

Here is a typical scenario for the exercise that we just completed. We would complete our model and develop the design table for various angles and sweep radii. But we would only confine ourselves to one body diameter. The configurator would then produce all of the variations of elbows for that one pipe size.

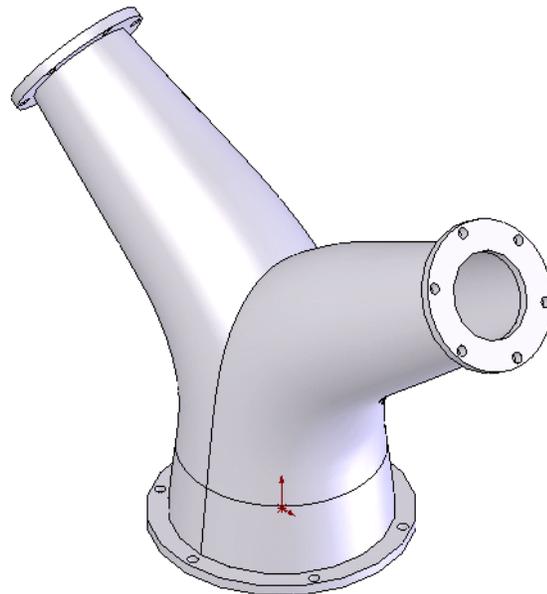
We could name the file with the pipe size as part of the name. We would then use the SAVEAS option to copy everything to a new file where we would edit the body diameter to get a new pipe size. After repeating this process for each major pipe size required, we would have an easily managed set of configurators that covered the range of pipe elbows produced by our company.

A More Complex Fitting

Let's take a few minutes to create the more complex fitting illustrated to the right.

Constructing this part is really quite simple if we take a moment to plan our attack prior to grabbing the old mouse!

Modeling more complex parts is easier if you remember the following important piece of advice:



Creating a Complex Part is Like Eating an Elephant!



Now eating an elephant is tough, because there is one heck of a lot of meat to swallow!

If you chop the elephant up and freeze the individual chunks, you can have fried elephant one day, elephant stew the next day, and so on until you eat the whole darn thing!

Complex models are tackled the same way. Break them up into a series of simpler construction steps and suddenly the problem isn't so bad!

For the part in question, we immediately notice that it is symmetrical about a vertical plane that passes through the center of the part.

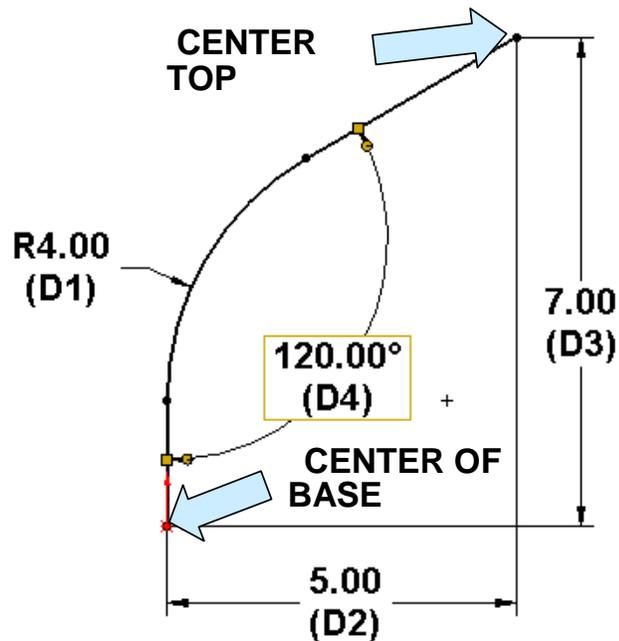
Secondly, we realize that the basic part requires a loft transition from the nominal 4 inch diameter of the base section to the nominal 2 inch diameter at the top port.

The loft transitions along a defined path that progresses upward from the base flange. The path then travels through an arc and finally extends along a straight section to the top flange.

The top flange is located on a plane that is at an angle to the horizontal plane of the base flange.

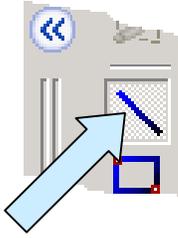
Think about what you would sketch on a cocktail napkin if you were describing this fitting to a colleague. (Yes, real engineers still sketch out ideas even though they have all of these fancy computers!)

The sketch at right shows the basic path.



Our plan of attack now takes shape:

1. We'll start by creating the path sketch on the Front Plane.
2. Then, we'll define a plane normal to the top of the oblique line in the sketch.
3. Next, we'll create the sketch for the lower profile.
4. Then, we'll create the sketch for the upper profile.
5. We'll use the two profile sketches and path to create a directed loft transition.
6. We'll then mirror the part to create the second half.
7. Next, we'll shell the part.
8. Then, we'll add the flanges and flange holes.

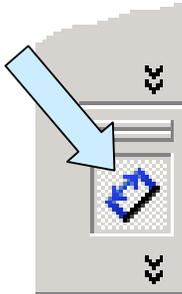
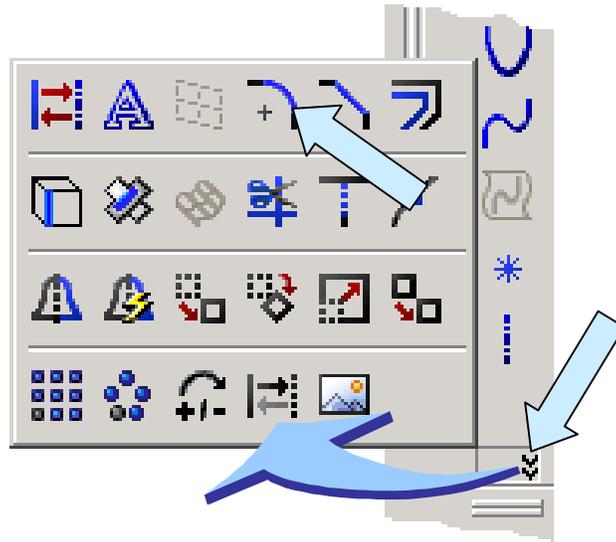


Let's start with a new part file in the default front view. Click on the Front Plane entry in the left pane.

Click on the LINE Icon. Create a short vertical line (about 3 inches long) with the bottom end starting at the origin.

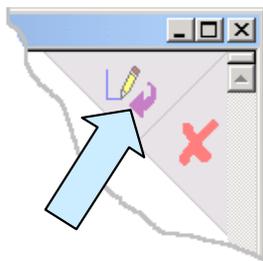
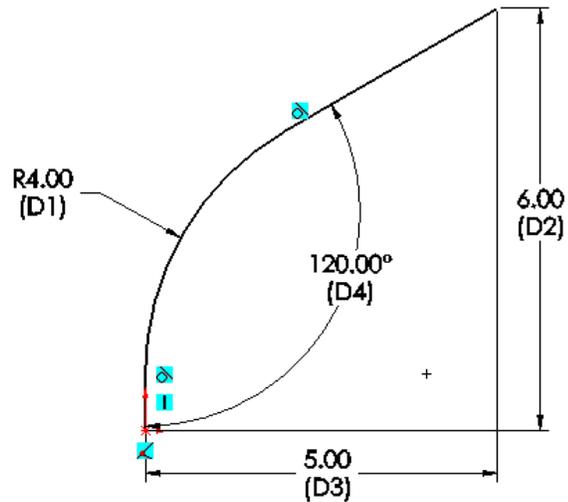
Then, create a second, connected line segment (About 4 inches long) that extends at an angle up and to the right.

Next, click on the SKETCH FILLET Icon and create a 4 inch radius fillet at the intersection of the two lines.

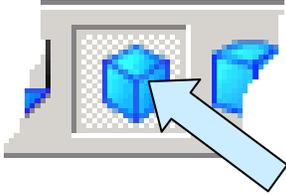


Now, click on the DIMENSION Icon and add the dimensions indicated in the illustration below to define the sketch.

Your sketch should look like this when you are done.



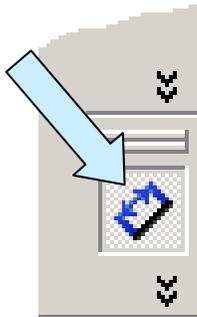
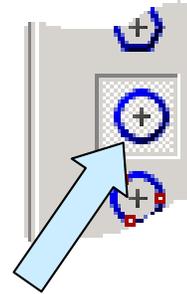
Click on the FINISH SKETCH Icon at the top, right corner of the drawing area to save the sketch.



Next, click on the Top Plane entry in the left pane.
Then, click on the ISOMETRIC VIEW Icon.

Next, click on the CIRCLE Icon.

Create a circle centered at the origin of the system.

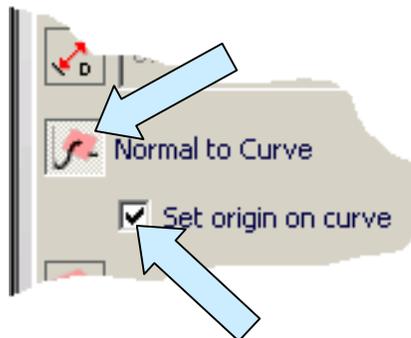
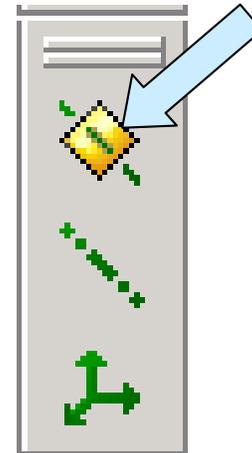


Dimension the circle at 4 inches diameter.

Then, click on the FINISH SKETCH Icon at the top, right corner of the drawing area to save the sketch.

We now need to create a sketch at the top end of the path.
First, we have to create a new oblique plane for the sketch.

Click on the PLANE Icon.



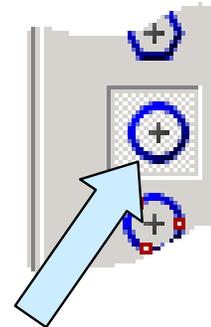
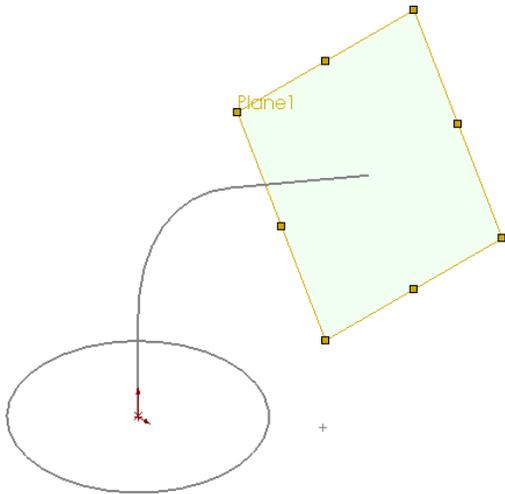
Then, click on the Normal To Curve option in the left pane.

Also, click to place a check in the Set Origin On Curve Box.

Now, click on the oblique line of the first sketch and then on the top end of the oblique line. (The line is the curve that the plane will be normal to and the top end of the line defines the actual location of the plane.)

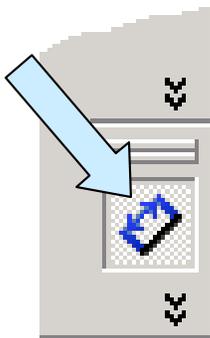
You will see a preview of the plane appear on the screen. Click on the checkmark to complete the operation.

Your screen should now look like this:

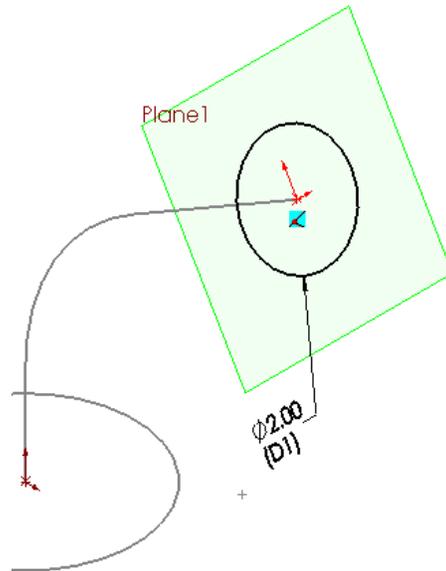


With the Plane1 highlighted, click on the CIRCLE Icon.

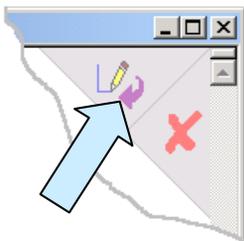
Create a circle centered on the origin of the new plane.



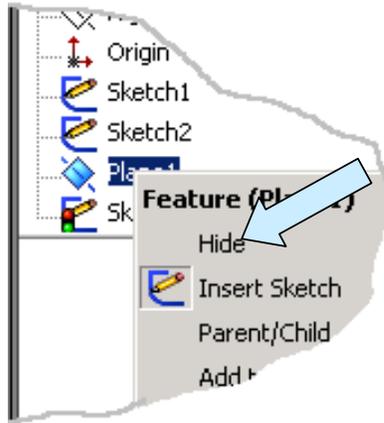
Next, dimension this circle at 2 inches diameter.



Your screen should now look like this:



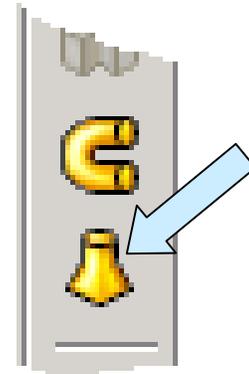
Then, click on the FINISH SKETCH Icon at the top, right corner of the drawing area to save the sketch.



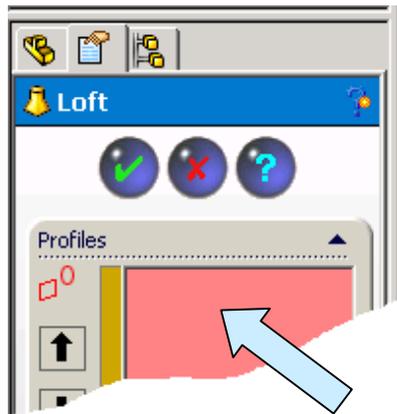
Take a moment to RIGHT MOUSE CLICK on the Plane1 entry in the left pane.

Then, click on the HIDE Option.

Now, let's create a lofted transition using the three sketches.

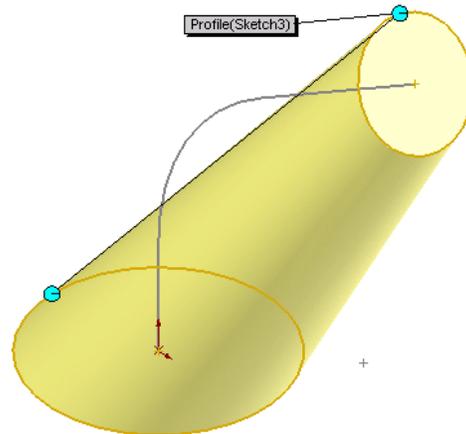


Click on the LOFT Icon.

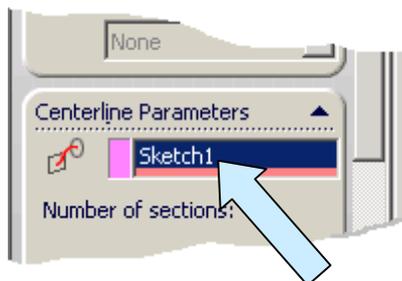


Notice that the top of the left pane is prompting you to select profiles for the loft.

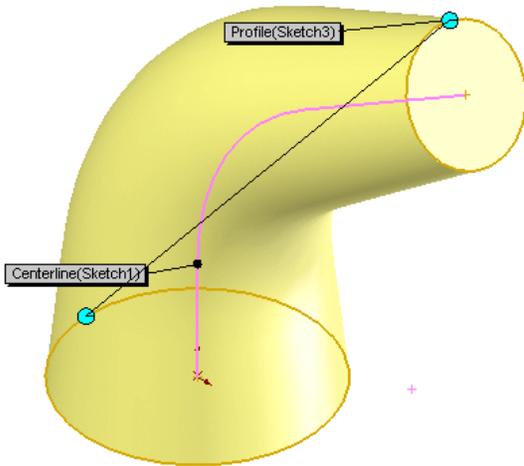
Click on each of the circles.



A preview of the loft based on just the two profiles appears.



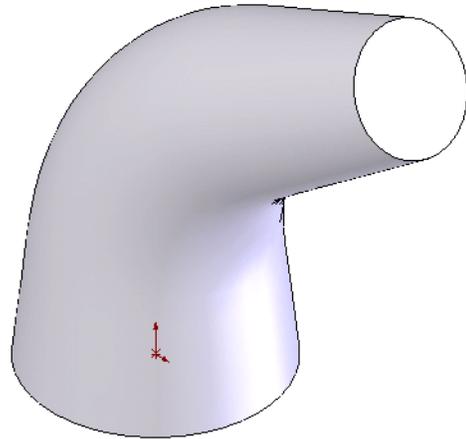
Next, click on the Pulldown Arrow to the right of the Centerline Parameters Option in the left pane. Then, click on the Sketch1 to add it to the Centerline Parameters field.



The loft preview now follows the centerline as it transitions from the larger circle to the smaller circle.

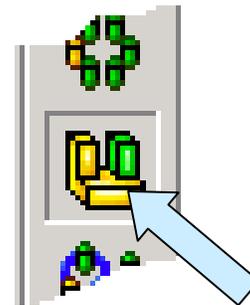
Click on the checkmark to finish.

Your part should now look like this:

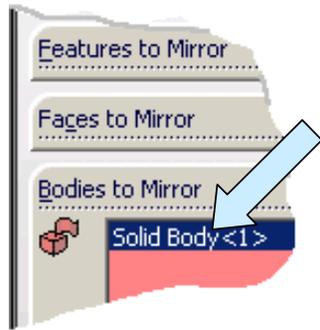


Next, click on the Right Plane Entry in the left pane.

With the right plane selected, click on the MIRROR Icon.

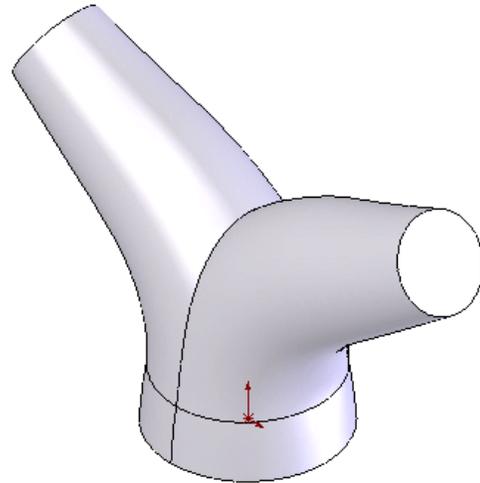


Notice that the right plane is automatically entered in the Mirror Face/Plane Field in the left pane.

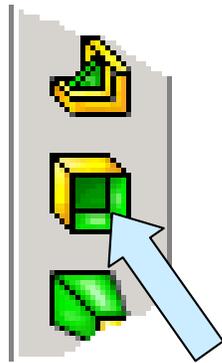


Click on the Bodies To Mirror option in the left pane. Then, click on the part.

Click on the checkmark to complete the operation.

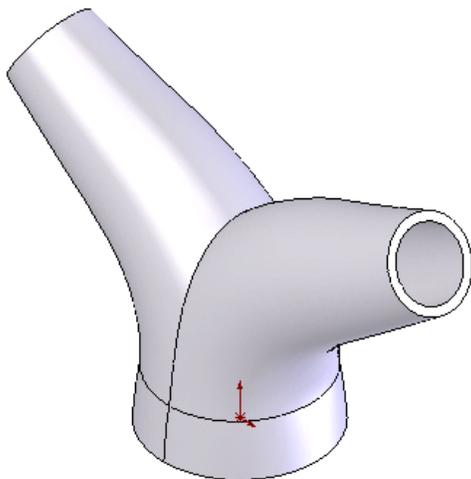
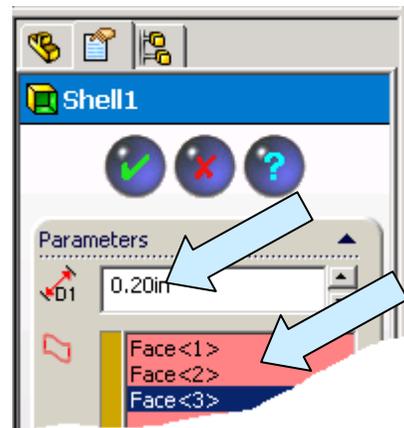


Your part should now look like this:

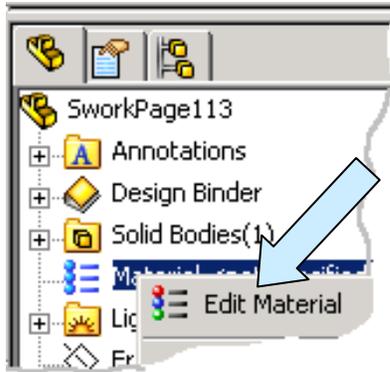


Next, click on the SHELL Icon. Type 0.2 for the Shell Thickness.

Then, select the three circular faces of the part. Click on the checkmark to complete the operation.



Your part should now look like this:

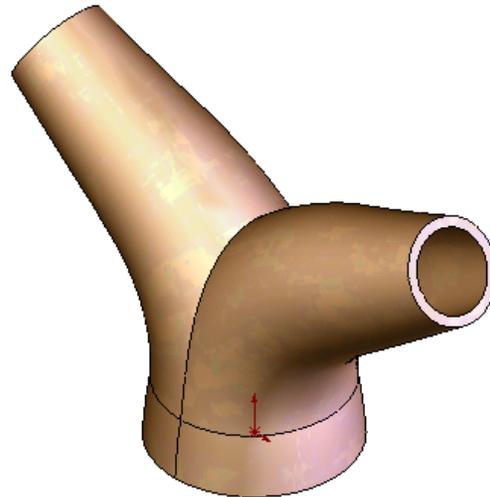


Before we proceed to the next construction step, let's take a moment to assign a material to our part.

By now, you should know the drill.

RIGHT MOUSE CLICK on the Material entry in the left pane and then click on the EDIT MATERIAL Option in the menu that appears.

I'm going to select Wrought Copper. (This part probably wouldn't be made from wrought copper, however, the color looks really snazzy in the page reprints!)



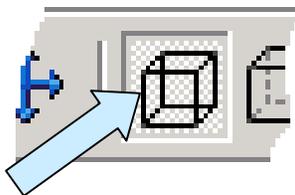
Next, let's create the flange on the bottom of the part.

In our construction of the elbow, we created the flanges first and then shelled the part.

This technique works quite well but can introduce design inflexibility if you want to make the flange thickness different from the shell thickness. Other issues also arise on more complex parts such as this one.

So let's look at a completely different way to generate the flanges that eliminates all of these issues!

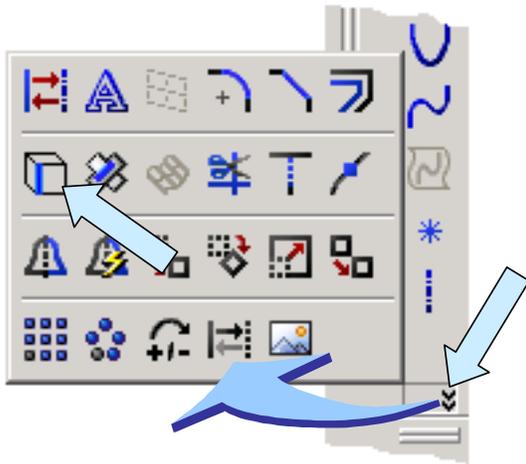
Click on the Top Plane entry in the left pane. (We'll remain in the isometric view.)



Click on the WIREFRAME Icon.

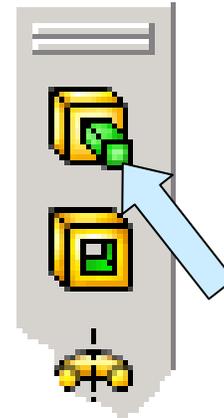
Then, click on the SKETCH Icon.



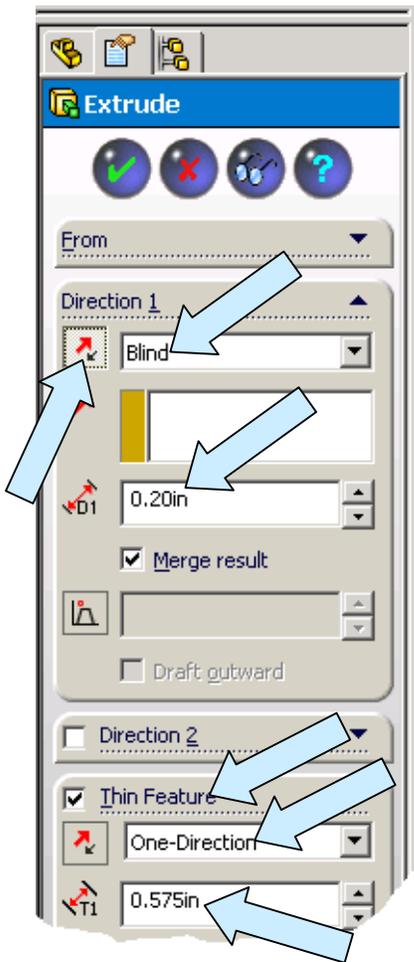


Now, depress the CTRL Key while you select the two inner arcs on the bottom of the part.

Then, click on the CONVERT ENTITIES Icon.



Next, click on the EXTRUDED BOSS/BASE Icon.



We'll use the default Blind Option.

Click on the REVERSE DIRECTION Icon to drive the extrusion downward from the sketch.

Type 0.2 for the Distance.

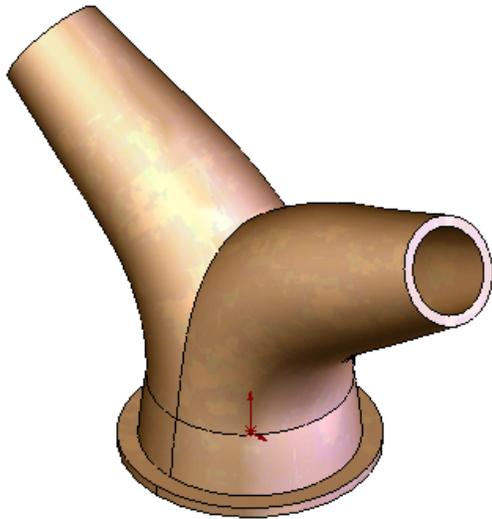
Now just using these settings would give us a solid disk across the bottom end of the part.

If you click on the Thin Feature Option in the lower half of the left pane, however, you now have the ability to create a washer-shaped flange.

We'll use the One-Direction Option.

Type 0.575 for the thickness.

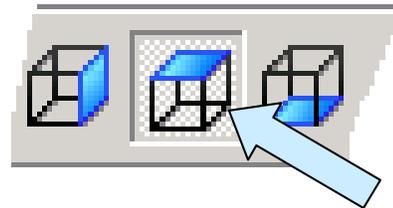
Then, click on the checkmark at the top of the left pane to finish.



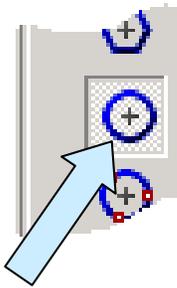
You will now have a flange on the bottom of the part.

Now, let's create a single hole on the flange.

Click on the Top Plane entry in the left pane.



Then, click on the TOP VIEW Icon.



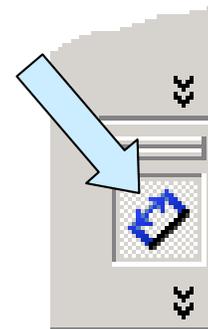
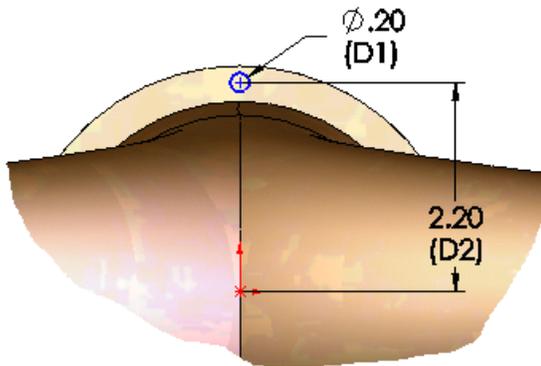
Next, click on the CIRCLE Icon.

Position the cursor over the origin until a solid dot appears.

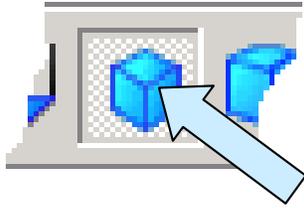
Then, move the cursor directly upward. Click while the dashed inference line indicating vertical alignment is visible.

Now, dimension the circle at 0.196 Diameter.

Also, place a 2.2 inch vertical dimension from the center of the circle to the origin.

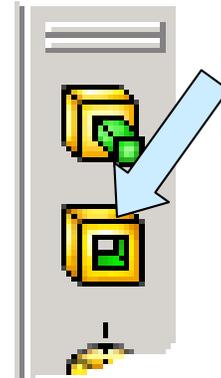


Your sketch should look like this:



Click on the ISOMETRIC VIEW Icon.

Then, click on the EXTRUDED CUT Icon.

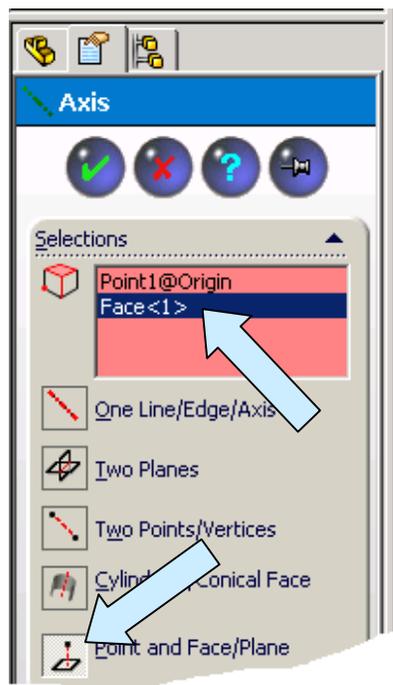
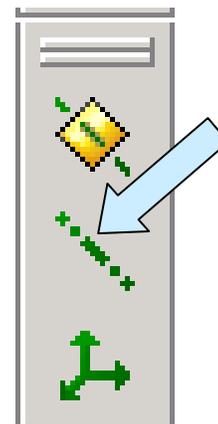


Click on the Up To Next Option in the left pane.

Then, click on the checkmark to complete the operation. You now have one hole in the flange.

Let's create the five additional holes on the flange.

First, click on the AXIS Icon.

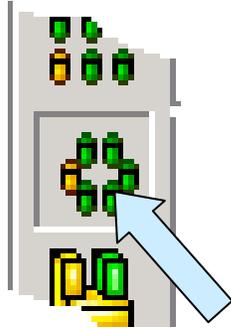


Next, click on the Point and Face/Plane Option in the left pane.

Now, click on the origin and then on the top surface of the base flange.

Click on the OK Button.

You will now have a vertical axis centered in the base of the part.



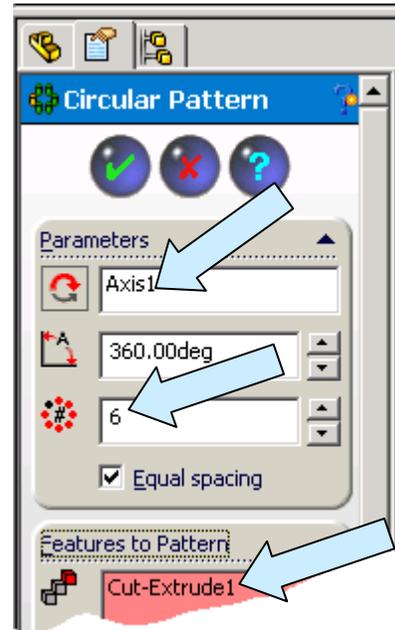
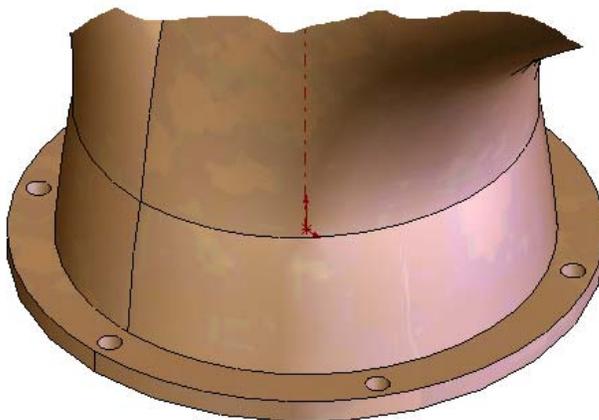
With the axis still selected, click on the CIRCULAR PATTERN Icon.

Notice that the axis is automatically entered in the Parameters field since it was selected when we entered the function.

Click on the Features To Pattern field and then click on the hole that you just created.

Type 6 for the number of instances. We'll use the default 360 degrees and Equal Spacing options.

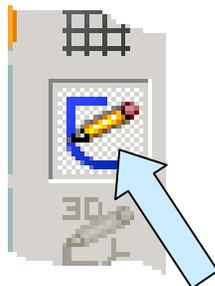
Click on the checkmark to finish.



Your part will now have six, equally spaced holes on the flange.

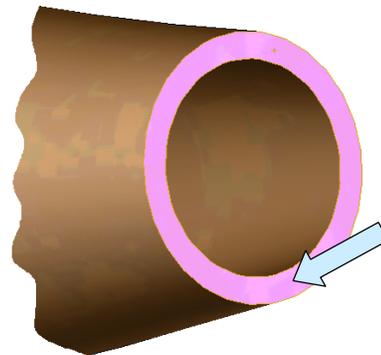
Let's use a similar approach to create a flange with six holes on the right arm of the part.

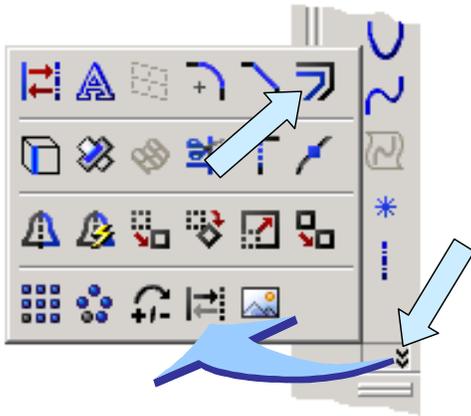
Start by selecting the washer-shaped rim surface of the right arm.



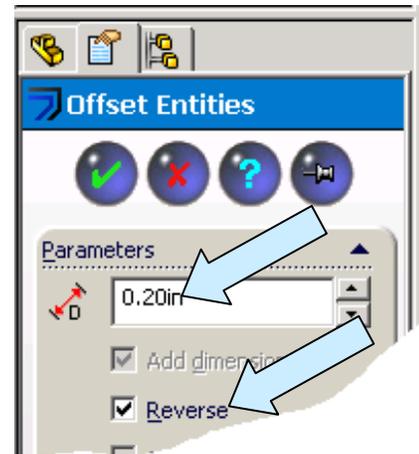
Click on the SKETCH Icon.

Select the outer circular edge on the rim surface.

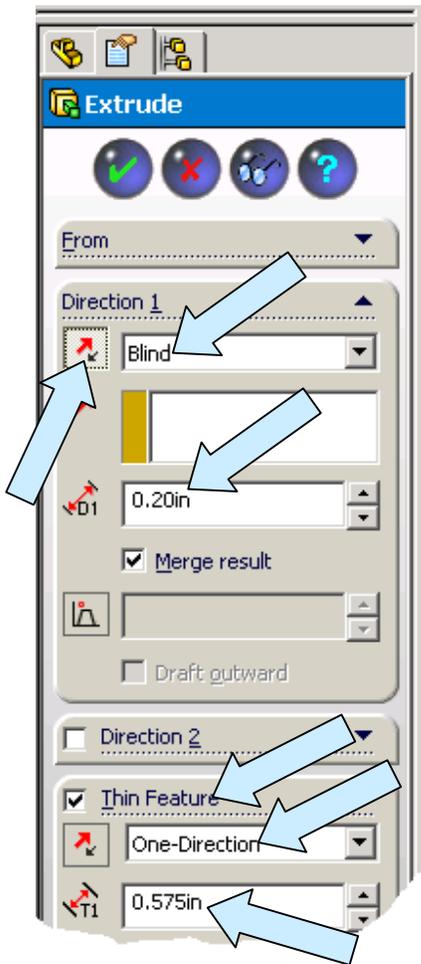




Then, click on the OFFSET ENTITIES Icon.

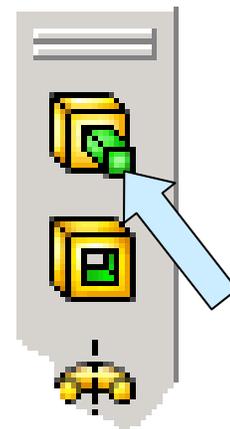


Type 0.20 and click on the Reverse Option. This creates a circle that lies superimposed on the inner edge of the opening. Click on the green checkmark.



NOTE: The outer edge of this arm is a true circular edge, derived from our original circle used as a profile for the loft. The inner edge, however, is not a true circle since it derived from the complex shell operation. We'll need our outer flange edge to be a true circle (If we offset it from a true circle, it will be a true circle.) for future operations.

Now, click on the EXTRUDED BOSS/BASE Icon.

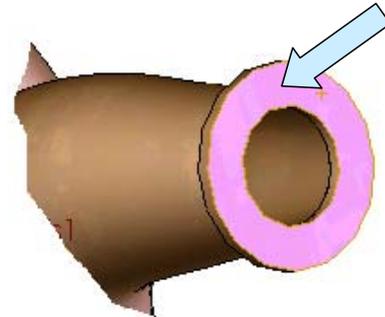
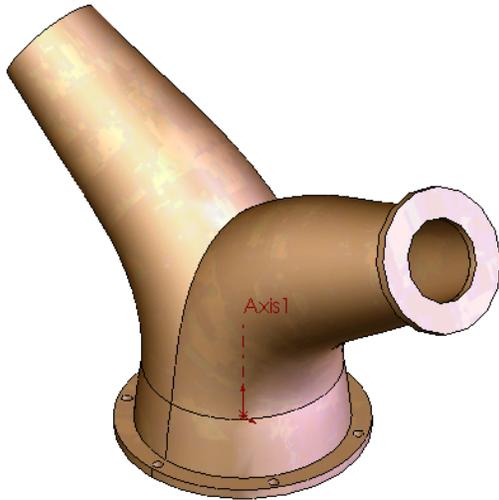


We'll use the same approach that we used for the base flange.

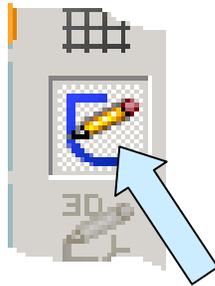
Use the default Blind Option and a Distance of 0.2 inches. Click on the Thin Feature Option.

Use the One-Direction Option and type 0.575 for the Thickness. Then, click on the checkmark to finish.

Your part should now look like this:



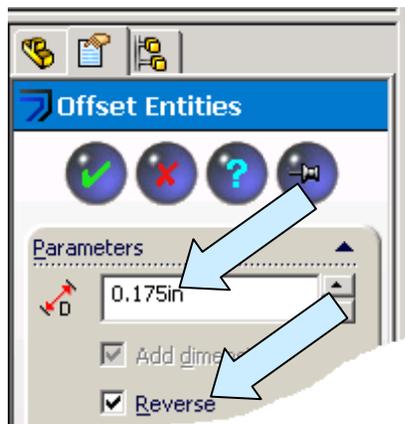
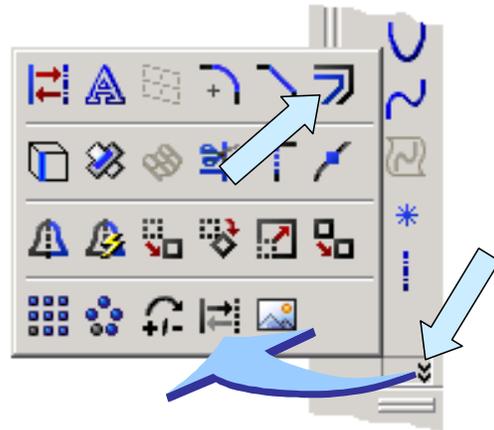
Now, let's create a hole in the top flange. Click on the flange surface.



Then, click on the SKETCH Icon.

Now, click on the outer, circular edge of the flange.

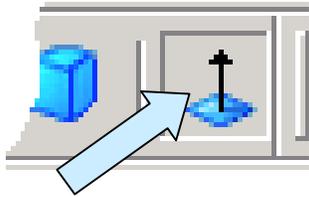
Then, click on the OFFSET ENTITIES Icon.



Type 0.175 for the Offset Distance. Then, click on the Reverse Option.

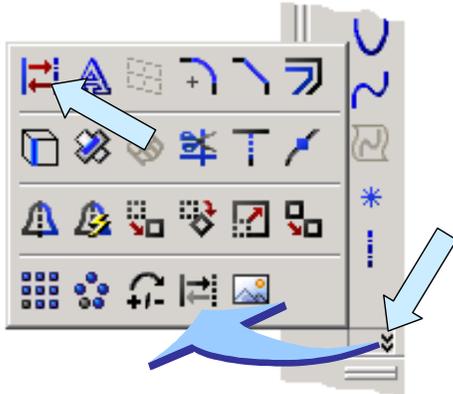
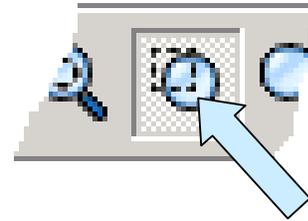
This is an alternate approach to creating a bolt circle for a flange. In this approach we specify the circle based on an offset distance from the flange edge.

Click on the checkmark to finish.



Then, click on the NORMAL TO VIEW Icon.

Now, click on the ZOOM TO AREA Icon and drag a selection window around the flange to magnify it.



Click on the circle that you just created to select it.

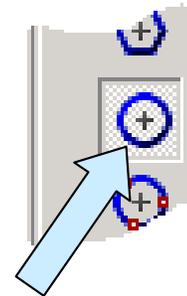
Then, click on the CONSTRUCTION GEOMETRY Icon.

This changes the circle to a piece of construction geometry.

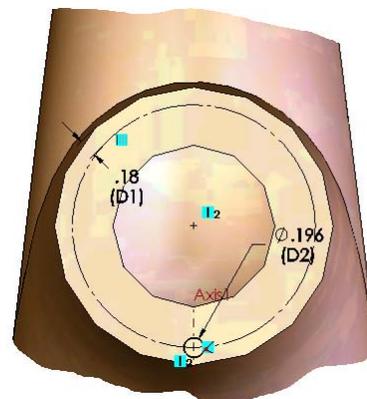
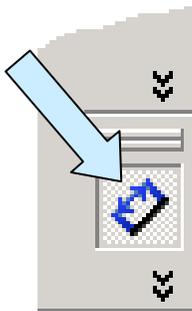
Next, click on the CIRCLE Icon.

Now, move the cursor over the lower, vertical axis on the part.

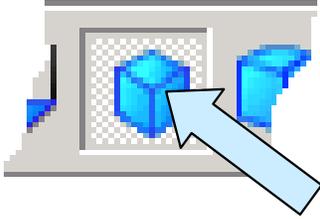
As you move upward, a dashed vertical inference line appears. Pause as you cross the lower edge of the construction circle and a Coincident Symbol appears indicating that the circle center is also inferred to lie on the circle. Click to fix the circle center at this point and then drag the cursor outward, clicking to set the circle.



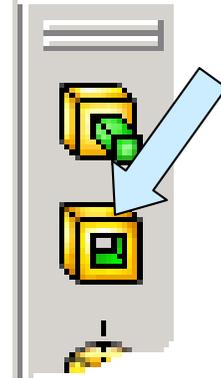
Dimension the circle at 0.196 Diameter.



Your sketch should now look like this:



Click on the ISOMETRIC VIEW Icon.

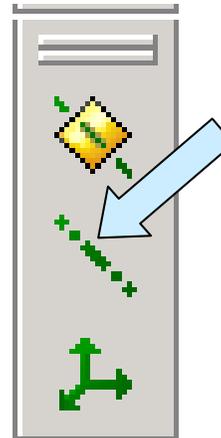


Now, click on the EXTRUDED CUT Icon.



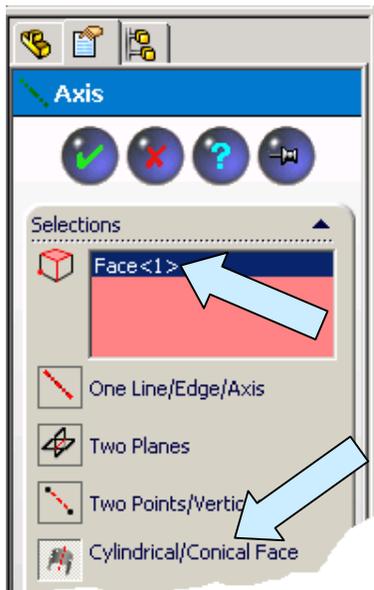
Click on the Up To Next Option in the left pane.

Then, click on the Checkmark to finish.



Next, let's create the other five holes on this flange.

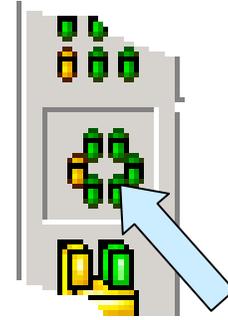
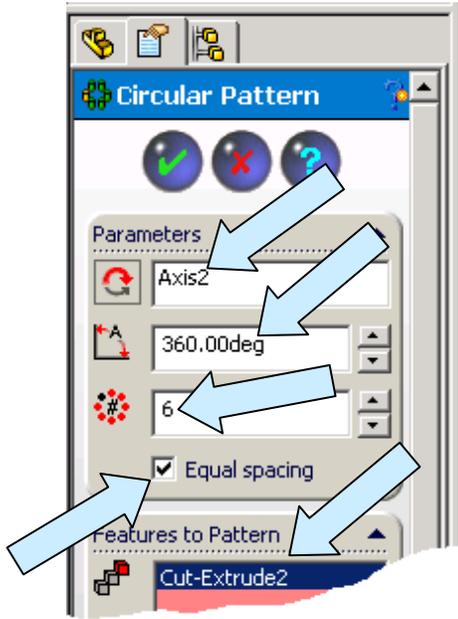
Start by clicking on the AXIS Icon.



Now, click on the Cylindrical/Conical Surface Option and then on the narrow, cylindrical surface at the rim of the flange.

You will now have an axis centered in the upper right flange.

With the Axis selected, click on the CIRCULAR PATTERN Icon.



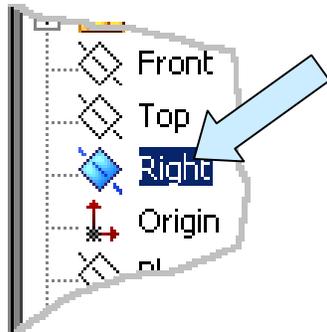
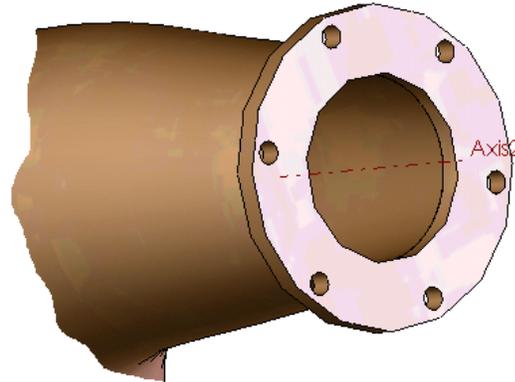
Notice that the pre-selected Axis2 is entered in the Parameters field.

We'll use the default 360 degree option. Type 6 for the Number of Instances. We'll also use the Equal Spacing Option.

Click on the Features to Pattern field in the left pane. Then, click on the single hole that you created in the flange.

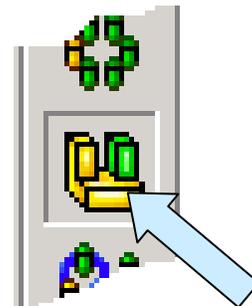
Notice that a preview of the five additional holes appears on the part. Click on the checkmark at the top of the left pane to complete the operation.

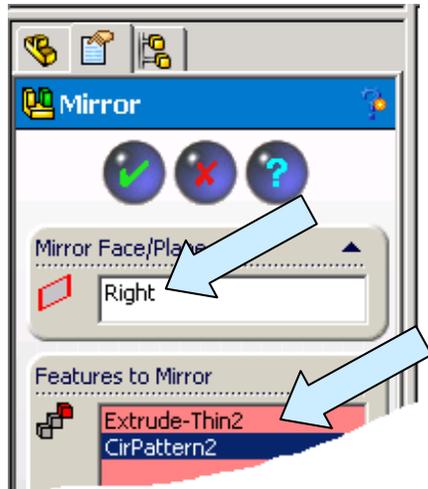
The side flange should now look like this:



Next, click on the Right Plane entry in the left pane.

Then, click on the MIRROR Icon.





Notice that the pre-selected Right Plane is entered in the Mirror Face/Plane field in the left pane.

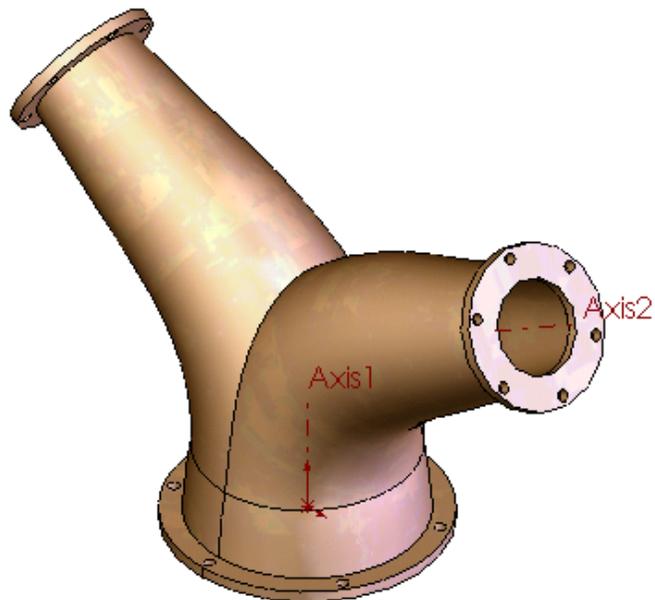
Next, click on the Features To Mirror field in the left pane.

Then, click on the flange and one of the holes in the circular pattern. These features are added to the Features To Mirror list. (Note: You only need to click on one of the hole copies. Since the hole belongs to a pattern, all of the holes in the pattern are automatically copied.)

Click on the Checkmark to finish.

Your completed part should look like this:

Take a moment to save the file. Let's call it "SworkPage125."



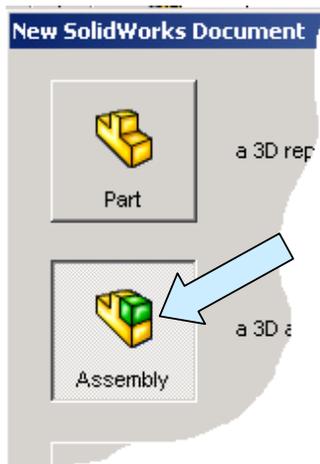
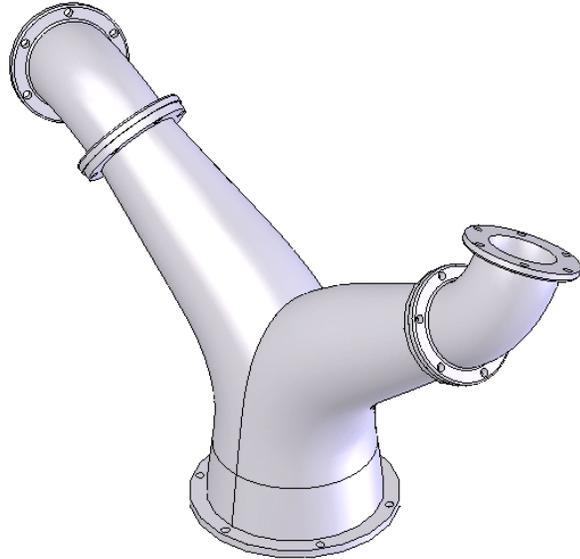
A Pipe Fitting Assembly

Let's take a moment to create a simple assembly consisting of two pipe elbows and our complex pipe fitting that we just completed.

I've illustrated the assembly to the right.

You should still have your SworkPage125 file open.

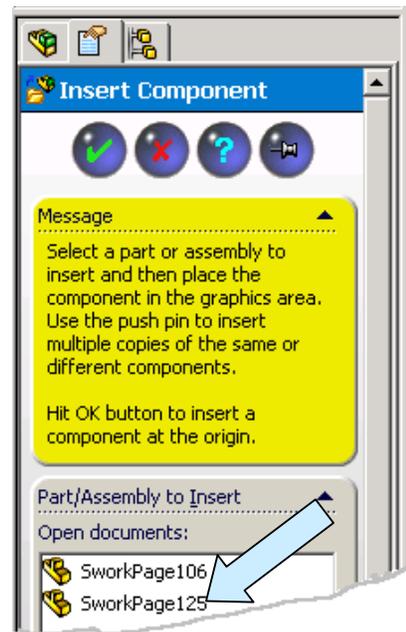
Also, open the SworkPage106 file.
(This is your elbow configurator file.)



Now, click on the FILE NEW Icon and then select the Assembly Option.

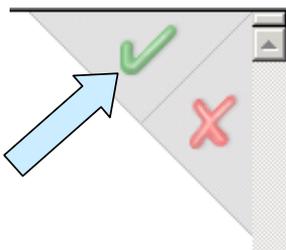
The Insert Component Dialog fills the left pane.

Notice that the two open part files appear in the Open Documents Field in the left pane.



Click on the SworkPage125 File.

Notice that a copy of the complex fitting appears in the graphics area.



Click on the green checkmark at the top, right corner of the graphics area to position this fitting at the origin of the assembly file.