create the rotating region by creating and extruding the sketch.

1. Create the sketch on the Top Plane, as shown in Figure 3-80.

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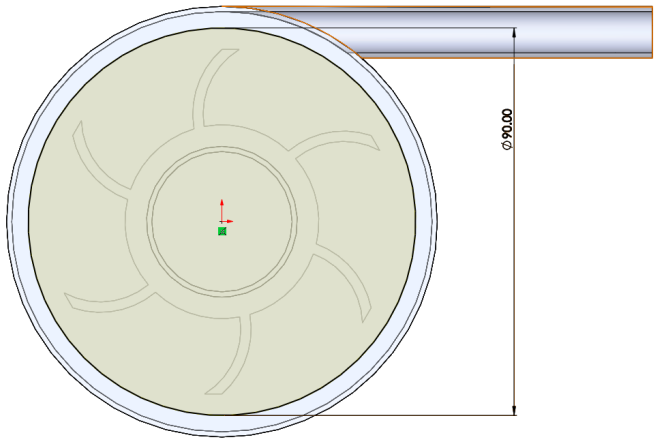


Figure 3-80 *Sketch after applying dimensions*

2. Create the solid model whose extrusion depth is **17** mm symmetrically, as shown in Figure 3-81.

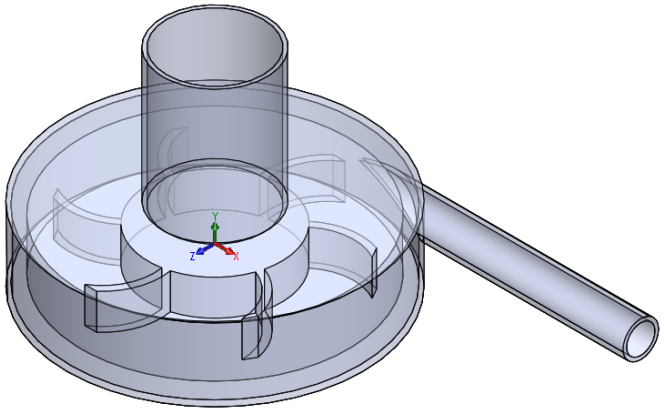


Figure 3-81 *The Solid model of rotating region*

Creating the Lids by Using the Extruded Feature

Next, you need to create Lid 1 and Lid 2. It will help you to specify the inlet and outlet conditions in flow simulation environment.

1. Invoke the **Extruded Boss/Base** tool, select the inlet face as the sketching plane, and create the sketch using the standard sketching tools. The sketch consists of a circle having 32.60 mm diameter.

2. Exit the sketching environment; the **Boss-Extrude PropertyManager** is displayed.

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3. Set **1** in the **Depth** spinner and clear the **Merge Result** check box. Choose the **OK** button from the **Boss-Extrude PropertyManager**; the selected contour is extruded, refer to Figure 4-82.

4. Similarly, create Lid 2 for outlet, refer to Figure 4-83.

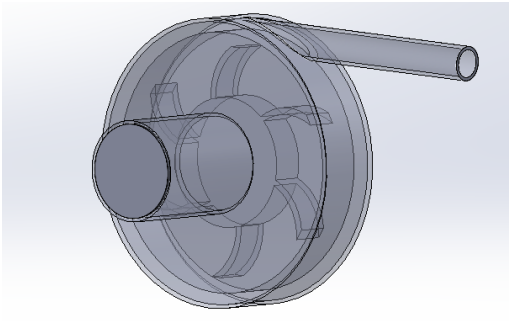


Figure 3-82 *Creating Lid 1 at inlet*

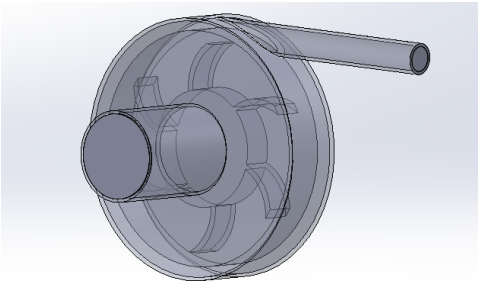


Figure 3-83 *Creating Lid 2 at outlet*

Next, rename the tree items to Lid 1, and Lid 2.

5. Right-click on **Boss-Extrude6** and select the **Rename tree item** option. Type the name **Lid 1**; the name is updated in the tree.

   Similarly, you can rename the other extruded feature as Lid 2.

Saving the Model

1. Save the part document with the name **c03_tut04** at the following location: *\Documents\SOLIDWORKS\c03*

2. Choose **File > Close** from the SOLIDWORKS menus to close the document.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The _____ tool is used to extrude a sketch in SOLIDWORKS.

2. The _____ tool is used to revolve a sketch in SOLIDWORKS.

3. The _____ option is used to create the base feature by extruding the sketch equally in both the directions of the plane on which the sketch is drawn.

4. The _____ button is used to specify a draft angle while extruding a sketch.

5. The Extruded Boss/Base tool is also used to extrude the open profile. (T/F)

6. The thin extruded features can be created using a closed sketch. (T/F)

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Review Questions

Answer the following questions:

1. Which of the following PropertyManagers is displayed when you choose the **Extruded Boss/Base** tool from the **Features CommandManager**?

   (a) **Boss** (b) **Extrude**
   (c) **Boss-Extrude** (d) None of these

2. Which of the following PropertyManagers is displayed on the left of the drawing area when you choose the **Swept Boss/Base** tool from the **Features CommandManager**?

   (a) **Swept** (b) **Boss**
   (c) **Sweep** (d) None of these

3. Which of the following options is used to create the extrude thickness?

   (a) **Thin Feature** (b) **Thin**
   (c) **Shell** (d) None of these

4. You cannot scoop out material from the model using the **Shell** tool. (T/F)

5. You can create shell features with different thickness of different faces. (T/F)

EXERCISES

Exercise 1

Create the solid model of the sphere shown in Figure 3-82. The sketch of the sphere is shown in Figure 3-83. **(Expected time: 30 min)**

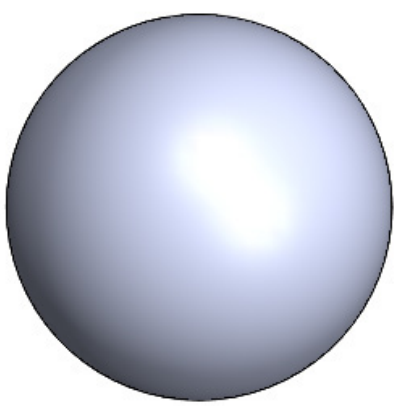


Figure 3-84 *Solid model for Exercise 1*

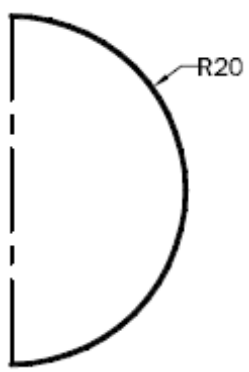


Figure 3-85 *Sketch for Exercise 1*

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Exercise 2

Create the solid model of the cylinder shown in Figure 3-84. The sketch of the cylinder is shown in Figure 3-85. **(Expected time: 10 min)**

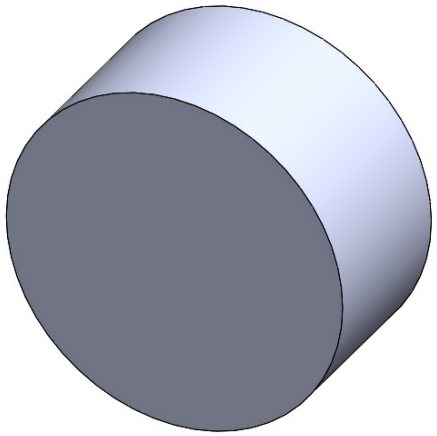


Figure 3-86 *Solid model for Exercise 2*

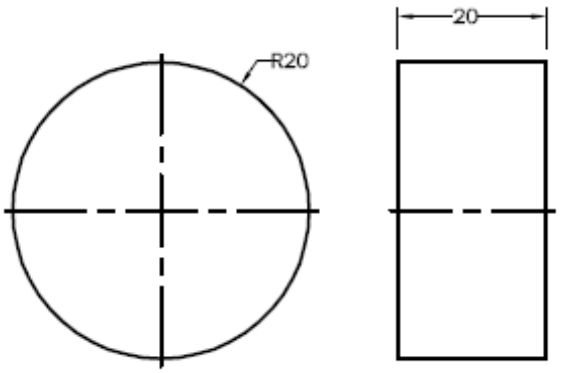


Figure 3-87 *Sketch for Exercise 2*

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Answers to Self-Evaluation Test

1. Extruded Boss/Base, **2. Revolved Boss/Base**, **3. Mid Plane**, **4. Draft On/Off**, **5.** T, **6.** T