AUTODESK UNIVERSITY 2014

Parametric Part Creation in AutoCAD MEP

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MP5037-L MvParts can be created in numerous ways, there are block based MvParts, then there are the wizards and finally there are parametric based MvParts. Getting started with the Parametric Parts can be intimidating for a new user of AutoCAD MEP so this class will step you through from a beginner level until you know everything required to be an expert in content creation. By the end of the class you will have the knowledge to create your own custom Air Terminals and Tee Fittings which will mean that you can make anything you need in AutoCAD MEP.

Learning Objectives

At the end of this class, you will be able to:

- Understand the Content Builder Interface
- Create a parametric Air Terminal
- Create a parametric Pipe Fitting
- Create a parametric Duct Fitting

About the Speaker

Add edited Speaker BIO as per AU website [Arial 10]
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Workflow of Part Creation

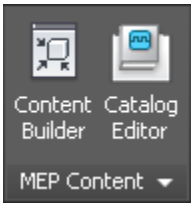
I wanted to start with an overview of the workflow for Parametric Part Creation, it's important to understand all of the steps below, both in their purpose and the method by which you can execute them within AutoCAD MEP.

1. Add Part to Catalog via Content Builder Tool
2. Define Part Configuration
3. Create Work Planes
4. Create Geometry and Profiles
5. Create Modifiers
6. Add Dimensions
7. Add Constraints
8. Add Connectors
9. Add Custom Parameters
10. Define Parameter Configuration
11. Define Parameter Calculations
12. Add Parameter values
13. Add Preview Bitmap
14. Confirm Options
15. Validate Part
16. Save Part
17. Use Part

By utilizing the information below, the Exercises and the accoumpanying videos I hope you will get a full understanding of how to create any Parametric Parts you require.

Understand the Content Builder Interface

Opening the Content Builder



The Content Builder tool is available on the Manage Tab -> MEP Content Panel. It is also available by typing AECBCONTENTBUILDER or CONTENTBUILDER.

Figure 1

Getting Started

The first thing to consider with the creation of your new content is under which Domain to place it. There are Duct, Cable Tray, Conduit and Pipe available for Fitting objects like Elbows and Tees then there is the Multi-view Part domain for Equipment and in-line objects like Valves and Battery Heaters.

If you are creating parts that will form the building blocks of other parts then you can set their domain as Primitive and Re-useable Parts.

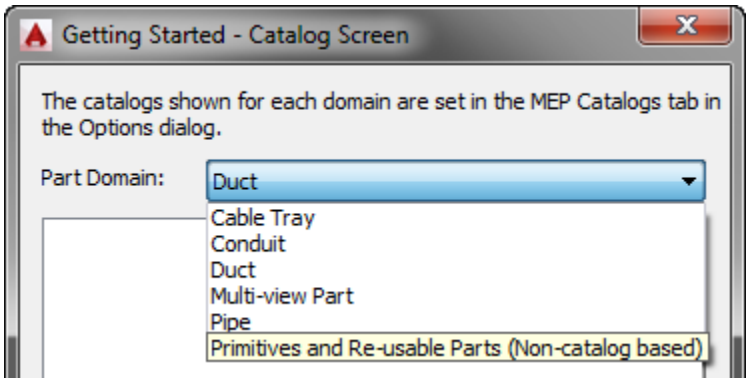


Figure 2

Creating New Parts

After defining the Domain you will see the folder structure of the current catalog for that selected domain (Figure 3) as defined on the MEP Catalogs Tab of the Options Dialog Box (Figure 5).

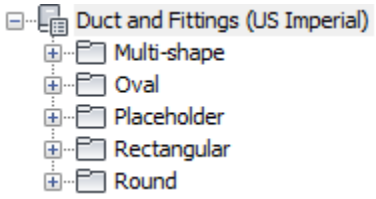


Figure 3

By selecting any of the Chapters within the folder structure you will have access to the tools for creation of new parts and chapters as shown in Figure 4

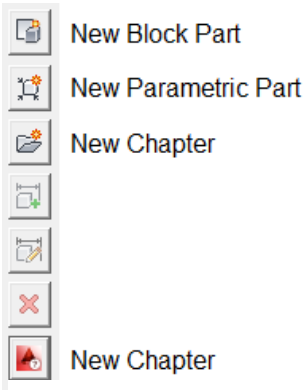


Figure 4

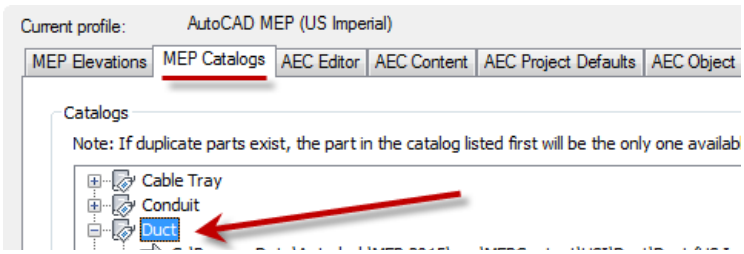


Figure 5

There are 2 tools available for Part Creation, Block Based and Parametric Based. You will find that the Block Based Part creation is easier to use for a beginner to MEP Content, however it is not as powerful as Parametric Part Creation.

For example if you had a piece of equipment with 5 variables regarding Length, Width, Height, Motor Size and Max Flow, and each of those variuables had 6 possible values then you have a total of 7,776 possible combinations (6^5). With Block Based Parts you would be kept busy making 7,776 DWG Blocks and assigning them to each of the combinations, however with a Parametric part you create 1 size, assign parameters to control the form and then put the variables into the parameters, creating 7,776 combinations in less time than perhaps 5 DWG Blocks would have taken.

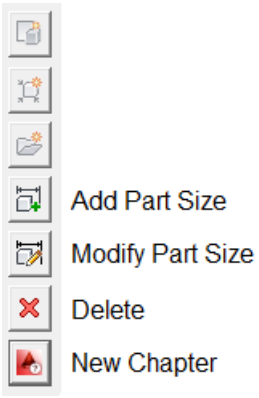


Figure 6

If you select one of the Parts in the Getting Started Dialog box you will see some different tool available as per Figure 6. These tools allow you to add additional Part Sizes (Block Based Parts) or Modify Part Sizes (Parametric and Block Based Parts). You can also delete Parts: note that if you delete the last part in a chapter the chapter will be removed from the Catalog.

Once you select the New Parametric Part tool, you will get the New Part Dialog Box (Figure 7), where you set the Name and Description of the New Part. The Name is the value you will see in the Content Builder while the Description is the value you will see upon placement of the Content in the Add Multi-view Parts Dialog Box, they can be the same.

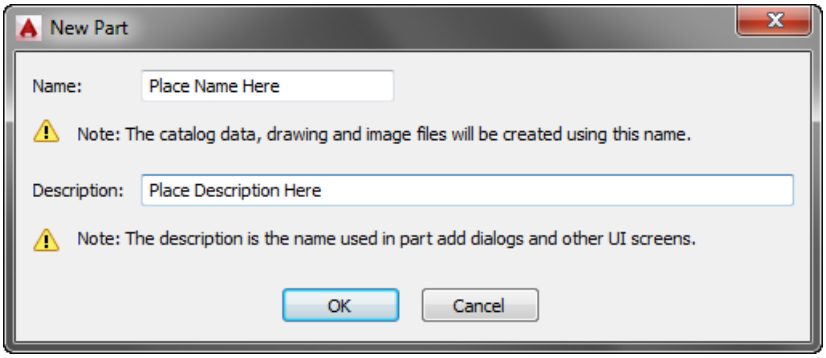


Figure 7

Content Builder Palette

Once you have selected to create a Parametric Part the interface will change and you will see the Content Builder Palette Anchored to the Left of the Screen by default, this is a standard AutoCAD Palatte and can be repositioned on the screen

There are a line of tools available along the top of the Palette, they are:

1. Save Part Family
2. Save Part Family As
3. Save Part Family As Primitive
4. Add Primitive
5. Generate Bitmap
6. Validate
7. Options

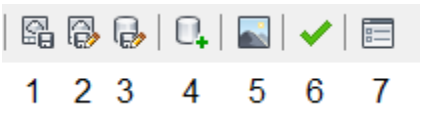


Figure 8

Save Part Family will save the Part to the Catalog location defined when the Part was created, a Part consists of a DWG, BMP and XML file. The DWG contains the geometry of the Part, the BMP is for the Preview image of the Content and the XML file stores the Parameters and their values. All 3 are required and so if you are distributing Content later, remember to package up all of the required files.

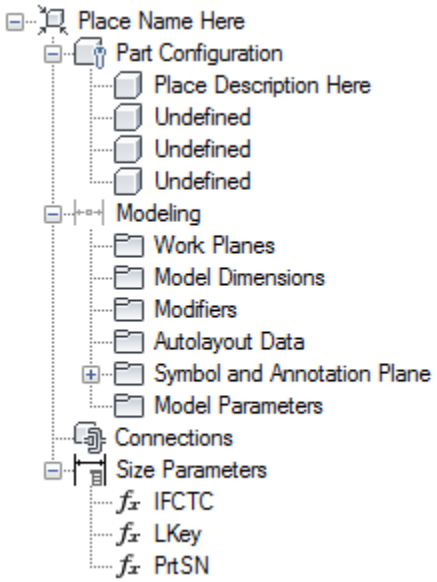


Figure 9

Primitives are pre-built forms that have pre-attached parameters, so if you are looking to place a Box with a Cylinder on top of it to define an Air Terminal then you may find the Primitives a good way to work to save time, however without a thorough understanding of the way form, Dimensions and Constraints work you may find that your Content does not perform as expected if you take this shortcut from the start.

You will find that a lot of the tools in the Content Builder are accessed via Right Clicking on the TreeNodes, for example in order to create a Work Plane to host your geometry you will need to Right Click on the Work Planes TreeNode.

Main TreeNode Headings

There are 3 main TreeNodes when you create a new Part

Part Configuration

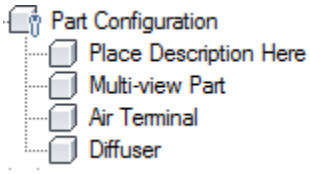


Figure 10

In here you will specify the top level category settings for the part, for example you could define the Domain as Multi-view Part, the category as Air Terminal and the Sub-Category as Diffuser, this will effect the overall settings of the Part and the requirements of the part for validation, for example an elbow will expect 2 connectors while a tee will require 3, without the correct number of connectors the Part will not be validated and therefore will not be available to place into your model.

Modeling

In here you will create the model geometry and datum's that define the solid form of the part. For example, you will create Work Planes to host lines that are used to define the center of extruded cylinders in the creation of a pipe fitting.

Size Parameters

In this final section of the Content Builder Interface you will find Parameters that you add, along with some default parameters. When you create a Parametric part, one of the benefits being parametric is the ability to create many versions of the part just by inputting size values, so here

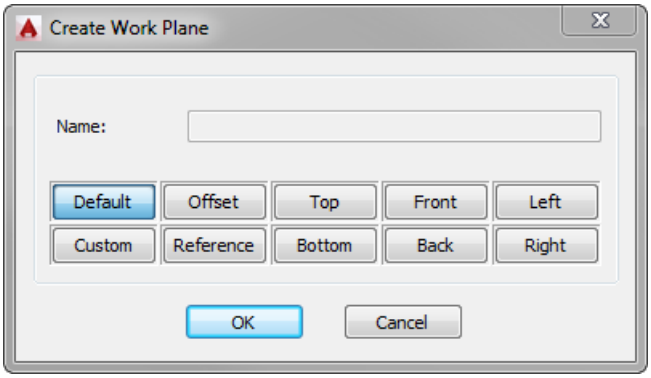
is where you will see the parameters you have created like Length, Width Connection Diameter for example.

There are some default parameters, in the example image above you will see one for IFC files, another for the Layer Key and finally one that stores a Part Serial Number.

Creating Content

Adding a Work Plane

Work Planes are infinite construction planes that can be placed at any orientation in the Content DWG file. They are used to host geometry, dimensions, constraints and profiles in order to define your Parts. The Work Plane is infinite, however in order to display it in AutoCAD MEP a rectangular object is shown that is 15 units squared in the Imperial AutoCAD MEP Profile, this cannot and does not need to be resized as it is just a visual representation of the inifinite plane. You can hide the Plane by Right Clicking it in the Content Builder Palette and selecting to turn off its visibility.



To Add a work plane Expand the Modeling TreeNode then Select and Right Click on the Work Planes TreeNode and select Add Work Plane.

You can define Work Planes for any of the 6 sides of a cube (Top, Bottom, Front, Back, Left and Right) along with Offset, Reference, Custom and Default.

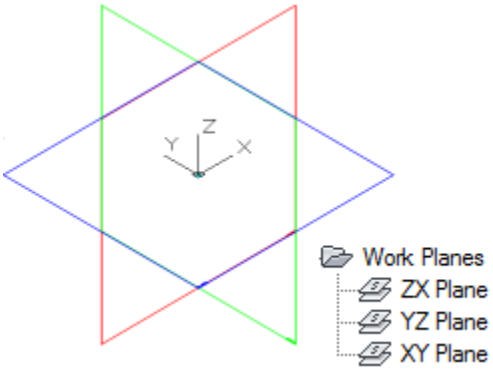


Figure 11

The Default Work Planes button will create 3 Work Planes, in the XY, YZ and ZX Plane orientation as shown in Figure 11, this can be a useful starting point.

The Custom button allows you to specify a Plane using vectors in the World Coordinate System, you will define the Origin and then then X and Y axis. This gives you

the most control over the created Plane, but is also the most complex of the creation methods.

The Offset button will create a Plan Parallel to an existing plane with an offset distance between them.

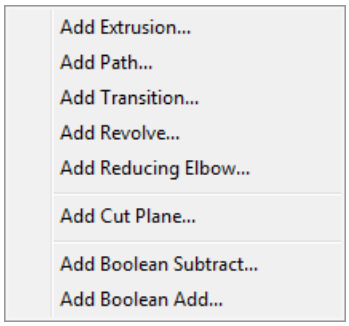
The Reference button enables you to create a Plane on the face of an existing modifier (solid form)

Adding Geometry

The main building blocks of your Parts will be Points, Lines, Arcs, Circles, and Profiles (which are closed shapes like Circles, Rectangles, Ovals and Custom Shapes which can be made from Geometry).

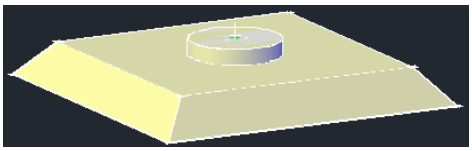
To Add Geometry Expand the Modeling TreeNode then the Work Planes TreeNode and then Select and Right Click on the Work Plane that you want to host the geometry on and select Add Geometry… or Add Profile…

Creating Solid Forms (modifiers)



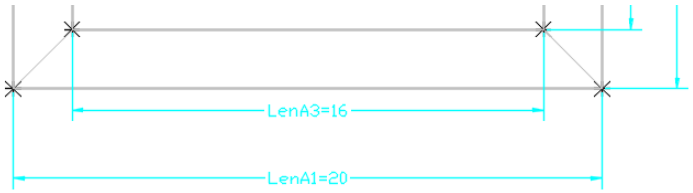
Solid Forms are created with modifiers in the AutoCAD MEP Content Builder. The modifers available are:

1. Extrusions,
2. Paths,
3. Transitions,
4. Revolves and
5. Reducing Elbows



You will need to provide geometry to the modifiers for them to create the Solid forms, for example an Extrusion requires a Profile, it will then request information to define how long the extrusion is, it can be a fixed distance, or it can be linked to another Work Plane for example.

Dimensions

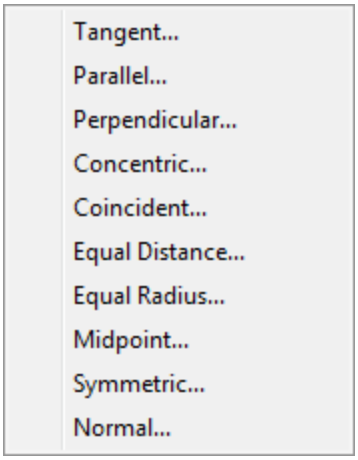


In order for the Part to be Parametric we will need to link parameters to the geometry, for example linking a Parameter LenA1 to the Width of the main body of an object. We do this via Dimensions. The dimensions connect to COLE 2d elements, for example at the COLE Points 2d at the end of Lines or the COLE line 2d's themselves.

Many Dimension types are available, for example the Horizontal and Vertical Distance Dimensions act line Linear Dimensions in regular AutoCAD while the Distance Dimension acts like the Aligned Dimension tool in regular AutoCAD.

To Add a Dimension Expand the Modeling TreeNode then the Work Planes TreeNode and then Select and Right Click on the appropriate Work Plane TreeNode select Add Dimension…

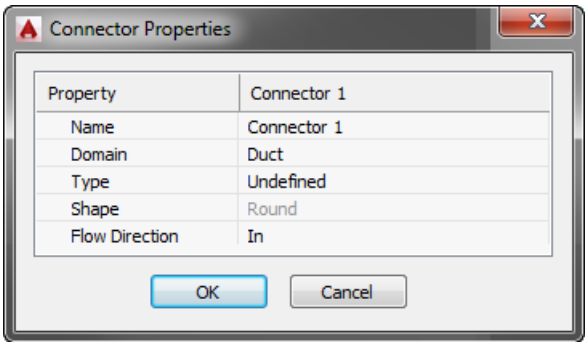
Constraints



In order to maintain relationships between geometry within the model you will utilize constraints, for example in order to keep a COLE Point 2d element in the center of a COLE CircArc 2d you can place a Concentric Constraint on the 2 elements, similarly if you required 2 COLE Line 2d elements to maintain a 90 Degree angle between them at all times then a Perpendicular Constraint can be employed. They act in a similar way to the Constraint tools that were introduced to AutoCAD in the 2010 release, however remember that these tools have been available since 2004 so there may be some differences.

To Add a Constraint, Expand the Modeling TreeNode then the Work Planes TreeNode and then Select and Right Click on the appropriate Work Plane TreeNode select Add Constraints…

Connectors



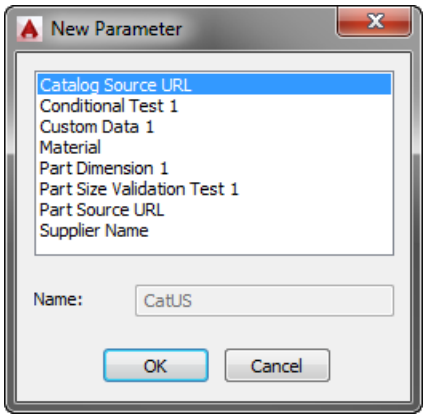
The thing that separates a Multi-view Block from a Multi-view Part is really the ability for the Part to interact with the Duct/Pipe/Electrical System that it is connected to. It is the presence of connectors that control this behaviour as they store the Domain and Flow Direction Settings along with the Shape and Size of the connector.

To Add a Connector Select and Right Click on the Connections TreeNode and select Add Connection…

Parameters

In order to connect the size information of a Part to the geometry you will need Parameters, these are either automatically generated in the case of Connector Dimensions and Work Plane Offsets, or manually created, for example if you required a Motor Size parameter.

To Add a Parameter Select and Right Click on the Size Parameters TreeNode and select Add… You will be presented with the New Parameter Dialog Box and in here you can select from different parameter types.



Finalizing

Prior to placing your new Part into your drawing you will need to Validate the Part, this takes a look at the part and checks to see if the correct number of connectors are in place, if the Preview image is available and if the configuration has been configured.

An example – Examining the Cylinder Primitive:

The following example you open up a simple primitive and examine the settings and content; this is a first step to becoming comfortable and familiar with the Content Browser.

1. Open the Content Builder from the Manage Tab –> MEP Content Panel
2. Select Primitives and Re-usable Parts (Non-catalog based) from the Part Domain Drop Down
3. Expand the Primitive Header and select Cylinder
4. Select Cylinder and then click the Modify Part Size tool  on the right hand side of the Getting Started – Catalog Screen Dialog box.
5. In the main drawwing window use the ViewCube to select the South West Isometric view, you should see something similar to Figure 12 – Cylinder Example.

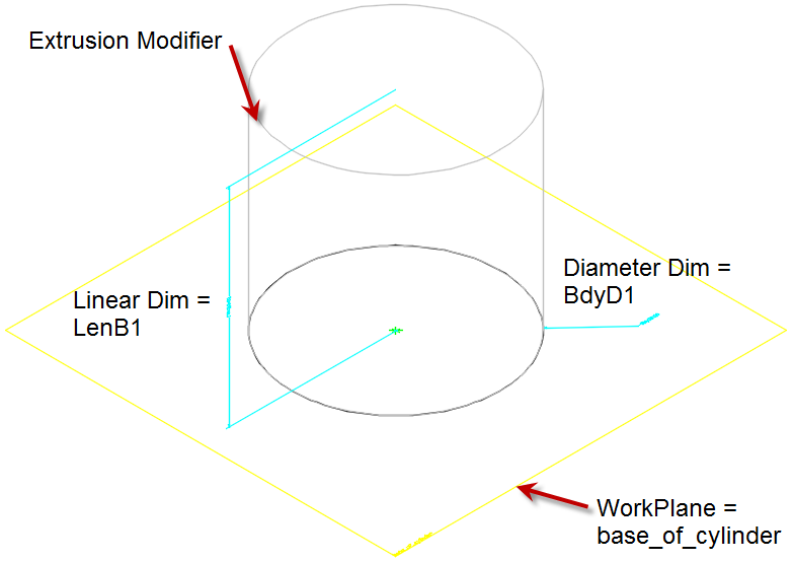


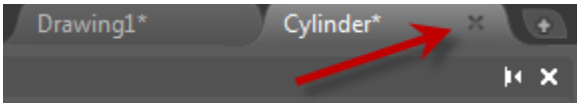
Figure 12 – Cylinder Example

6. Note that there is a Yellow Work Plane, pick the workplane and inspect its properties, note that there is no descriptive information held within the Properties Pane other than it being described as a COLE Work Plane.
7. Take a look at the Content Builder Palette on the Left (by default) of the screen, expand the Modeling TreeNode and you will see the following TreeNodes – Work Planes, Model Dimensions, Modifiers and Model Parameters.
8. Expand the Work Planes TreeNode and you will see the base_of_cylinder Work Plane, if you click to select the Work Plane in the Conten Builder you will see if highlights in the drawing window, try right clicking on the Work Plane and you will see many options available.

9. Expand the base_of_cylinder Work Plane TreeNode in the Conten Builder and you will see the following TreeNodes – bottom_cylinder_profile, Geometry, Construaints and Dimensions. Now we are starting to see how information is organized in the Content Builder Palette, by expanding the bottom_cylinder_profile TreeNode you will see that the profile is constructed of a Circle 2D element, there is also a Point 2D geometry element and Concentric Constraint and finally a BdyD1 Dimension.
10. Right click on the Model Parameters and select Edit
11. In the Equation column, double click the values and change them to other values then click Close, you will see that the cylinder has changed size on the screen.

Name	Value	Equation	Description
BdyD1	20.0000	20	Diameter
LenB1	10.0000	10	Height

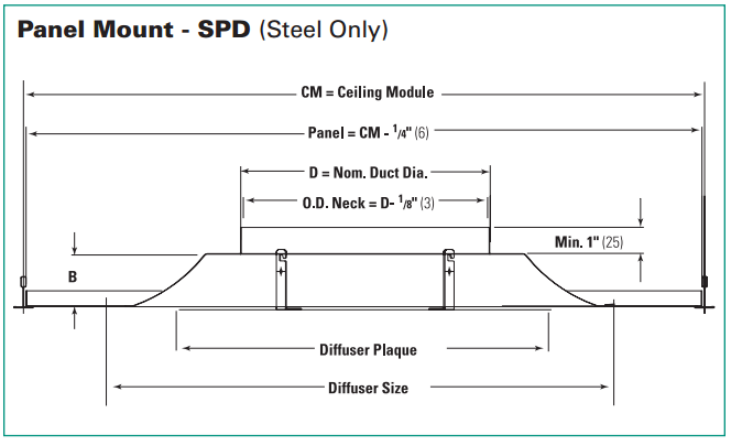
12. Close the Cylinder drawing Tab to exit the Cylinder content file.



An Example – Create a parametric Air Terminal

During this example we will create an Air Terminal MvPart using the Parametric Content Builder, the Air Terminal will be a simplified version of the following EH Price SPD Supply Diffuser:

The basic dimensions of the Object are shown in Figure 13 SPD Dimensions, we will use a bit of a simplification, but accurate enough for the needs of a MEP model. For example where the object tapers from the face to the neck we will use a linear taper (Dimension B) rather than an arc taper. This will allow the object to not be too much of a drag on the graphics system while still working as an accurate representation for design and coordination.



Face View

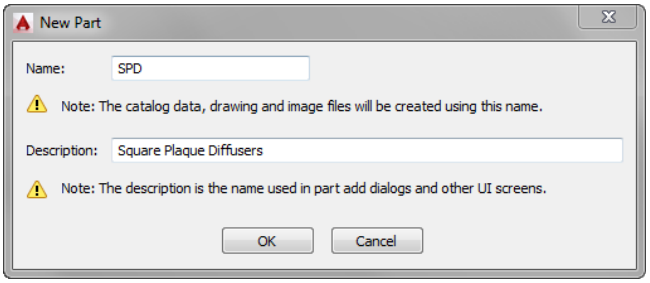
Modules	CM = Nom. Panel Size	
Diffuser Size 24x24 (600x600)	Imperial (inches)	Metric (mm)
	48 x 24	1200 x 600
Diffuser Size 12x12 (300x300)	Imperial (inches)	Metric (mm)
	24 x 12	600 x 300
	48 x 12	1200 x 300
	20 x 20 24 x 24	500 x 500 600 x 600

Dimensional Data — Imperial (in.) / Metric [mm]

Diffuser Size Imperial	Diffuser Size Metric	B	Duct Size
24 x 24	600 x 600	$2^{1}/_{2}$ [64]	6, 8, 10, 12, 14, 15 [152, 203, 254, 305, 356, 381]
12 x 12	300 x 300	$1^{1}/_{8}$ [29]	4, 5, 6, 7, 8 [102, 127, 152, 178, 203]

Figure 13 SPD Dimensions

1. Lets start by opening the Content Builder from the Manage Tab –> MEP Content Panel
2. Select Multi-view Part from the Part Domain Drop Down
3. Expand the Mechanical TreeNode then expand the Air Terminals TreeNode.
4. Select Diffusers then click on the Add Chapter  tool, set the Name as AU2014 then click OK
5. Click the Add Parametric Part  tool
   a. Set the Name as SPD
   b. Set the Description as Square Plaque Diffusers
6. Click OK
7. Expand the Part Configuration TreeNode in the Content Builder Palette, verify that the Description



and Domain are already set as Square Plaque Diffusers and Multi-view Part respectively

8. Double Click on the first Undefined TreeNode in the Content Builder Palette, you will then have a drop down list of available Multi-view Part Types, select Air Terminal
9. Double click the next TreeNode down, also named Undefined and again there will be a drop down, select Diffuser from the list

By setting the Type and Sub Type for the content the software will be able to complete some verification on the object, for example the software now knows to expect a duct connector in order to complete verification of the MVPart.

10. Next we need to create a Work Plane as this is required for the hosting of Profiles and Geometry, Expand the Modeling TreeNode
11. Right Click on Work Planes and select Add Work Plane…
12. Select *Bottom* and click OK, this will create a Wrok Plane that we can use for the face of the Diffuser
13. In the main drawing window set the view to a South West Isometric view
14. Right Click on Work Planes and select Add Work Plane…
15. Select Offset and set the Name to *Top of Main Body* then click OK

We are now creating a work plane based on an offset from an existing Work Plane, we will need to select a Work Plane and specify a distance between the existing Work Plane and the new Work Plane

16. Select the Bottom Work Plane then position the cursor so you have a vertical polar line as per Figure 14 Offset Work Plane then type 2.5 inches (2.5”) to create a new work plane 2.5” above and parallel to the Bottom Work Plane
17. Repeat the Process to create another Work Plane, this time 1 inch (1”) above the Top of Main Body Work Place, and call this new Work Plane *Connection*
18. Set the View Direction to Top
19. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Profile -> Rectanglar....
20. Select 2 points within the scope of the yellow Work Planes to create a Rectangle, the exact size is not important

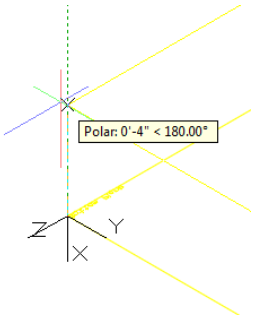


Figure 14 Offset Work Plane

You will have created 8 elements, 4 COLE Line 2d’s and 4 COLE Point 2d’s We now need to generate a center point of the rectangle which will assist us with lining up the other geometry

21. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Geometry -> point then select a point somewhere close to the center of area of the rectangle

22. Next we will constrain the Point to the center of area of the rectangle using Equal Distance Constraints
23. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Constraints -> Equal Distance then select the COLE Point 2d at the top left corner of the rectangle followed by the COLE Point 2d in the center, then the COLE Point 2d at the bottom right corner followed by the COLE Point 2d in the center
24. Repeat the process with the following pairs
    a. Top Right and Center
    b. Bottom Left and Center
25. We can now create center of areas for the 2 additional Work Plane using Point References, these take the position of a point on 1 Work Plane and create a similar point on the other Work Plane, if the original point moves then so will the Point Reference
26. Set the View Direction to South West Isometric
27. Select and then Right Click on the Top of Main Body Plane in the Content Builder and select Add Geometry -> Point Reference then select the middle point on the Bottom Work Plane, a new green Point Reference will be created on the Top of Main Body Work Plane
28. Repeat the Process for the Connection Work Plane the result should look like Figure 15

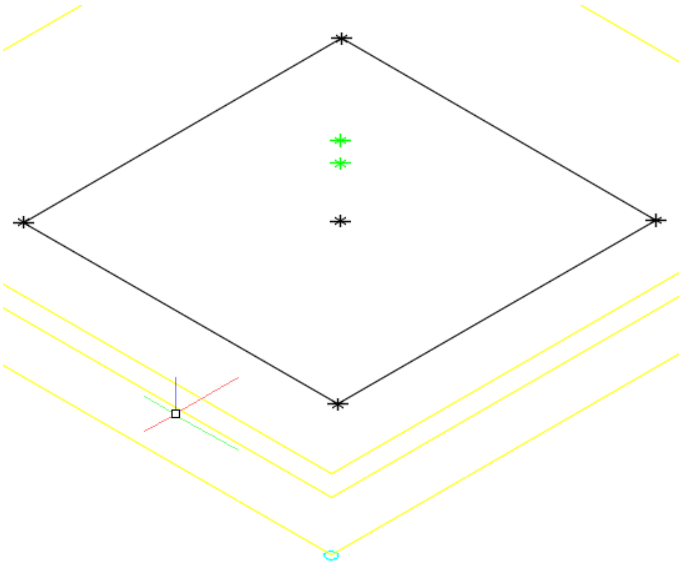


Figure 15

29. You can now start to generate profiles on the upper 2 Work Planes to help with the generation of the solid modifiers later
30. Set the View Direction to Top
31. Select and then Right Click on the Top of Main Body Work Plane in the Content Builder and select Add Profile -> Rectanglar....
32. Select 2 points within the rectangle on the Bottom Work Plane to create a Rectangle, the exact size is not important

33. Select and then Right Click on the Top of Main Body Work Plane in the Content Builder and select Add Constraints -> Equal Distance then select the COLE Point 2d at the top left corner of the rectangle followed by the COLE Point 2d in the center, then the COLE Point 2d at the bottom right corner followed by the COLE Point 2d in the center
34. Repeat the process with the following pairs
    a. Top Right and Center
    b. Bottom Left and Center
35. Select and then Right Click on the Connection Work Plane in the Content Builder and select Add Profile -> Circular....
36. Click a point for the center of the Profile, try to create it off of the center, we will use a constraint to position it correctly later
37. Click a point to define the radius of the Profile you should see something like Figure 16

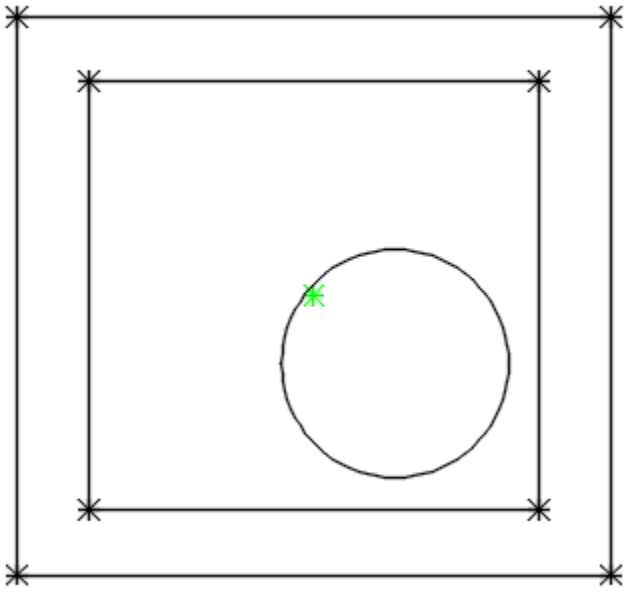


Figure 16

38. Set the View Direction to South West Isometric
39. Select and then Right Click on the Connection Work Plane in the Content Builder and select Add Constraints ->Concentric
40. Pick the Circular Profile followed by the Point on the Connection Work Plane

You now have everything in place for the framework of the Air Terminal, the only thing left is to create the solid form that will be visible in the model

41. Close the Work Planes TreeNode
42. Select and then Right click on the Modifiers TreeNode and select Add Transition…
43. Select the Bottom Rectangular Profile then select the Upper Rectangular Profile
44. Select and then Right click on the Modifiers TreeNode and select Add Extrusion… in the Extrusion Modifier Dialog box set the Type to Plane and the To to Top of Main Body as shown in Figure 17
45. Set the Visual Style to Conceptual and use the 3DORBIT command to inspect your Air Terminal

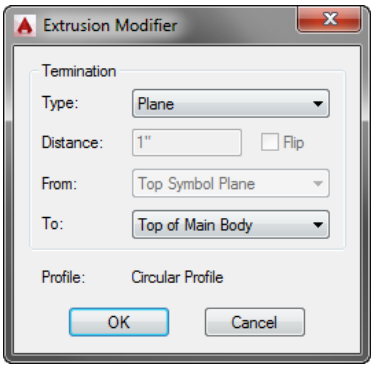


Figure 17

Next you need to add a connector to the MvPart so that the software knows how to attach Duct to the MvPart

46. Select and then Right click on the Connections TreeNode and select Add Connection… move the cursor over the top surface of the Air terminal and you will see a Connection Preview as shown in Figure 18
47. Click to place the Connector on this top face surface
48. Enter 1 on the command line to specify the connector number
49. Pick a point on the circumference of the Circular Profile to locate the poisition of the dimension for the connector

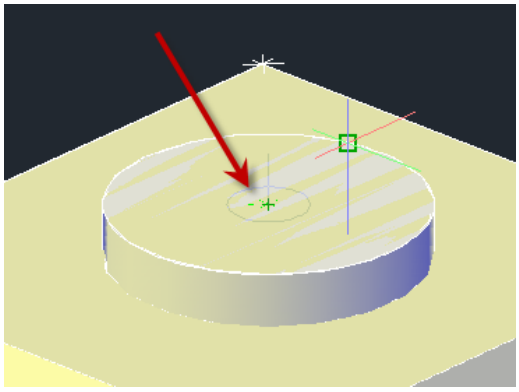


Figure 18

50. Expand the Connections TreeNode and then Select and Right Click on the Connector 1 TreeNode, select Edit…
51. Set the Domain to Duct and the Flow Direction to In, Click OK
52. Set the Visual Style to 2D Wireframe
53. Set the View Direction to Top
54. Expand the Size Parameters TreeNode, here you will see that the Dimension has created a new Size Parameter D1. Note there are also Parameters named WPOf1 and WPOf2, these are the offset distances between the Work Planes and were automatically created

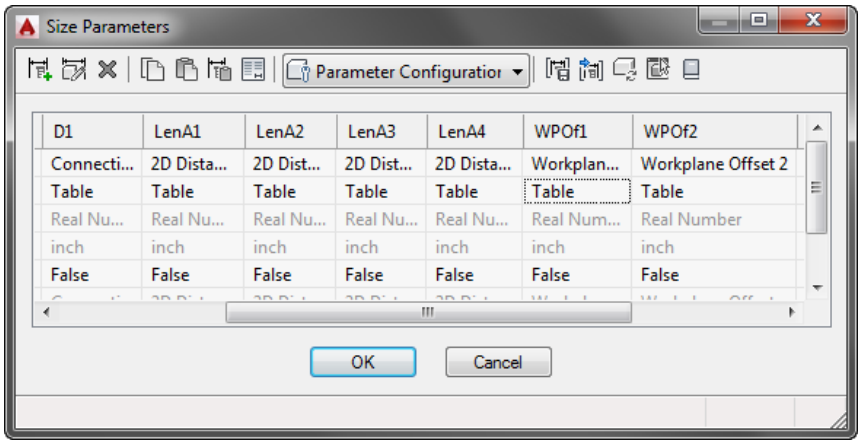
Next we need to specify paramerters to control the overall size of the Air Terminal, we will place Horixontal and Vertical Dimensions on the Bottom and Top of Main Body Work Planes

55. Expand the Work Planes TreeNode and Minimize the Bottom Plane, Top of Main Body and Connection TreeNodes
56. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Horizontal Dimension… then pick the top left COLE Point 2d followed by the top right COLE Point 2d pick a point above the Rectangular Profile and towards the middle to place the Dimension outside the extents of the Air Terminal then enter a value of 12 inches (12”) for the Dimension value
57. Repeat the process with a Vertical Dimension on the top right COLE Point 2d and the bottom right COLE Point 2d
58. Repeat the process for the Top of Main Body Geometry, this time using values of 9 inches (9”)

Now we need to create multiple sizes by putting data into the Size Parameters. There are different ways we can populate the data, for example if all widths andheight and diameter combinations were possible then we could simply store the data as Lists, then if there were

5 widths, 5 heights and 5 Diameters there would be 125 sizes available (5x5x5) However only certain combinations are available so we must store the data in a size Table (like an excel worksheet)

59. Select and Right Click on the Size Parameters TreeNode in the Conten Builder Palette and select Edit Configuration…
60. Set the Data Storage Type to Table for D1, LenA1, LenA2, LenA3, LenA4, WPOf1 & WPOf2 then Click OK



61. Select and Right Click on the Size Parameters TreeNode in the Conten Builder Palette and select Edit Values…
62. Enter the following values, 8, 12, 12, 9, 9, 1.125, 1 like the image below:

	Part Size Name	Connecti...	Geometry						Catalog E...	Dis...
	PrtSN	D1	LenA1	LenA2	LenA3	LenA4	WPOf1	WPOf2	IFCTC	LKey
1	Square Plaque Diffusers	8"	12"	12"	9"	9"	1 1/8"	1"	Undefined	Undefinec

63. Click the New Button  4 times to add a 4 additional rows to the Table, set the D1 value to 4, 5, 6 & 7 respectively for the new rows:

	Part Size Name	Connecti...	Geometry						Catalog E...	Dis...
	PrtSN	D1	LenA1	LenA2	LenA3	LenA4	WPOf1	WPOf2	IFCTC	LKey
1	Square Plaque Diffusers	4"	12"	12"	9"	9"	1 1/8"	1"	Undefined	Undefinec
2	Square Plaque Diffusers	5"	12"	12"	9"	9"	1 1/8"	1"	Undefined	Undefinec
3	Square Plaque Diffusers	6"	12"	12"	9"	9"	1 1/8"	1"	Undefined	Undefinec
4	Square Plaque Diffusers	7"	12"	12"	9"	9"	1 1/8"	1"	Undefined	Undefinec
5	Square Plaque Diffusers	8"	12"	12"	9"	9"	1 1/8"	1"	Undefined	Undefinec

64. Click the New Button  again to add a new row and set its values to 6, 24, 24, 16, 16, 2.5, 1 like the image below:

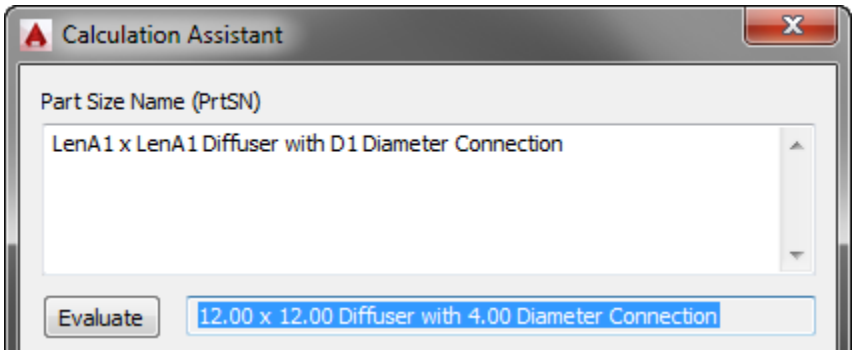
6	Square Plaque Diffusers	6"	24"	24"	16"	16"	2 1/2"	1"	Undefined	Undefinec

65. Click the New Button  5 times to add a 5 additional rows to the Table, set the D1 value to 8, 10, 12, 14 & 15 respectively for the new rows

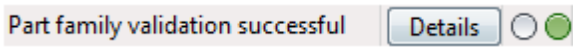
You will now have11 sizes but note that the PrtSN are all identical, we can change them manually, or we can have them auto name based on the other parameters, it makes sense

to use the auto naming especially if we were to have hundreds of sizes and maybe some that may change in the future

66. Change the drop down at the top of the Size Parameters Dialog box from Value to Calculations then double click on the Part Size Name Row 1 value, the Calculation Assistant will open up here we can make a combination of text and variables to construct a Part Size Name
67. Set the Part Size Name to LenA1 x LenA1 Diffuser with D1 Diameter Connection and click Evaluate to confirm it gives a value of 12.00 x 12.00 Diffuser with 4.00 Diameter Connection



68. Click OK twice to exit the Size Parameters Dialog boxes
69. Expand the Autolayout Data TreeNode in the Conten Builder Palette, Select and Right Click on the Layout Data TreeNode and select Select Placement Point, Pick the bottom left corner COLE Point 2d entity
70. To create a Preview Image click on the Generate Bitmap tool
71. Click Browse… to select a bmp file, navigate to the class folder and pick the file SPD.BMP then click open then click OK
72. Click on the Save tool to save the files to your content folder
73. Note at the bottom of the Content Builder there is a line stating that Part family validation failed with a red circle
74. Click on the Validate tool, the green circle should now display

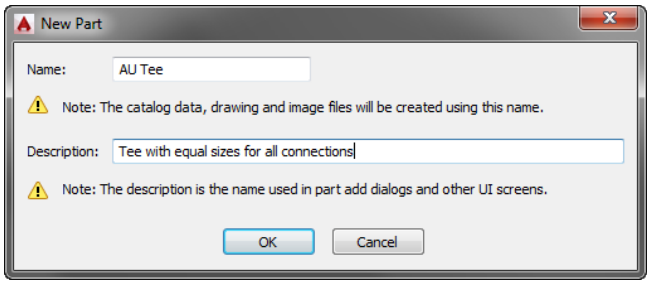


75. Click on the Options tool and Uncheck the Hide Part Flag, Select the Layer Key cell and set it to M-MV-DIFFUSERS, click OK to exit the Options Dialog box
76. Click on the Save tool to save the files to your content folder
77. Close the SPD Tab.
78. Use the MVPARTADD command to test your part, it should be ready to go!

An Example – Create a parametric Pipe Fitting

During this example we will create an Pipe Fitting using the Parametric Content Builder, the Pipe Fitting will be a simply Tee Fitting just to show how easy it is to create fittings with the Conten Builder

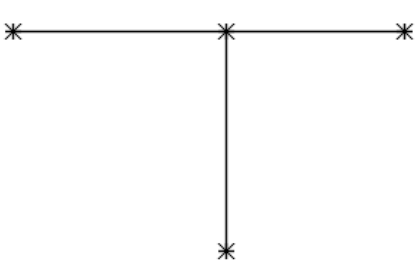
1. Lets start by opening the Content Builder from the Manage Tab –> MEP Content Panel
2. Select Pipe from the Part Domain Drop Down
3. Expand the General pipe (US Imperial) TreeNode then the Generic TreeNode
4. Select Butt Welded then click the Add Parametric Part  tool
    a. Set the Name as AU Tee
    b. Set the Description as Tee with equal sizes for all connections



5. Click OK
6. Expand the Part Configuration TreeNode in the Content Builder Palette, verify that the Description and Domain are already set as Tee with equal sizes for all connections and Pipe respectively
7. Double Click on the first Undefined TreeNode in the Content Builder Palette, you will then have a drop down list of available Pipe Part Types, select Tee

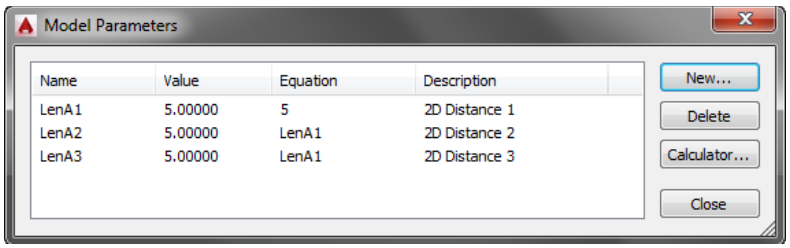
By setting the Type for the content the software will be able to complete some verification on the object, for example the software now knows to expect 3 Pipe fitting connectors in order to complete verification of the Tee

8. Next we need to create a Work Plane as this is required for the hosting of Profiles and Geometry, Expand the Modeling TreeNode
9. Right Click on Work Planes and select Add Work Plane…
10. Select *Bottom* and click OK, this will create a Work Plane that we can use to define the center line path of the Tee
11. Expand the Work Planes TreeNode
12. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Geometry -> Line....
13. Select 2 points within the scope of the yellow Work Planes to create a vertical line, the exact size is not important
14. Repeat the process to create 2 more lines, this time horizontal extending away from the top point of the vertical line like the image to the right

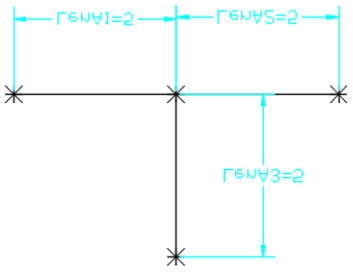


We are now going to place dimensions and constraints to control the size and position of the Tee

15. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Horizontal Distance....
16. Select the COLE Point 2d at the top left end of the COLE Line 2d and the COLE Point 2d at the intersection of all of the lines setting the value to 5 inches (5”)
17. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Horizontal Distance....
18. Select the COLE Point 2d at the top right end of the COLE Line 2d and the COLE Point 2d at the intersection of all of the lines setting the value to 5 inches (5”)
19. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Vertical Distance....
20. Select the COLE Point 2d at the bottom end of the vertical COLE Line 2d and the COLE Point 2d at the intersection of all of the lines setting the value to 5 inches (5”)
21. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Constraints -> Perpendicular....
22. Select the vertical COLE Line 2d then the left COLE Line 2d
23. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Constraints -> Perpendicular....
24. Select the vertical COLE Line 2d then the right COLE Line 2d
25. Now we will link the values of LenA1 and LenA2 to LenA3
26. Select and then Right Click on the Model Parameters TreeNode in the Content Builder and select Edit....
27. Set the Equation values as shown below



28. Click Close and your geometry should look like the image to the right

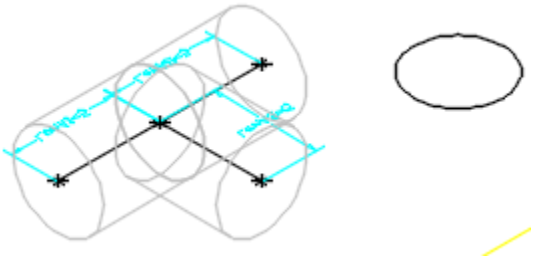


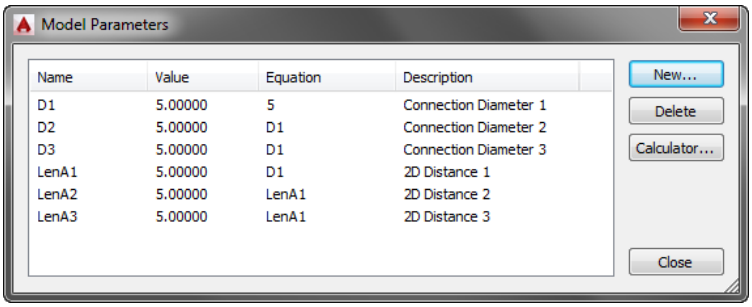
Next we are going to create Profiles for the end connections, we will need 1 Profile as all of the connectors in this example are going to be the same size

29. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Profile -> Circular....
30. Select 2 points within the scope of the yellow Work Planes to create a Circular Profile, the exact size is not important, but aim for approx 3 inches in Diameter

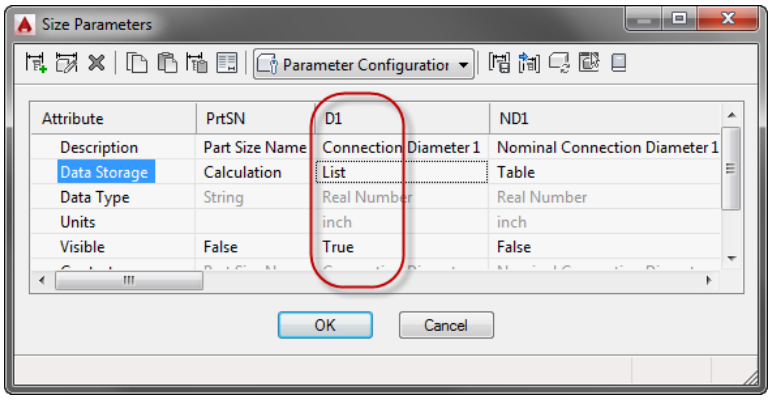
Now we can create the solid form of the Tee, we will use a Path Modifier for this

31. Select and then Right Click on the Modifiers TreeNode in the Content Builder and select Add Path…
32. Select the Vertical COLE Line 2d then select the Circular Profile twice to define the start and end profiles
33. Repeat the process for the 2 horizontal COLE Line 2d's
34. Set the View to South West Isometric
35. Select and then Right click on the Connections TreeNode and select Add Connection… move the cursor to the face on the left of the view in the South West Isometric View. Click to place the Connector on this face
36. Enter 1 on the command line to specify the connector number
37. Click a Point close to but outside the Profile to position the Dimension
38. Select and then Right click on the Connections TreeNode and select Add Connection… move the cursor to the face at the end of the vertical line. Click to place the Connector on this face
39. Enter 2 on the command line to specify the connector number
40. Select and then Right click on the Connections TreeNode and select Add Connection… move the cursor to the remaining face in the South West Isometric View. Click to place the Connector on this face
41. Enter 3 on the command line to specify the connector number
42. Select and then Right Click on the Model Parameters TreeNode in the Content Builder and select Edit....
43. Set the Equation values as shown below

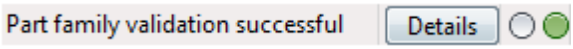




44. Select and Right Click on the Size Parameters TreeNode in the Content Builder Palette and select Edit Configuration…
45. Set the Data Storage value for D1 to List and the Visible setting for D1 to True



46. Change the drop down at the top of the Size Parameters Dialog box from Configuration to Values then select the cell under D1
47. Click the New tool then use the Add button to add the following sizes: 2, 4, 6, 8, 12, 14, 16, 18 & 20
48. Click on the Save tool to save the files to your content folder
49. To create a Preview Image click on the Generate Bitmap tool
50. Click on the South West Isometric tool in the Generate View area to create an automatic view for the element
51. Click on the Validate tool, the green circle should now display

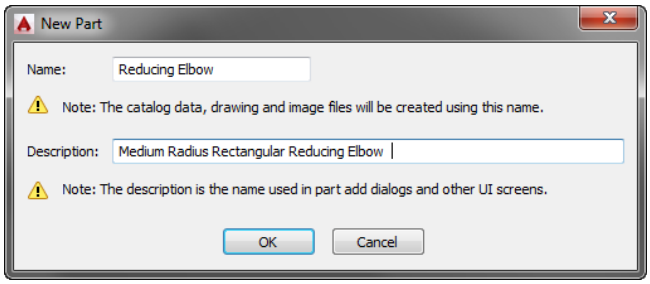


52. Click on the Options tool and Uncheck the Hide Part Flag, and check the Custom Sizing Flag, click OK to exit the Options Dialog box
53. Click on the Save tool to save the files to your content folder
54. Close the AU Tee drawing Tab
55. Use the PIPEFITTINGADD command to test the new part

An Example – Create a parametric Duct Fitting

During this example we will create an Duct Fitting using the Parametric Content Builder, the Duct Fitting will be a simpe Reducing Elbow Fitting just to show how easy it is to create fittings with the Content Builder

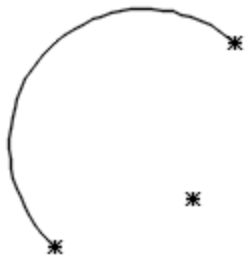
1. Lets start by opening the Content Builder from the Manage Tab –> MEP Content Panel
2. Select Duct from the Part Domain Drop Down
3. Expand the Rectangular TreeNode
4. Select Elbows then click the Add Parametric Part  tool
    a. Set the Name as Reducing Elbow
    b. Set the Description as Medium Radius Rectangular Reducing Elbow



5. Click OK
6. Expand the Part Configuration TreeNode in the Content Builder Palette, verify that the Description and Domain are already set as Medium Radius Rectangular Reducing Elbow and Duct respectively
7. Double Click on the first Undefined TreeNode in the Content Builder Palette, you will then have a drop down list of available Duct Part Types, select Elbow

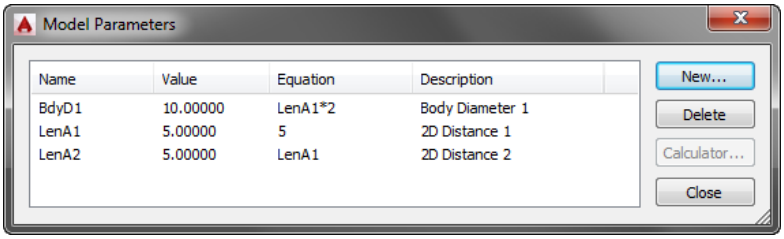
By setting the Type and Sub Type for the content the software will be able to complete some verification on the object, for example the software now knows to expect 2 duct fitting connectors in order to complete verification of the Elbow

8. Next we need to create a Work Plane as this is required for the hosting of Profiles and Geometry, Expand the Modeling TreeNode
9. Right Click on Work Planes and select Add Work Plane…
10. Select *Bottom* and click OK, this will create a Work Plane that we can use to define the center line path of the elbow
11. Expand the Work Planes TreeNode
12. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Geometry -> Arc....
13. Select 3 points within the scope of the yellow Work Planes to create n arc, the exact size is not important
14. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Geometry -> Point....
15. Place the Point close to the center of the arc geometry, the exact location is not important

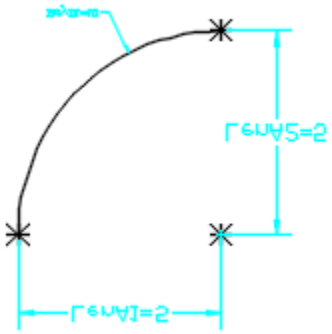


We are now going to place dimensions and constraints to control the size and position of the arc and point

16. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Horizontal Distance....
17. Select the COLE Point 2d at the lower end of the arc and the COLE Point 2d then place the dimension to the bottom of the geometry setting the value to 5 inches (5”)
18. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Vertical Distance....
19. Select the COLE Point 2d at the upper end of the arc and the COLE Point 2d then place the dimension to the right of the geometry setting the value to 5 inches (5”)
20. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Dimension -> Diameter....
21. Select the COLE CircArc 2d then place the dimension above the geometry
22. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Constraints -> Concentric....
23. Select the COLE CircArc 2d then the single COLE Point 2d
24. Now we will link the values of LenA2 and D1 to LenA1
25. Select and then Right Click on the Model Parameters TreeNode in the Content Builder and select Edit....
26. Set the Equation values as shown below



27. Click Close and your geometry should look like the image to the right

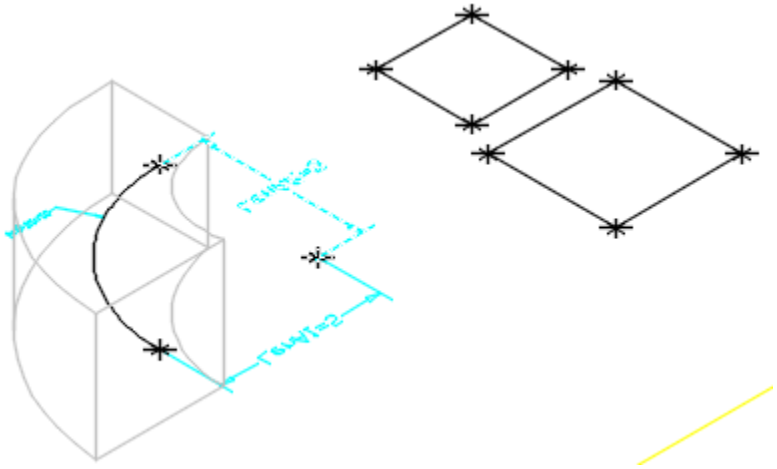


Next we are going to create Profiles for the end connections, we will need 2 Profiles as there will be two different size connections in order to create the reducing form

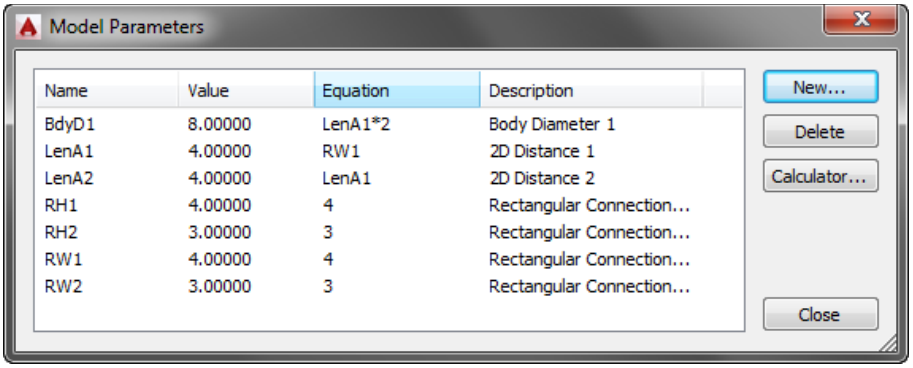
28. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Profile -> Rectanglar....
29. Select 2 points within the scope of the yellow Work Planes to create a Rectangle, the exact size is not important, but aim for approx 4x4
30. Select and then Right Click on the Bottom Plane in the Content Builder and select Add Profile -> Rectanglar....
31. Select 2 points within the scope of the yellow Work Planes to create a Rectangle, the exact size is not important, but aim for approx 3x3

Now we can create the solid form of the reducing elbow, we will use a Path Modifier for this

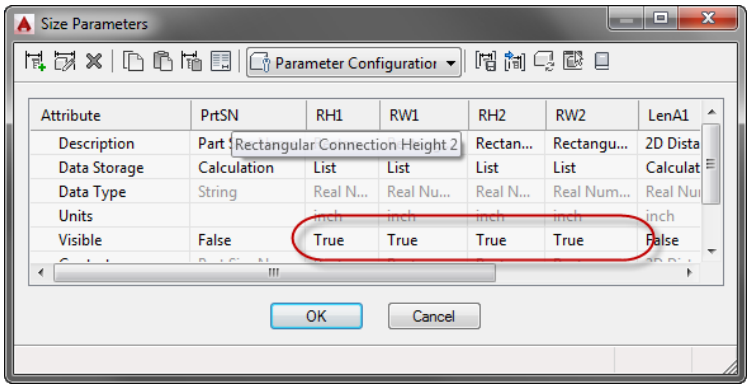
32. Select and then Right Click on the Modifiers TreeNode in the Content Builder and select Add Path…
33. Select the COLE CircArc 2d then select the Larger Profile followed by the Smaller Profile
34. Set the View to South West Isometric
35. Select and then Right click on the Connections TreeNode and select Add Connection… move the cursor to the face closest to you in the South West Isometric View. Click to place the Connector on this face



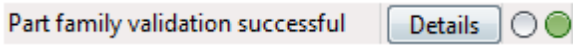
36. Enter 1 on the command line to specify the connector number
37. Select the 2 COLE Point 2d elements along the right side of the larger Profile and place the dimension to the right of the profile
38. Select the 2 COLE Point 2d elements along the bottom side of the larger Profile and place the dimension to the bottom of the profile
39. Select and then Right click on the Connections TreeNode and select Add Connection… move the cursor to the face farthest from you in the South West Isometric View. Click to place the Connector on this face
40. Enter 2 on the command line to specify the connector number
41. Select the 2 COLE Point 2d elements along the right side of the smaller Profile and place the dimension to the right of the profile
42. Select the 2 COLE Point 2d elements along the top side of the smaller Profile and place the dimension to the top of the profile
43. Select and then Right Click on the Model Parameters TreeNode in the Content Builder and select Edit....
44. Set the Equation values as shown below



45. Select and Right Click on the Size Parameters TreeNode in the Content Builder Palette and select Edit Configuration…
46. Set the Visible setting to True for RH1, RH2, RW1 & RW2 then Click OK
47. Change the drop down at the top of the Size Parameters Dialog box from Configuration to Values then select the cell under RH1



48. Click the New tool then use the Add button to add the following sizes: 2, 4, 6, 8, 12, 14, 16, 18 & 20
49. Repeat this process for RW1, RH2 and RW2
50. You have just created 10,000 part sizes (10x10x10x10)
51. To create a Preview Image click on the Generate Bitmap tool
52. Click Browse… to select a bmp file, navigate to the class folder and pick the file RRE.BMP then click open then click OK
53. Click on the Validate tool, the green circle should now display

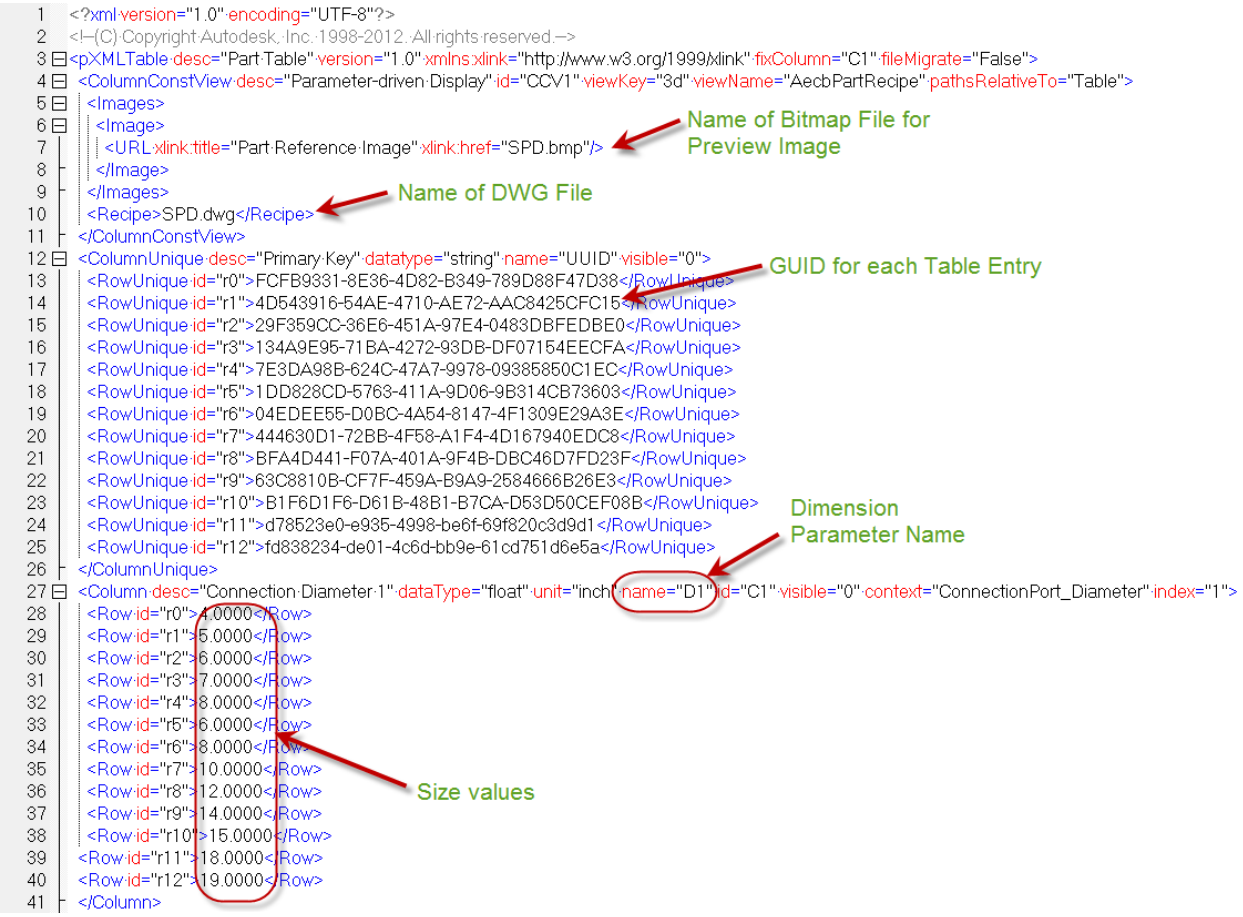


54. Click on the Options tool and Uncheck the Hide Part Flag, and check the Custom Sizing Flag, click OK to exit the Options Dialog box
55. Click on the Save tool to save the files to your content folder
56. Close the Reducing Elbow drawing Tab
57. Use the DUCTFITTINGADD command to test the new part

Appendix A – Editing Parts with an XML Editor

Parametric Multi-view Parts have 3 files associated with them as mentioned above; a DWG, a BMP and a XML. XML files can be directly edited via an ASCII text editor or a XML Specific editor. I use XML Copy Editor (http://xml-copy-editor.sourceforge.net/), but I am sure there are others out there with other features, but for ease of use and the fact that its free it's a great choice. You will need to be somewhat familiar with the parameters within the content you want to edit, but its fairly easy to understand.

As an example, let's take a look at the SPD.xml file that you created in the Air Terminal example.




```
<?xml version="1.0" encoding="UTF-8"?>
<!--(C) Copyright Autodesk, Inc. 1998-2012. All rights reserved.-->
<pXMLTable desc="Part Table" version="1.0" xmlns:xlink="http://www.w3.org/1999/xlink" fixColumn="C1" fileMigrate="False">
 <ColumnConstView desc="Parameter-driven Display" id="CCV1" viewKey="3d" viewName="AecbPartRecipe" pathsRelativeTo="Table">
  <Images>
   <Image>
    <URL xlink:title="Part Reference Image" xlink:href="SPD.bmp"/>
   </Image>
  </Images>
  <Recipe>SPD.dwg</Recipe>
 </ColumnConstView>
 <ColumnUnique desc="Primary Key" datatype="string" name="UUID" visible="0">
  <RowUnique id="r0">FCFB9331-8E36-4D82-B349-789D88F47D38</RowUnique>
  <RowUnique id="r1">4D543916-54AE-4710-AE72-AAC8425CFC15</RowUnique>
  <RowUnique id="r2">29F359CC-36E6-451A-97E4-0483DBFEDBE0</RowUnique>
  <RowUnique id="r3">134A9E95-71BA-4272-93DB-DF07154EECFA</RowUnique>
  <RowUnique id="r4">7E3DA98B-624C-47A7-9978-09385850C1EC</RowUnique>
  <RowUnique id="r5">1DD828CD-5763-411A-9D06-9B314CB73603</RowUnique>
  <RowUnique id="r6">04EDEE55-D0BC-4A54-8147-4F1309E29A3E</RowUnique>
  <RowUnique id="r7">444630D1-72BB-4F58-A1F4-4D167940EDC8</RowUnique>
  <RowUnique id="r8">BFA4D441-F07A-401A-9F4B-DBC46D7FD23F</RowUnique>
  <RowUnique id="r9">63C8810B-CF7F-459A-B9A9-2584666B26E3</RowUnique>
  <RowUnique id="r10">B1F6D1F6-D61B-48B1-B7CA-D53D50CEF08B</RowUnique>
  <RowUnique id="r11">d78523e0-e935-4998-be6f-69f820c3d9d1</RowUnique>
  <RowUnique id="r12">fd838234-de01-4c6d-bb9e-61cd751d6e5a</RowUnique>
 </ColumnUnique>
 <Column desc="Connection Diameter 1" dataType="float" unit="inch" name="D1" id="C1" visible="0" context="ConnectionPort_Diameter" index="1">
  <Row id="r0">4.0000</Row>
  <Row id="r1">5.0000</Row>
  <Row id="r2">6.0000</Row>
  <Row id="r3">7.0000</Row>
  <Row id="r4">8.0000</Row>
  <Row id="r5">6.0000</Row>
  <Row id="r6">8.0000</Row>
  <Row id="r7">10.0000</Row>
  <Row id="r8">12.0000</Row>
  <Row id="r9">14.0000</Row>
  <Row id="r10">15.0000</Row>
  <Row id="r11">18.0000</Row>
  <Row id="r12">19.0000</Row>
 </Column>
```

So you can see there are easily identifiable data values within the xml file, I have actually added 2 additional rows (r11 & r12) to the file with 2 new values for D1 of 18 and 19 respectively.

I used the following website to make the GUID:

http://www.guidgenerator.com/online-guid-generator.aspx

It's a useful resource for programming as well as anywhere you happen to need a 128 bit unique value.