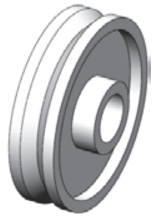
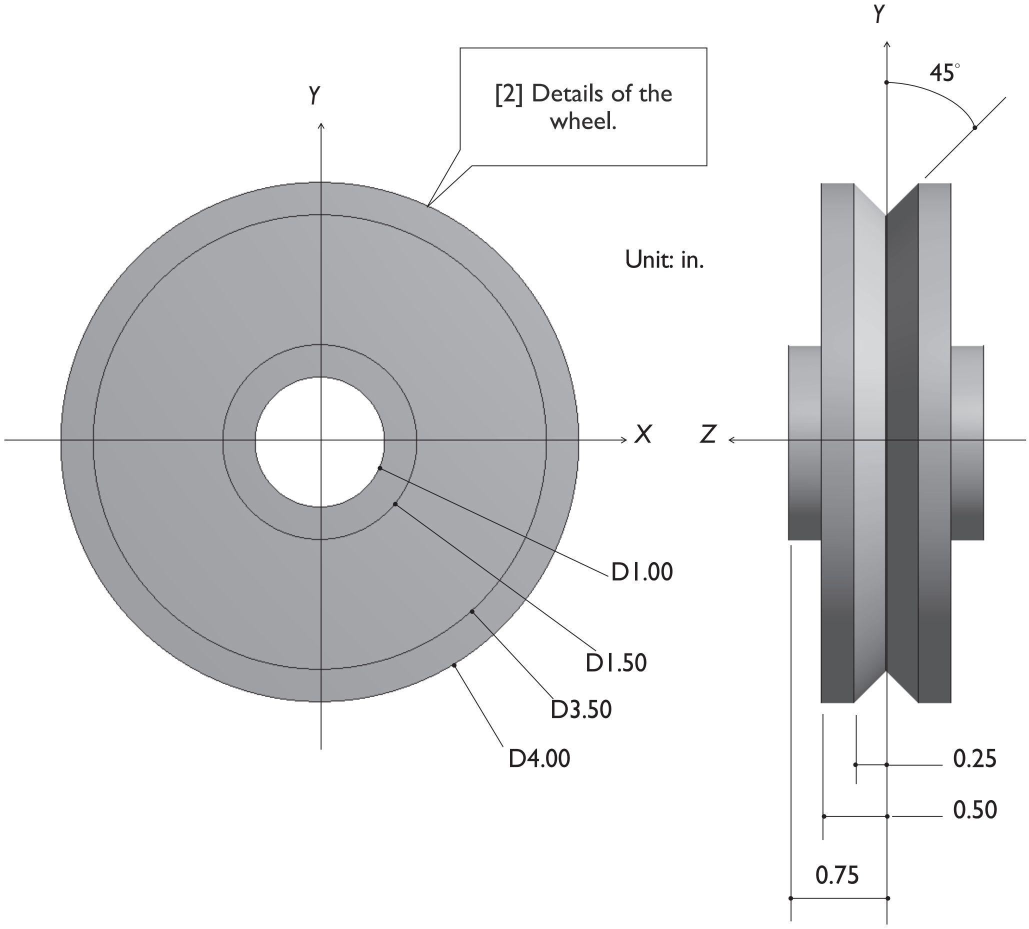
Section 2.5

Wheel



2.5-1 About the Wheel

[1] So far, we exclusively used **Extrude** command to create 3D solids. In this section, we introduce another command to create 3D solids: **Revolve**, which takes a sketch as the profile and revolves about an axis to create a 3D solid body. We'll create a 3D solid model for a wheel [2]. The wheel is axisymmetric. An axisymmetric body always can be created by drawing a **profile** then revolving about its **axis** to generate the 3D solid body.

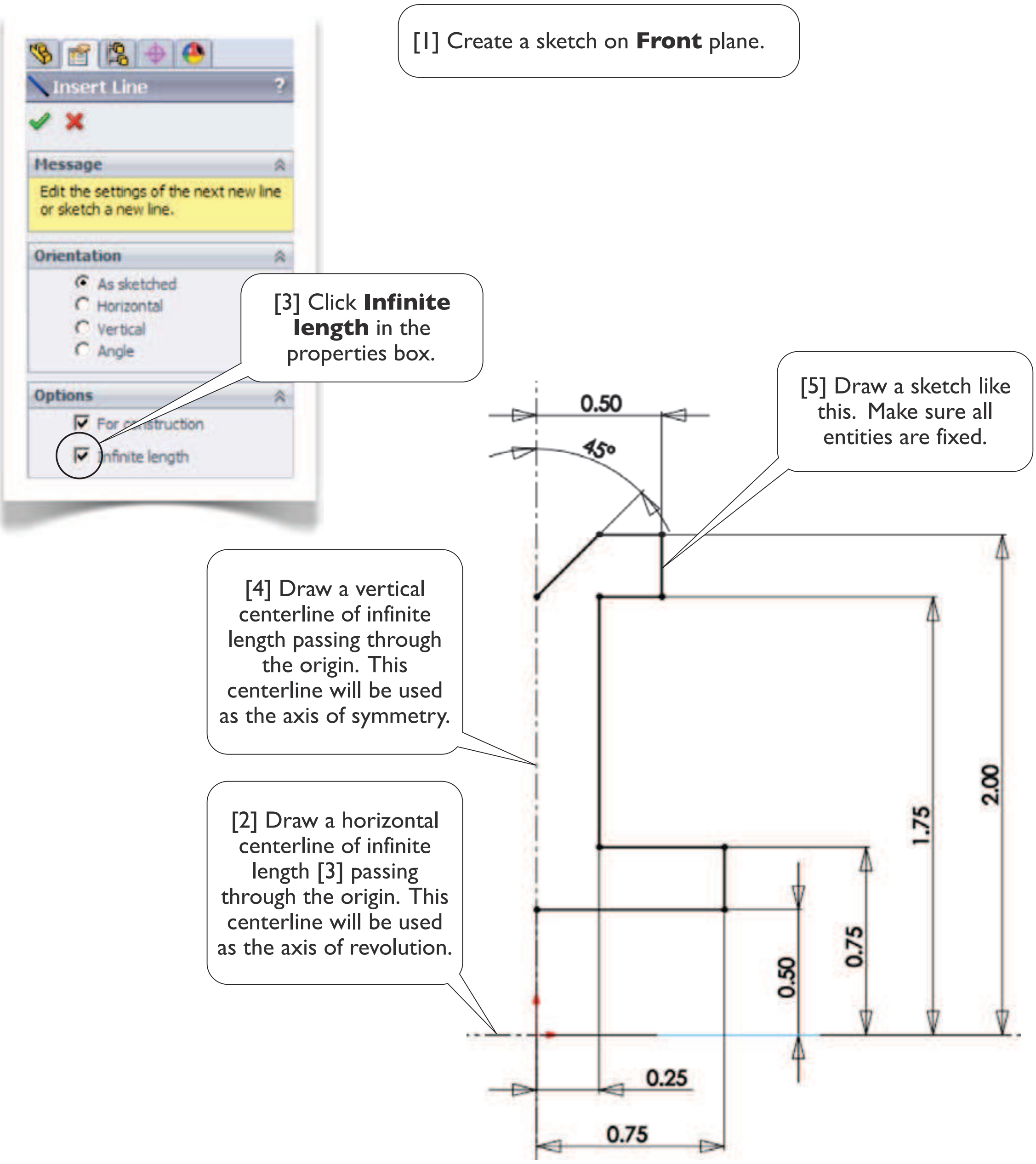


2.5-2 Start Up

[1] Launch **SolidWorks** and create a new part. Set up **IPS** unit system with 2 decimal places for the length unit.

2.5-3 Create a Sketch for the Profile

[1] Create a sketch on **Front** plane.

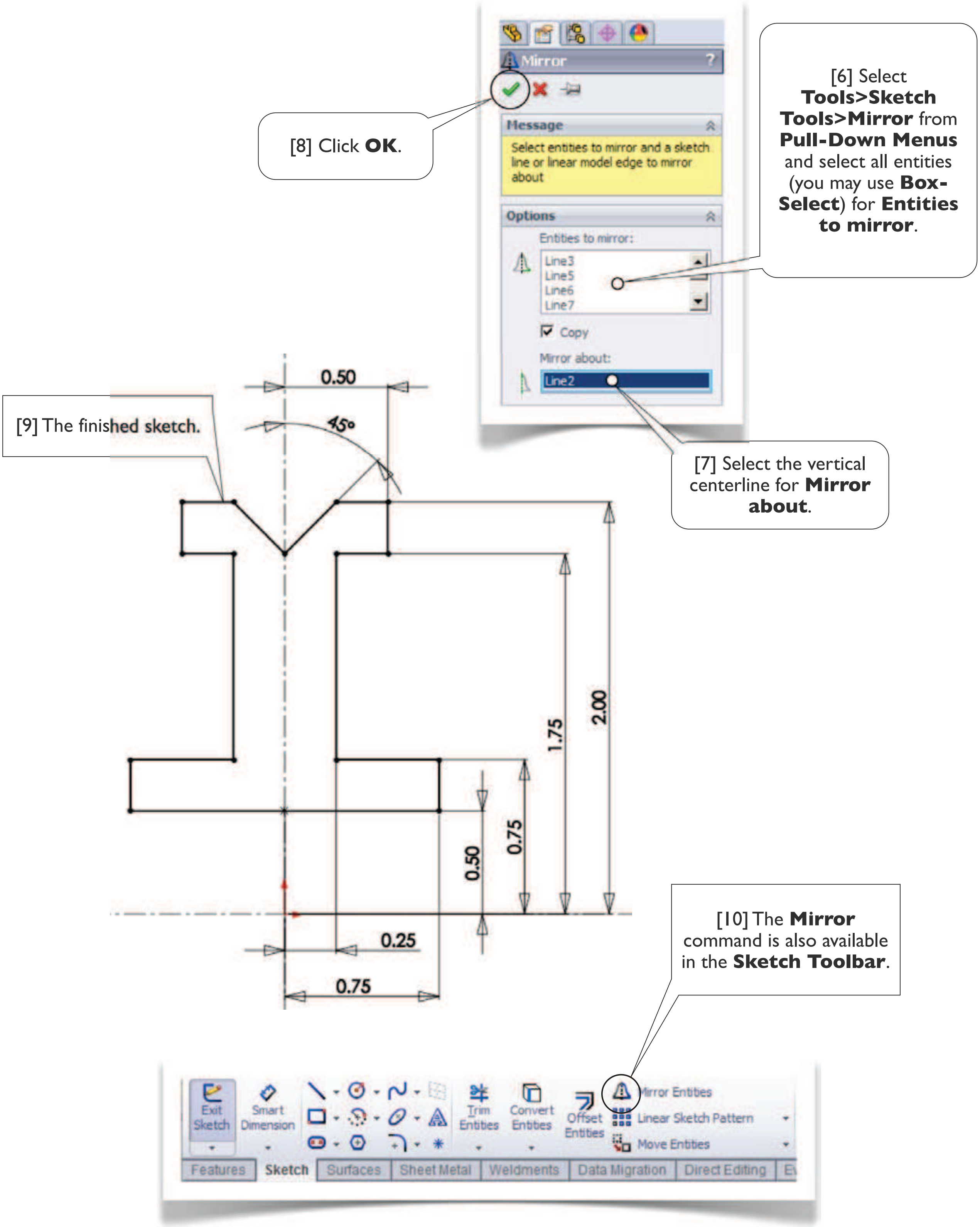


[2] Draw a horizontal centerline of infinite length [3] passing through the origin. This centerline will be used as the axis of revolution.

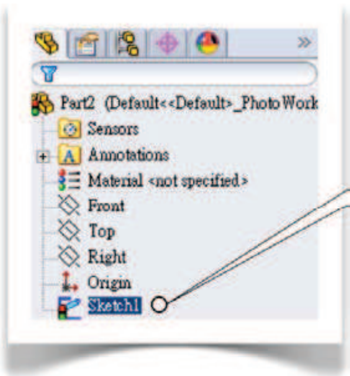
[3] Click **Infinite length** in the properties box.

[4] Draw a vertical centerline of infinite length passing through the origin. This centerline will be used as the axis of symmetry.

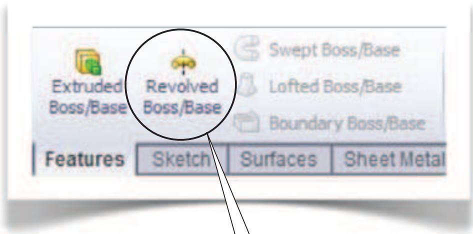
[5] Draw a sketch like this. Make sure all entities are fixed.


Mirror
Message
Select entities to mirror and a sketch line or linear model edge to mirror about
Options
Entities to mirror:
Line3
Line5
Line6
Line7
Copy
Mirror about:
Line2
[6] Select **Tools>Sketch Tools>Mirror** from **Pull-Down Menus** and select all entities (you may use **Box-Select**) for **Entities to mirror**.
[8] Click **OK**.
[7] Select the vertical centerline for **Mirror about**.
[9] The finished sketch.
0.50
45°
2.00
1.75
0.75
0.50
0.25
0.75
[10] The **Mirror** command is also available in the **Sketch Toolbar**.
Exit Sketch
Smart Dimension
Trim Entities
Convert Entities
Offset Entities
Mirror Entities
Linear Sketch Pattern
Move Entities
Features
Sketch
Surfaces
Sheet Metal
Weldments
Data Migration
Direct Editing

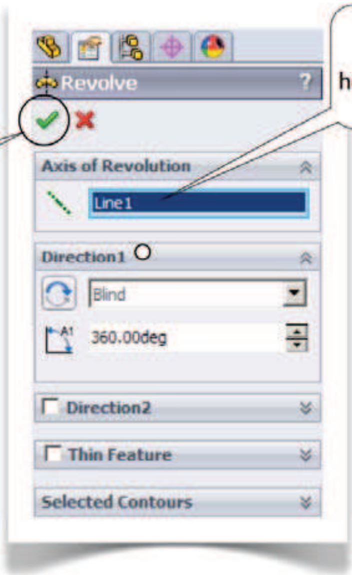
2.5-4 Revolve the Sketch



[1] While the sketch is highlighted, select **Insert>Boss/ Base>Revolve...**



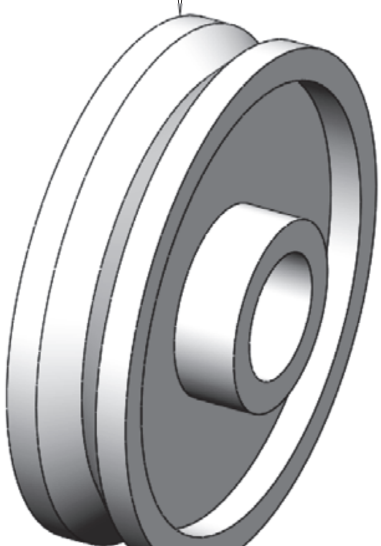
[2] The **Revolve** command is also available in the **Features Toolbar**.



[3] Select the horizontal centerline.

[4] Click **OK**.

[5] The finished 3D model.



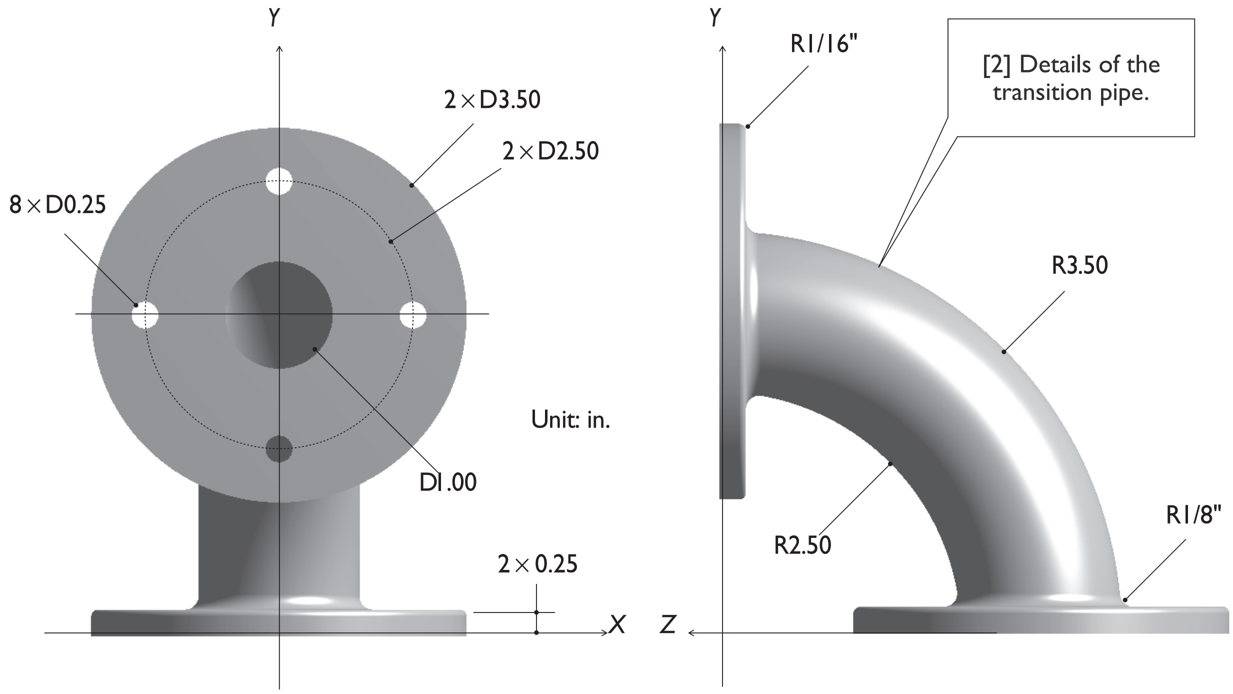
[6] Save the part with the file name **Wheel**. Close the file and exit **SolidWorks**.

Section 2.6

Transition Pipe

2.6-1 About the Transition Pipe

[1] In this section, we introduce another command to create 3D solids: **Sweep**, which takes a sketch as the **path** and another sketch as the **profile**; the **profile** then "sweeps" along the **path** to create a 3D solid body. In this exercise, we'll create a 3D solid model for a transition pipe, which is used to connect two pipe segments.



2.6-2 Start Up

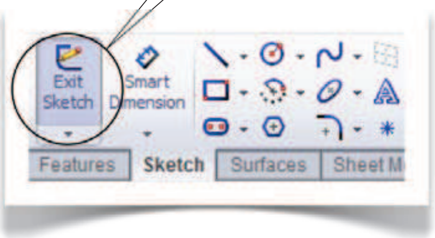
[1] Launch **SolidWorks** and create a new part. Set up **IPS** unit system with 2 decimal places for the length unit.

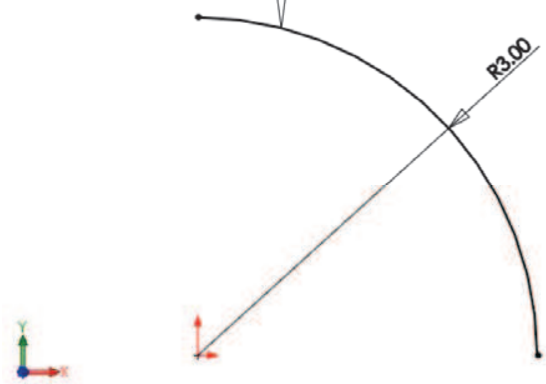
2.6-3 Create a Sketch for the **Path**

[1] Create a sketch on **Front** plane.

[2] Draw a sketch like this. This sketch will be used as a sweeping **path**. Note that, each end point aligns with the origin either vertically or horizontally.

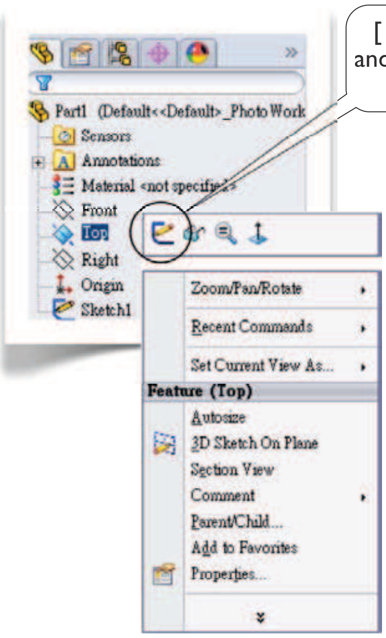
[3] Click **Exit Sketch** in the **Sketch Toolbar**.





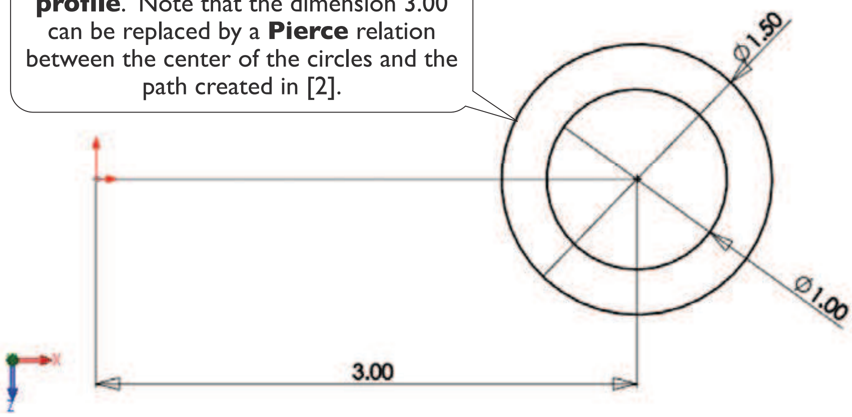
2.6-4 Create a Sketch for the **Profile**

[1] Right-click **Top** plane and select **Sketch** to create a second sketch.



[2] In the **Standard Views Toolbar**, click **Normal To**.

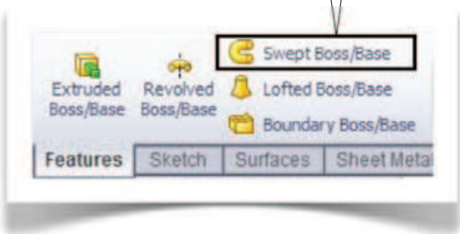
[3] Draw two concentric circles like this. This sketch will be used as a sweeping **profile**. Note that the dimension 3.00 can be replaced by a **Pierce** relation between the center of the circles and the path created in [2].

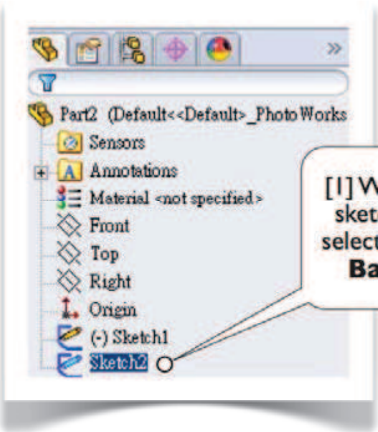


[4] Click Exit Sketch in the **Sketch Toolbar**.

2.6-5 Create the Curved Pipe

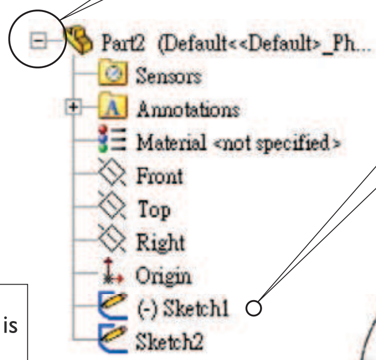
[2] The **Sweep** command is also available in the **Features Toolbar**.





[1] While the **Profile** sketch is highlighted, select **Insert>Boss/Base>Sweep...**

[5] In the **Graphics Area**, click "+" sign to expand the **Features Tree**; the "+" sign becomes "-" sign.

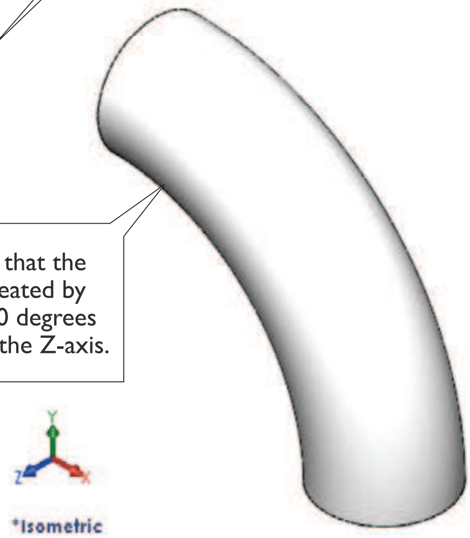


[6] Select the **Path** sketch.

[7] Click **OK**.

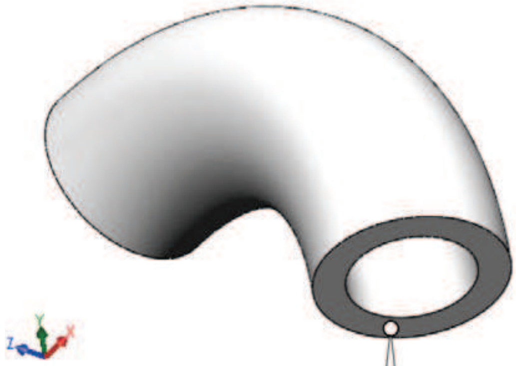
[3]The **profile** sketch is pre-selected.

[4] Click to activate **Path** box.

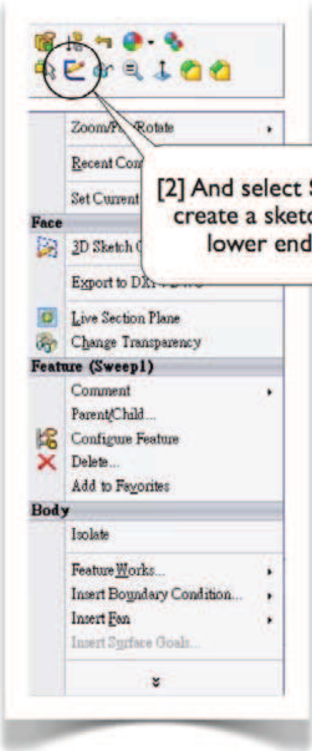


[8]The curved pipe. Note that the curved pipe also can be created by **Revolving** the **Profile** 90 degrees with an axis coincident with the Z-axis.

2.6-6 Create the Lower End Plate



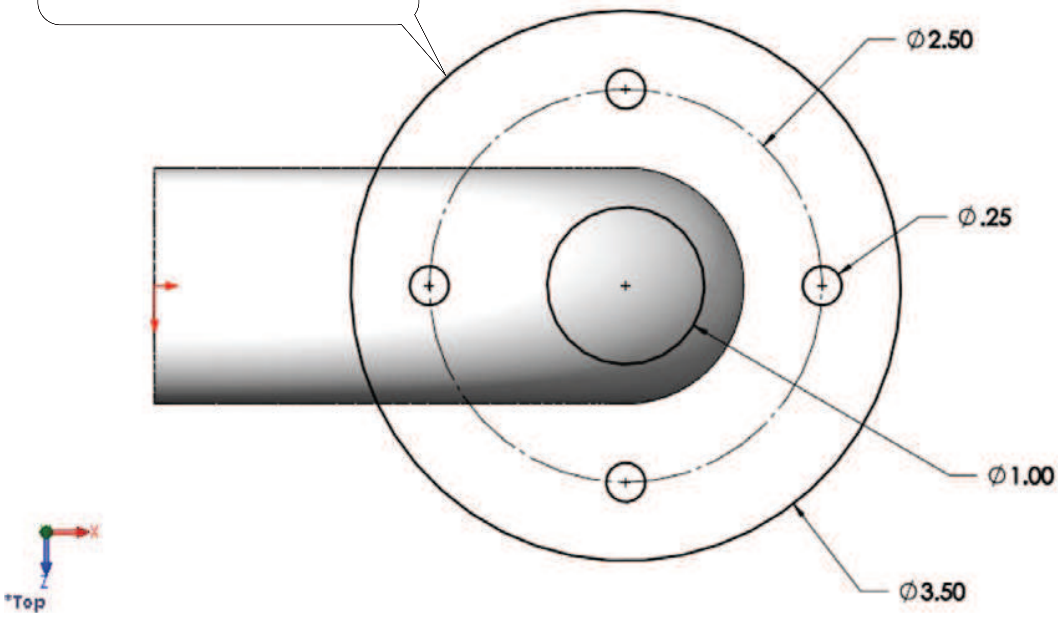
[1] Right-click the bottom face of the curved pipe.



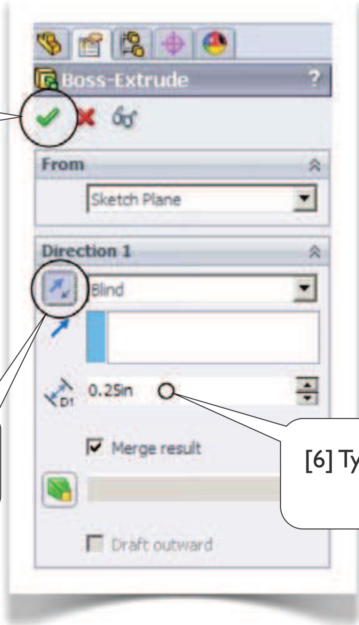
[2] And select **Sketch** to create a sketch for the lower end plate.

[3] In the **Standard Views Toolbar**, click **Normal To**.

[4] Draw a sketch like this. The sketch consists of 7 circles, including a construction circle of diameter 2.5 inches and four circles of diameter 0.25 inches.



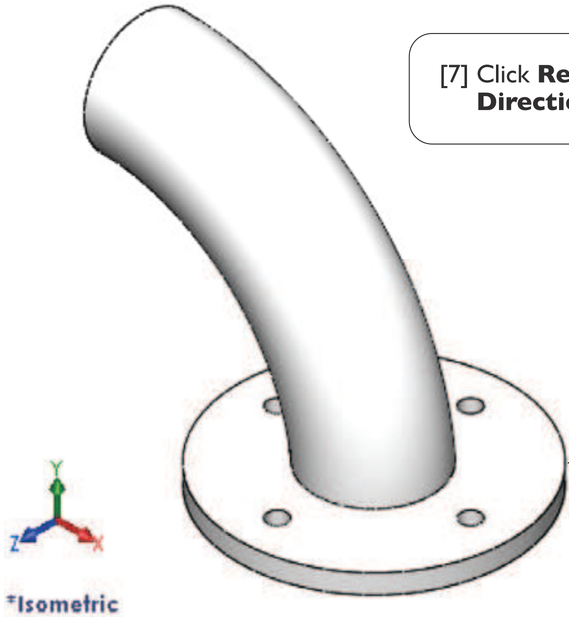
[5] In the **Features Toolbar**, click **Extruded Boss/Base**.



[8] Click **OK**.

[7] Click **Reverse Direction**.

[6] Type 0.25 (in) for **Depth**.

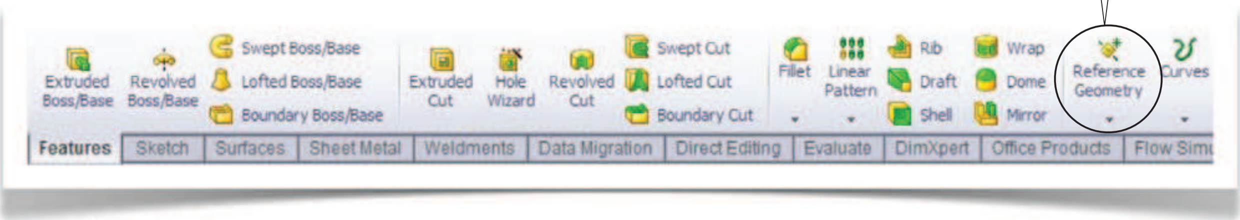


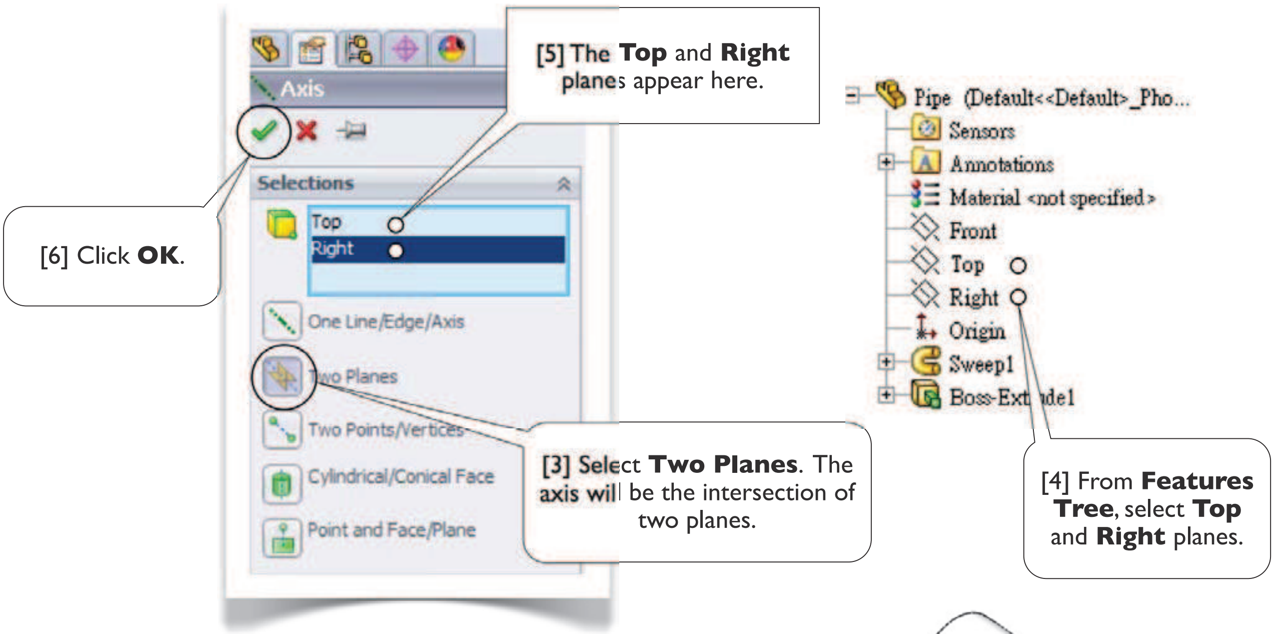
[9] The lower end plate.

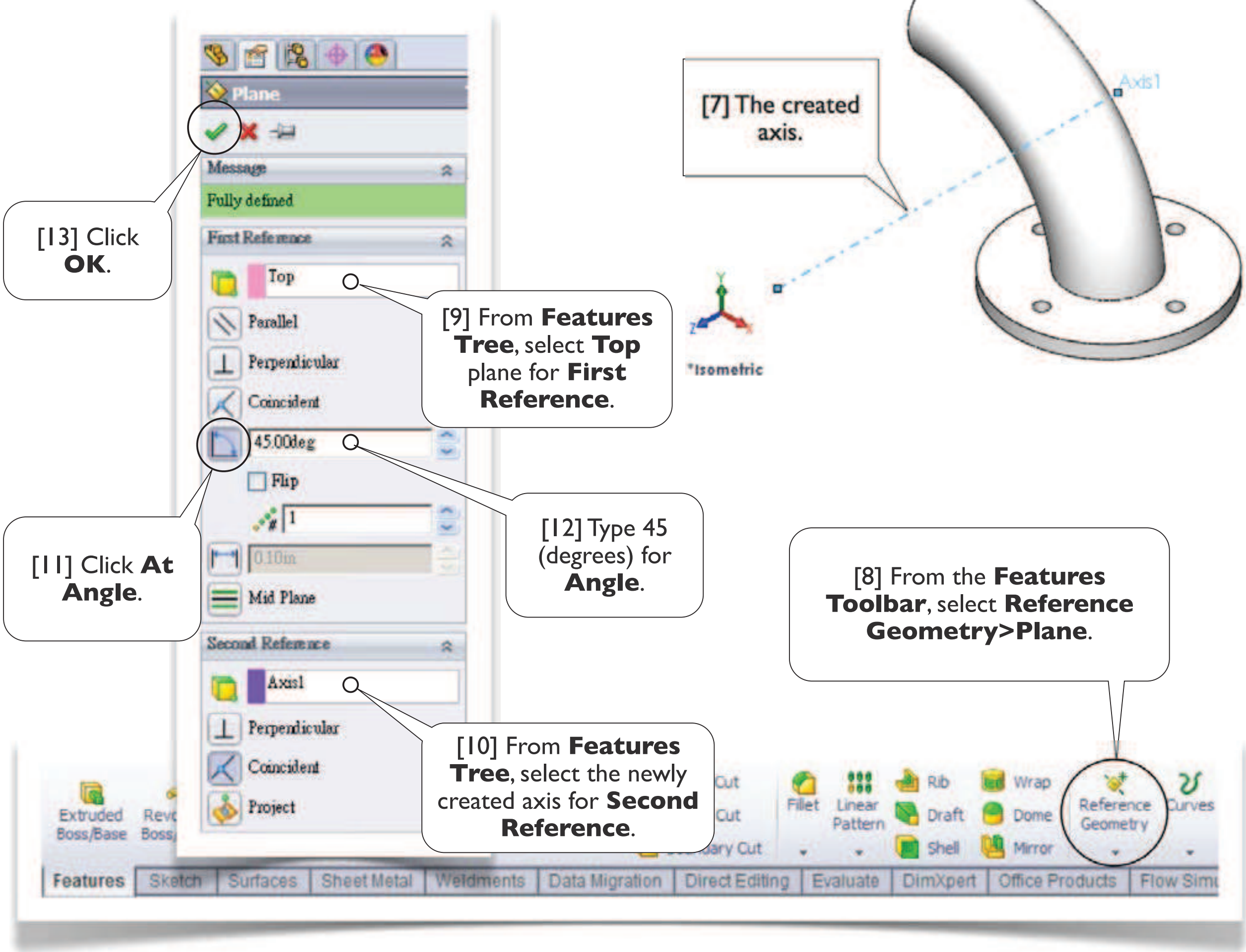
2.6-7 Create a Mirroring Plane

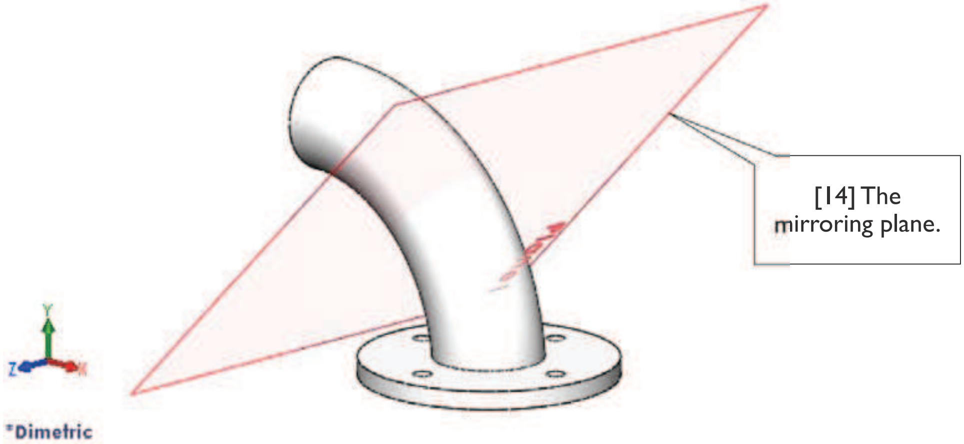
[1] Next, we want to create the upper end plate by using **Mirror** command. The mirroring plane will be created by rotating the **Top** plane 45 degrees about an axis coincident with the Z-axis. First, we create the axis.

[2] From **Features Toolbar**, select **Reference Geometry>Axis**.

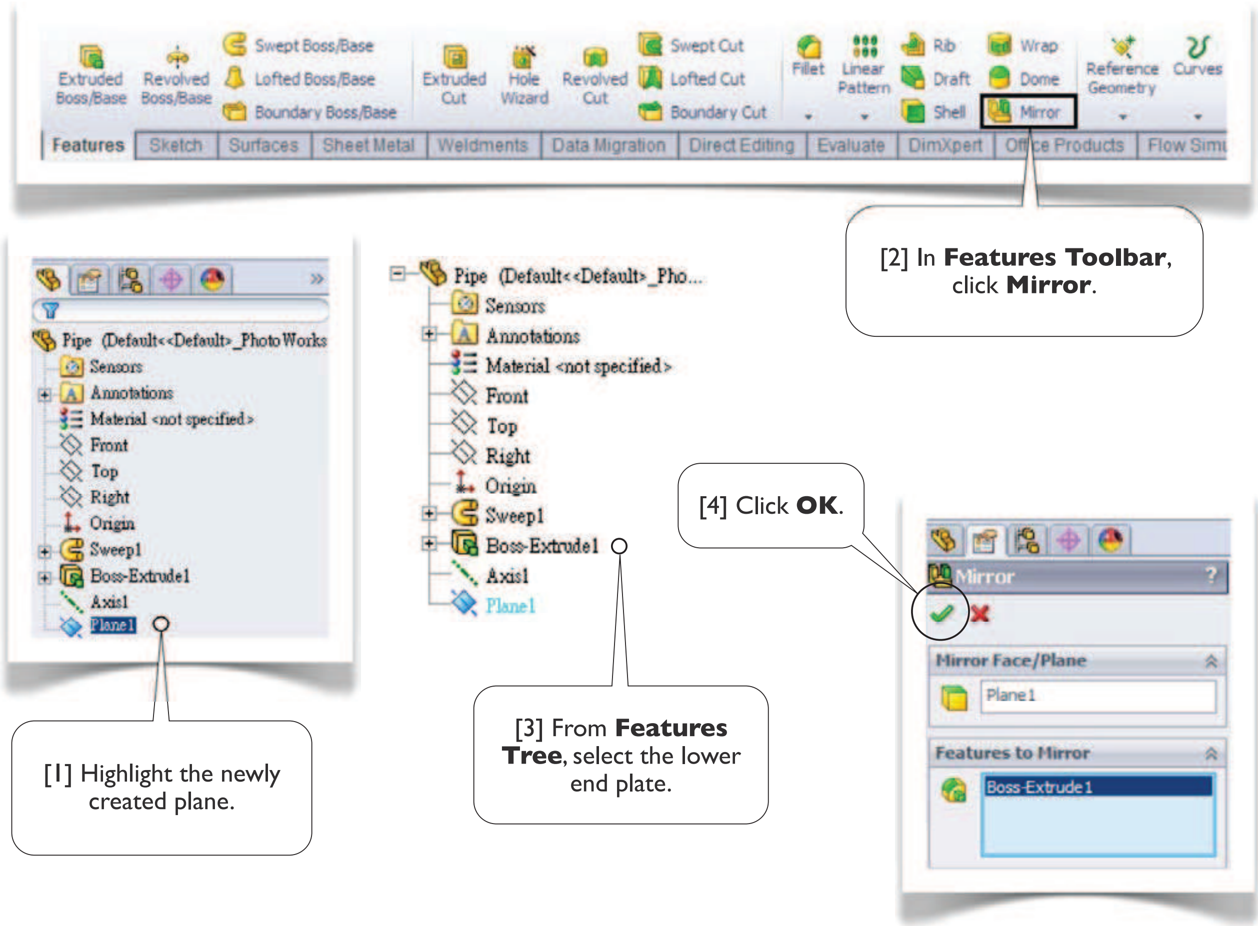


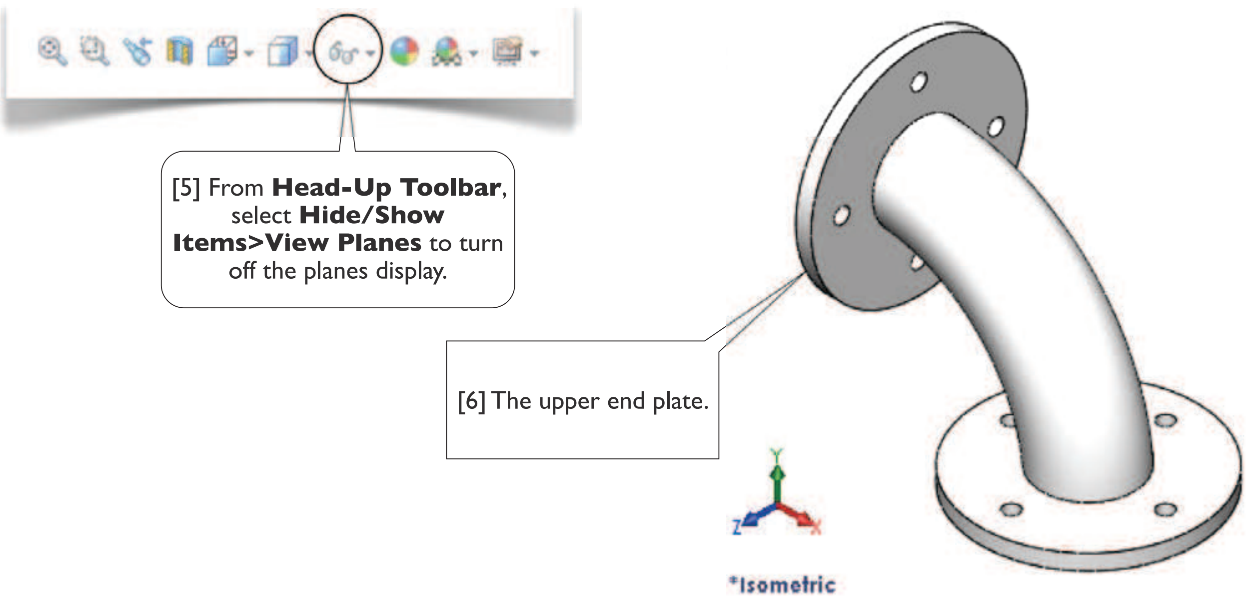

[5] The **Top** and **Right** planes appear here.
[6] Click **OK**.
Axis
Selections
Top
Right
One Line/Edge/Axis
Two Planes
Two Points/Vertices
Cylindrical/Conical Face
Point and Face/Plane
[3] Select **Two Planes**. The axis will be the intersection of two planes.
Pipe (Default<<Default>_Pho...
Sensors
Annotations
Material <not specified>
Front
Top
Right
Origin
Sweep1
Boss-Extrude1
[4] From **Features Tree**, select **Top** and **Right** planes.


Plane
Message
Fully defined
First Reference
Top
Parallel
Perpendicular
Coincident
45.00deg
Flip
1
0.10in
Mid Plane
Second Reference
Axis1
Perpendicular
Coincident
Project
[13] Click **OK**.
[11] Click **At Angle**.
[9] From **Features Tree**, select **Top** plane for **First Reference**.
[12] Type 45 (degrees) for **Angle**.
[10] From **Features Tree**, select the newly created axis for **Second Reference**.
[7] The created axis.
Axis1
*Isometric
[8] From the **Features Toolbar**, select **Reference Geometry>Plane**.
Extruded Boss/Base
Fillet
Linear Pattern
Rib
Draft
Shell
Wrap
Dome
Mirror
Reference Geometry
Curves
Features
Sketch
Surfaces
Sheet Metal
Weldments
Data Migration
Direct Editing
Evaluate
DimXpert
Office Products

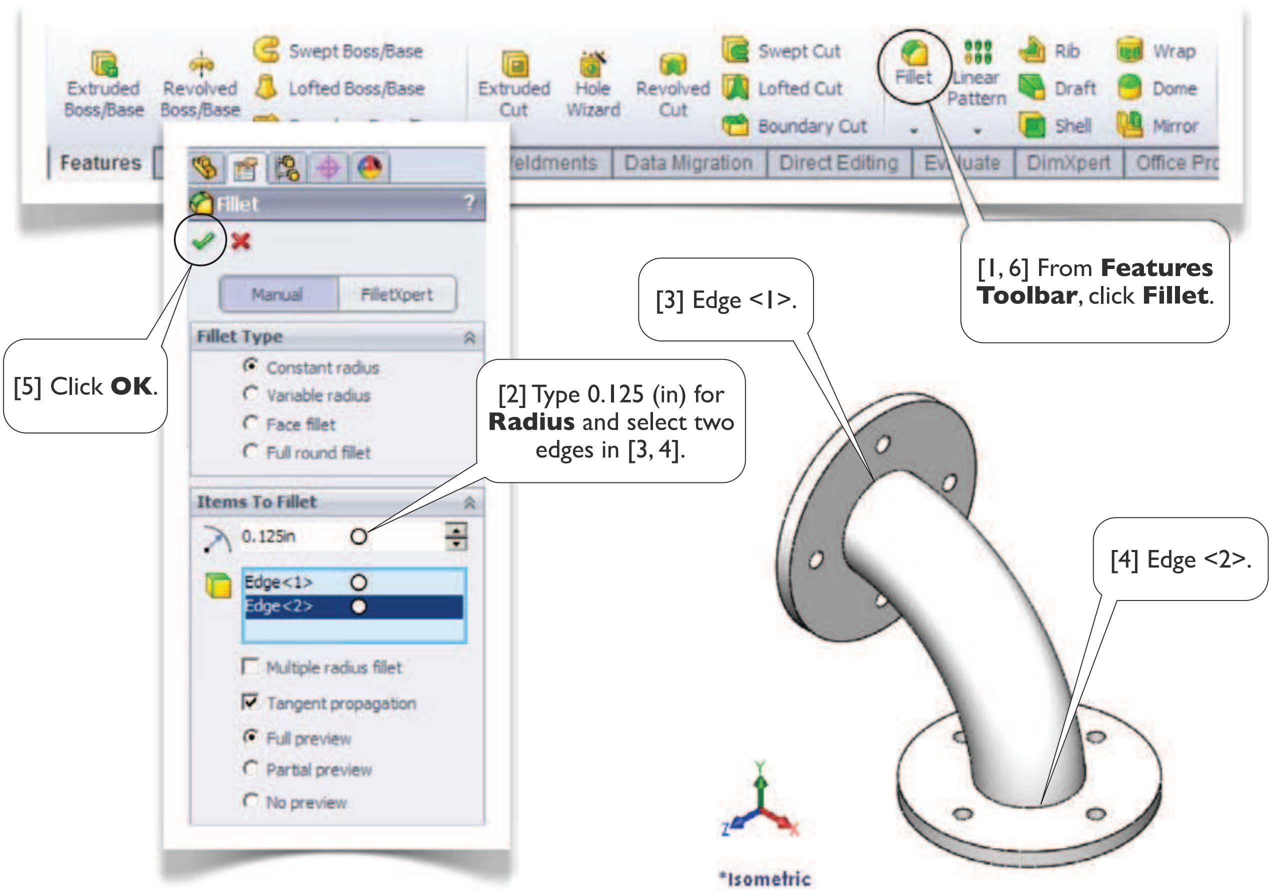


2.6-8 Create the Upper End Plate

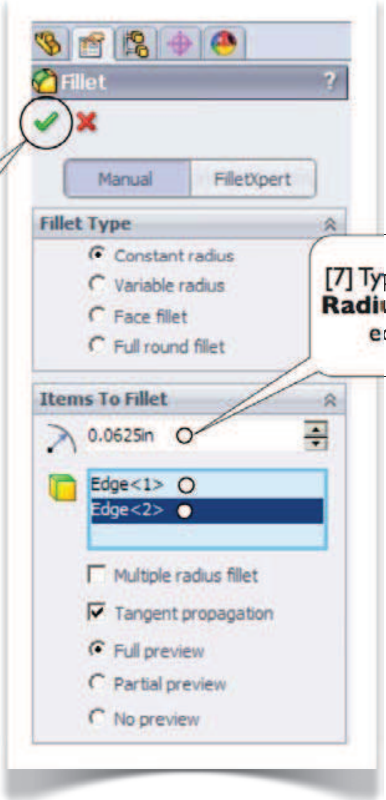




2.6-9 Create Fillets and Rounds

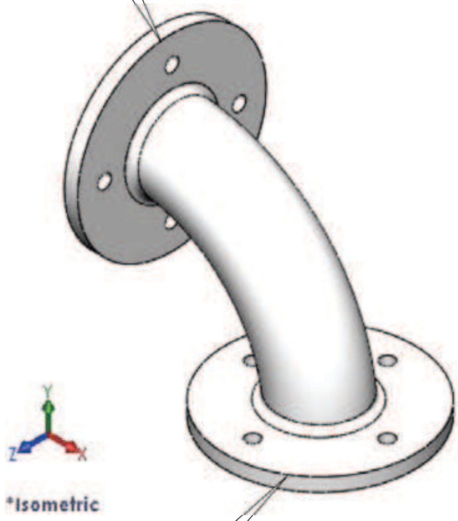


[10] Click **OK**.

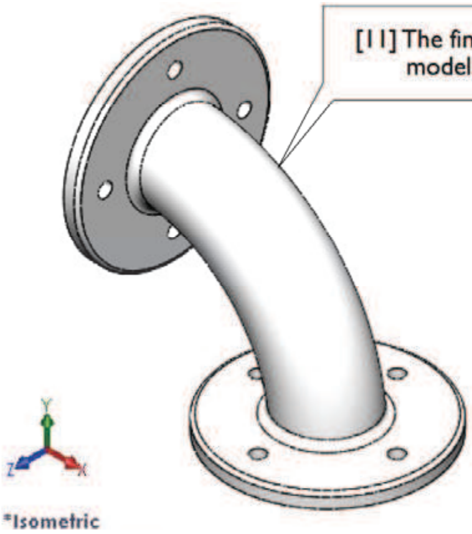


[7] Type 0.0625 (in) for **Radius** and select two edges in [8, 9].

[8] Edge <1>.



[9] Edge <2>.



[11] The finished model.

[12] From **Head-Up Toolbar**, select **Display Style>Shaded** to remove the edge display.

[13] Shaded model without edge display.

[14] Save the part with the file name **Pipe**. Close the file and exit **SolidWorks**.

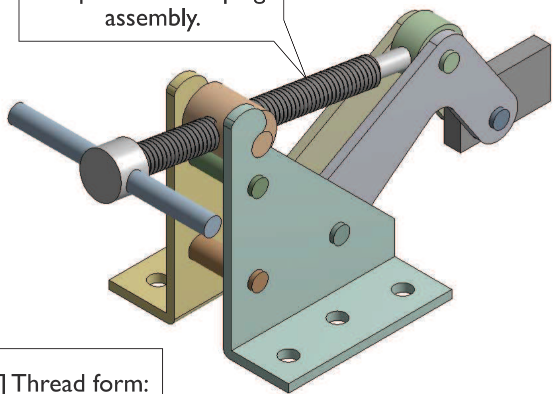
Section 2.7

Threaded Shaft

2.7-1 About the Threaded Shaft

[1] The threaded shaft is a part of the clamping mechanism mentioned in Sections 1.1 and 2.4 [2]. In this exercise, we will create a 3D solid model for the threaded shaft.

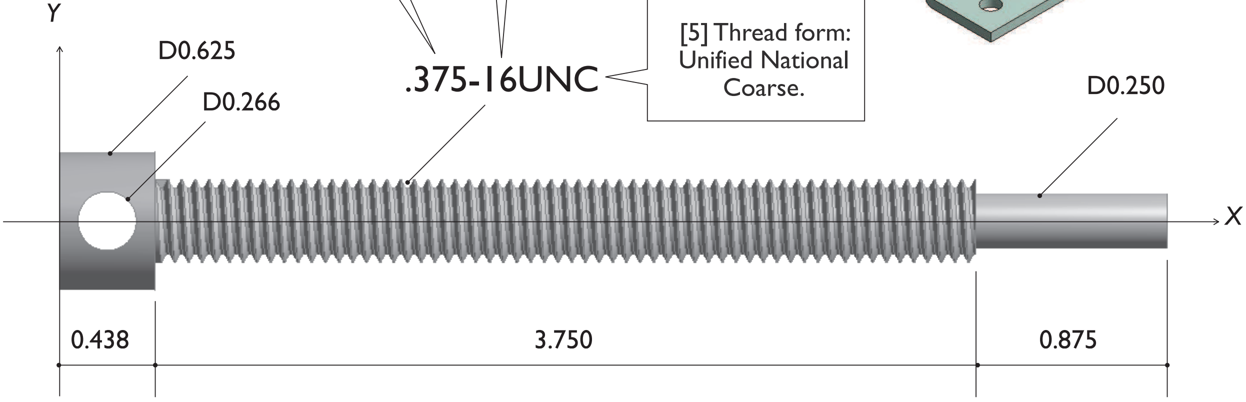
[2] The threaded shaft is a part of a clamping assembly.



[3] Major diameter d = .375 in.

[4] Pitch p = 1/16 in.

[5] Thread form: Unified National Coarse.



[6] Details of the threads.

$d = 0.375$ in

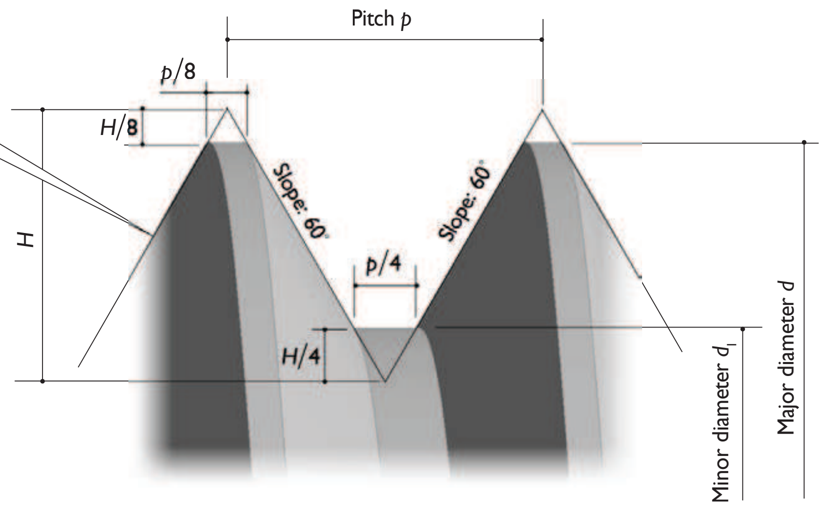
$p = 0.0625$ in

$H = (\sqrt{3}/2)p = 0.0541266$ in

$d_1 = d - \frac{5H}{8} \times 2 = 0.307342$ in

$\frac{p}{4} = 0.015625$ in

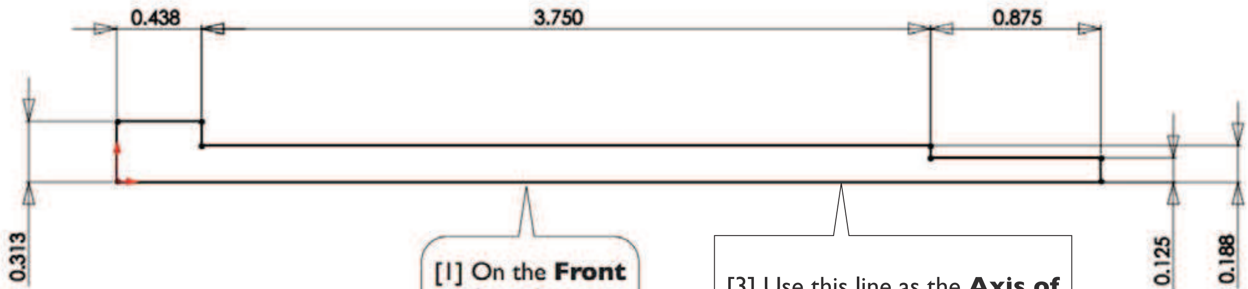
$\frac{p}{8} = 0.0078125$ in



2.7-2 Start Up

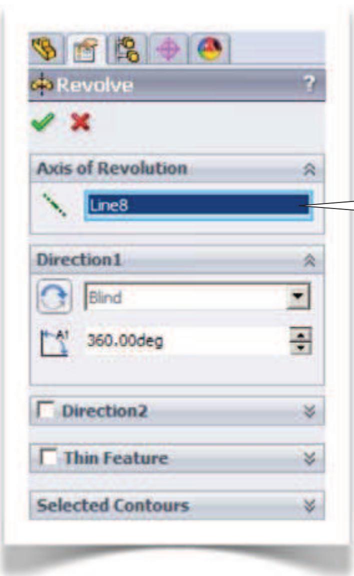
[1] Launch **SolidWorks** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

2.7-3 Create a Shaft Base



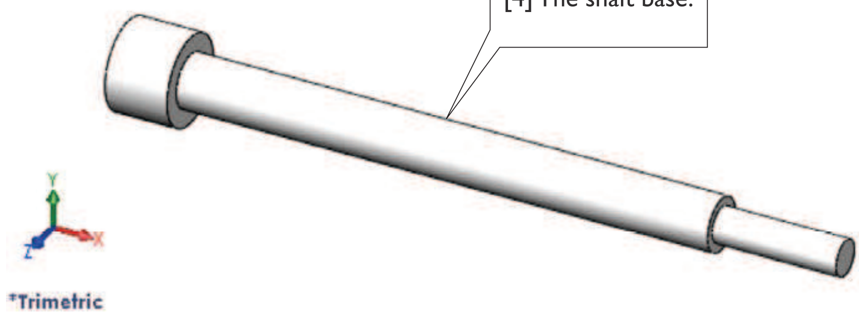
[1] On the **Front** plane, draw a sketch like this.

[3] Use this line as the **Axis of Revolution**.

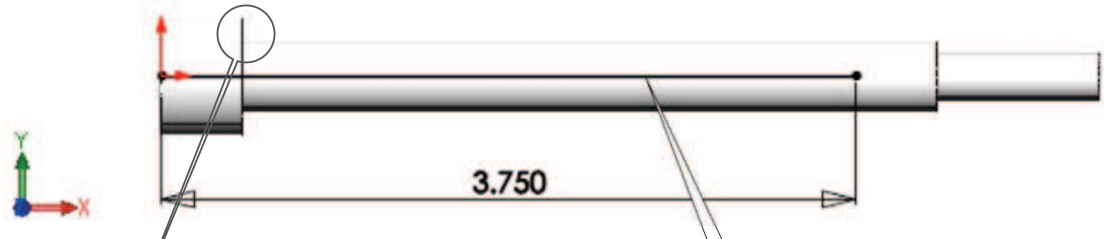


[2] **Revolve** the sketch 360 degrees to create the shaft base. Use the bottom line of the sketch as **Axis of Revolution** [3].

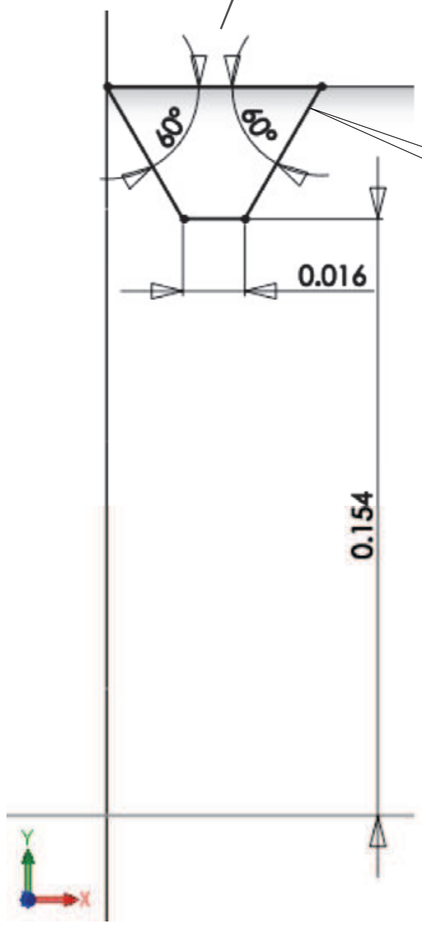
[4] The shaft base.



2.7-4 Create Threads



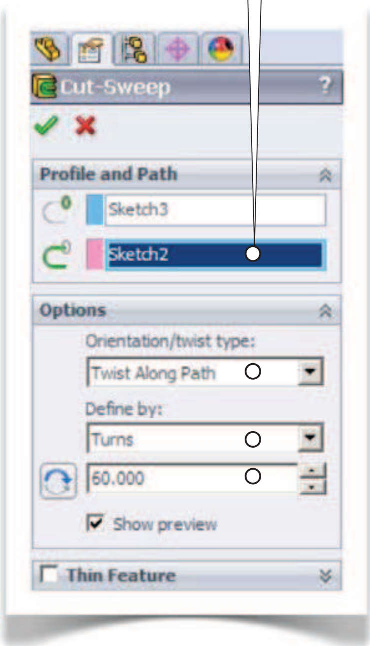
[1] On the **Front** plane, draw a single line of length 3.75 inches like this. Remember to click **Exit Sketch**. This sketch will be used as the sweeping **Path**.

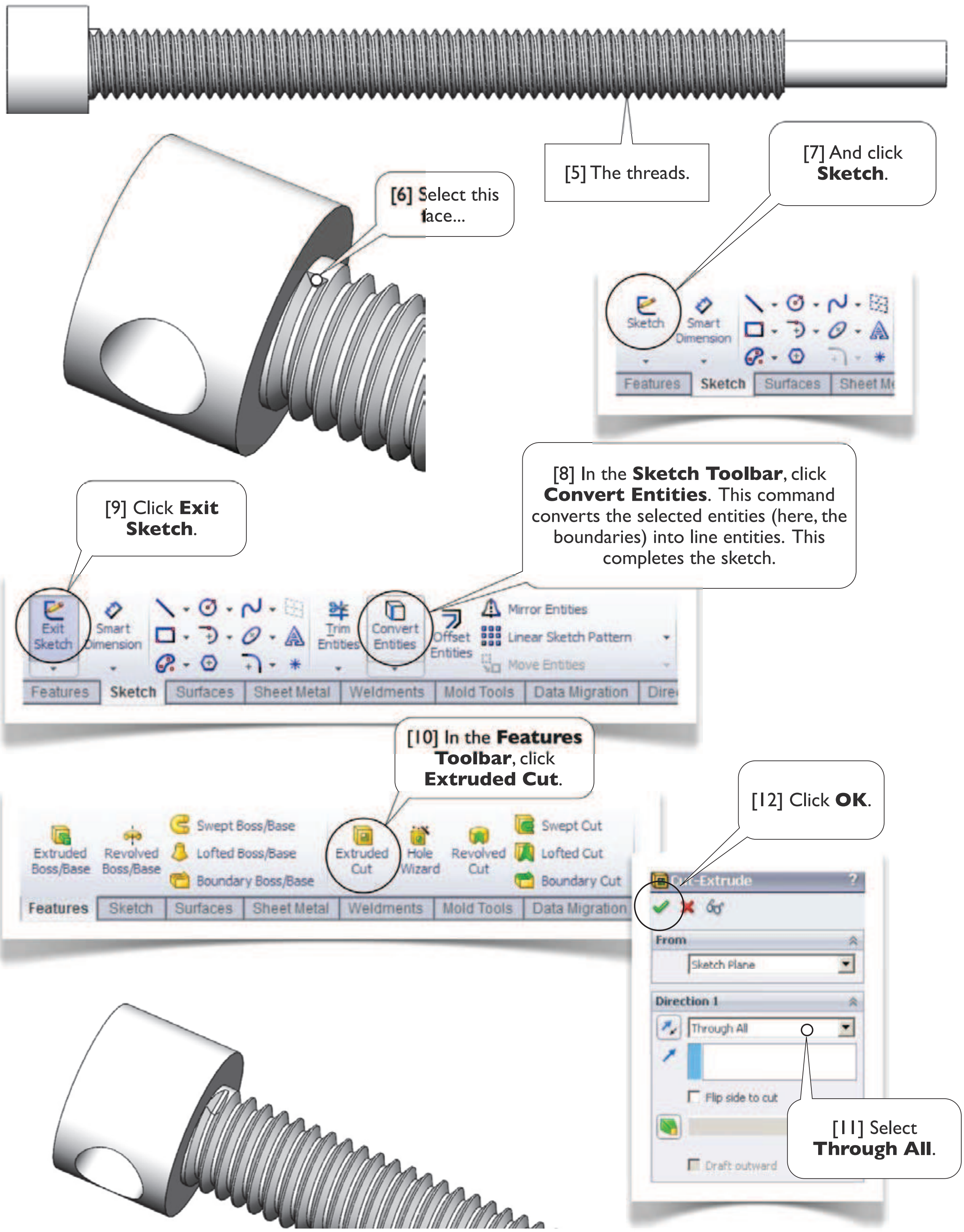


[2] On the **Front** plane, draw a sketch of trapezoid like this. Remember to click **Exit Sketch**. This sketch will be used as the sweeping **Profile**.

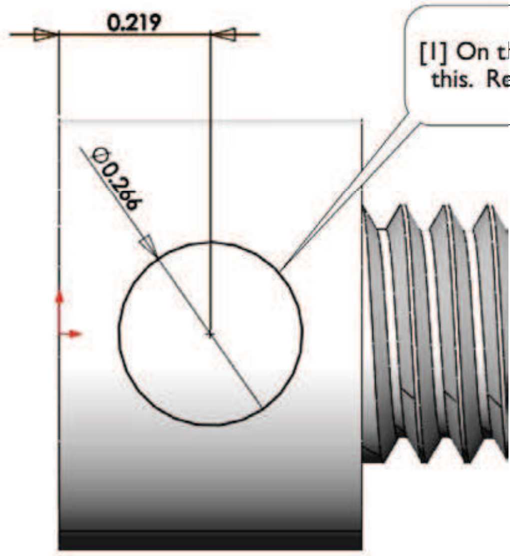
[3] With the **Profile** (**Sketch3**) highlighted, from **Features Toolbar**, click **Swept Cut**.

[4] Select the **Path** (**Sketch2**) and set up other parameters like this. Note that the number of turns (60) is calculated by 3.75/0.0625, where 0.0625 (in) is the thread pitch. Click **OK**.

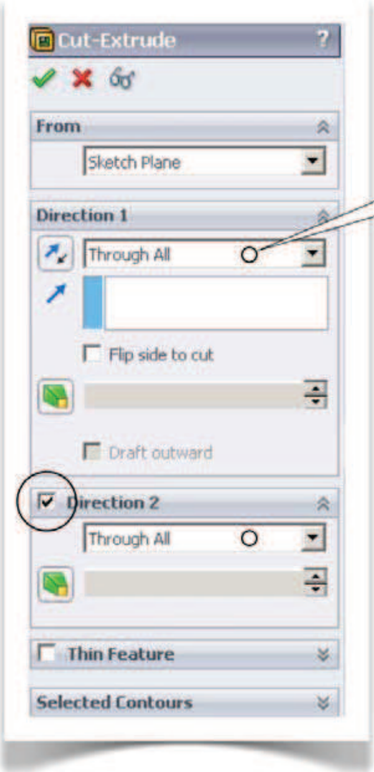



[5] The threads.
[6] Select this face...
[7] And click **Sketch**.
Sketch
Smart Dimension
Features
Sketch
Surfaces
[8] In the **Sketch Toolbar**, click **Convert Entities**. This command converts the selected entities (here, the boundaries) into line entities. This completes the sketch.
[9] Click **Exit Sketch**.
Exit Sketch
Smart Dimension
Trim Entities
Convert Entities
Offset Entities
Mirror Entities
Linear Sketch Pattern
Move Entities
Features
Sketch
Surfaces
Sheet Metal
Weldments
Mold Tools
Data Migration
[10] In the **Features Toolbar**, click **Extruded Cut**.
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Features
Sketch
Surfaces
Sheet Metal
Weldments
Mold Tools
Data Migration
[12] Click **OK**.
Cut-Extrude
From
Sketch Plane
Direction 1
Through All
Flip side to cut
Draft outward
[11] Select **Through All**.

2.7-5 Create a Hole



[1] On the **Front** plane, draw a circle like this. Remember to click **Exit Sketch**.



[2] With the sketch highlighted, from the **Features Toolbar**, select **Extruded Cut** and set up the parameters like this.

[3] The finished model.

[4] Save the part with the file name **Shaft**. Close the file and exit **SolidWorks**.