Autodesk University 2012

Fast Content for AutoCAD® MEP 2013

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MP1465-L

AutoCAD MEP 2013 software, a world-class design and drafting application, continues to improve. This release adds more power and flexibility and goes a long way toward improving the user experience. In this information-packed hands-on lab, we will take a look at creating custom content. We will start by using Autodesk® Inventor® Fusion to create a quick 3D DWG™ model. Next, we will use this model to create a new multi-view part. Once the part is defined, you will learn a quick way to use the new symbol and annotation planes to add line-based symbols, and then create a new catalog to store your custom content. Every AutoCAD MEP user needs to learn how to use the overwhelming volume of manufacturer's 3D content—this lab will help you learn how to do this!

Learning Objectives

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

- Use Inventor Fusion to create 3D DWG models
- Create a multi-view part quickly from manufacturer content
- Add and edit symbol and annotation planes for multi-view parts
- Create and store custom content catalogs

About the Speaker

David is a BIM Specialist for Gannett Fleming, a multi-discipline engineering firm based in Camp Hill, PA, with 60 offices in the US and overseas. Based in the Raleigh, NC office, he provides BIM Implementation and training for the firm's engineering design software, including Revit, Navisworks, AutoCAD MEP/P&ID and more. He has 27 years of experience in both the design and Autodesk VAR channel, spending 13 years working as an instructor and consultant for the Autodesk building design product line. David also worked as a training manager while in the channel, and was a member of the Autodesk ATC Advisory Board for 2009-10. He is a Revit Architecture Certified Professional, and also earned the MEP Implementation Certified Expert title.

David has spoken at AU for several years, and was named the Top Speaker for both labs and lectures at AU 2011. As an author, he also contributes to 4D Technology's CADLearning training programs and has written several training manuals on Revit MEP.

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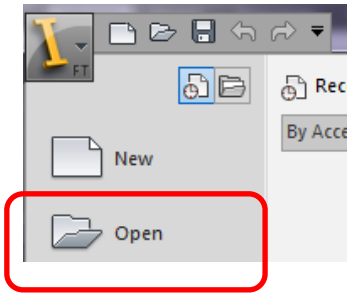
Introduction

AutoCAD MEP is the old man in the room when it comes to 3D modeling in the MEP world. And it's still one of my favorites, with an ever-improving set of tools that make the move to BIM easier than ever. But as the industry progress into the 3D world, we really need to learn how to get our content made quickly – whether it's a custom model, or content provided by a manufacturer. This session will take us from the part to the catalog, so let's get rolling!

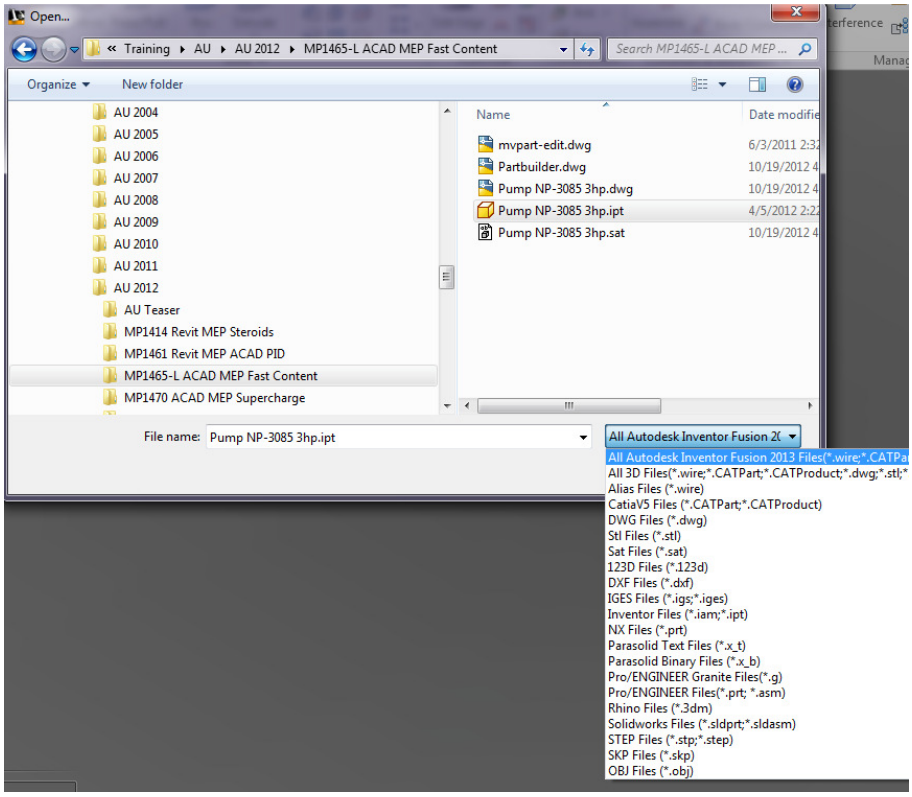
Converting Manufacturer Content with Inventor Fusion

Autodesk released **Inventor Fusion** a couple of years ago. This tool allows users to take nearly ever modeling format, including .IPT, .SAT, .IGES, .STEP and more, and make minor edits. The main reason we use it is for converting these formats to 3D DWG, so we can use the parts to create our own intelligent content. In this lesson, we're going to learn how to use this software package to save a pump model from ***Flygt***, a major pump manufacturer, and create the DWG file we need to use this in a project.

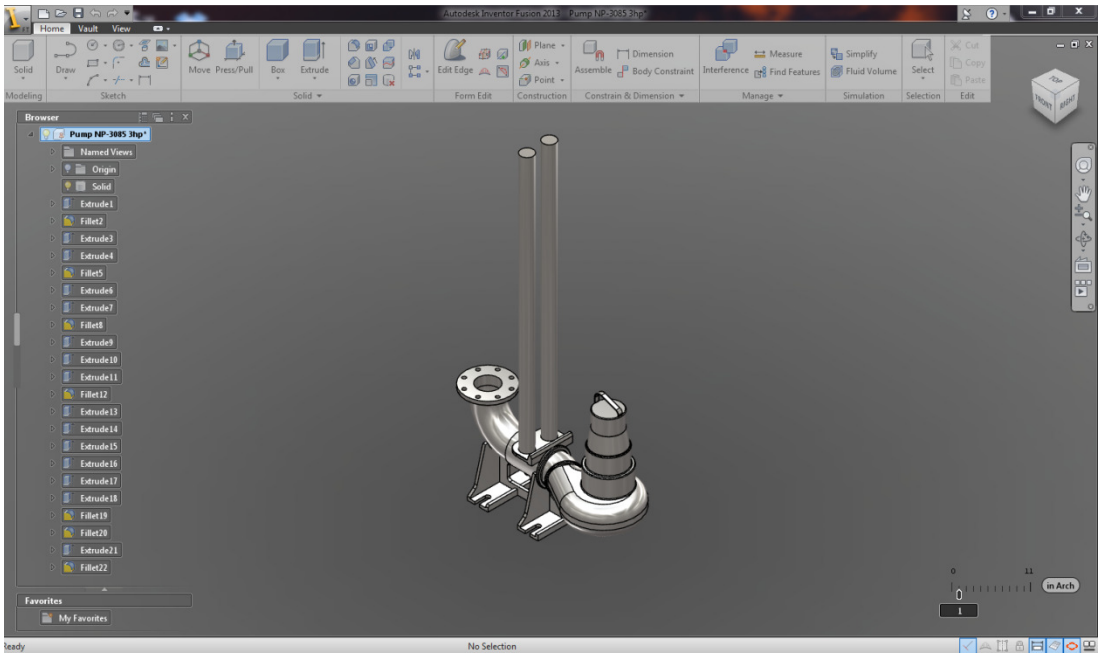
1. From Fusion, use the *application menu* to open the model file:



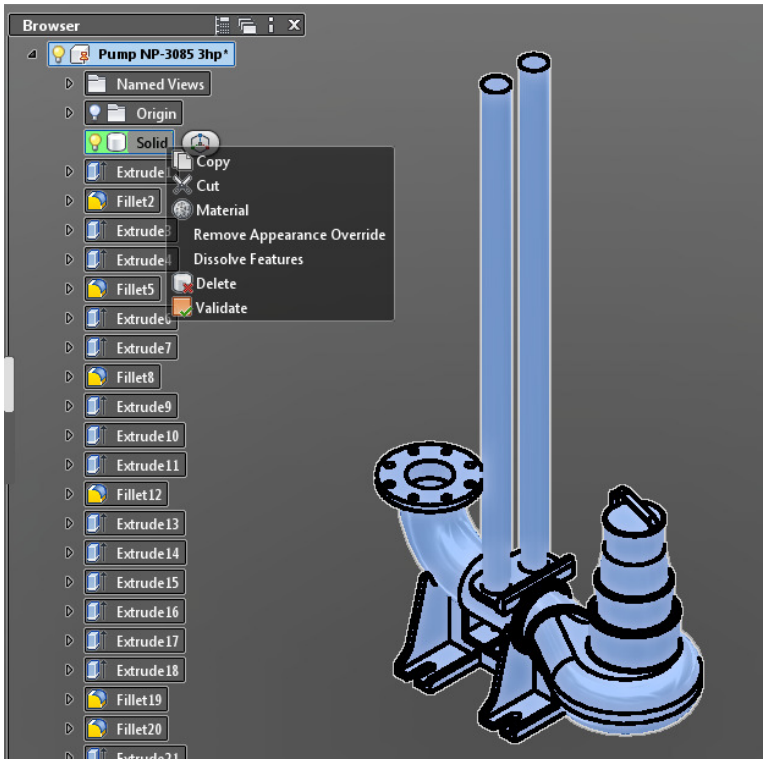
2. When the Open dialog appears, check out the file types you can use:



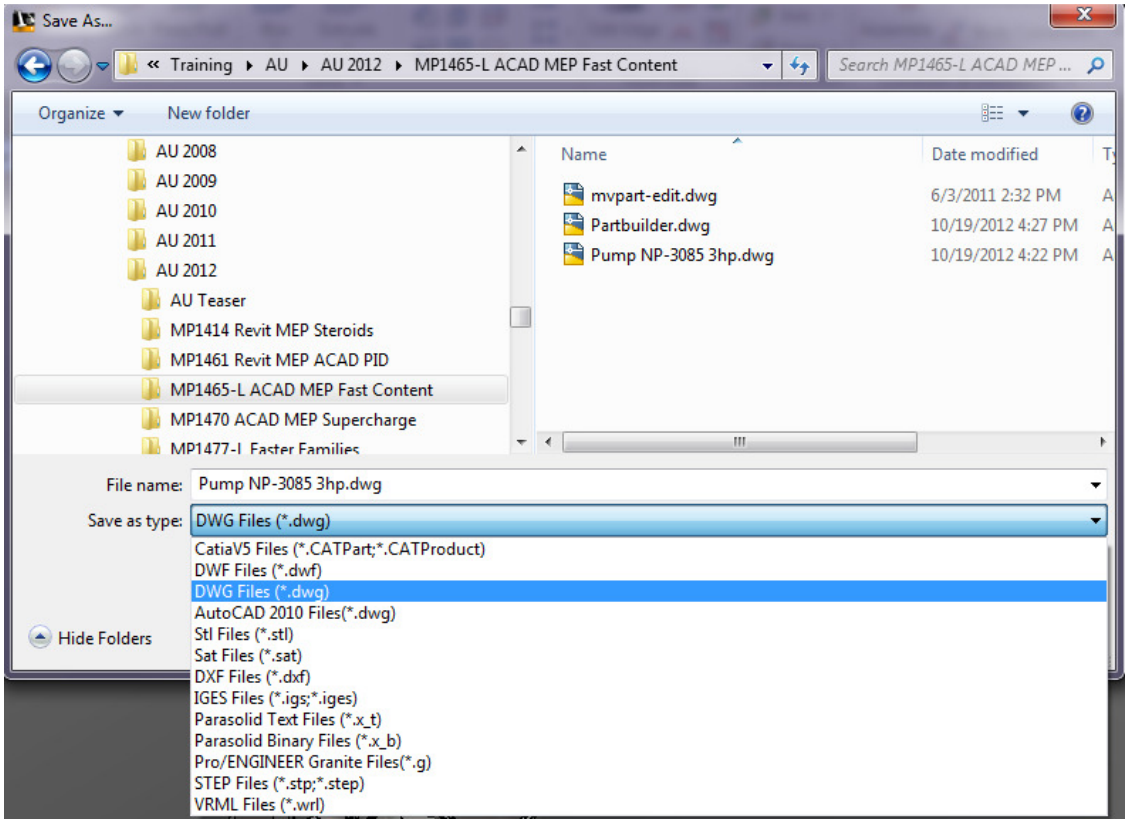
3. You can see how this program becomes the “jack of all trades” when it comes to file conversion. Select the **Pump NP 3085 3hp.ipt** file. Select *Open* to continue.
4. When the file opens, you’ll see a 3D model of the part:



5. Fusion includes a browser that lets you select and review items in the screen area. If you select a component in the list, it is highlighted in the view. If you right click on the component, you can *copy, cut, edit material, dissolve features, delete* and *validate*.



6. The tool features depend on the type of component you select, so an extrude will list different tools than a solid, for example. While this class does not go into editing in Fusion, be aware that you can make some changes to file before you convert it.
7. To convert a file, this is really tough – go to the *Save As* command on the application menu. When the dialog opens, you have the following options:



8. You can save to DWG, DWF, STL, SAT, DXF, IGES, STEP and other modeling formats. When you select *Save*, you've created the DWG you need to make the part.

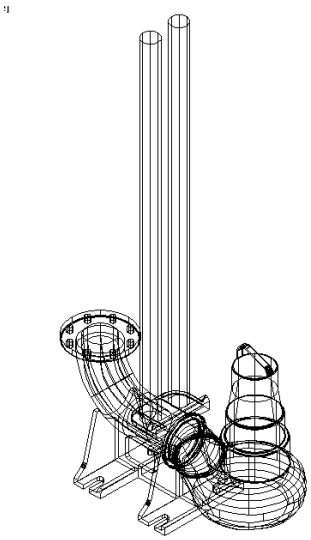
And that's all there is to it – you can save the file to the AU dataset folder, or any location you want. With Fusion, it doesn't take a lot to convert the files. This should expand your part universe exponentially!

Create the Multi-view Part from Manufacturer Content

Once you get the 3D model into DWG format, you can use it to build your multiview part. There's a lot of different ways to approach this, but it boils down to a few simple things. You need a 3D model to make the part. The Content Builder will use this to generate 2D plan and elevation views based on the model, but you can also define your own view blocks. If you want to use a custom symbol, you'll need that defined before you start the content builder. No matter what the part type is, the process is basically the same, for most equipment. In this exercise, we'll walk through the steps of adding this part, so let's get started.

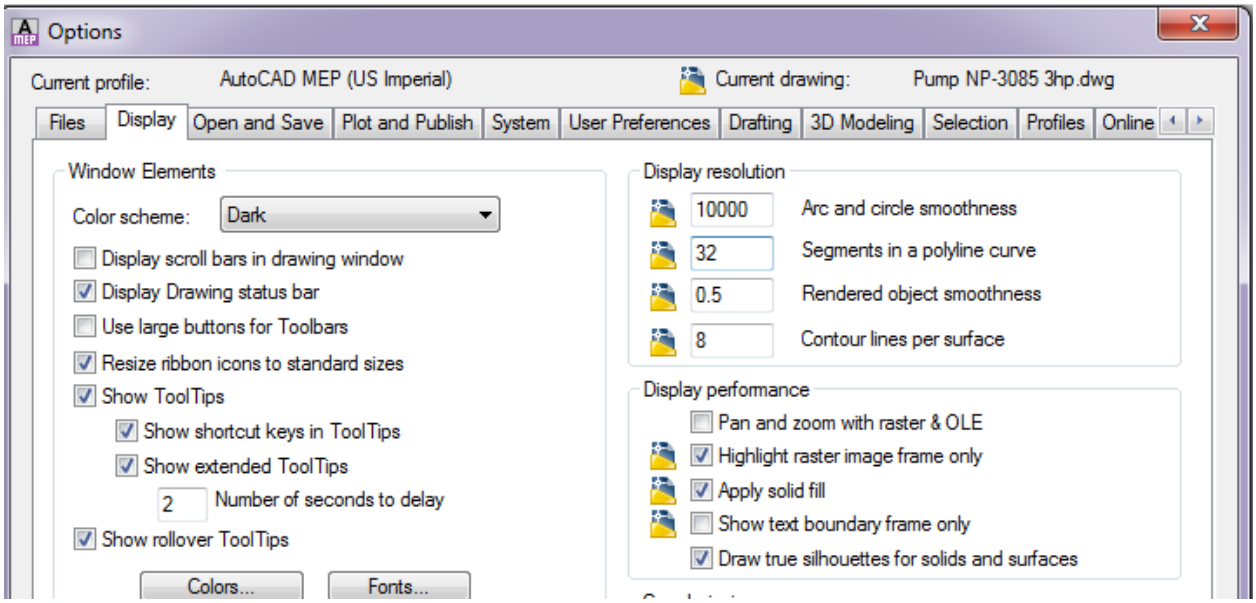
Creating 2D View Blocks from a Model

To start, open the drawing **Pump NP – 3085 3hp.dwg**. The part should be displayed from a 3D point of view:

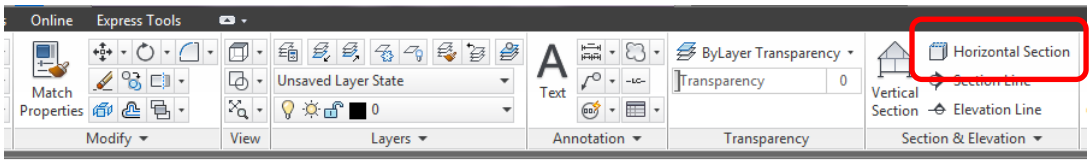


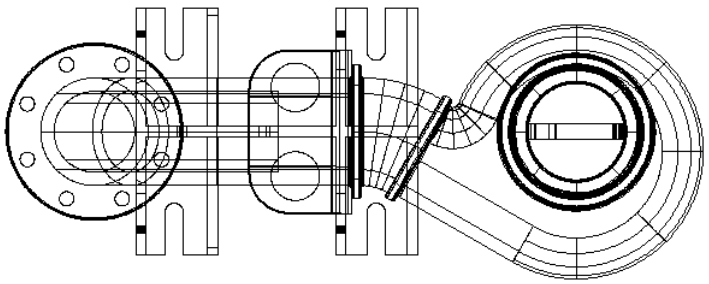
When I'm working with a part like this, I like to make a few changes to options, to make the linework as smooth as possible.

1. Right-click on the command line and choose *Options*. On the display, I like to change a few items:

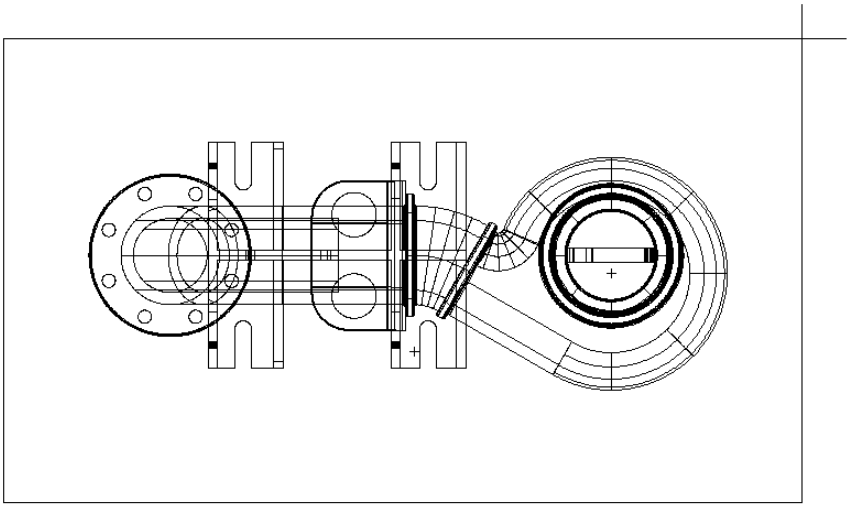


2. Set the *arc and circle smoothness* to a higher number, such as **10000**. Set the *segments in a polyline curve* to **32**, and check the option to *draw true silhouettes for solids and surfaces*. This will help the part look a little better when placed in a drawing. Select *OK* to close the options dialog.
3. Normally the *Content Builder* will generate 2D views for you, but if you want to create your own version quickly, here's a couple of tips. Autodesk has had the command CREATEHLR around since the first release of Architectural Desktop, but when working with solids, you may get segment lines along round objects. You can leverage the *Horizontal* and *Vertical Section* tools to create a plan section, then use CREATEHLR to create a nice, clean 2D block. This block can be edited separately, and used to replace the default blocks created in the Content Builder. To start, change to the *top* view.
4. From the *Home* tab, *Section and Elevation* panel, select the *Horizontal Section* tool:

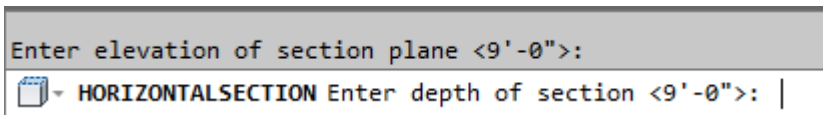




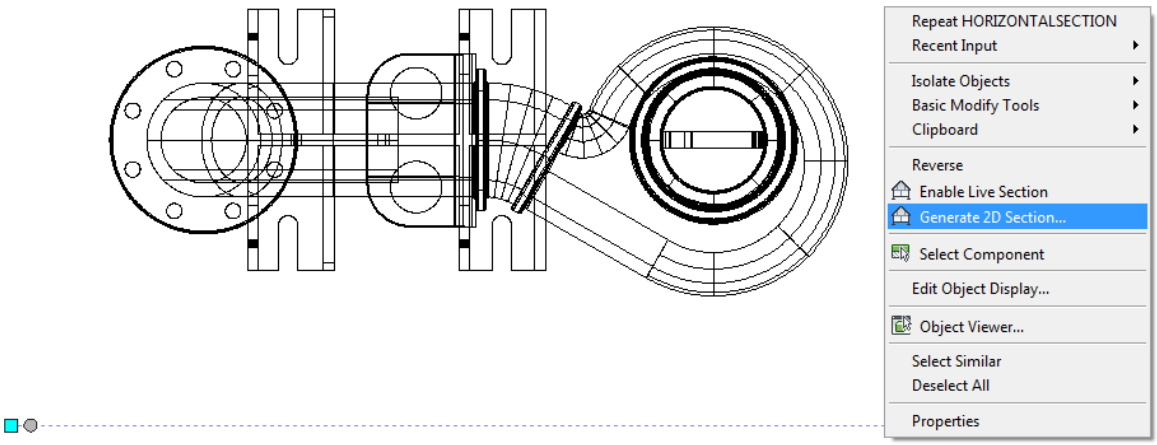
5. Pick two points to place a box around the pump:



6. When prompted, press enter twice to accept the *elevation of the section plane* and *depth of section* at *9'*:

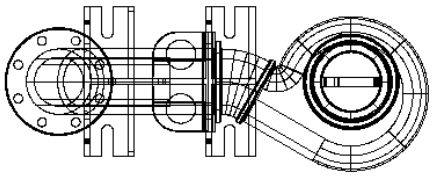


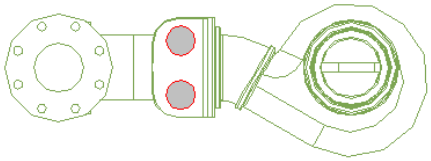
7. You'll see the section line appear. Pick it, right-click and choose *Generate 2D Section*:



8. From the dialog, pick *Select Objects*, and then pick the pump. Once the pump is selected, right-click to return to the dialog.

9. Under *Placement*, use the *pick point* command to select a point to the right of the pump. Once these items are set, leave the remaining settings as-is, and press *OK* to close the dialog.

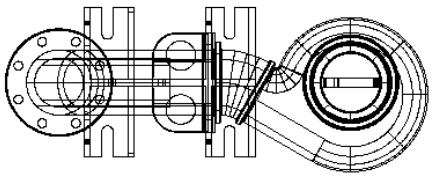


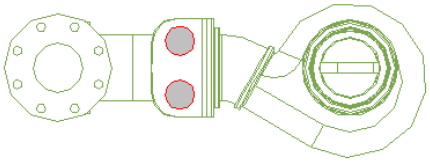


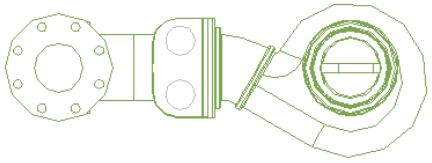
New 2D Section Block

10. The 2D Section will appear. Next, type in CREATEHLR on the command line and press enter. When prompted, selected the new section view:

11. Press enter after selecting the section block. Pick a point below the section, and then press enter to accept the default (placing the block in a plan view).

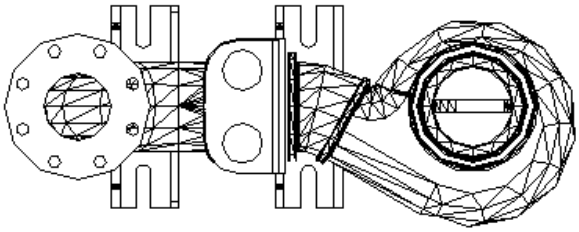




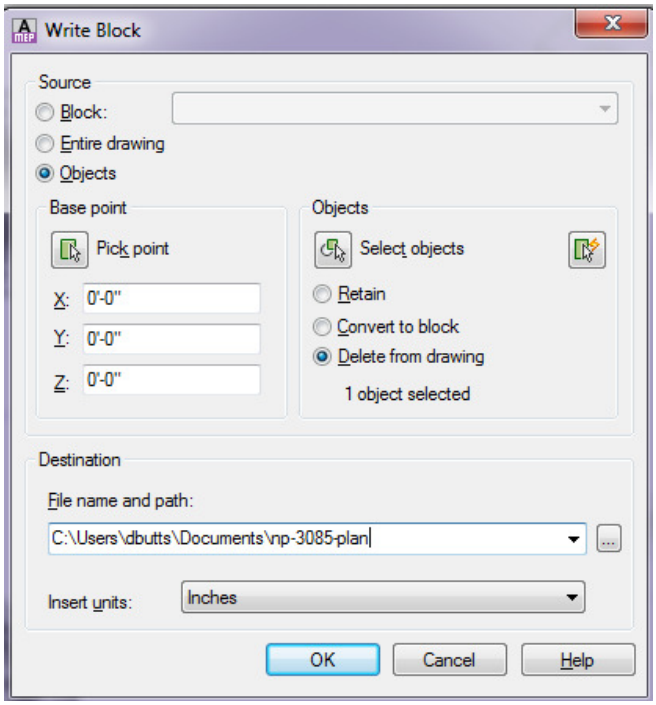


New **CreateHLR** Block

12. Notice that shading objects carried from the style don't carry across, which is what I'm looking for. As stated before, you can't go straight to CREATEHLR from the model, as it creates an image like this:



13. Let's clean this file up. Type in WBLOCK, and press enter. Pick the *Select objects* icon, and pick the new plan view block. Once it's selected, select the *Delete from drawing* option, and then set the *file name and path location* for the new block, so you can locate it later.



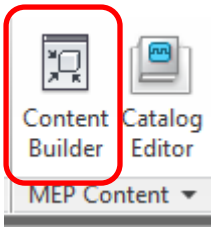
Select *OK* to close the dialog and save the block. Delete the section and section block from the drawing to continue.

I only use this step if I'm going to customize the plan view block from something other than what Content Builder creates. It's important to know how to leverage the model to make this block, and save yourself some steps.

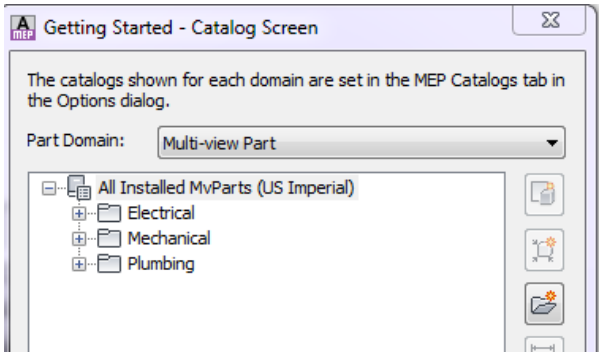
Using Content Builder to Define the Multi-View Part

Once you have the 3D model, you can move right into creating the part. To use the Content Builder, make sure you have one of the MEP workspaces set current, such as HVAC or Electrical.

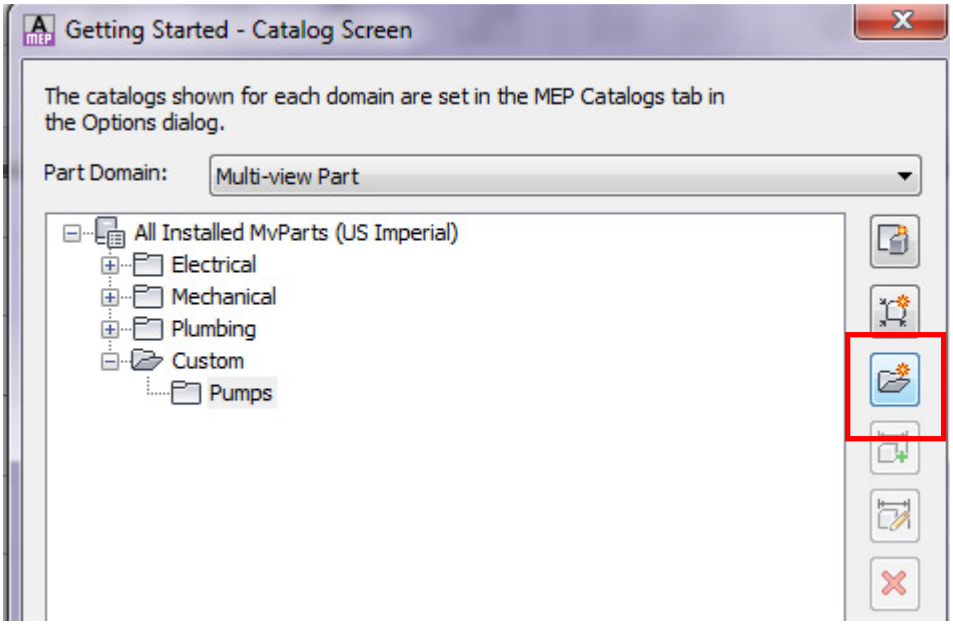
1. Make sure you are looking at the model from a 3D isometric view, so you can easily place connections. From the *Manage* tab, select the *Content Builder* tool:



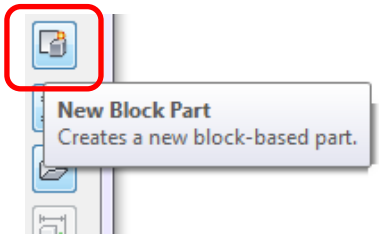
2. When the dialog opens, set the *Part Domain* to *Multi-view Parts*:



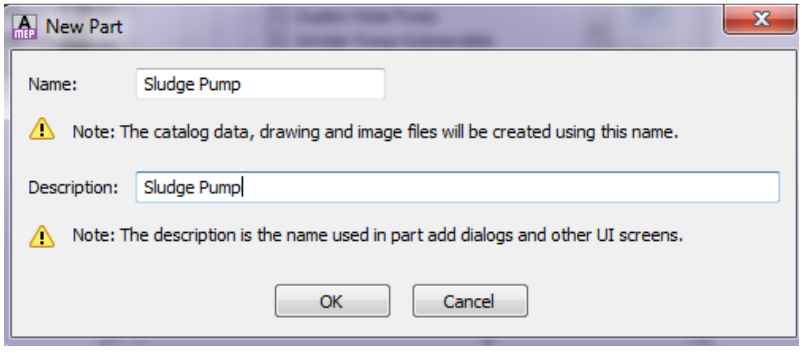
3. Select the *New Chapter* icon. Add a chapter under All *Installed MvParts (US Imperial)* named **Custom**, and then add a sub-chapter named **Pumps**:



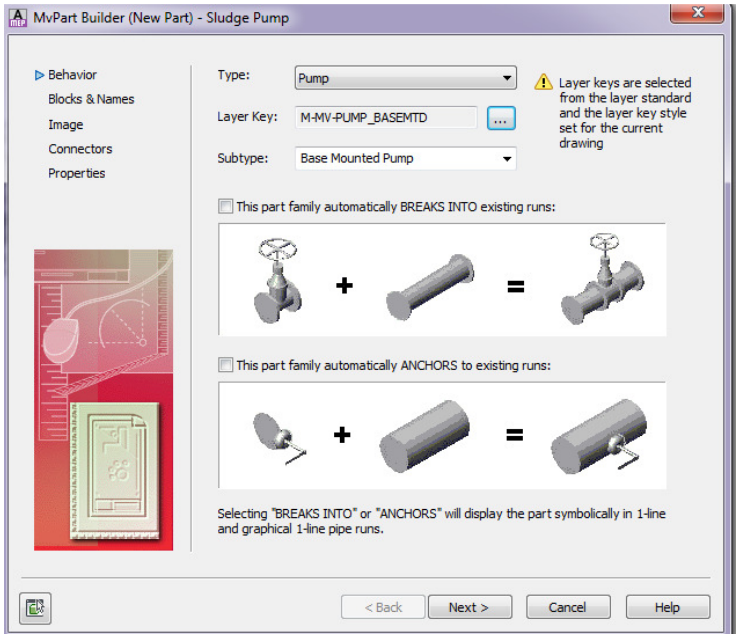
4. Next, select the *New Block Part* Tool:



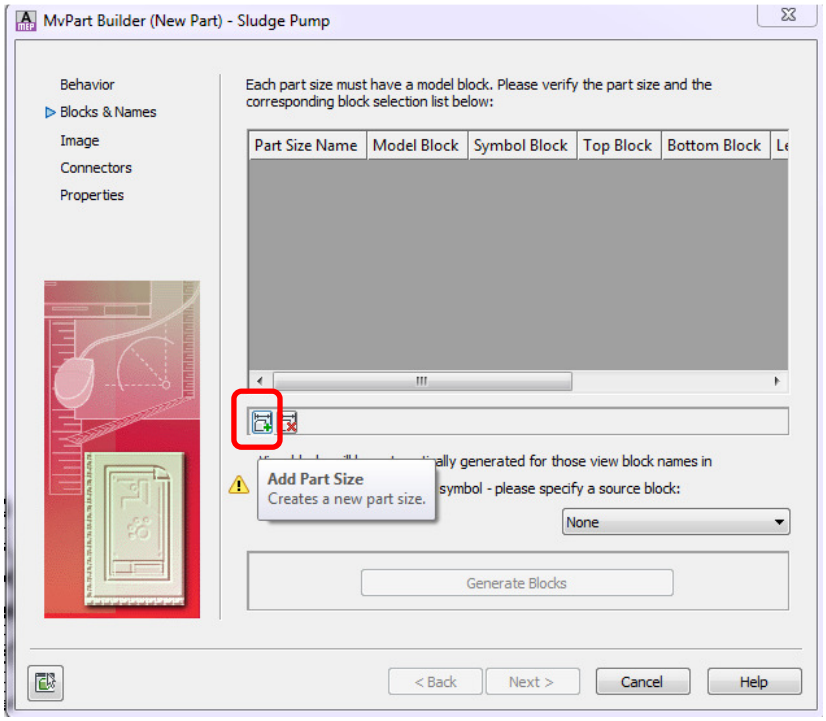
5. For the name, enter **Sludge Pump**. When you pick the description field, the name will automatically be populated by the part name:



6. Click *OK*. The *MvPart Builder* will appear. Set the type to *Pump*, and then pick the *M-MV-PUMP_BASEMTD* layer key from the default layer key style list. Make sure the Subtype is set to *Base Mounted Pump*:

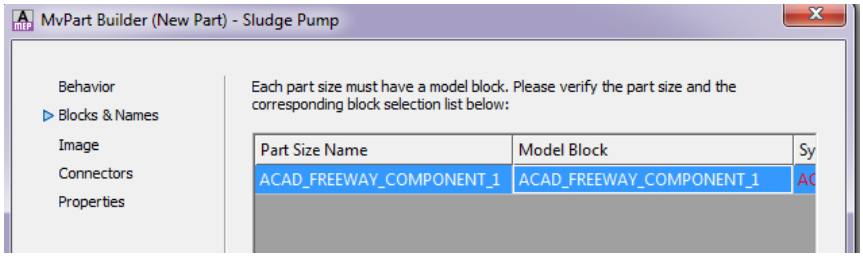


7. Now that the behavior is set, click *Next*. For the *Blocks and Names* page, pick the *Add Part Size* icon:

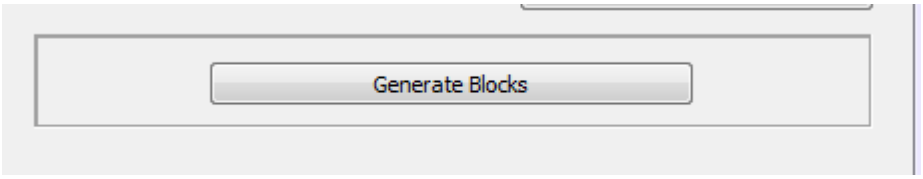


8. Pick the 3D model block, which should be the only block in the drawing (*note: you can rename this block as needed, prior to making the part – the block name is used to define*

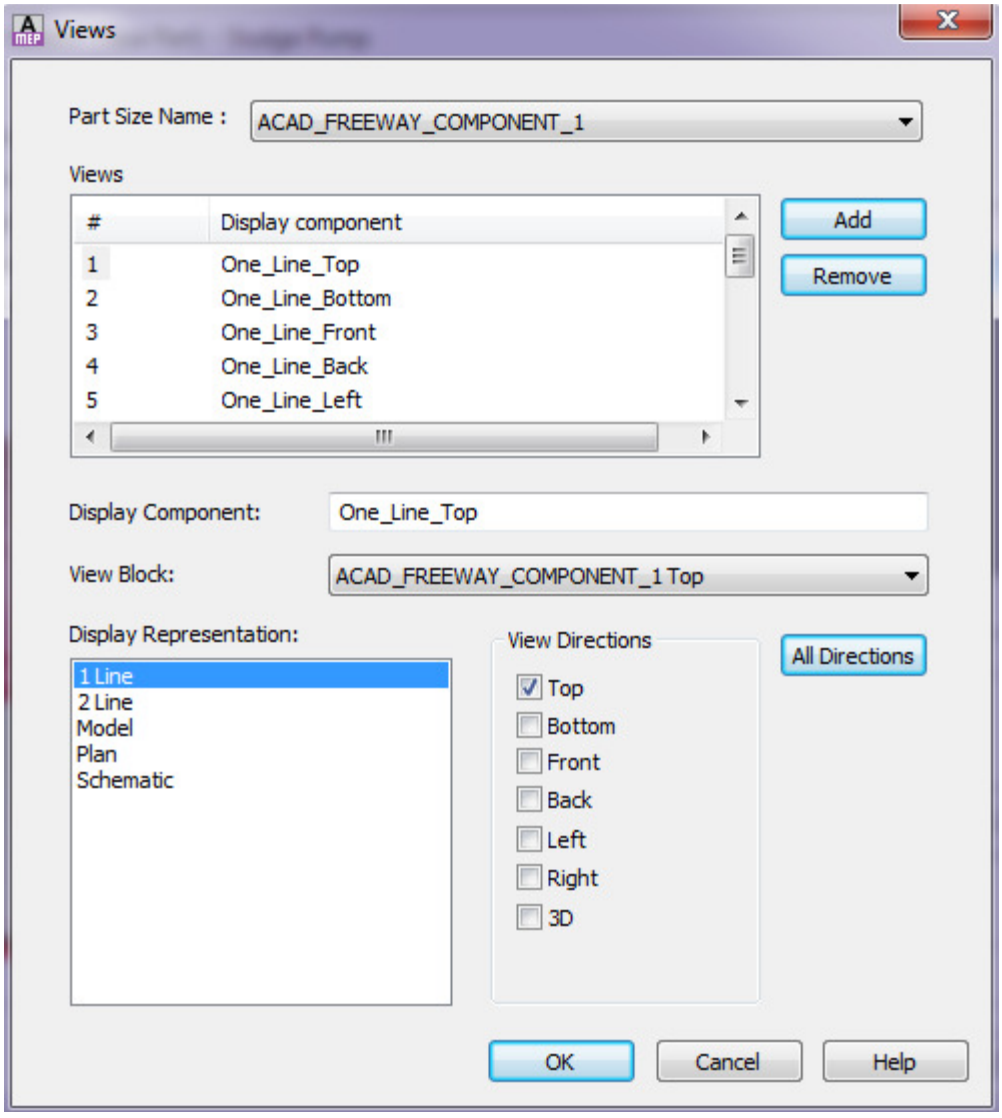
the part name, which we'll see later). Once the part is selected, the dialog will show what blocks will be defined for the elevation and plan views:



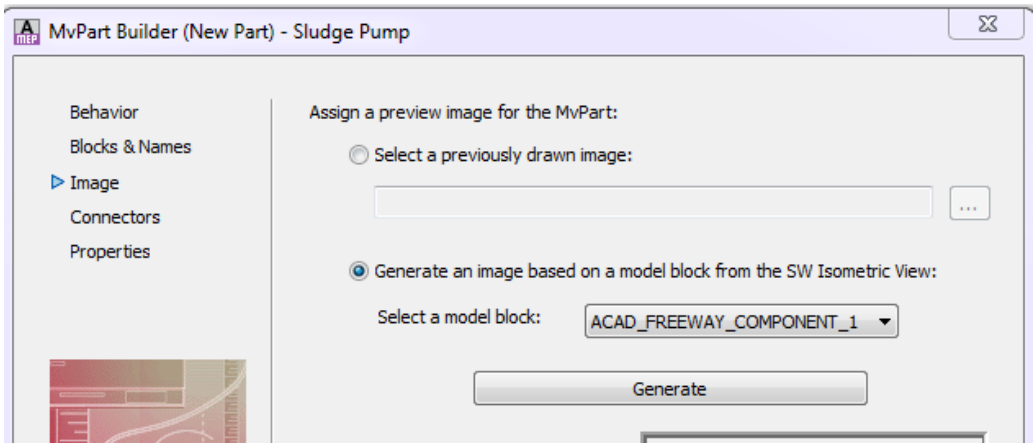
9. Select *Generate Blocks* at the bottom of the dialog:



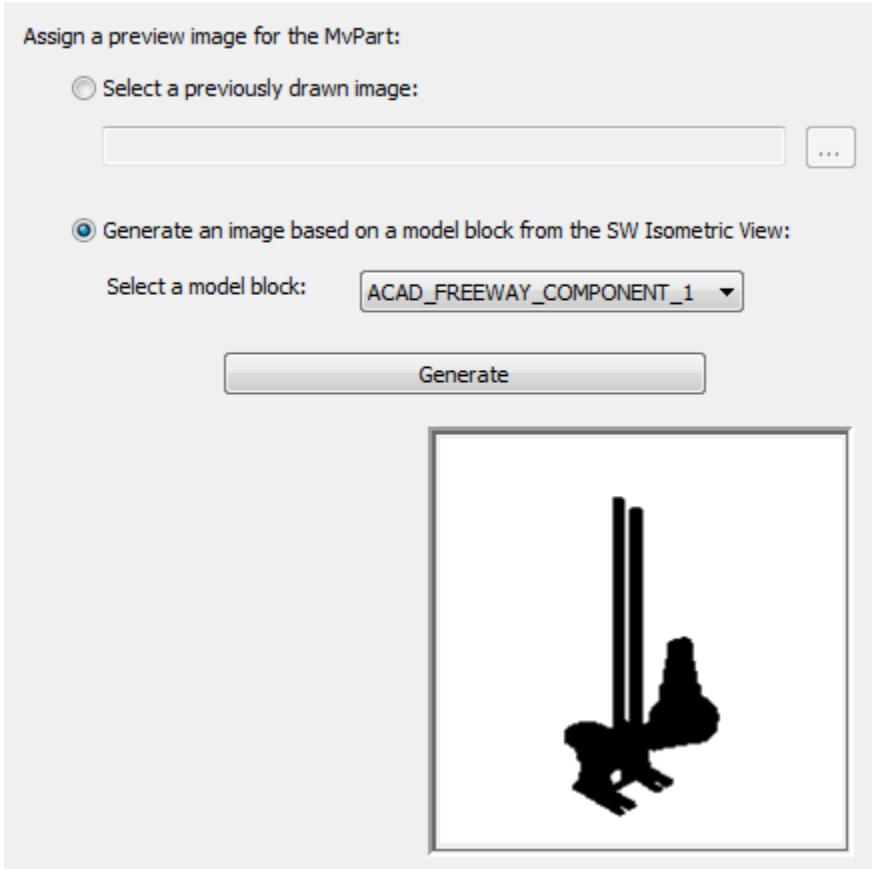
10. The *Views* dialog will appear, and list all of the blocks that have been defined. This tool also sets what blocks will appear, in specific directions, based on the type of view (single line or double line):



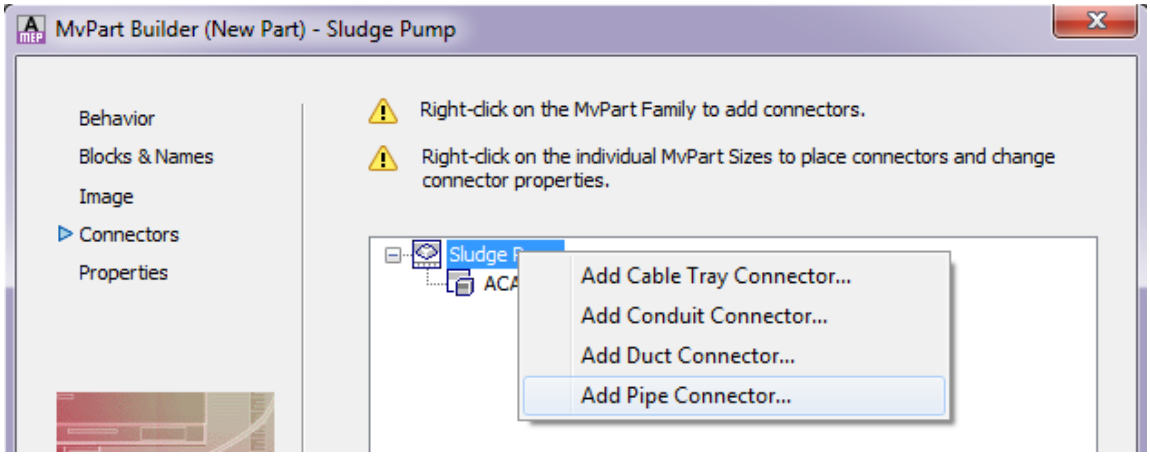
11. Select *OK*. Click *Next* to continue. From the *Image* page, select the "*Generate an Image…"* option. The model block will appear as the default, since it's the only one loaded in the drawing:



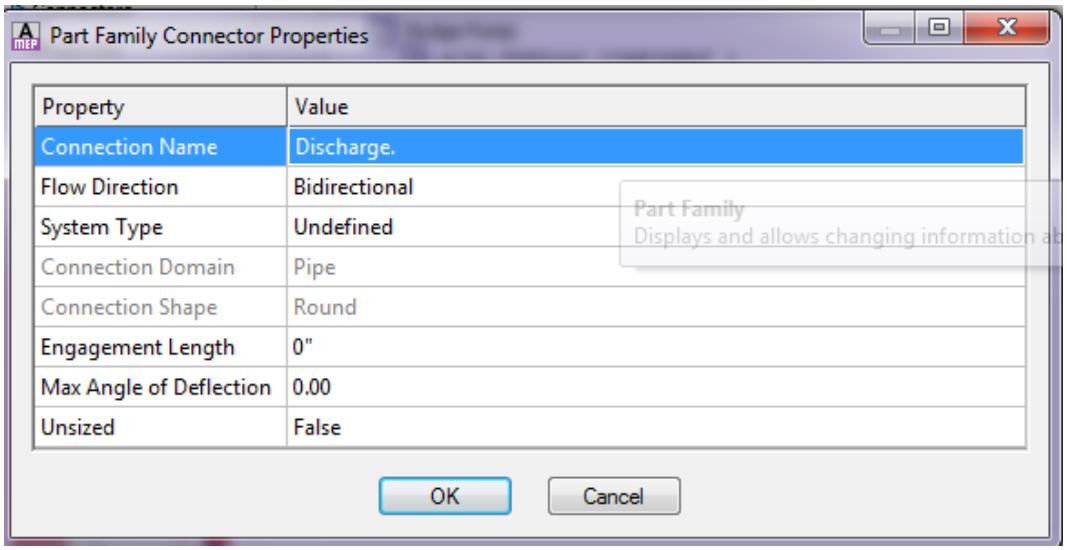
12. Select *Generate* to create the image, which honestly, is a little ugly. You can also use saved image files when using the “Previously drawn image” option which I prefer:



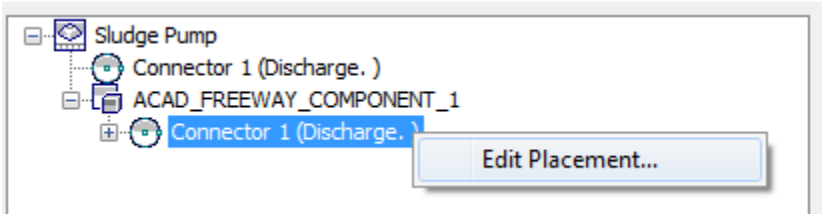
13. Select *Next*. On the *connectors* page, we’ll add a connection, for the pump discharge. Right click on the *Sludge Pump* type, and choose *Add Pipe Connector*:



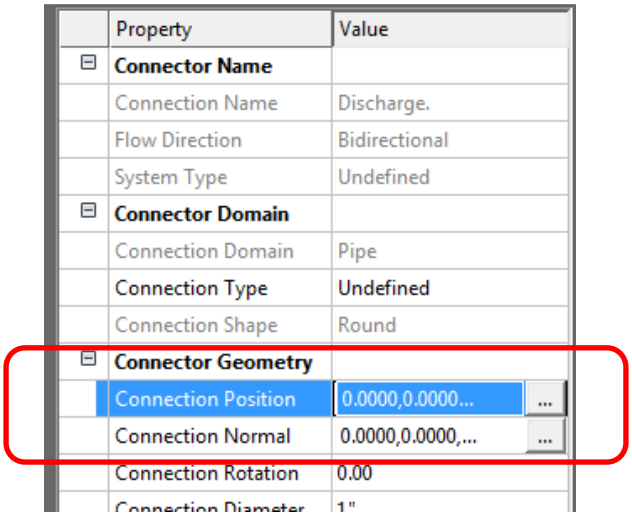
14. For the name, type in **Discharge**. Leave the remaining settings as-is.



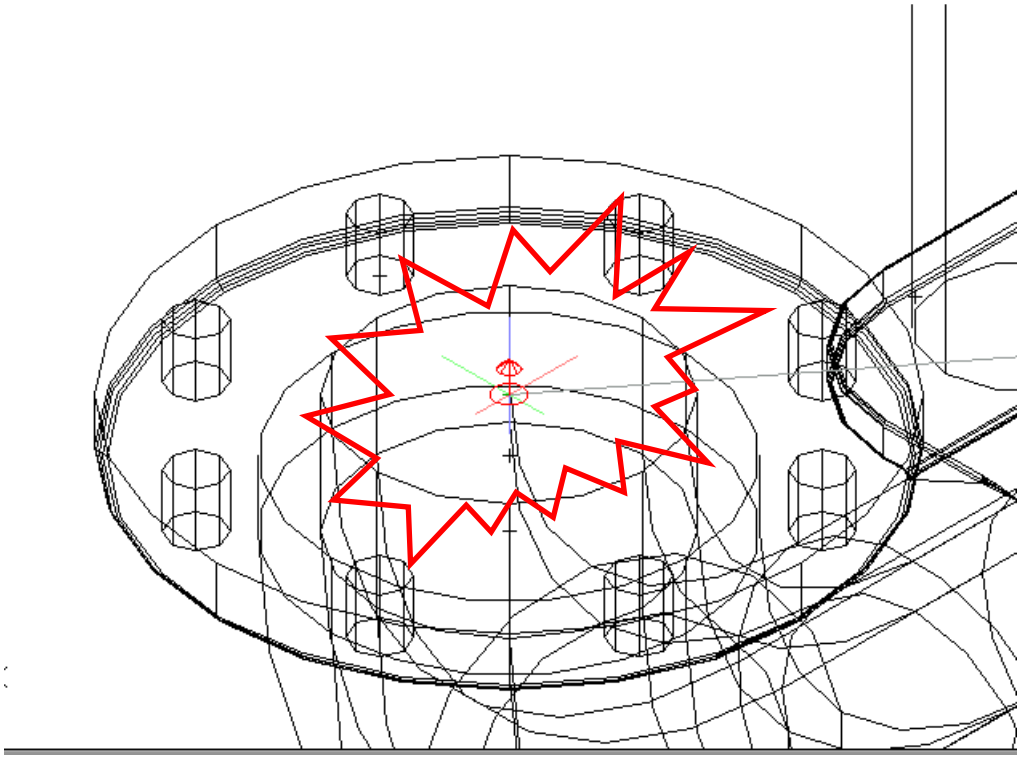
15. Select *OK*. Next, right click on *Connector 1 (Discharge)* and choose *Edit Placement*:



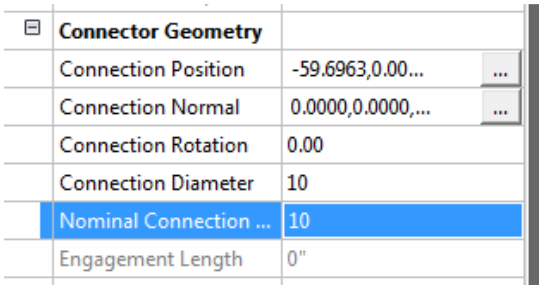
16. When the part builder tool opens, you see now why you start from a 3D view. It's easier to add connections from this point. Select the *connector position* placement tool:



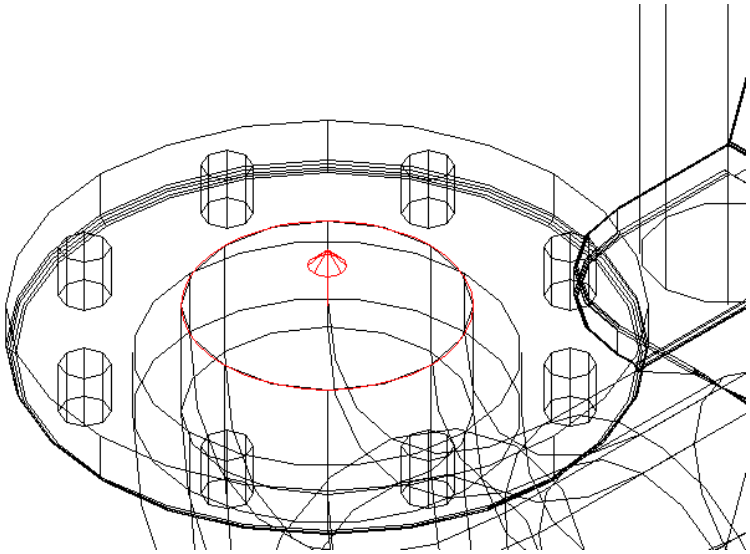
17. This lets you pick the placement point. Use the center snap to locate the center of the discharge connection:



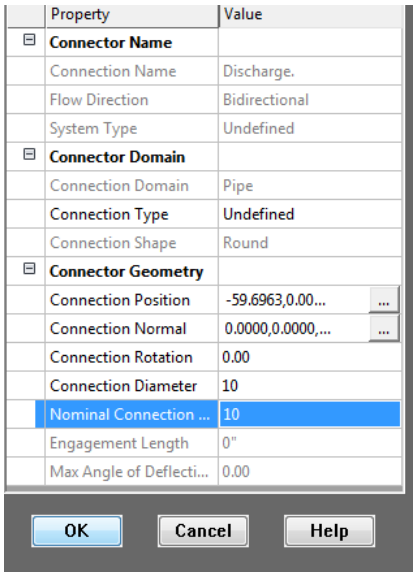
18. Once the connection is placed, the direction arrow is pointing up, since this is the default. You can change the direction by editing the *Normal* values, but since this is correct, we'll leave it as-is. Next, change the *connection diameter* and *nominal connection diameter* to **10"** (you don't need to add the inch mark):



19. The connector is now defined to match the size of the nominal pipe connection:

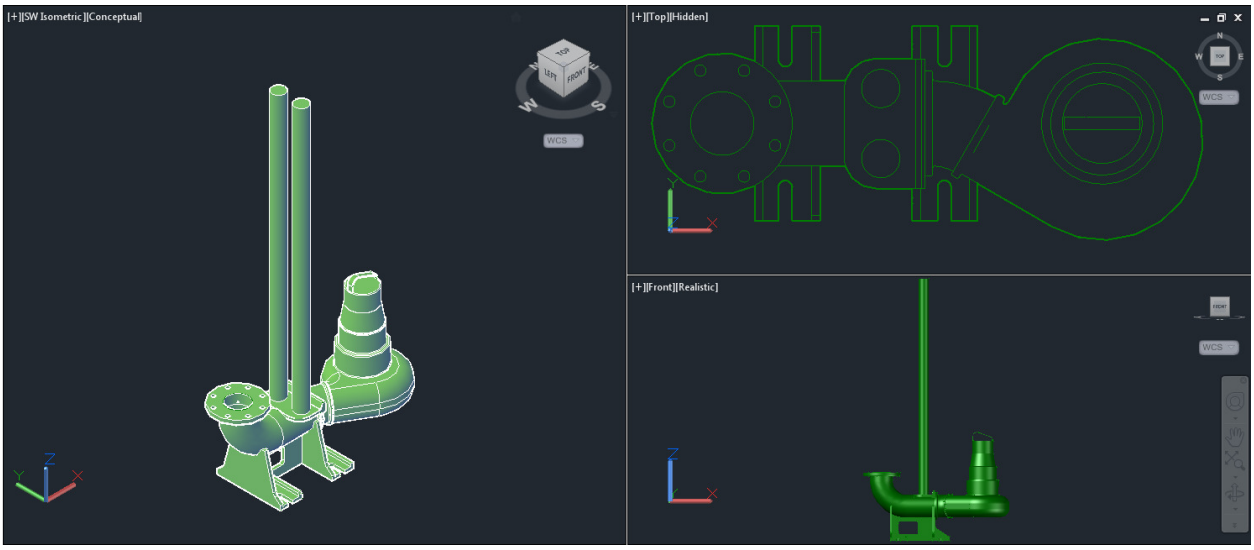


20. At the bottom of the property palette, select *OK* to save the changes and close the part builder.



21. Select *Next*. You can now edit any additional properties, or *Finish* the command. Properties can be edited at any time, but there's one important point – once you start down the part builder path, if you cancel out of the command, your steps are NOT saved. Work all the way through this, and edit the part later if needed.

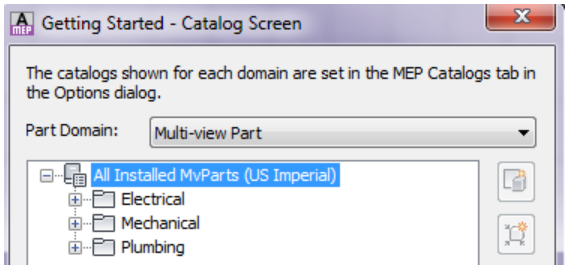
Congratulations! You've now added your part as a multi-view part, and can now place it in a drawing.



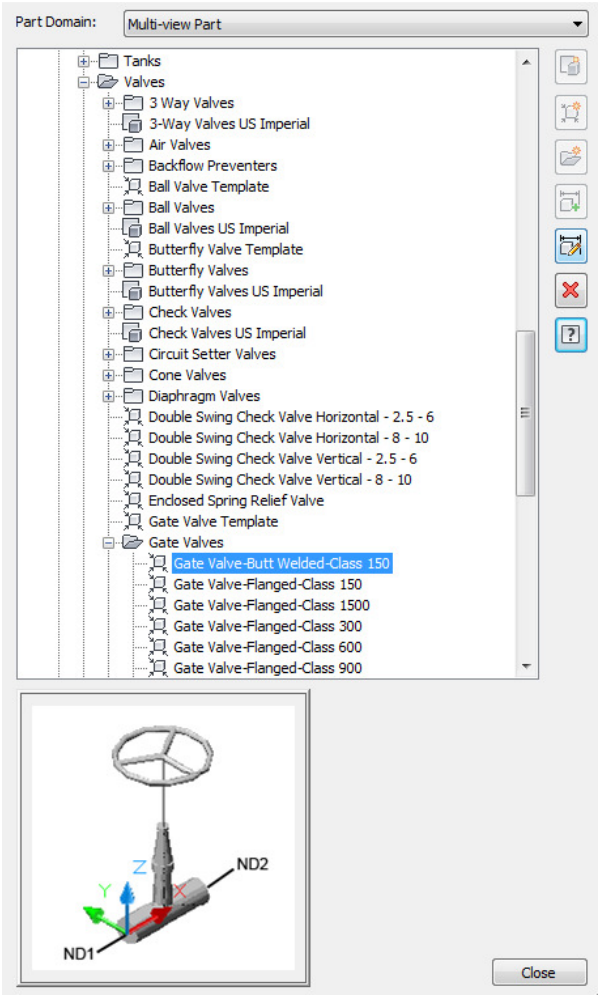
Adding and Editing Symbols and Annotation Planes

Working in 3D gives you the advantage of seeing your design from different viewpoints. IT also helps to create section and elevation views automatically. AutoCAD MEP 2013 includes a new feature that makes the engineering models provide clearer detail levels for specific parts. This feature is the ability to add front and left side working planes, and create custom 2D schematic symbols. Let's take a look at how this works.

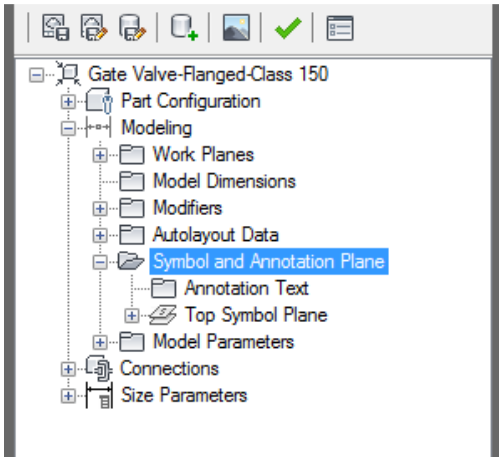
1. Start a new drawing from the *AECB Model* template.
2. On the ribbon, *Manage* Tab, *MEP Content* Panel, select the *Content Builder* tool.



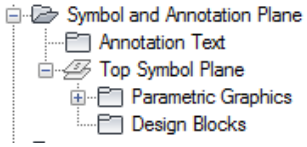
3. We're going to edit an existing valve, and add the graphic symbol to the front view of the model.
4. When the dialog opens, make sure the *Part Domain* is set to *Multi-View Part*. Under the *Mechanical Equipment* section, browse to valves; expand the section, and then select *Gate Valves*. One note: look for the symbol that has three arrows pointing to a box. This indicates that the part is a block-based part.



5. Select the *Gate Valve-Butt-Welded-Class 150* part. Click the *Modify Part Size* icon to continue.
6. The file will be opened, and the *Content Builder* dialog will appear. The default view is for a plan view, so let's take a look at how the current plan symbol is displayed. On the dialog, expand the *Modeling* section, and then expand the *Symbol and Annotation Plane* section.

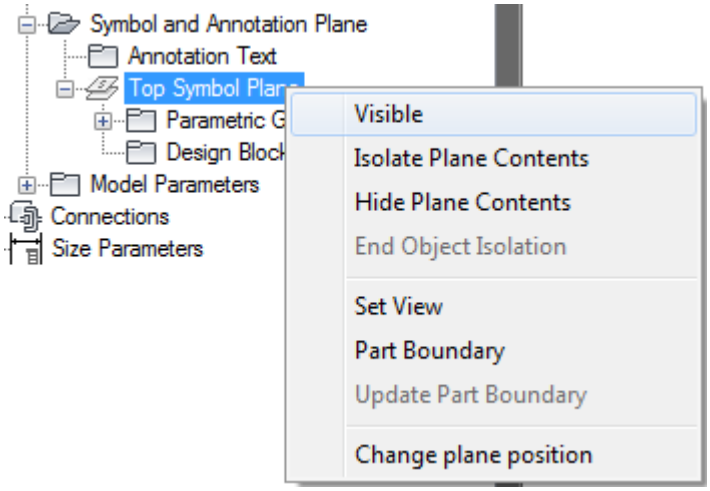


7. Expand the *Top Symbol Plane*. You'll see two categories:

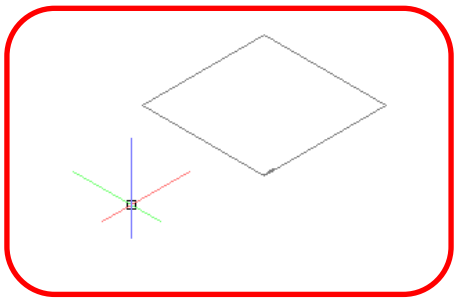


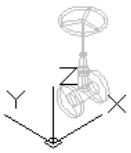
- *Parametric Graphics* allows you to create your own 2D symbol, and leverage the existing model to define your part.
- *Design Blocks* allows you to import a 2D Symbol block and lets it be used to represent the part in the plan, or in the front or left view planes.

8. To add our own symbol, the first important step is to turn on the workplane. Right-click on *Top Symbol Plane*, and click *Visible*.

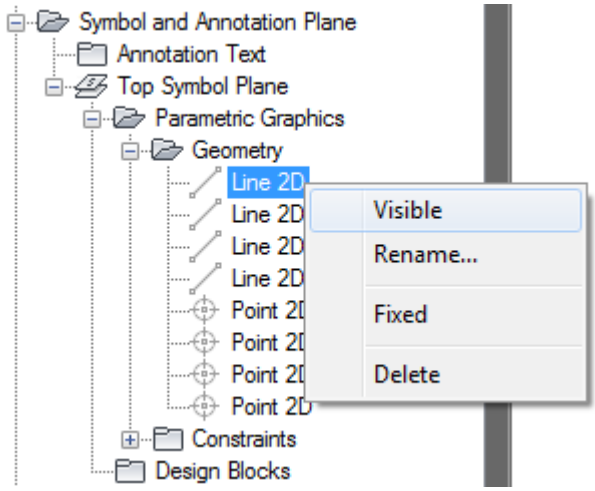


9. This will show the working plane. Use the view cube see where the plane is located in the model:

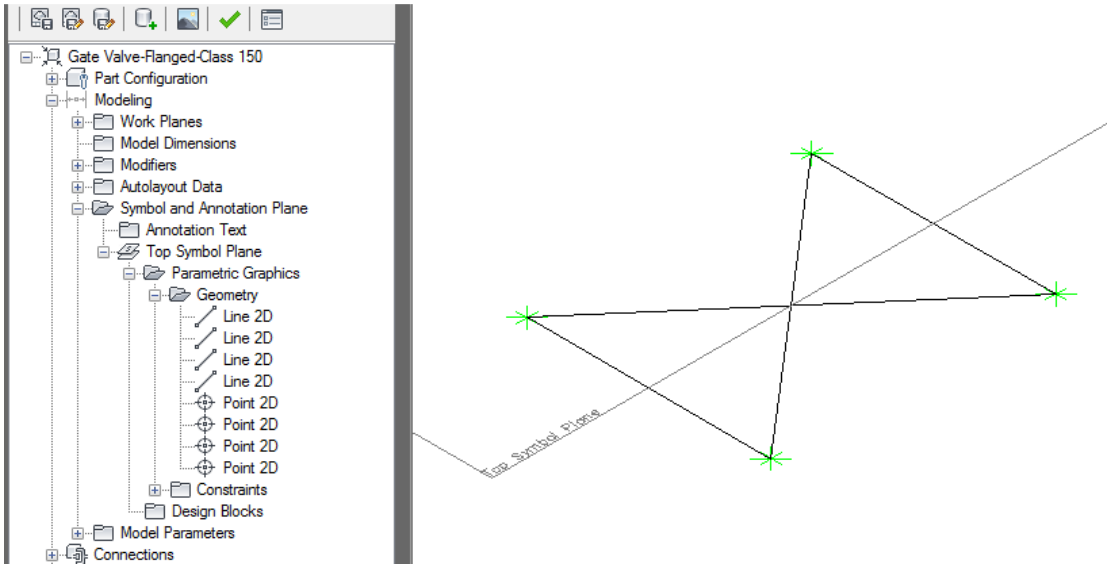




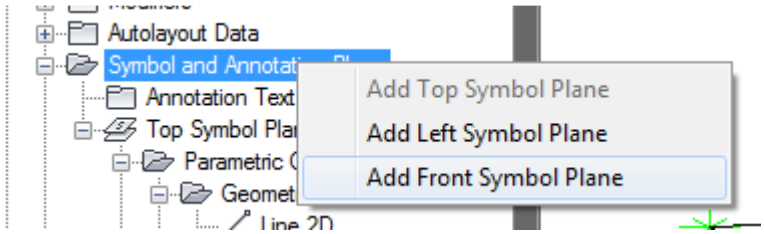
10. As an example, let's check out the existing 2D geometry. Expand the *Geometry* section and then right-click on the first *Line 2D*. Click *Visible* to turn this on.



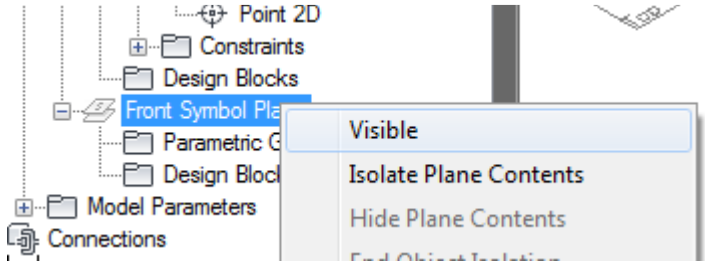
11. To see the rest of the parts, repeat the steps for all of the lines and points. Note how the icon becomes bold as the line or point is made visible:



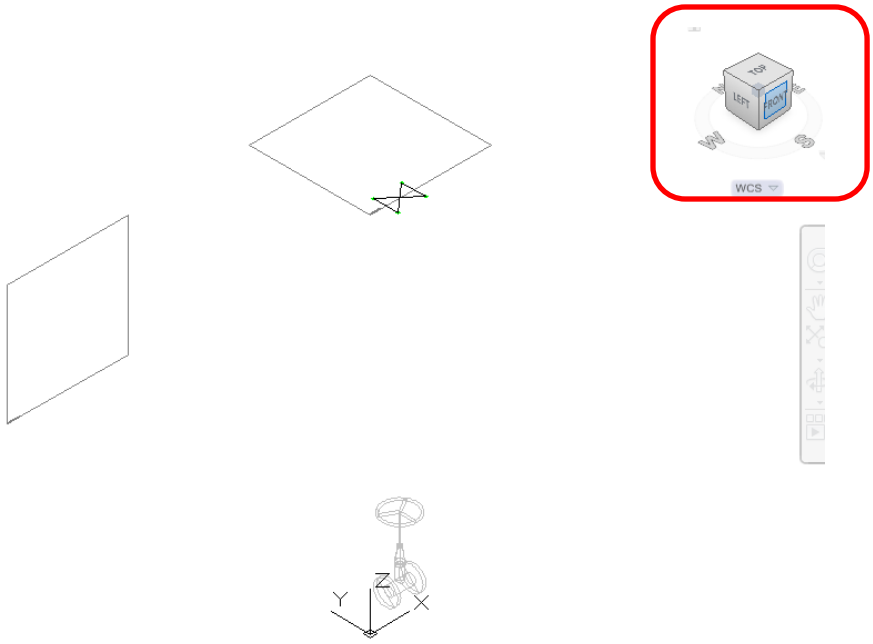
12. This displays the linework that is used when the view is set to the top plane, which is where plan views are displayed. Our next step would be to turn on the *Front Symbol plane* and use it to define a symbol for that point of view.
13. Right-click on the *Symbol and Annotation Plane*. Click *Add Front Symbol Plane:*



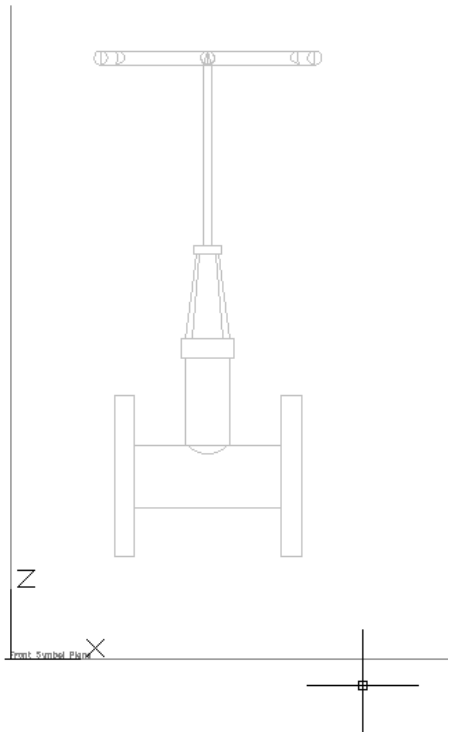
14. Once the plane is created, right-click on the new plane, and then click *Visible* to turn it on.



15. Once it's visible, you can change to the front view.

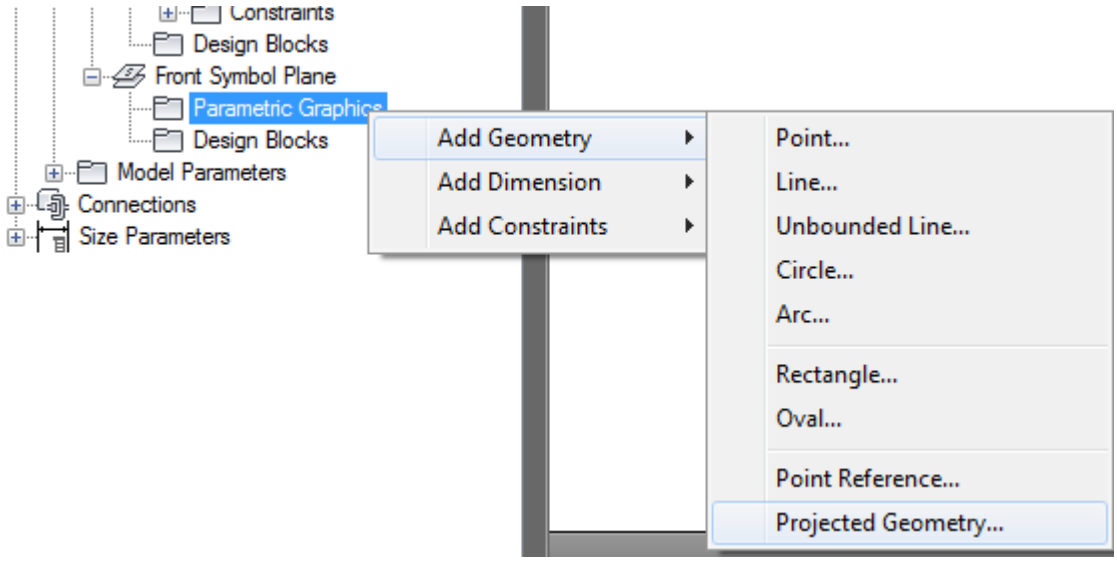


16. Zoom into the view so you can see what you're adding.

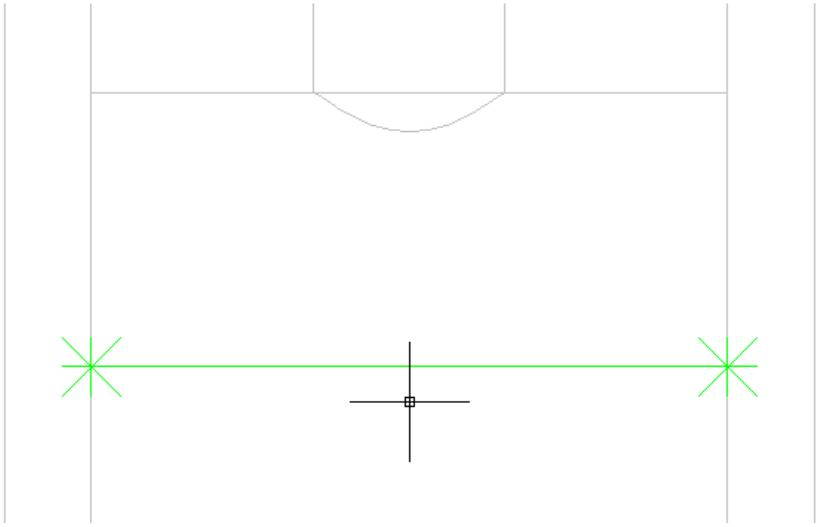


17. In order to link the geometry from the model to the symbol, you need to create *Coincident Points*. These points link the symbol lines you create to the size of the model. The quick way to create the reference points needed to link the new linework to the

model is to use the *Projected Geometry* tool. Right-click on *Parametric Graphics*, and then click *Add Geometry*. Once the section expands, select the *Project Geometry…* tool:

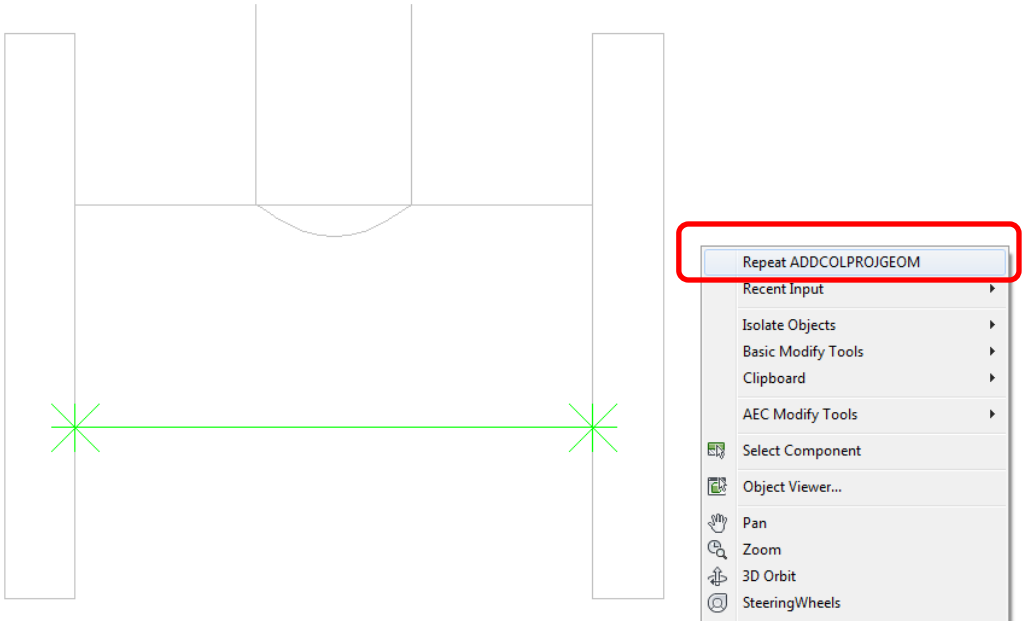


18. You'll be prompted to select a modifier – this is the body of the valve. Pick the shape; as you move your mouse around the body, you'll see a green preview line, which indicates which edge is being used to create the line and points. Select the bottom of the body:

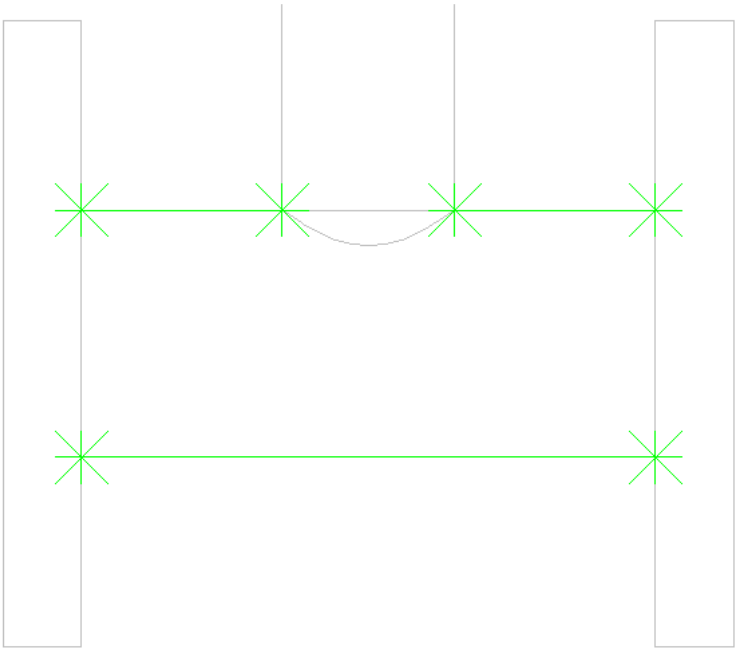


The line and points will be created.

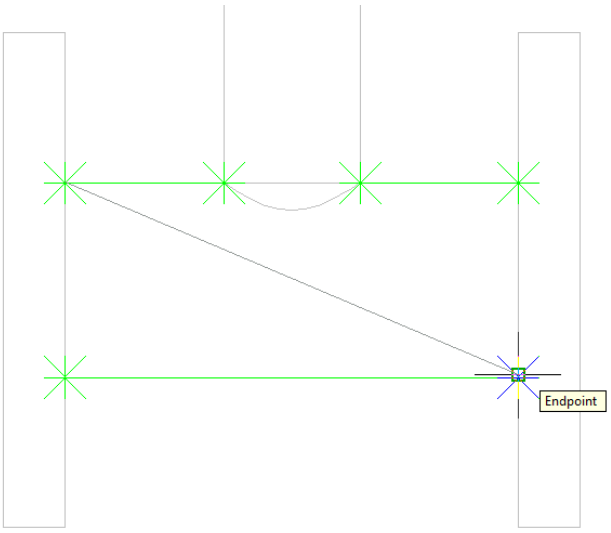
19. Repeat this tool by right-clicking in the view, and selecting the Repeat *ADDCOLPROJGEOM* tool.



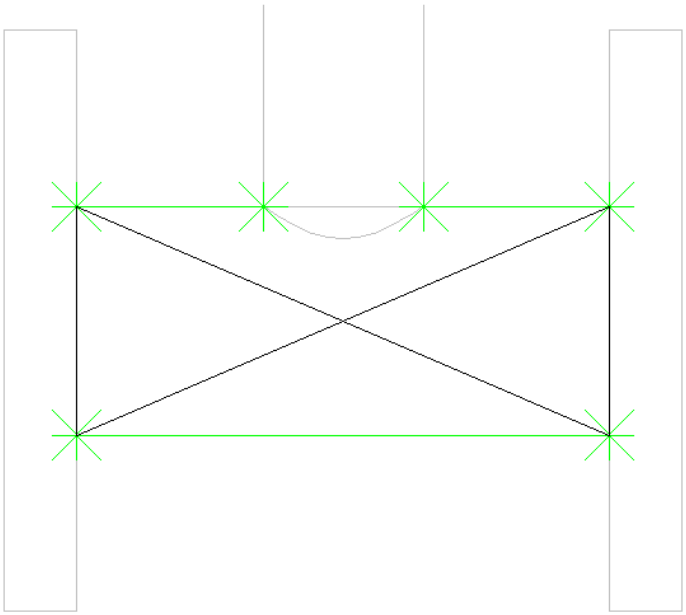
20. Select the two lines on either side of the top of the valve body, as shown here. Once you've added all three lines, you'll have the points you need; and they are automatically constrained to the 3D model.



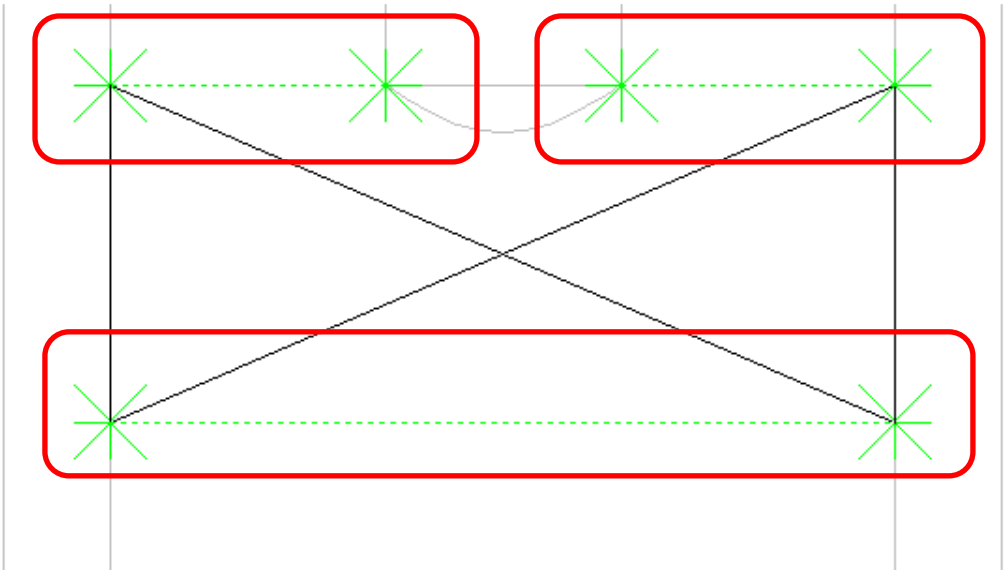
21. Next, right-click on *Parametric Graphics* under the *Front symbol Plane*, and then click *Add Geometry*. Select the *Line…* tool. Using the points as snap points, draw the four lines as shown to create the symbol.



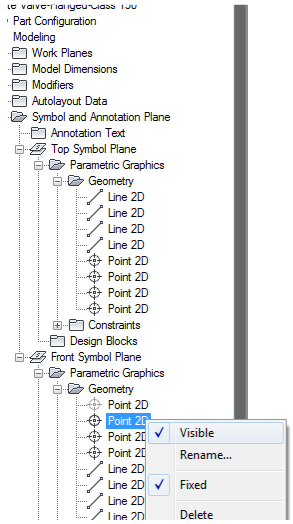
22. Make sure you have the Ortho tool turned off. The linework will be created similar to a polyline. When finished, press ENTER to complete the command.

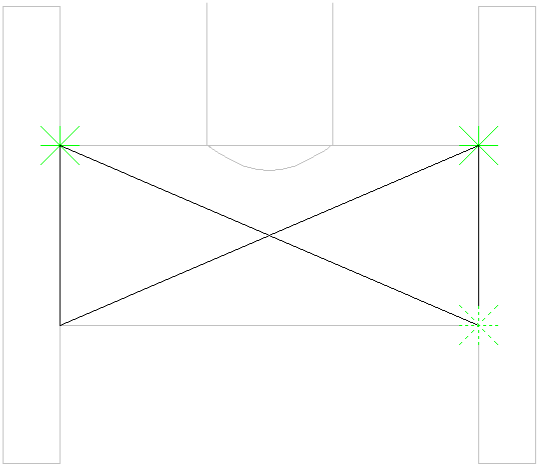


23. You can delete any lines or points that aren't needed to create the symbol, so delete the highlighted below. Pick each segment or point and then use the DEL key to remove them.

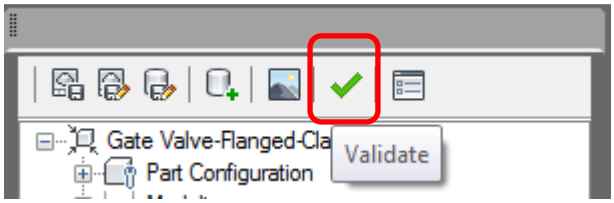


24. The remaining points that constrain the lines to the model can be turned off. By the way, don't delete these; the symbol will not work correctly if you remove them from the model. To hide them, expand the Geometry section and then right-click on each point. Deselect the Visible option, and the point will be turned off. Notice that these are fixed, which means they are properly constrained to the model.

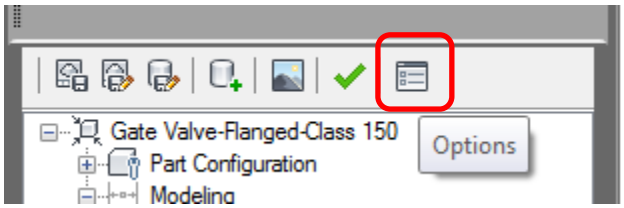




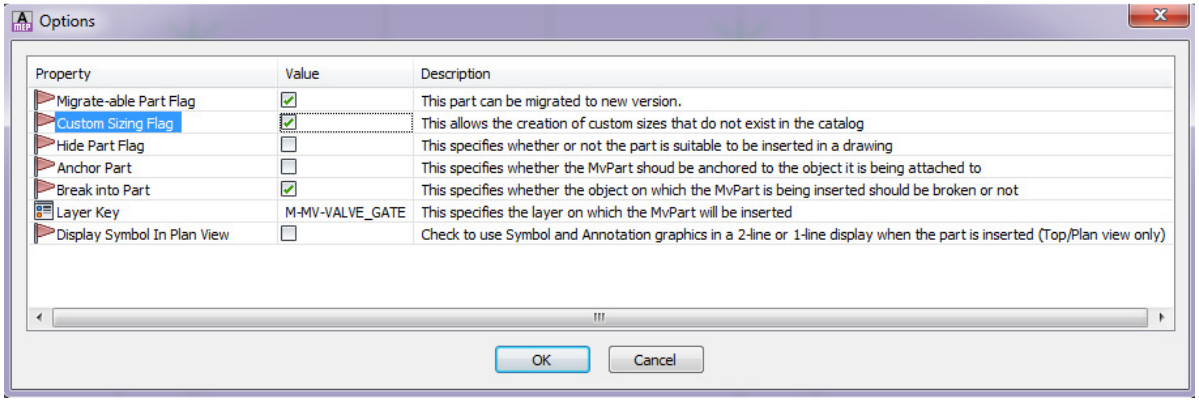
25. The symbol is ready to use, but you aren't finished yet. On the Content Builder dialog, select the *Validate* tool.



26. If there are any errors in the model, they will show up here. Make sure the check mark is green, and your model is correctly defined. Select the Options tool.



27. Make sure the following options are checked:



- Migrate-able Part Flag – so you can upgrade this to new releases;
- Custom Sizing Flag – so you can add custom sizes

A new feature in AutoCAD MEP lets you assign the layer key style from this dialog. Previously, you had to edit this setting with the *Catalog Editor*. Make sure the *Display symbol in Plan View* is deselected, so the new symbol does not appear from the top view.

You've finished the task now and properly defined the symbol. Close the *Content Builder* dialog, saving the changes when prompted. You can now load the valve into a drawing, and test it to make sure it works. Use this tool to get the construction documents looking the way you want!

Creating Custom Content Catalogs

Now that we've made some custom content, let's learn how to separate it from all the other riff-raff that comes with the program (just kidding, there's a ton of great content here).By keeping this content separate, the user will be able to receive new content from Autodesk without having to be concerned with their content being lost or overwritten. In the event a user creates a large number of parts, they may want to move their custom content to a separate location. In order to do this, the user catalog must contain at least one part, and the folder location must already exist. The MEP Catalog path will need to include this new path, and can only be added after the content is located in the new folder.

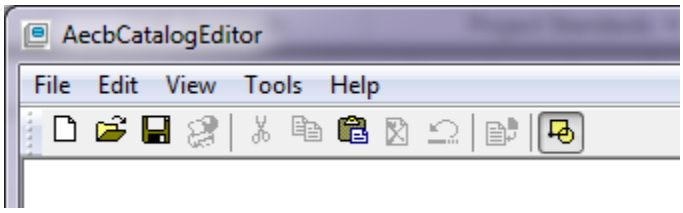
Note: this exercise should take five minutes to complete once the user has become proficient with the steps listed.

Before getting started, Autodesk includes these tips to make the creation of parts simpler:

1. Save a backup copy of the part catalogs before using Content Builder, in case you need to revert to the original catalogs provided with AutoCAD MEP. You can use a browser application, such as Windows® Explorer, to copy and paste the catalogs and their sub-folders to a new location.
2. Use the order of the folders in the part browser as a guide to the steps involved in the creation process.
3. From Windows Explorer in Windows 7, browse to the following folder (this is the default location as installed using the defaults for AutoCAD MEP):

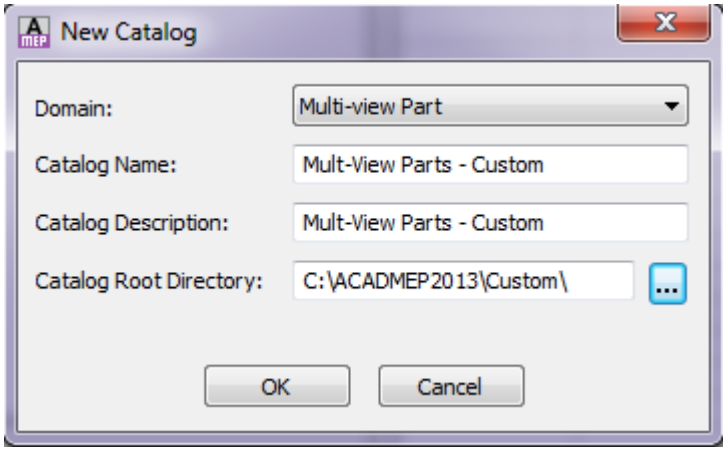
    - *C:\ProgramData\Autodesk\MEP 2013\enu\MEPContent\USI\MvParts*

4. To begin creating the part, a new folder should be created and used for the custom content. Personally, I'd use the default location above and just add the folder, but you can also locate it on a server or local hard drive. The user must have full permissions and administrative rights to the folder. For this class, create a new folder directly under the c:\ drive named *c:\ACADMEP2013\Custom*.
5. From the manage tab of the ribbon, select the *Catalog Editor*. In the Catalog editor dialog, select the *New* icon to create a new catalog:

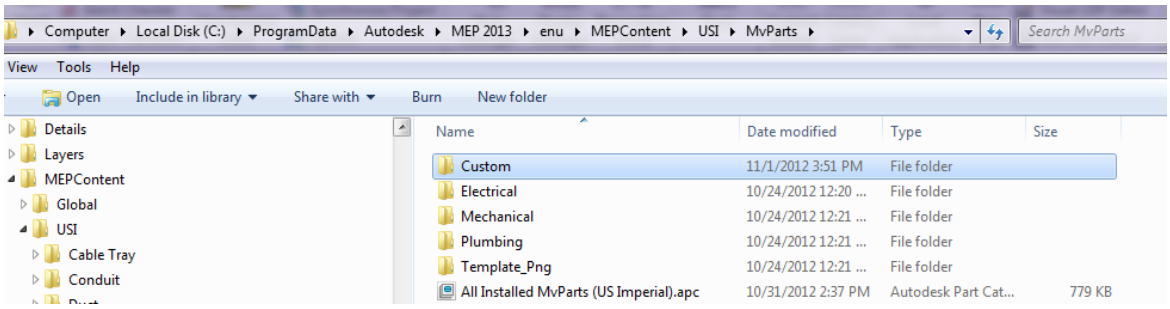


6. In the new catalog dialog, change the part Domain *to Multi-View Part*. Name the new

catalog **Multi-View Parts – Custom**. Match the description to the name, and change the catalog root directory to *C:\ACADMEP2013\Custom* (note: this folder must already exist on the system to be used):

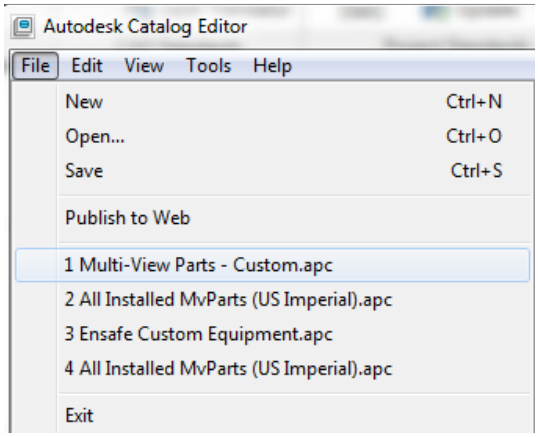


7. Select *OK* to continue and return to the catalog editor.
8. On the right panel, change the following values by double-clicking on the field:
   - *Supplier Name* – your company name;
   - *Supplier URL* – your company web address
   - *Supplier Image* – if you have an image file that can be used, select the image – make sure it's in a path that can be seen by all users.
9. Save the catalog.
10. Remain in the catalog editor. The next step is to copy the new part from its original location to the new catalog. Open the existing catalog where the new part is located (see the previous path locations for the default catalog paths):



11. To copy the new part out of the main folder, expand the MvParts catalog to the chapter where the custom pump was created. Right mouse click on the chapter and choose *Copy.*

12. To re-open the new catalog, choose the *File* pull down menu, and then select the new catalog:



13. Once the new catalog is open, right click on the catalog and choose *Paste*. The *pumps* chapter should appear. From the *Tools* menu, choose *Regenerate Catalog* tool to update the catalog and use the part. The user can now return to the original catalog using content builder to remove the custom part. To see the parts, add the catalog to the MvParts path located in the *Options* palette under *MEP Catalogs*.

Now you've setup a place to keep your custom content. Get busy – and set up as much as you need. Migrating to newer releases is a lot easier from now on!

Conclusion

Taking the time to get AutoCAD MEP setup and customized the way you want isn't that difficult once you learn the fast way to do things. Hopefully these tips will help you integrate the program more tightly into your design process. If you want to learn more about how AutoCAD MEP works, checkout the training videos I produced from 4D Technologies at www.CADLearning.com.

For more tips and trick, you can also check out to my blog, The MEP BIM Engineer, at http://mep-cad.blogspot.com.

Thanks for attending!