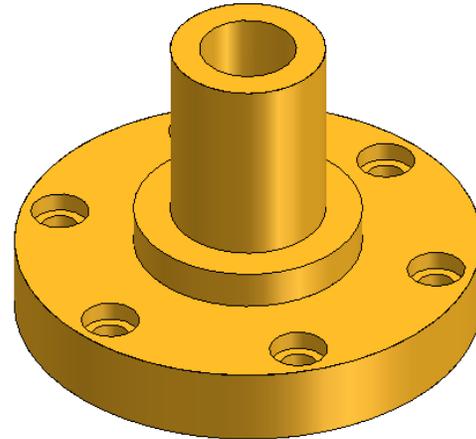


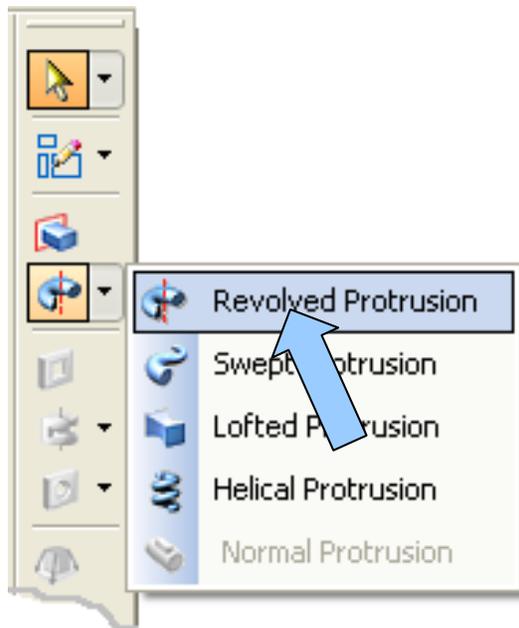
Chapter Three

More Basic Parts

Let's construct some additional parts to reinforce the skills that you have already acquired and to introduce some additional modeling tools. We'll start with the simple part illustrated to the right.



This is an example of what we call an axial symmetric part. Axial symmetric parts, as their name implies, are symmetrical about a long body axis. These parts can typically be machined using a lathe.



We'll start with a new part file.

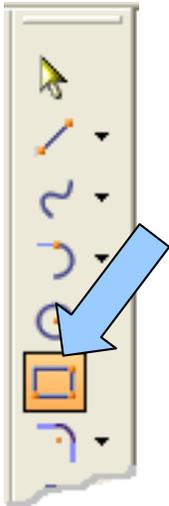
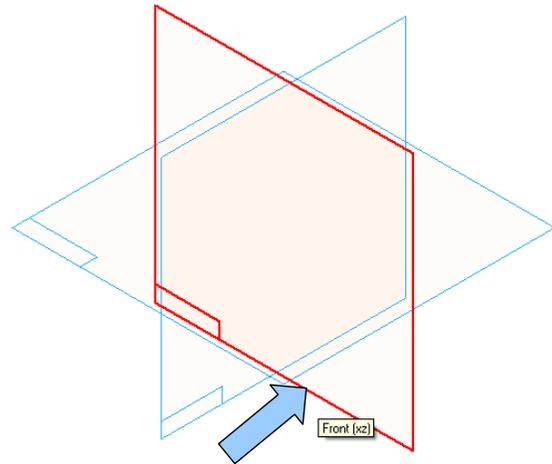
Click on the REVOLVED PROTRUSION Icon.

Notice that the Ribbon Bar is prompting you to select a Reference Plane.



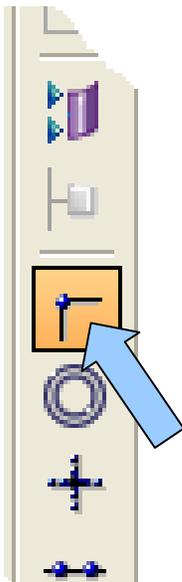
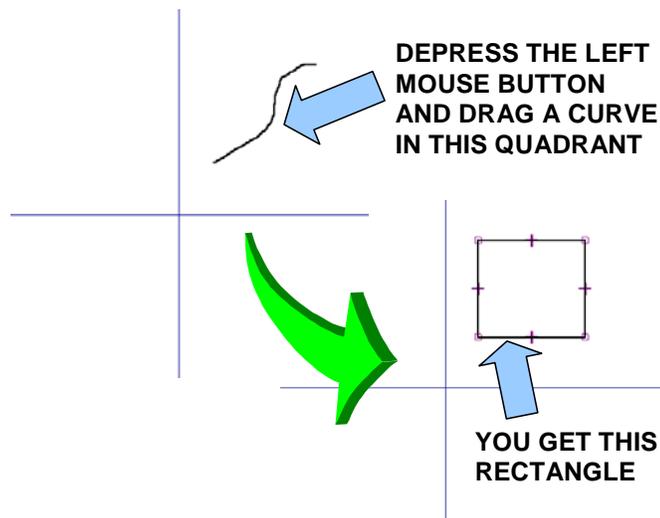
Click on the Reference Plane indicated by an arrow in the illustration to the right.

The display dynamically morphs to the Sketch View.



Notice that the tools change on the Feature Toolbar. If you look closely, you'll notice that the assortment of tools is a little different than the ones that appear when you create a sketch for a protrusion.

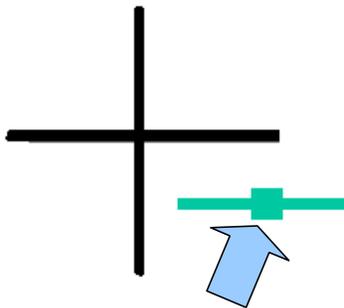
Let's start by clicking on the CREATE RECTANGLE Icon. Drag a curve in the top, right quadrant of the screen.



A rectangle appears with opposite corners located at the ends of the curve.

Click on the CONNECT Icon.

Then, move the cursor over the lower, left corner of the rectangle and click when the ENDPPOINT Symbol appears.

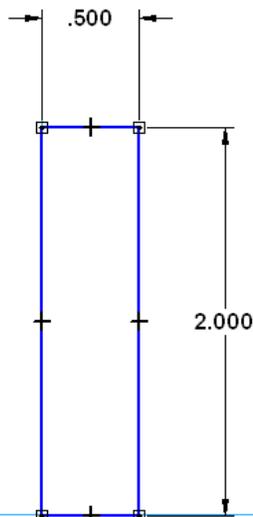
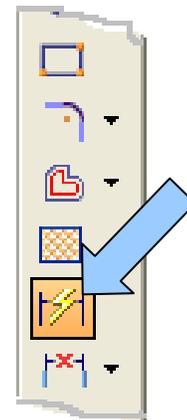


MIDPOINT

Now, move the cursor over the intersection of the axes and click when the MIDPOINT Symbol appears. This pins the lower, left corner of the rectangle to the origin of the file.

Click on the SMART DIMENSION Icon.

Next, click on the top edge of the rectangle and move the cursor upward. Click to set the text position and type 0.5 for the dimension value. Hit the ENTER Key.



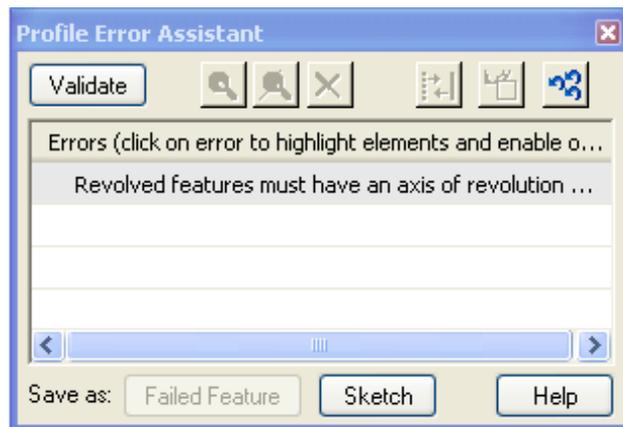
Repeat this process, placing a 2 inch dimension on the right side of the rectangle.

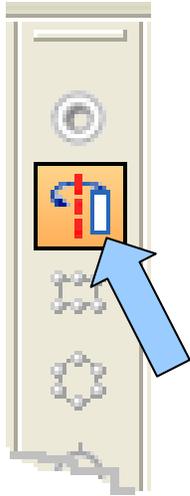
Your screen should look like this:

Click on the RETURN Button on the Ribbon Bar.

Notice that a warning Dialog Box appears!

When we create a revolved protrusion we must specify an axis of revolution in the profile step.





Click on the AXIS OF REVOLUTION Icon.

Move the cursor over the vertical axis and click on it when it highlights.

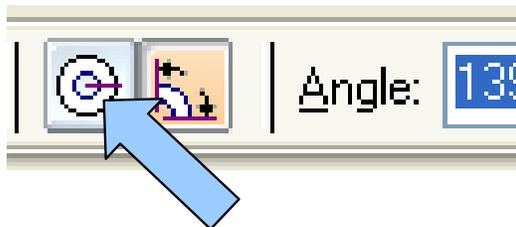
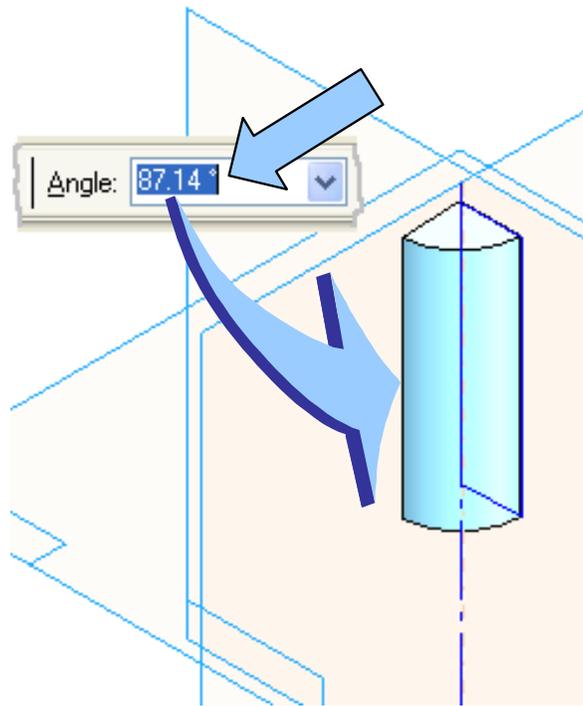
Notice that a dashed centerline is created on the axis.

Now click on the RETURN Button and you are returned to the Isometric view.

Notice that as you move the cursor, a ghost solid revolves out from the profile.

The value in the ANGLE Field on the Ribbon Bar also keeps updating.

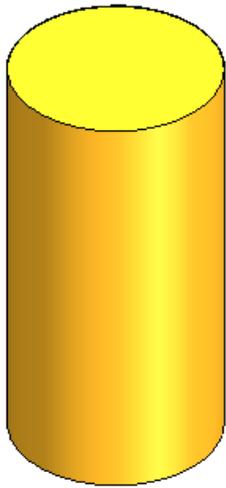
You could stop at any point and type a specific angle if you want to create a revolved solid that is less than 360 degrees.



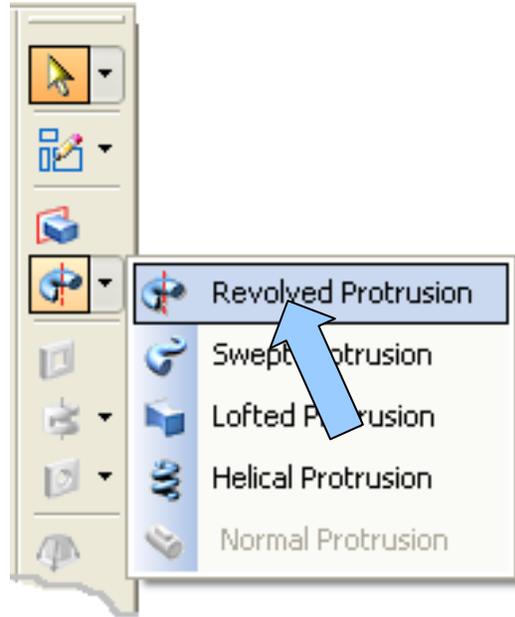
For our model, however, we need a complete 360 degree revolution of the profile.

So let's use a shortcut and simply click on the REVOLVE 360 Icon on the Ribbon Bar.

Then, click on the FINISH Button on the Ribbon Bar.



Your part should look like this:



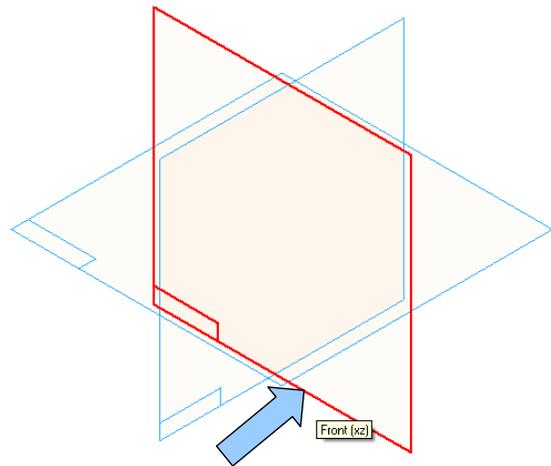
Next, let's create the stepped flange on the part. Once again, click on the REVOLVED PROTRUSION Icon.



Notice that the Ribbon Bar is prompting you to select a Reference Plane.

Click on the Reference Plane indicated by an arrow in the illustration to the right.

The display dynamically morphs to the Sketch View.

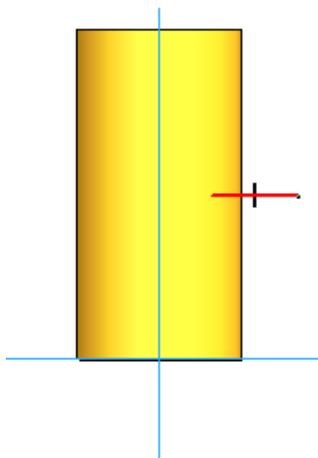




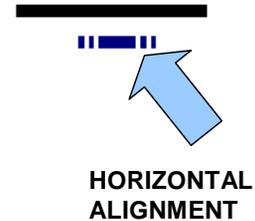
Next, click on the CREATE LINE Icon.

Move the cursor just to the left of the right vertical edge of the cylinder. Your vertical position should be near the middle. Click to set the starting point of the line.

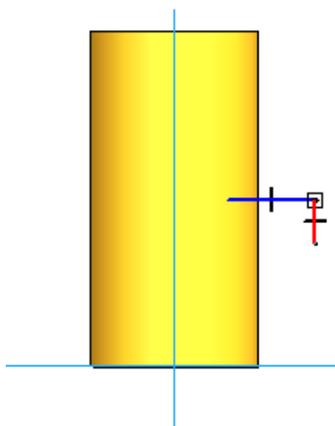
Now, move the cursor to the right about 0.5 inches and when the HORIZONTAL ALIGNMENT Symbol appears, click to set the other end of the line.



Your screen should look like this:

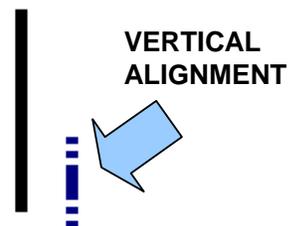


Now, continue, moving the cursor downward about 0.25 inches.



Position the cursor so that the VERTICAL ALIGNMENT Symbol appears and click while the symbol is visible.

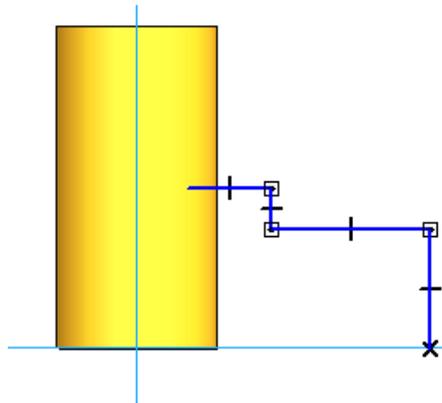
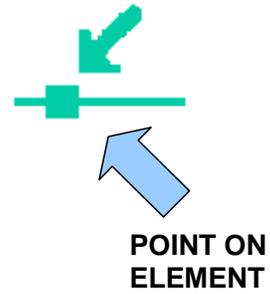
Your screen should look like this:



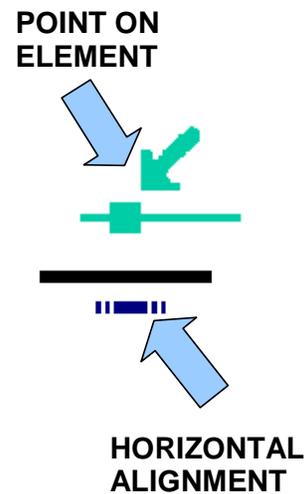
Move your cursor to the right about 1 inch and with the HORIZONTAL ALIGNMENT Symbol visible, click to set the line endpoint.

Move the cursor downward until it is positioned on the horizontal axis.

With both the POINT ON ELEMENT Symbol and the VERTICAL ALIGNMENT Symbol visible, click to set the endpoint of the line.

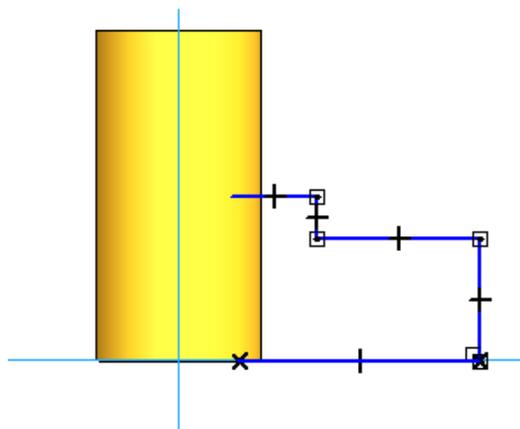


Your screen should look like this:

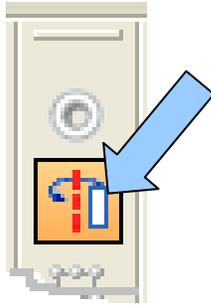


Now, move your cursor to the left so that it is just inside the right edge of the cylinder and on the horizontal axis.

With the POINT ON ELEMENT Symbol visible, click to set the endpoint of the line.



Your screen should look like this:



Next, click on the **AXIS OF REVOLUTION** Icon.
Move the cursor over the Vertical Axis and click on it.

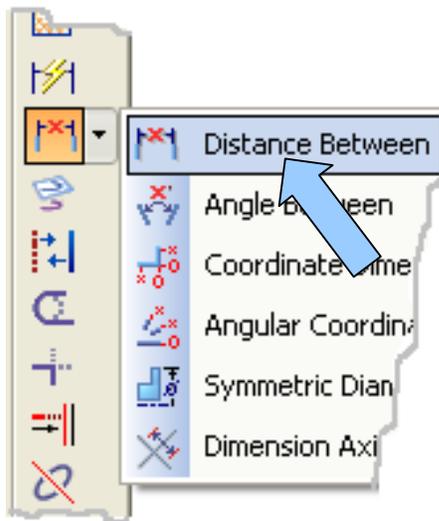
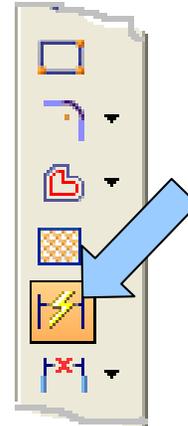
A centerline is created on the axis.

Now, let's place some dimensions on the sketch.

Click on the **SMART DIMENSION** Icon.

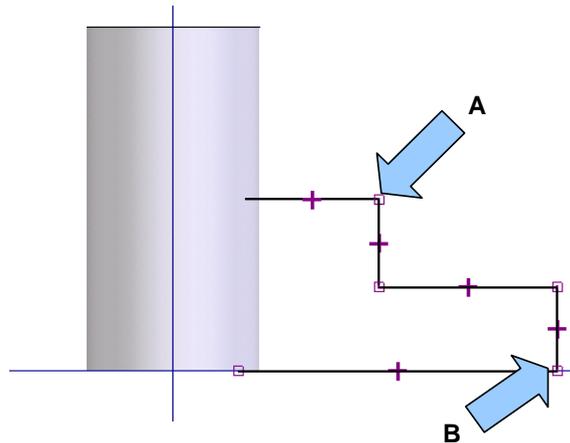
Then, click on the short vertical line at the far right of the sketch and move the cursor to the right.

Click to set the text position and type 0.5 for the value. Hit the **ENTER** Key.



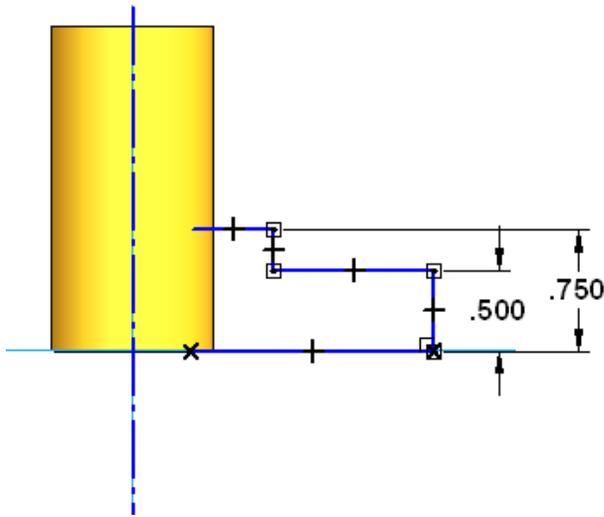
Click on the **DISTANCE BETWEEN** Icon.

Now, move the cursor over the corner labeled "A" in the illustration to the right.



Click when the **ENDPOINT** Symbol appears.

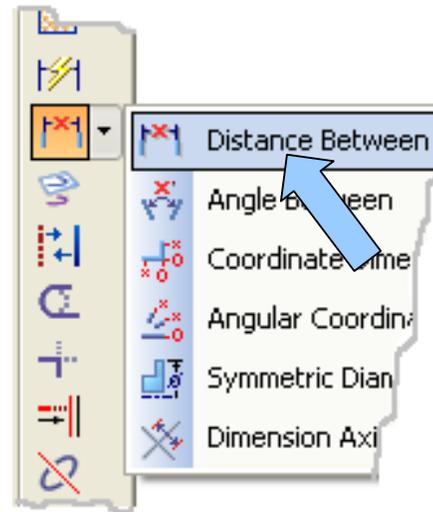
Then, move the cursor over the corner labeled "B" and click when the **ENDPOINT** Symbol appears.



Move the cursor to the right and click to set the text position.

Type 0.75 for the value and hit the ENTER Key.

Your sketch should look like this:



Still using the DISTANCE BETWEEN Tool, move the cursor over the upper end of the Vertical Axis.



ENDPOINT

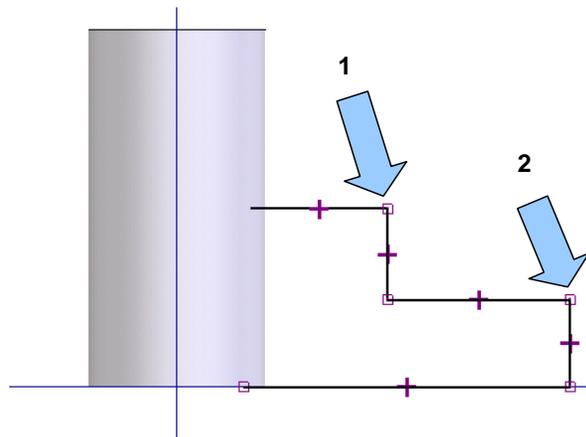
Click when the ENDPOINT Symbol appears.

Then, move the cursor over the corner labeled "1" in the illustration to the right and click on it when the ENDPOINT Symbol appears.

Then, move the cursor upward and click to set the text position.

Type 0.75 for the value and hit the ENTER Key.

Repeat this process, creating a 1.5 inch horizontal dimension between the Vertical Axis and the corner labeled "2."

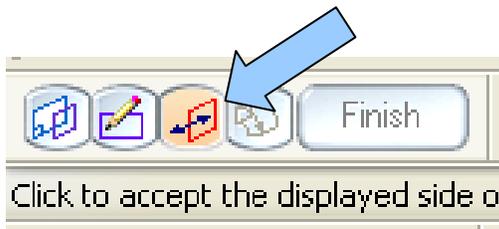
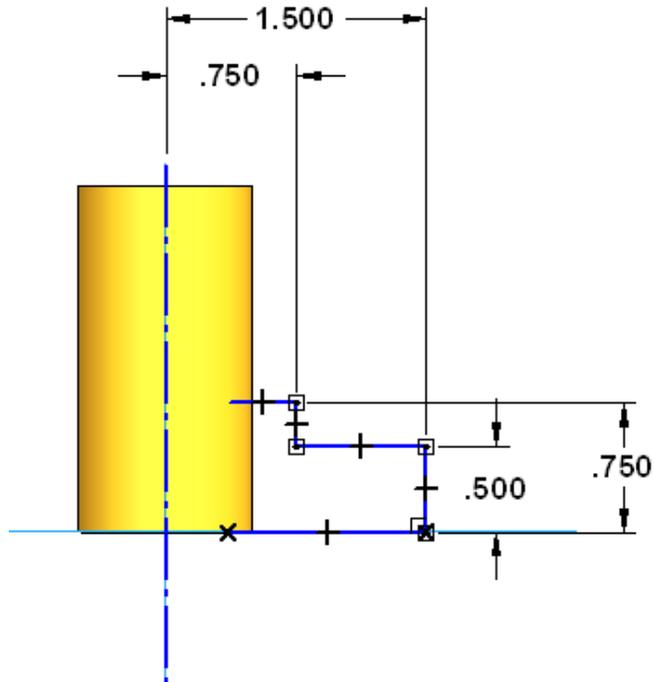


Your sketch should now look like this:

Notice the vertical alignment, horizontal alignment, and connection symbols.

Combined with the dimensions, this completely defines our profile.

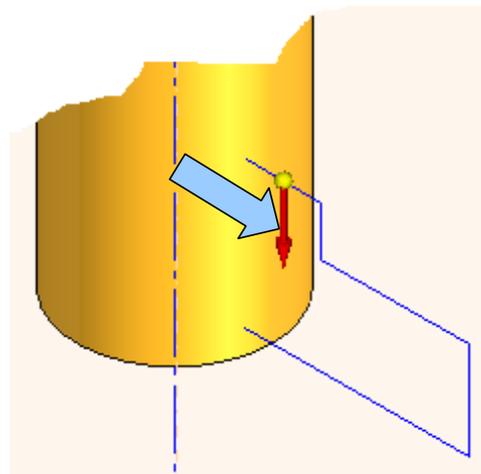
Now, click on the RETURN Button on the Ribbon Bar.

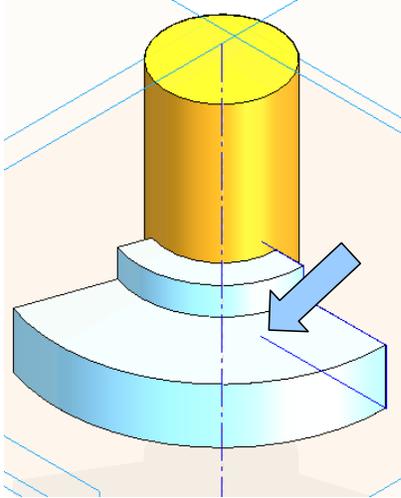


Notice that since the profile is open, the Ribbon Bar is prompting us with the SIDE STEP Icon.

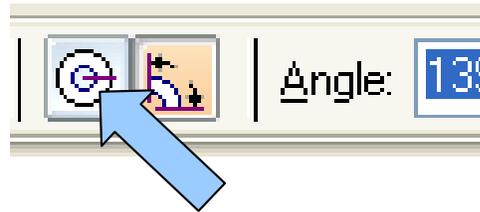
As you move the cursor, a side step vector toggles on the profile.

Position the cursor so that the vector points into the profile and click to set the direction.



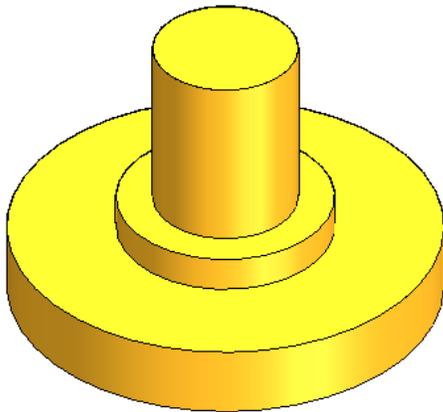


Now, as you move the cursor, a ghost revolved protrusion grows from the profile.

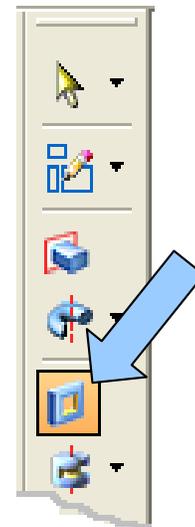


Click on the REVOLVE 360 Icon on the Ribbon Bar.

Then, click on the FINISH Button on the Ribbon Bar.



Your screen should look like this:

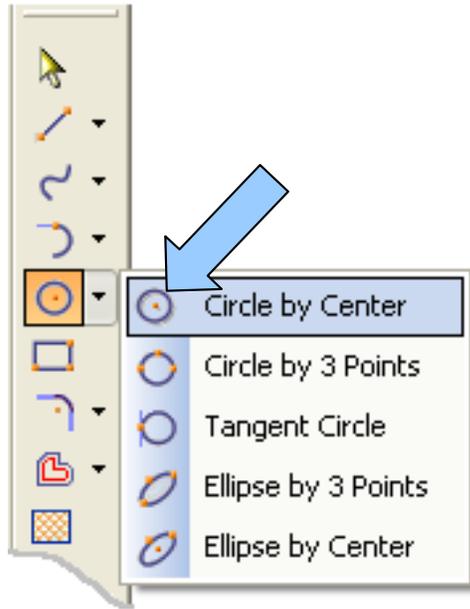


Next, let's create the bore hole in the center of the hub.

Click on the CUTOUT Icon.

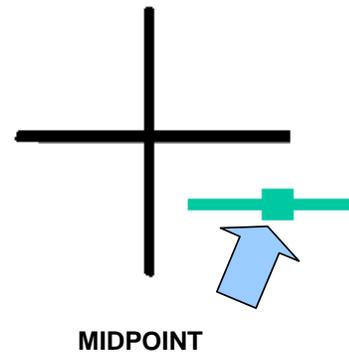


Notice that the Ribbon Bar is prompting you to select a Reference Plane.



Click on the Horizontal Reference Plane. The display transforms to the Sketch Window.

Now, click on the CREATE CIRCLE BY CENTER Icon.



Move the cursor over the intersection of the axes so that the MIDPOINT Symbol appears.

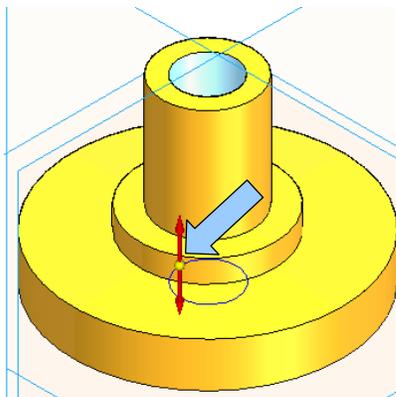
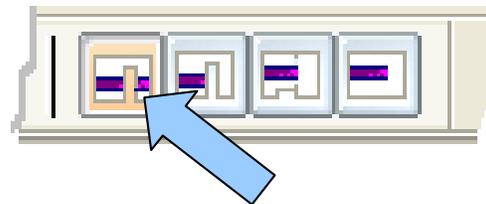
Click when the symbol is visible.

As you drag the cursor, a ghost circle grows on the screen.

Type 0.625 for the Diameter and hit the ENTER Key.

Then, click on the RETURN Button on the Ribbon Bar. The display transforms to the Isometric View.

Click on the THROUGH ALL Icon on the Ribbon Bar.

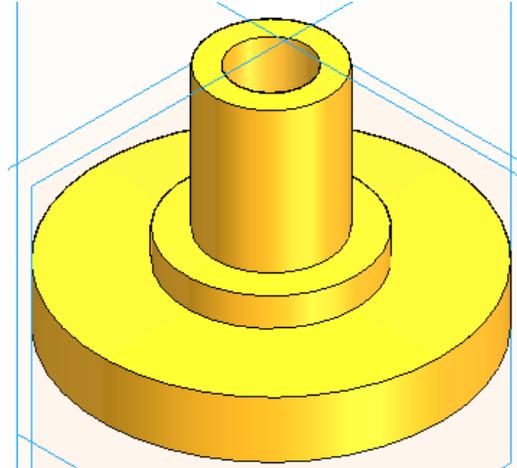


As you move the cursor over the sketch, notice that an Extent Vector toggles direction.

When it changes to a double-headed vector, click on it.

Then, click on the FINISH Button on the Ribbon Bar.

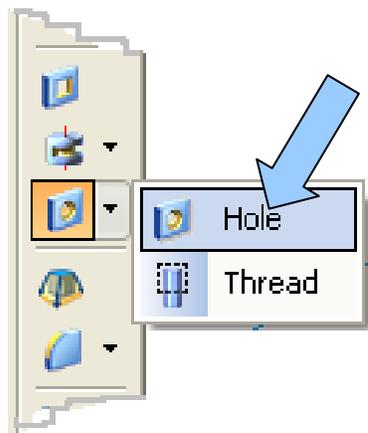
Your screen should look like this:



Next, let's create the six counterbored holes on the flange of the part.

Before we do that, however, let's talk about the design intent for this part. We might possibly modify the size of the counterbore or the through hole that is associated with it as the design of the part evolves.

However, whatever we do, all six counterbored holes will have the same dimensions and will always be located on a 2.5 inch diameter bolt circle, sixty degrees apart.

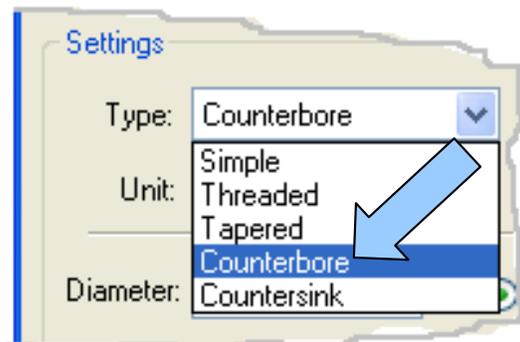


Because of this design intent, we'll create just one counterbored hole and then use a copy function to create the five duplicate features. Then, if we need to make future changes the task will be much simpler!

Click on the HOLE Icon.

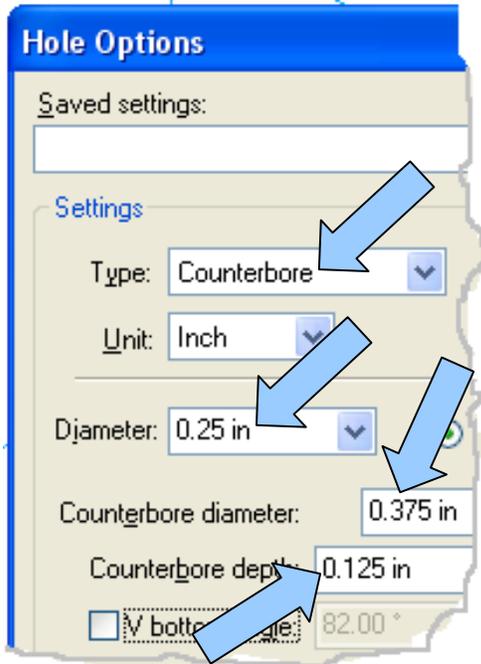


Click on the HOLE PROPERTIES Icon on the Ribbon Bar.



The HOLE OPTIONS Dialog Box appears.

Click on the Pull-down Arrow to the right of the Type Field and select the Counterbore Option.

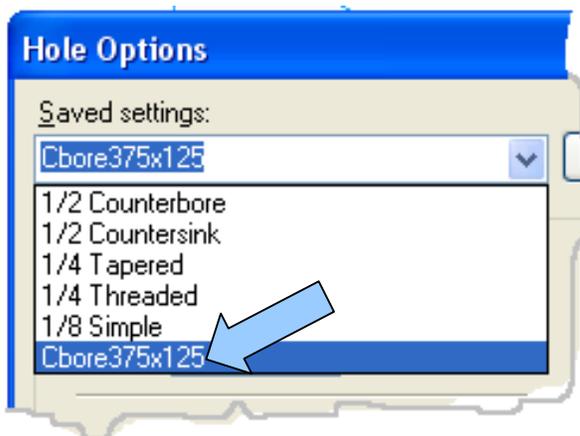
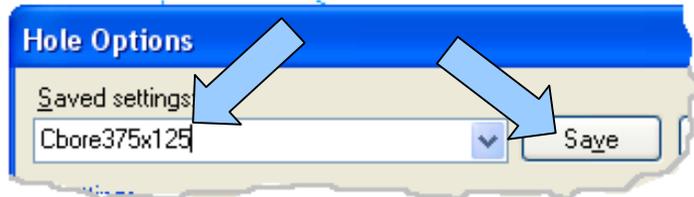


Next, select 0.25 from the Pulldown Options for Diameter.

Proceed to select a Counterbore Diameter of 0.375 and a Counterbore Depth of 0.125.

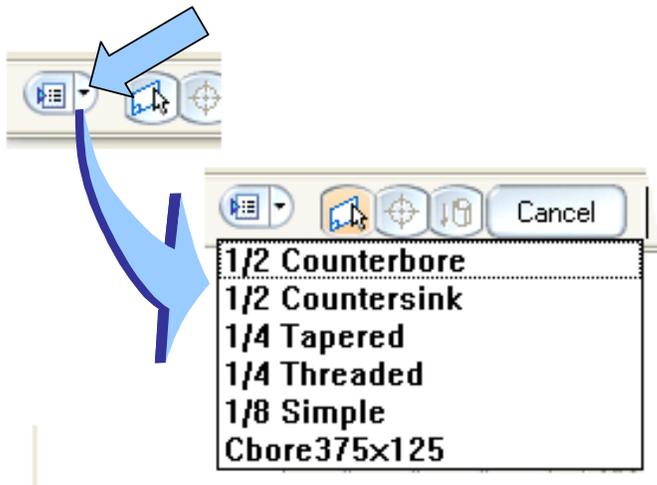
Before we exit the Dialog Box, let's suppose that we plan on using this particular set of parameters for additional features on this part.

We can type a name in the Saved Settings Field at the top of the box. Then, click on the SAVE Button to the right of the field.



If you enter the box at a later time during work on the part, you can click on the Pulldown Arrow to the right of the field and select the particular counterbore/hole settings to reuse.

Click on the OK Button at the bottom of the Dialog Box.



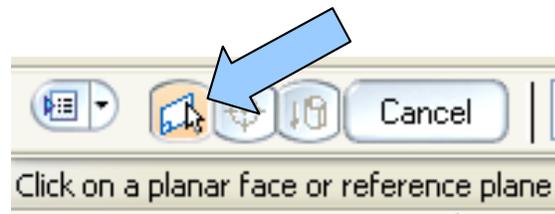
If you look at the Ribbon Bar, you'll notice that there is also a Pulldown Arrow to the right of the HOLE OPTIONS Icon.

Click on this Pulldown and the saved options appear directly below.

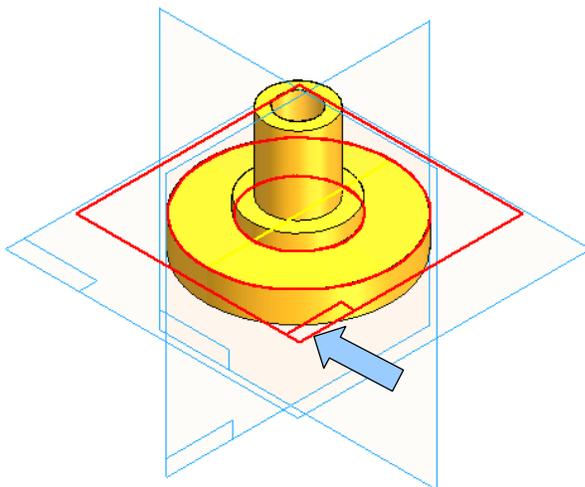
Meanwhile, back at our hole drilling operation!!

The Ribbon Bar is now prompting us to select a Reference Plane.

We want to locate the counterbored hole from the top surface of the broad flange on the part.



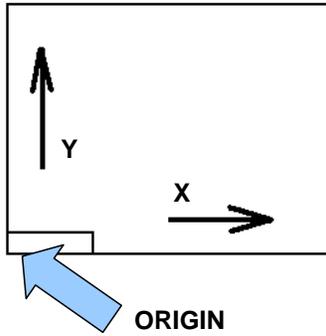
In our past exercises, the default System Planes were always positioned where we needed them! Well, as you'll soon discover, there are hundreds of situations where that isn't going to be true.



This is not a problem, however, since we can instantly pick any surface on a part to use as a Local Reference Plane for a current operation.

To do this, move the cursor onto the broad top surface of the flange.

Notice that a local reference plane is displayed. You can click on this displayed plane to pick it.



NOTE: The small rectangle at the corner of a reference plane indicates the X and Y axes of the plane.

I've illustrated the orientation in the picture to the left.

While the plane is displayed, you can use the following Hot Keys to toggle the orientation of the plane:

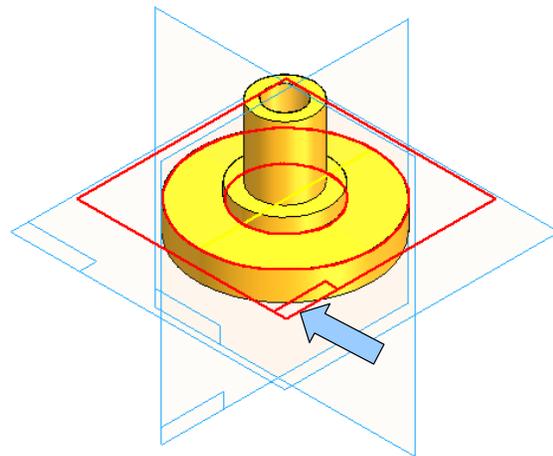
Hot Key Action

N or n Rotates reference plane counterclockwise

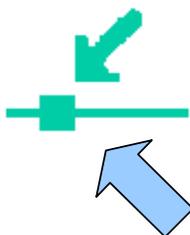
B or b Switches orientation of reference plane to previous status

T or t Switches X axis to opposite corner

Click on the plane while the display looks like this:



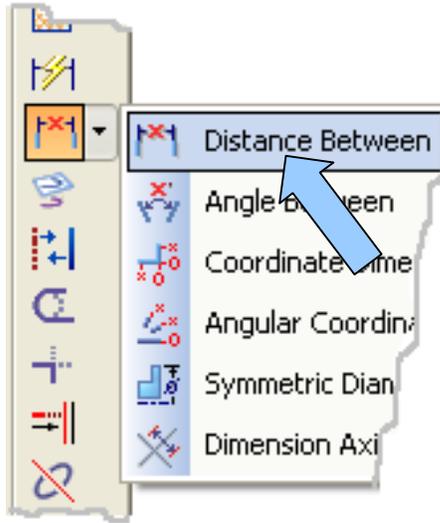
The display transforms to the Sketch View. As you move your cursor over the screen, a ghost counterbored hole follows.



POINT ON ELEMENT

Move the cursor over the right section of the horizontal axis until the POINT ON ELEMENT Symbol appears.

Click while the symbol is visible.



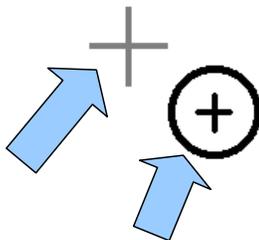
We want the counterbored hole to lie on a 2.5 inch diameter bolt circle.

To accomplish this, click on the DISTANCE BETWEEN Icon.

Move the cursor over the lower end of the vertical axis until the ENDPOINT Symbol appears.



Click while the symbol is visible.



Next, move the cursor over the counterbored hole that you just placed so that it highlights.

Then, as you move over the center marker for the hole, notice that the CENTERPOINT Symbol appears.

Click when the symbol is visible.

MOVE THE CURSOR OVER THE CROSS. WHEN THE CENTERPOINT SYMBOL APPEARS, CLICK TO SET THE POSITION

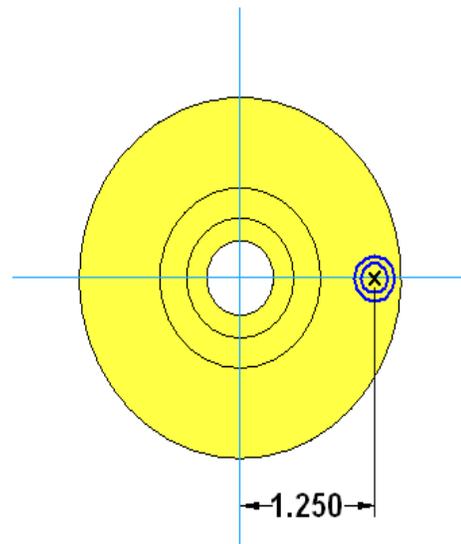
Now move the cursor downward and click to place the dimension text.

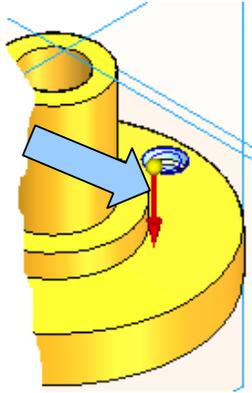
Type 1.25 for the value and hit the ENTER Key.

Your sketch should look like this:

Click on the RETURN Button on the Ribbon Bar.

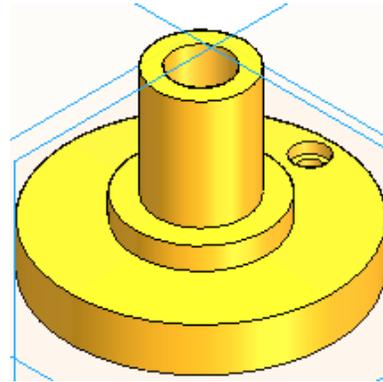
The display dynamically switches to the Isometric view.



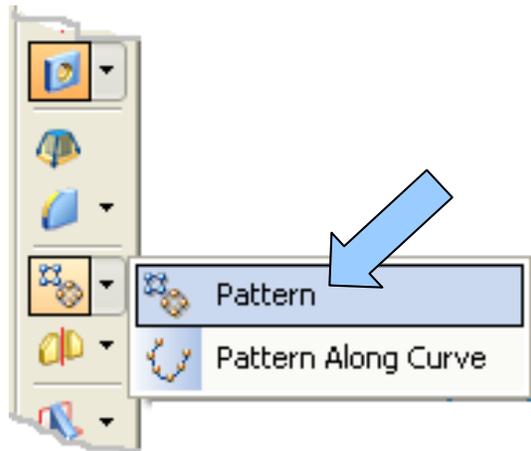


As you move the cursor over the part, notice that a vector toggles on the counterbored hole location.

Click when the vector is facing downward.
Then, click on the FINISH Button on the Ribbon Bar.



Your part should now look like this:



Now, let's create five additional counterbored holes that are duplicates of the one we just finished.

To do this, click on the PATTERN Icon.

You'll find it in the lower half of the Feature Toolbar.

Notice that the Ribbon Bar is prompting you to select a feature.

Move the cursor over the counterbore and hole and click on this feature when it highlights.



SMART

FAST

Once again, look at the Ribbon Bar. You'll notice that there are two Options that you can use: FAST or SMART.

Let's click on the FAST Icon.