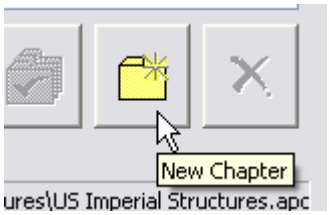
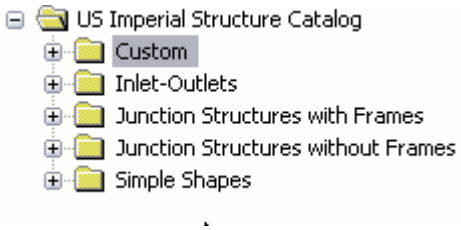
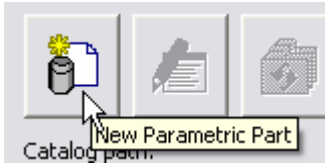
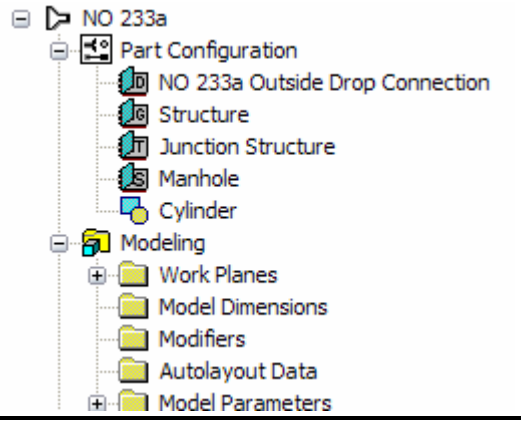
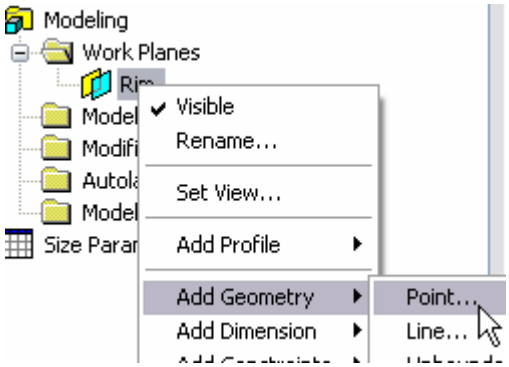
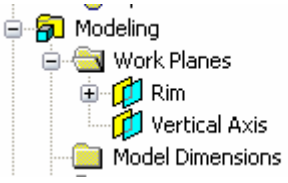
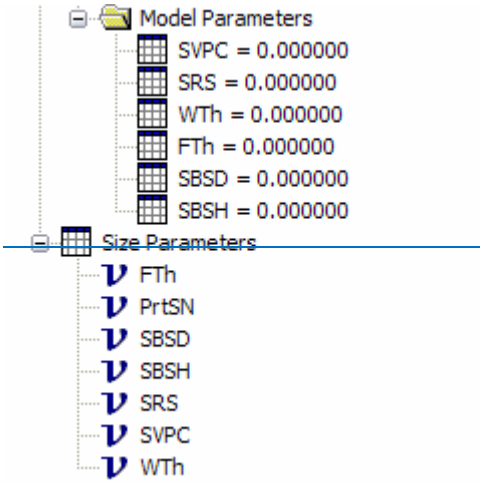
NO 233a
Part 1: Getting Started

In this exercise you create a new part chapter, a new part family within the new chapter, and configure the Part Builder environment to begin modeling the part. We'll use an alternate technique than that used in the previous exercise.

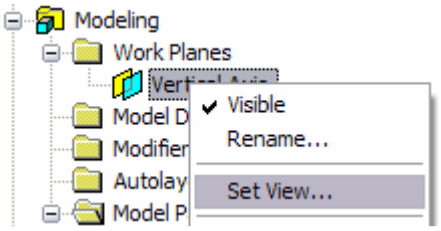
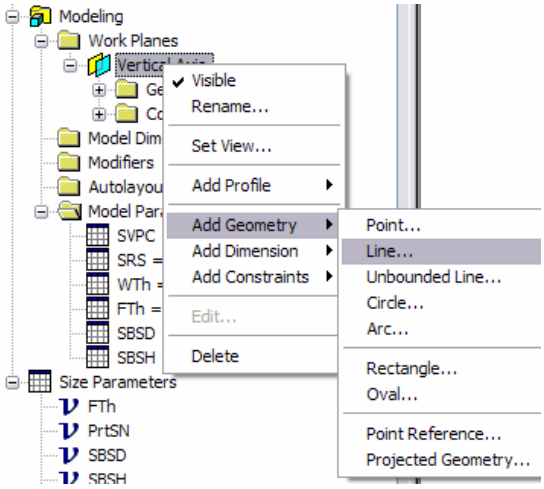
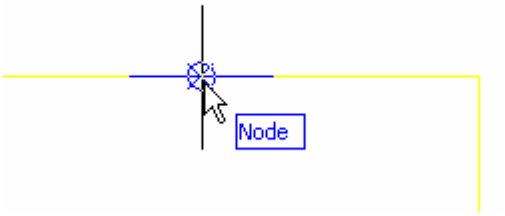
Step	Action	Result
1.	Click the Pipes menu, then click Part Builder	The Getting Started – Catalog Screen dialog appears.
2.	Click the US Imperial Structures Folder, then click New Chapter. Enter Custom for name and click OK. 	A new Chapter is created for custom structures. 
3.	Verify that the Custom Folder is selected, then click New Parametric Part. 	The New Part dialog appears
4.	For Name enter "NO 233a". Click in the box next to Description. Add "Outside Drop Connection" to the default Description, Click OK.	The Part Builder environment launches.
5.	Expand Part Configuration. Change Undefined Part Type to Junction Structure. Change Undefined to Manhole. Change Undefined Bounded Shape to Cylinder.	The Part is configured as a cylinder shape with the properties of a junction structure.

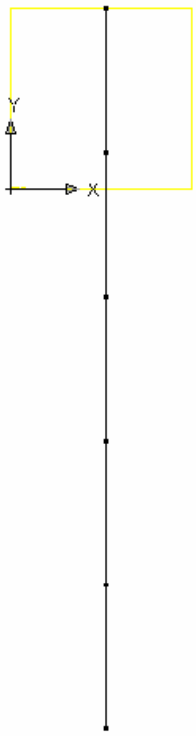
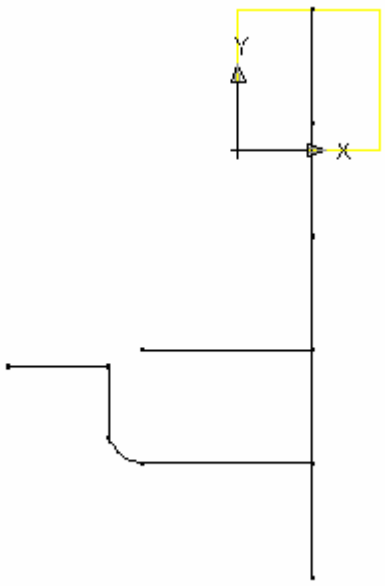
		
6.	Expand Modeling. Right-click Work Planes and click Add Work Plane. 	The Create Work Plane dialog appears.
7.	Click Top, enter “Rim” for Name, then click OK.	A top work plane is created which will represent the rim elevation of the structure.
8.	Expand Work Planes. Right-click Rim, then click Add Geometry, Point. 	You are prompted to pick a point.
9.	Click a point near the center of the yellow rectangle. Press ESC.	A point is created on the Rim work plane near the center. This will become a reference point to begin the construction of the part.
10.	Right-click Work Planes, then click Add Work Plane.	The Create Work Plane dialog opens.
11.	Click Right, enter “Vertical Axis” for Name then click OK.	The Vertical Axis work plane is created. 

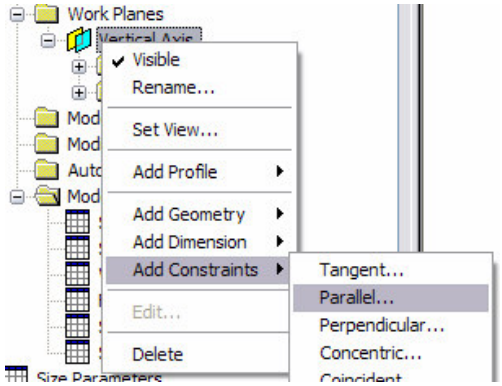
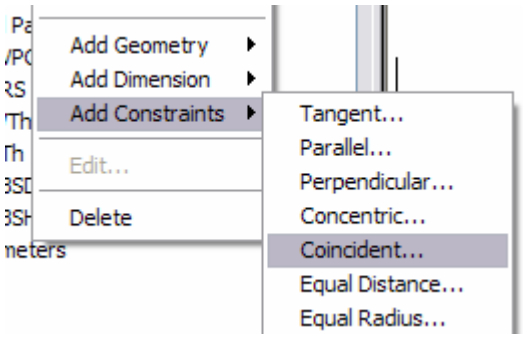
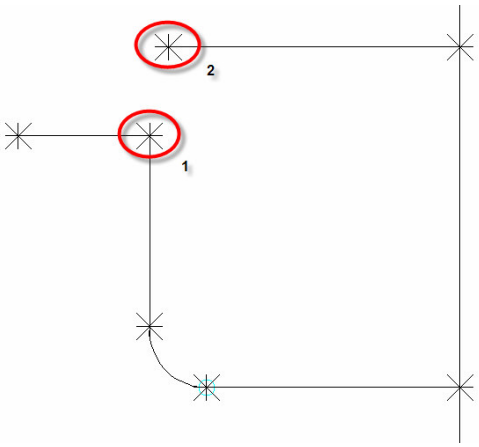
12.	Click Save Part Family. Click Yes. Stay in the Part Builder environment for the next exercise. 	The part is validated and saved. Depending on the part Type and on the Bounding Shape selected, certain Model Parameters and Size Parameters are automatically added to the part definition. 

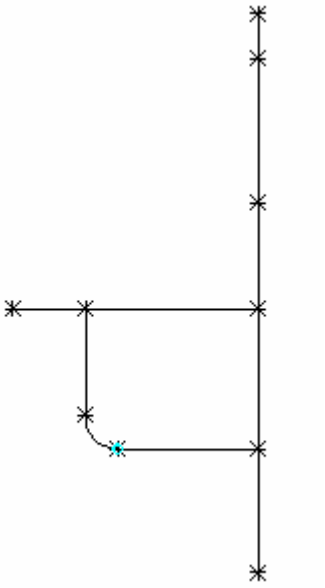
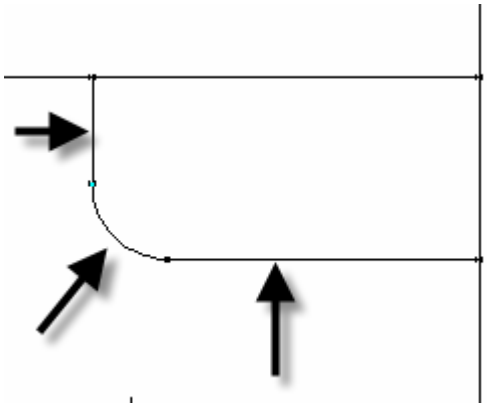
Part 2 – Defining the Manhole Geometry

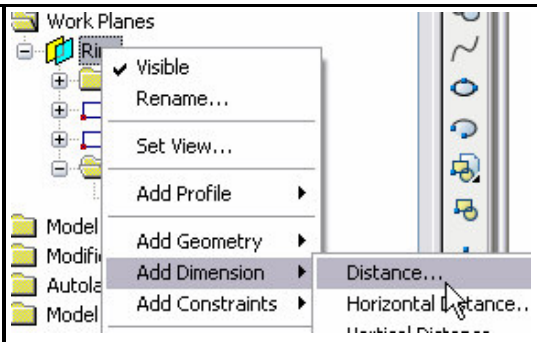
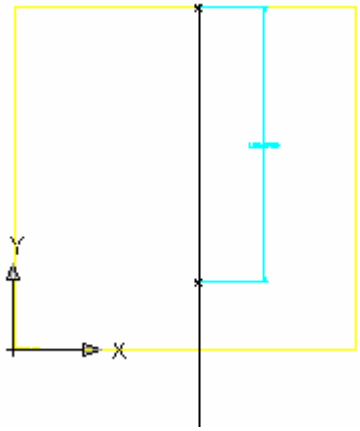
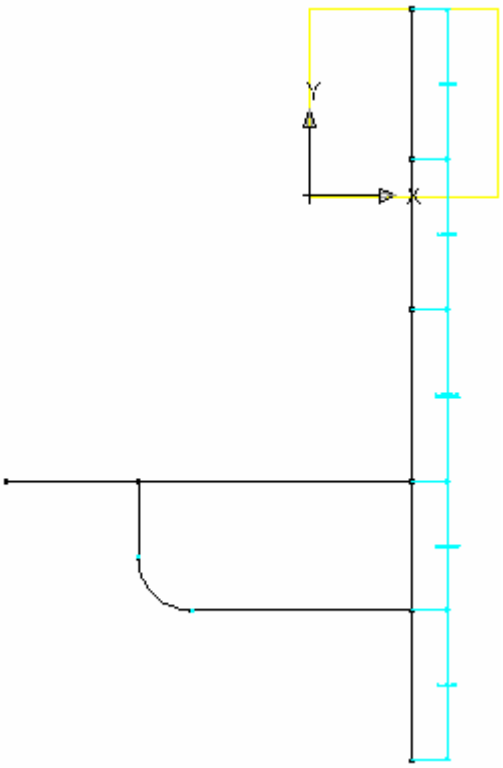
In this exercise you will define the geometry of the drop manhole by creating a simple schematic of the structure profile. You will build this portion with dimensions that can be modified from within Civil 3D when the part is in use.

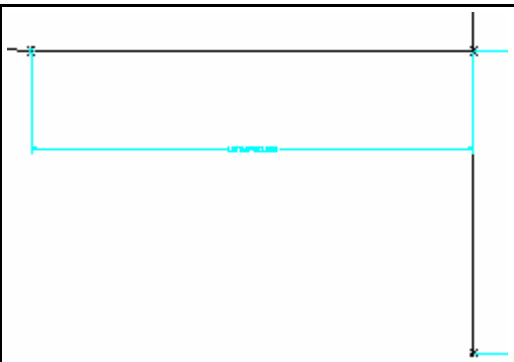
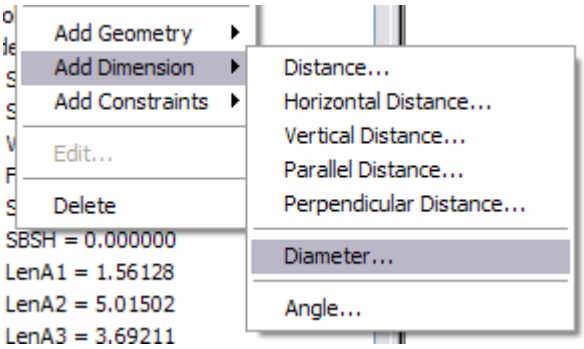
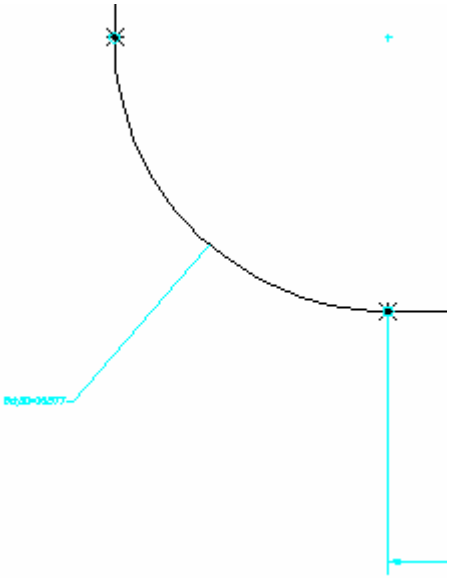
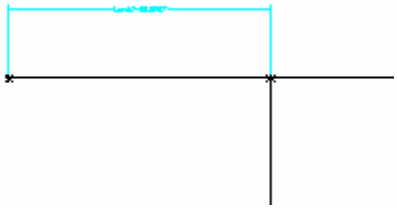
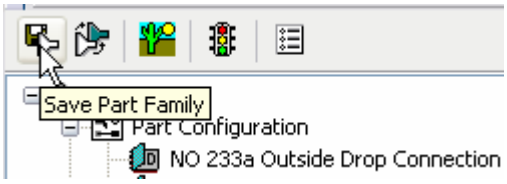
Step	Action	Result
1.	Under Modeling, Work Planes, right-click Vertical Axis and click Set View. 	The current view and UCS is set to match the work plane.
2.	Right-click Vertical Axis. Click Add Geometry, Line. Create a line with 5 segments on the work plane. Begin by snapping to the node of the reference point on the Rim work plane and Use Ortho to make it easier to draw a straight line. Make the segments about 24 units long.  	A line geometry object is shown in the drawing. This line represents the vertical axis of the manhole. Each segment represents a component of the structure. Starting from the top, the segments represent: the frame; the cone; and the last 3 segments represent the barrel. We'll use the extra vertices to place the incoming Dip Tee and the Drop 90° Elbow in the next steps.

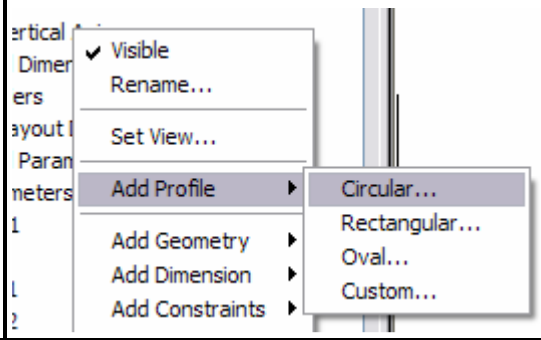
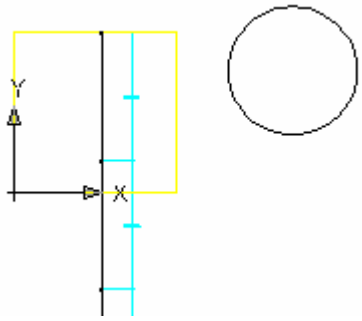
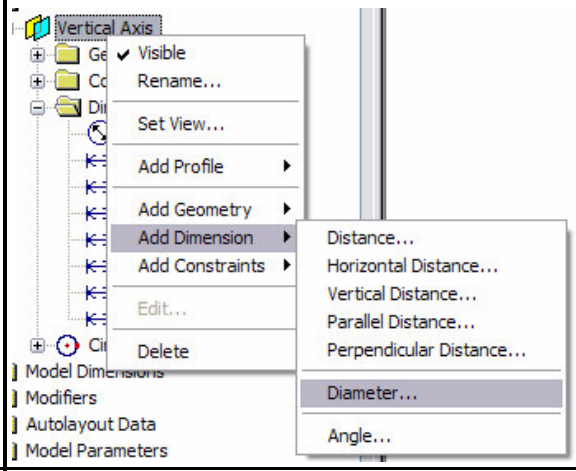
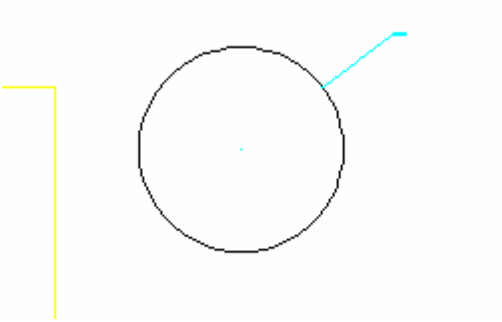
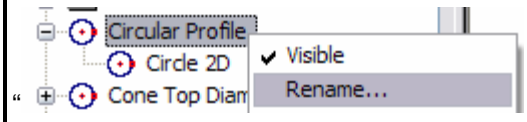
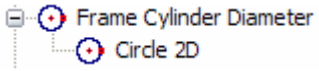
		
3.	Use the Add Geometry, Line and Add Geometry, Arc to draw the schematic of the drop assembly. Don't worry about making the parts perfectly meet. We'll use constraints to make the geometry match up properly. Make the two horizontal lines that connect to the vertical line about 36 units long.	
4.	Next you will establish some constraints to keep the components of the profile in the correct location relative to one another. Right-click Vertical Axis, Add Constraints, Parallel. Select the bottom line segment of the manhole centerline and then click the segment	The bottom two segments are now constrained such that they are parallel to each other.

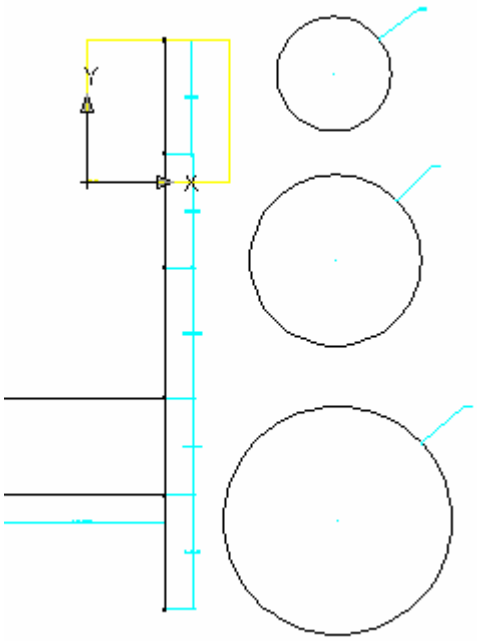
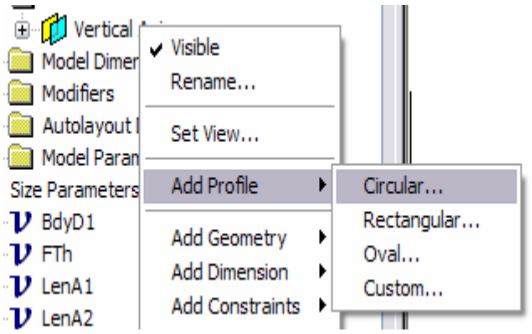
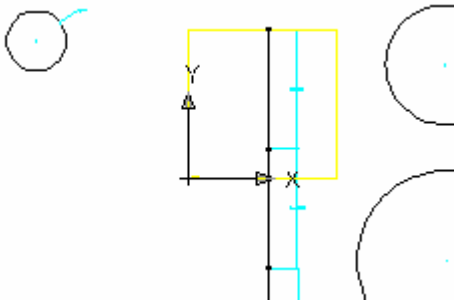
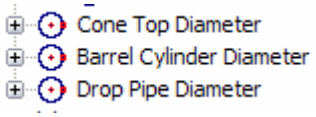
	directly above it.  	
5.	Repeat the process, working your way up the centerline, constraining adjacent line segments to Parallel.	All segments representing the centerline of the structure are constrained to be parallel to one another.
6.	Right-click Vertical Axis, Add Constraints, Perpendicular. Select the bottom segment of the structure centerline and the lower horizontal line. Repeat for the upper horizontal line	The lower and upper horizontal components of the drop pipe are constrained to perpendicular to the structure centerline.
7.	Right-click Vertical Axis, Add Constraints, Parallel. Select the bottom line segment of the manhole centerline and then click the vertical segment of the drop pipe.	The vertical drop pipe is constrained to parallel to the structure centerline.
8.	Right-click Vertical Axis. Click Add Constraints, Coincident. Click the point at the top of the vertical drop pipe and then the left end of the upper horizontal line.  	This will position the rectangle so that its center is located at the fixed point. 

9.		
	Right-click Vertical Axis, then click Add Constraints, Parallel. Select the right upper horizontal segment and then the left upper horizontal segment.	The two upper segments are constrained to parallel.
	Right-click Vertical Axis, Add Constraints, Tangent. Select the lower horizontal line and then the arc. Repeat for the arc and the vertical segment of the drop pipe.	The drop pipe bend arc is constrained to be tangent with the horizontal and vertical segments of the pipe. 
	Right-click Vertical Axis, then click Add Dimension, Distance. Click the bottom and then top points at the ends of the top segment of the centerline. Click a point to set the location of the dimension.	A dimension named LenA1 is created for the line segment representing the frame height.

		
	Repeat these steps for each segment of the centerline, starting at the top and ending at the bottom segment.	Dimensions named LenA2 through LenA5 are created for the centerline of the structure. **Note: for this exercise, make sure you dimension the segments in the order shown below.** 
	Add a Distance dimension to the lower horizontal line by picking the points at each end. This represents the distance from the structure centerline to the start of the elbow and will be used to ensure the drop stays on the outside of the structure.	LenA6 is created.

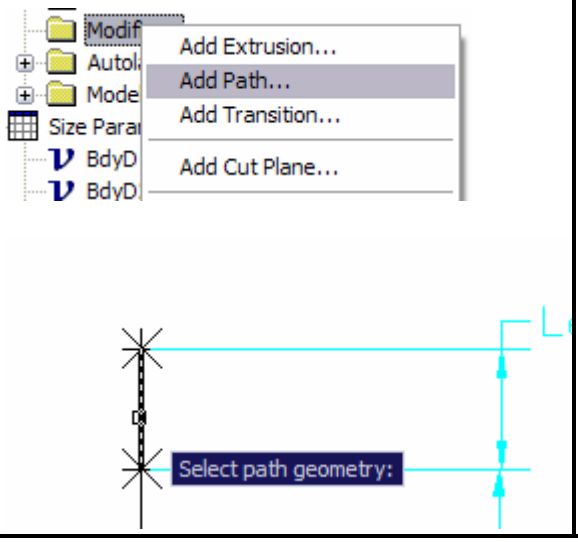
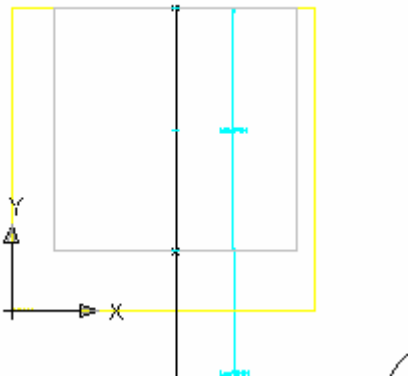
		
	Right-click Vertical Axis, then click Add Dimension, Diameter. Select the arc that represents the elbow. Click a point to set the location of the dimension. 	BdyD1 is added to the arc. 
	Add one final length dimension to the left upper horizontal line segment.	LenA7 is added. 
	Click Save Part Family. 	The part is saved.

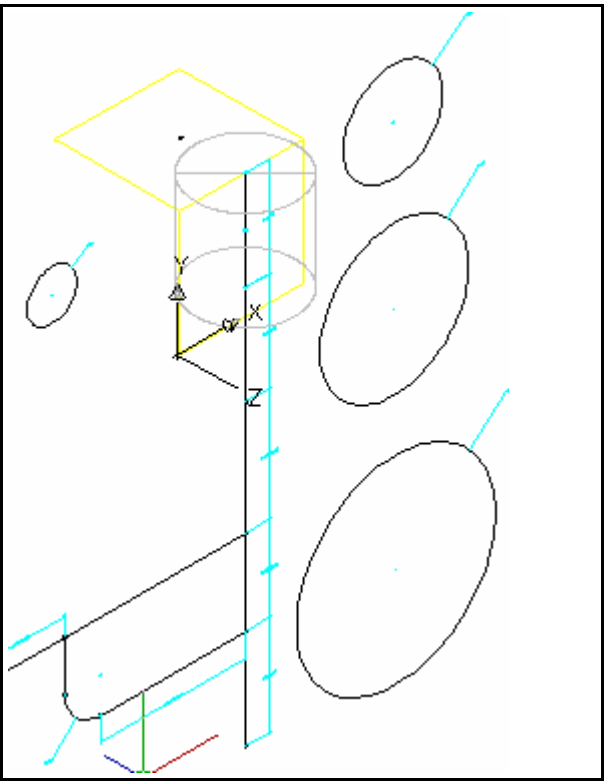
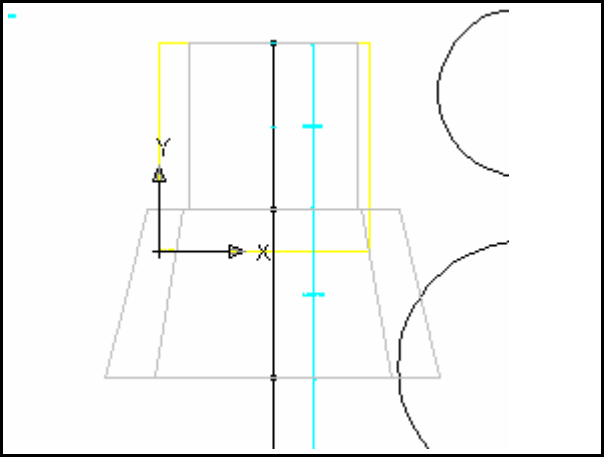
	Next you will add Profiles that represent the diameters of the frame, top of cone, barrel, and drop pipe. First, you'll create the profile for the frame diameter. Right-click on Vertical Axis, select Add Profile, Circular. Click an open area near the top right of the vertical axis to define the center and then click again about 12 units away to define the diameter. 	A circle profile is drawn. 
	Right-click Vertical Axis, click Add Dimension, Diameter. Click the circle drawn in the previous step. Click a point to set the location of the dimension. 	BdyD2 is created for the frame diameter. 
	Expand Vertical Axis. Right-click on Circular Profile, select Rename. Enter "Frame Cylinder Diameter" 	The profile is renamed Frame Cylinder Diameter. This will make it easier to work with later. 
	Next, repeat the previous two steps to create and dimension the top of cone profile with a radius of about 18 units and the barrel profile with a radius of about 24 units.	The top of cone profile is drawn and dimensioned with BdyD3. The barrel diameter profile is drawn and dimensioned with BdyD4.

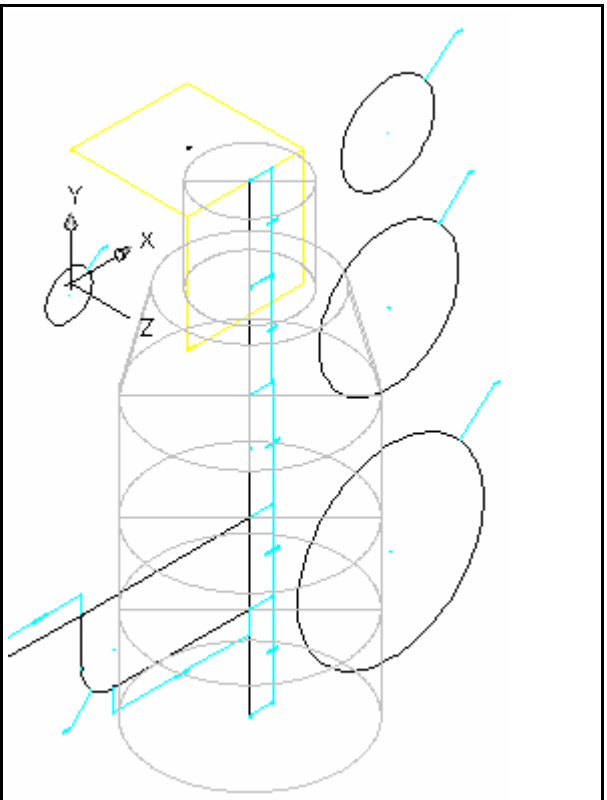
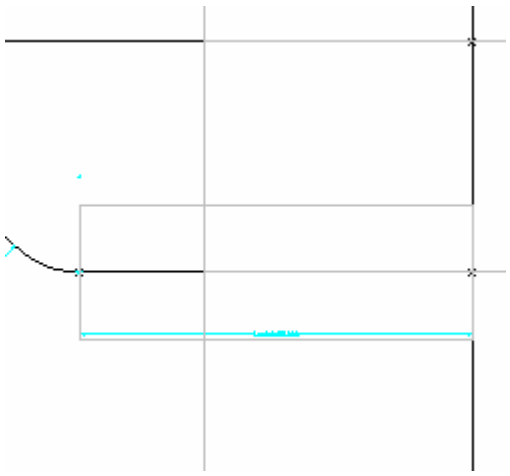
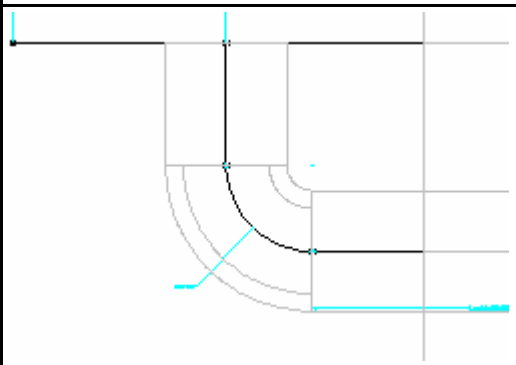
		
	Next, create and dimension the drop pipe profile. Right-click on Vertical Axis, select Add Profile, Circular. Click an open area to the left of the upper end of the vertical axis to define the center and then click again about 6 units away to define the diameter. Add a diameter dimension to the pipe profile. 	The pipe profile is created and dimensioned with BdyD5. 
	Rename the three Circular Profiles to Cone Top Diameter, Barrel Cylinder Diameter, and Drop Pipe Diameter.	Renaming the Profiles will make them easier to work with later. 
	Click Save Part Family. We will continue working with this part in the next section of this exercise.	

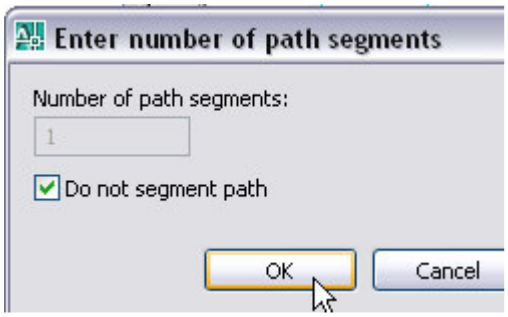
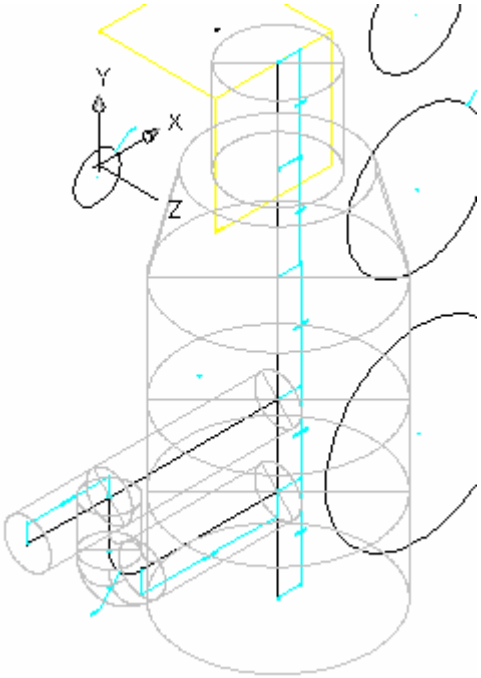
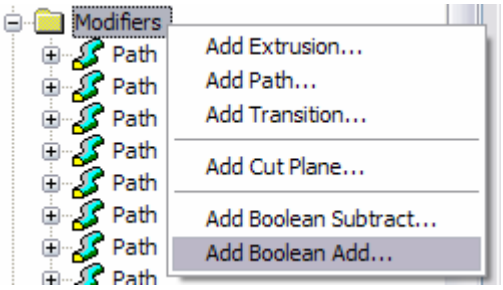
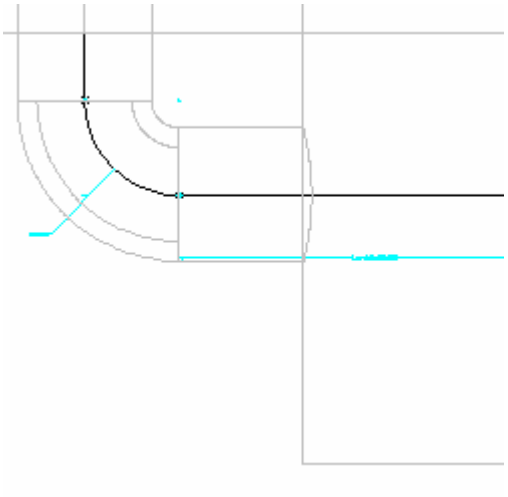
Part 3 – Create Profiles and Establish Parameters

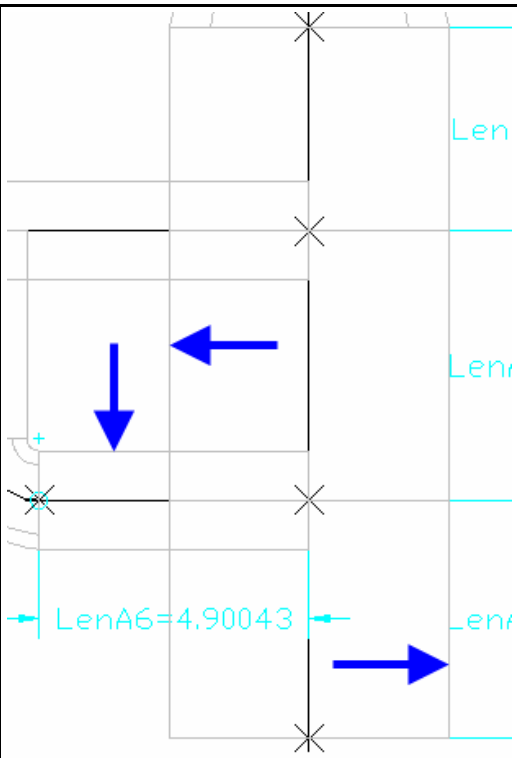
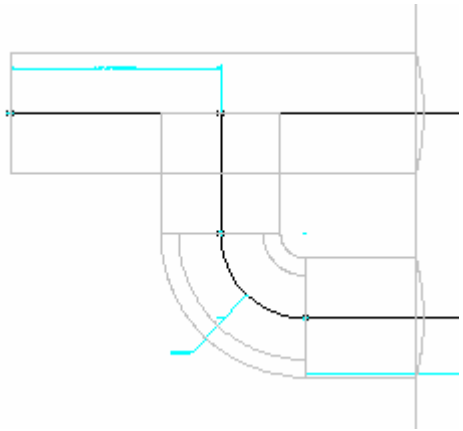
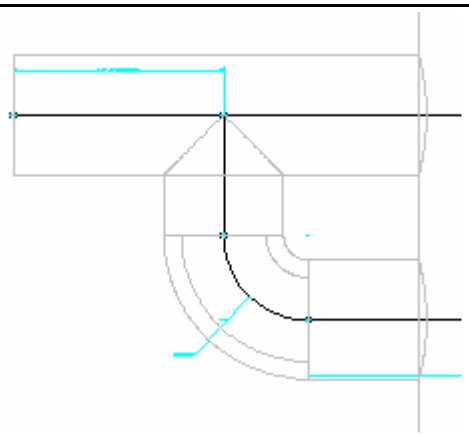
In this exercise you will extrude the part profiles to create the 3D model. You will then establish the model parameters to control the sizing and dimensions of the manhole.

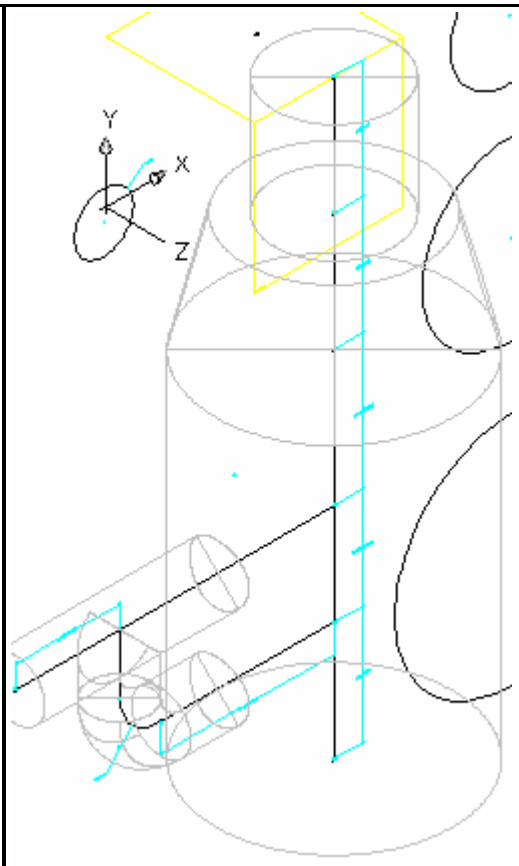
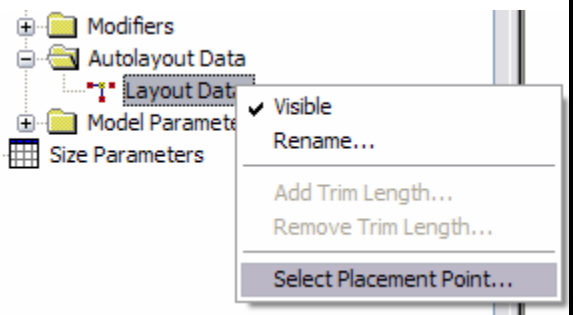
Step	Action	Result
1.	Right-click Modifiers. Click Add Path. When prompted for path, select the top line segment (the segment dimensioned LenA1) then select the Frame Cylinder Diameter profile (the circle dimensioned BdyD2) for the start profile and select it again for the end profile. 	The Frame Cylinder Diameter profile is applied along the path (length) of the top line segment. 

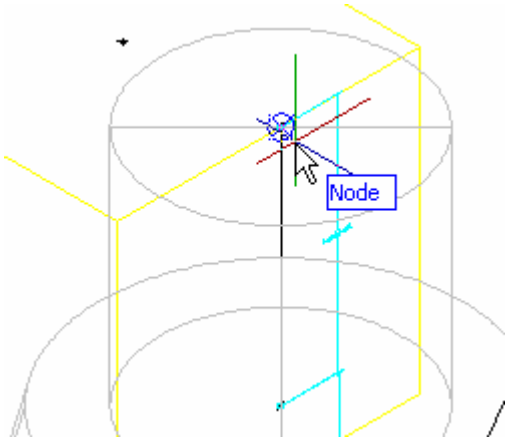
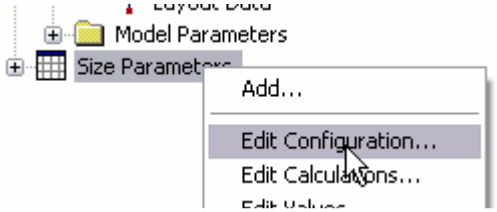
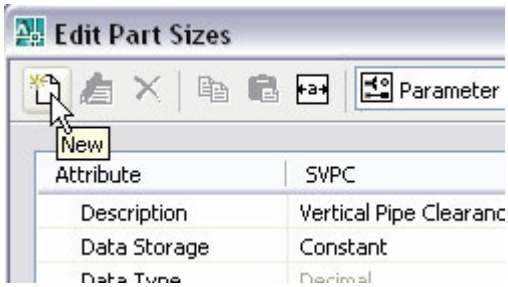
2.	Change the view to SE Isometric to get a better view of the Path. Change the view back to Right.	
3.	Repeat the Add Path command for the cone segment. For the start profile, select the Cone Top Diameter profile (dimensioned with BdyD3). For the end profile, select the Barrel Cylinder Diameter profile (dimensioned with BdyD4).	

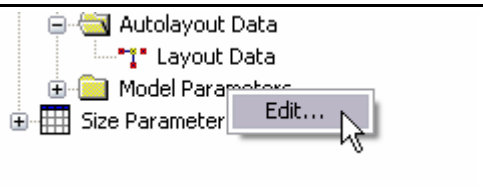
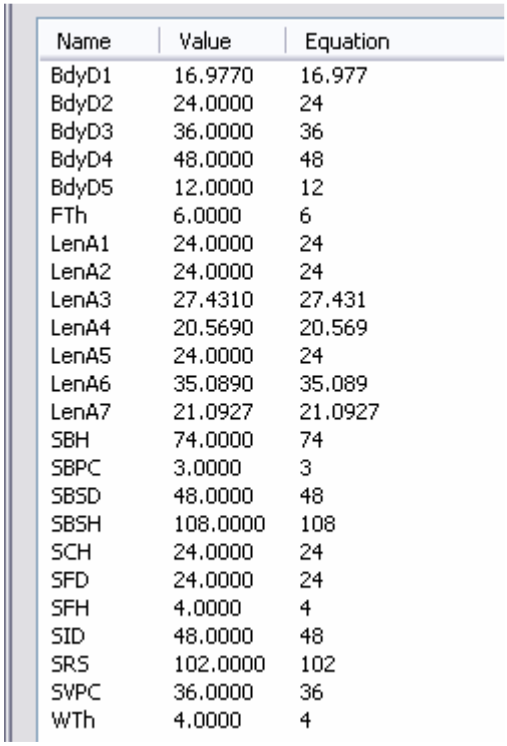
4.	Repeat the Add Path command for the remaining three line segments. Use the Barrel Cylinder Diameter profile for both the start and end profiles for each of the three segments. Change your view to SE Isometric to see the part in 3D. Change view back to Right	
5.	Next, you'll add paths for the drop pipe assembly. Right-click Modifiers. Click Add Path. Select the lower horizontal segment for the path and the Drop Pipe Diameter profile for the start and end profiles.	
6.	Add Path for the curved elbow. When the Enter number of path segments dialog opens, verify that "Do not segment path" is checked, and click OK. Then add the path modifier for the vertical segment of the drop pipe. Use the Drop Pipe Diameter profile.	

		
7.	Finish the drop pipe assembly by adding paths for the upper segments. Start with the right upper segment. Switch to SE isometric view.	In SE Isometric view, your part looks like this: 
8.	Switch back to Right view. Next, you will merge the structure components with the drop pipe assembly components. Right-click Modifiers and select Boolean Add. When prompted to select objects, select the bottom two barrel segments and the lower horizontal pipe segment and press Enter 	The parts are merged. 

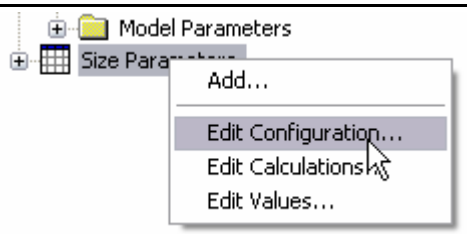
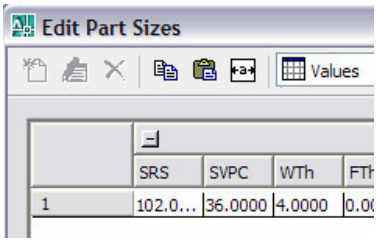
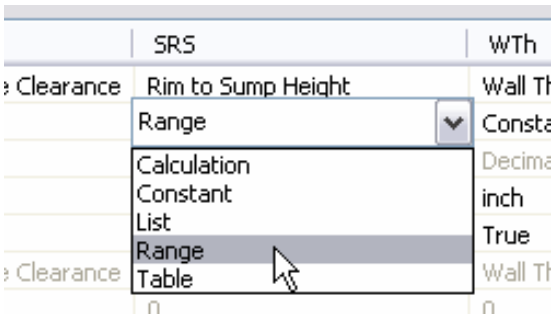
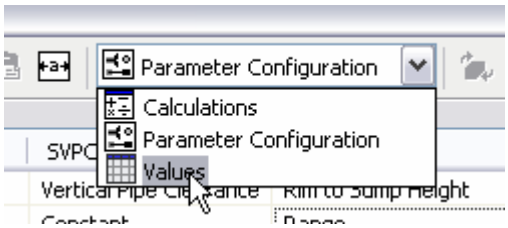
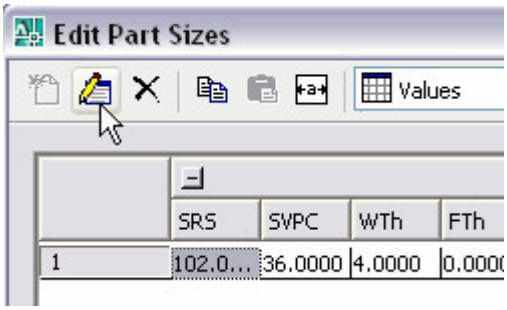
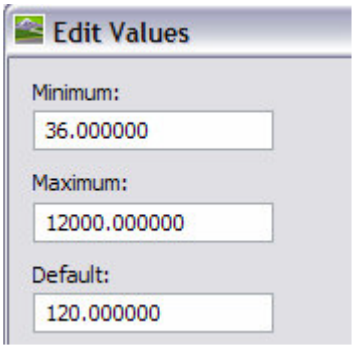
		
9.	Right-click Modifiers and select Boolean Add. When prompted to select objects, select the top two barrel segments and the upper two horizontal pipe segments and press Enter.	
10.	Right-click Modifiers and select Boolean Add. When prompted to select objects, select the upper horizontal pipe segment and the vertical pipe segment and press Enter.	

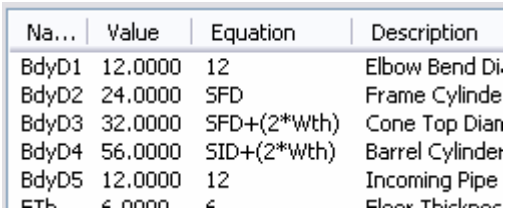
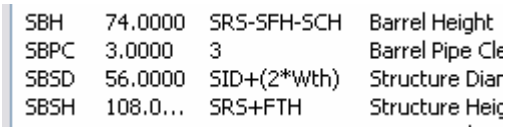
11.	Switch the view to SE Isometric.	
12.	Next you will set the placement point for the part. Expand AutoLayout Data. Right click Layout Data, then click Set Placement Point. Use the node object snap to click the top point on the vertical axis. 	This is the point at which the part will be inserted into the drawing.

		
13.	Click Save Part Family.	
14.	Right-click on Size Parameters and click Edit Configuration. 	
15.	Click the New button.  Add the following Parameters clicking the New button each time. • Barrel Height • Frame Height (SFH) • Frame Diameter (SFD) • Cone Height (SCH) • Inner Structure Diameter (SID) • Barrel Pipe Clearance (SBPC) Click OK when all of the parameters have been added.	
16.	Click Save Part Family. Right-click Model Parameters, then click Edit.	Saving the part will cause the Model Parameters to update, including the new Size Parameters.

	 Double-click in the Equation column next to each of the following and enter the values shown. • FTh 6 • SBH 74 • SBPC 3 • SBSD 48 • SBSH 108 • SCH 24 • SFD 24 • SFH 4 • SID 48 • SRS 102 • SVPC 36 • WTh 4 	
17.	Right-click on Size Parameters, click Edit Configuration...	The Edit Part Size Dialog opens

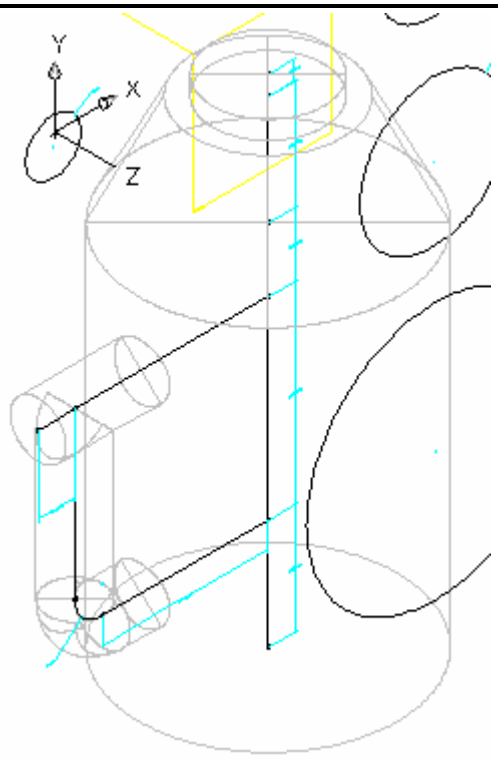
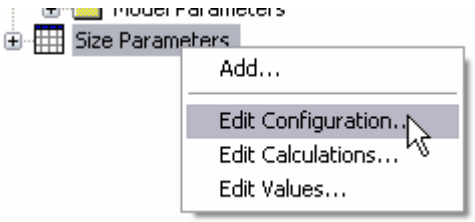
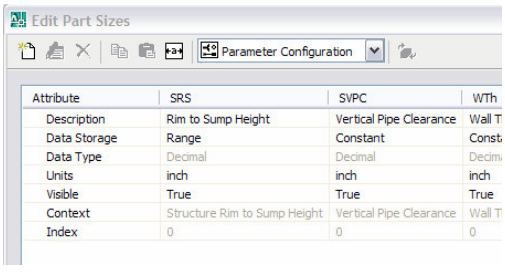
Name	Value	Equation
BdyD1	16.9770	16.977
BdyD2	24.0000	24
BdyD3	36.0000	36
BdyD4	48.0000	48
BdyD5	12.0000	12
FTh	6.0000	6
LenA1	24.0000	24
LenA2	24.0000	24
LenA3	27.4310	27.431
LenA4	20.5690	20.569
LenA5	24.0000	24
LenA6	35.0890	35.089
LenA7	21.0927	21.0927
SBH	74.0000	74
SBPC	3.0000	3
SBSD	48.0000	48
SBSH	108.0000	108
SCH	24.0000	24
SFD	24.0000	24
SFH	4.0000	4
SID	48.0000	48
SRS	102.0000	102
SVPC	36.0000	36
WTh	4.0000	4

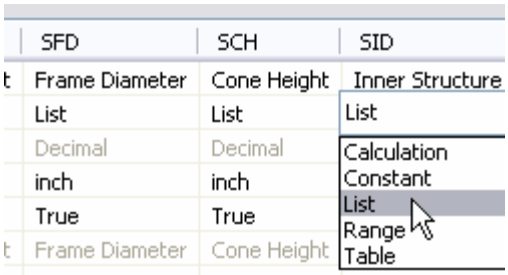
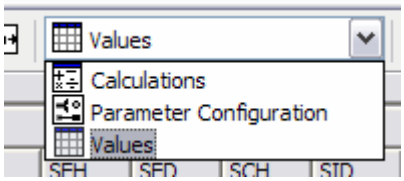
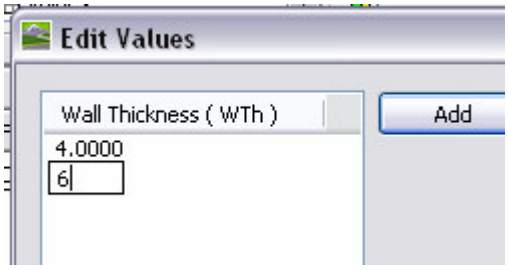
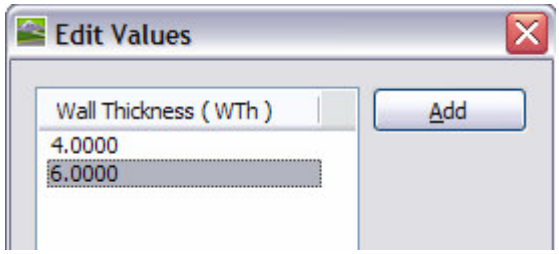
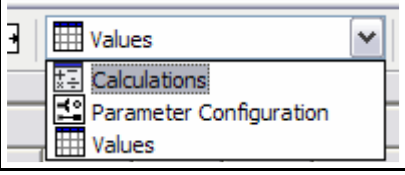
		
18.	In the SRS column, click on Constant and change it to Range.  Click Parameter Configuration and change it to Values. 	The SRS parameter will is now formatted as a range of values. The view is changed to show the value of each parameter.
	Click the cell in the SRS Column, then click the Edit button.  Set the Minimum to 36, the Maximum to 12000 and the Default to 120 and click OK.	
19.		
20.	Right-click Model Parameters, click Edit. Edit	

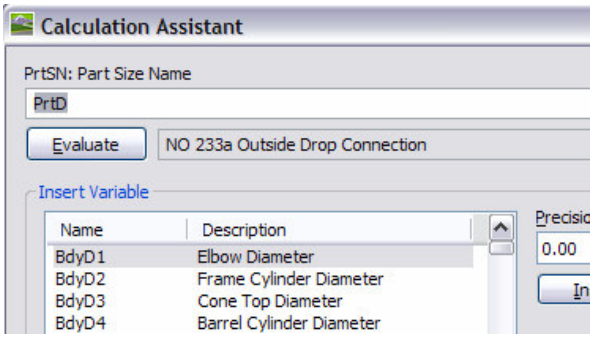
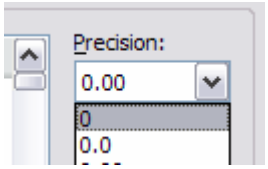
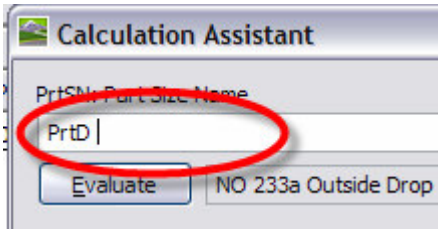
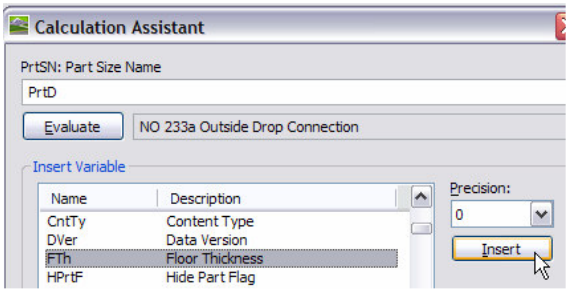
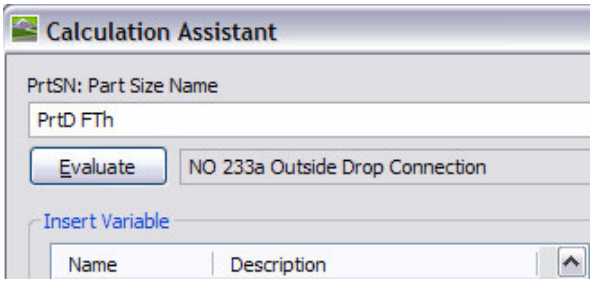
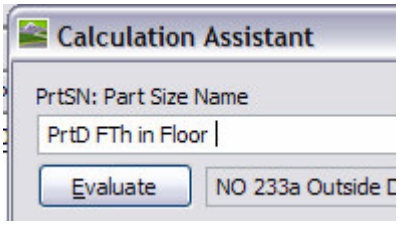
	the Equations and Descriptions for the BdyD# parameters as shown: • BdyD1 12 Elbow Bend Diameter • BdyD2 SFD Frame Cylinder Diameter • BdyD3 SFD+(2*Wth) Cone Top Diameter • BdyD4 SID+(2*Wth) Barrel Cylinder Diameter • BdyD5 12 Incoming Pipe Diameter 	
21.	Make the following additional edits • SBH SRS-SFH-SCH • SBPC 3.0 • SBSD SID+(2*Wth) • SBSH SRS+FTh 	
22.	Edit the Equations and Descriptions for the LenA# parameters as shown: • LenA1 SFH Frame Cylinder Height • LenA2 SCH Cone Cylinder Height • LenA3 SVPC-SFH-SCH+(BdyD5/2) Top Pipe CL • LenA4 SRS+FTh-SFH-SCH-LenA3-LenA5 Top Pipe CL to Bottom Pipe CL • LenA5 24 Bottom Pipe CL to Struct Bottom • LenA6 (SID/2)+WTh+(BdyD5/2)+2 Struct CL to Start of Elbow • LenA7 (BdyD5/2)+2 CL Vert Pipe to Incoming Pipe	

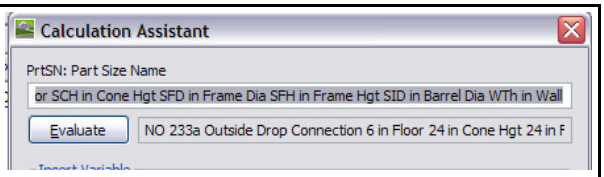
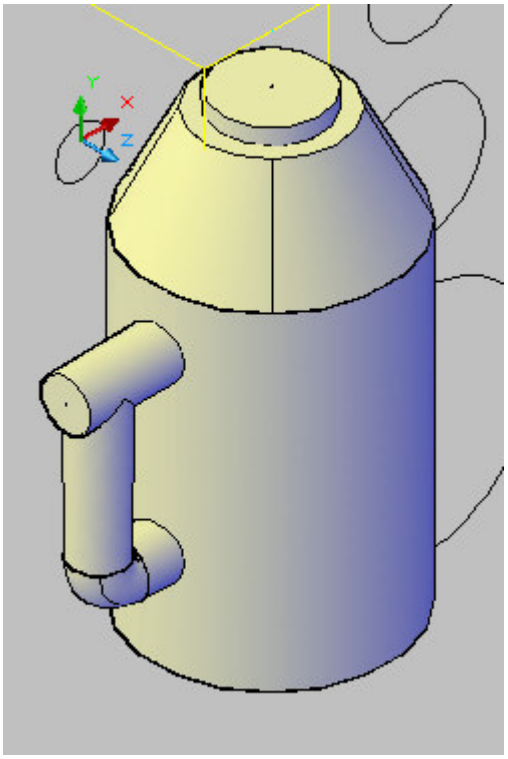
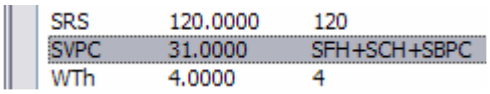
Na...	Value	Equation	Description
BdyD1	12.0000	12	Elbow Bend Di
BdyD2	24.0000	SFD	Frame Cylinde
BdyD3	32.0000	SFD+(2*Wth)	Cone Top Dian
BdyD4	56.0000	SID+(2*Wth)	Barrel Cylinder
BdyD5	12.0000	12	Incoming Pipe

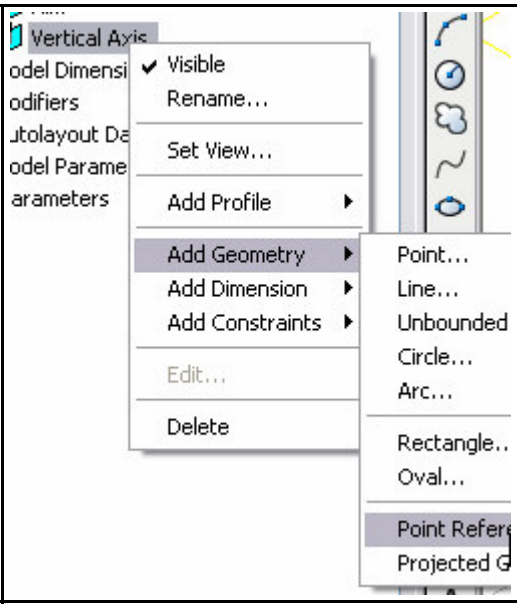
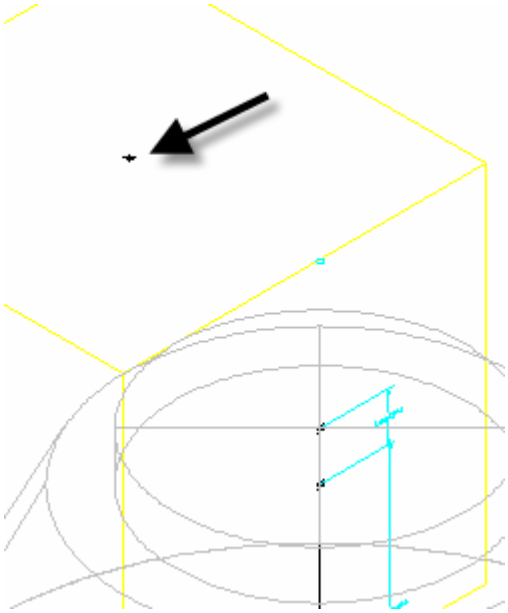
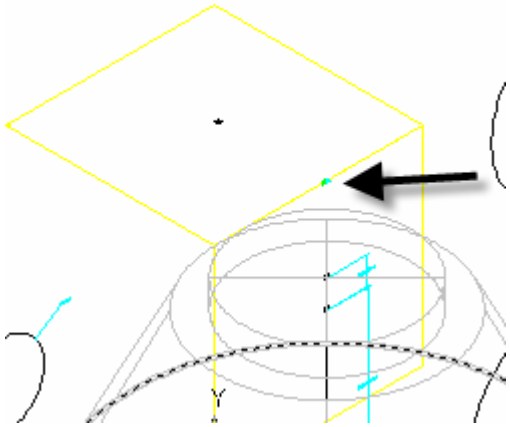
SBH	74.0000	SRS-SFH-SCH	Barrel Height
SBPC	3.0000	3	Barrel Pipe Cle
SBSD	56.0000	SID+(2*Wth)	Structure Diar
SBSH	108.0...	SRS+FTH	Structure Heig

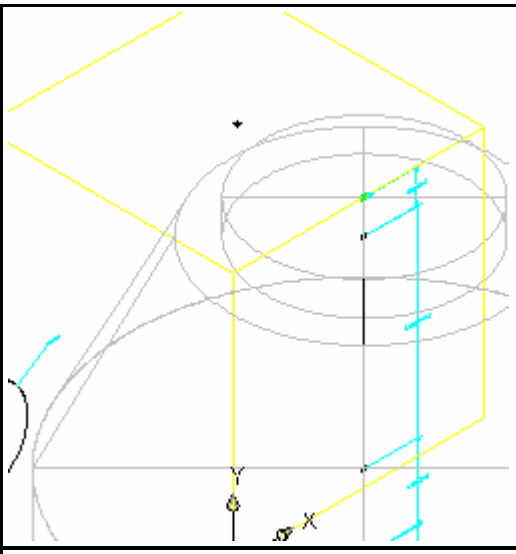
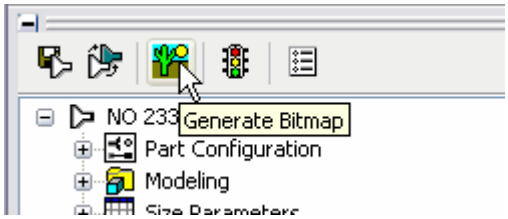
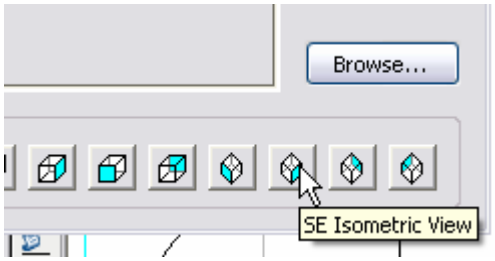
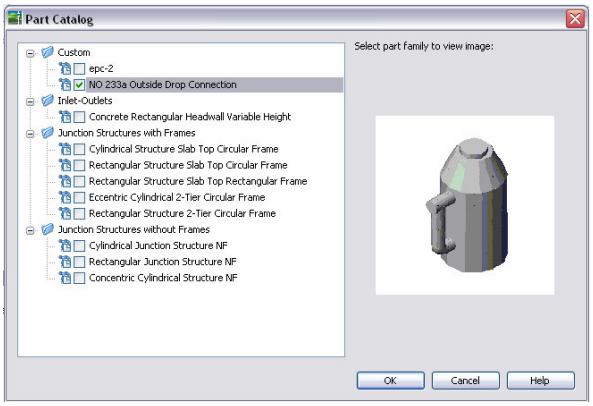
	LenA1 4.0000 SFH Frame Cylinder LenA2 24.0000 SCH Cone Cylinder LenA3 14.0000 SVPC-SFH-... Top Pipe CL LenA4 42.0000 SRS+FTh-S... Top Pipe CL to LenA5 24.0000 24 Bot Pipe CL to LenA6 36.0000 (SID/2)+W... Struct CL to St LenA7 8.0000 (BdyD5/2)+2 CL Vertical Pipe	
	Click Close. Change to SE Isometric view and note the changes to the geometry of the structure.	
	Click Save Part Family. Close out of the Part Builder Environment and then re-open the Part.	After all the changes that have been made, now is a good time to close out and re-open the part so that all of the data is freshly loaded into the part builder environment.
26.	Right-click Size Parameters, Click Edit Configuration. 	
27.	Change the Data Storage type to List for the following parameters: • WTh • FTh • SFH • SFD	

	• SCH • SID 	
28.	Click on Parameter Configuration and select Values from the drop-down menu. 	
29.	Select the 4.000 in the WTh column, click Edit and add the following values: 4.0, 6.0. Click OK. 	
30.	Repeat the previous step for each of the following parameters: • FTh: 6.0, 8.0, 12.0 • SFH: 4.0, 6.0, 8.0 • SFD: 24.0, 36.0 • SCH: 24.0, 36.0 • SID: 48.0, 60.0, 72.0	
31.	Click on Value and select Calculations from the drop-down menu. 	

32.	Double click on the cell in the PrtSN to open the Calculation Assistant	
33.	Change the Precision to 0. 	
34.	Click in the text box, right after PrtD and press the space bar.	
35.	From the Insert Variable list, select FTh and click Insert 	The FTh variable is inserted into the Part size name. 
36.	Type “in Floor ” after the FTh variable. (Exclude the quotation marks.)	
37.	Repeat the previous steps, adding variables and text for SCH, SFD, SFH, SID, WTh. The complete string should look something like this: PrtD FTh in Floor SCH in Cone Hgt SFD in	This is a required step to ensure that each part has a unique part name when added to the part list.

	Frame Dia SFH in Frame Hgt SID in Barrel Dia WTh in Wall. Click Evaluate to see the resultant part name (note that the name is long and partially cut off on the right. You can click on the name and use your keyboard arrow keys to see the rest of the name)	
38.	Click OK twice to close all dialog boxes.	
39.	Save the Part. Switch Visual Style to Conceptual. Part should look like the image at right.	
40.	Right-click Model Parameters, click Edit	
41.	Double click the Equation for SVPC and enter: SFH+SCH+SBPC.	
	Change the Visual Style to 2D Wireframe. Right-click Vertical Axis, then click Add Geometry, Point Reference.	

		
	Click the point in the center of the Rim work plane. 	A reference point (green) is created where the two planes meet in line with the vertical axis of the structure. 
	Right-click Vertical Axis, Add Constraints, Coincident. Click the top point of the vertical axis, then click the reference point created in the previous step.	The entire structure moves upward so that the rim elevation matches the top work plane.

		
	Click Generate Bitmap.  Click SE Isometric View.  Click OK.	A bitmap image is generated for the part that will be visible in the part catalog. 
42.	Click Save Part Family.	