

Next, let's enter a standard formula for calculation of the Bolt Circle Radius in each cell of column F. In cell F3 type the formula  $= (E3 - 0.35) / 2$  to replace the current 1.2 value.

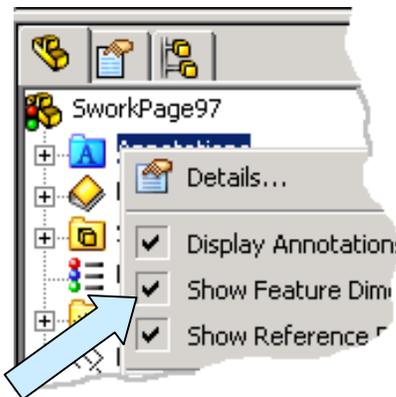
Notice that the value remains at 1.2.

Now copy the formula from cell F3 into cells F4 through F7.

Finally, let's enter formulas in the cells of column C to automatically make the Sweep Radius equal to the Body Diameter plus 0.5 inches. I'll let you do that on your own.

When you are done, your design table should look like this:

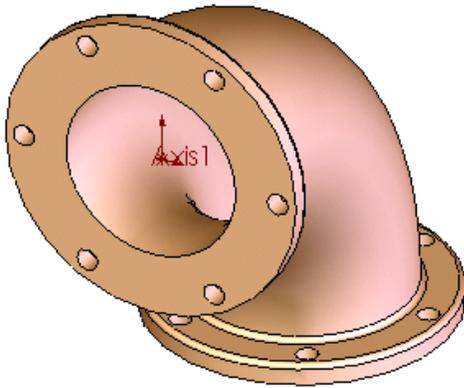
	A	B	C	D	E	F
1	Design Table for: SworkPage97					
2		BodyDia@Sketch1	SweepRad@Sketch1	HalfAngle@Revolve1	FlangeDia@Sketch2	BoltCirRad@Sketch3
3	2 Inch 90 Degree Elbow	2	2.5	45	2.75	1.2
4	2 Inch 60 Degree Elbow	2	2.5	30	2.75	1.2
5	2 Inch 45 Degree Elbow	2	2.5	22.5	2.75	1.2
6	3 Inch 90 Degree Elbow	3	3.5	45	3.75	1.7
7	3 Inch 60 Degree Elbow	3	3.5	30	3.75	1.7



Click anywhere outside of the design table to return to the SolidWorks environment.

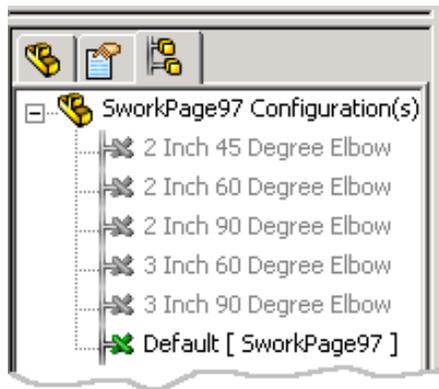
RIGHT MOUSE CLICK on the Annotations entry in the left pane.

Then, click on the Show Feature Dimensions to clear them from the display.



Your part should now look like this:

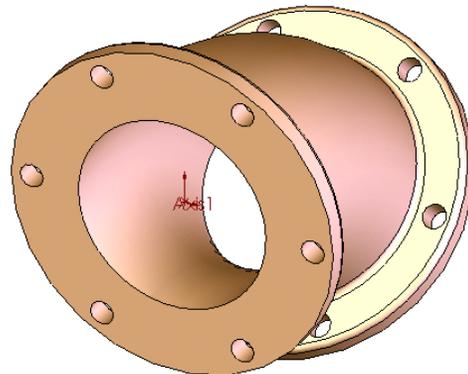
Now, click on the CONFIGURATION MANAGER Icon at the top of the left pane.



The left pane changes to present you with a list of the configurations generated by the design table.

Double click on the 2 Inch-45 Degree Elbow entry.

Notice that the model rebuilds to this configuration.



Try each of the other configurations to see the results. Save your file as “SworkPage106.”

Now in this exercise we built a simple configurator where we can change the body diameter and angle of sweep. We could have made a much more elaborate configurator where the number of holes on the flanges, thickness of the flanges, and other characteristics were also controlled by columns in the design table.

There may be situations where this will be appropriate, however, you want to carefully weigh the tradeoffs involved when creating an overly complex design table. In some situations, design teams will opt for making an initial simple configurator in a part file, and then using the SAVEAS Option to create several duplicates of the part file. Each Part file then can be modified to represent one particular characteristic of the part.

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.

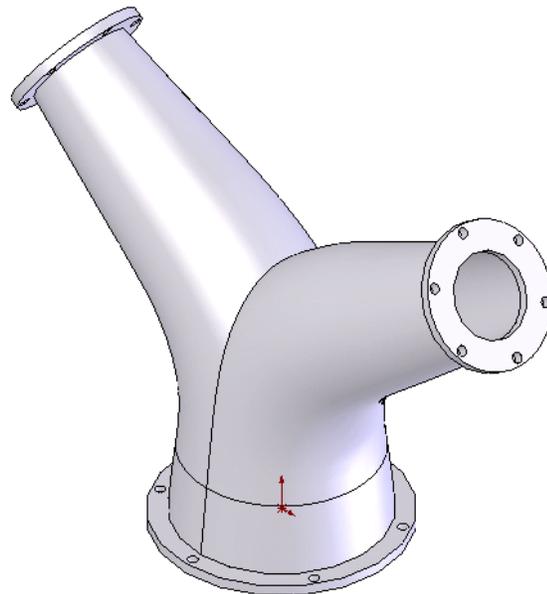
We could name the file with the pipe size as part of the name. We would then use the SAVEAS option to copy everything to a new file where we would edit the body diameter to get a new pipe size. After repeating this process for each major pipe size required, we would have an easily managed set of configurators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

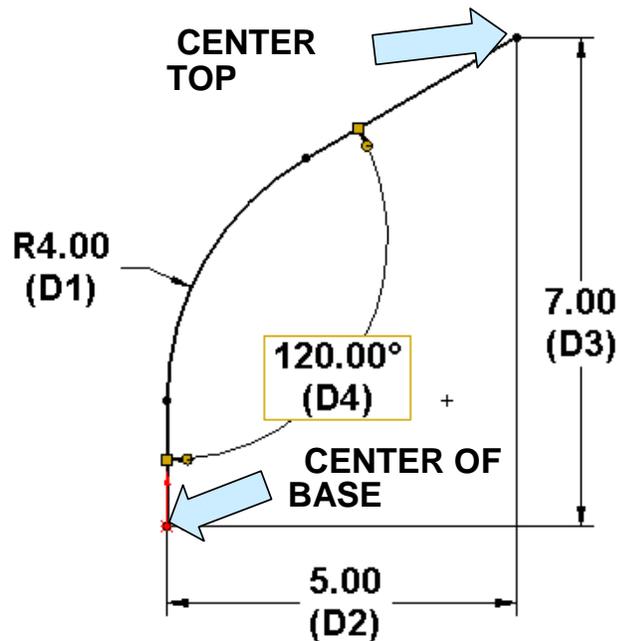
Secondly, we realize that the basic part requires a loft transition from the nominal 4 inch diameter of the base section to the nominal 2 inch diameter at the top port.

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.

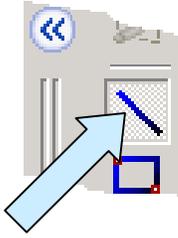
Think about what you would sketch on a cocktail napkin if you were describing this fitting to a colleague. (Yes, real engineers still sketch out ideas even though they have all of these fancy computers!)

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.

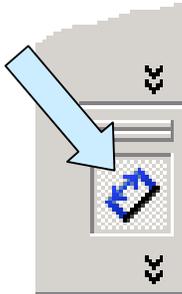
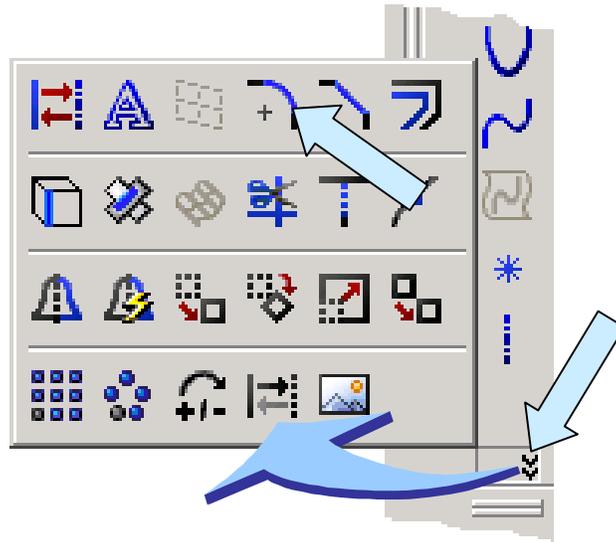


Let's start with a new part file in the default front view.  
Click on the Front Plane entry in the left pane.

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

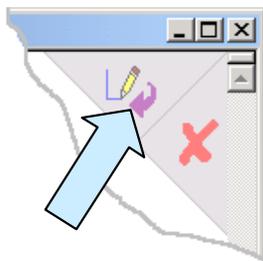
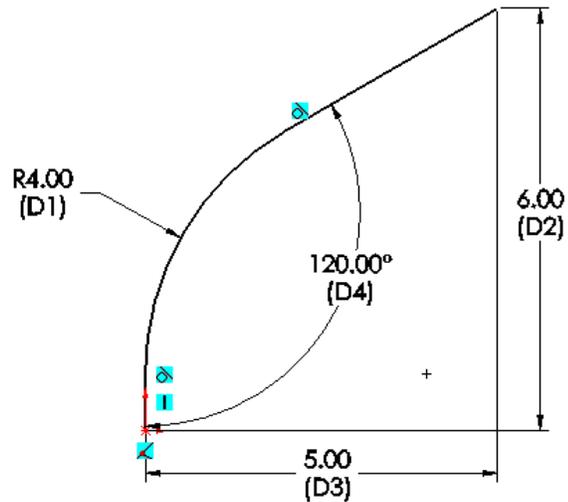
Then, create a second, connected line segment (About 4 inches long) that extends at an angle up and to the right.

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

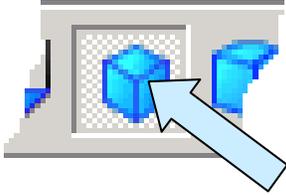


Now, click on the DIMENSION Icon and add the dimensions indicated in the illustration below to define the sketch.

Your sketch should look like this when you are done.



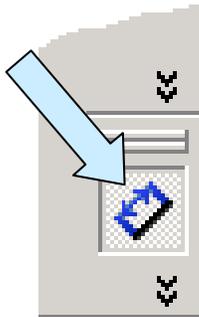
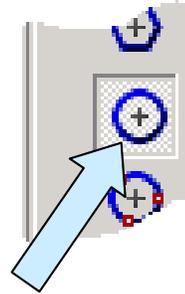
Click on the FINISH SKETCH Icon at the top, right corner of the drawing area to save the sketch.



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.

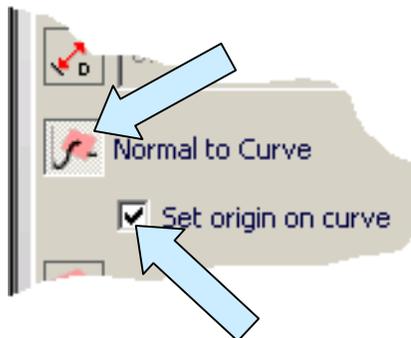
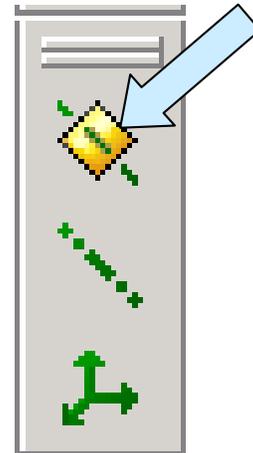


Dimension the circle at 4 inches diameter.

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.

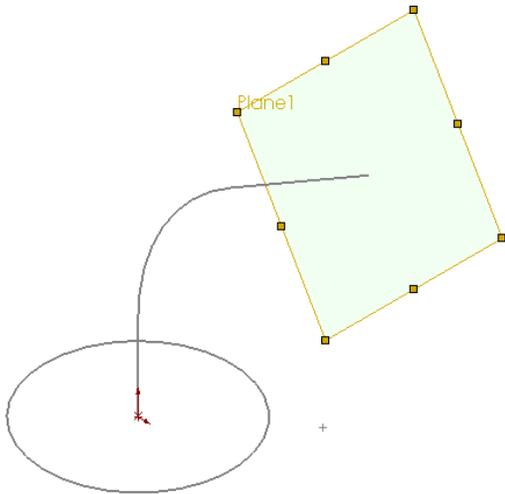
Click on the PLANE Icon.



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

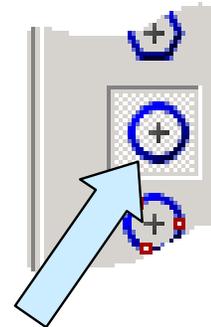
Also, click to place a check in the Set Origin On Curve Box.

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.)



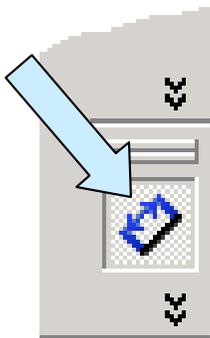
You will see a preview of the plane appear on the screen. Click on the checkmark to complete the operation.

Your screen should now look like this:

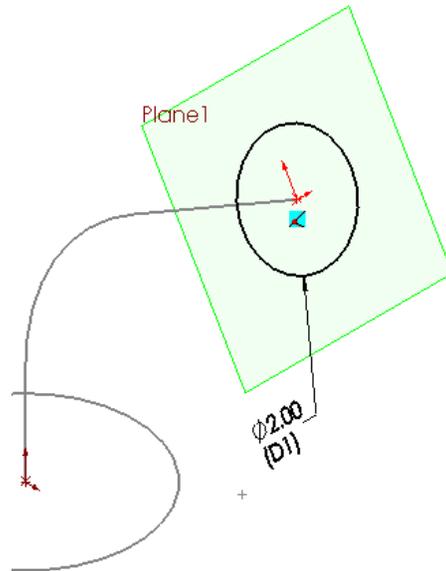


With the Plane1 highlighted, click on the CIRCLE Icon.

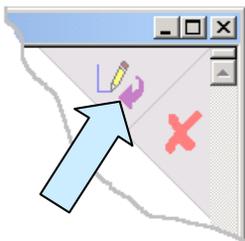
Create a circle centered on the origin of the new plane.



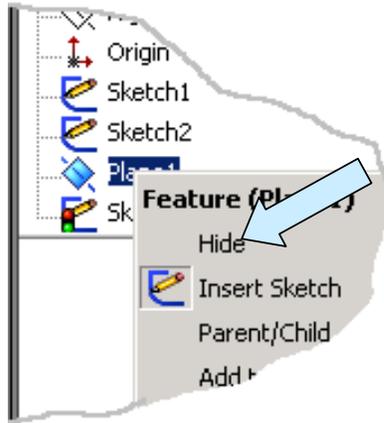
Next, dimension this circle at 2 inches diameter.



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.

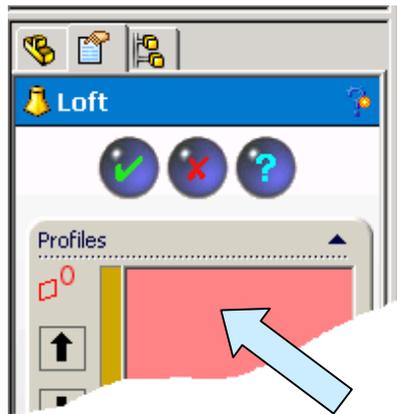
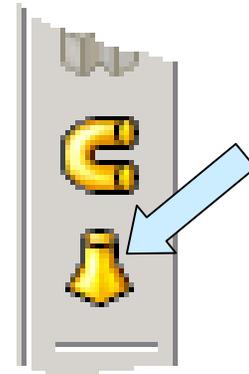


Take a moment to RIGHT MOUSE CLICK on the Plane1 entry in the left pane.

Then, click on the HIDE Option.

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

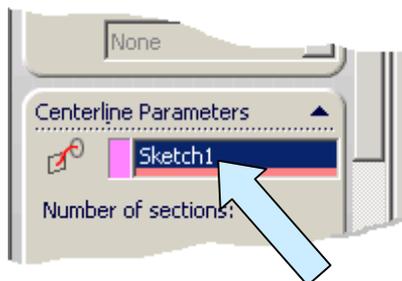
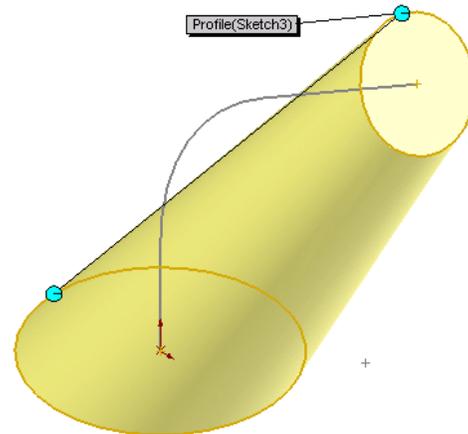
Click on the LOFT Icon.



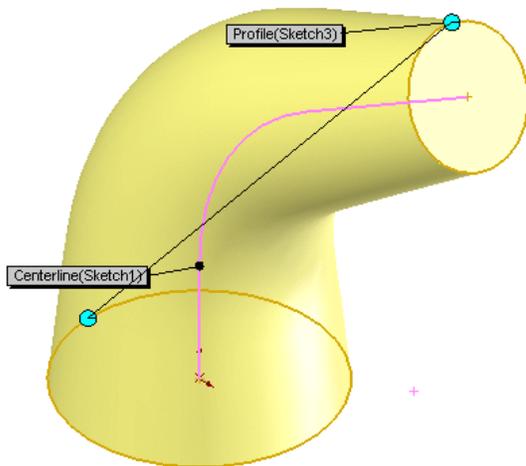
Notice that the top of the left pane is prompting you to select profiles for the loft.

Click on each of the circles.

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



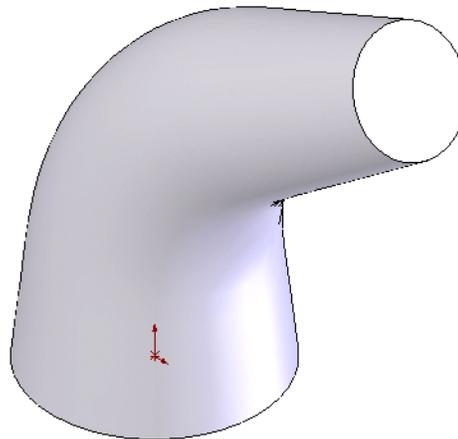
Next, click on the Pulldown Arrow to the right of the Centerline Parameters Option in the left pane. Then, click on the Sketch1 to add it to the Centerline Parameters field.



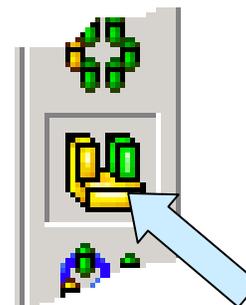
The loft preview now follows the centerline as it transitions from the larger circle to the smaller circle.

Click on the checkmark to finish.

Your part should now look like this:



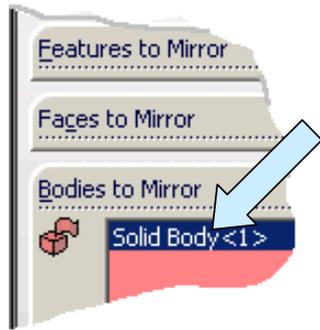
Next, click on the Right Plane Entry in the left pane.



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

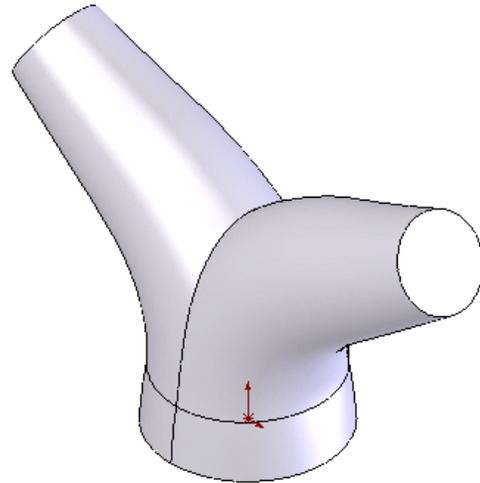


Notice that the right plane is automatically entered in the Mirror Face/Plane Field in the left pane.

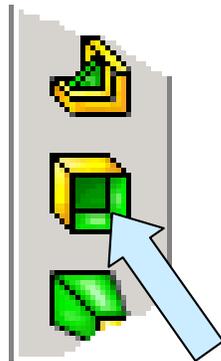


Click on the Bodies To Mirror option in the left pane. Then, click on the part.

Click on the checkmark to complete the operation.

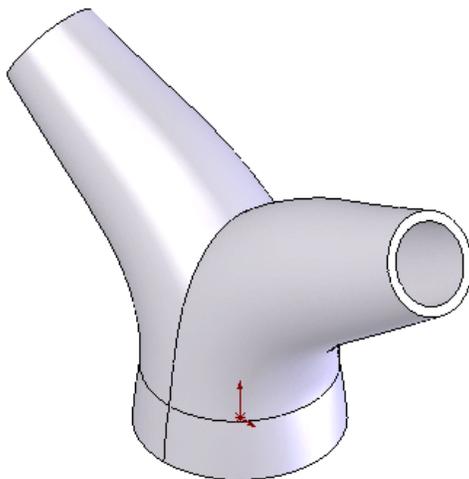
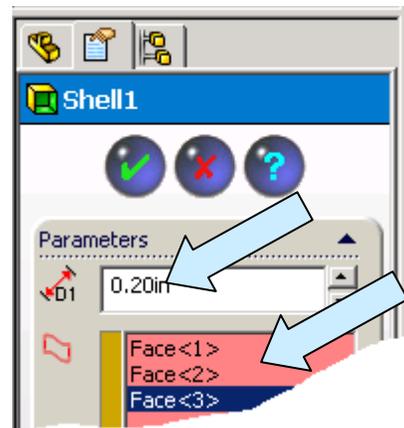


Your part should now look like this:



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:

