

	A	B	C	D	E	F	G
1	Design Table for: SworkPage97						
2		BodyDia@Sketch1	SweepRad@Sketch1	HalfAngle@Revolve1	FlangeDia@Sketch2	BoltCirRad@Sketch3	
3	Default	2	2.5	45	2.75	1.2	
4							
5							
6							
7							
8							
9							
10							

Your Design Table should now look like this:

Also notice that the toolbars at the top of your screen have been replaced by Spreadsheet Toolbars.

Notice that there is a column for each of the variables that you selected. The first column in row 3 contains the Default label cells B3 through F3 contain the values of the respective variables.

Now we could double-click on a given cell from B3 through F3 and type a new value to generate a different elbow.

For certain design situations, this may be all you will want to do. However, let's look at a more sophisticated way to use the Design Table to create a whole family of fittings.

Start by positioning the cursor over the vertical line between the "A" Column heading and the "B" Column heading. When the cursor changes to a double-headed arrow, drag the border to the right about two inches to make the "A" column wider. (Remember that this is just a simple spreadsheet, so the standard tricks that you already use in spreadsheets will work here also!)

	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						

Next, double-click on the cell A3 and type "2 Inch-90 Degree Elbow" for the entry.

Next, select cells A3 through F3 in the design table.

	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 90 Degree Elbow	2	2.5	45	2.75	1.2
5						

Then use the Hot Key CTRL + C to copy the selection to the Windows Clipboard.

Now, click in cell A4 and use the Hot Key CTRL + V to paste your selection into the fourth row.

Your design table should now look like this:

Paste the same information into the next three rows using the Hot Key CTRL + V.

Your design table should now 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 90 Degree Elbow	2	2.5	45	2.75	1.2
5	2 Inch 90 Degree Elbow	2	2.5	45	2.75	1.2
6	2 Inch 90 Degree Elbow	2	2.5	45	2.75	1.2
7	2 Inch 90 Degree Elbow	2	2.5	45	2.75	1.2

Now, edit the names in cells A4 through A7 so that they match the following list:

- 2 Inch-60 Degree Elbow
- 2 Inch-45 Degree Elbow
- 3 Inch-90 Degree Elbow
- 3 Inch-60 Degree Elbow

Column A in the design table should now look like this:

2	
3	2 Inch 90 Degree Elbow
4	2 Inch 60 Degree Elbow
5	2 Inch 45 Degree Elbow
6	3 Inch 90 Degree Elbow
7	3 Inch 60 Degree Elbow

		Body	Sweep	Half	Flange	Bolts
2						
3	2 Inch 90 Degree Elbow	2	2.5	45	2.75	1.2
4	2 Inch 60 Degree Elbow	2	2.5	45	2.75	1.2
5	2 Inch 45 Degree Elbow	2	2.5	45	2.75	1.2
6	3 Inch 90 Degree Elbow	3	2.5	45	3.75	1.2
7	3 Inch 60 Degree Elbow	3	2.5	45	3.75	1.2
8						

Now the flange diameter values in column E will automatically update whenever you enter new body diameter values in column B.

Next, edit the Half Angle values so that they are half the value indicated in the Column A labels.

Cells D3 and D6 remain at 45 degrees.
 Cells D4 and D7 should be 30 degrees.
 Cell D5 should be 22.5 degrees.

Increase the width of column D so that the entire 22.5 degree entry is visible.

Your design table should now 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	2.5	45	3.75	1.2
7	3 Inch 60 Degree Elbow	3	2.5	30	3.75	1.2
8						

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.

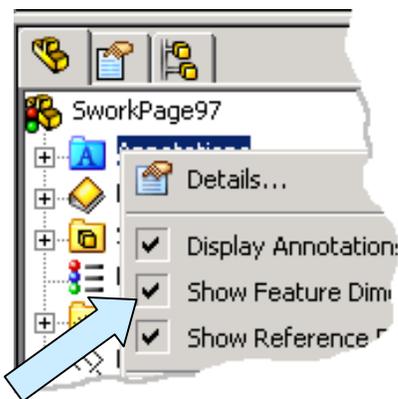
Notice that the value remains at 1.2.

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:

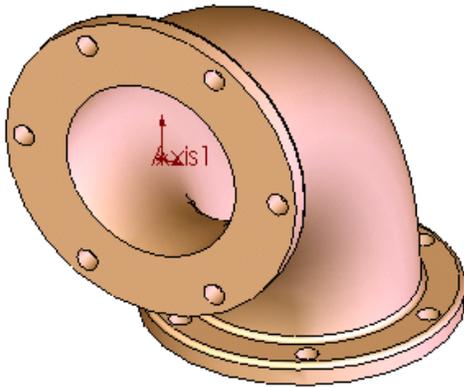
	A	B	C	D	E	F
1	Design Table for: SworkPage97					
2		BodyDia@Sketch1	SweepRad@Sketch1	HalfAngle@Revolve1	FlangeDia@Sketch2	BoltCirRad@Sketch3
3	2 Inch 90 Degree Elbow	2	2.5	45	2.75	1.2
4	2 Inch 60 Degree Elbow	2	2.5	30	2.75	1.2
5	2 Inch 45 Degree Elbow	2	2.5	22.5	2.75	1.2
6	3 Inch 90 Degree Elbow	3	3.5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

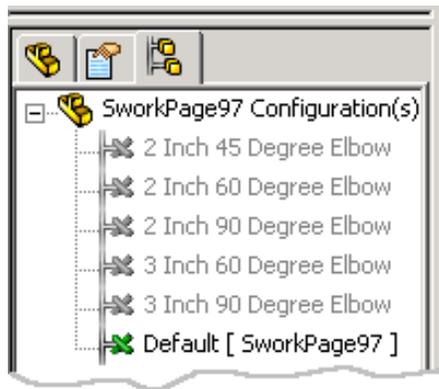
RIGHT MOUSE CLICK on the Annotations entry in the left pane.

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



Your part should now look like this:

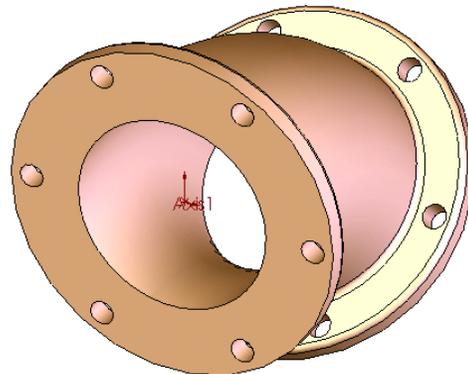
Now, click on the CONFIGURATION MANAGER Icon at the top of the left pane.



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.

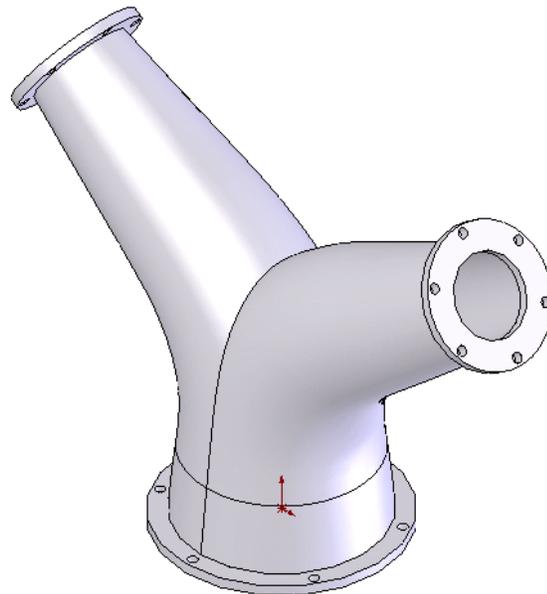
We could name the file with the pipe size as part of the name. We would then use the SAVEAS option to copy everything to a new file where we would edit the body diameter to get a new pipe size. After repeating this process for each major pipe size required, we would have an easily managed set of configurators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

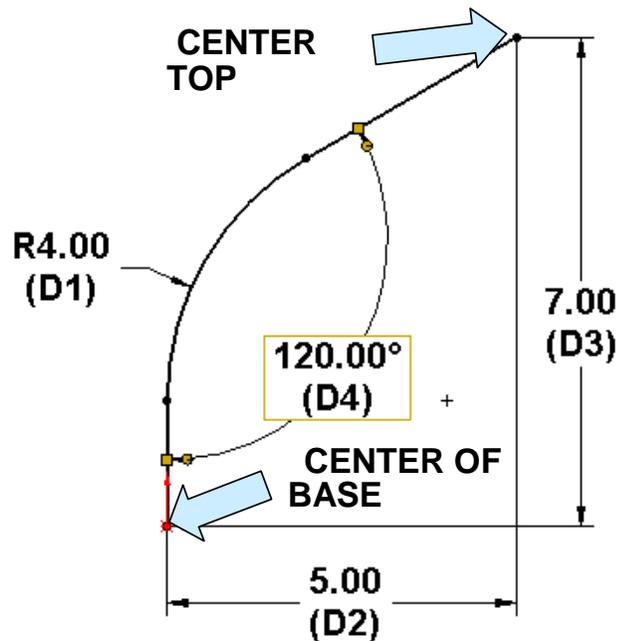
Secondly, we realize that the basic part requires a loft transition from the nominal 4 inch diameter of the base section to the nominal 2 inch diameter at the top port.

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.

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