

Still using the SMART DIMENSION Tool, click on the left vertical edge of the sketch.

Move the cursor to the left and click to set the text position.
Type 1.5 for the value and hit the ENTER Key.

Next, click on the right vertical edge of the sketch.

Move the cursor to the right and click to set the text position.
Type 0.5 for the value and hit the ENTER Key.

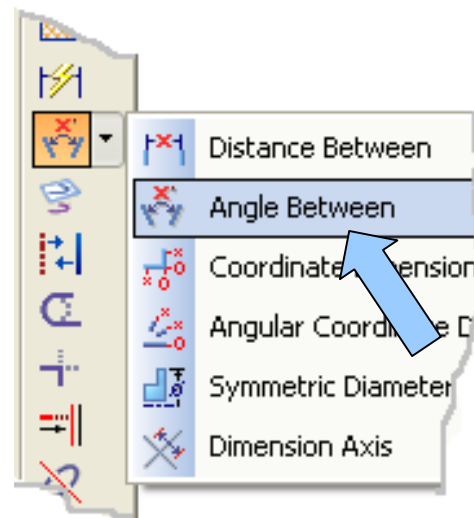
This sets the vertical dimensions for the part.

Your sketch should now look like this:

We're going to use a new dimensioning tool to specify the angle of the oblique line.

Click on the ANGLE BETWEEN Icon.

It is located in the flyout menu that contains the DISTANCE BETWEEN Icon that we have been using.



First, click on the right end of the horizontal axis.

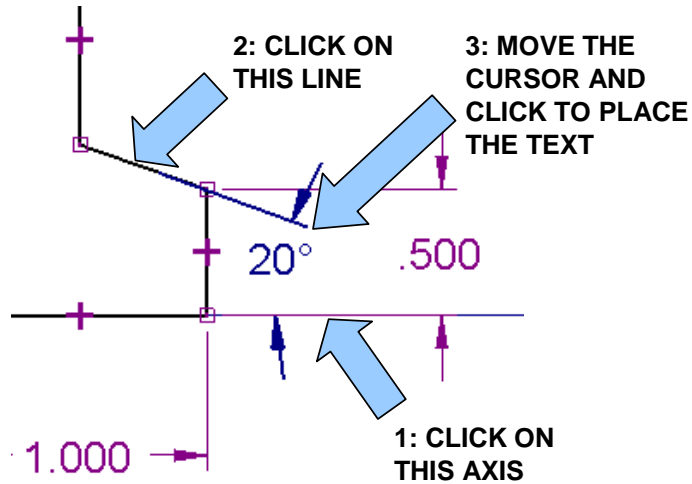
Second, click on the oblique line in the sketch.

Third, move the cursor and click to place the angle.

You'll notice that you can get several different angular relationships based on cursor location.

Type 20 and hit the ENTER Key.

Your sketch is now fully constrained.

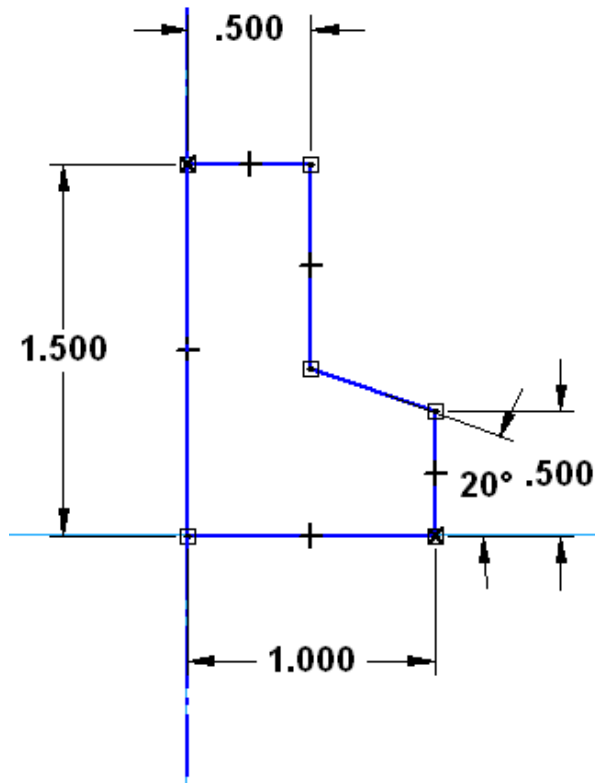


I've shown the sketch with dimensions in the illustration to the left.

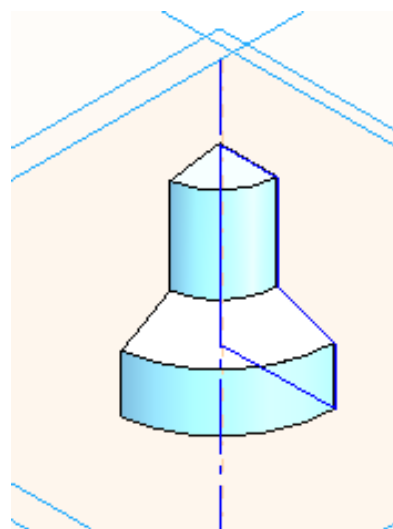
Notice the vertical and horizontal alignment symbols and the connection points.

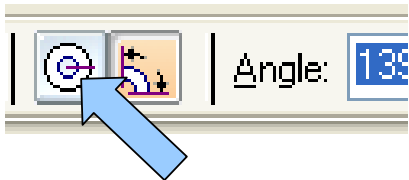
Click on the RETURN Button on the Ribbon Bar.

You are returned to the Isometric View.



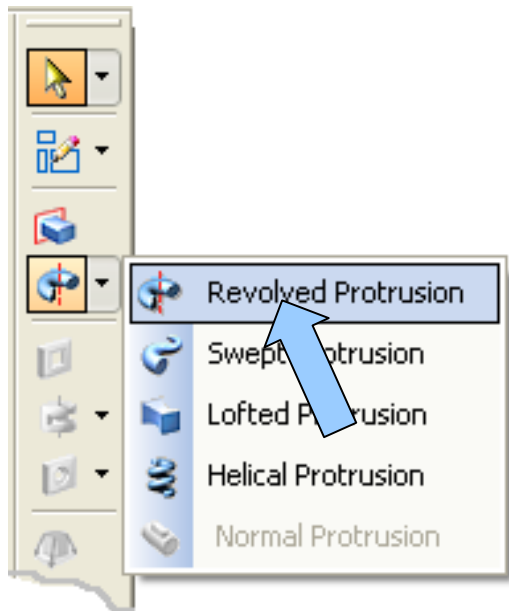
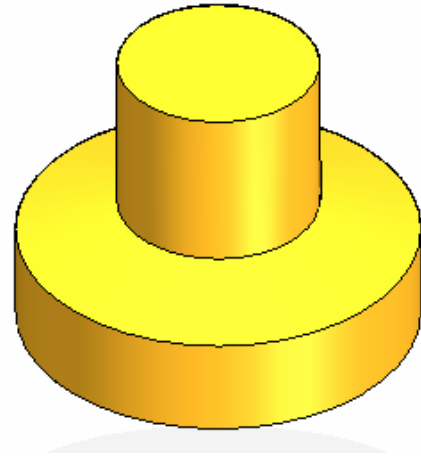
As you move the cursor, a ghost revolved solid grows on the screen.





Click on the REVOLVE 360 Icon on the Ribbon Bar. Then, click on the FINISH Button.

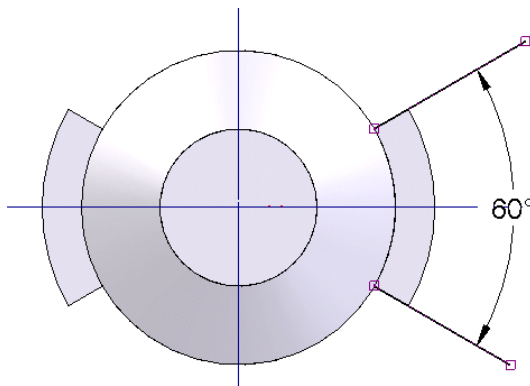
Your part should look like this:



Next, let's create the two, symmetrical lugs that are located on the rim of the part.

Start by clicking on the REVOLVED PROTRUSION Icon.

The Ribbon Bar prompts you to select a reference plane for the sketch.



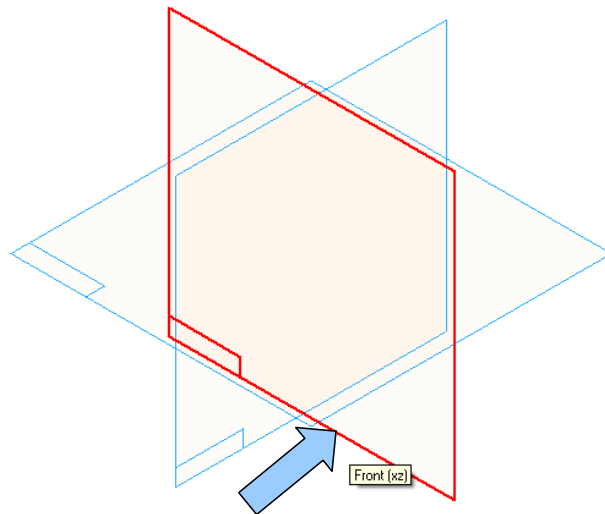
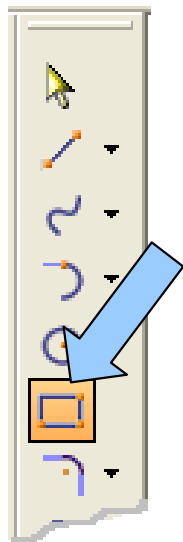
The lugs are located directly opposite each other and each lug is developed along a 60 degree arc as illustrated in the picture to the left.

Here's our game plan.

1. Create a profile of the lug on the vertical reference plane that lies at the midpoint of the 60 degree sweep.
2. Use the Symmetrical Extent Option on the Ribbon Bar to revolve a lug through 30 degrees on both sides of the reference plane.
3. Mirror the feature across the other vertical reference plane to create the matching lug.

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

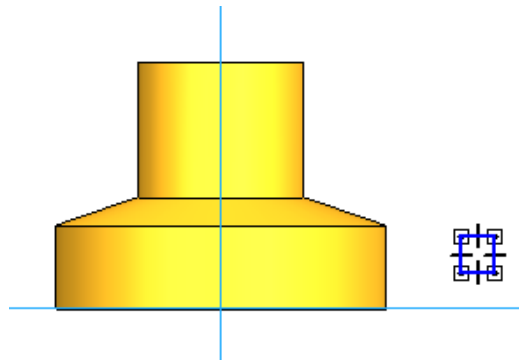
The screen transforms to the Sketch Window.

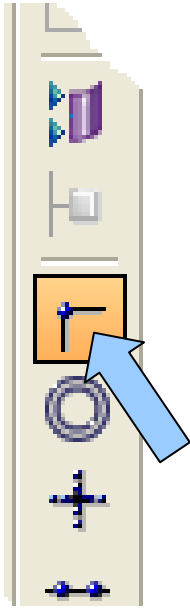


Click on the CREATE RECTANGLE Icon.

Click and depress the LEFT MOUSE BUTTON to the right of the part and drag downward and to the right a short distance. Then, let go of the mouse button at the end of the stroke.

A small rectangle should appear on the screen.





Next, click on the CONNECT Icon.

Move the cursor over the lower left corner of the rectangle until the ENDPOINT Symbol appears.



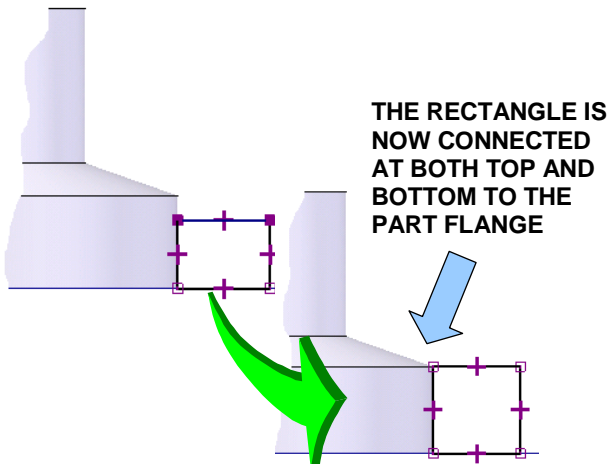
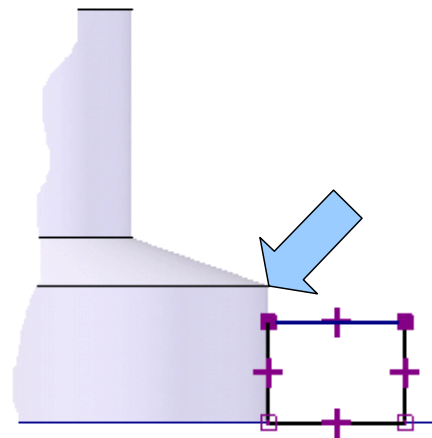
Click when the symbol is visible.

Next, move the cursor over the lower, right corner of the part and click when the CONNECT Symbol appears.



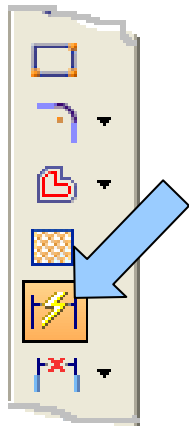
Repeat this process, moving the cursor over the top, left corner of the rectangle and clicking on it when the ENDPOINT Symbol appears.

Then, move the cursor over the corner indicated by an arrow in the illustration to the right and click when the CONNECT Symbol appears.



We will always want the lug to be equal in height to the height of the outside flange wall.

By creating these connection points, we have specified our design intent.

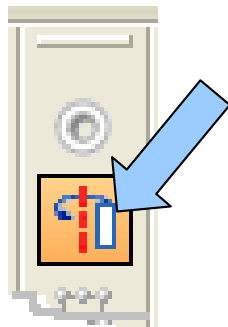
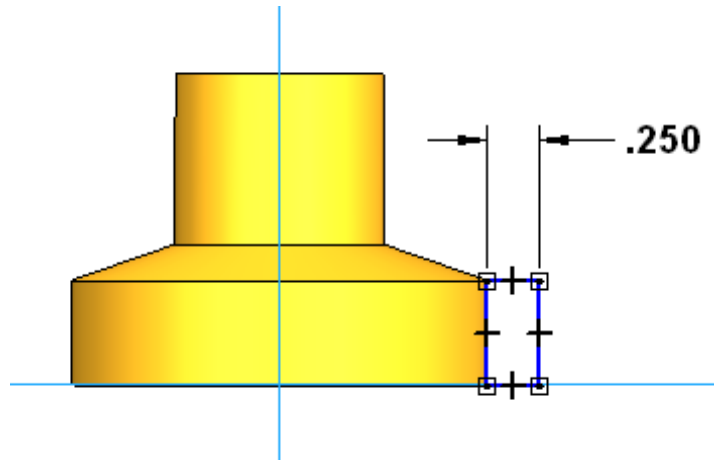


Next, click on the SMART DIMENSION Icon.

Then, click on the top edge of the rectangle that you just created. Move the cursor upward and click to set the text position.

Type 0.25 for the value and hit the ENTER Key.

Your sketch should now look like this:



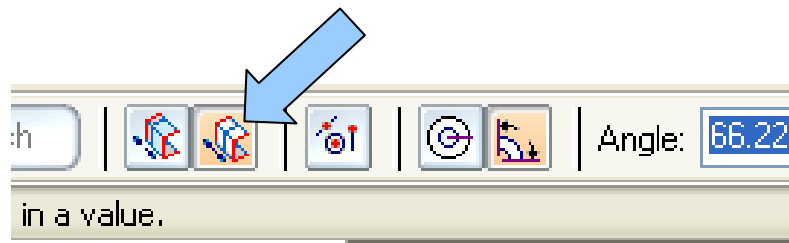
Click on the AXIS OF REVOLUTION Icon.

Then, click on the lower end of the vertical axis. Now, click on the RETURN Button.

The display dynamically transforms to the Isometric view.

As you move the cursor on the screen, a revolved protrusion grows from the rectangle.

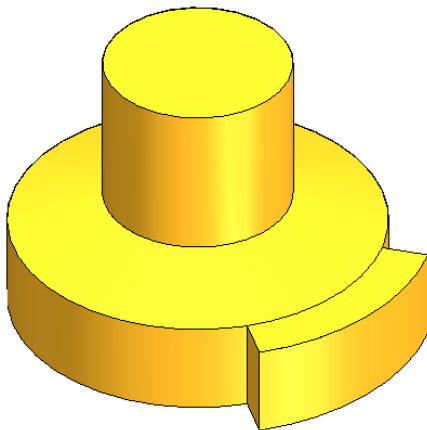
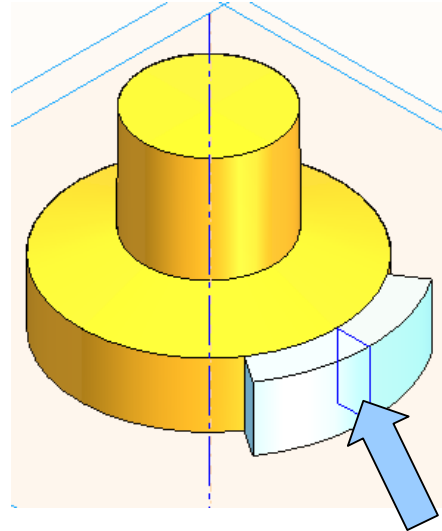
Let's click on the SYMMETRIC REVOLVE Icon on the Ribbon Bar.



Now as we move the cursor, the revolved protrusion grows out of both sides of the profile an equal amount.

Type 60 in the ANGLE Field on the Ribbon Bar and hit the ENTER Key.

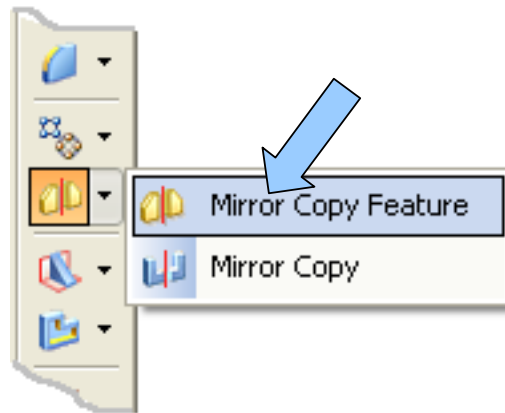
Then, click on the FINISH Button.



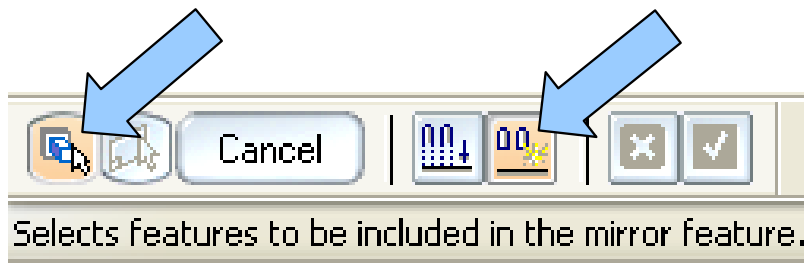
Your part should now look like this:

Next, we'll create a duplicate of the lug on the opposite side of the part.

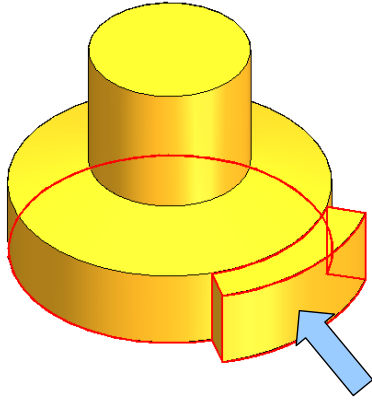
To do this, click on the MIRROR COPY FEATURE Icon.



Notice that the Ribbon Bar is set to FAST copy.



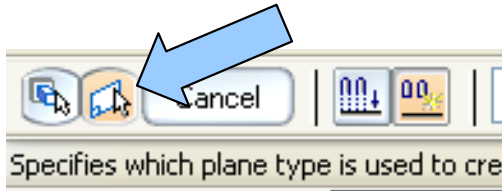
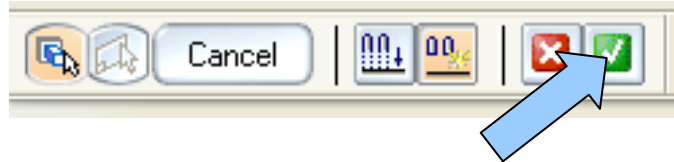
The first Icon is prompting you to select a feature.



Now, move the cursor over the lug and the feature highlights.

Click on it.

Now, click on the green checkmark on the Ribbon Bar.



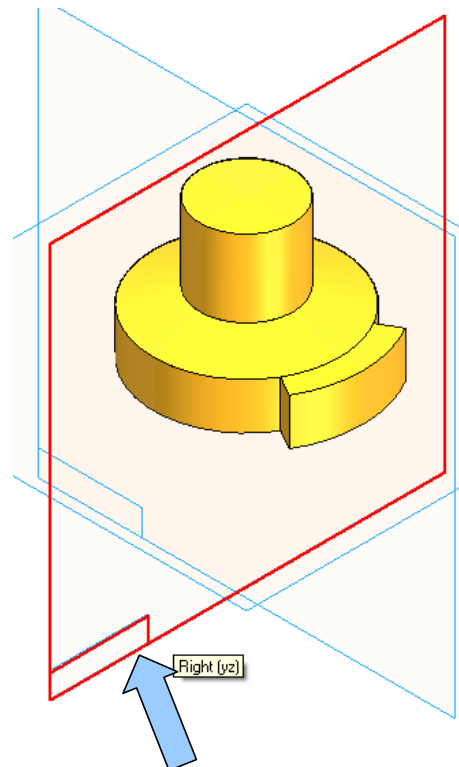
Notice that the Ribbon Bar is now prompting you to select a Reference Plane for the mirror operation.

Click on the vertical reference plane indicated by an arrow in the illustration to the right.

A Dialog Box appears, advising you that the operation has failed. You are advised to try again using the SMART Option.

I led you down this road on purpose to illustrate the difference between the smart and fast options. The fast option requires less processing power and will increase throughput when working with large amounts of simple geometry.

However, when a feature interacts with overall part topology, such as this external lug does, you must use the smart option.



Click on the OK Button in the Warning Dialog Box.

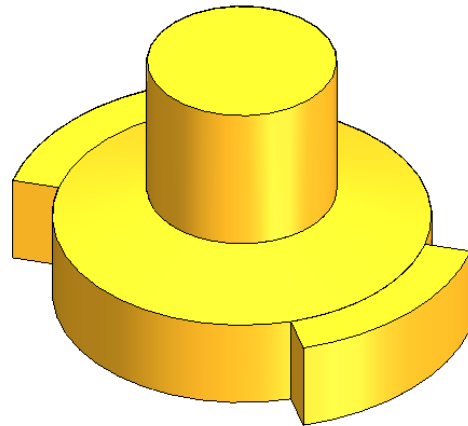


Then, click on the SMART Icon on the Ribbon Bar.

Notice that a PREVIEW Button appears on the Ribbon Bar. Click on the PREVIEW Button.

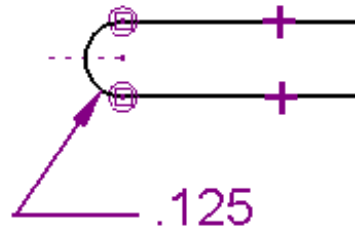
Then, click on the FINISH Button that appears on the Ribbon Bar to complete the operation.

Your part should look like this:

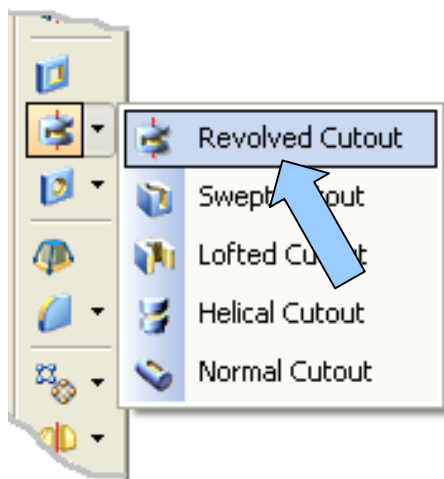


In our next step we'll create the groove that runs around the outer perimeter of the flange. It also passes through the two lugs. This groove could be created by using a profile cutter on a lathe.

The cutter would have the cross-section shown in the illustration to the right:



We'll use the REVOLVED CUTOUT Tool to simulate this manufacturing step in our design.

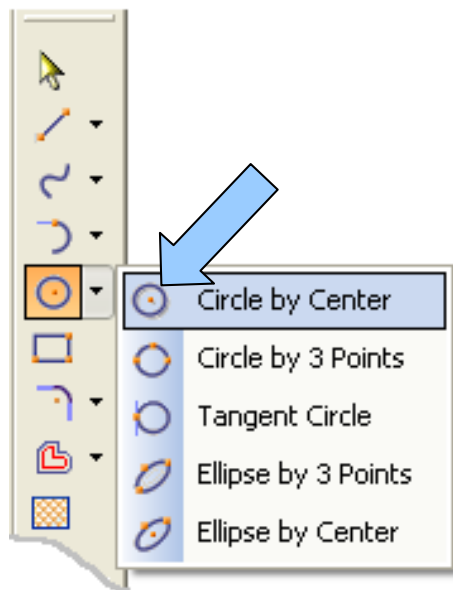
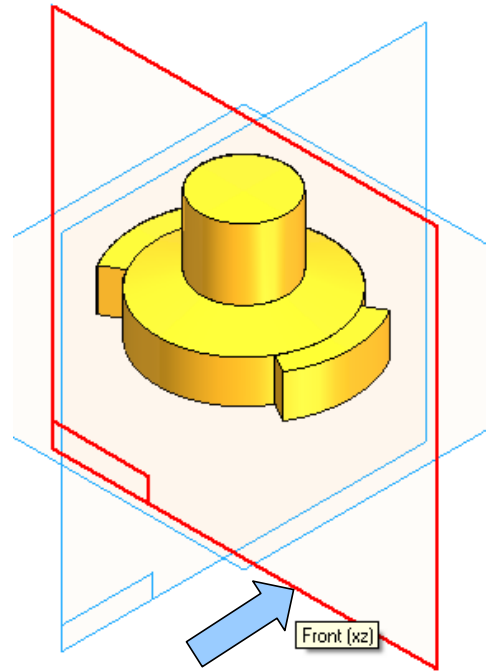


Click on the REVOLVED CUTOUT Icon.

You are prompted to select a reference plane.

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

The display transforms to the Sketch Window.



Click on the CREATE CIRCLE BY CENTER Icon.

Now let's suppose that we will always want the groove that we are creating in the perimeter of this part to be located at the midpoint of the vertical edge.

For instance, at present, the vertical edge height is 0.5 inches, so the vertical height of the groove centerline from the baseline of the part will be 0.25 inches.

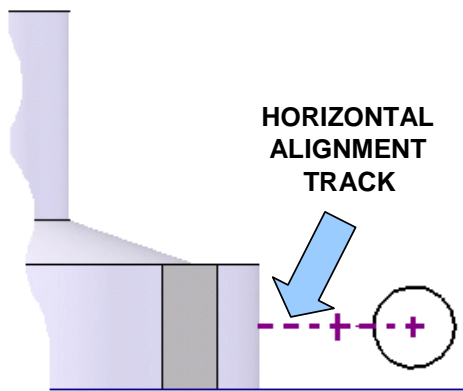
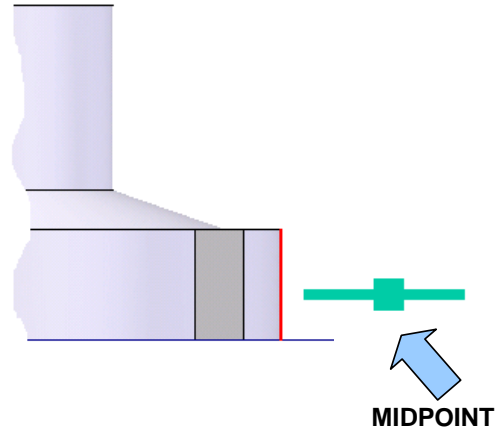
If we later increase the vertical edge height to 1 inch, we'll want the height of the groove centerline to be 0.5 inches.

We could dimension the height of the groove centerline and modify this dimension each time we change the part.

There is, however, a much simpler and more elegant way to get the same result. Let's use a connecting relationship to force the relationship.

With the CREATE CIRCLE Tool selected, move the cursor over the short, vertical edge on the right side of the part.

Notice that the MIDPOINT Symbol appears. DO NOT click to set the circle position.



Instead, move the cursor carefully to the right.

Notice that a dashed line extends from the center of the vertical edge to the cursor position.

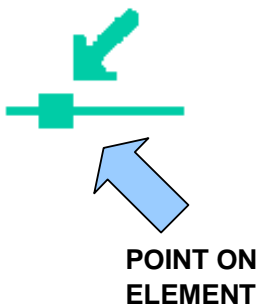
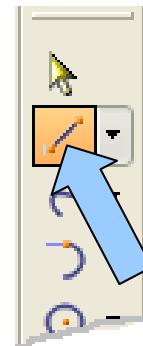
Click to set an arbitrary horizontal position for the center of the circle.

Now, move the cursor and a ghost circle grows. Click to set the circle.

This imaginary alignment axis will now be embedded in the design intent of the part. The relationship will act just as though you clicked when a POINT ON ELEMENT Symbol was present!

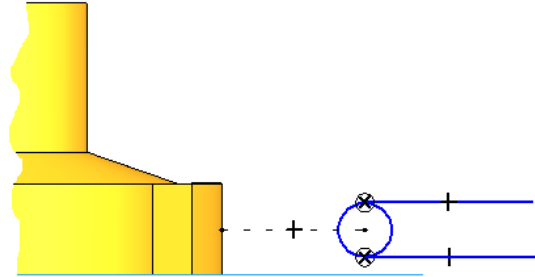
Click on the CREATE LINE Icon.

Move the cursor over the circle at about the 11 O'Clock position and click when the POINT ON ELEMENT Symbol appears.

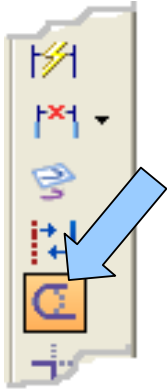


Now, move the cursor to the right and notice that the line tracks around the circle, both connected and tangent to the circle. When the line is at a horizontal orientation, the HORIZONTAL ALIGNMENT Symbol appears. Click when the symbol is visible and the line is approximately 1.5 inches long.

Repeat this process, clicking on the circle at about the 7 O'Clock position. Create a second tangent, horizontal line the same approximate length.

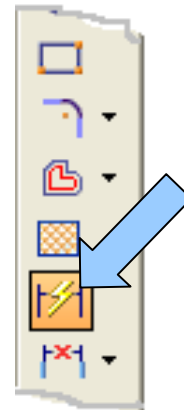


Your sketch should look like this:



Click on the TRIM Icon.

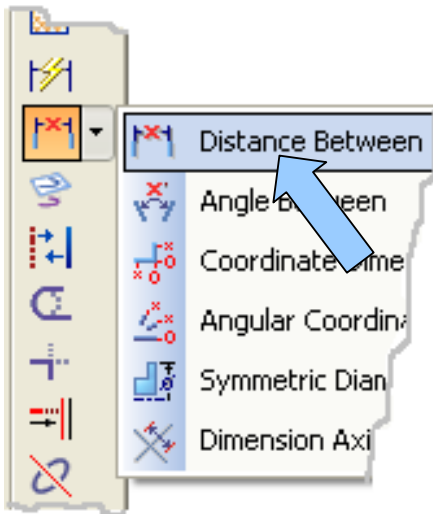
Drag a curve through the right side of the circle to eliminate it from the sketch.



Now, click on the SMART DIMENSION Icon.

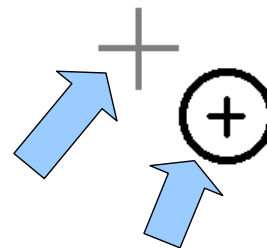
Click on the remains of the circle and move the cursor down and to the left. Click to set the text position.

Type 0.125 for the radius value and hit the ENTER Key.



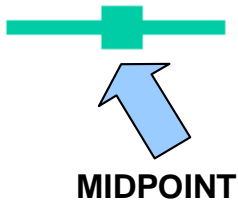
Next, click on the DISTANCE BETWEEN Icon.

Move the cursor over the circular arc and a point appears at the center.



Move over the point and the CENTERPOINT Symbol appears. Click while the Symbol is visible.

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



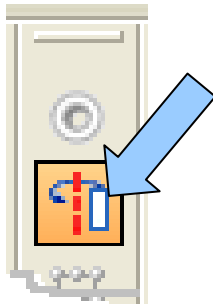
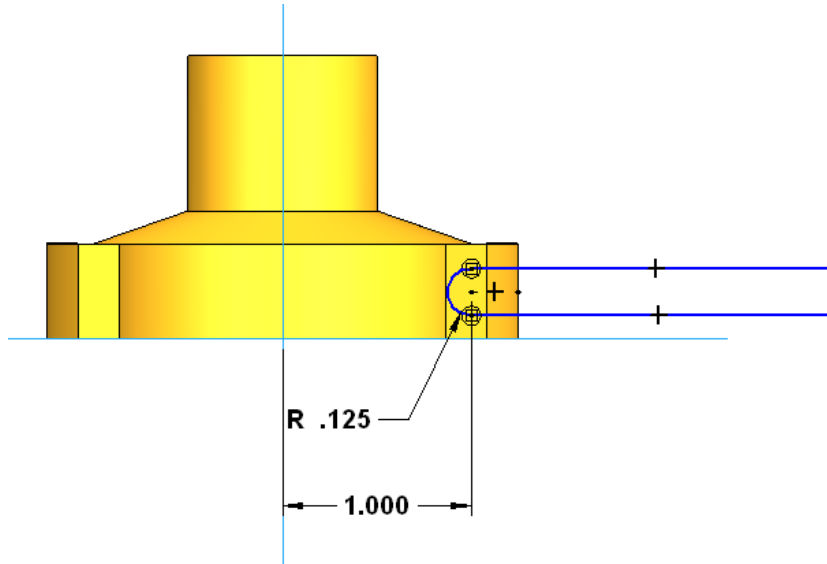
Next, move the cursor over the bottom edge of the part. When you reach the center, the MIDPOINT Symbol appears.

Click while the symbol is visible.

Move the cursor downward and a ghost dimension follows. Click to set the text position.

Type 1 and hit the ENTER Key.

Your sketch should look like this:



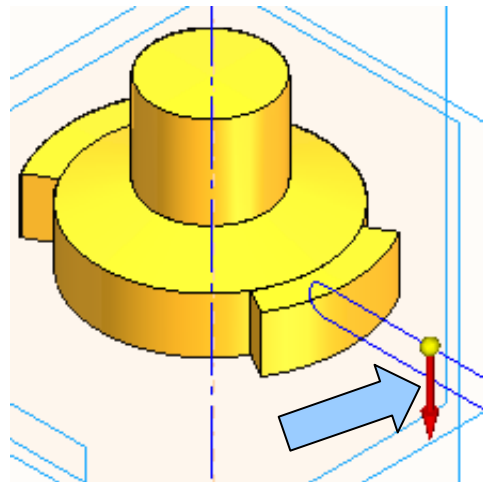
Click on the AXIS OF REVOLUTION Icon. Then, click on the lower end of the vertical axis.

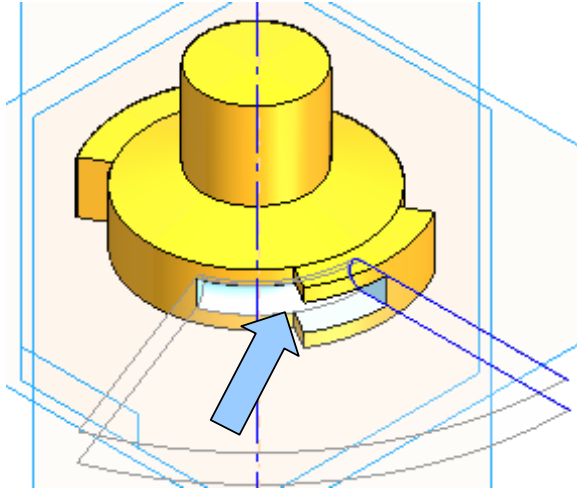
Finally, click on the RETURN Button on the Ribbon Bar.

The display transforms to the Isometric view.

Notice that the Ribbon Bar is prompting you with the SIDESTEP Icon and a vector toggles on the profile as you move your cursor over it.

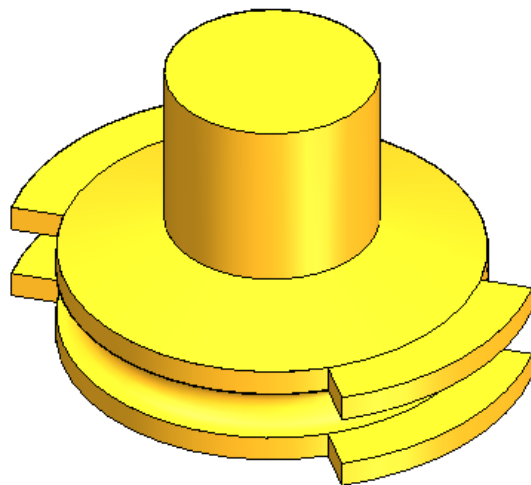
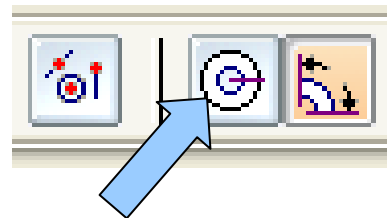
Click when the vector points into the profile.





Now, as you move your cursor, a ghost cutout path grows in one direction of rotation.

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:

Let's create the bore hole and the keyway.

Click on the CUTOUT Icon.

Next, click on the Horizontal Reference Plane.

The screen transforms to the Sketch View.

