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AutoCAD® MEP Content Builder: Simplified Workflow for Engineers

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Whether you are an experienced AutoCAD MEP user or you recently learned to use AutoCAD MEP, when the time comes to create parts in your engineering designs, the workflow for creating parts can feel very daunting. AutoCAD MEP has a very powerful content creation tool called Content Builder, and understanding how to use it systematically can help you streamline the creation workflow to create flexible parts very quickly. This lecture class will describe the steps for planning out a basic 3D model to a final and functional part, covering concepts such as work planes, dimensions, and paths in AutoCAD MEP 2011. Join us to learn how easy it can be to use Content Builder tools that you already have at your fingertips!

About the Speaker:
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AutoCAD® MEP Content Builder: Simplified Workflow for Engineers

Today I'm going to share what I know about content builder. It has been a blessing to have such great friends and co-workers that have been willing to teach and pass on their knowledge. And so i would like to pass that information forward, with one request that you also pass it along. Thank you Gregg Stanley, Gerry Huot, Martin Schmid, Daryl McClelland, Bob Bell and all my co-workers.

There is something else that should be shared, and that has been my experience at Autodesk. Let's pull back the veil a bit. Autodesk, like everything in life is not perfect, but I honestly believe we strive to be. We strive to be a team at all levels and it's a team I'm proud to be a part of. I'm a better person for my experience here at Autodesk. It is simply amazing what we as a company set our sights on, and as a team make happen. We want to share that experience with you, and coming from a customer support perspective, we want you to consider us as part of your extended team.

Feel all warm inside? Ha ha, maybe. Well, the good intention is there. This gives some insight into who I am and what I'm about. Let's get started and learn!

Updates and Hot Fixes:

AutoCAD MEP has a Web Update and several hot fixes posted on www.Autodesk.com please remember to install all service packs and hotfixes.

Subscription Extensions

AutoCAD MEP also has Subscription Extensions available for Subscription customers.

- Piping Content Extension
- Duct Transition Utility
- AutoCAD Architecture Productivity Extension
- AutoCAD Architecture Renovations Extension

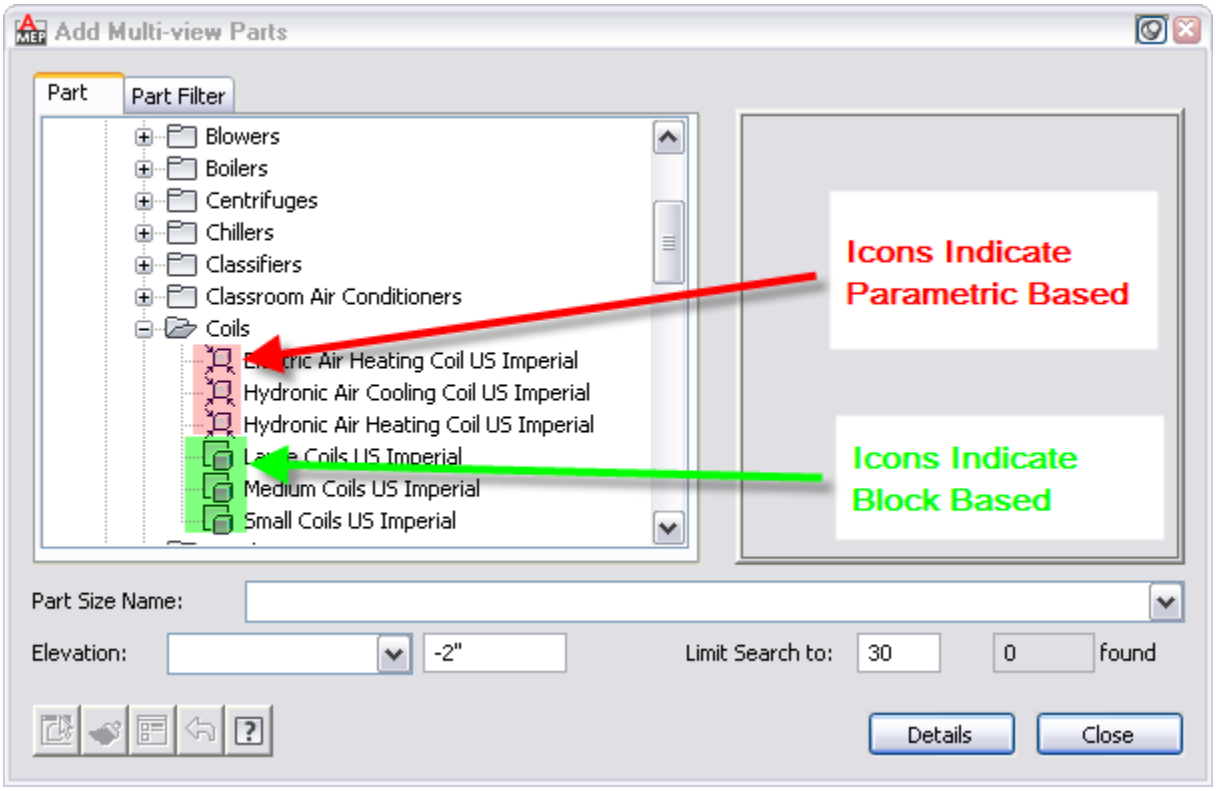
Additional Resources

The book "Mastering AutoCAD MEP 2010" from Paul Aubin, Darryl McClelland, Martin Schmid and Gregg Stanley is now available at www.paulaubin.com and book stores everywhere.

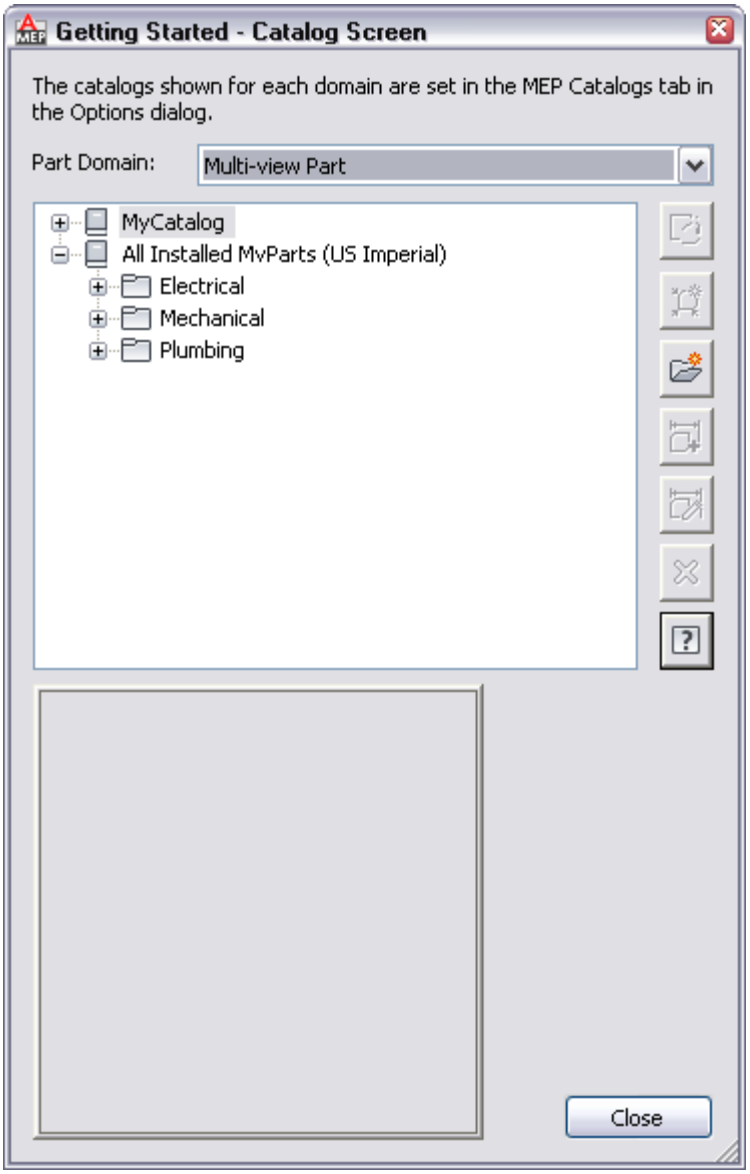
Content Builder Is Easy!

You just need to know the rules of the road. If you have never seen an automobile, would you feel comfortable driving it? No. But let someone explain it to you; give yourself some experience and eventually you find that it becomes second nature. I'm going to repeat this throughout this whitepaper. Tell yourself this over and over and believe it.

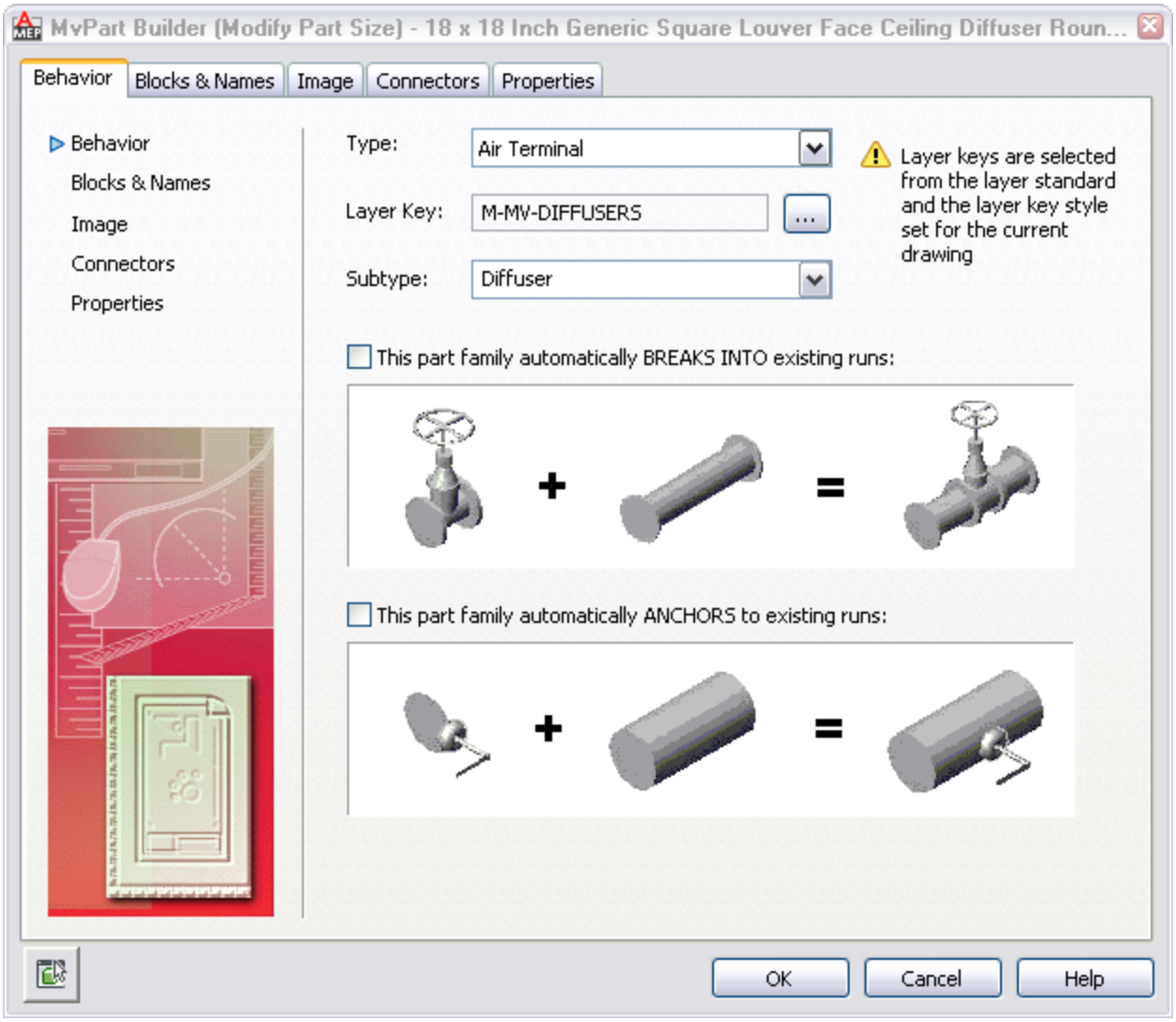
Content Builder, What is it?



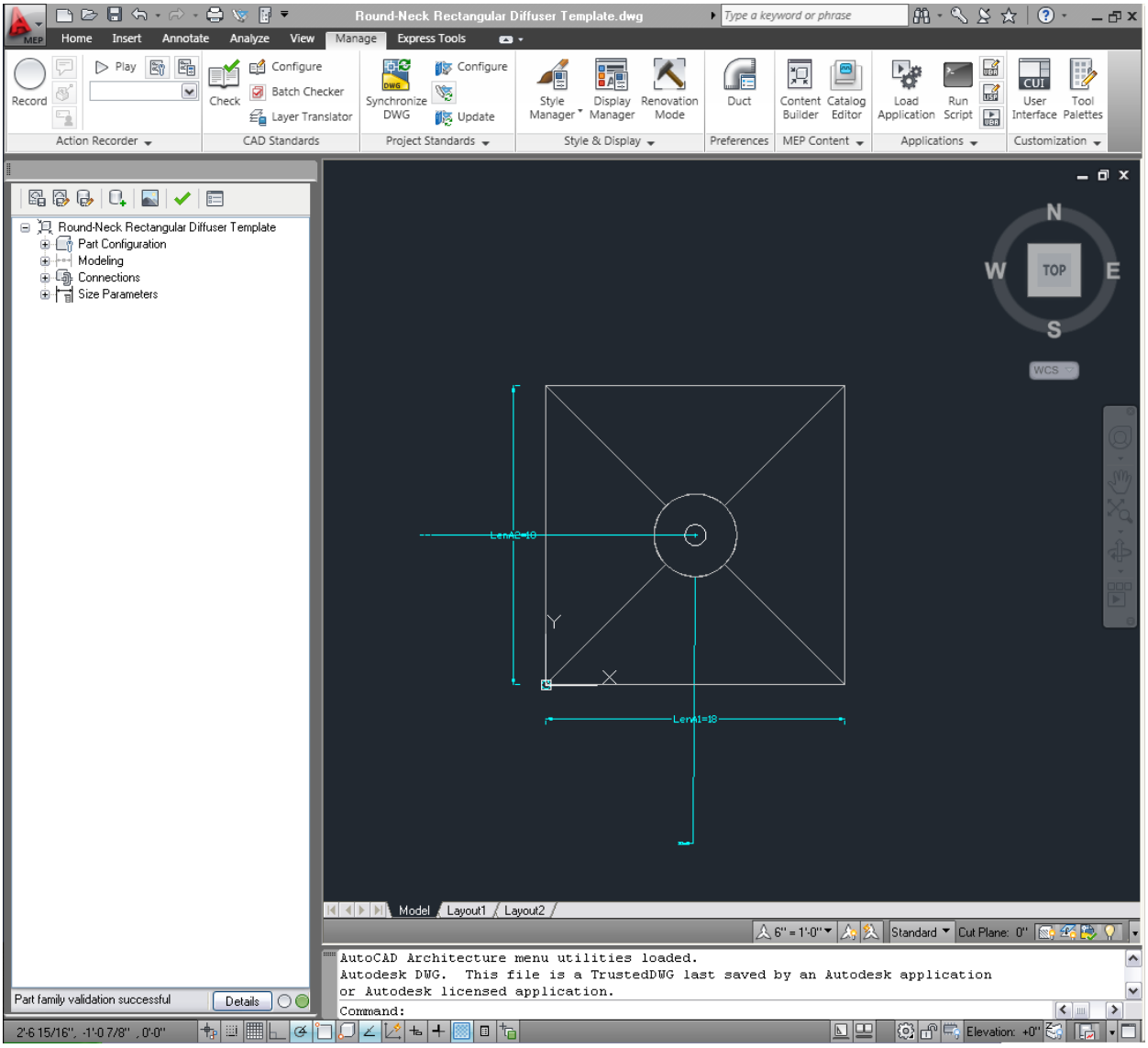
Content Builder makes both block-based and parametric-based mvparts and parametric fittings. We are going to focus on parametric based parts and fittings, since it is the more complex and least understood side of content builder. The program within content builder that allows us to create parametric parts is often referred as parametric part builder.



Content Builder – Initial Screen



Content Builder - Block Based Wizard



Content Builder - Parametric Part Builder

For all intents and purposes, parametric part builder is content builder and the words are synonymous. Content builder is simpler so we will continue using that terminology.

What is it not? It is not AutoCAD and not AutoCAD MEP. You should consider it a separate program that launches from within AutoCAD MEP. Lines, arcs, points, dimensions, etc. Found in content builder have no association with the objects found in AutoCAD. In fact, in content builder, the interface may say “line” but at the command line, it says AddColLine. Every command in content builder, may feel similar to commands familiar in AutoCAD. You must resist the association.

Why? Because most of ribbon commands are still available, though should not be used. Do not ever use move, copy, paste, array, fillet, chamfer, etc. in content builder. Do not use the undo! It breaks the instruction set that you are spelling out for the computer! AutoCAD commands are used "to achieve some graphics intent." Content builder commands are used "to instruct the computer how to fabricate a part."

What is parametric content builder? Its purpose is to fabricate a flexible part that you can insert using any dimension specified. What you create with content builder, is a set of instructions. The concept is similar to how software developers write computer code in a language. The language is something that humans can grasp and elaborate on how software is to behave. The language that software developers write in, is not the language that computers think in. A compiler is a program that acts to translate programmer language into computer language. Content builder, is a very simple program that allows us to draw lines, arcs, dimensions, etc. It forms an instruction set that the program uses to interpret "your intent into a 3d part."

Think of a set of instructions to cook a meal.

Step 1: Identify what you want to bake

Step 2: Identify ingredients and pots pans

Step 3: Try to get wife or mom to bake for you. (just seeing if you are awake).

Step 4: Buy missing ingredients (but only after trying step 3!)

Step 5: Follow a sequence to cook

Step 6: Taste test

Baking an mvpart is not too different. The steps look like:

Step 1: Identify the part desired

Step 2: Identify the shapes, the dimensions, the connector locations

Step 3: See if part exists in the catalog; on Autodesk seek website, on Augi, on Google, etc.

Step 4: Find manufacturer's cut sheet with dimensions and maybe options/accessories

Step 5: Use content builder, follow a sequence to specify intent

Step 6: Test test test

The tools used in content builder are sequential. What you draw first and second affects things down the line. So like a brick foundation, the first bricks laid form the base for the rest of the bricks. Pull out the first or bottom brick and the rest fall. We will get into that later. The take-away here is that you are making an instruction set for the computer to follow. You are not making graphics for the computer to mimic.

Content Builder Is Easy!

You just need to know the rules of the road. If you have never seen an automobile, would you feel comfortable driving it? No. But let someone explain it to you; give yourself some experience and eventually you find that it becomes second nature.

Content Builder Strategy

Now we know that content builder is a separate program and it should be thought of and treated that way. I'm going to say this message several times and in several ways. Do not associate the familiar lines, arcs, circles and commands from AutoCAD, with Content Builder. You cannot use them inside of Content Builder, and Content Builder has its own special versions of these objects. Content Builder does not stop you from using most AutoCAD commands, though it probably should. They are ignored and not part of the Content Builder workflow or tool set. Prior to launching Content Builder, in model space you can use AutoCAD lines, Arcs, commands to visualize and plan your part. Just don't bring them into Content Builder.

My personal preference is to use AutoCAD or AutoCAD MEP to plan your model, prior to opening content builder. You must realize, though, that any lines, arcs, dimensions that you draw in AutoCAD model space, are not transferrable into content builder. Content builder uses different objects, so do not attempt to cut copy or paste any work from outside content builder into content builder.

Plan out the part using model space prior to content builder. It may sound like extra work, and it is but can also be a resource. I think of it as a digital version of the manufacturer's cut sheet that we are using for the part. On a print out of the cut sheet, we could use a highlighter and a pen to identify or draw necessary dimensions, points, lines, etc. We could do that on paper. Paper is 2d. Sometimes we need to plan our parts in 3d. In AutoCAD model space, we have the added benefit that we can use move, copy, rotate. We can add solids, points, lines and dimensions. We can also orbit around our part in 3d as we plan.

Let's face it, we know classic AutoCAD and you would not be reading this if you weren't learning content builder. So use tools you are familiar with to plan the model.

Example:

In preparation for this AU class, I built a model of a rectangular duct elbow with straight legs that were editable in length. We will dig into more detail later, but I'll summarize for purposes of this example. This elbow could handle any angle 0 to 90 degrees. When I changed the angle dimension, it did not retain the shape of the elbow. The center point of the curve kept moving in unexpected ways. Content builder can get cluttered with lines, points, and dimensions. I needed to step outside of content builder. I fell back to my model in model space, which allowed me to visualize what was happening in content builder. I planned and tried some different approaches. Ultimately, I realized that we needed to anchor the center point for the curve of the elbow, and allow dimensions and lines to rotate about that point. We will see this as we dig into how to build this elbow part.

How should we plan our part in model space? We have a chick and egg situation here. To be able to plan, you need to know how content builder works, and… so what we will do is teach a few concepts needed before going on. Feel free to skip this next section down to "Launch Content Builder," and come back to it when you are more comfortable with the workings of Content Builder. We will be discussing some advanced topics that require some familiarity with the program already. Come back when you are comfortable.

Content builder has the ability to generate a 3D body, using three methods or modifiers. These are referred to, inside content builder, as modifiers. The three Modifier methods are: Add Path, Add Extrusion, and Add Transition.

Let's talk about "Add Path." Similar to classic AutoCAD extrusions, content builder has this concept of a path and profile. You create a 2D profile, like a rectangle or circle or something more complex. You create a path on a flat 2D plane. A plane is essentially a flat surface that you can draw 2D shapes onto. An example of a plane is the x-y ucs plane. The x-y plane goes on forever in the x and y directions. It is located at the 0 z elevation. It's a flat surface. If you draw a line or series of lines and arc's on that surface, it makes a path. When that profile is extruded along that path, it makes a 3D shape. Paths and profiles are used most often when making fittings, like elbows, transitions and tees.

Let's talk about "Add Extrusion." We mentioned that content builder has planes that you can define, that allow you to draw lines and profiles onto. Instead of taking a profile and extruding along a path tied to that plane, an extrusion is outwards from that plane, but some specified distance, or to another plane. Your 2D profile is drawn on the plane. Add extrusion modifier just says to extrude that profile straight out from the plane by some distance, or to another plane. To make a rectangular rooftop unit, we may draw a rectangle for a profile and then extrude it some distance out from the plane. There is one drawback to this method. The distance out from the plane does not generate a model parameter. We cannot change that dimension once it is made. So you make an offset plane, which does generate a dimension in the model parameters.

Let's talk about "Add Transition." The transition modifier is not to be confused with a duct transition. We know we can make planes. We can make two planes that are parallel, separated by some distance, and those are called offset planes. Content builder generates a dimension between offset planes that we can edit. We have control over the distance between the two planes. We can draw a profile on both planes of any shape and location along that plane. The add transition modifier generates a shape from one profile to profile, between the two planes. It is a better modifier than the add extrusion.

Now you know a bit more about the three methods of generating a 3D body in content builder. It's enough to know what I'm referring to in some of the steps about planning prior to going into content builder. These steps will have more meaning after we make a part in content builder and you can reflect on them. For now, they make good reference material. Let's get back to our question. How should we plan our part in model space?

1. Content builder makes equipment (mvparts) and fittings for duct, pipe, conduit, cable tray. Identify what type of part it is.
2. Is it going to be a block-based or parametric mvpart, or a parametric fitting? FYI: Fittings can be made block-based but content builder forces it to be a mvpart instead of a fitting. Fittings are inserted automatically as we draw duct or pipe and must be parametric. Sometimes we need a complex shaped fitting. Maybe we don't have time or the geometry is too complex to be made parametrically. You may be very familiar with AutoCAD solids, and can make any shape imaginable. You can use AutoCAD solids to make a block based mvpart. So if you don't mind inserting a fitting manually, you can create it as block based and just know that it will be added as an mvpart one at a time. In summary, step 2 is about planning if you are going to make your part as block or parametric based.
   What is the difference between block and parametric based? A block based part starts out as an AutoCAD solid. You use the solids editing tools to carve it into the shape you need. Then convert your solid(s) into a block. Then use Content Builder to convert the block into a block-based part that has context and connectors. The shape of the part cannot be changed. Whatever dimensions, your solid was when you converted it is what you get. You can make other sizes by making more solids of the size needed. Add those sizes to the first block-based mvpart. A parametric based mvpart, has dimensions tied to an internal database. When you want a new size, you change the dimension and the part stretches. Parametric parts can be intimidating to make, but like learning to drive a car it becomes familiar and second nature. Parametric parts can be made with complex geometries, but do you want to? Not with the current state of the technology. The more complex a parametric part, the slower they make your project file. Add a handful of very complex parametric parts, and watch your project come to a crawl. That's the nature of parametric "rule" driven parts. Parametric parts are great for simpler shaped parts. A diffuser or roof top unit would be a fine parametric part. A screw chiller or a watertube boiler, maybe not. You would have to test your part in a project file to see how it affects performance. If performance becomes an issue, make it as a block based

mvpart. AutoCAD solids have made great strides over the past decade in performance. You can make a highly complex solid and your project file will still have a decent performance.

3. Start by making a solid 3D body of the final part. Make one size. You do not need to make every size. We just need something to help us visualize the part and plan.
4. Looking at the 3D solid, identify which modifier we need. Add path, add extrusion or add transition. Do we need a combination? Add path is the simplest, so use that as much as possible.
5. Plan your modifier:
    a. Add path: What is the path(s)? What is the profile? Is the path 3D, or can be drawn on single plane? A 3D path is a path on one plane that ends where another path starts on another plane. At the center of the 3D body, draw a path and imagine a profile skirting that path. Our elbow example has a straight segment, an arc, and a straight segment.
    b. Add transition: What are the two profiles? Each profile has to be on a plane at either end. Where are the two ends where the planes will be located? Transition modifiers cannot follow a curve between planes. You get a straight shot, from one profile on one plane, to the opposite profile. Planes must be parallel. If they are not, you will get unexpected results. Is it a straight shot acceptable, or do you need a more complex path between profiles? If so, transition modifier is not your choice. Use add path.
    c. Add extrusion: Is one profile acceptable? Are you extruding in one direction or both directions out from the plane? Are you extruding to an offset plane? What is the extrusion distance?
6. You now have a feel for how many planes, profiles and paths will be needed. Your part is going to be parametric, so how is it going to grow and shrink between part sizes? What dimensions are necessary? An elbow may be a 90 degree or 45 degree or any angle in between. As it bends to support the shape, what is the center point of the bend? If you extrude along a path and the path bends, what is the center point of that bend? If you are extruding or transitioning from a profile to a plane or another profile, what is the location of the profiles on their respective planes?
7. The same lines we use to make paths can also be used as construction lines. We can dimension them, allowing them to grow and shrink. These can push the profiles around to get them into position between different sizes of your part. Example: Rooftop unit has supply and return connections. These connections are located in different positions from one size of rooftop unit to another. You may need a construction line from a fixed point, out to the location of the profiles for the connections. You need a plane and a line on that plane, to be able to control the dimension. In house, we refer to the construction lines as the frame. You know the big caterpillar tractors with scoops and arms. The frame is like the arm. Once we dimension the frame, we can edit those dimension values. The frame lines shrink and stretch with the dimension value. So the dimensions

are like the hydraulic rams that move the arm. Well, your arm has to move about a pivot point and that would generally be an endpoint of a line or center point of an arc.

8. Newton, 1st law: A body in motion tends to stay in motion unless acted upon by an outside force. Newton, 3rd law: The forces between two bodies are equal and opposite. What does this have to do with the price of beans? Well, when we are flexing a part by changing the value of their dimensions, which direction should the part as a body move? Example: We have nothing but a line. We change the value of the length of that line. Shall it grow to the left, right or a little in both directions? Every line inside content builder has a point at either end. You can edit that point and declare it to be fixed. With a fixed point at one end, content builder knows that if the line grows in length, it holds that point fixed in position. If you declare both end points to be fixed, that line can still grow. It's going to hold the first point and move the second one declared to be fixed. I.e. It breaks its own rules in the opposite order of creation. That line with two fixed points is still fixed in angle. So if we have a line angling off the fixed line, and change the dimension angle between lines, the unfixed line moves or rotates. So step 7 is more about figuring out what points need to be declared fixed. Our origin point for the whole part is probably a good candidate. For fittings, the point at which we have the first connector should be fixed.
9. Sequence. We just gave an example of how content builder will break its own rules in the opposite order of creation. With parametric parts, it's possible to overload it with too many dimensions. A rectangle. How many dimensions do you need to control the shape of that part? Four? Three? Two? One? You need only two dimensions. Once you add a third, content builder is going to ignore it in favor of the first two dimensions added. First come, first serve. If you flex the third dimension, the rectangle will not change shape. The dimension driving the rectangle is one of the first two dimensions. Now, you can add the third or fourth dimension, and set it to always equal one of the first two. The first two are in control anyway, and the third and fourth become informational only.

10. We have most of the body and dimensions planned. You may want to plan out which dimensions are going to be visible and editable to the user. You don't want to overload the user. They should not have to pull out a manufacturers cut sheet every time they insert a new part. But you do want to give them some flexibility in case they need to edit a value.
11. Incidentals. There is a means to attach property sets and hyperlinks. The hyperlinks will not be live, but available to the user to copy and paste into a web browser. So you may want to create the property sets in style manager ahead of time or have that data handy.

With some concept of a plan in place, we can launch content builder and make our part. The creation of the part will take a fraction of the time than if you had not planned it out.

If this is your first time creating a new part, create it inside one of the out of the box catalogs. If you are going to make a bunch of content that you may want to migrate to future versions,

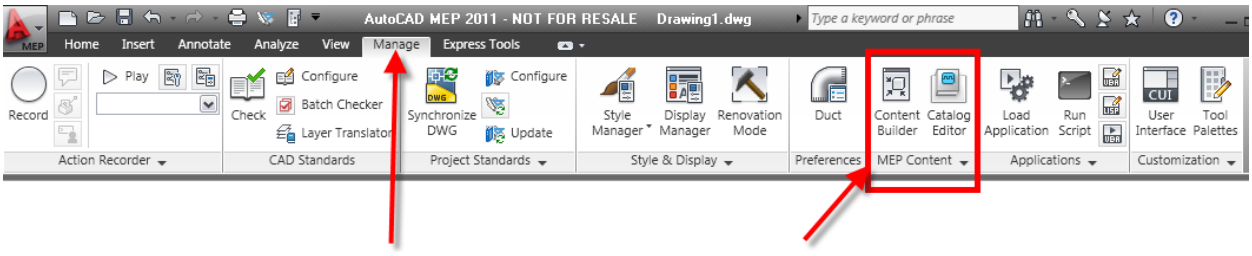
consider creating a company catalog. Your company catalog could sit off to the side from our out of the box catalog parts. When the next release comes out, you only need to migrate the company catalog by adding a file path to the new release. There is no intermixing of custom parts and older version, out of the box parts to detangle. Why would you need to detangle them? Our catalog parts don't change every release. If you migrate an old catalog that contains our old out of the box parts, the risk is that they have not changed in the new release. We may add more, or fix some but their unique identifiers (GUID's) are still the same between releases. GUID is a software developer term, stands for "Globally Unique IDentifier." When our product sees two identical parts (same GUID), you get an error message asking you to regenerate your catalog. Their GUID's are not so unique anymore! This error pops up every time you launch the program. Please regenerate your catalogs. So if you migrate a catalog that contains some out of the box parts from an older release, chances are they did not change. You get the error message in the new release to regenerate since you now have two of the same part. The Appendix of this whitepaper will have some steps to create your own company catalog.

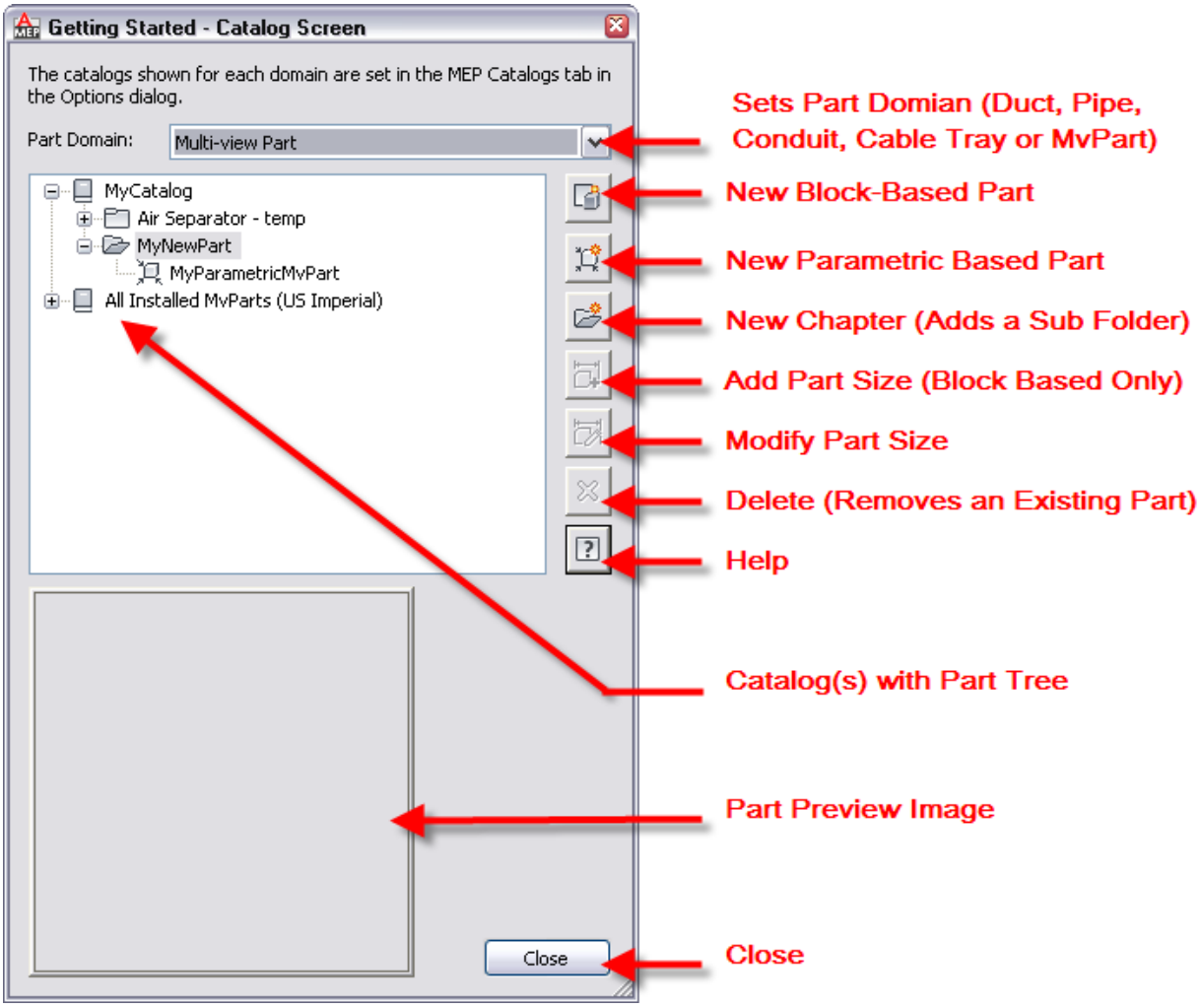
Content Builder Is Easy!

You just need to know the rules of the road. If you have never seen an automobile, would you feel comfortable driving it? No. But let someone explain it to you; give yourself some experience and eventually you find that it becomes second nature.

Launch Content Builder

To launch content builder, find the manage tab of the ribbon. Find the "MEP Content" group and click "Content Builder."





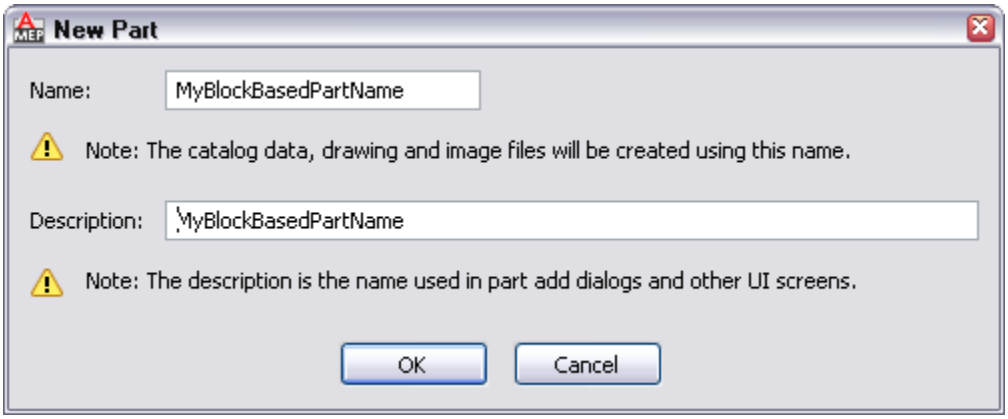
Initial Content Builder Screen

Part Domain: This determines what catalog to display and in what catalog it creates a new part when you click the “New Block-Based Part” or “New Parametric Based Part” buttons.

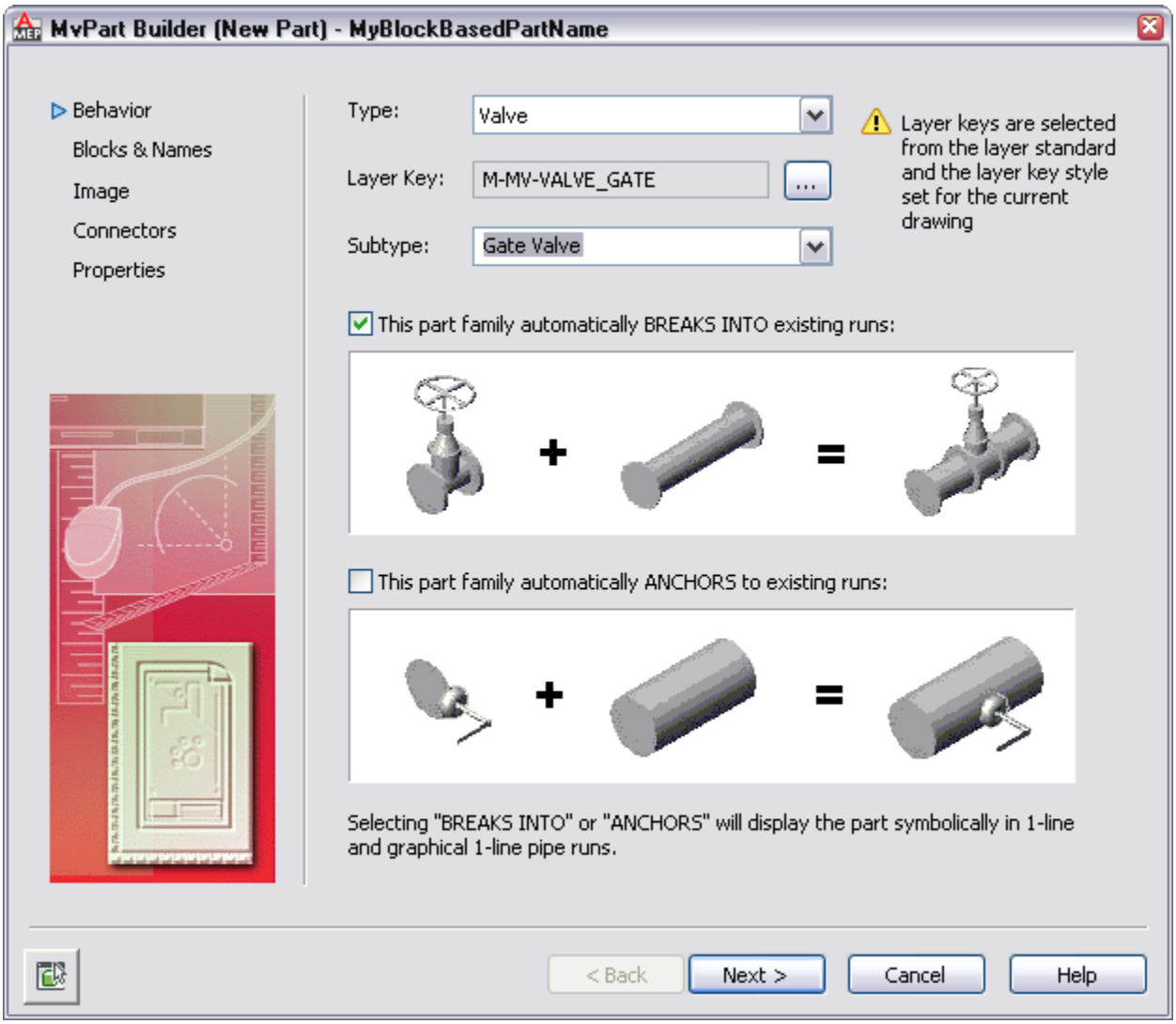
New Block Based Part: Creates a new block based part in the catalog. You must create a solid and from that, turn into a block, prior to generating a block based part. The wizard that runs after clicking this new block based part button, will be needing a solid > block to generate the respective left, right, top, bottom, front, back and model view blocks. You may also wish to create a flat 2D schematic symbol block and a bitmap image prior to creating a new block based part. The schematic symbol and image are optional. I’m going to cover block based part creation only very briefly, as we want to focus on parametric part creation.

The block based part wizard will:

1. Ask you to name the part

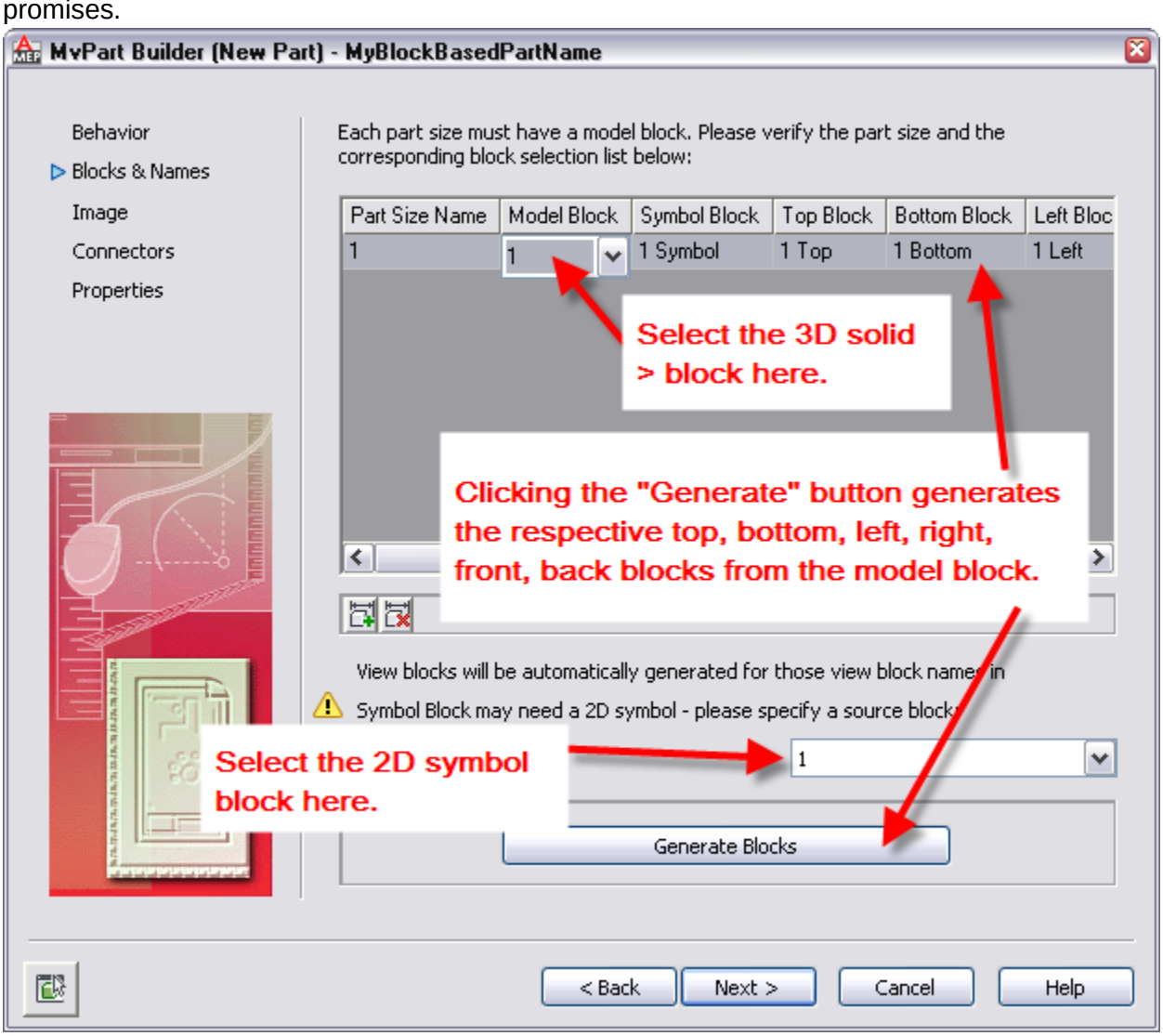


2. Set the behavior of the part. Classify the part type and sub-type (Ie. Part Type: Valve > Sub-Type: Gate Valve)

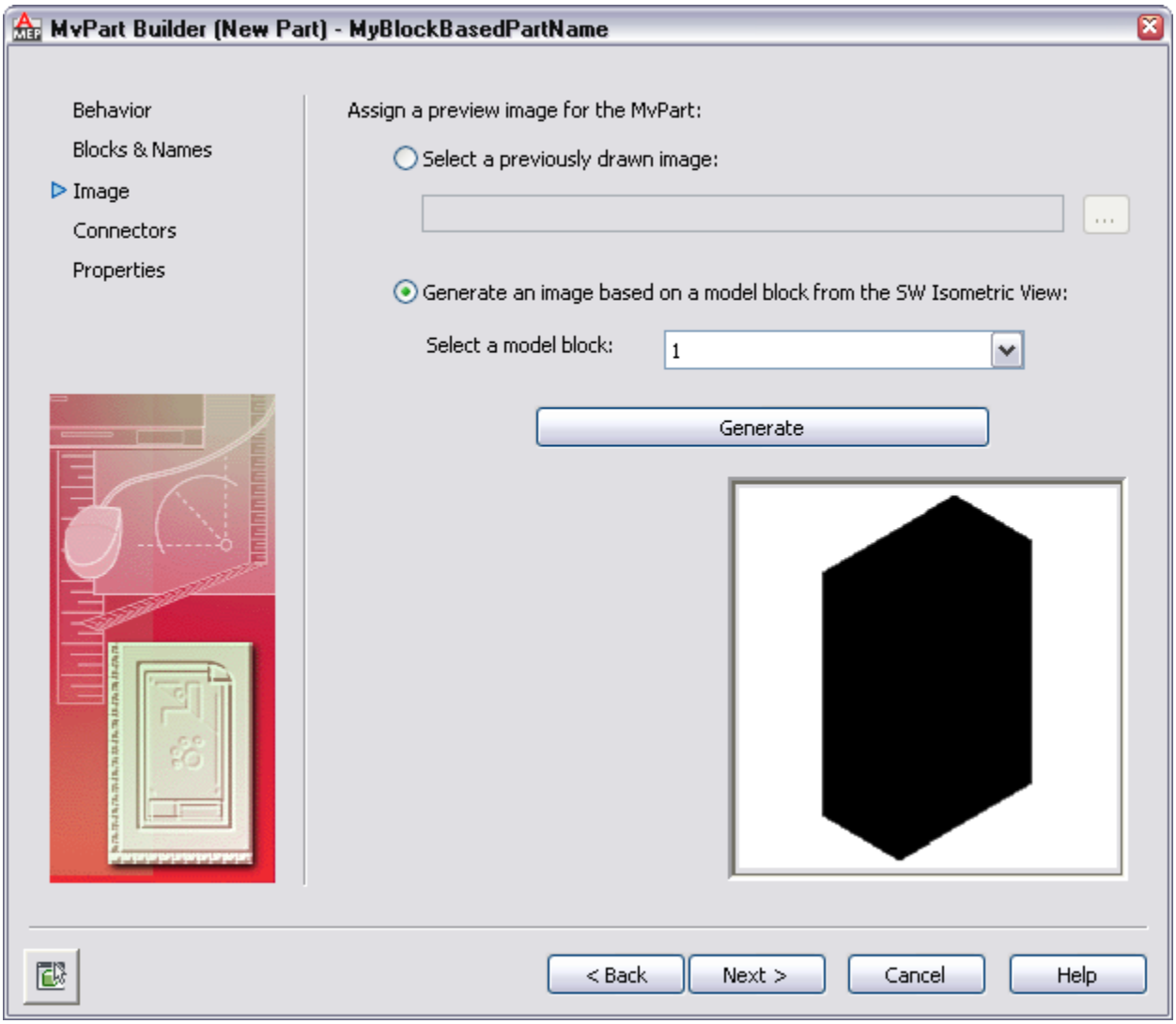


3. Assign a layer key. See image above.
4. For “behavior” it wants to know if upon insertion, does it “break into” a duct/pipe segment splitting it into two segments, or does it “attach to” or sit inside a single segment? Ex. An inline pipe valve needs a pipe segment on either side, versus a duct balancing damper which sits inside a duct segment. See above image.

5. On the next screen called Blocks and Names, you supply the solid > block, and allow it to generate the respective views like I described above. It also asks for a schematic symbol block on this screen. NOTE: When you create the solid, prior to making a block or using this wizard, move your solid back to 0,0,0 WCS. Make sure 0,0,0 WCS is at the point of insertion. Then convert your solid into a block with its origin at 0,0,0 WCS. Then use the content builder wizard to make a new block based part. If you do not make your part at 0,0,0 WCS, when you go to add new connectors, the body of the part will not show up in the screen that allows you to put connectors on that 3D body. Kind of hard to add connectors when you cannot see the part, and you will not be able to zoom out to the part. It is a limitation we hope to address at some point in the future, but no promises.

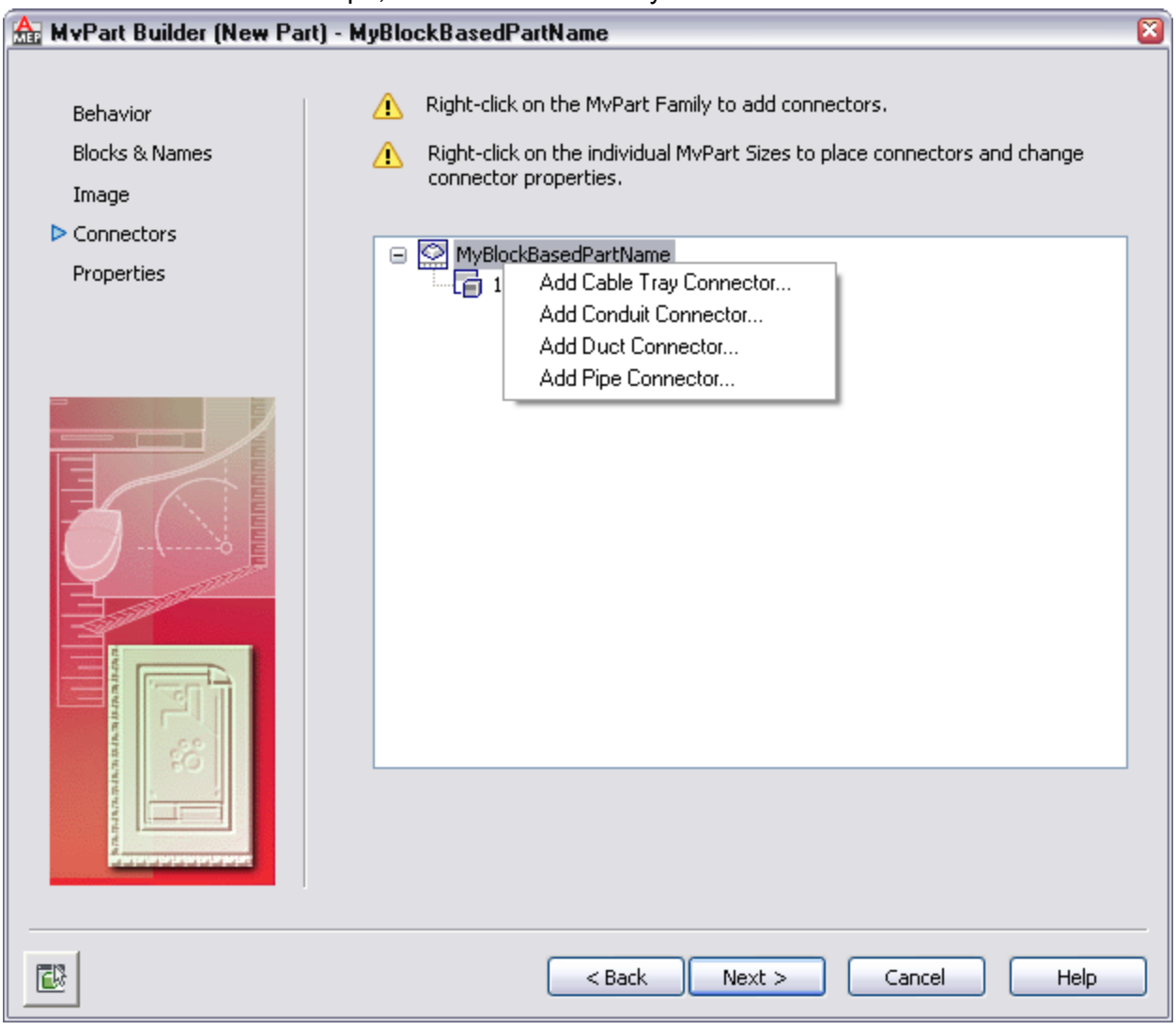


6. Next screen asks for a pre-made bitmap image, or you have an option to generate an image from the solid > block. It will use this image in the MVPARTADD dialog.

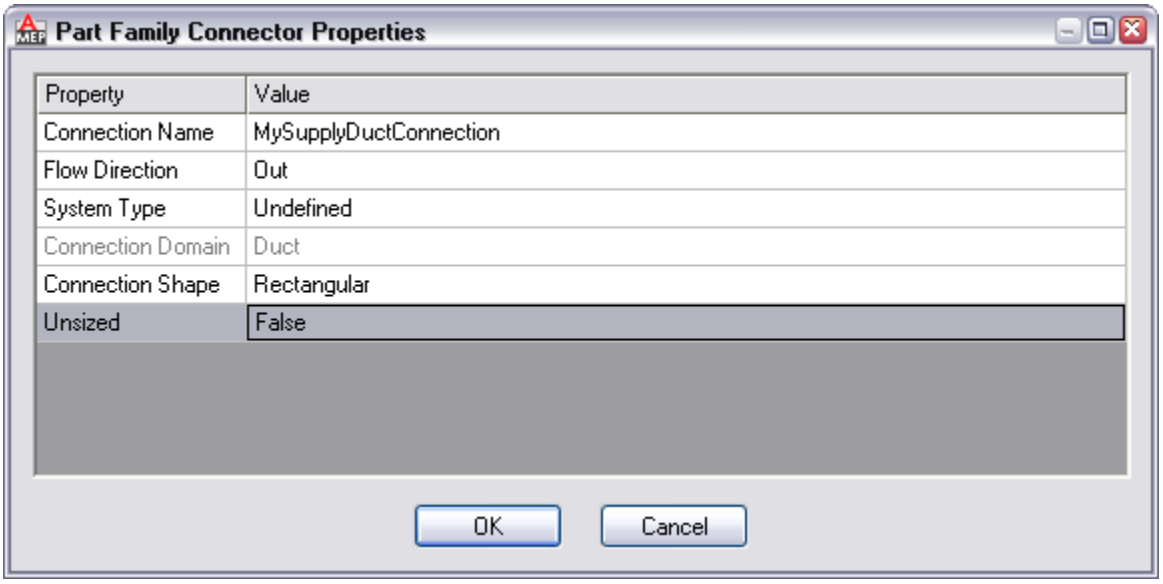


If you have an image editor, you can create some really nice images to represent your part. As you can see, generating an image has some limitations. You can make out the outline, but no internal detail.

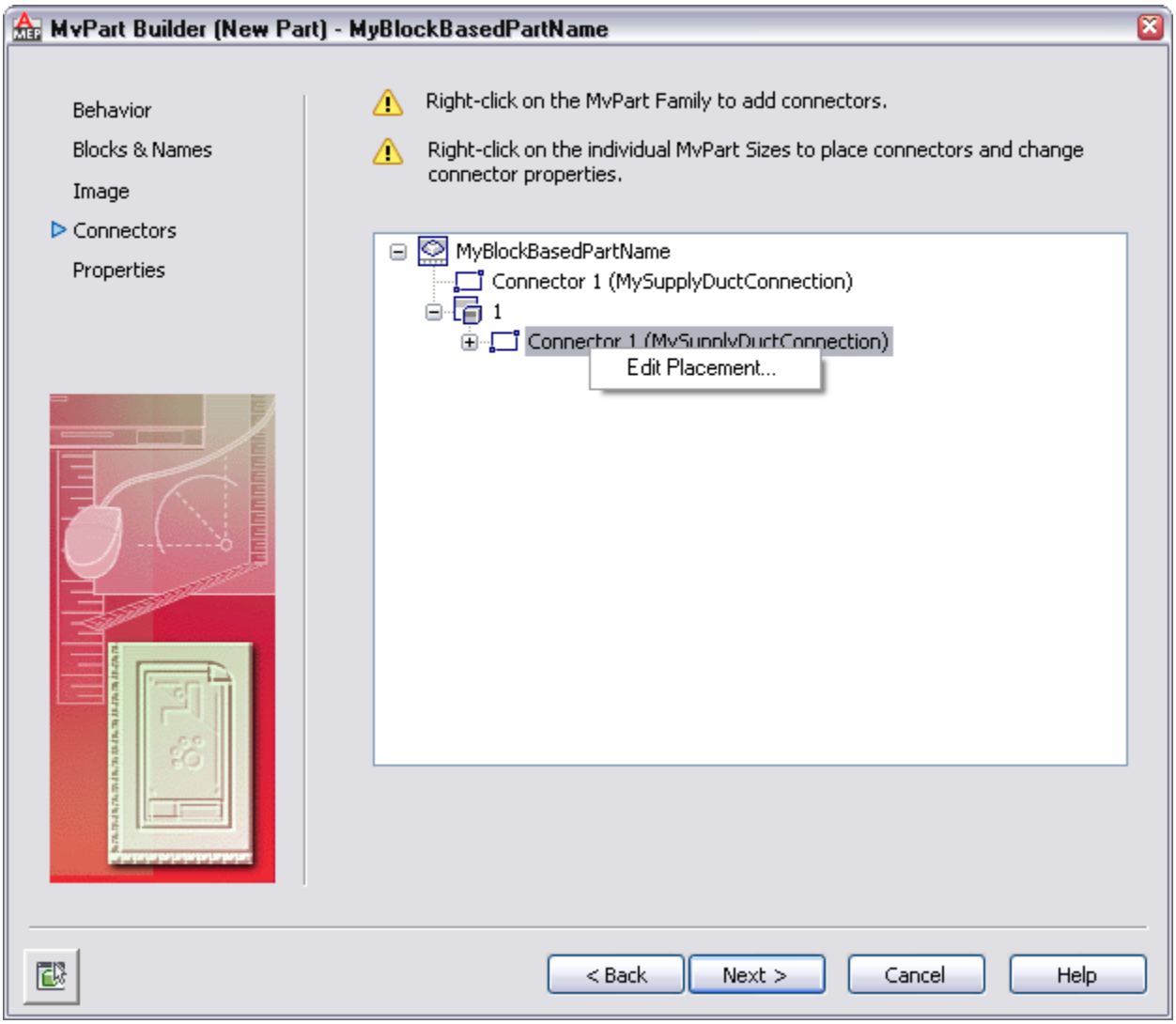
7. Next screen allows you to add connectors to your 3D part. Connectors are required, but we don't force that on you. If you fail to add a connector, we don't stop you. However, it's the connector that stores the part elevation information that is seen on the property palette. So if you want to be able to change elevation from the property palette, you need a connector somewhere on your part. There are connectors for pipe, duct, conduit and cable tray, but not wire. Wire is not a 3D object. Our parts are often at elevation in a ceiling or on a roof, whereas our wires are drawn in plan view and reside at 0 WCS z-elevation. To add a connector, you right-click on the tree in the connector screen and "Add Duct Connector." Or Pipe, Conduit or Cable Tray.



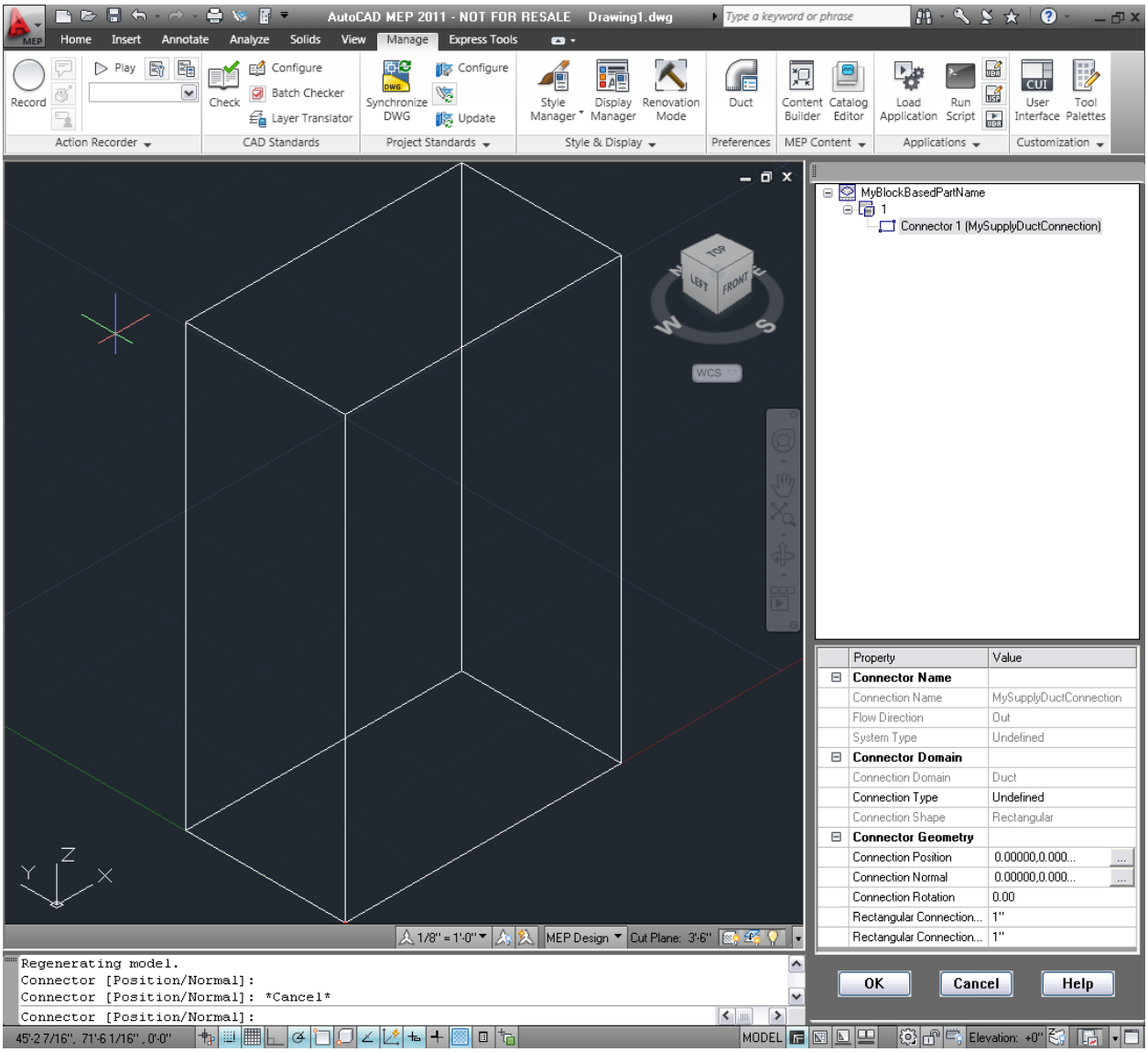
8. A dialog pops up asking us to name the connector, and provide more detail about the connector. Give it a name, but feel free to leave the other settings alone. We can set the size and shape of the connector when editing the connector placement.



9. We now need to edit the connector's location on the part. Select the connector in the tree, as seen below. Right-click and select Edit Placement.



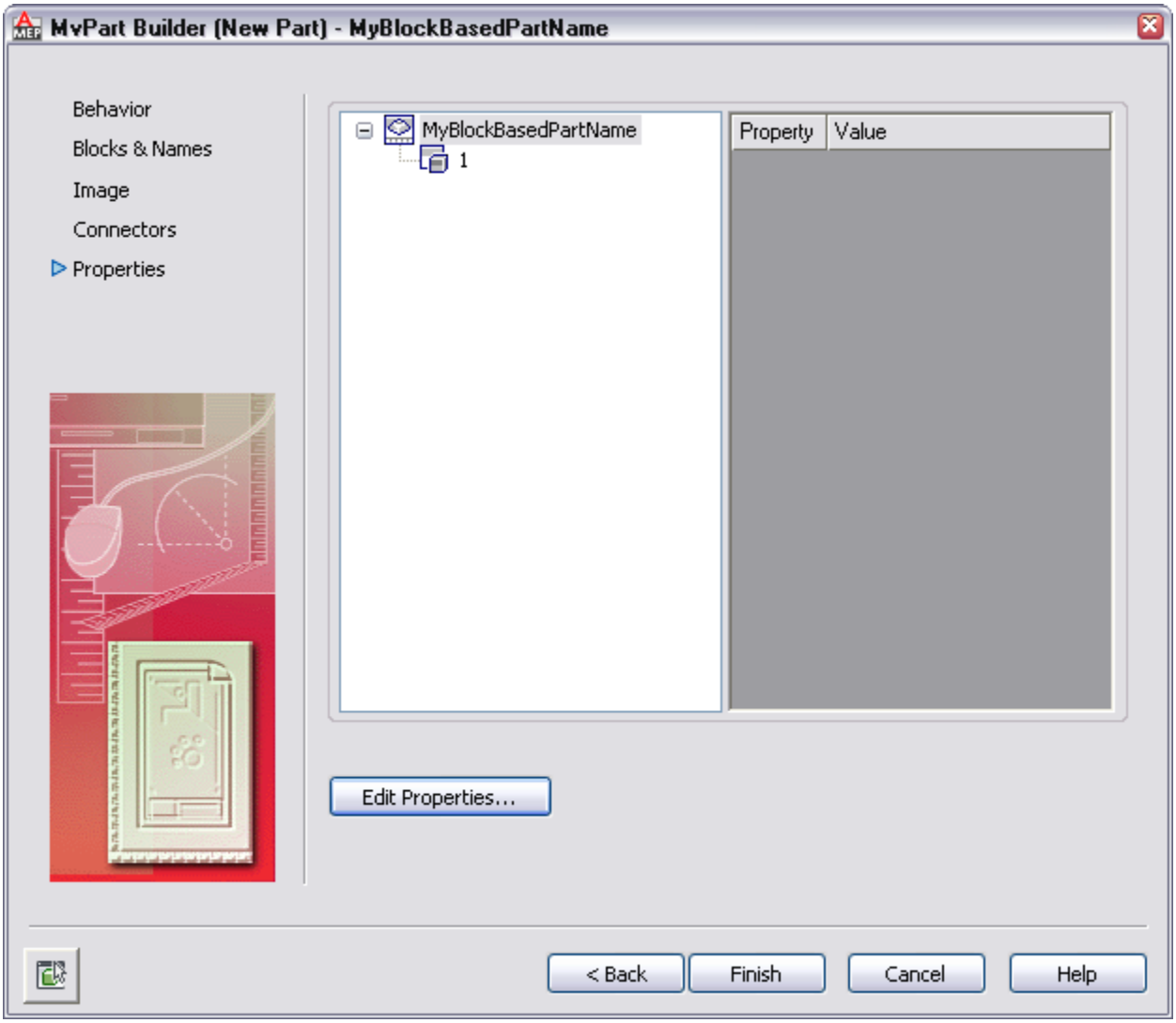
10. It then opens up your part in model space. Settings will be on the right side in what looks like a docked palette or window pane.



If your solid > block was originally created at 0,0,0 WCS, it will appear in model space. You will not see it here, if you fail to do so, and you will not be able to place connectors on the part. To place connectors, find the heading for “Connector Geometry” and select the button “…” to the right of “Connector Position.” In model space, the connector will be stuck to the end of the curser. Your Osnaps will work if turned on. When you have snapped the curser to a point, click your left mouse button to place the connector. You next will want to orient the connector to face away from the part. The arrow on the connector should point away, or put another way, it should point in the direction that your duct or pipe will approach the part. The direction that the arrow points is called the “Connector Normal.” You click the “…” button to the right of “Connector Normal.” You can Osnap, from any point to point, in order to orient the arrow in that direction. Instead

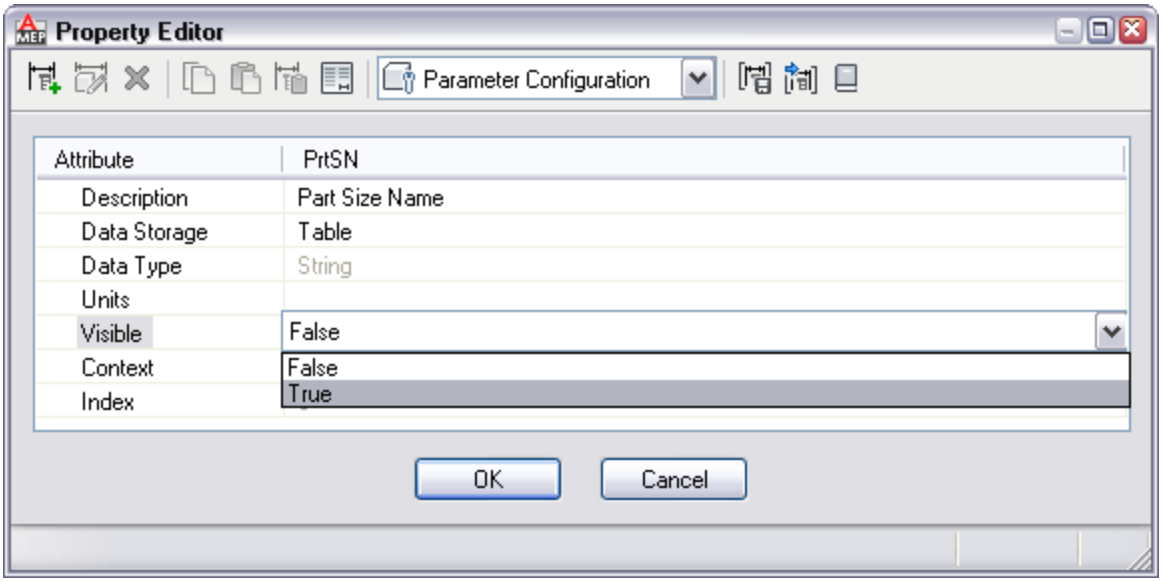
of clicking the "…" button, you can select in the text box and manually type in a normal direction. The value ranges from -1 to 0 to 1, in the x, y, z coordinates.
A connector normal value of:
+1,0,0 points the normal arrow down the positive x-axis
-1,0,0 points the normal arrow down the negative x-axis
0,+1,0 points the normal arrow down the positive y-axis
0,-1,0 points the normal arrow down the negative y-axis
0,0,+1 points the normal arrow down the positive z-axis
0,0,-1 points the normal arrow down the negative z-axis
Connector Rotation will rotate the connector. For a round connection, this may not have much effect, until you draw a round duct or pipe away from your part and attempt to elbow. For a rectangular duct connection, you will see the rectangle rotated by this much on the face of your part. This rotation setting is rarely if ever used. You may want to swap "Connector Height" and "Connector Width" dimensions, and rotate your connector by 90 degrees. Below Connector Rotation are "Rectangular Connector Height" and "Rectangular Connector Width" settings. If we had a round connector, it would have "Round Connector Diameter." Most equipment has a specific connection dimension, so this is where you set those dimensions. This dimension cannot be changed outside of this block based content builder wizard. You can always go back and edit the part size in content builder. Click the Ok button to exit, or in the tree above the connector settings, you can select a different connector and edit it's placement settings before Ok'ing out. Click Next to move on to the next and last step of the block based part wizard.

11. The next screen is the Properties screen.



Here you will want to click the Edit Properties button.

12. Under the “PartSizename” column, make the “visible” setting True. The part size name will then show up in the MVPARTADD dialog.



There are tricks to attach custom property sets to your MvPart, that you can also do from this dialog. See TS1053724: http://usa.autodesk.com/adsk/servlet/ps/dl/item?linkID=9240937&id=6100819&siteID=123112

This TS1053724 will be updated soon, but requires some more information for newer releases, 2009 onwards. In the past releases, the file mentioned in the TS Step 8 (*C:\Documents and Settings\All Users\Application Data\Autodesk\ABS 2006\enu\Aecb Shared Content\AecbPropertySetDefinitions.dwg)* was removed due to the duplication of all its property set definitions inside template drawings and in the Style content folder > System Definition.dwg.

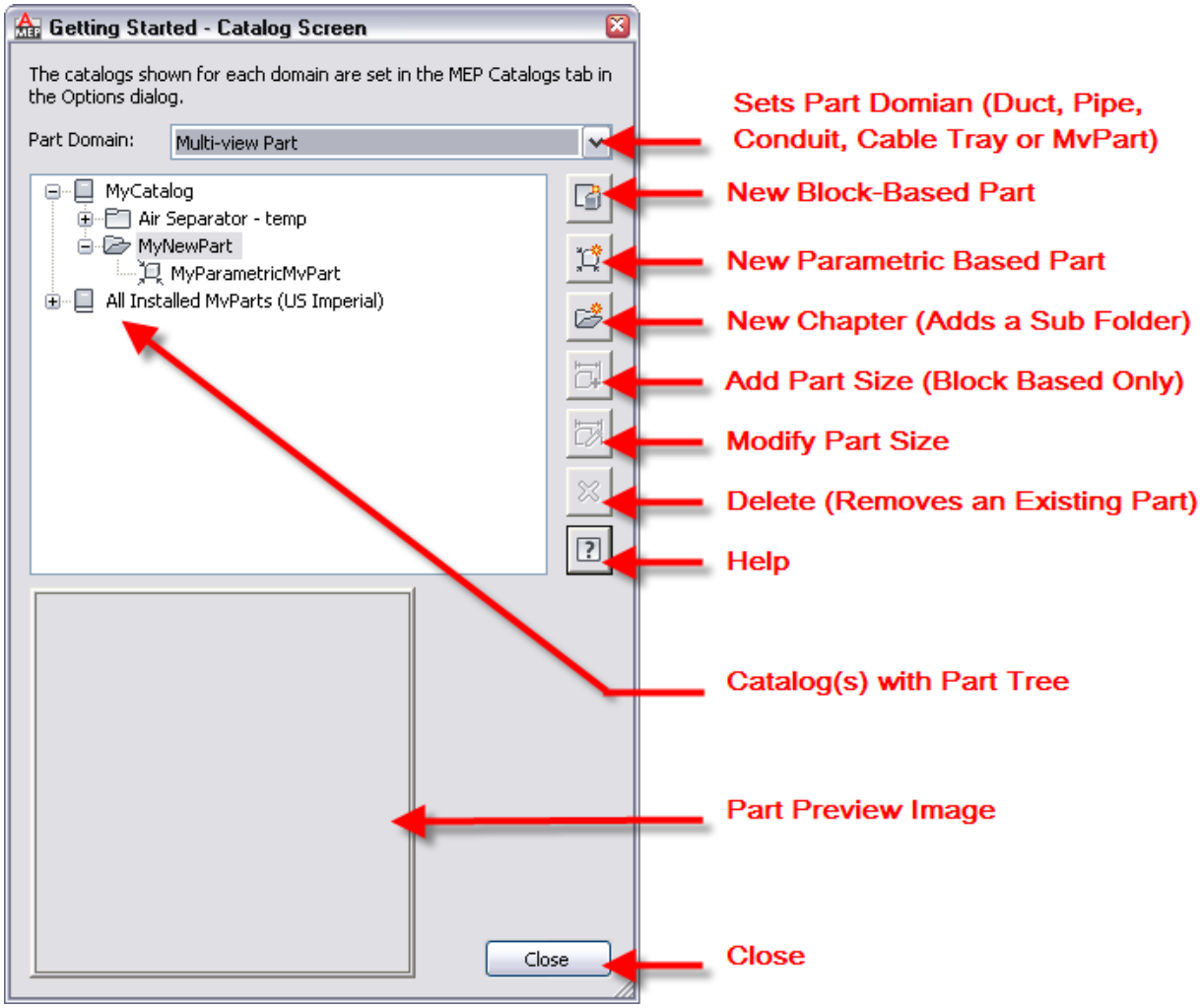
So this Step 8, would be changed like this:

8. Open C:\ProgramData\Autodesk\MEP 2011\enu\Styles\Imperial\Schedule Tables (Imperial).dwg

The rest of the steps would be the same to automatically attach a custom property set. Click Ok to exit the Property Editor.

13. Click the “finish” button to finish creating a block based part. You can add your part to a drawing by typing in MVPARTADD at the command line.

Back to the buttons on:



The next button after New Block-Based Part is New Parametric Based Part. This launches the Parametric Part Wizard that we often just refer to as “Content Builder.” This is where we will be focusing on. Before we do, lets briefly finish the remaining buttons on the above dialog.

Below New Parametric Based Part, is the New Chapter Button. It adds a sub-folder to your catalog. It allows you, for example, to keep your valve parts separate from your chiller parts. In the content folders on your hard drive, it will create a new folder to contain the part files.

Below the new Chapter button is the Add Part Size button. This button is only available if you are selecting on a block based part in the catalog part tree. You can create more solids, convert each to blocks, each represent more sizes for your block based part. This button allows us to add new sizes to our existing block based part. It runs the block based part wizard again to specify the model and symbol blocks, and add connectors for that size that you are adding to the existing block based part.

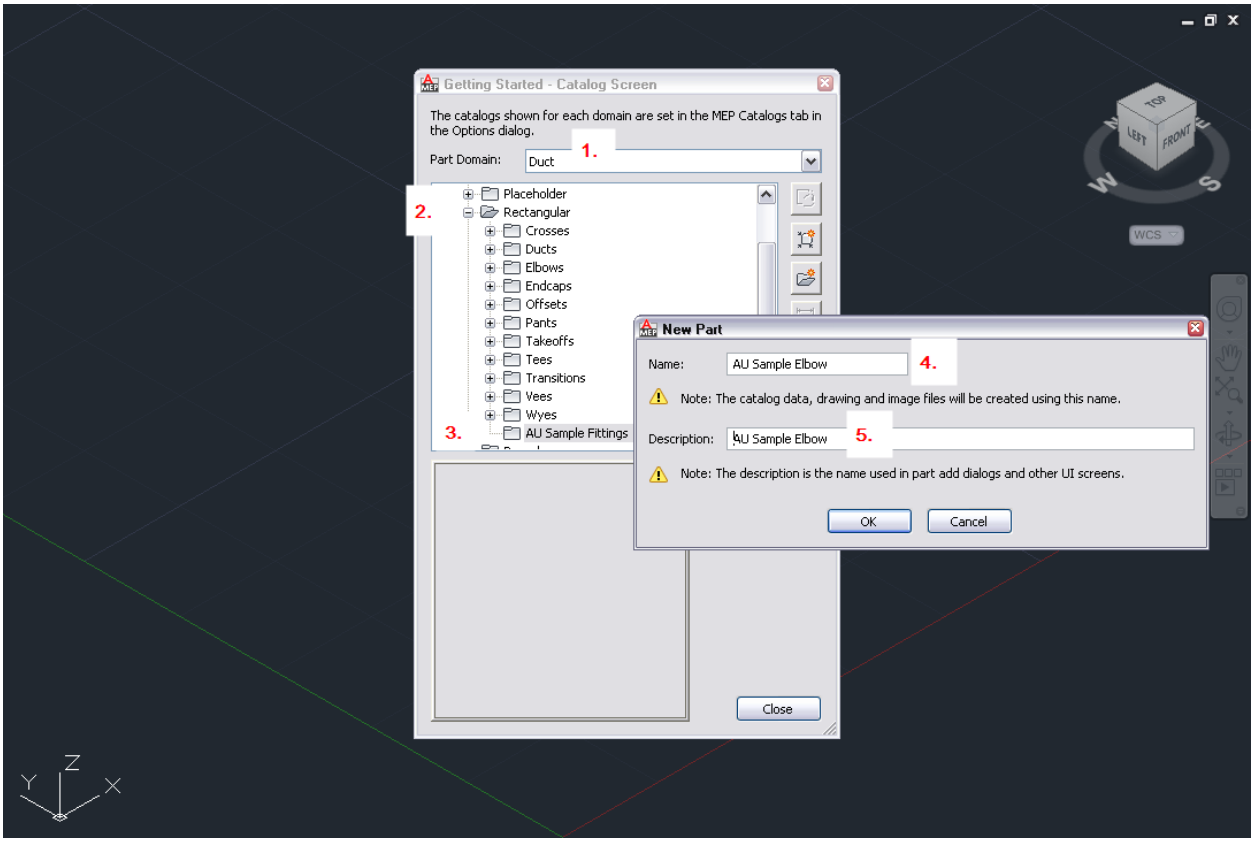
Below the Add Part Size button is the Modify Part Size button. This works for both block and parametric based parts, and launches either the block based part wizard, or the parametric based part wizard (Aka. Content Builder). If you ever want to edit our out of the box parametric parts, maybe to add more sizes, this is the button you want.

The next button is the Delete button. It deletes the selected part. It does ask you to confirm before deleting. There is no trash bin, so be cautious. After the Delete button, is a button to pull up a help dialog on content builder.

Parametric Based Part Wizard Aka. Content Builder

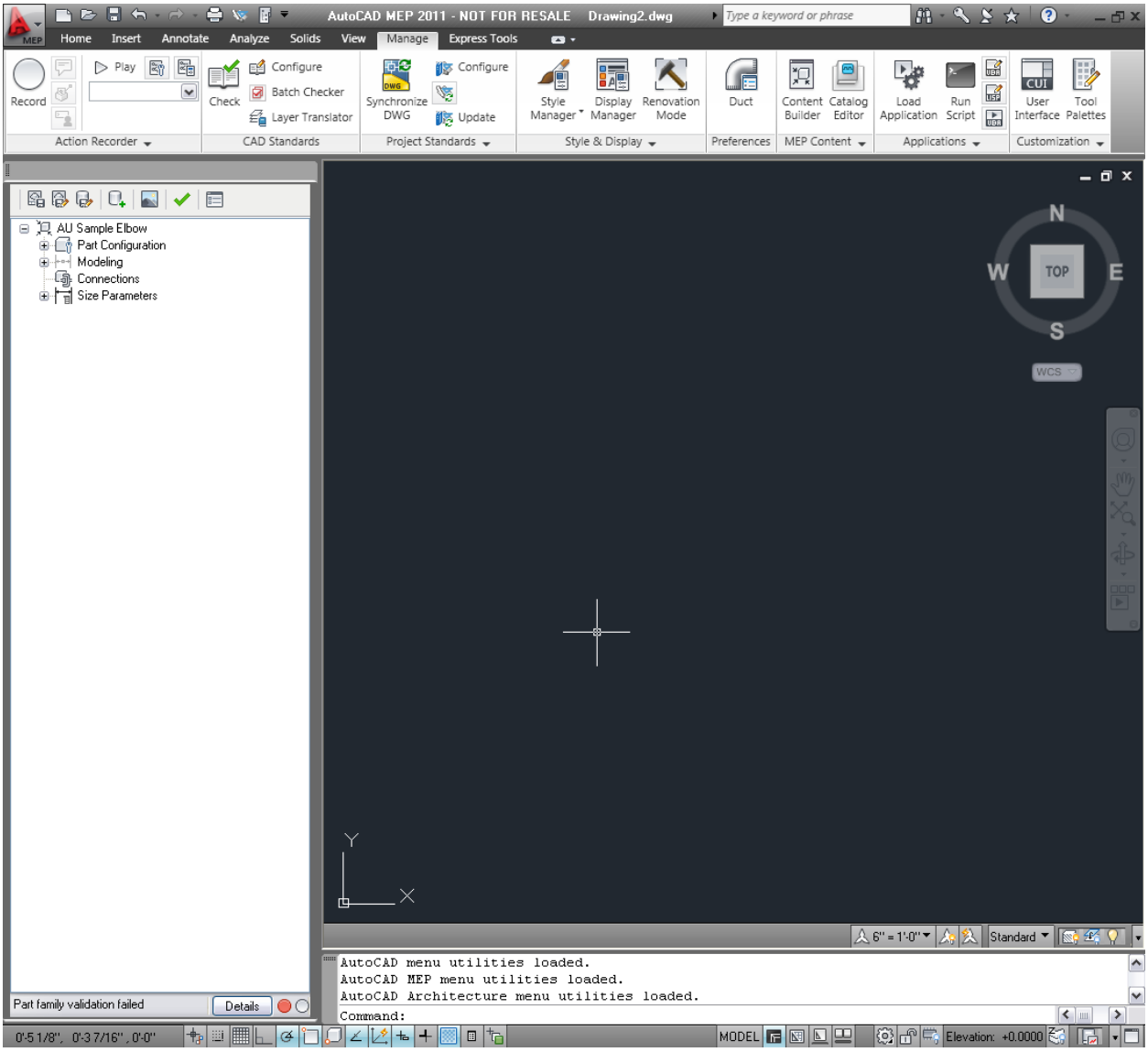
To learn the parametric part wizard, we are going to create a duct elbow fitting that allows us to edit the length of the legs, the angle of the elbow and the radius of the elbow.

1. Let's change the part domain to "Duct."
2. In the tree below, select on the "Rectangular" folder.
3. Let's then create ourselves a new chapter (Ex. AU Sample Fittings) in the rectangular folder to store our new part.
4. Select on our new folder "AU Sample Fittings." Click the "New Parametric Based Part" button. Let's name the new part "AU Sample Elbow."
5. After you type in the name, simply placing the curser down in the "Description" field, fills out the description to match the name. It is important that the description always match the part name.

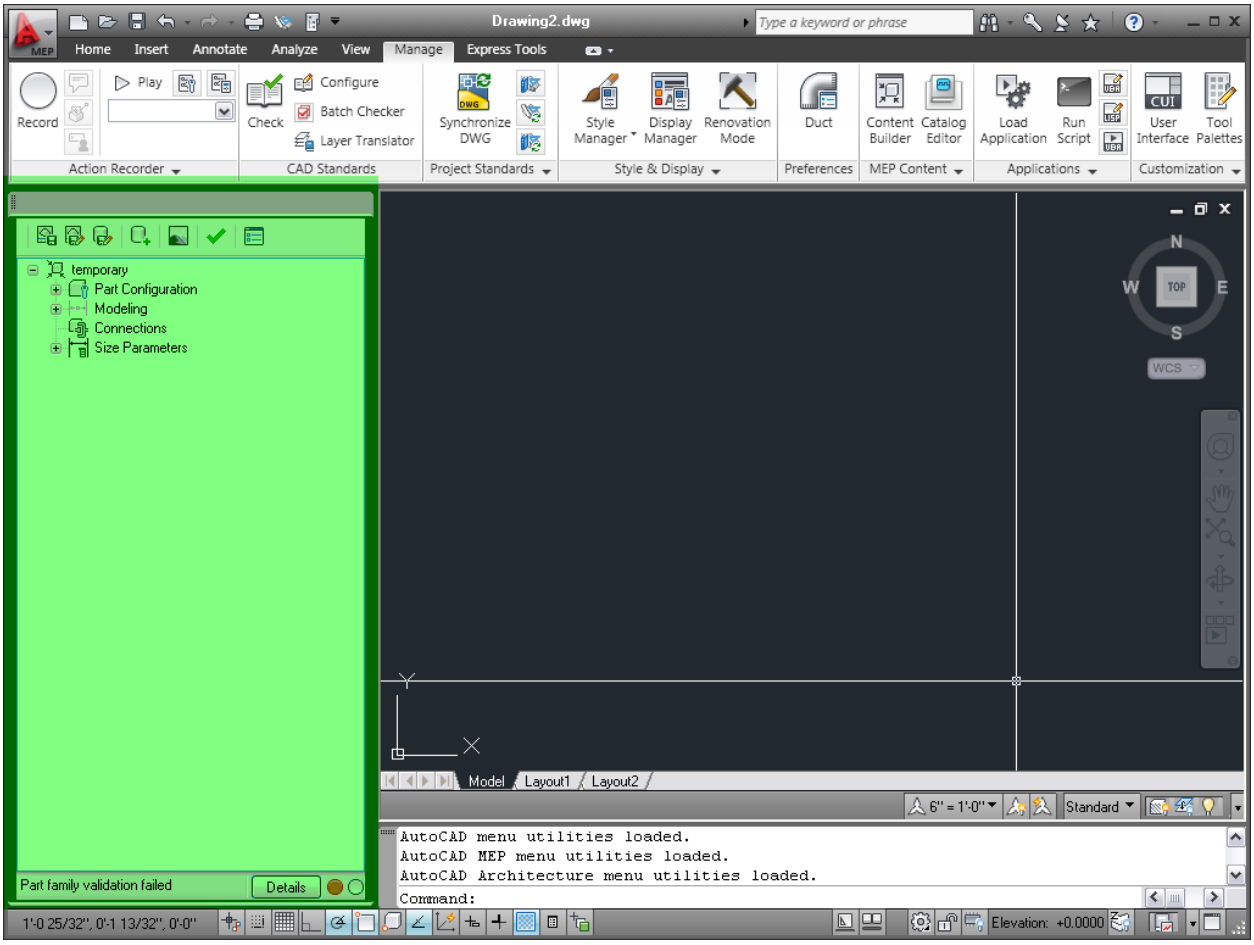


In the MVPARTADD dialog, it is the “Description” that is shown, not the part name. This becomes important if you ever use the “Save As” button inside Parametric Part Wizard. The same dialog pops up allowing you to specify a new name. You have to remember to change the “Description” to match the new Part name. Otherwise, when you pull up the MVPARTADD dialog, you will have two parts with the same name and not tell them apart.

6. Click ok and the Parametric Part Wizard will open.



Unlike the Block Based Part Wizard > Edit Connector Position screen, the Parametric Part Wizard has what looks like a docked palette or window pane on the left side. I'm going to call it a window pane.

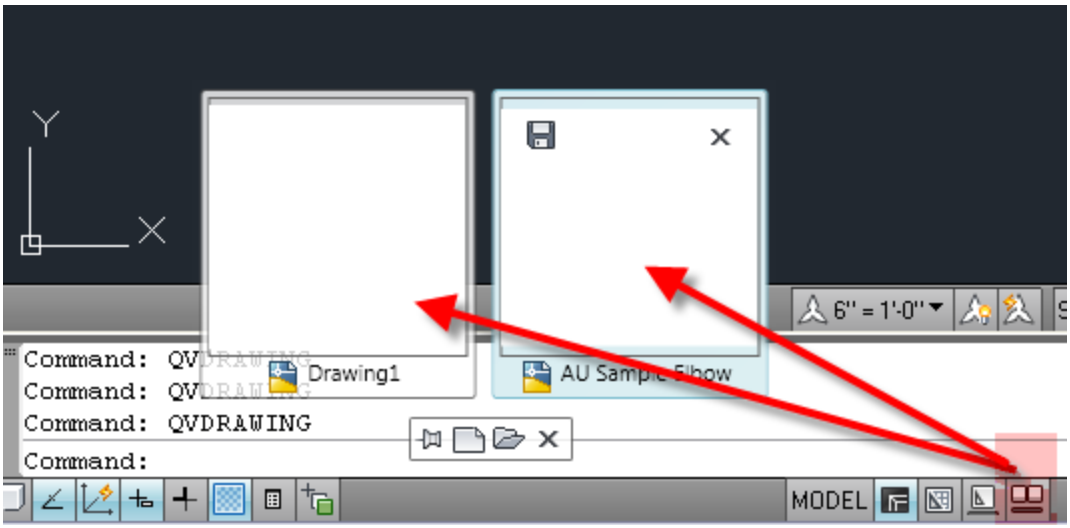


What you need to know right now: Every Content Builder tool and command is started from a right-click on this tree in this left window pane. What you see up in the Ribbon, is AutoCAD and AutoCAD MEP. We don't prevent you from using the Ribbon, but you should avoid it. This is Content Builder, not AutoCAD/MEP. The only AutoCAD/MEP commands that are intended to be used with Content Builder are the UCS commands, Visual Styles, Zoom/Pan/Orbit, SW ISo commands. The view cube works and can be used.

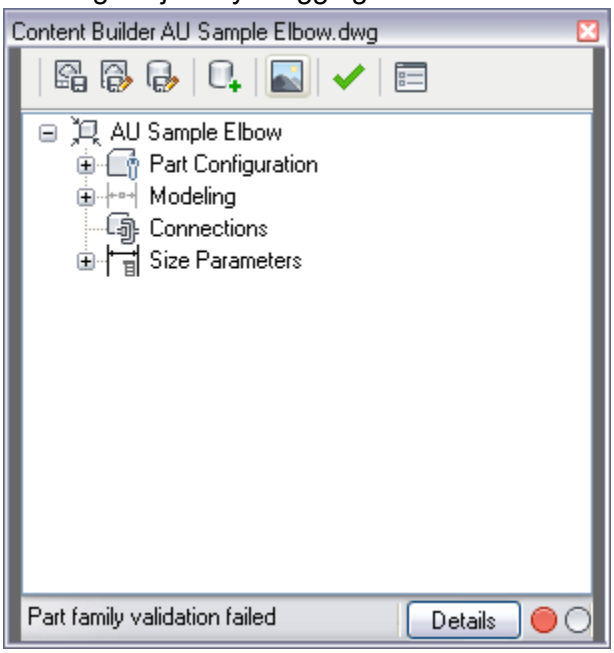
Do NOT use: The Undo button! Use the delete button instead and redo your actions. Remember, Content Builder is recording the steps and sequence you take to make your part. If you over-constrain, it needs to know what dimension to ignore (the last one created). The Undo button can corrupt the part if used at the wrong step, due to the interdependencies of objects and sequence. Do NOT use most of the Ribbon tools or menu commands. Ex. AutoCAD lines, arcs, circles, mtext, dimensions, annotation scale, font styles or any other object. Only use the lines and dimensions we can create from the docked palette on the left. They have different commands than the AutoCAD/MEP commands. Similarly, the AutoCAD MEP objects and displays do not

have a place in content builder. Don't draw duct or pipe. We don't stop you, but know that these objects are not part of the workflow in content builder. The content builder objects are not meant to interact with them. The only effect they may have is to corrupt the content builder part.

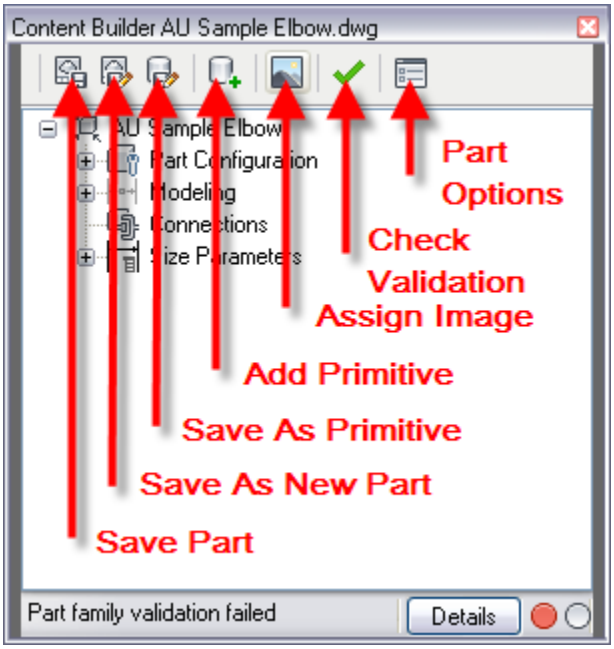
Next thing to know. We are in a new drawing file that content builder creates and places in the content folder. The original file we were in, prior to launching content builder is still there and open side by side. You can switch between content builder and the other file and you may want to do so to refer to any model you built in your planning session.



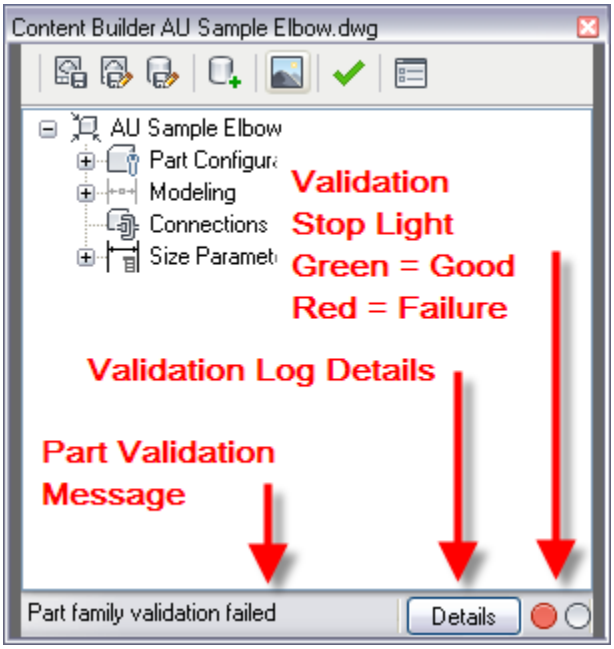
You can right-click the window pane on the left and undock it by unchecking "Allow Docking" or just by dragging it off the left side.



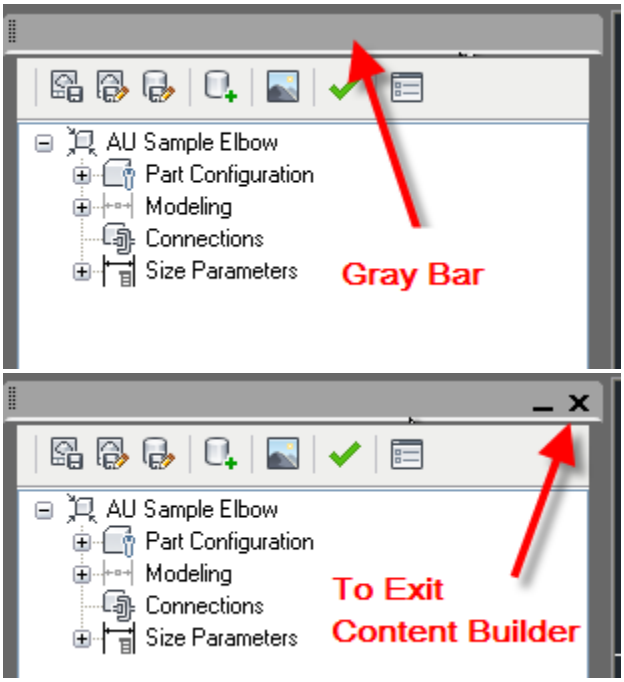
At the top, we have a row of icons. See image below for descriptions.



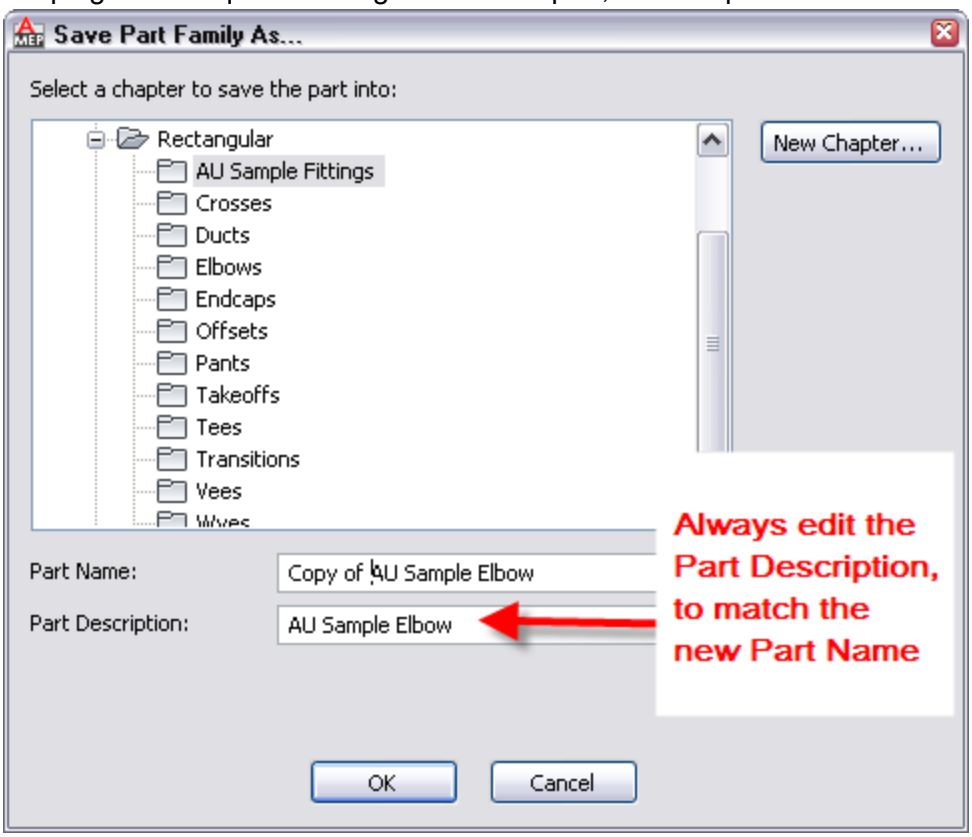
At the bottom, we have information. See image below for descriptions.



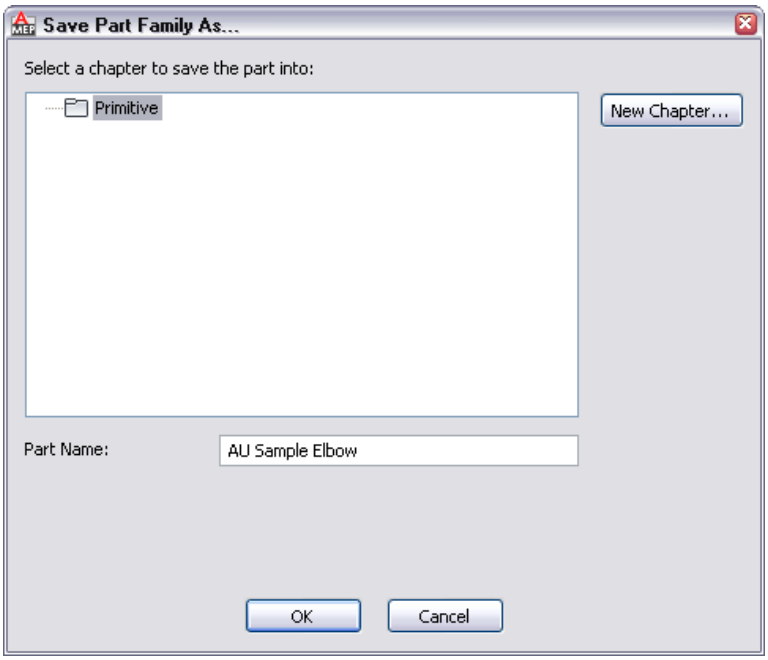
Above the top row of icons, is a gray bar. Hovering your curser over this gray bar, reveals the – and X. Click the X to exit content builder. It will prompt you to save the part.



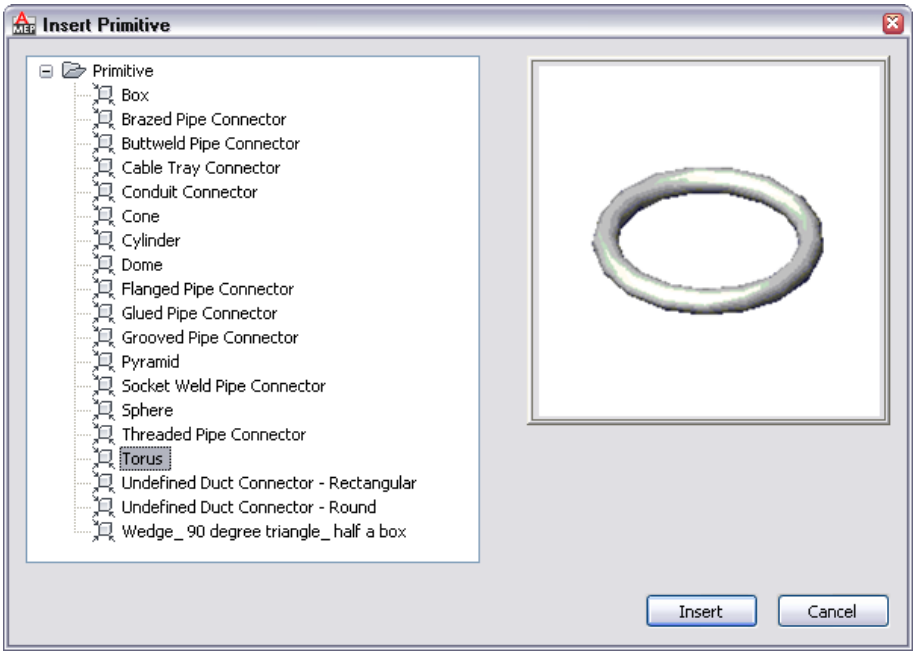
Save Part button: Has no dialog. It just saves the part.
Save As New Part: Creates a duplicate of the current part and allows you to specify a new part name and description. This is where you must be cautious. Be sure to edit the part description to match any new part name. The description, will by default, have the original part name. If you create a new part from the old one, and fail to edit the description to match the new part name, in the MVPARTADDDIALOG you will have two parts with the same name. TIP: Never use Windows Explorer to make a copy of your parts. Use this Save As New Part button. It will generate a GUID for the new part, thus keeping the "Unique ID" assigned to each part, well unique.



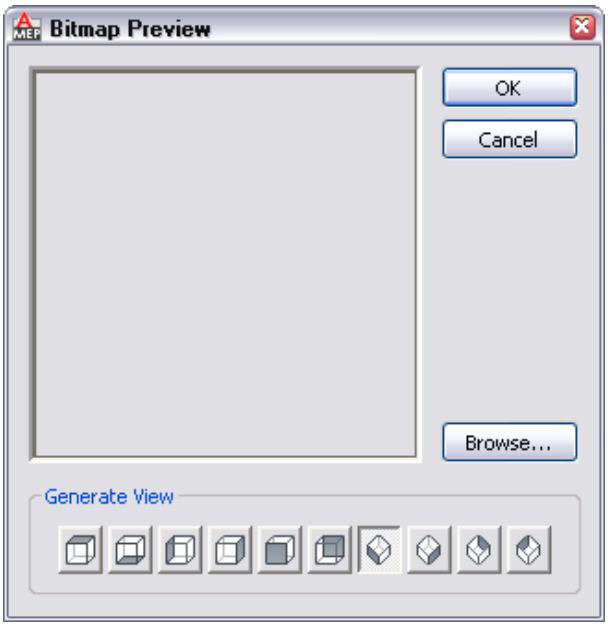
The Save As Primitive dialog is similar, but does not have Part Description since Primitives are shapes used to make future parametric part extrusions.



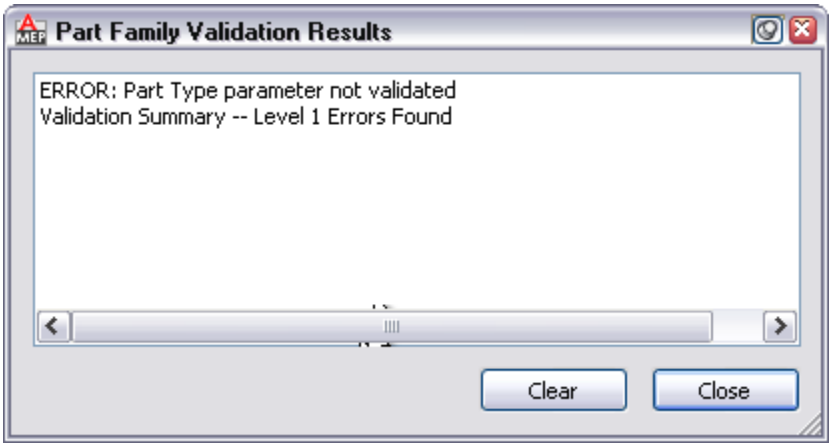
This is the dialog for Insert Primitive. Again, a Primitive is a shape that you can insert into your part to rough-in a shape. You cannot use the AutoCAD commands to move, copy, paste, rotate or array these primitives. They insert where they insert, and their orientation is what it is. You cannot change their position or orientation after initial insertion.



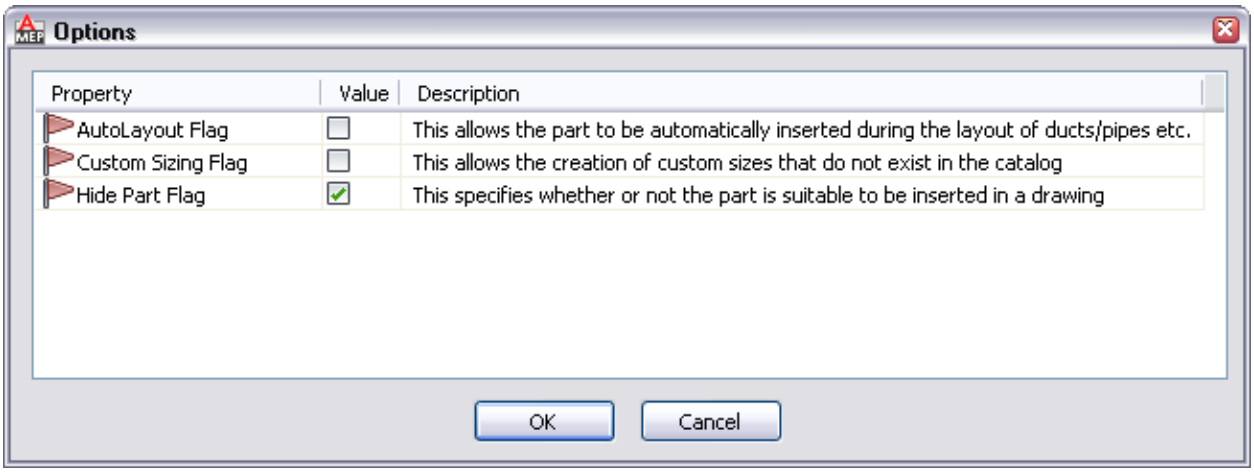
The Assign Image dialog allows us to generate an image of our part from the 3D model. You can only generate an image, after you have created the 3D model. There will be no preview until the part has a 3D body. Similar to the block based part wizard, you can create an image using an image editor and browse to that bitmap.



The next button in the top row of icons, is the Validate button. Clicking it pulls up this dialog, the message will change depending on the status of your part. The same dialog comes up if we click the “Details” button at the bottom.



The last icon in the top row, is the Part Options button. Clicking it pulls up a dialog the contents will change depending on the part domain and part type.



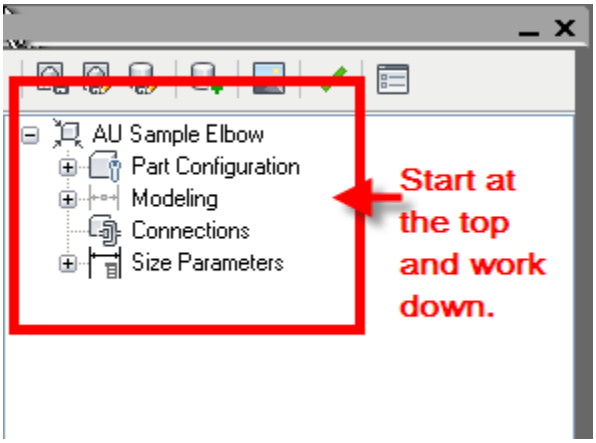
The “AutoLayout Flag” tells content builder if this part can be used in the Duct Part Preferences or for Pipe and Conduit, in the Routing Preferences. The Part or Routing Preferences tells AutoCAD MEP, what fitting to use in different situations. As you draw duct and turn a corner, what specific elbow should it use “Automatically?” A mitered elbow or a long radius elbow, etc? We call this “AutoLayout.” Some parts are unique enough that we would never use it in “AutoLayout” but prefer users to manually place the fitting.

The “Custom Sizing Flag” tells content builder if a user can insert a part with a custom size. We can build any size we need into our parametric part. Sometimes as users, we need a new part for a specific project and don’t have the time or inclination to build-in every size imaginable. Maybe we intend to revisit the part at a later date and finish those sizes. So during “AutoLayout” when it goes to insert the part, if the user encounters a size not yet built into the part, we are asking here if it is ok to allow users to add the part anyway.

The “Hide Part Flag” tells content builder if this part can be used at all. CAD Managers may be working on a number of parts on a centralized catalog. The CAD Manager may not want their part to be able to be inserted into a project drawing until they are finished creating the part. They may not have all sizes built into the part, and don’t want that part available in the part add dialogs until the final product is complete.

The Meat of Content Builder

The below image shows the meat of content builder. This is where most of the work is done.



The workflow in content builder is flexible, unlike the block based part wizard. Earlier we showed the workflow for the block based part wizard. It steps us thru the screens in a set order. The screens are Behavior, Blocks and Names, Connectors, then Properties. The exact same workflow is seen above, different names.

Part Configuration is about defining the “Behavior” of the part. The Part Configuration has settings that tells Parametric Part Wizard how this part is to behave. In Part Configuration, we select the Part Type and Sub-Type. If we choose a “valve” part type, it knows already if the part “Breaks Into” a pipe segment because that’s how all valves behave. So the intelligence behind our selections is improved over the block based wizard.

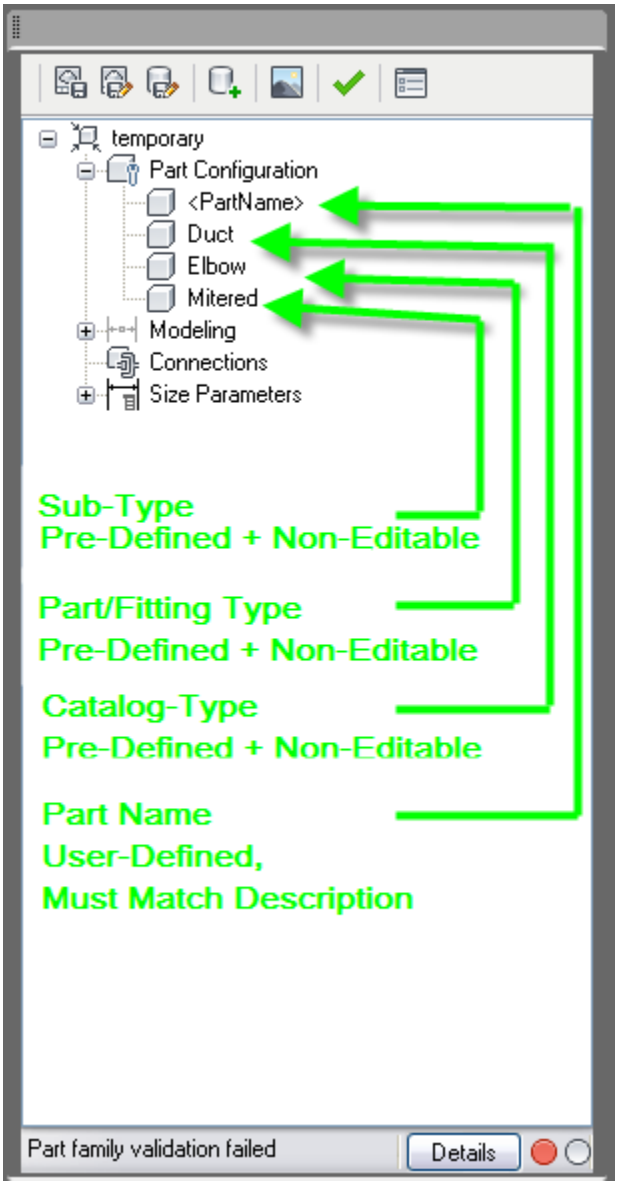
Modeling, is about defining the shape of the part and defining it’s dimensions. Very similar to “Blocks and Names” in the sense that we are defining the shape and size of the part.

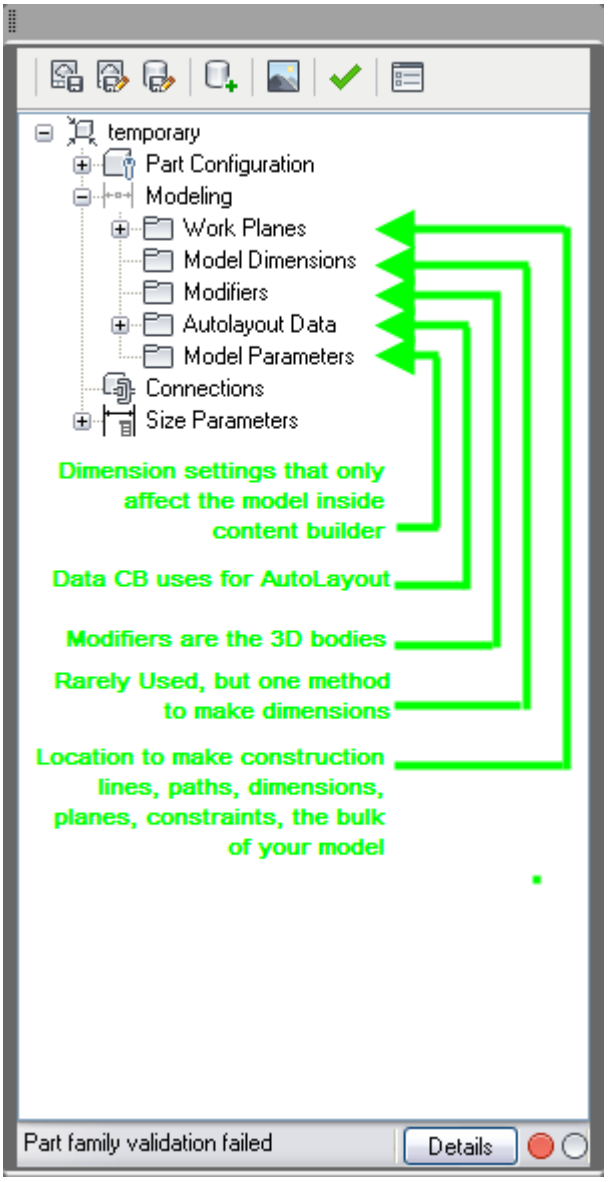
Connections is about defining the “Connector” types and locations. It’s very similar to the block based part wizard “Connectors” except that we are already in model space. We do not have to define a normal or size because parametric part wizard is more intelligent and can figure out the normal. Our Size Parameter dimensions can drive the connector size, or like for fittings, the pipe or duct size (determined at draw time) can drive the connector size.

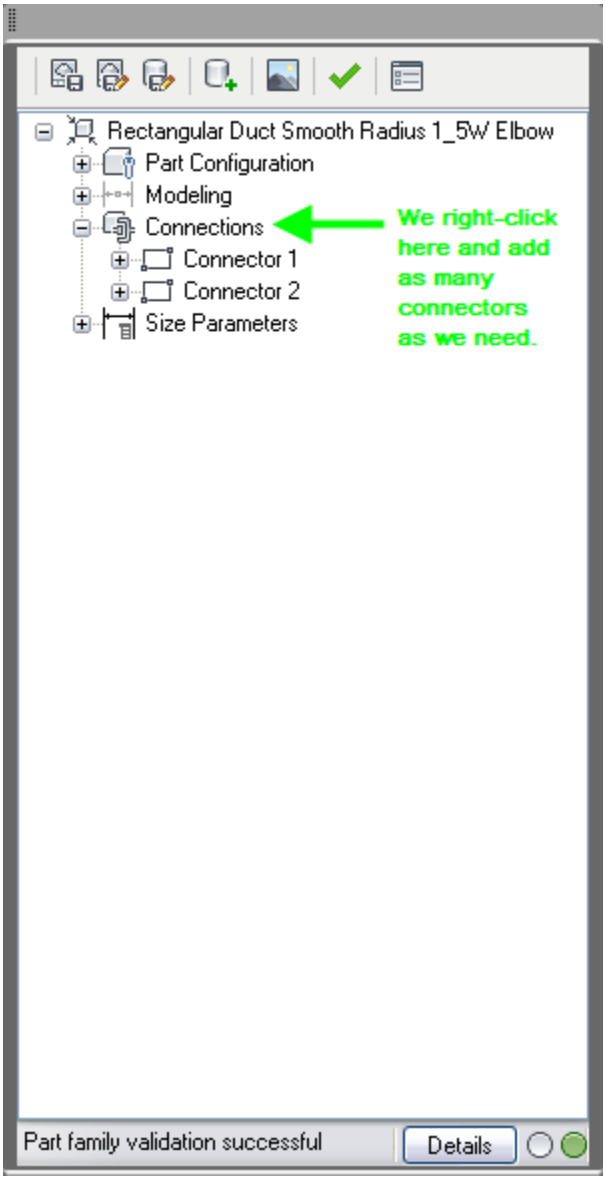
Size Parameters is about specifying the values that drive the dimensions for our parametric part. Not all dimensions have a list or table of values. Some dimensions use formulas. Some can be constants. We also specify in the size parameters if a dimension is visible, or available in the part add dialog. Just like the “Properties” section of the block based part wizard, we can

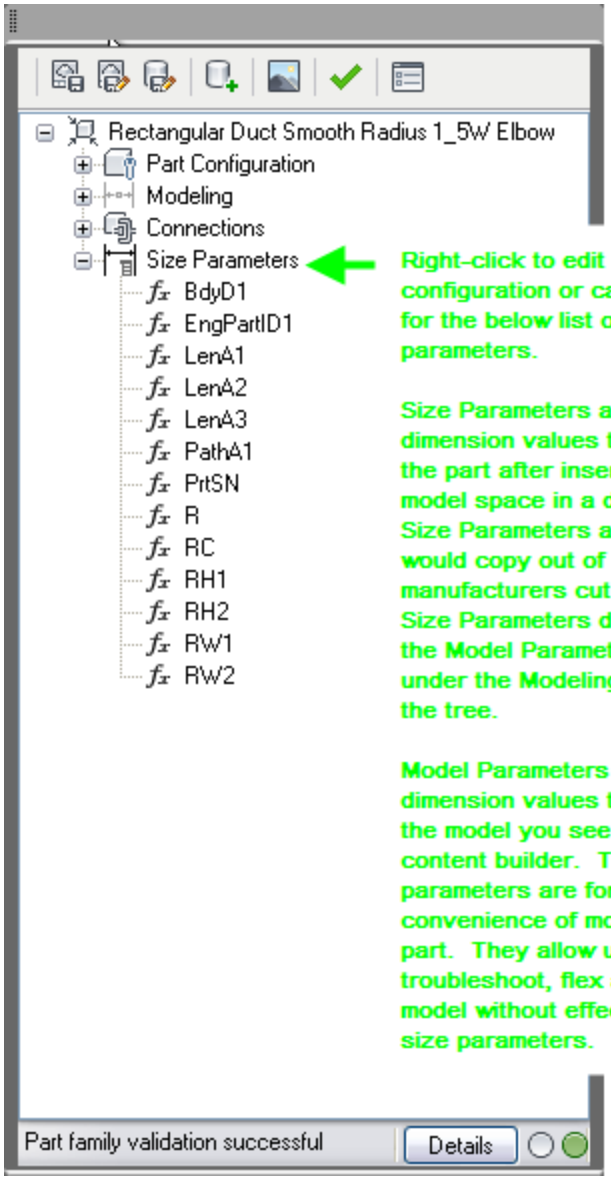
also define a property set that attaches to our part automatically upon insertion. See the steps given in our discussion on the block based part wizard.

The content builder workflow is to start with “part configuration,” then work on “modeling,” then “connections” and end on “size parameters.” Certain choices that you make in part configuration, drive options that are available in later sections. It also affects the part validation. If you select a “tee” part type, it’s going to expect 3 connectors added later in the connections section. If you validate the part, it’s going to fuss until you have 3 connectors. You cannot add connectors till you have a 3D body, which you build in the modeling section. We have to start with the part configuration, then build a model plus temporary model dimensions, add connectors to the model, add values that drive the final part dimensions, called size parameters.






Rectangular Duct Smooth Radius 1_5W Elbow
Part Configuration
Modeling
Connections
Connector 1
Connector 2
Size Parameters
We right-click here and add as many connectors as we need.
Part family validation successful
Details

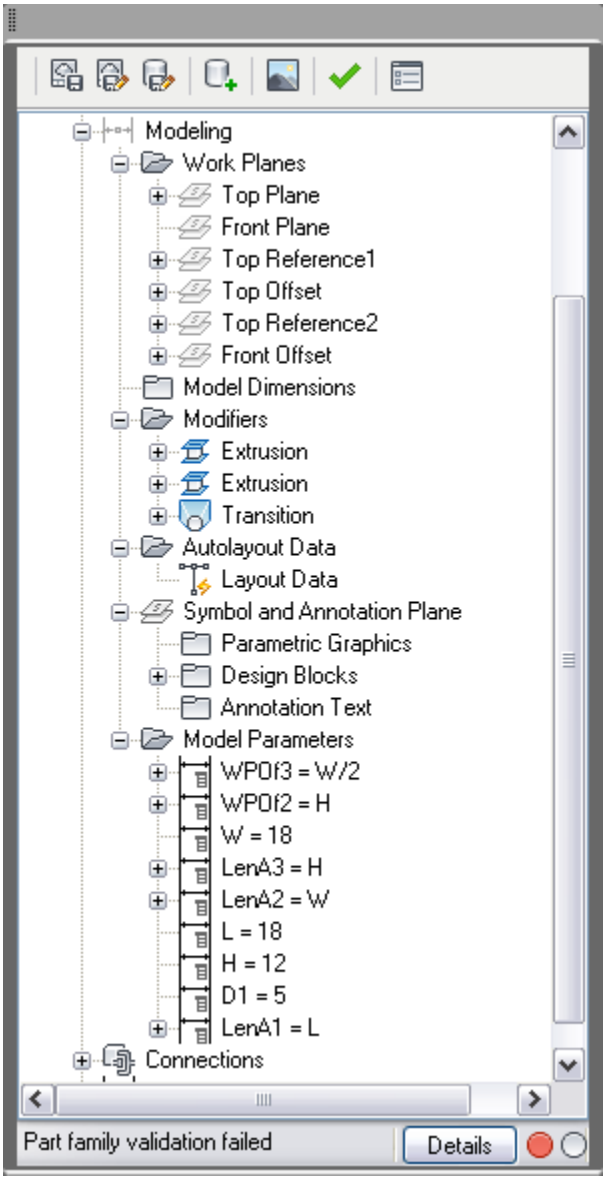

Rectangular Duct Smooth Radius 1_5W Elbow
Part Configuration
Modeling
Connections
Size Parameters
BdyD1
EngPartID1
LenA1
LenA2
LenA3
PathA1
PrtSN
R
RC
RH1
RH2
RW1
RW2
Right-click to edit the values, configuration or calculations for the below list of parameters.
Size Parameters are the dimension values that drive the part after insertion into model space in a drawing. Size Parameters are what you would copy out of a manufacturers cut sheet. Size Parameters differ from the Model Parameters found under the Modeling section of the tree.
Model Parameters are the dimension values that drive the model you see here in content builder. The model parameters are for convenience of modeling the part. They allow us to troubleshoot, flex and test the model without effecting the size parameters.
Part family validation successful
Details

Part Configuration

Let's start with Part Configuration settings. There is the Part Name, the catalog type Aka. Part Domain, Part Type and Part Sub-Type. The Part Name will match the part name. You can double-click it to edit the part name. The part domain was set in stone from the Getting Started screen. The Part Type and Sub-Type has a drop down list of available types. You cannot add your own types or sub-types, but feel free to leave one undefined. Older versions of AutoCAD MEP allowed us to edit the sub-type but we found this could lead to issues. It is fine to select the closest part type or subtype even if it doesn't match exactly. What we are defining here is not a classification, or layer key, but a behavior. Some examples: Does your part behave like a valve and break into a pipe segment, or does it sit inside anchoring into the segment. Does it behave like a tee or an elbow. Think in terms of software, not reality. We can care less that a chiller cools water and a boiler heats water. So we are not asking that kind of "behavior" questions. We just want to know how the part should behave when inserted into a drawing file. How many connectors? Does it turn a corner like an elbow? Does it assist a duct or pipe to transition in size? Does it move with the pipe segment like a valve, or does it anchor the segment down in position, like an immoveable boiler. If you are having difficulty picking a part type or sub-type, look at some of our out of the box parts. Open an out of the box part in parametric part builder and see how we classified them.

Modeling

Modeling can be a complex process to teach or learn, but like a bicycle once you catch on it's easy. I'm going to describe the concepts of this section and it gets deep. At the end, I'm going to step us thru an example. The example really drives home the use, the workflow, and the tools used to make our model. You may wish to do the example first, and then read these concepts.



For a Modeling workflow, start at the top with work planes, work on model parameters, then on modifiers. I have never used Model Dimensions and do not recommend their use. We create dimensions on the planes themselves. If we were to expand on “Top Plane” in the tree above, we would see another folder for “Dimensions.” That is where we work with dimensions, and is why you don’t see anything under “Model Dimensions” in the image above. AutoLayout Data is another one we don’t ever work with. Content builder will automatically create “Layout Data” for us after we create our Connectors. So it’s one category that we don’t need to do anything with. Symbol and Annotation Plane is where we define a plane for which to draw or add a 2D symbol block on top of. Symbol blocks sometimes have text, so the ability is there.

Model Parameters is where we tweak the values of the dimensions we add in the Work Planes section. So every time we add a dimension in the work planes section, it adds a parameter under the Model Parameters for us. We can edit a model parameter's value to make a dimension grow or shrink to that value. If we want to remove a value and replace it with a formula we can do it here. Model Parameters are the initial parameters that drive the model we see in content builder. They are a way for us to play with the model dimensions without changing the actual dimensions used. It decouples our testing activities from the values that drive the final shape and size.

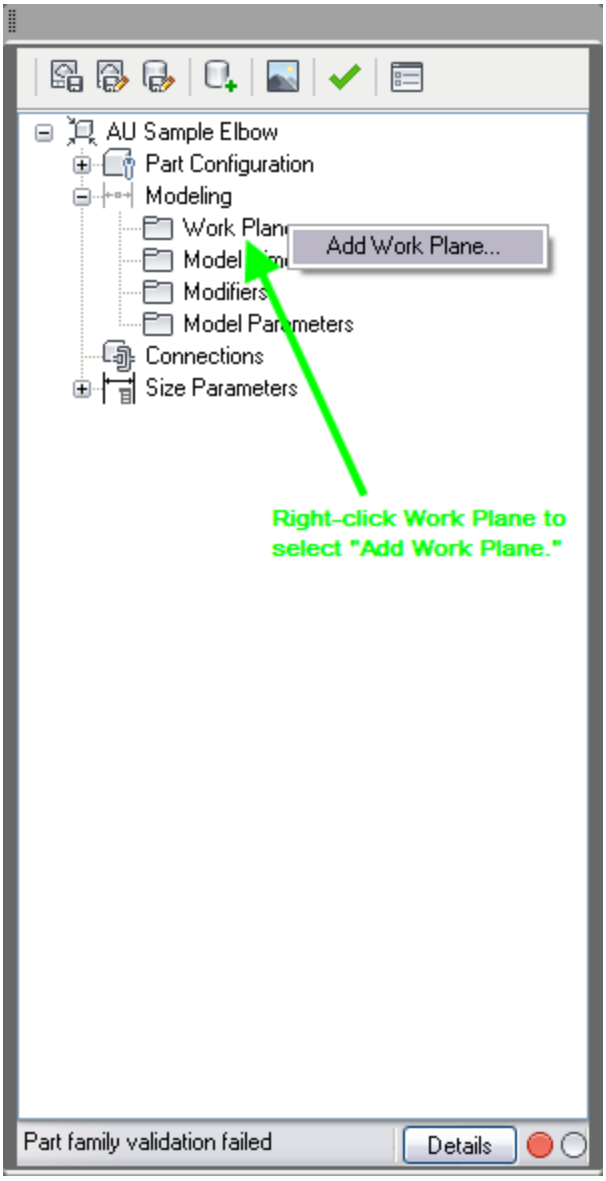
Model parameters are here for our convenience in modeling inside content builder. When we are first modeling our part, we just want to rough-in the size and shape. We want to change the dimensions often, to see what the model does. What happens after we make a change? Does the model behave or break? We don't want to refer to a manufacturer's cut sheet for values, every time we change a value. When we are laying out lines and dimensions, we just want to slap some dimensions down in the approximate shape and fill in the cut sheet values later.

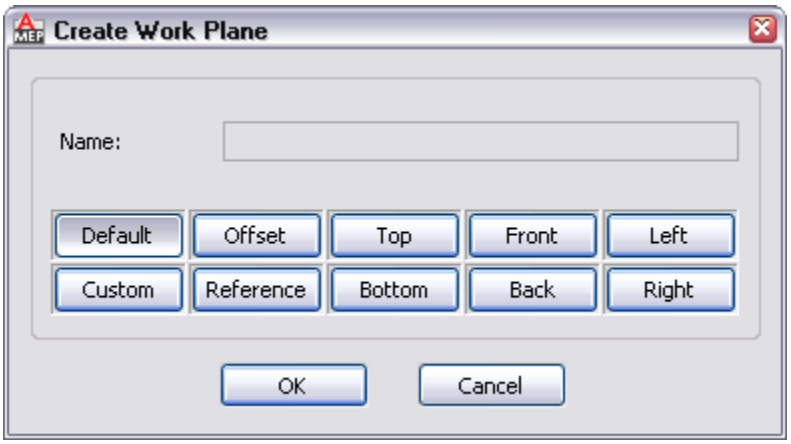
Model parameters allow us to be sloppy, act quick to rough-in the framework of our model. They allow us to test our model as we change the model values and watch the model grow and shrink, and break. Yep, we want to break the model so that we can learn and correct. It is not always obvious when we break a model. Most common break is discovered when we go to change the value of a model dimension, nothing happens. Your model should change size or shape when you change a dimension value. If nothing happens, the model is over-constrained. Too many dimensions. Some other dimension is driving the size of your model.

Size Parameters are the final values. They are the cut sheet values from a manufacturer. They should not change much once entered. They drive the same dimensions, but only after we insert the part in the drawing file after we build it in content builder.

Work Planes

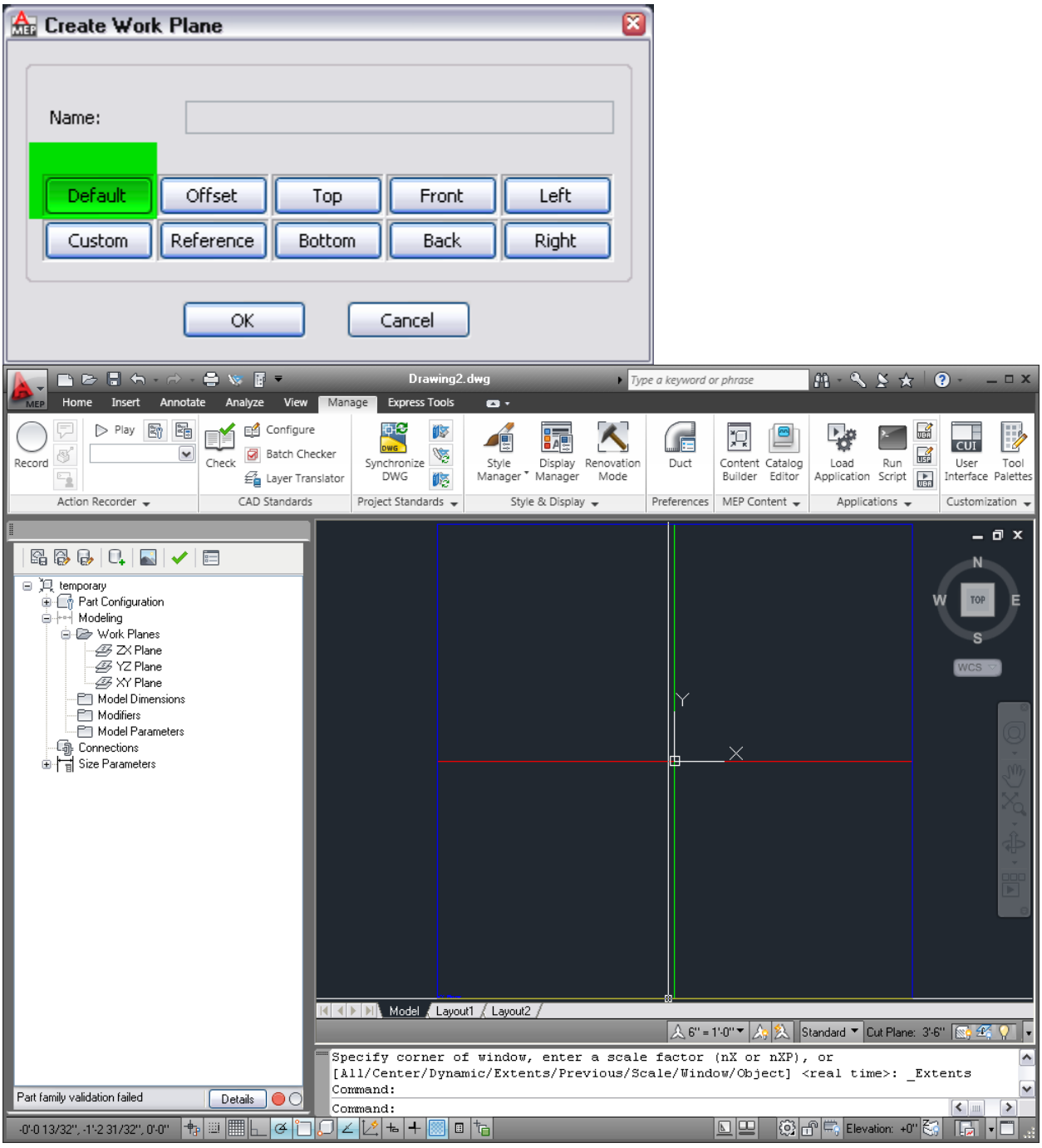
Let's discuss the work planes in depth under the Modeling section.



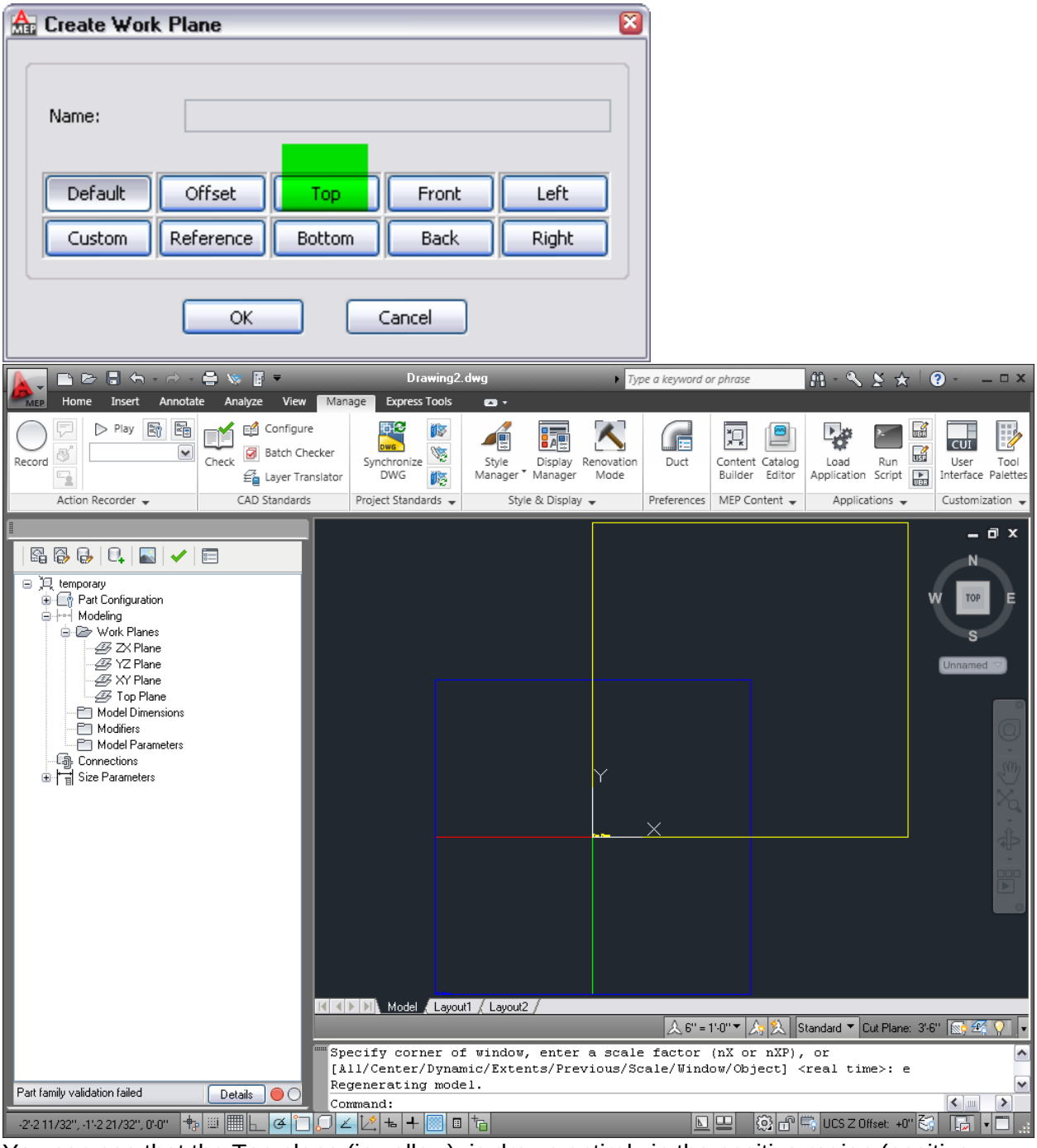


We must first learn how to create work planes and learn what they are. Creating them is easy, learning what they are is more difficult, but easy. The planes are like a surface that we can draw on. The planes are flat, so you cannot draw a 3D object within the plane. We can draw points, lines, rectangles, ovals, circles and any other 2D flat object like dimensions. What do we do with these flat objects? Extrude them along a path (Add Path), or in a direction straight out from the plane (Add Extrusion), or from one plane to another (Add Transition). These are called Modifiers, and there are three types. Add Path, Add Extrusion and Add Transition.

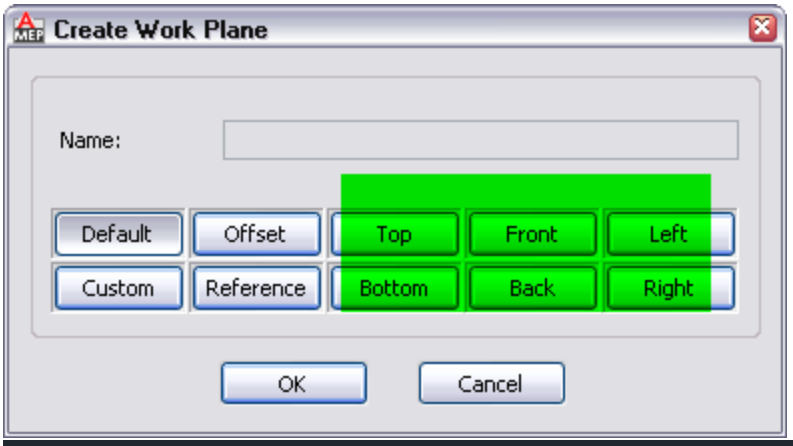
So let's focus on how to create a plane. The following images are fairly self explanatory. Click on the button highlighted in green and see the plane created in the next image.

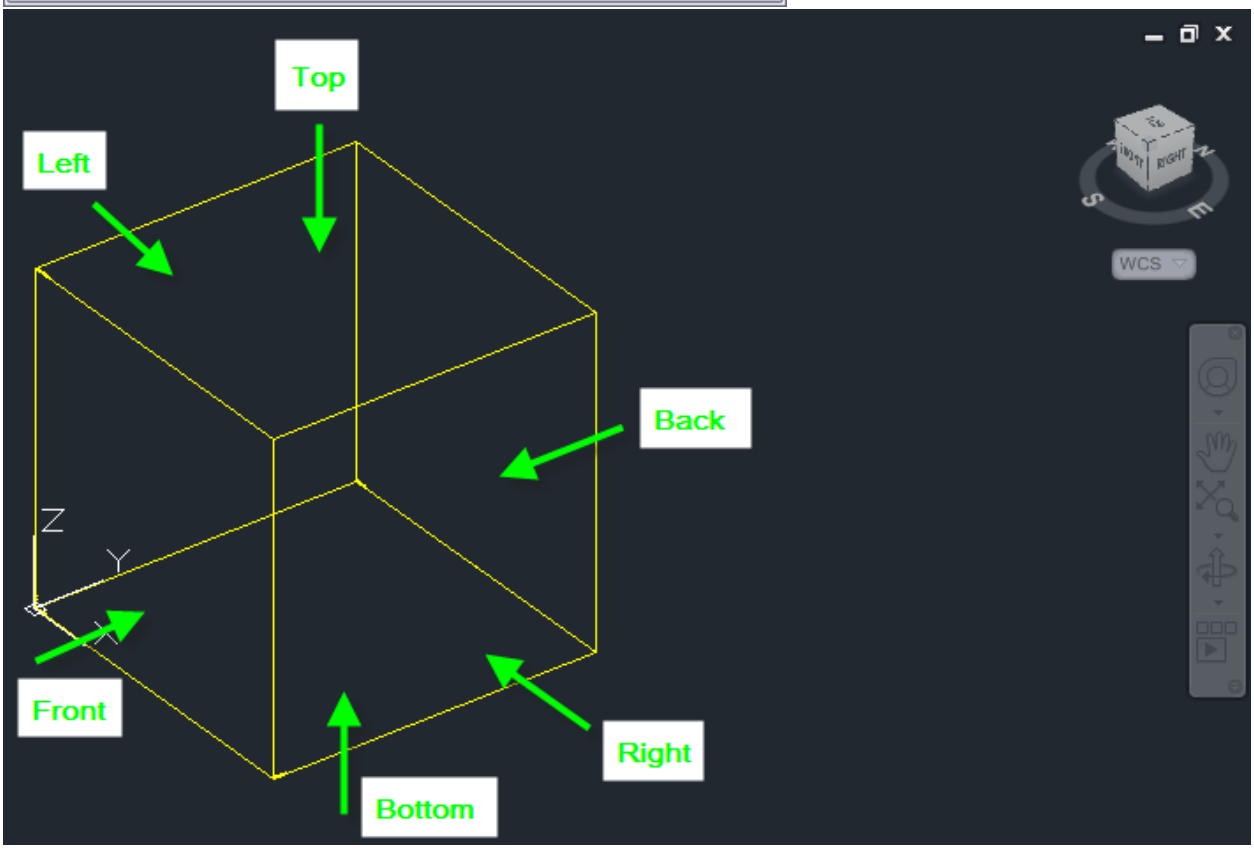


These planes created by clicking “Deafult” are the X-Y plane of the WCS, the X-Z and the Y-Z planes. The planes in content builder are represented by a rectangle. The rectangle gives us a visual clue as to the location and orientation of the plane, but not to the size. These planes go on forever in their respective directions. So don’t let the rectangle fool you, it’s not a boundary. I’ll also caution you about using the “Default” planes. At one point, we had an issue in content builder where our parts would get corrupted if you drew the model in the negative portions. For example, in the image above, you see the origin dead center. The positive X-axis is to the left of the origin, and the positive Y-axis is above the origin. The negative sections (negative x-axis and negative y-axis) are to the left of the origin or below the origin. Don’t model in these areas. We haven’t seen any issues in many releases, but never hurts to be safe, right? Notice that the “Default” rectangles representing the planes are centered on the origin. So I don’t like using the Default planes, and don’t recommend them. Let’s explore the other options.

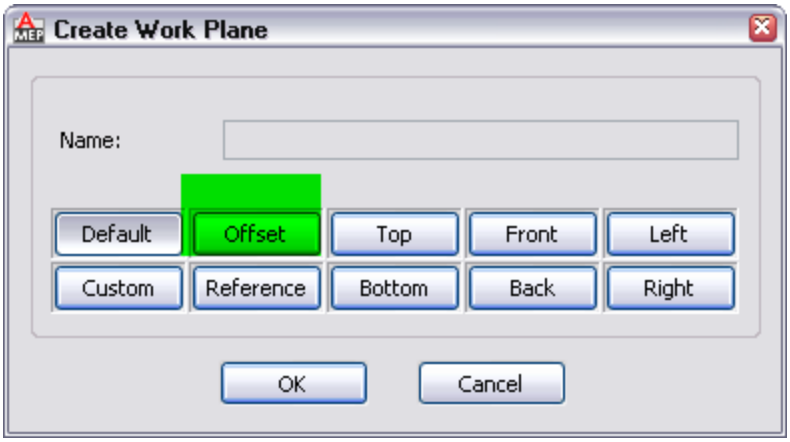


You can see that the Top plane (in yellow), is drawn entirely in the positive region (positive x-axis and positive y-axis). Use this plane instead. This plane is also 30 inches or units straight up (in z-axis) from the origin. No worries, the UCS in content builder has NO relationship to the UCS of your part after insertion, NO relationship to the part insertion point. That rectangle, by the way, is 30 units per side.

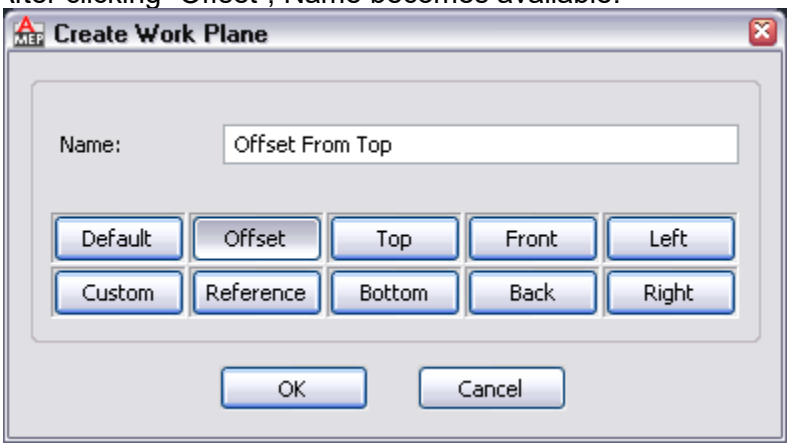




I have deleted the default planes, added the other planes highlighted in green. We are no longer looking down from Plan view, and switched to a SE Iso view. This cube is the other work planes, top, bottom, left, right, front, back. The cube dimensions are 30” x 30” x 30”.



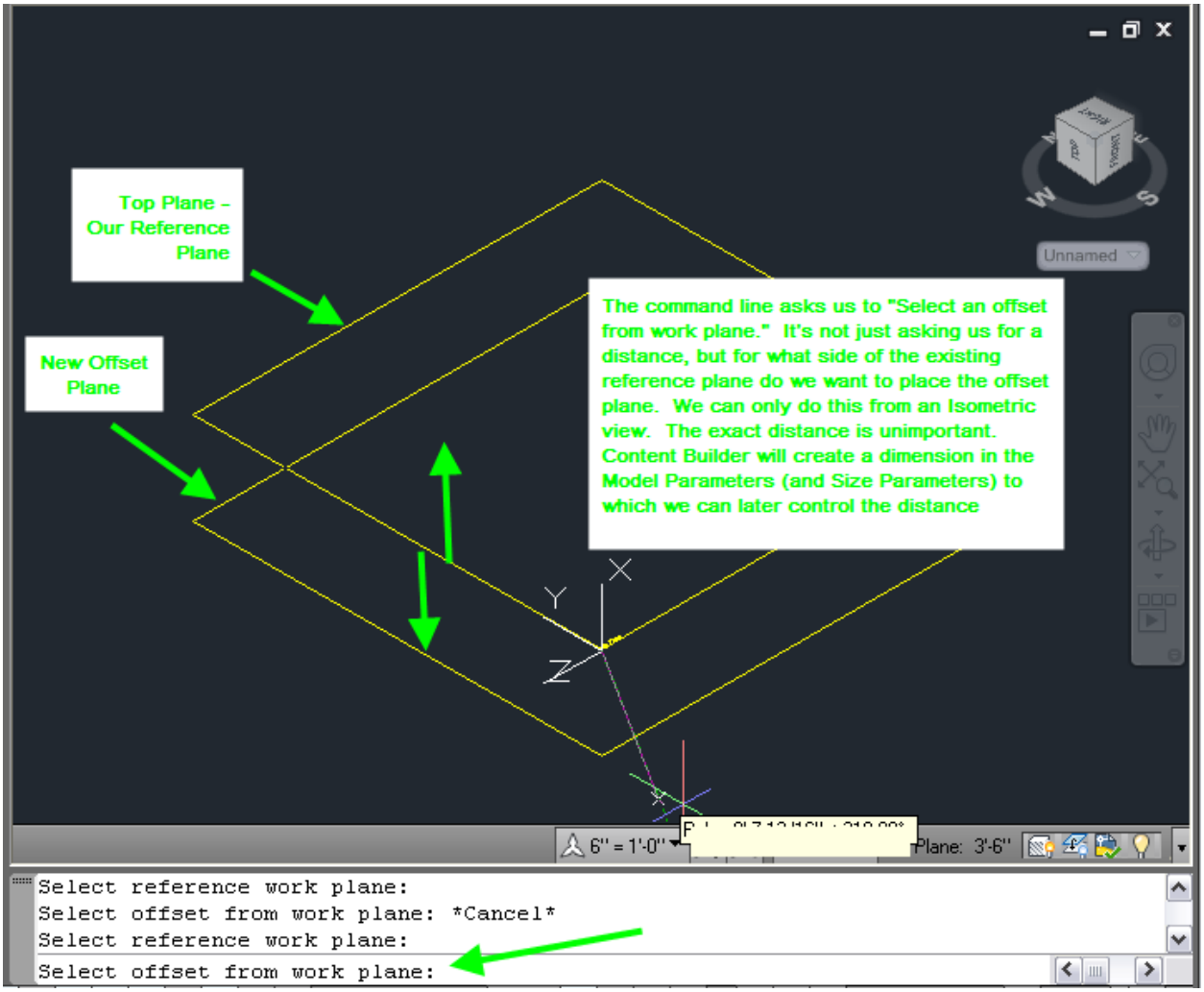
After clicking “Offset”, Name becomes available.



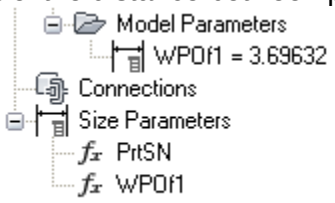
After clicking “Ok”, the command line prompts us to select an existing plane as a reference to create an offset plane.

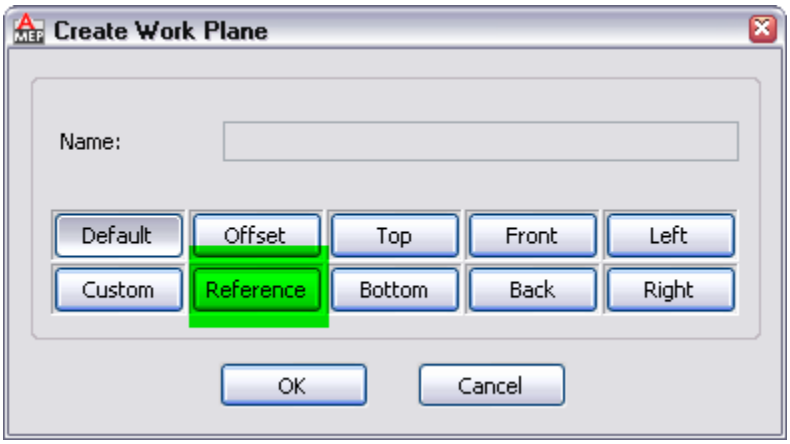
```
Resuming ADDCOLWORKPLANE command.
Select reference work plane:
Select offset from work plane: *Cancel*
Select reference work plane:
```

At this point, if you are not already, switch to a SE Isometric view. Use the view cube, or switch to the View tab of the Ribbon. The views do work within content builder.

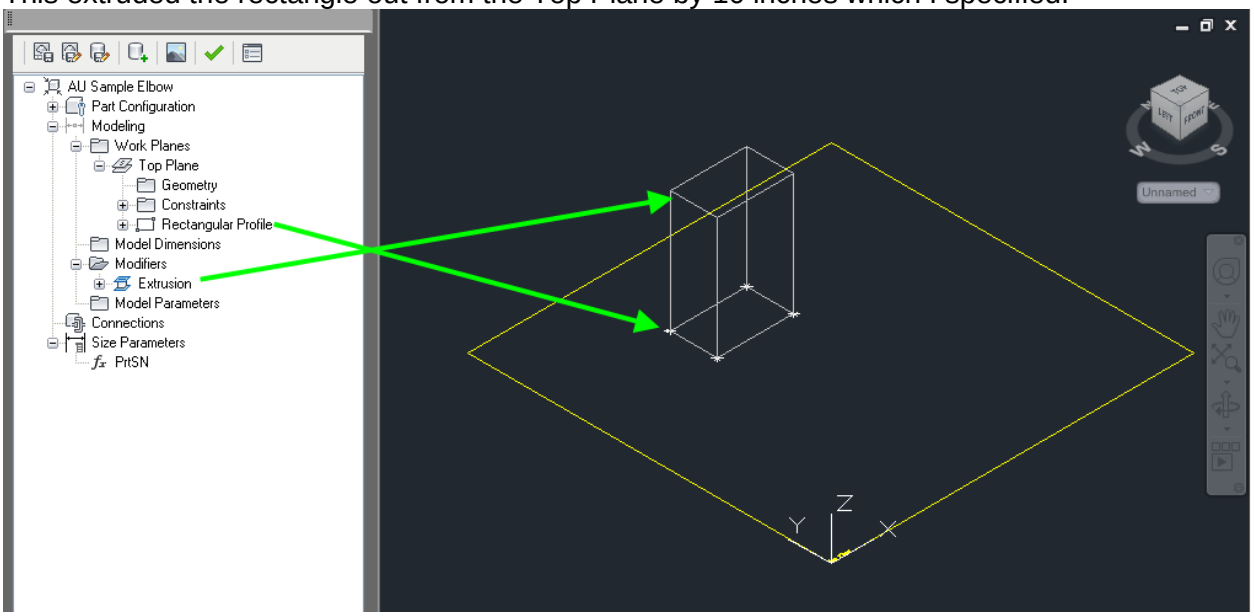


The command line asks us to "Select an offset from work plane." It's not just asking us for a distance, but for what side of the existing reference plane do we want to place the offset plane. We can only do this from an Isometric view. The exact distance is unimportant. Content Builder will create a dimension in the Model Parameters (and Size Parameters) to which we can later control the distance between planes.

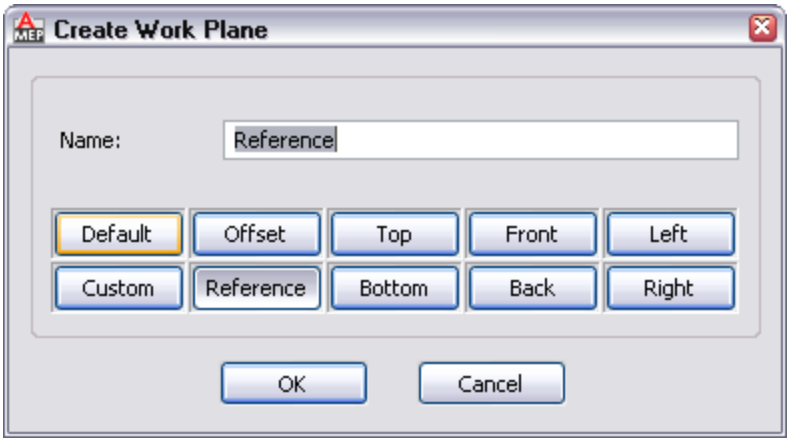




In order to understand a Refernce plane, we need a Modifier. In the following image, on the Top plane, I added a rectangular Profile. From this profile, I added a Modifier "Add Extrusion." This extruded the rectangle out from the Top Plane by 10 inches which I specified.



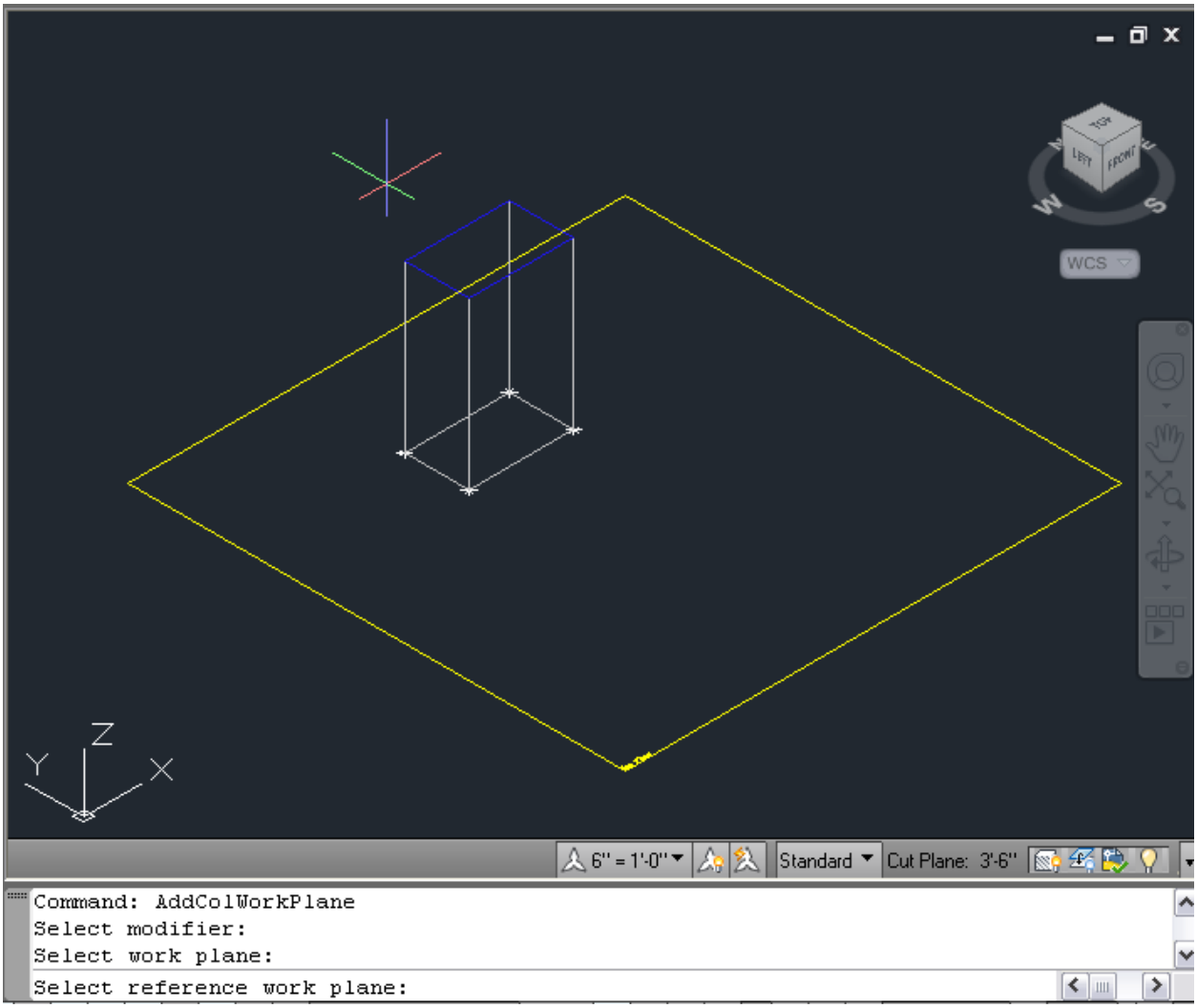
We need the Modifier and another plane prior to making a Reference plane. Reference planes are not created often, so since this is more difficult to make, it's used infrequently. Creating an offset plane is a much easier method.



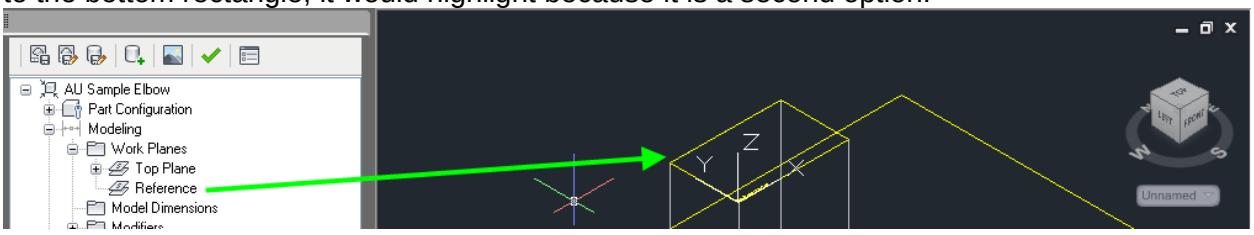
After we click the Reference button, the Name field becomes available.

The command line asks us to select our Modifier.

The command line next asks us to select a work plane. It will generate a Reference plane parallel with whatever work plane we select here.

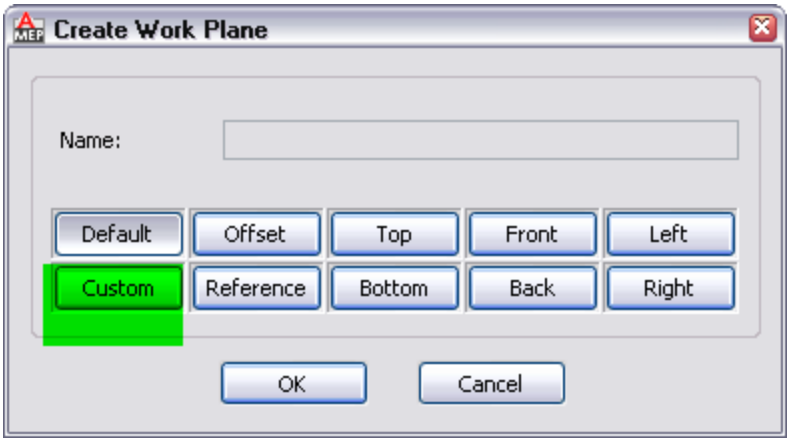


While it asks us to select a reference work plane, the surfaces on our Modifier, that are parallel with the work plane we previously selected are highlighted in blue. If I brought my curser closer to the bottom rectangle, it would highlight because it is a second option.

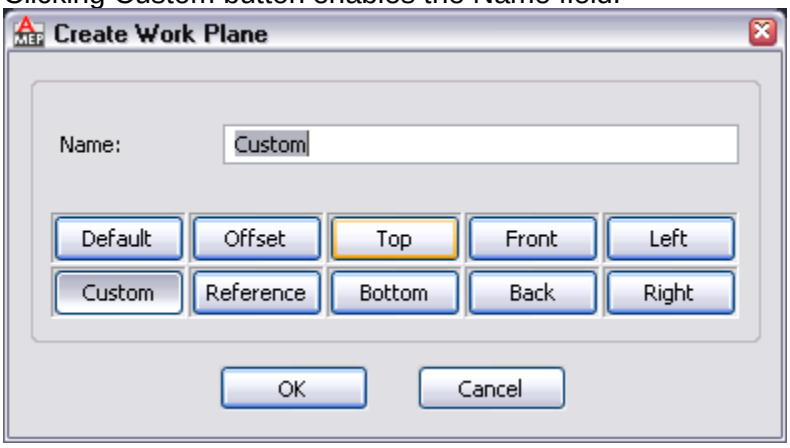


In the above image, you can see the final Reference Plane in yellow. It is attached to that surface of the Modifier. If some dimension, were to change the size of the Modifier, the Reference plane would move with that surface. Hence, the “Reference.”

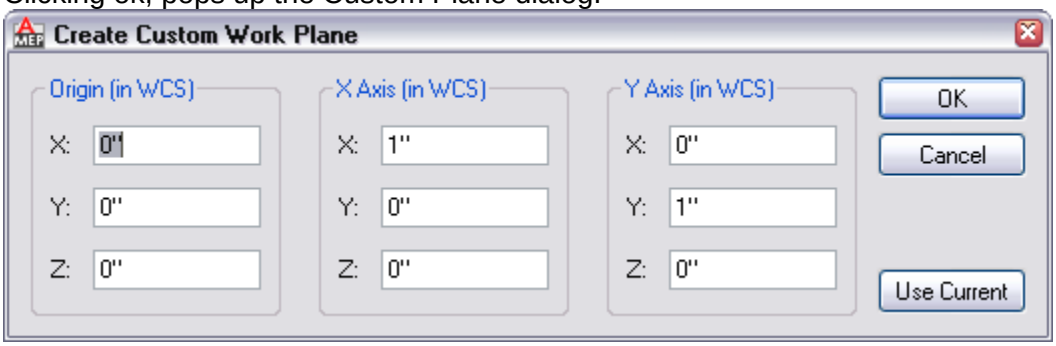
We use an “Offset” plane when we want planes to drive geometry by controlling the distance between two planes. We can use that to drive the geometry of a (Add Transition) Modifier between the two planes. On the other hand, we use a “Reference” plane when we want geometry to drive the location of a plane.



Clicking Custom button enables the Name field.



Clicking ok, pops up the Custom Plane dialog.

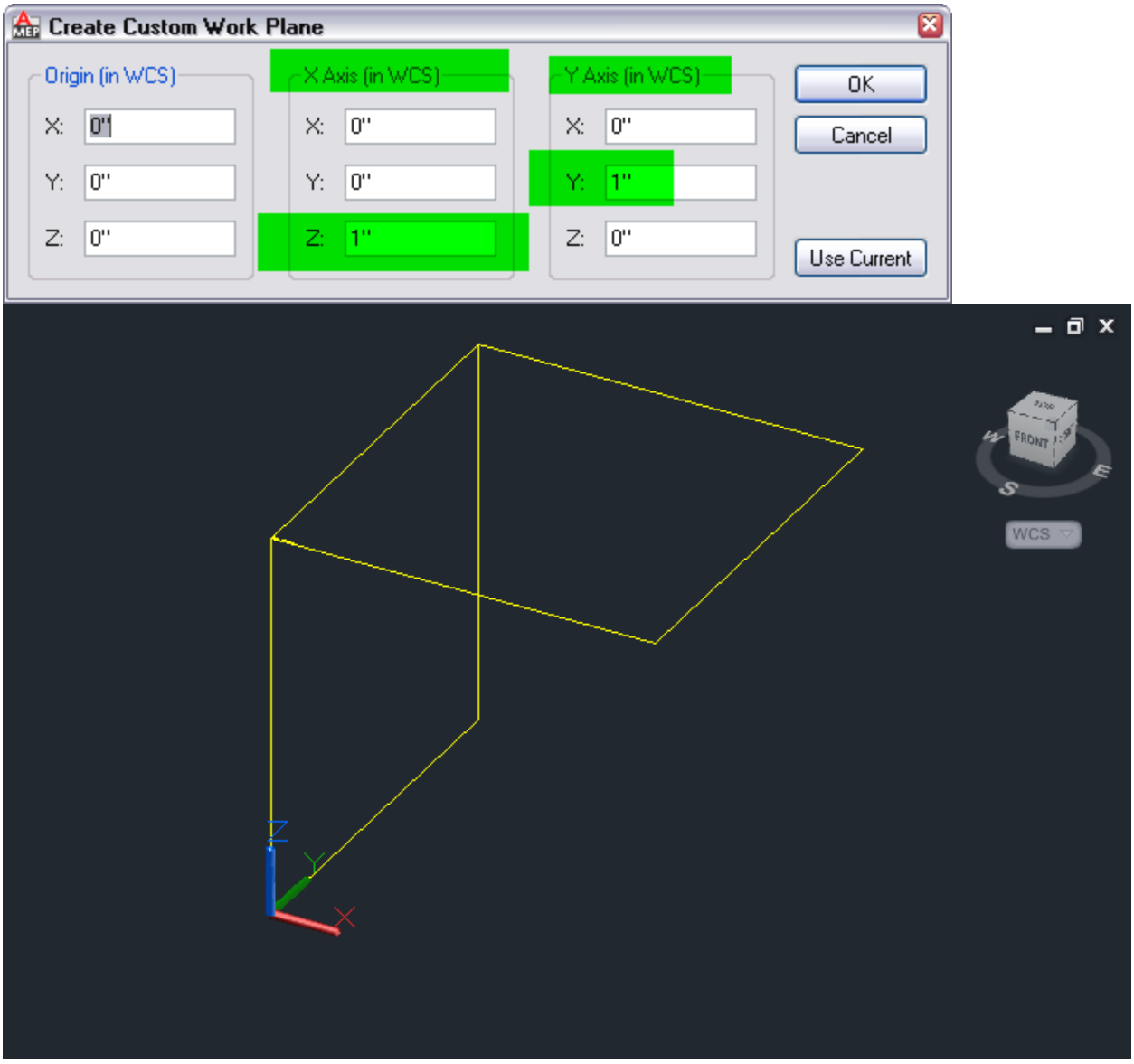


This dialog is really simple. The Origin is relative to the WCS. Tell it the 0,0,0 coordinates to start from. The proposed custom plane has an X-axis, and a Y-axis. In the image above, the X-axis moves 1" in the X, 0 in the Y, and 0 in the Z. The X-axis is pointing in the positive x WCS direction. Likewise, the Y-axis is pointing in the positive y WCS. Our custom plane, the x and y axes must always be perpendicular. We can rotate our plane, by moving our custom plane's x-axis over a little in the x WCS and up in the y WCS. Our custom plane's y-axis must rotate by the same amount, but relative to the y WCS.

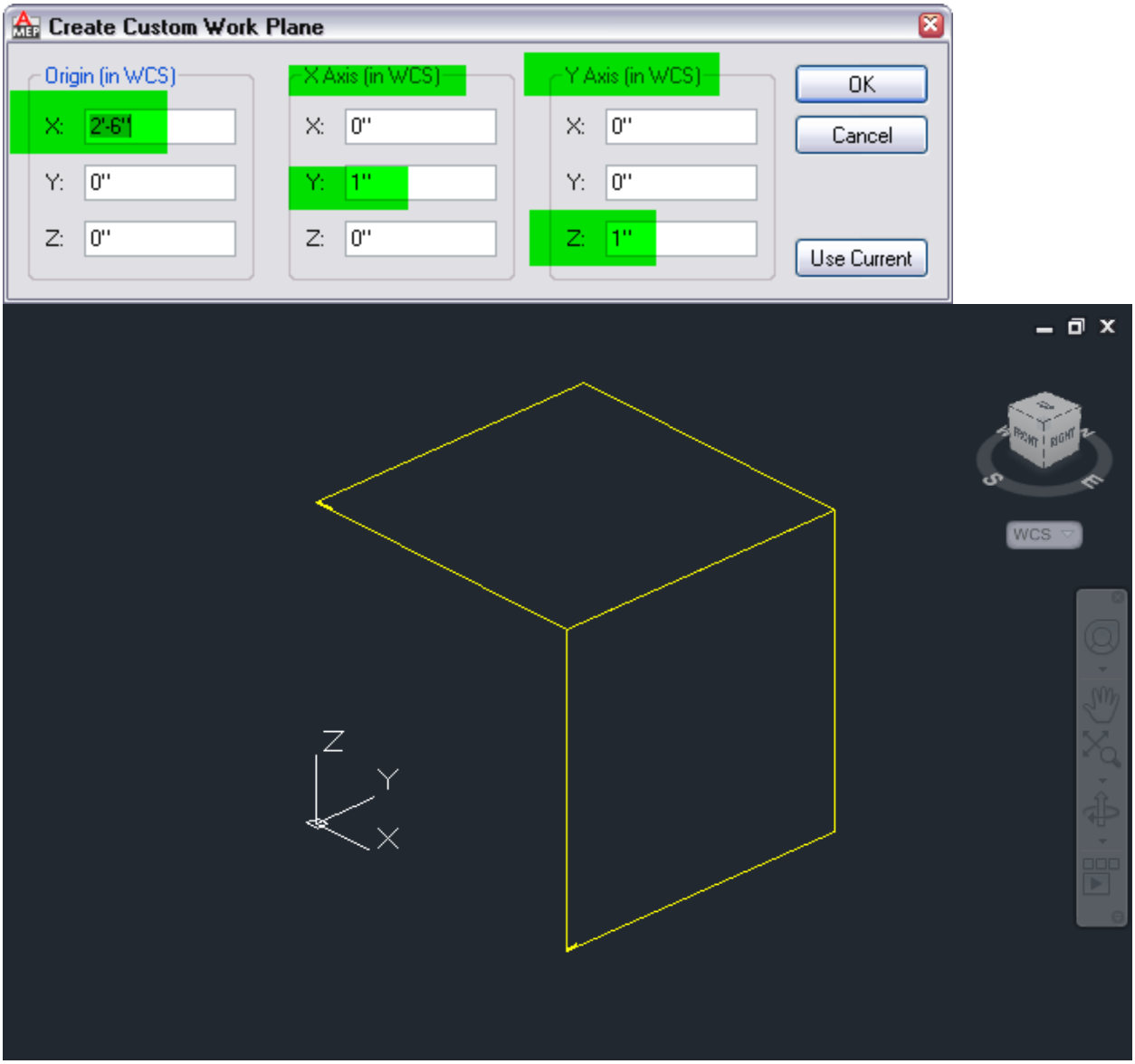
If necessary, find a T-square. Set a T-square on your desk. One leg is the x-axis and the other is the y-axis of our custom plane. It just helps to visualize what is happening.

If the Y-axis moves up in the z-direction, it rotates about the X-axis. The x-axis does not move, it is still positive in the x WCS direction. But if the Y-axis rotates about the Z-axis, the X-axis must rotate with it. So with a custom plane, we can create a plane at any angle and location. You just cannot move or rotate it after creation. In fact, Offset and Refernce planes are the only plane types that will move after creation by the object or plane+dimension that they refer to. None of the planes can change angle after creation. We have a wish request logged for this.

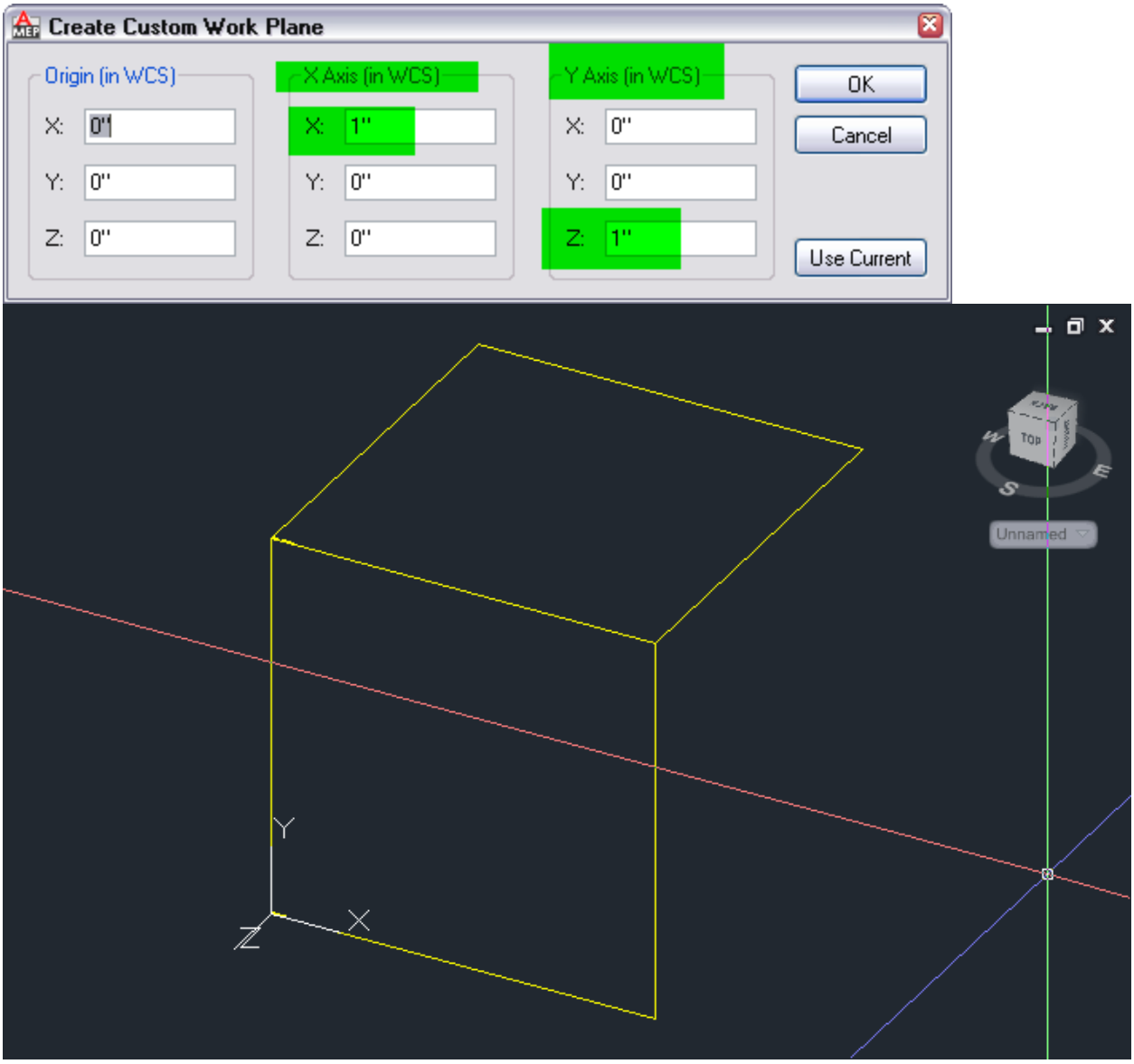
Here are some custom settings and the resulting planes. In the following images, you will see the Top plane as a frame of reference. Don't forget, the Top plane is 30" above the WCS origin in the z WCS direction..



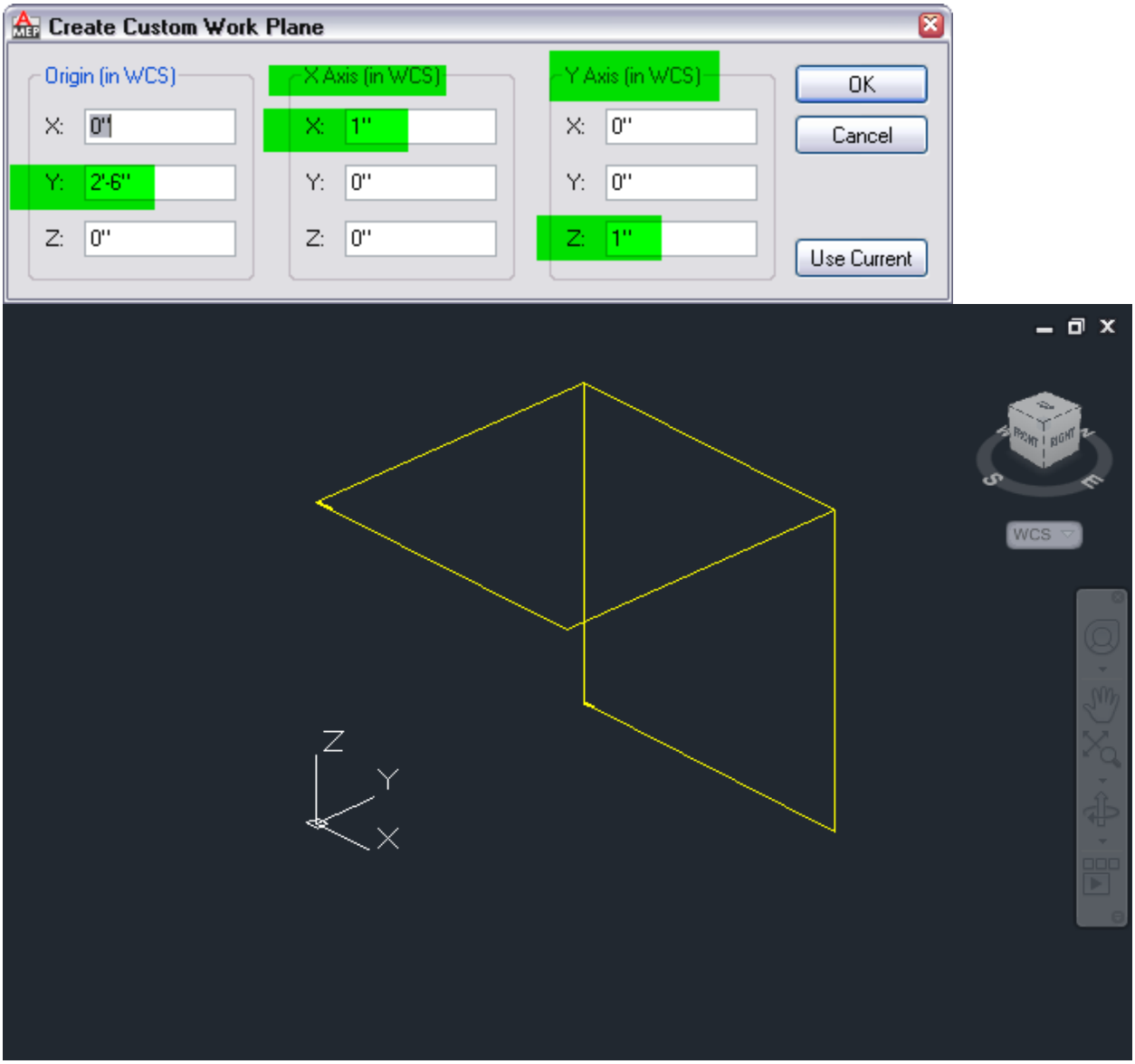
Here we are asking the custom x-axis to go 1 unit in the z WCS direction, straiight up. The y axis still points 1 unit in the positive y WCS direction. (Aka. LEFT)



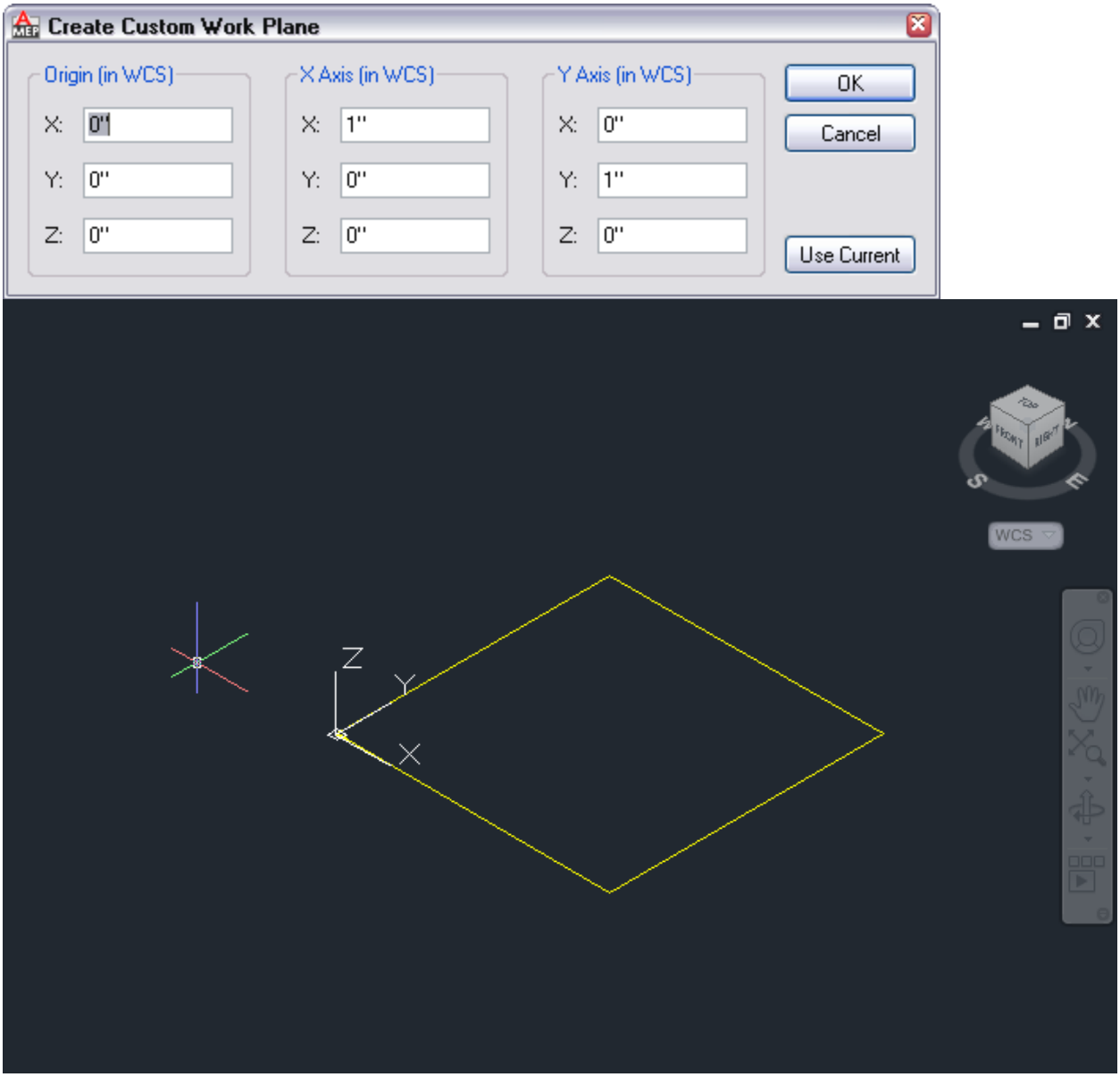
Here we ask the origin to shift 30 inches in the x WCS direction. Same as in prior image, we are asking the custom x-axis to go 1 unit in the z WCS direction, straiight up. The y axis still points 1 unit in the positive y WCS direction. (Aka. RIGHT)



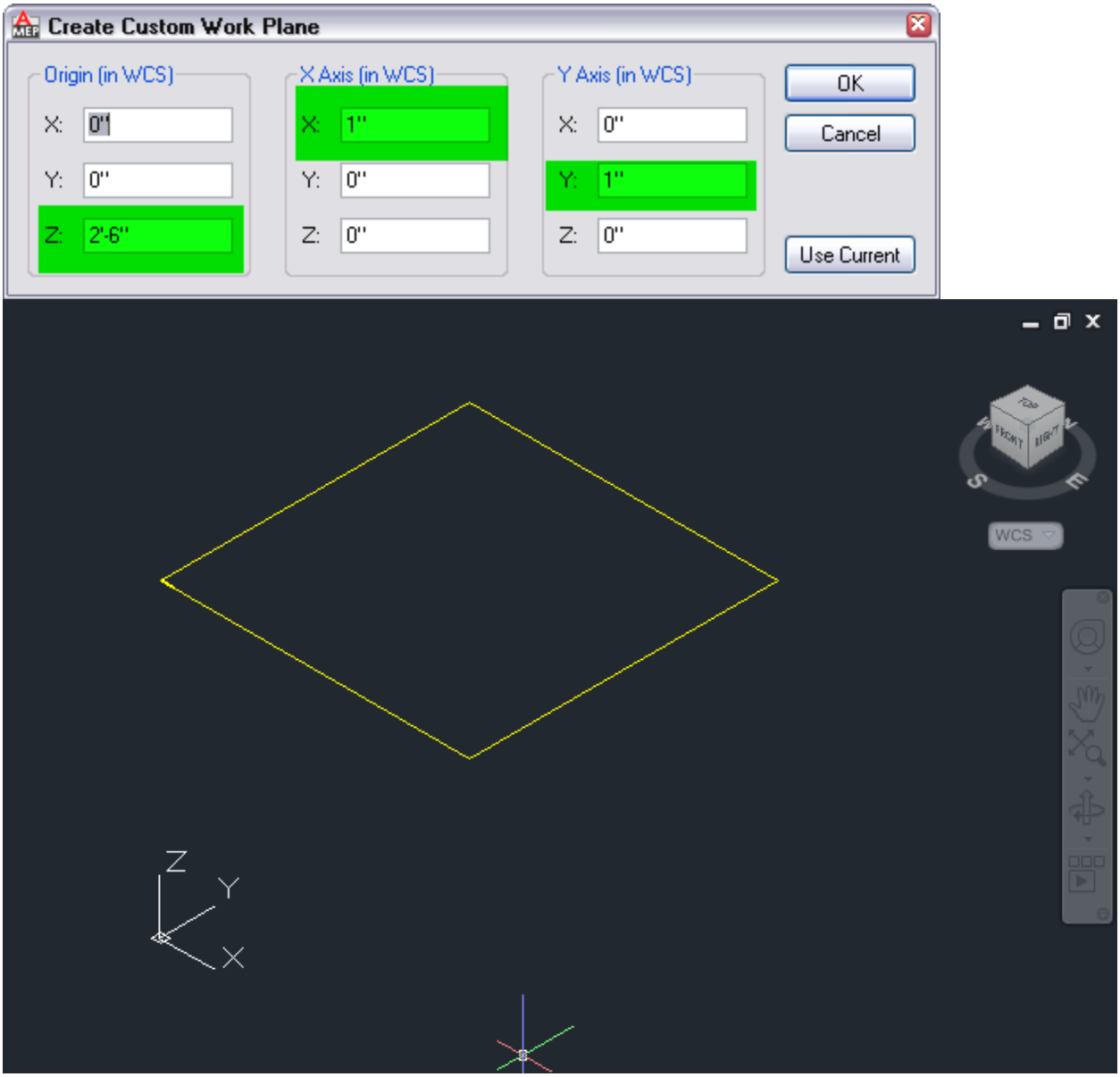
Here we are asking the custom y-axis to go 1 unit in the z WCS direction, straiight up. The x axis still points 1 unit in the positive x WCS direction. (Aka. FRONT)



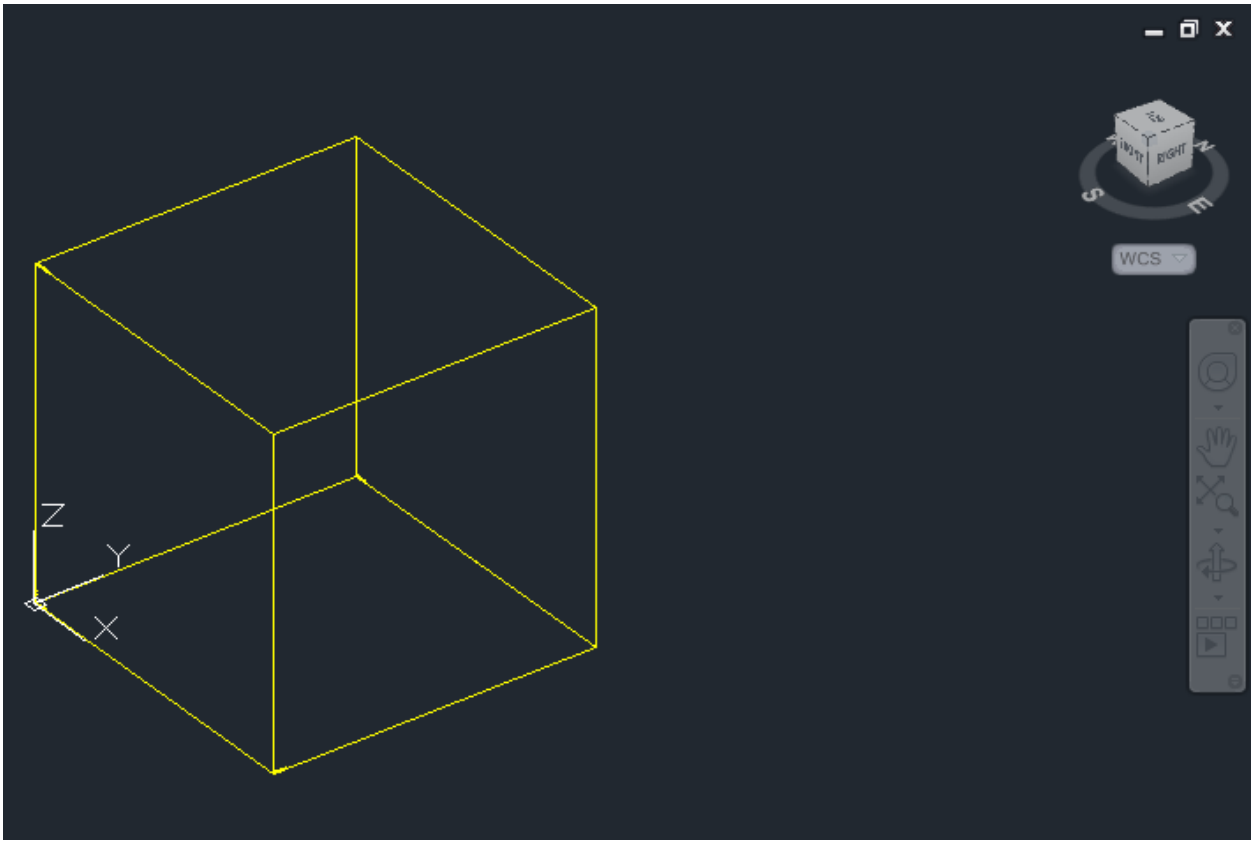
Here we ask the origin to shift 30 inches in the y WCS direction. Same as in prior image, we are asking the custom y-axis to go 1 unit in the z WCS direction, straiight up. The x axis still points 1 unit in the positive x WCS direction. (Aka. BACK)



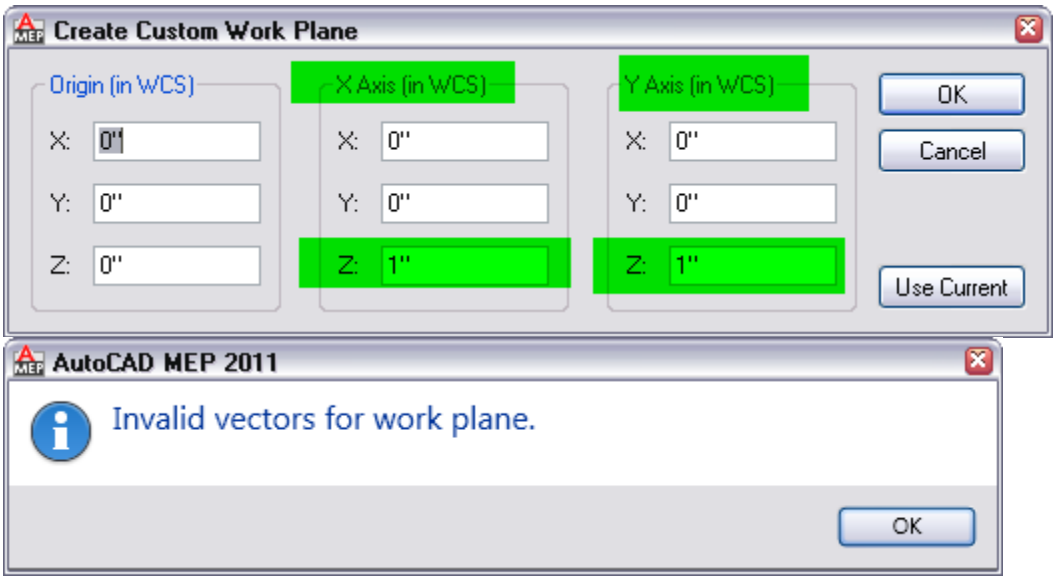
We have seen this one before. It was the default values when we first launched the custom plane dialog. Origin = 0,0,0. "In the image above, the X-axis moves 1" in the X, 0 in the Y, and 0 in the Z. The X-axis is pointing in the positive x WCS direction. Likewise, the Y-axis is pointing in the positive y WCS." (Aka. BOTTOM)



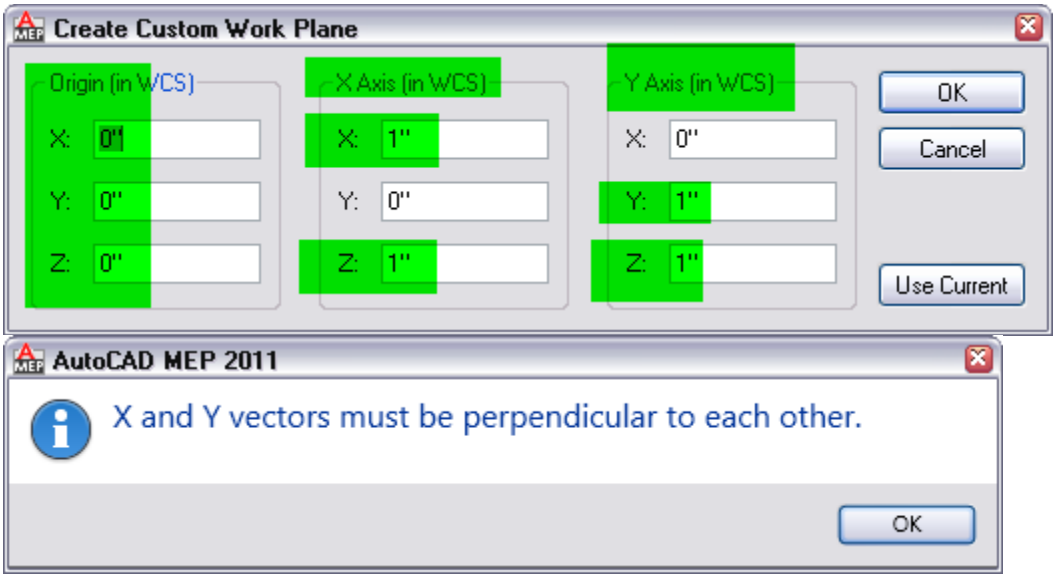
Same as above, but origin moved 30” up in the z WCS axis. Same as in prior image, we are asking the custom x-axis to go 1 unit in the x WCS direction. The y axis still points 1 unit in the positive y WCS direction. (Aka. TOP)



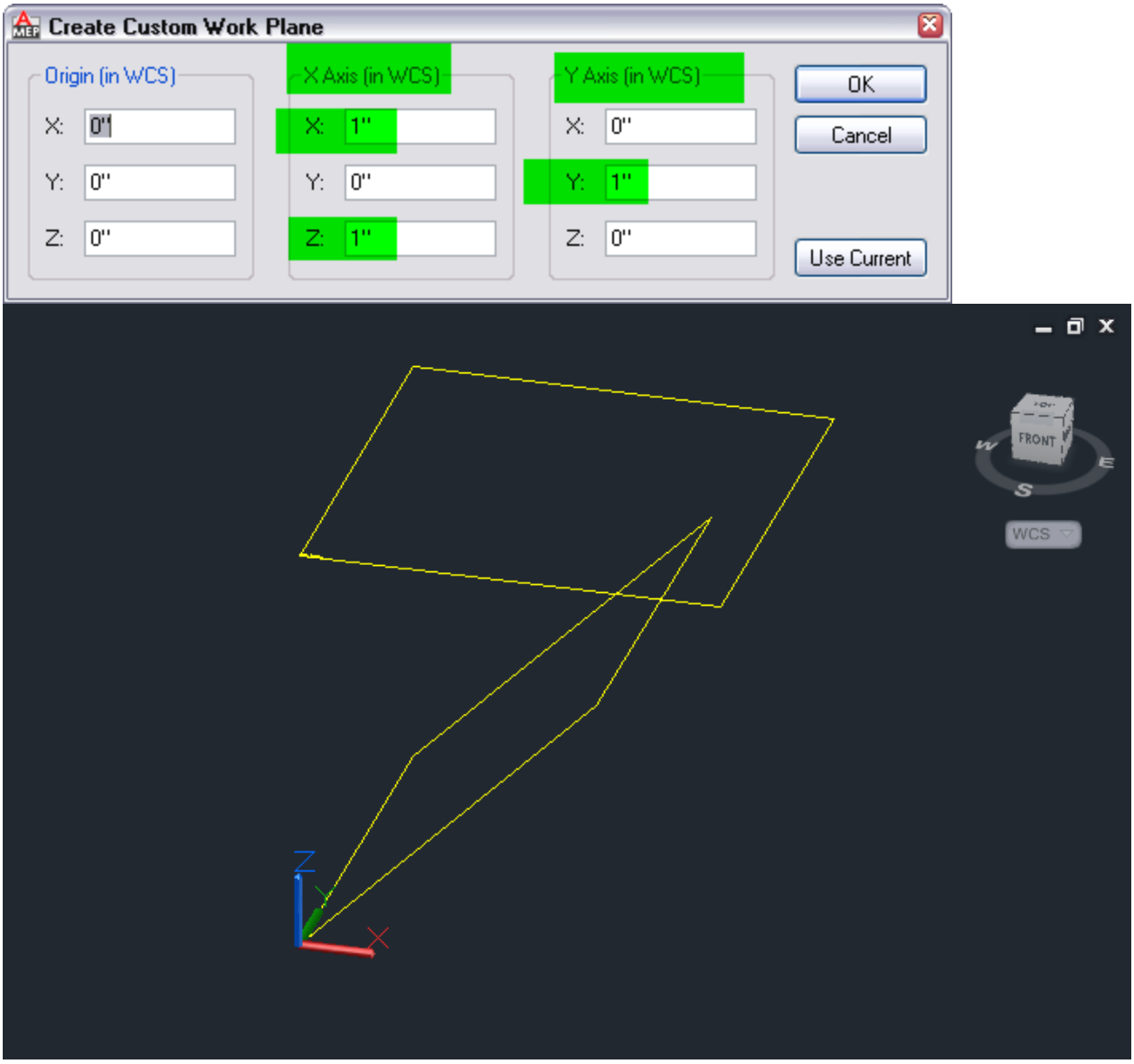
Guess what, we now have enough data to make a work plane for Top, Bottom, Left, Right, Front and Back planes out of Custom planes. Did you notice the Aka labels?



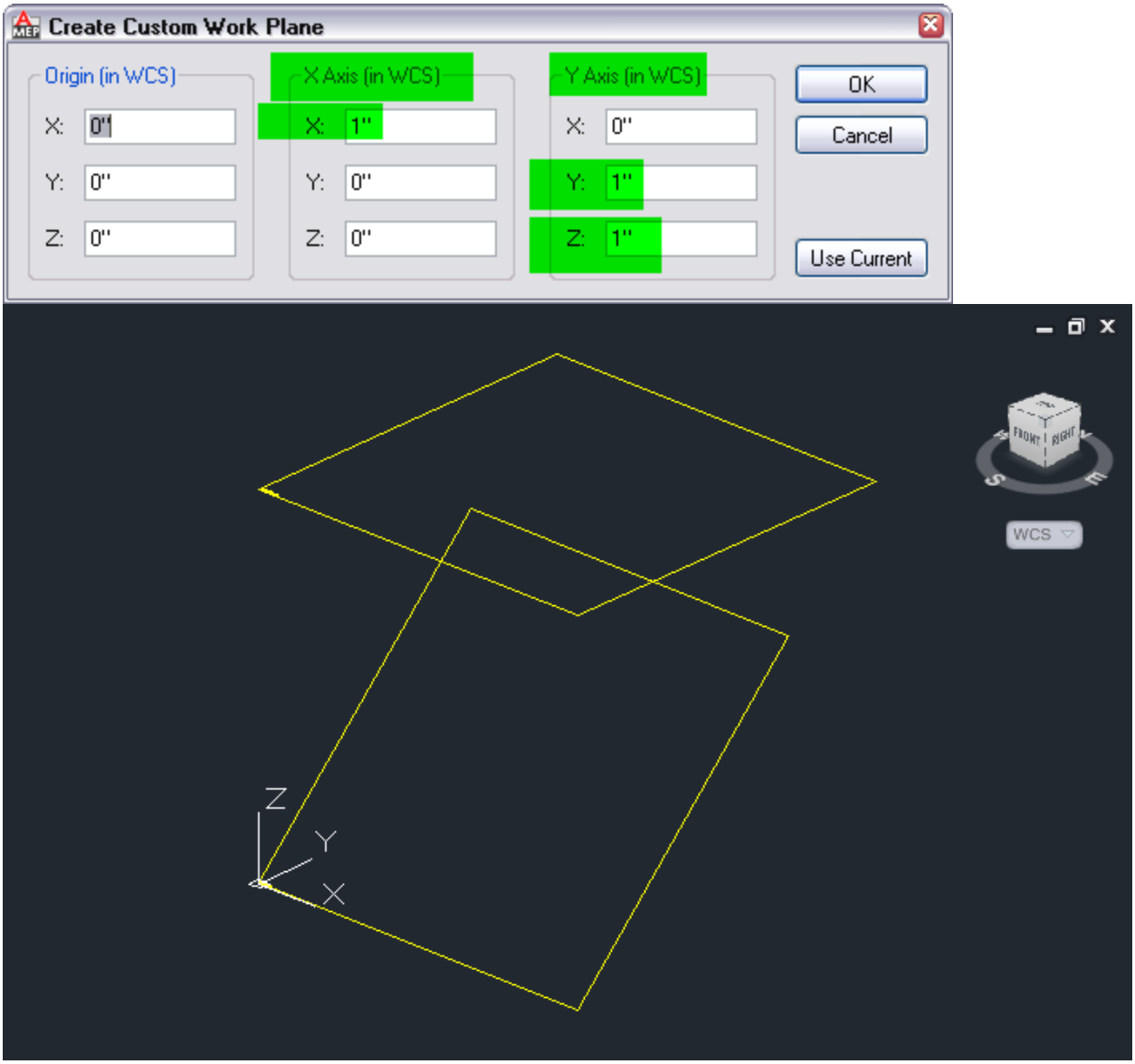
The custom plane's x and y-axes cannot both be pointed in the same direction. They would not be perpendicular to each other.



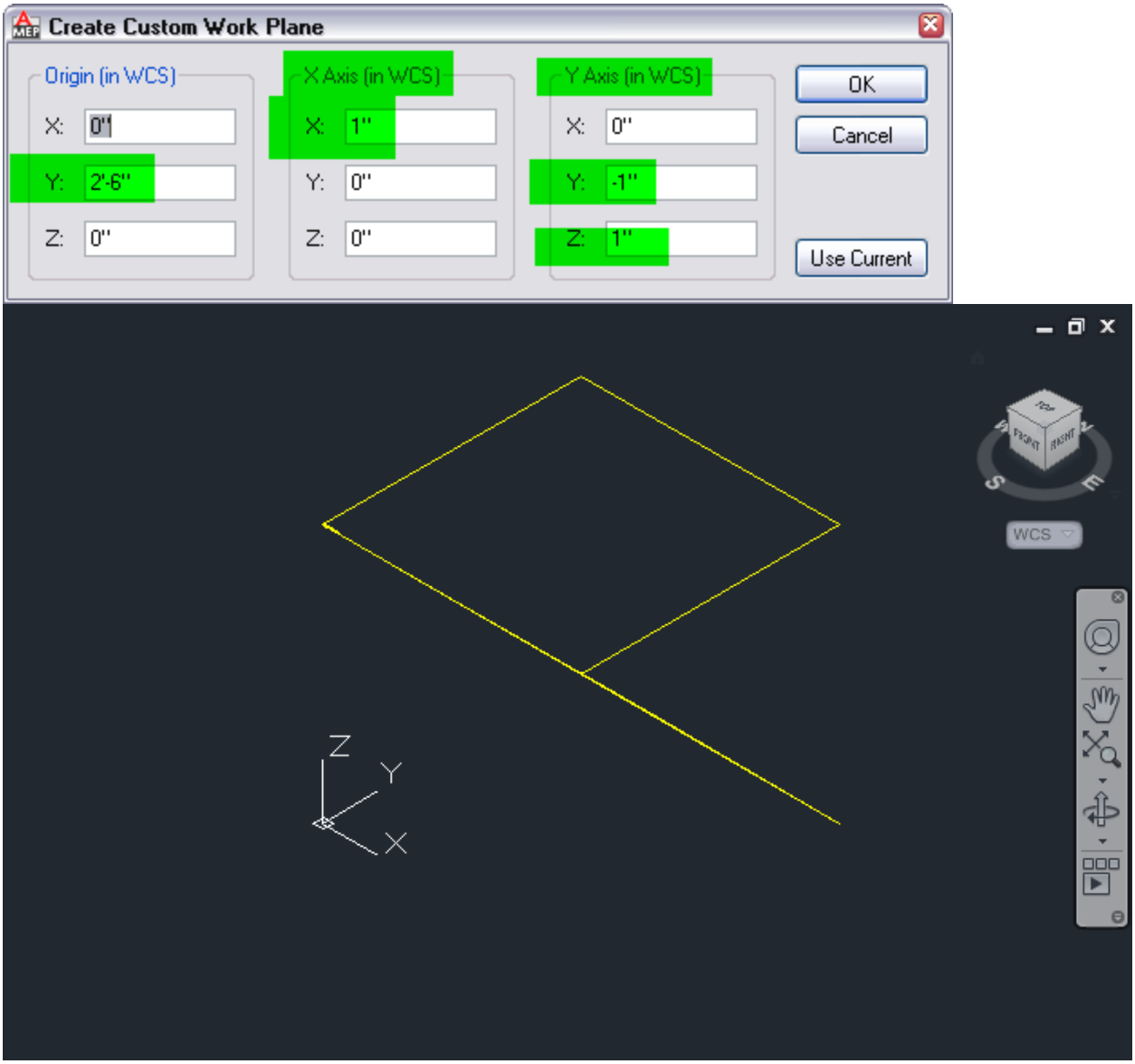
This is where our T-square helps to visualize. Lay it flat and square with the edge of your desk. We ask the y-axis to point a unit in the y direction and up a unit in the z direction, nothing in the x-direction. The T-square rotates about the x-axis as we raise up the y-axis in the z-direction. Now try to raise the x-axis up in the z-direction, at the same time. Your T-square leg that represents the y-axis no longer points down the true y-axis. It begins to wander down the negative x-axis. As we raise up the x-axis, the y-axis rotates into the x-axis a few degrees. So we need to correct the above settings by giving the x: value under the Y Axis (in WCS) a slightly negative value.



Here we are asking the custom x-axis to go 1 unit in the x WCS direction, and 1 unit up in the z WCS direction. The y axis still points 1 unit in the positive y WCS direction.

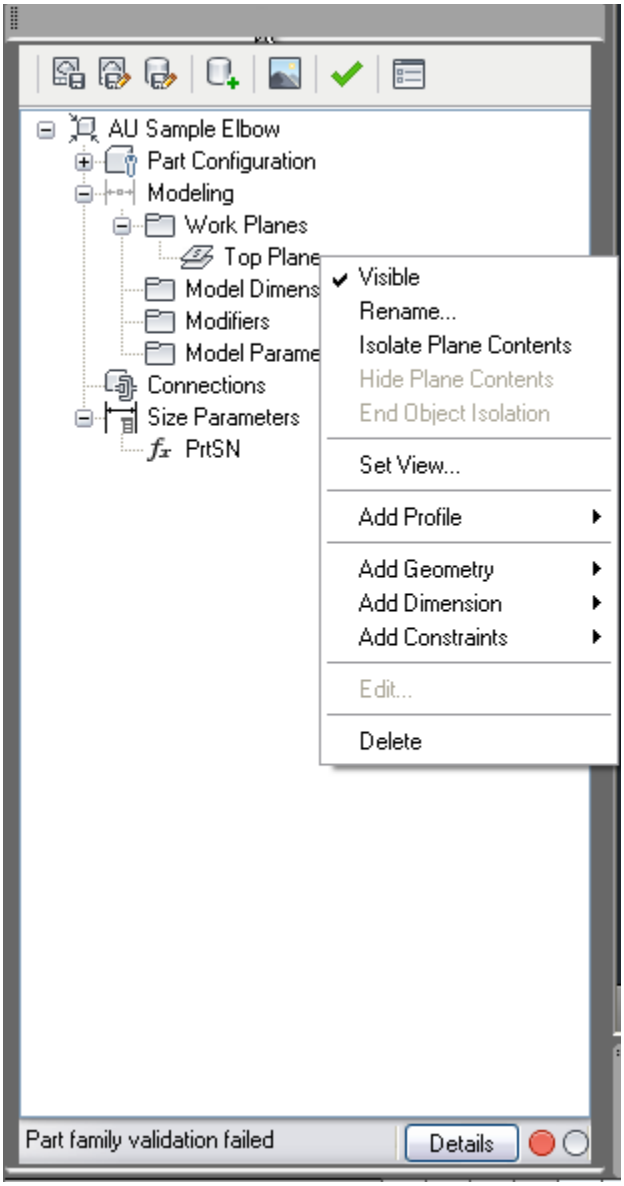


Here we are asking the custom y-axis to go 1 unit in the y WCS direction, and 1 unit up in the z WCS direction. The x axis still points 1 unit in the positive x WCS direction.



Here we are asking the origin to move 30” in the y-direction. We ask the custom y-axis to go -1 unit in the y WCS direction, and 1 unit up in the z WCS direction. The x axis still points 1 unit in the positive x WCS direction.

Are we comfortable with how to create work planes at any location and angle yet? Let’s move on.



When we right-click a work plane in the tree, this is the list of options.

Visible: This controls the visibility of the plane in model space. It's like layer > freeze, but content builder's way. You will want to use this visibility setting when and if you model has too many things in the way. Remember, we cannot use the move, copy, rotate commands from AutoCAD/MEP. So if an object is in the way, shut it off.

Rename: Pretty self-explanatory. Allows us to rename a plane.

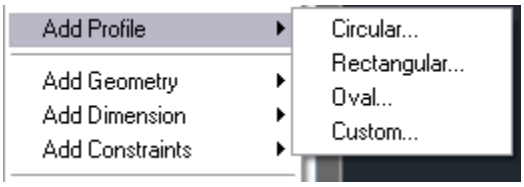
Isolate Plane Contents: Uses Visibility again. It shuts off the Visibility setting on all objects that are not on this plane. All objects on this plane stay on, or maintains their current visibility setting.

Hide Plane Contents: It's the flip of "Isolate Plane Contents." It shuts off the visibility of all objects on this plane. It maintains the visibility setting of all objects outside of this plane.

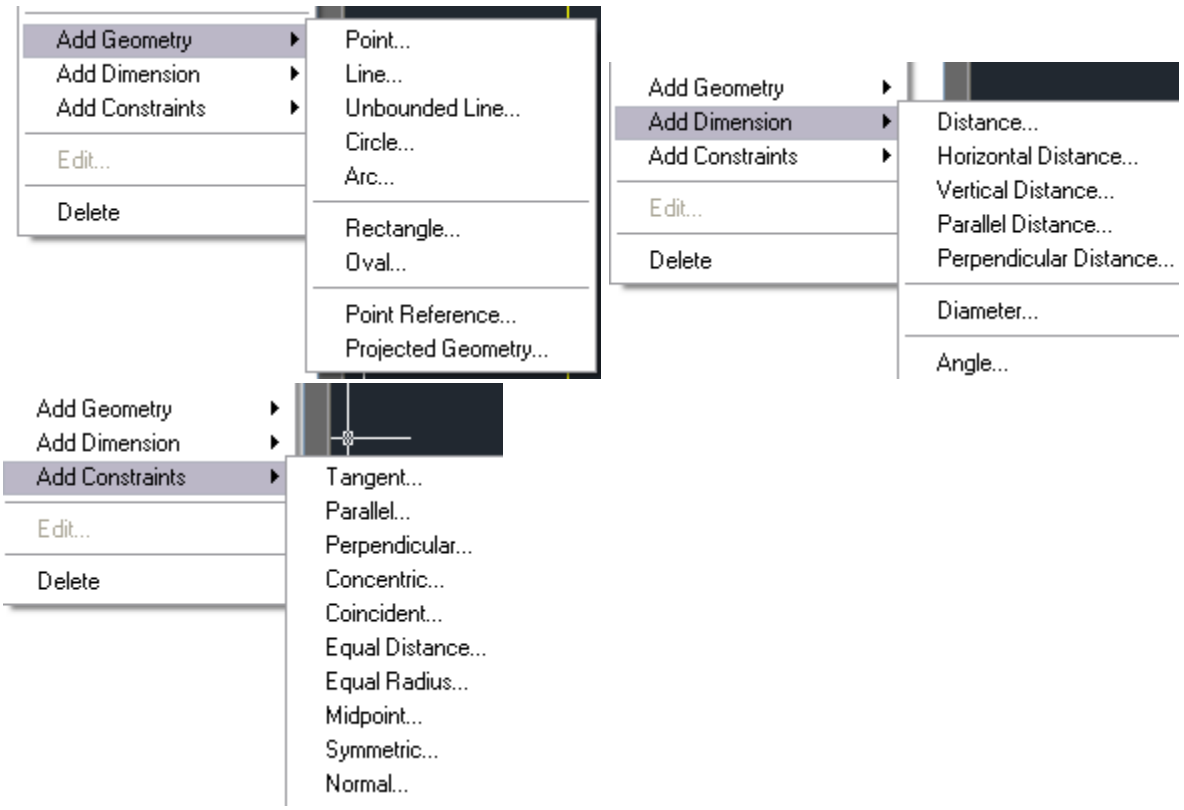
End Object Isolation: It turns the Visibility settings back to what they were, prior to isolating or hiding plane contents.

Set View: If we are in a SW Isometric view, and right-click the Top view > select "Set View," it will return us to Plan view. It brings whatever plane we "Set View" on, to a plan view perspective.

Add Profile: The choices are Circular, Rectangular, Oval and Custom. A Profile is a flat 2D set of lines and/or arcs used to extrude into a 3D object. We draw a profile on a plane. We then later use a Modifier to extrude the Profile either: Out from a Plane, Along a Path, or Between Two Planes.



Add Geometry, Add Dimension and Add Constraints: Geometry is used as construction lines or as paths. A path is what a Profile will follow to generate a 3D shape. You can pick any line to be a path and you don't have to use all lines, or any at all. Note: Rectangle and Oval geometry cannot be used as a Profile. They are for construction lines. We often refer to construction lines as the frame. Think of the frame of a large construction tractor. Construction tractors have big scoops and buckets on long arms. They have hydraulic pistons that drive the motion of the arms. We add dimensions to the geometry lines. These dimensions are like the hydraulic pistons. They push and pull as we change the value of the dimension. This stretches a line. Line is connected to another line at their end-points, so like the arm on a tractor, it pushes it around. We have constraints that hold two lines perpendicular, or parallel, or concentric, etc. When lines are constrained to be perpendicular, it forms a frame that can be pushed around by dimensions. We will see this in action in the example.

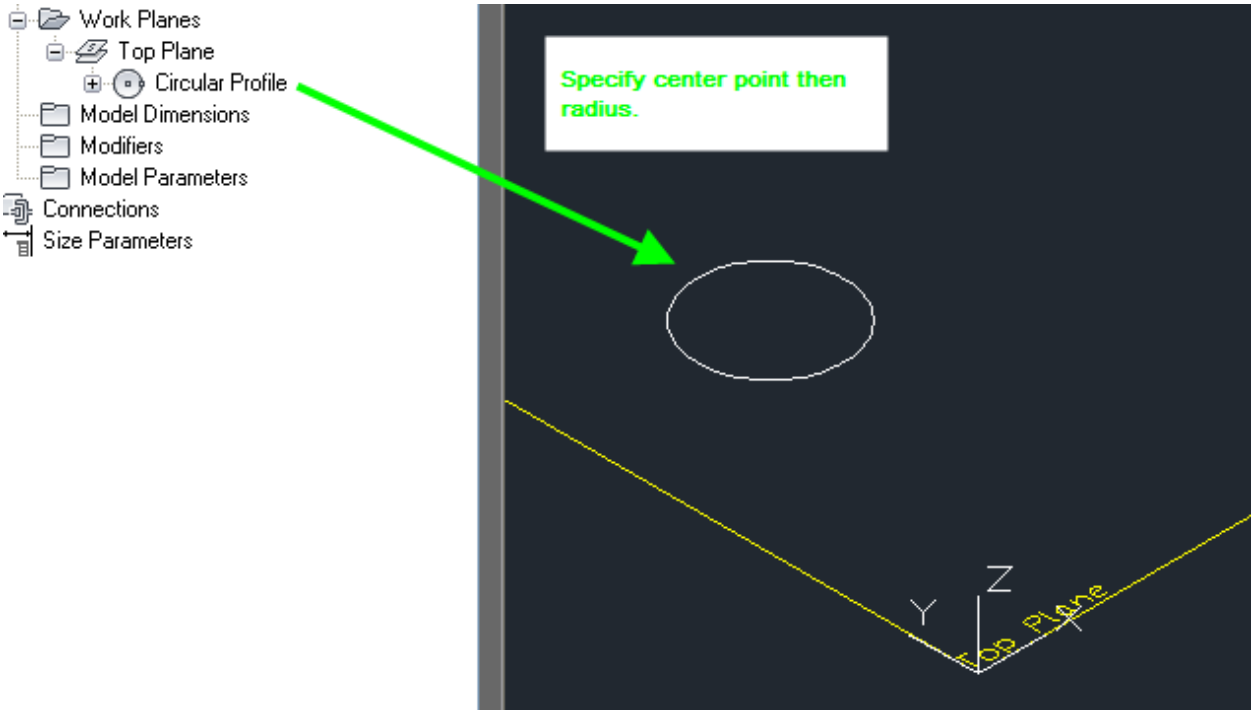


Edit: Allows you to edit an existing plane's settings if it was created as an Offset, Reference or Custom plane type.

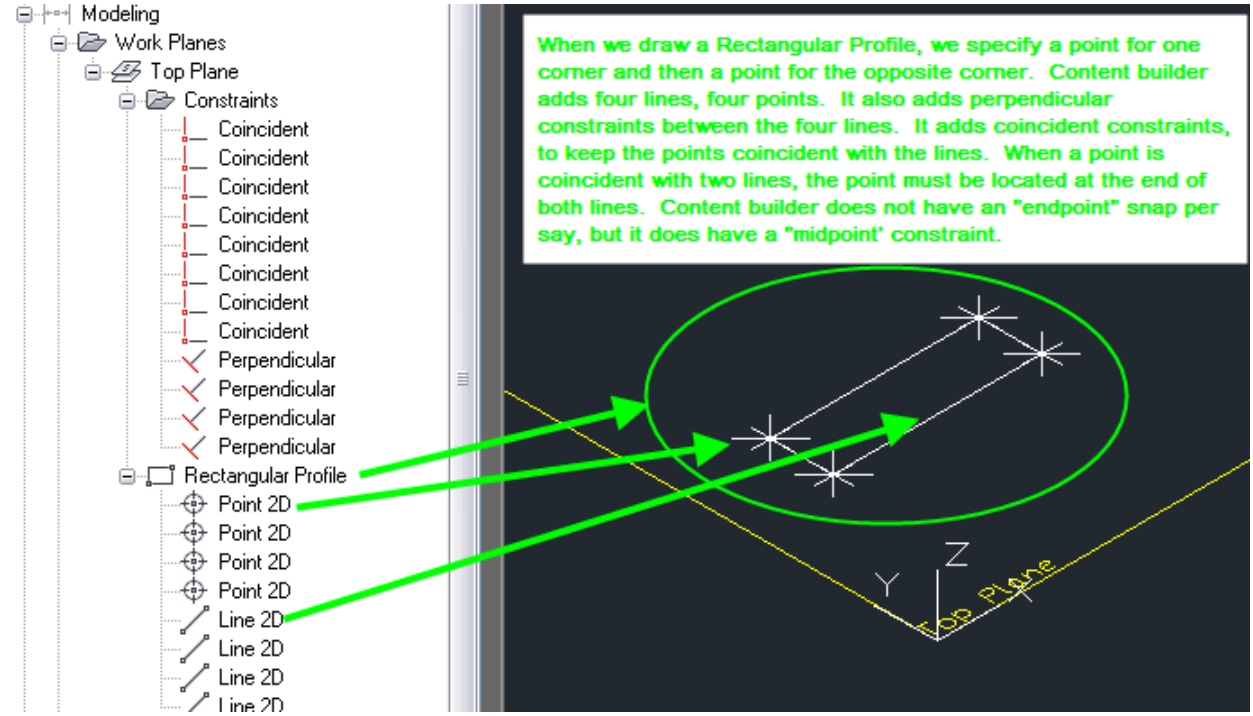
Delete: Allows us to delete a work plane. Note: Remember, we build objects on top of other objects. Deleting a plane, deletes all objects built on that plane. You draw geometry, profiles, dimensions and constraints on a plane. You build a Modifier based on a profile related to the plane. You add a connector to the Modifier. You eventually add Size Parameters to the

dimensions. Deleting a plane, would delete the geometry, profiles, modifiers, connectors and size parameters. May be the quickest way to start from scratch again if that is the goal. Otherwise, be cautious. Always try to delete items in the reverse order they were created in, to minimize unintended casualties.

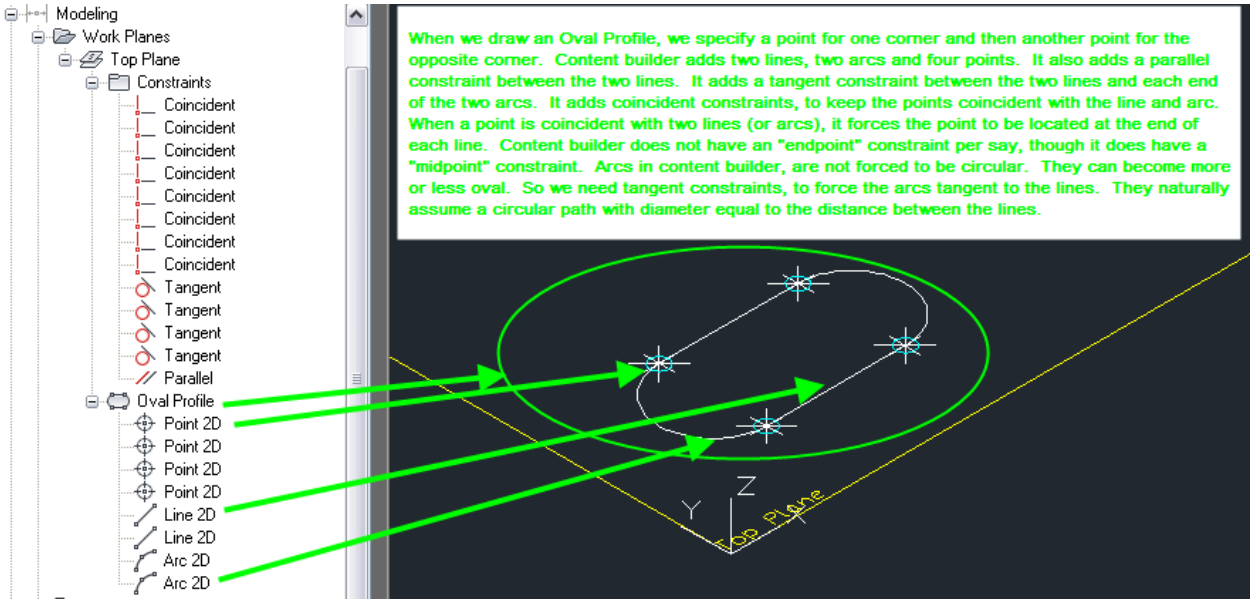
Add Geometry, Dimensions and Constraints



Circular Profile

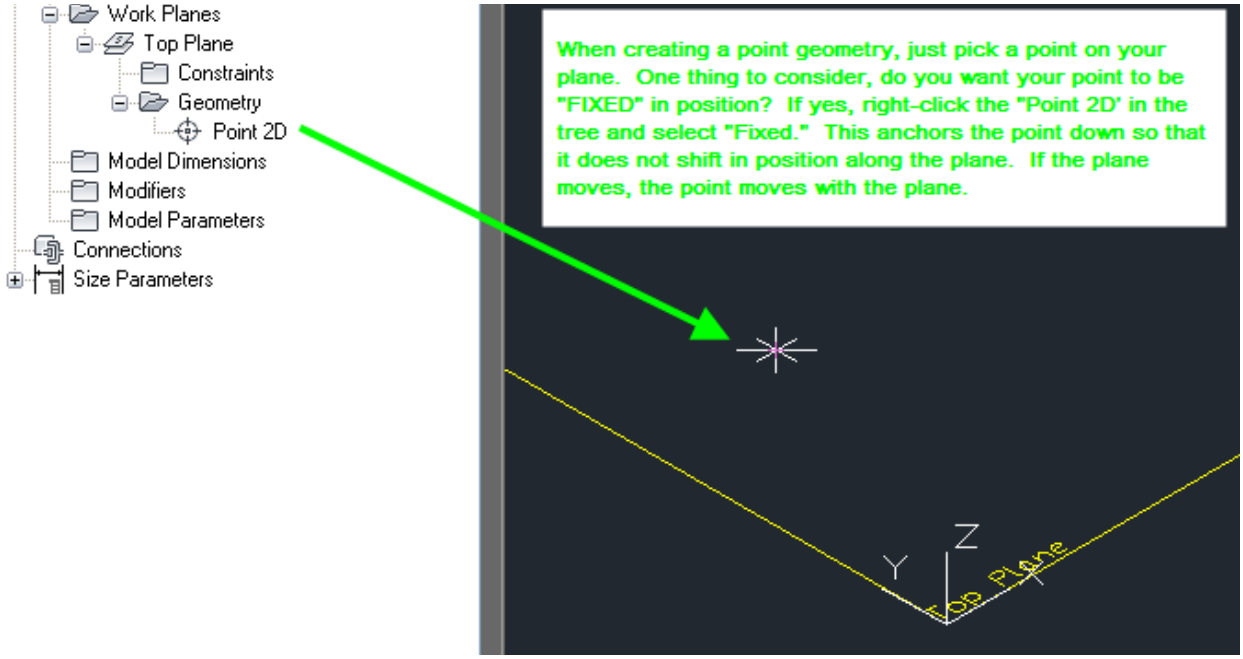


Rectangular Profile

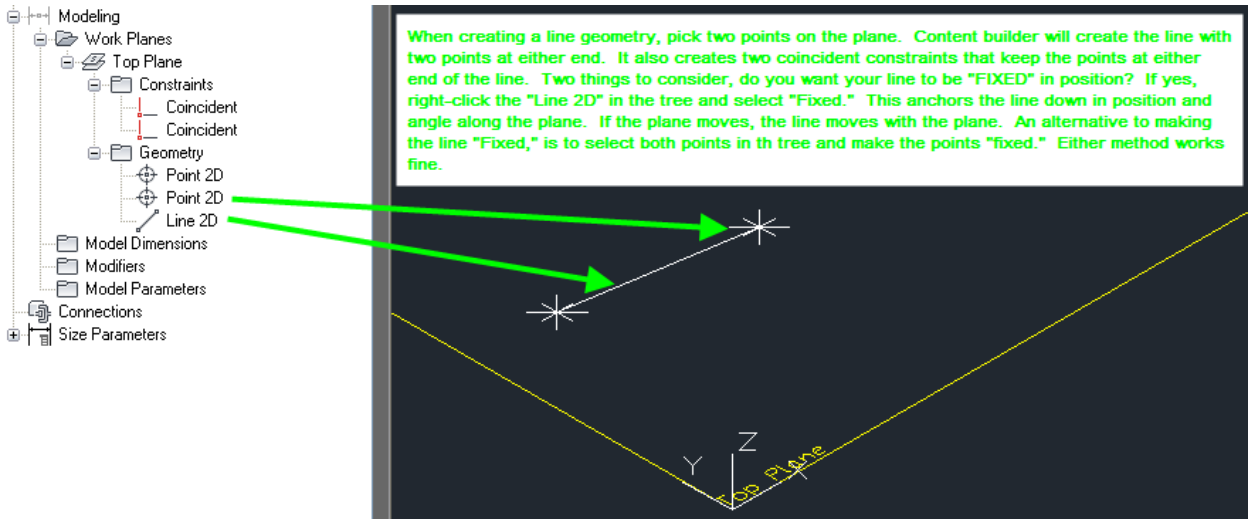


Oval Profile

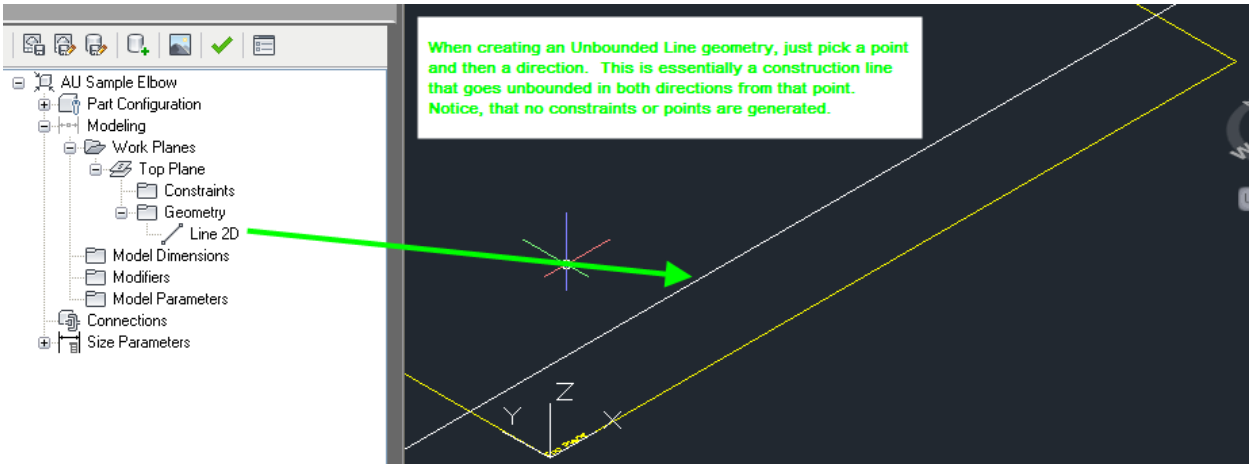
Custom Profile – Requires that Geometry be drawn first.



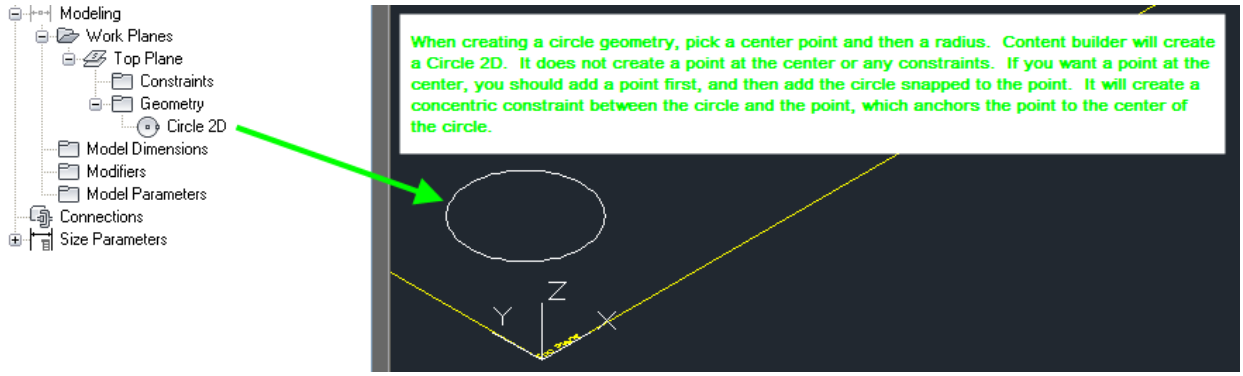
Point Geometry



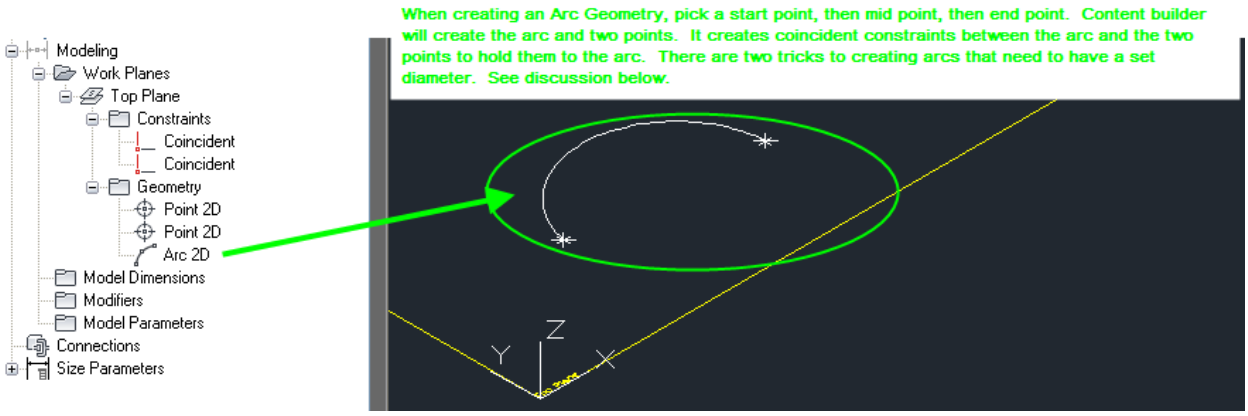
Line Geometry



Unbounded Line Geometry

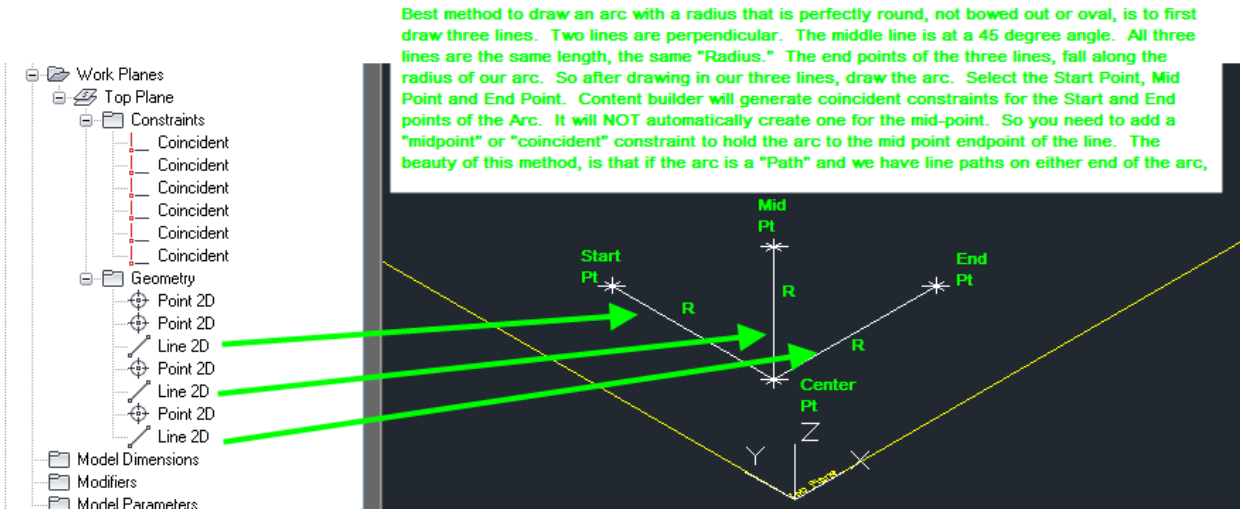


Circle Geometry

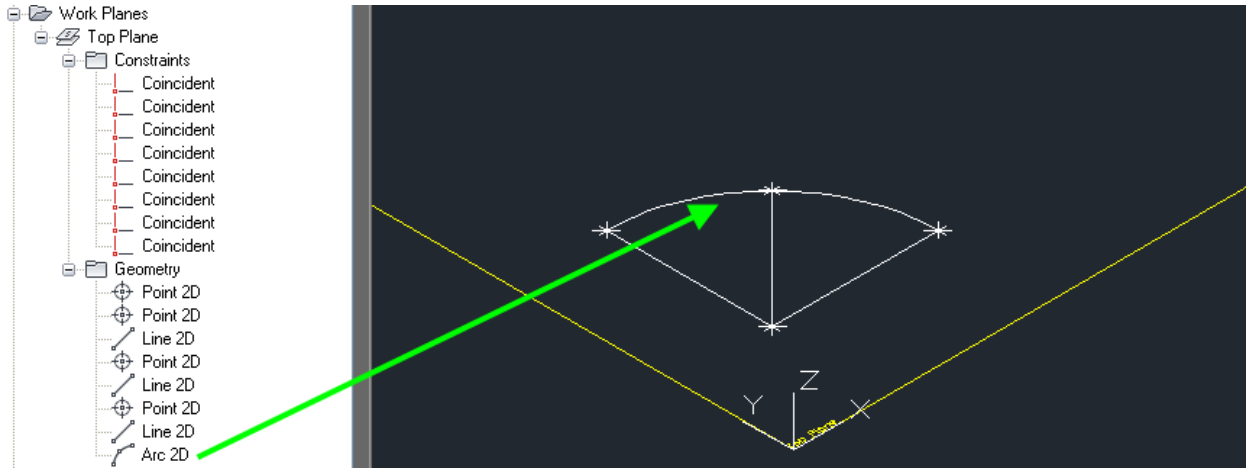


Arc Geometry

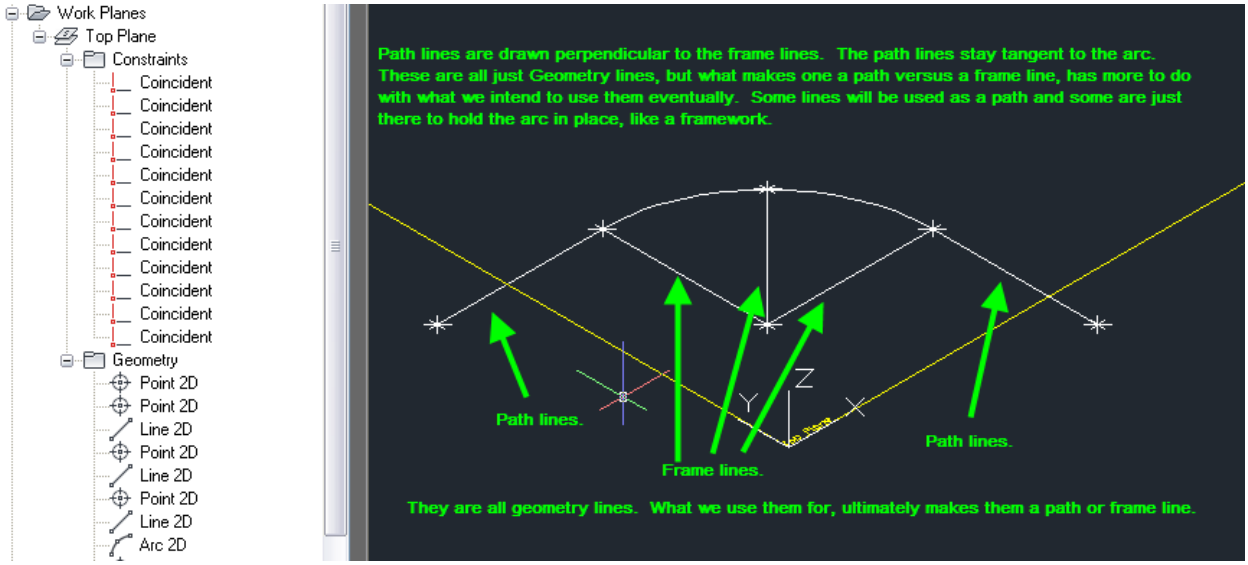
There are two tricks to creating arcs, when you need the arc to be round, not bowed outwards like an oval, and held to a diameter.



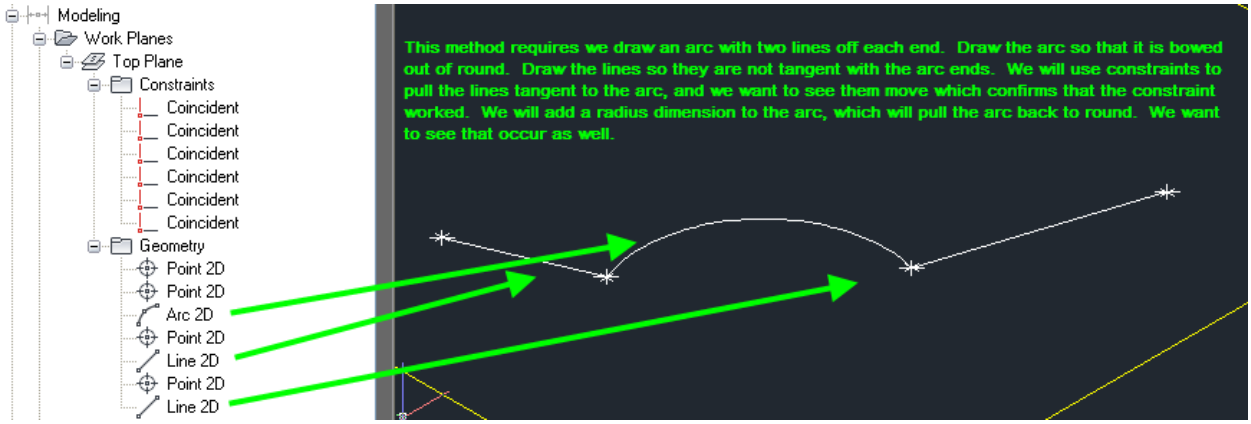
Best Method – This method ties an arc to a frame of points and lines. The lines are equal to the radius of the arc.



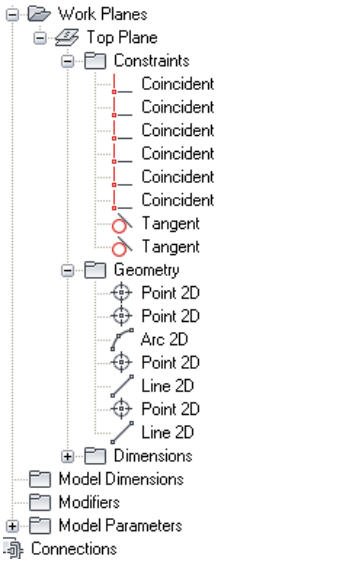
This frame is nice if we have paths leading away fromeither end of the arc, and we may want those path lines to always be tangent to the arc, or perpendicular to the frame supporting the arc.

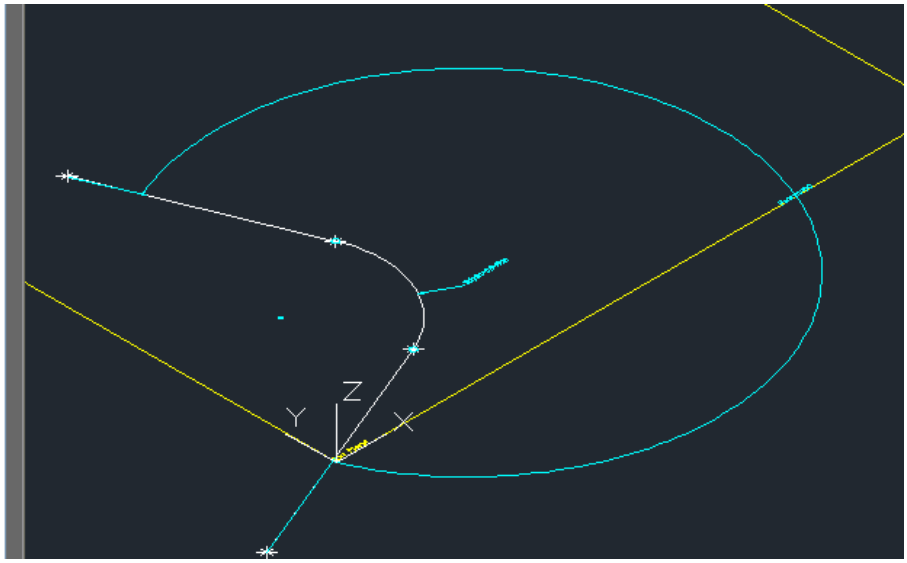


This frame method is great if we want better control over the center point of the arc, and the frame around the arc.

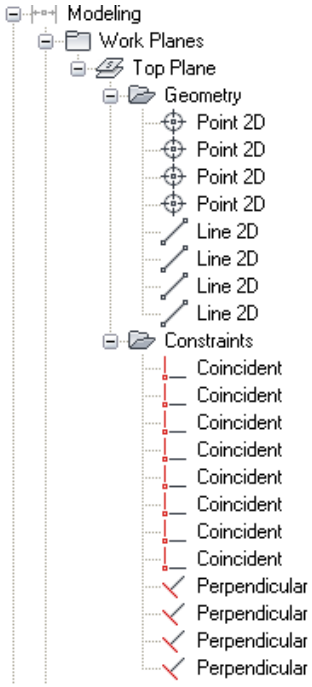


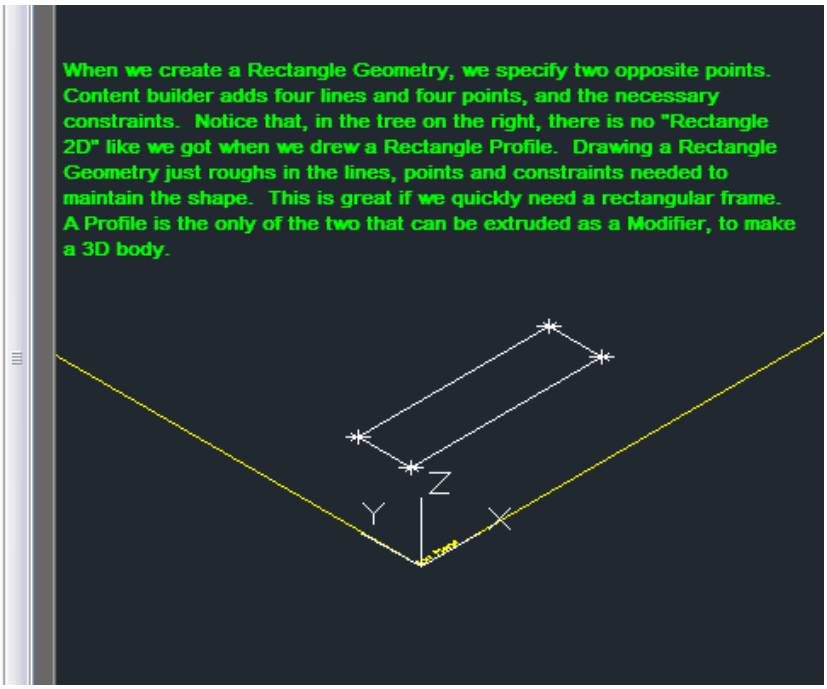
This is the second method to define an arc. It has fewer frame lines, and relies more on constraints. There is little control over the arc center point.



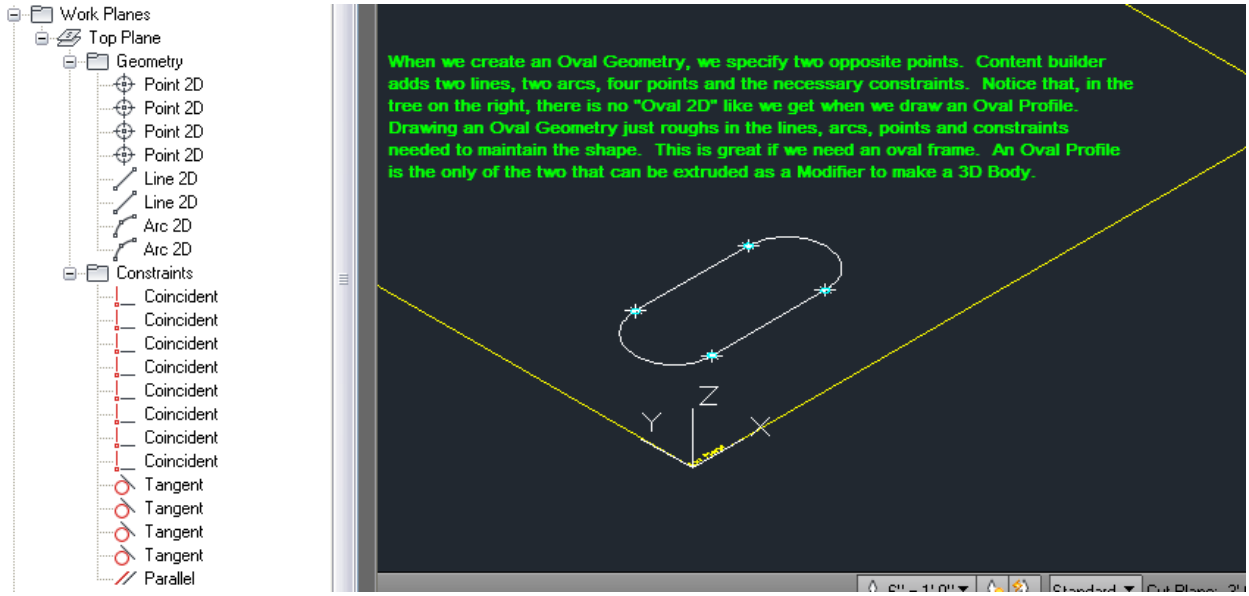


This is the result. We have two lines tangent to an arc that has a radius dimension. The two lines have an angle dimension between them, holding them to 90 degrees.

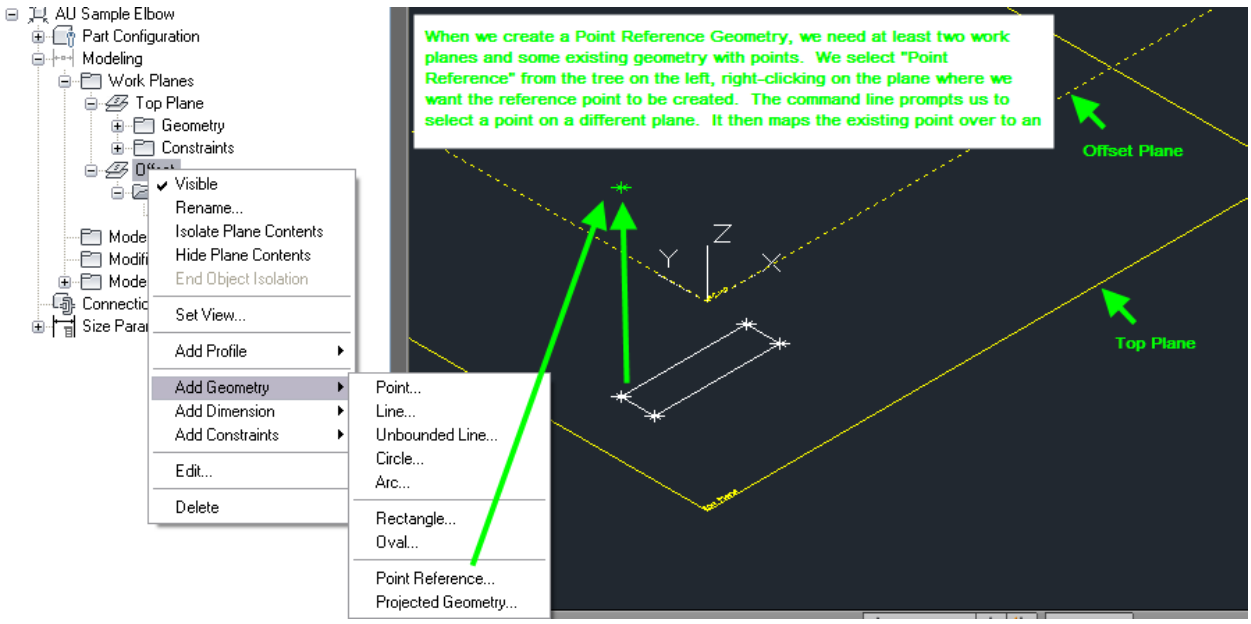




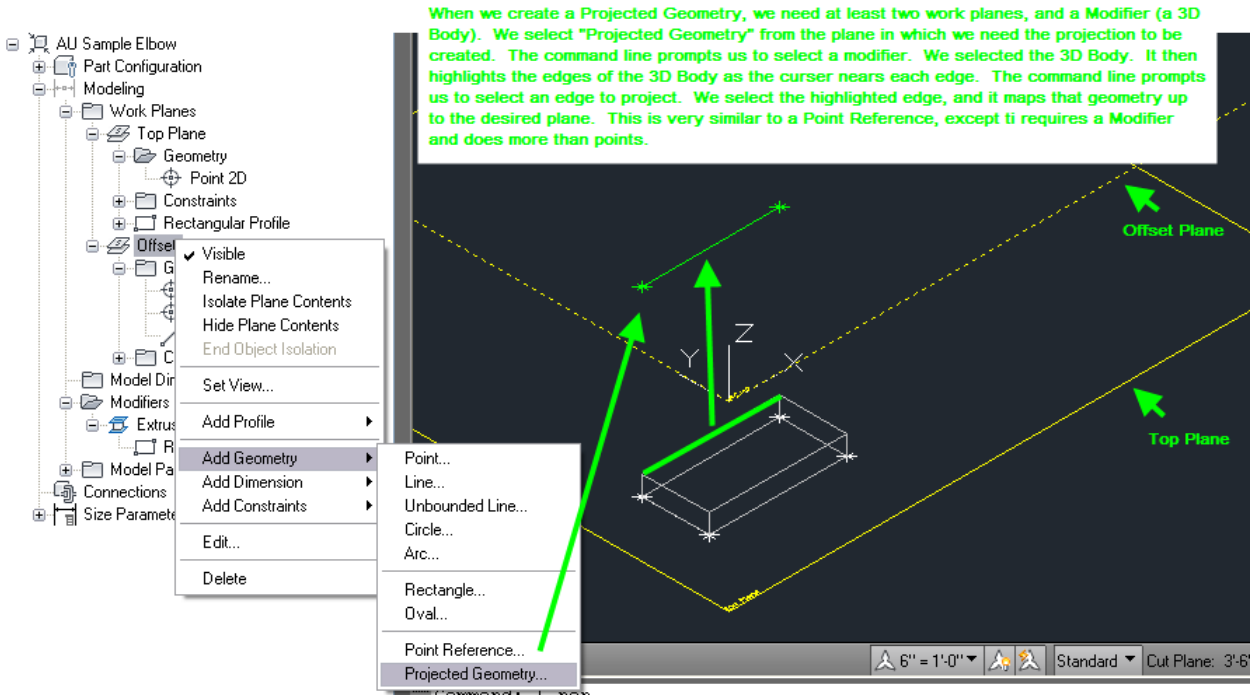
Rectangle Geometry



Oval Geometry

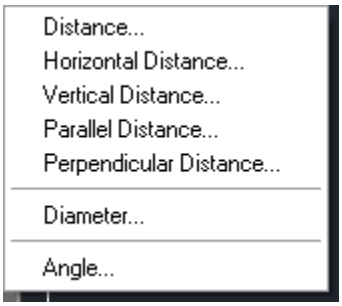


Point Reference Geometry

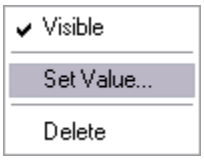


Reference Geometry

Dimensions

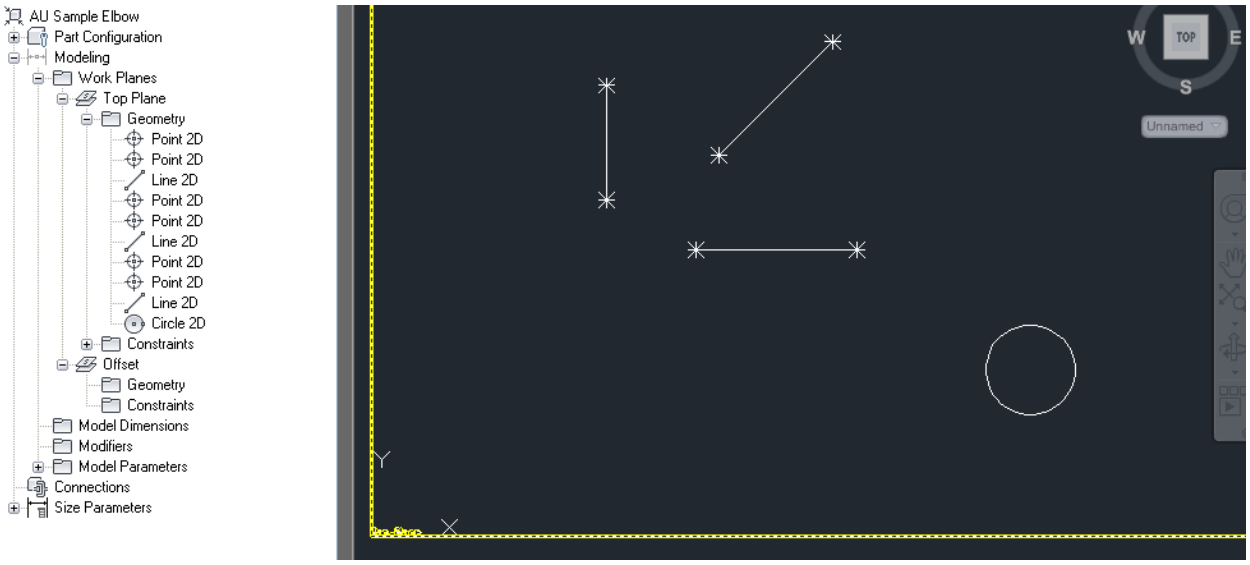


These are the dimension choices.

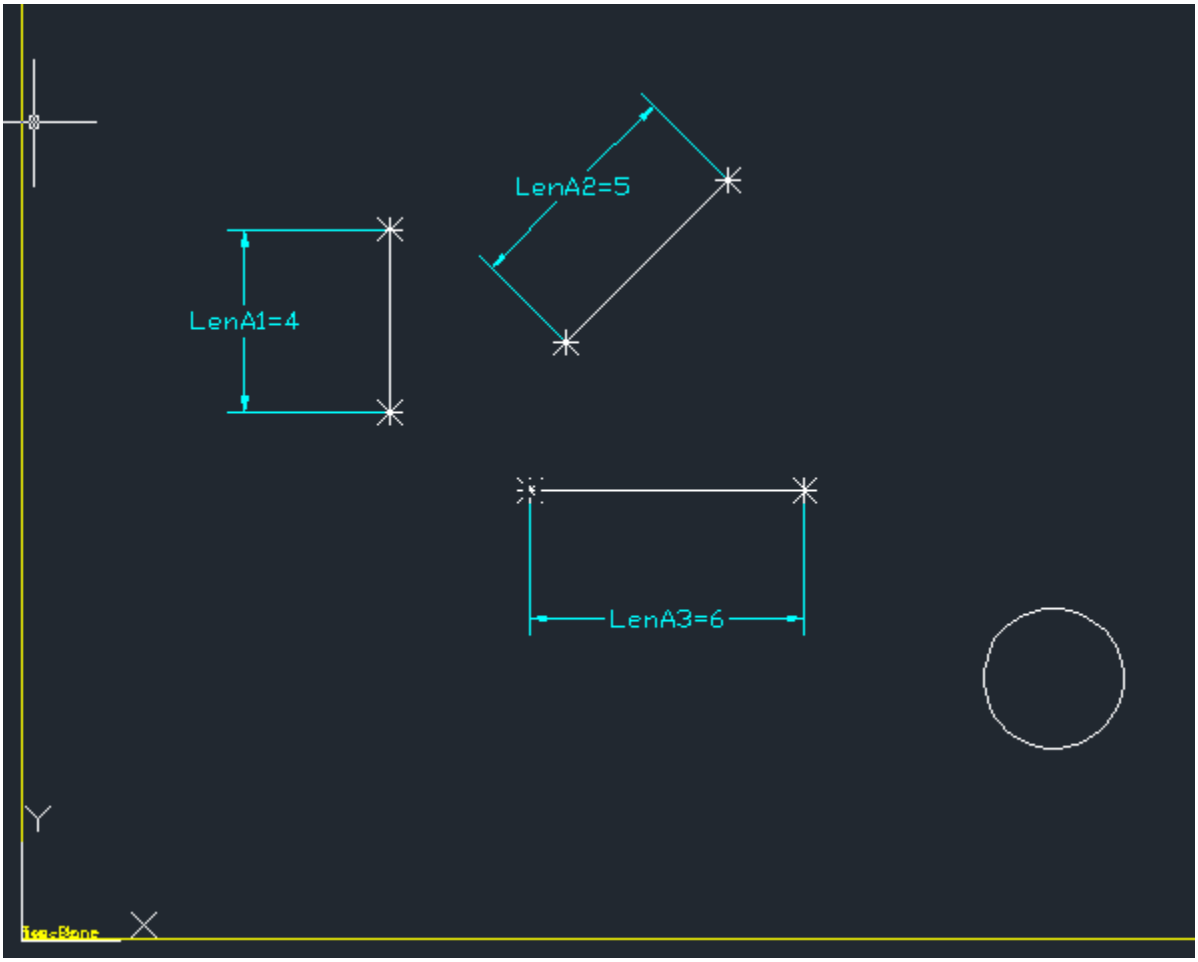


After creating a dimension, we can right-click the dimension, in the tree, and get these options. Set Value can be useful to change the value of a dimension quickly. Changing the value of a

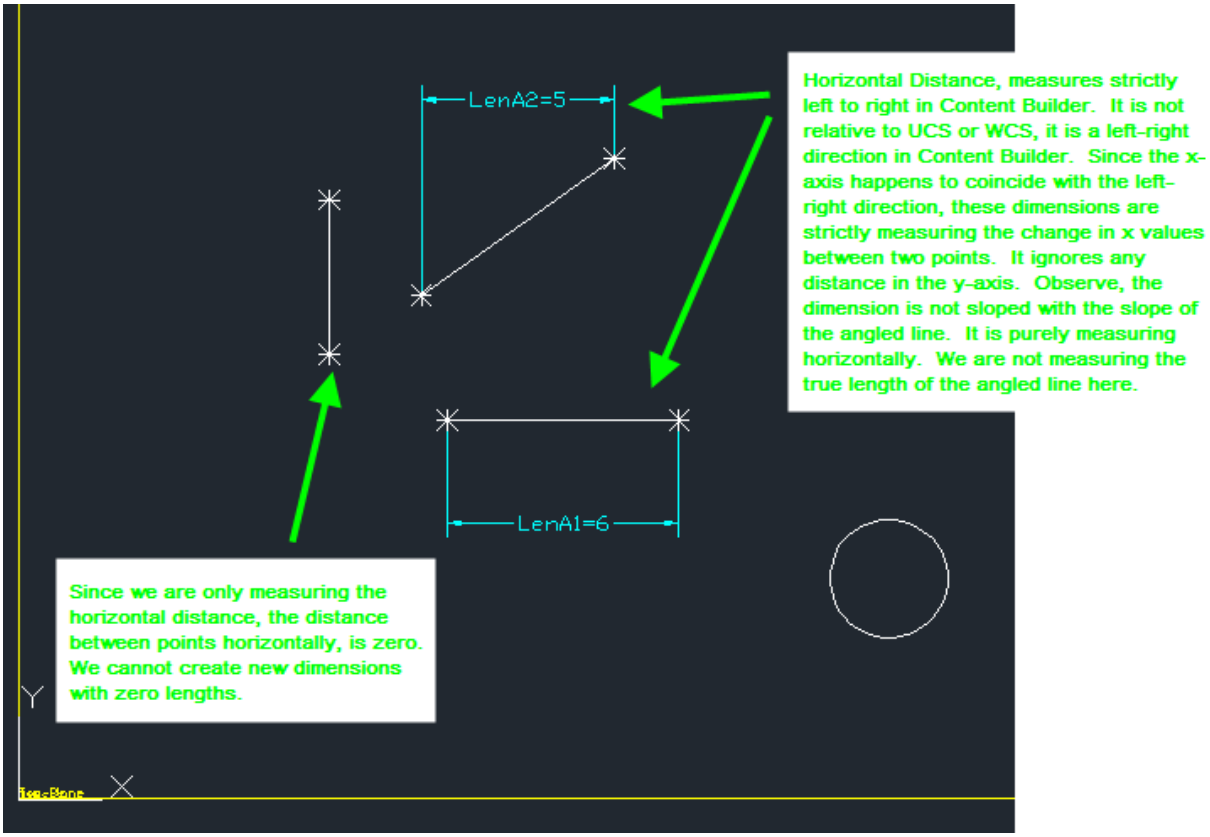
dimension, only changes the Model Parameter value. It affects the Model Parameter values, not the Size Parameter values.



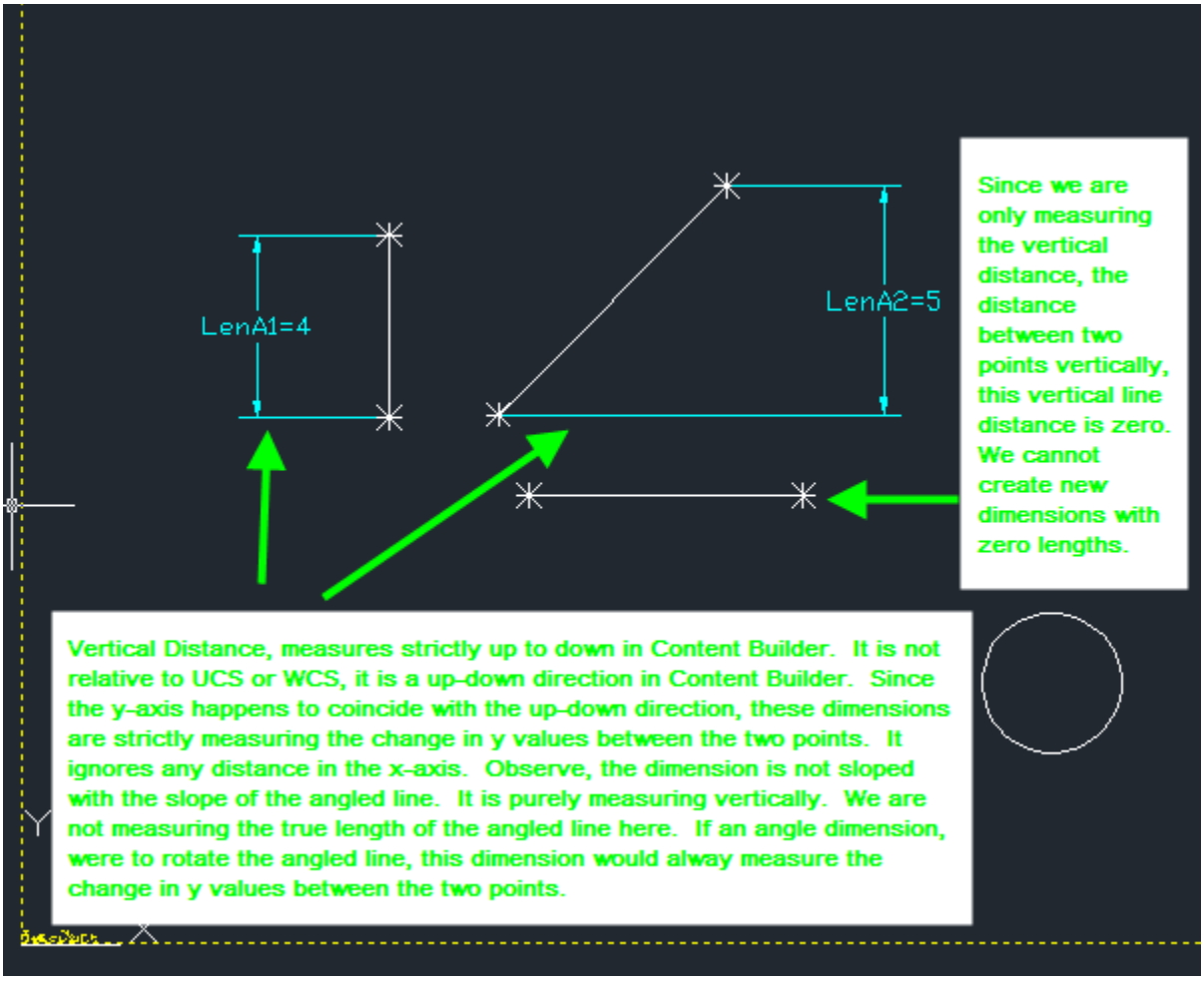
What do content builder dimensions do? Let's dimension the above lines and circle, using each type.



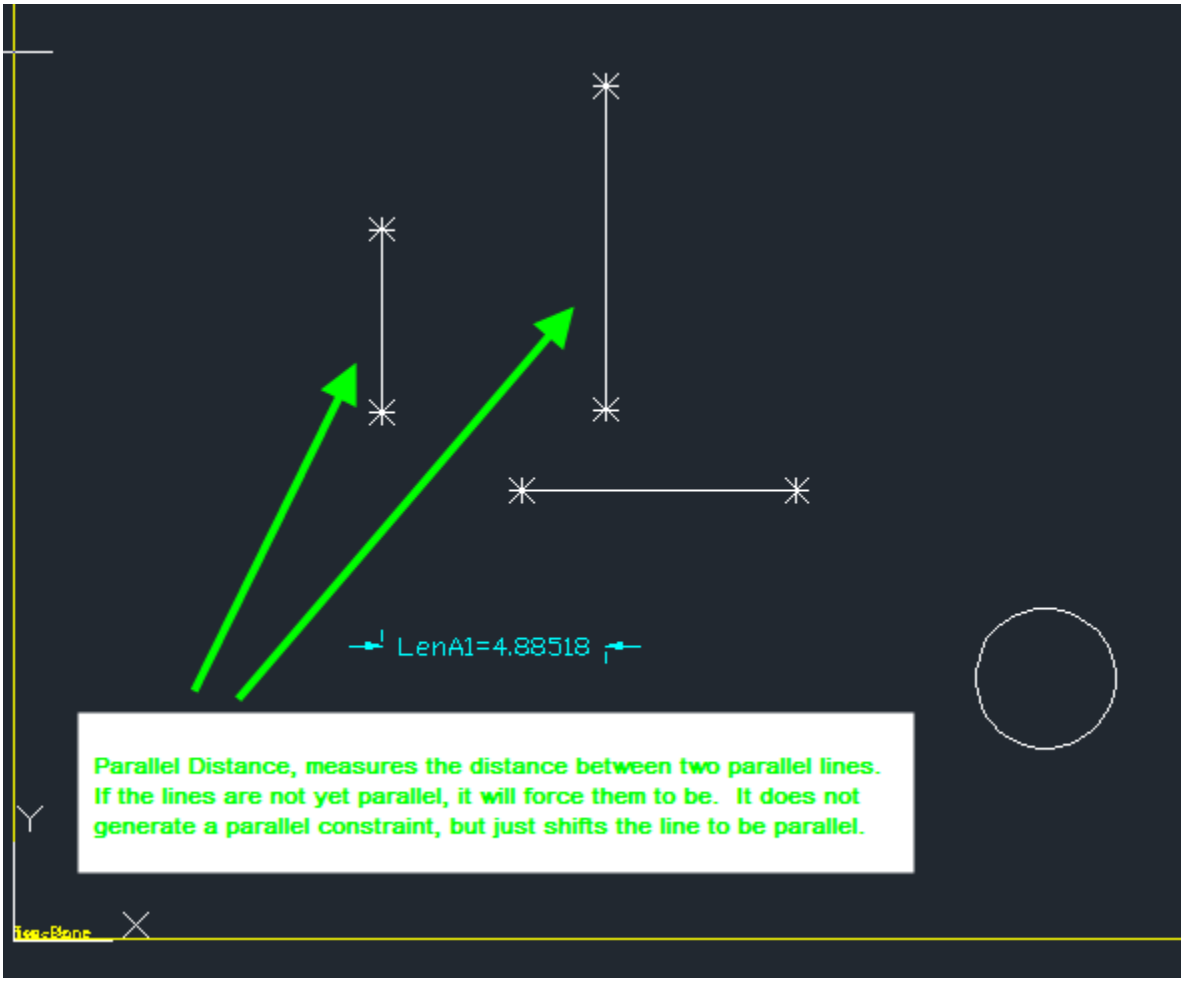
Distance Dimensions – Measures the length at any angle.



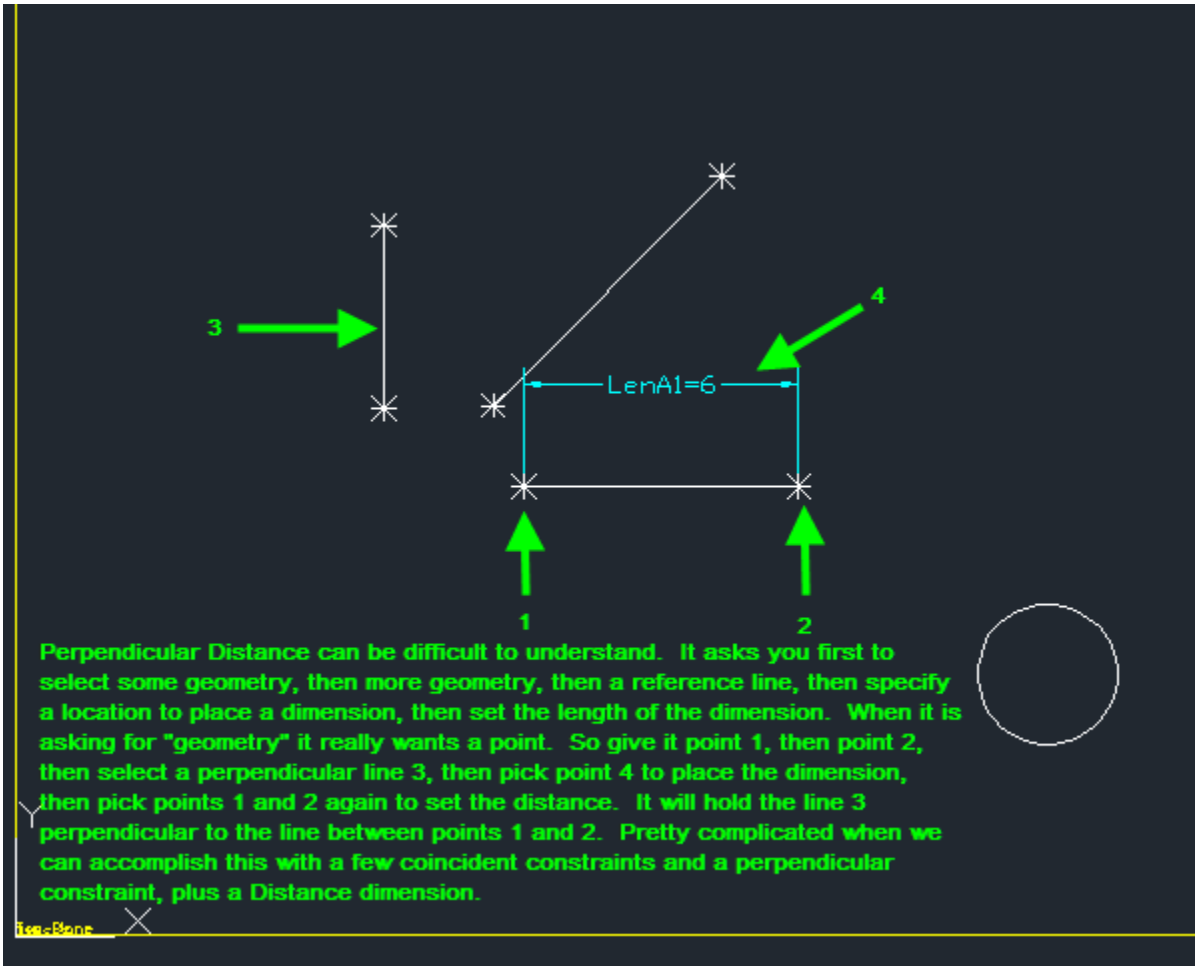
Horizontal Distance Dimension – Measures left-right distances



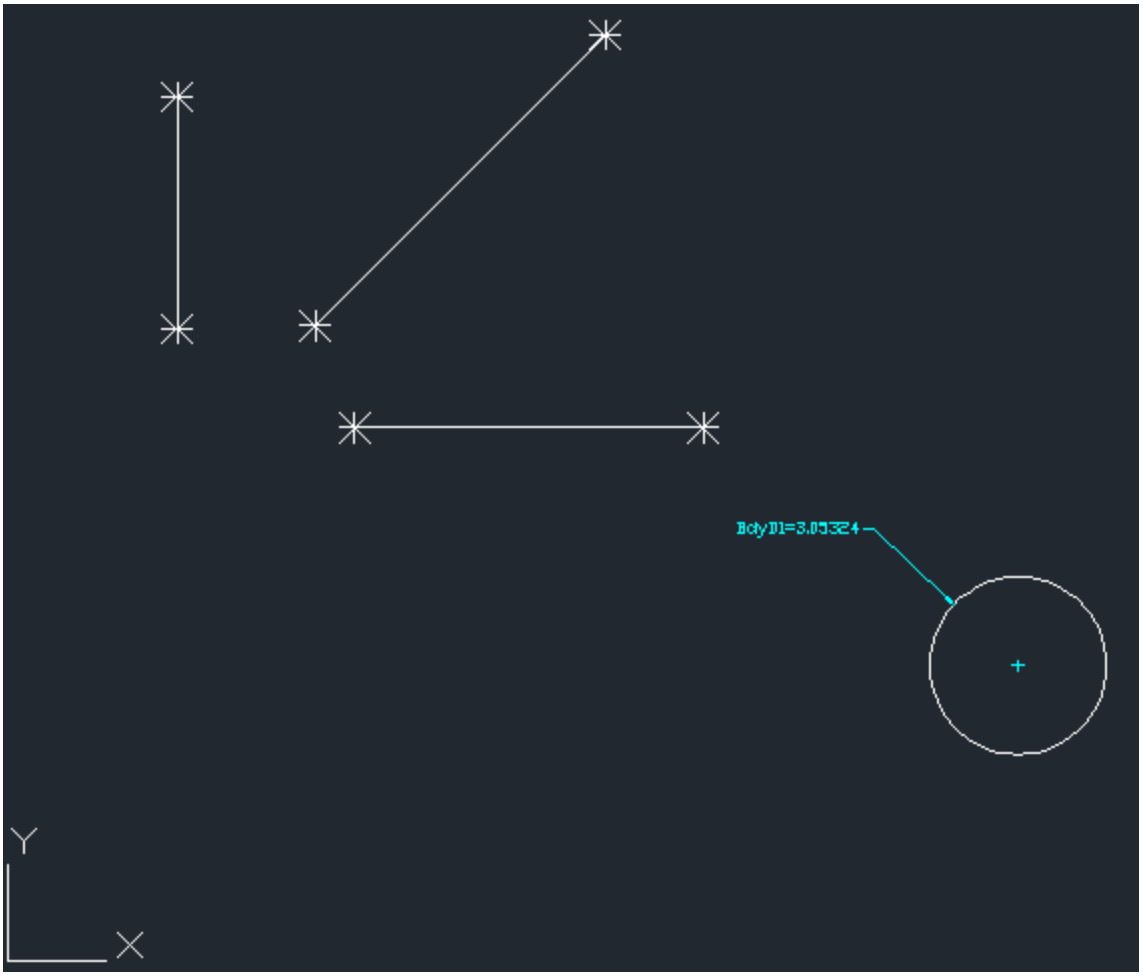
Vertical Distance Dimension – Measures up-down distances



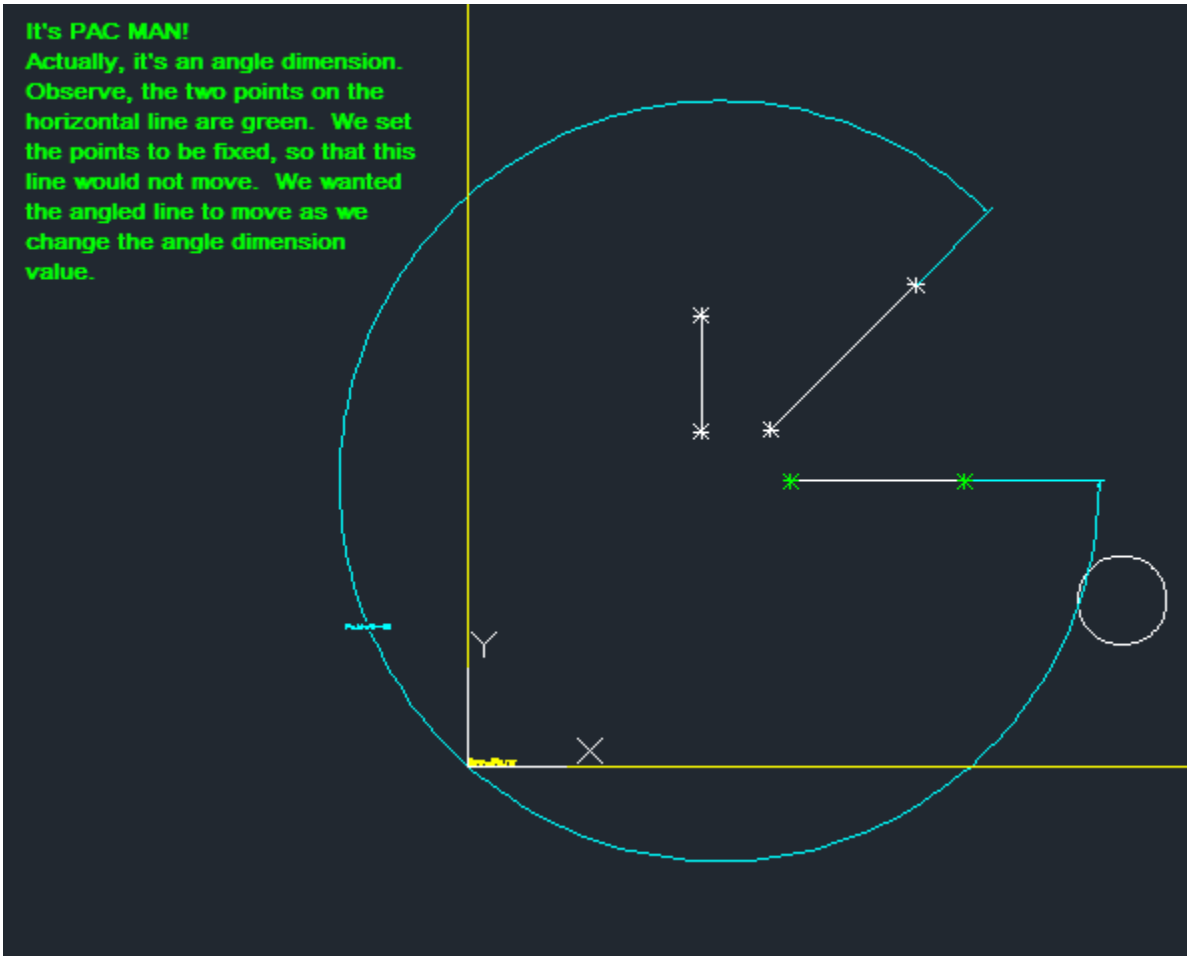
Parallel Distance – Note, when the command line asks for “Geometry,” it really wants the points not the lines.



Perpendicular Distance – Note, when the command asks for “Geometry” it really wants the points, not the lines.

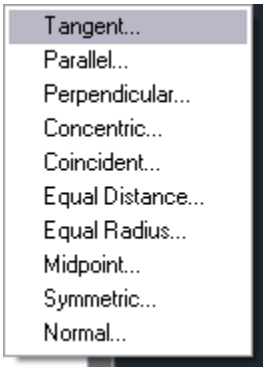


Diameter Dimension



Angle Dimension

Constraints



Tangent – Keeps lines and arcs tangent to arcs.

Parallel – Keeps lines parallel.

Perpendicular – Keeps lines perpendicular.

Concentric – We need a point. Keeps an arc or circle concentric about a point, or two circles concentric about each other.

Coincident – This means two objects touch. So a point coincident on a line or arc, just means that they stay in contact with each other. It's exactly the same as the "nearest" Osnap back in AutoCAD. When Content Builder automatically generates a line with points, or arc with points, it makes the points coincident, and forces them to the ends. It acts like end points.

Equal Distance – Makes two lines have the same length.

Equal Radius – Keeps two arcs or two circles to have an equal radius.

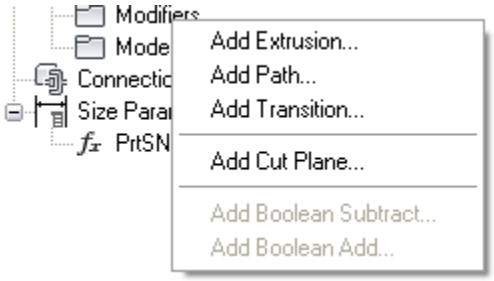
Midpoint – Keeps a point coincident on a line and held tight to the midpoint of that line.

Symmetric - When we have a line in the middle, this will force two points or two parallel lines to remain "equal distance" on either side, from the center line.

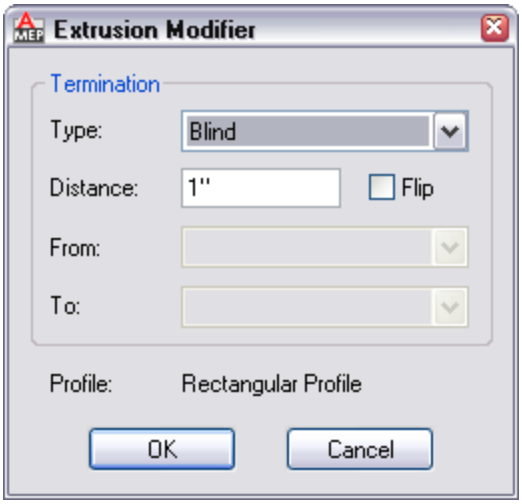
Normal – The opposite of Tangent, this forces a line to be perpendicular or "Normal" to an arc or circle.

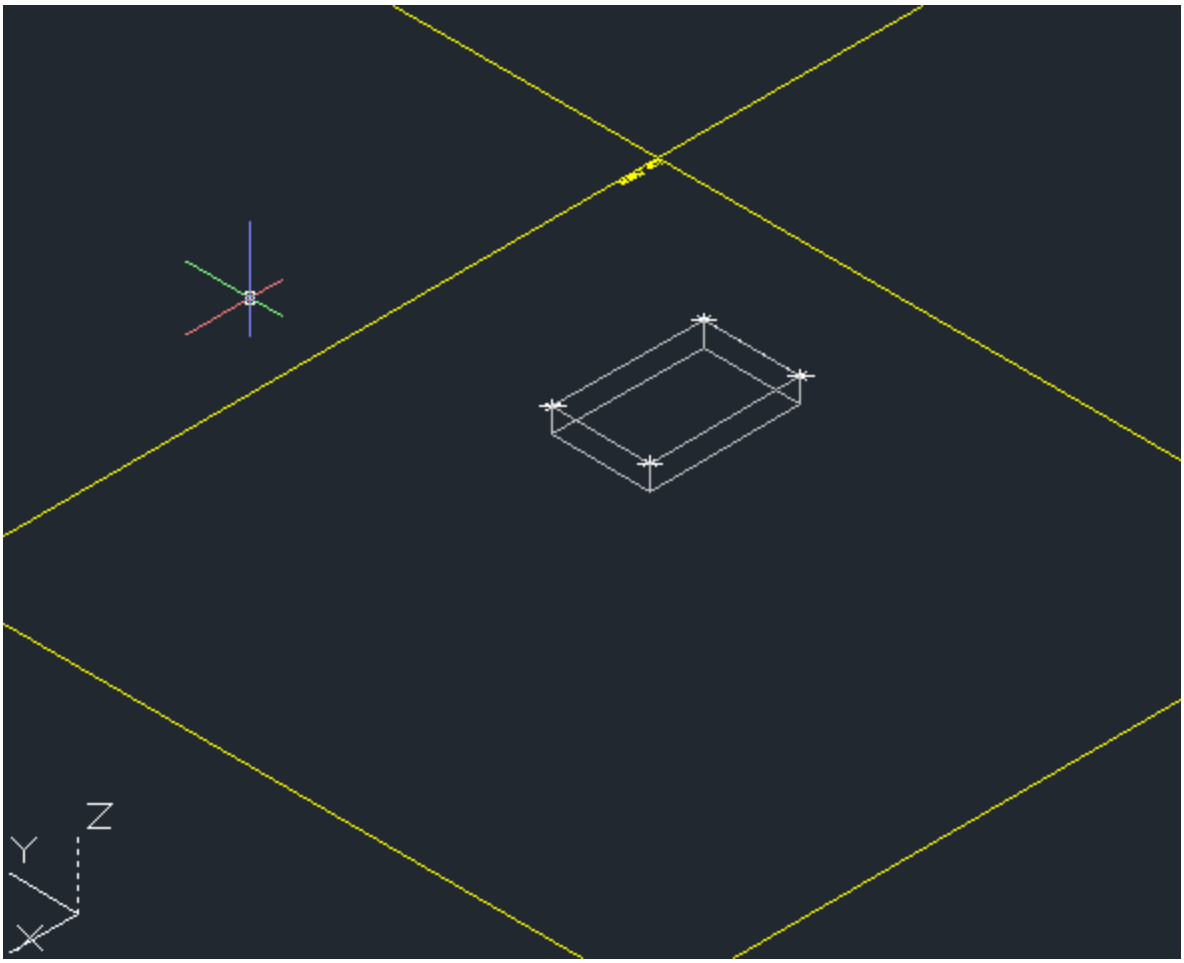
Modifiers

Let's discuss Modifiers.

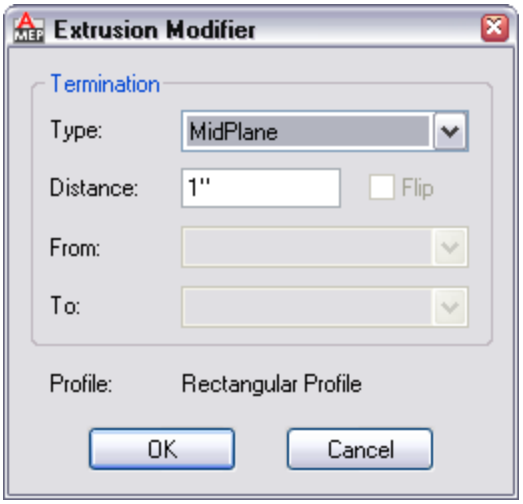


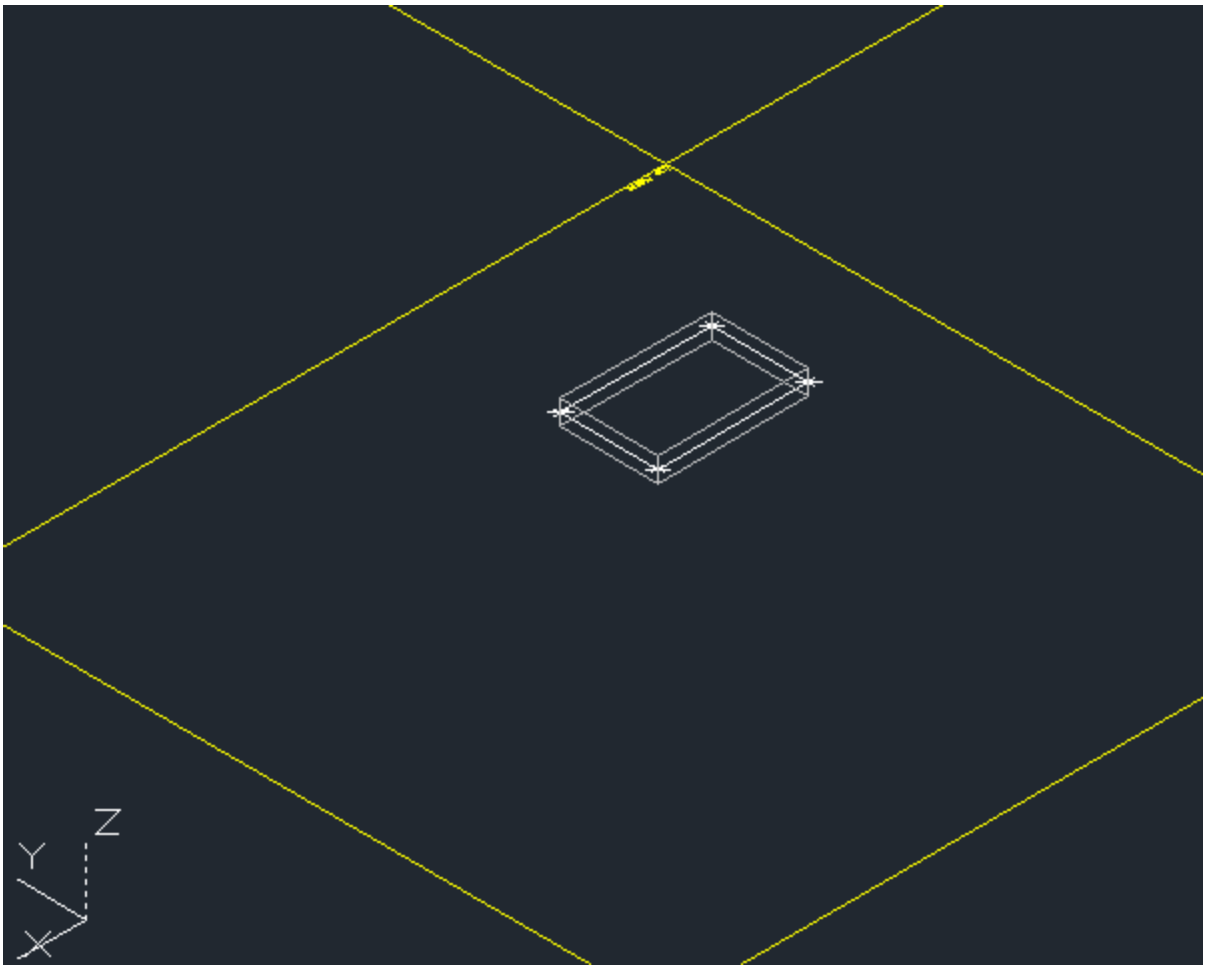
After selecting Add Extrusion, the command line prompts to select a Profile.



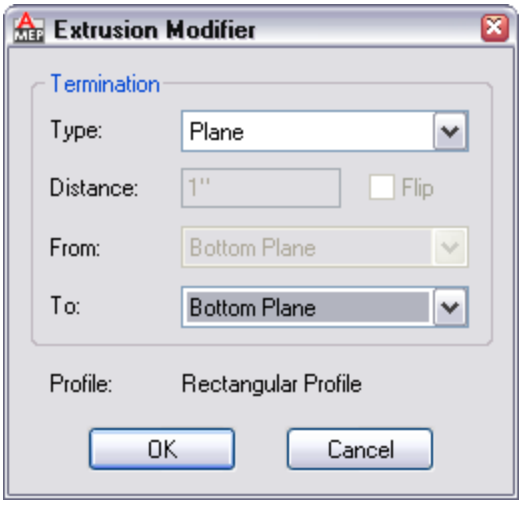


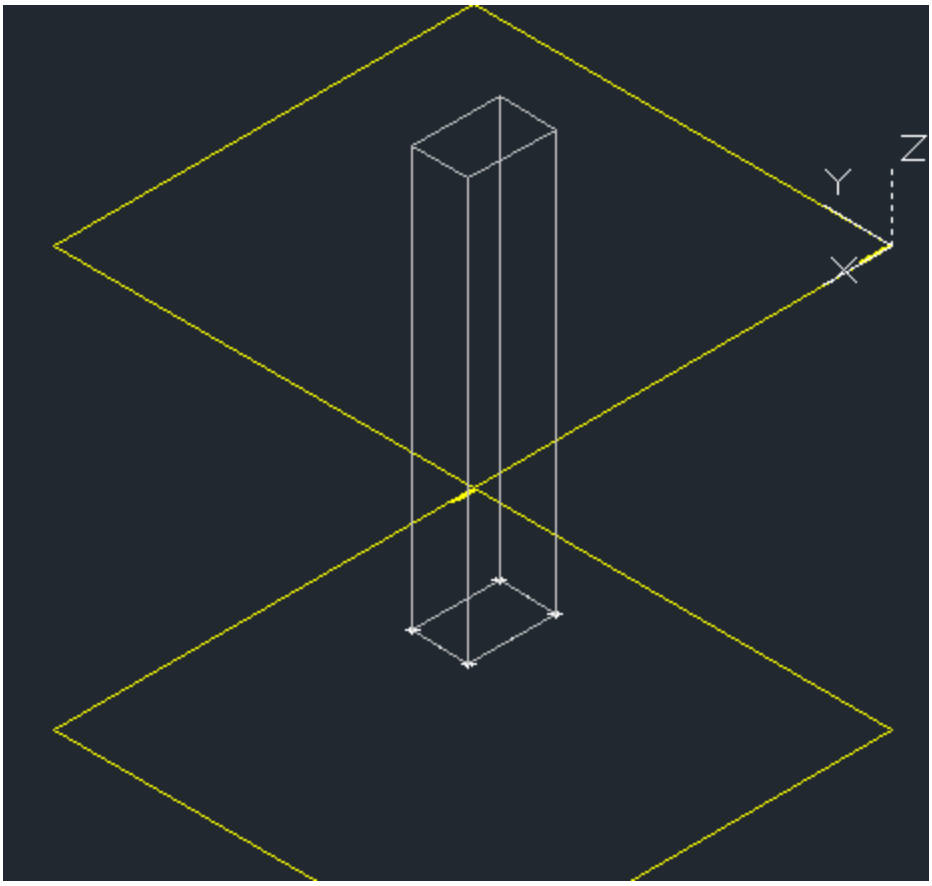
Blind Extrusion – Uses a Profile, and extrudes that profile out perpendicular to the plane surface by the distance specified. To go the opposite direction, enable the “Flip” checkbox.



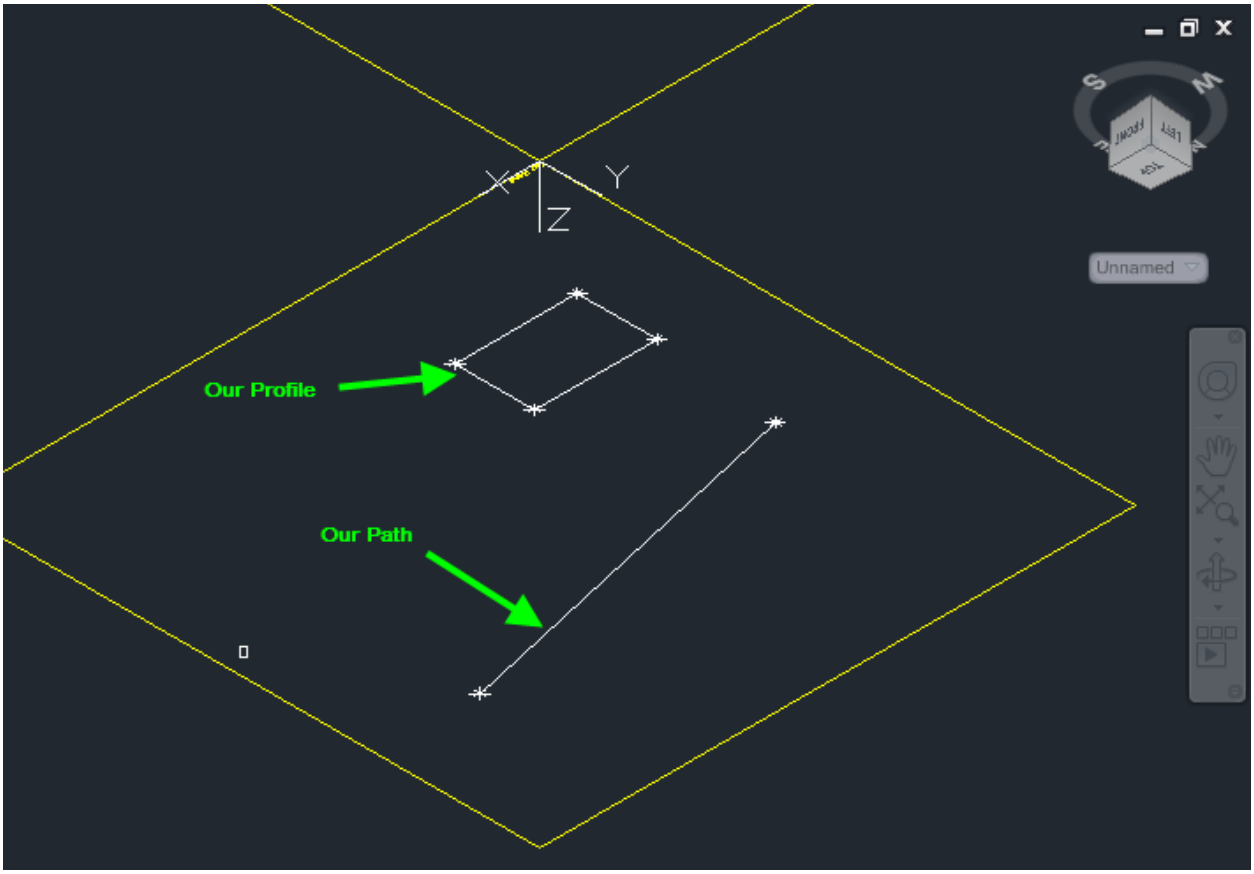


MidPlane Extrusion – Uses a Profile, and extrudes that profile out perpendicular to the plane surface by the distance specified. It extrudes in both directions equally by the specified distance. So the “Flip” checkbox is not necessary.





Plane Extrusion – Uses a Profile, and extrudes that profile out to the specified plane surface, by whatever distance is between the planes. So the “Flip” checkbox is not necessary.



Add Path

```
Pick end point: *Cancel*
Underconstrained by 6 dimensions.
Command: AddColModPath
Select path geometry:
```

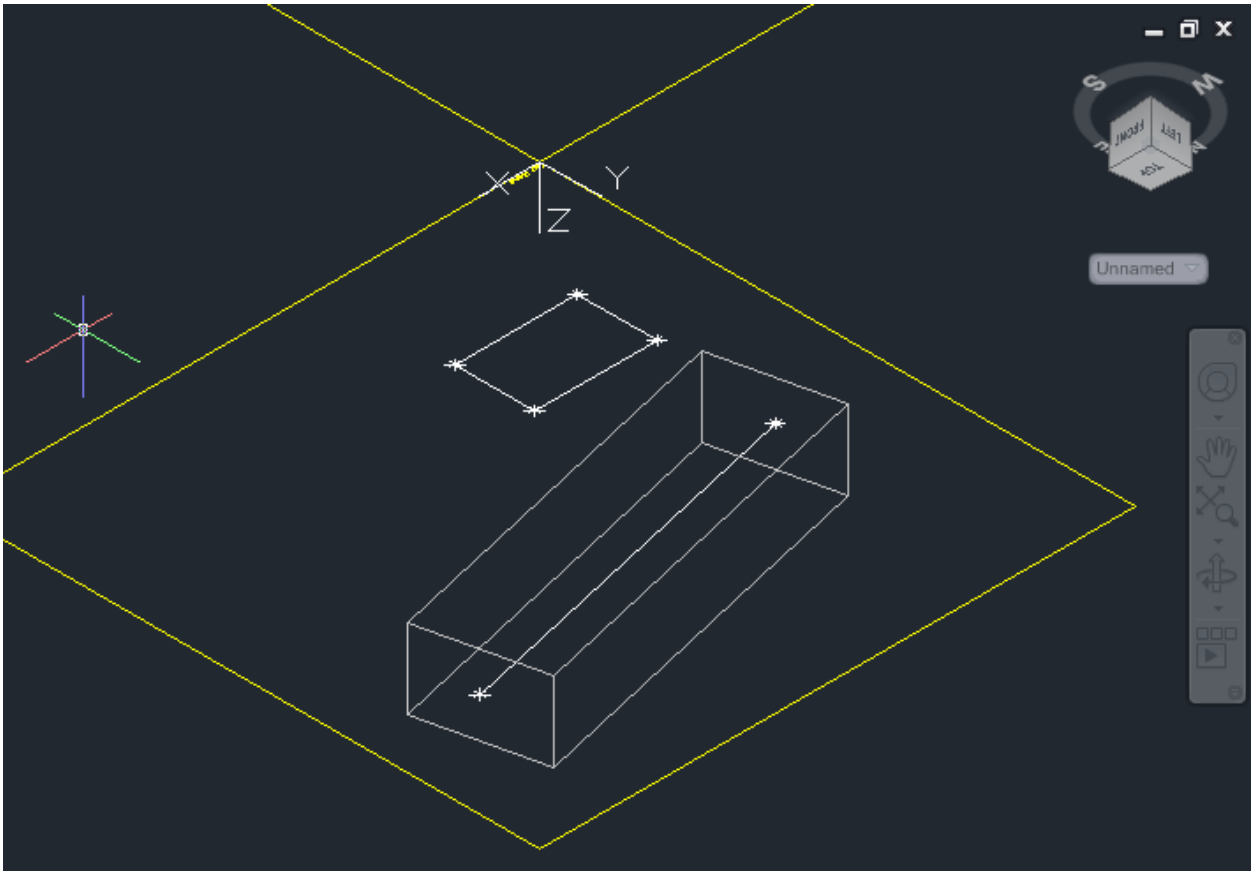
The command line requests a path, select the line.

```
Select end profile or <Enter>: *Cancel*
Command: AddColModPath
Select path geometry:
Select start profile:
```

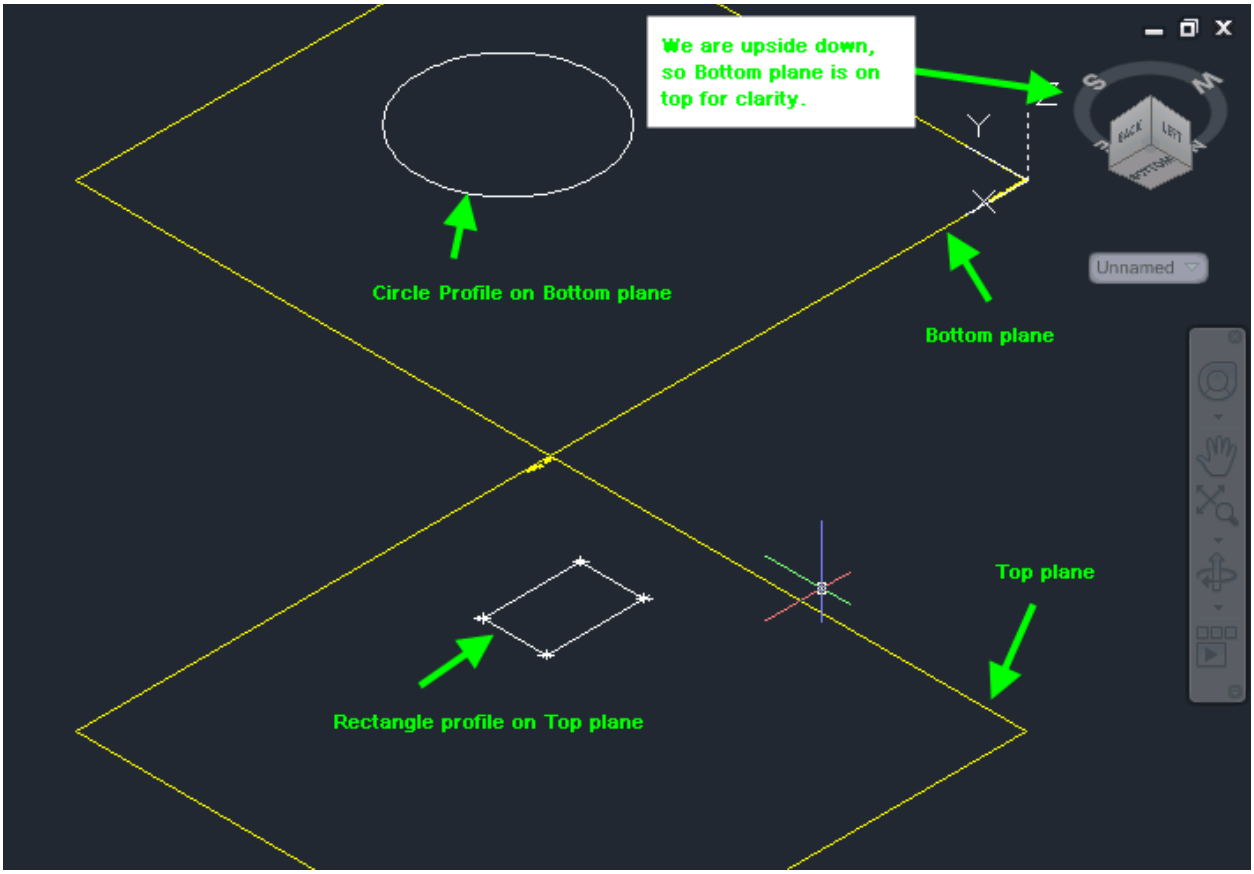
The command line requests a start profile, select the profile anywhere.

```
Command: AddColModPath
Select path geometry:
Select start profile:
Select end profile or <Enter>:
```

The command line requests a final profile. If we had a second profile, of any shape, we could select it. It would generate a transition along the path, as it transitions from one profile to the next across the length of the path. If we don't have a second profile, we can tap the enter key to accept the first profile as the second. It would not transition and would hold a constant profile across the length of the path.



This is the final result. We have a 3D Body made by that profile swept along that path.



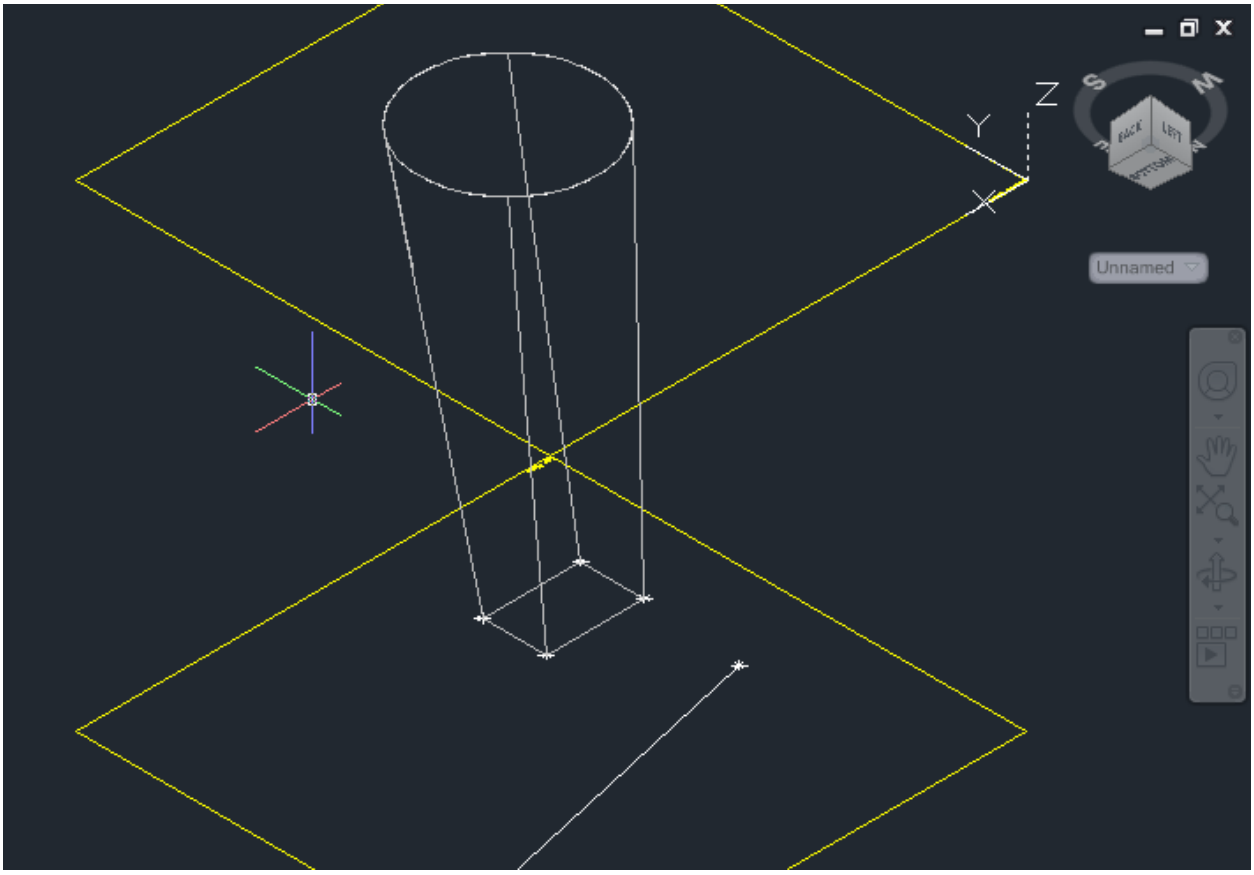
Add Transition

```
Select start profile:
Select end profile or <Enter>:
Command: AddColModTransition
Select start profile:
```

The command line requests a start profile (on one plane). We selected the Rectangle first.

```
Underconstrained by 1 dimensions.
Command: AddColModTransition
Select start profile:
Select end profile:
```

The command line requests an end profile (on opposite plane). Planes do not have to be parallel, but you will get odd and possibly unexpected results if they are not parallel.

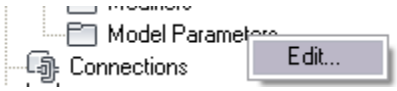


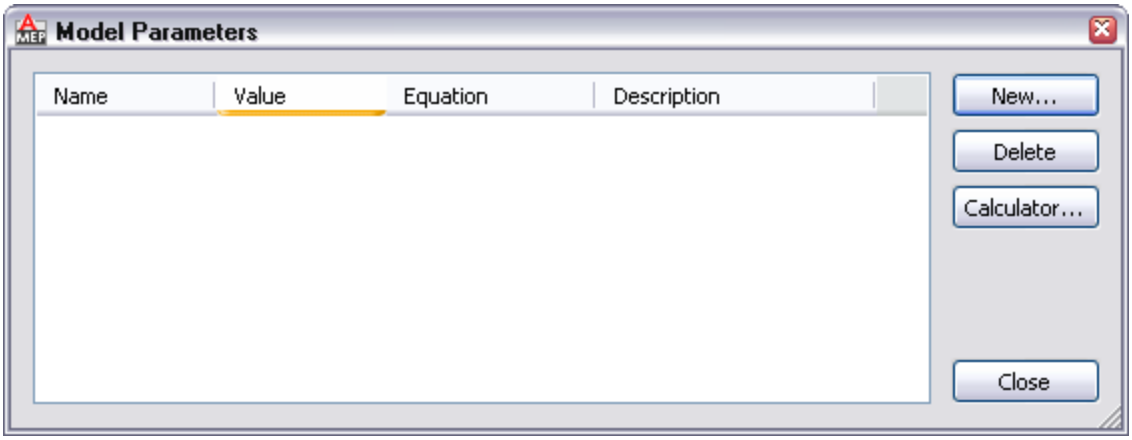
This is the final result. We have a 3D Body made by ttransitioning from one profile to another between planes.

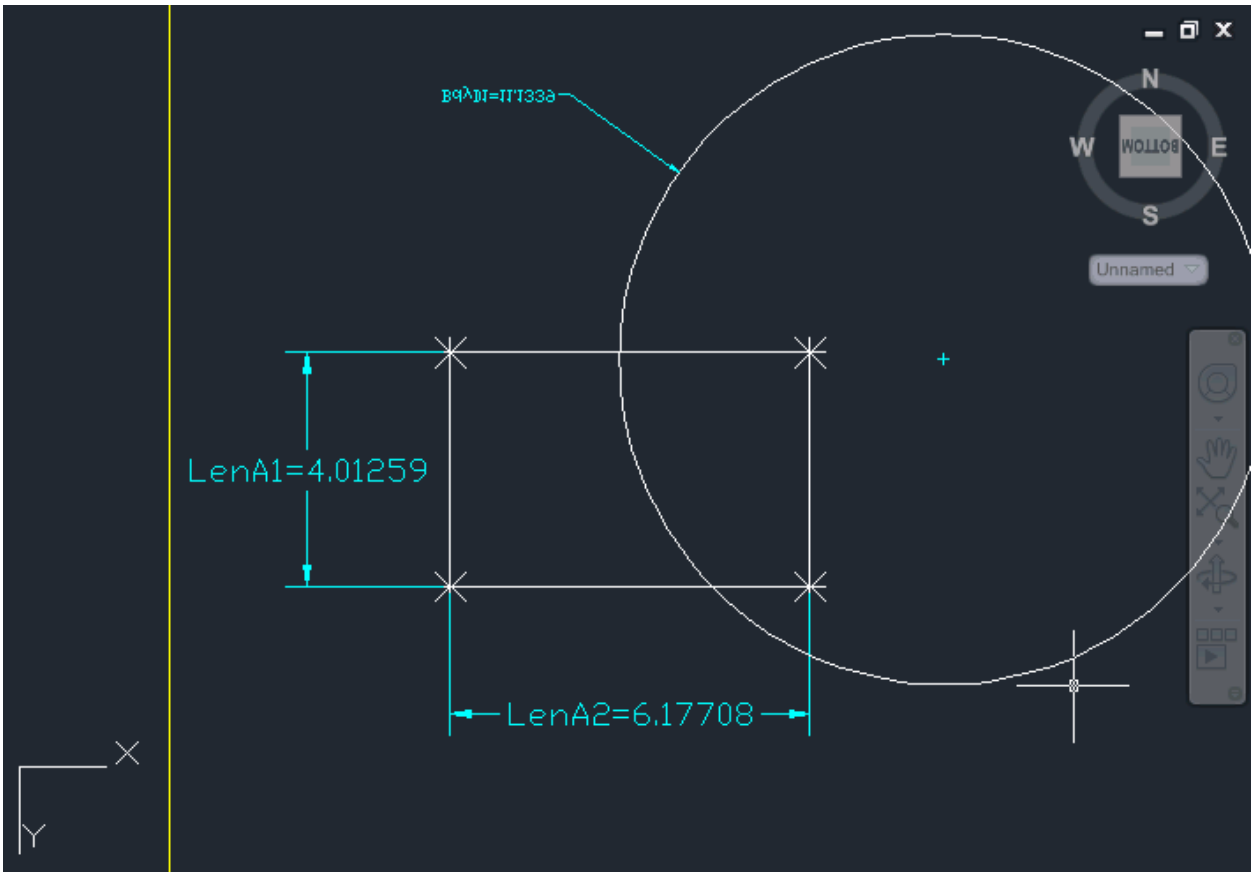
The Extrusion and Transition Modifiers, are linear in the sense that they come perpendicular out of the plane. If you want a profile to follow an arc or other route you must use a Path Modifier. The Path Modifier is the quickest and easiest method to get a 3D Body. Many fittings are made on one plane, with simply a profile and a path. It does not have to be complicated. A tee fitting is three paths and a rectangular profile. An elbow is an arc, maybe two straight legs, and a profile.

Model Parameters

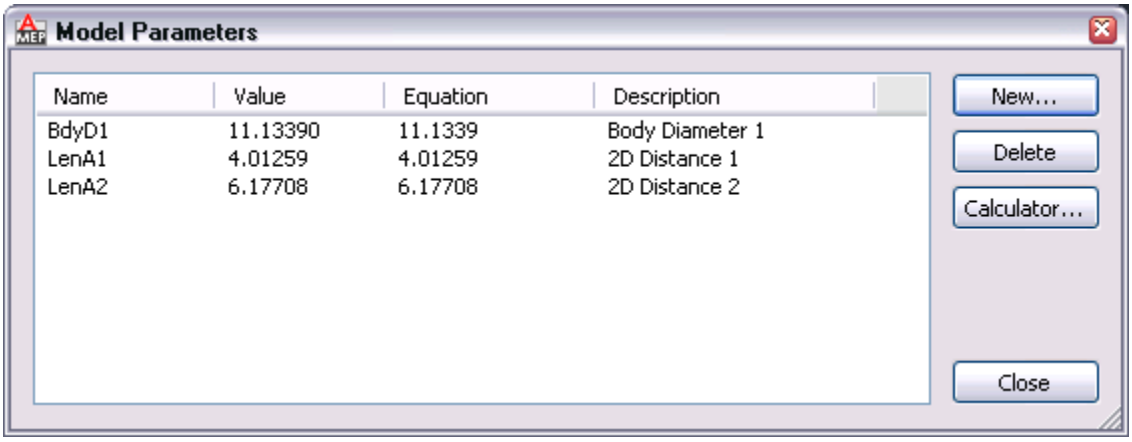
Let's discuss Model Parameters. Model Parameters do not exist until we add some geometry with geometry dimensions.



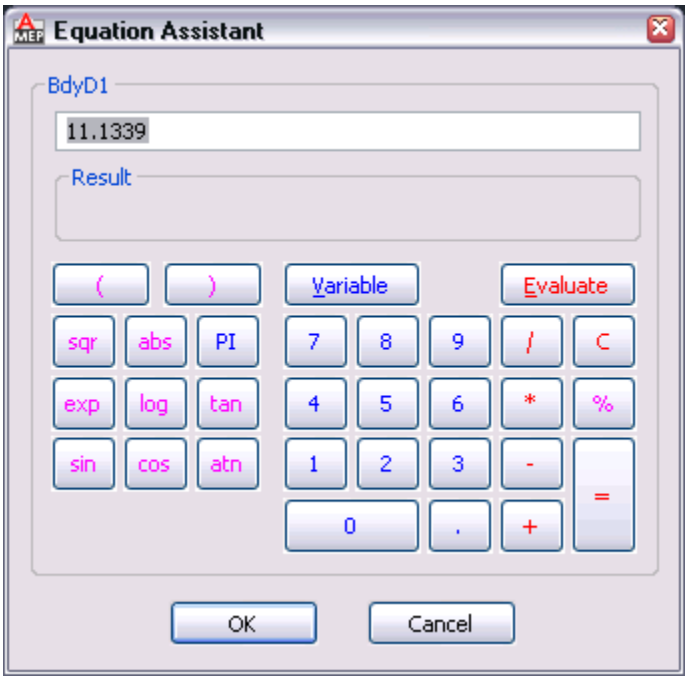




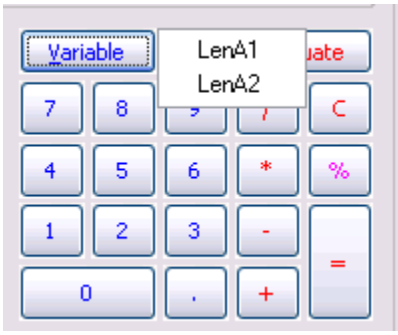
We added a rectangular profile to the Top plane, and a round profile to the Bottom plane. We added two Distance dimensions to the rectangle and a Diameter dimension. Notice when the dimensions insert, content builder automatically names the dimensions, LenA1, LenA2 and BdyD1. It not only adds these labels to the dimension lines, it adds them into the tree under the Model Parameters.



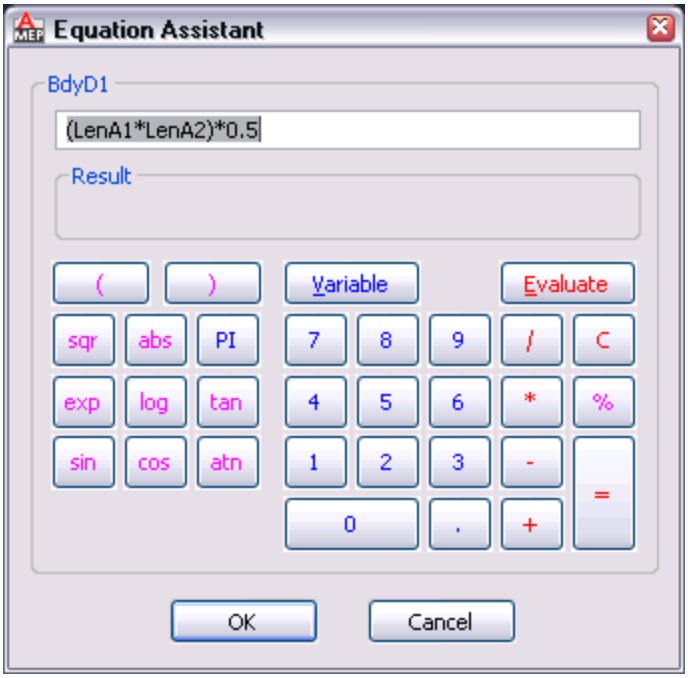
In this dialog, we can create new or delete or edit Model Parameters with the calculator button. Why would we create New Model Parameters? Well, BdyD1 is not very descriptive and when we pull out the Manufacturer Cut sheets, the cut sheet will have it's own nomenclature. So create new parameters with the abbreviations used on the cut sheet. Then set one equal to the other with a formula. So let's talk formulas. Select BdyD1 on the left and click the Calculator button.



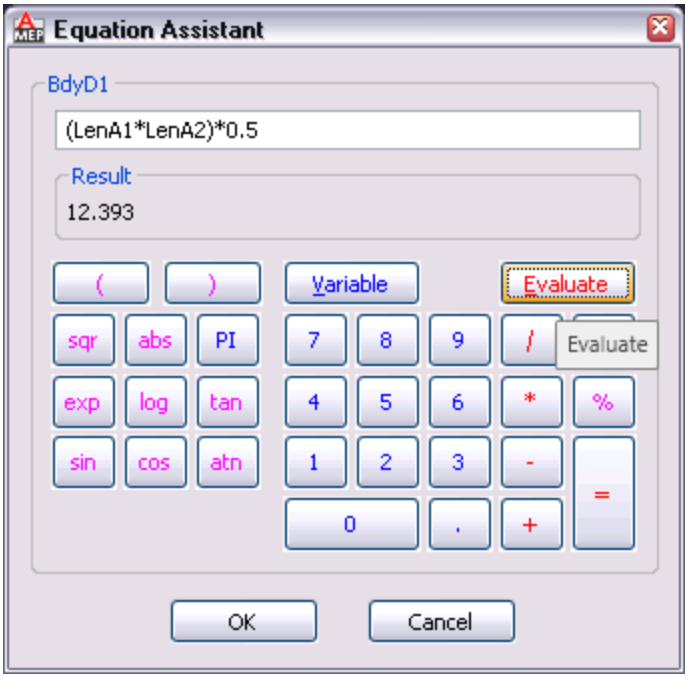
The equation assistant pops up. This allows us to insert a variable, LenA1, LenA2 and BdyD1 are variables. Clicking on the “Variable” button pops up:



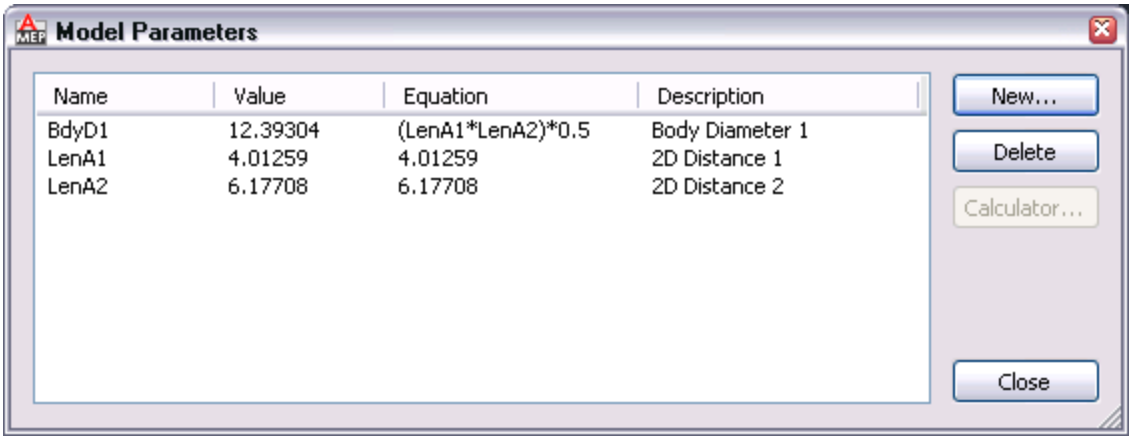
This allows us to select from the list of remaining variables. It inserts “LenA1” on the text line. In the image, we are editing BdyD1, so any equation we type, will set it equal to BdyD1.



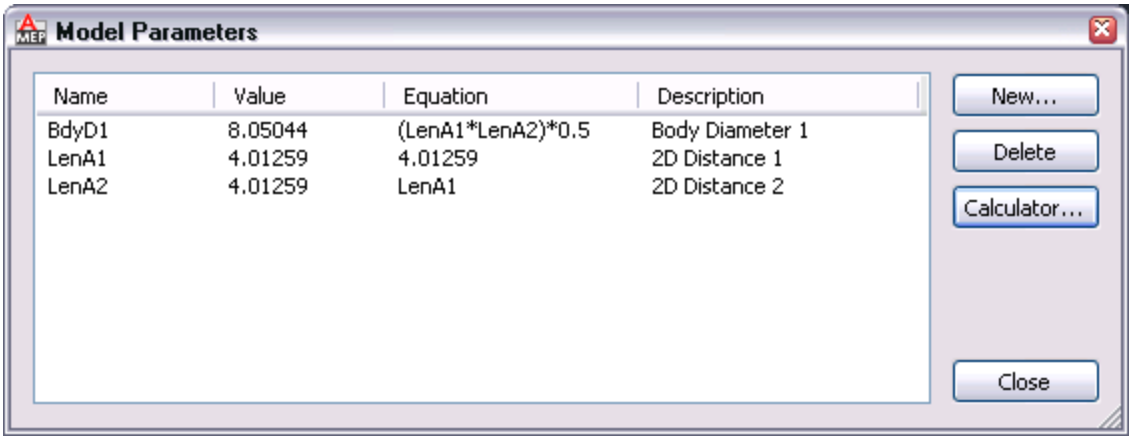
We can quickly create formulas like (LenA1 * LenA2) * 0.5, and it sets this equation equal to BdyD1.



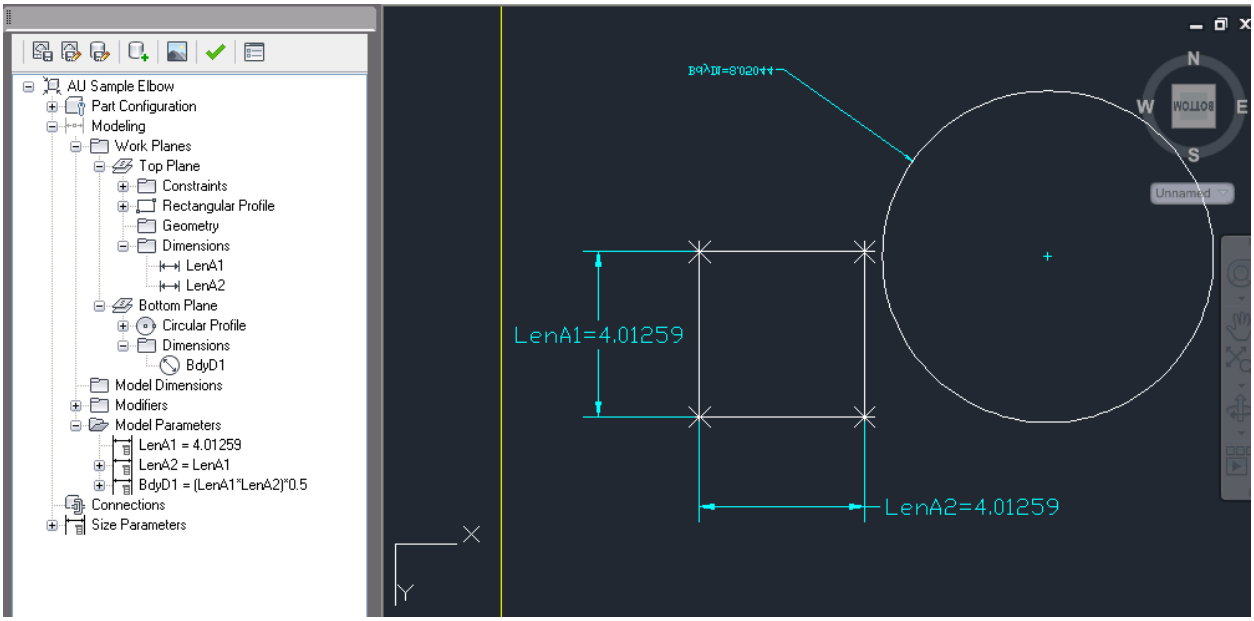
Click the “Evaluate” to see the “Result,” prior to exiting the assistant.



You can see the entered formula in the Equation column, and the result in the Value column. The Description field is directly editable, and can help you if you add reminders describing what the dimension controls.



Here we have set LenA2 equal to LenA1. This will make our rectangle profile square.



Here are the results.

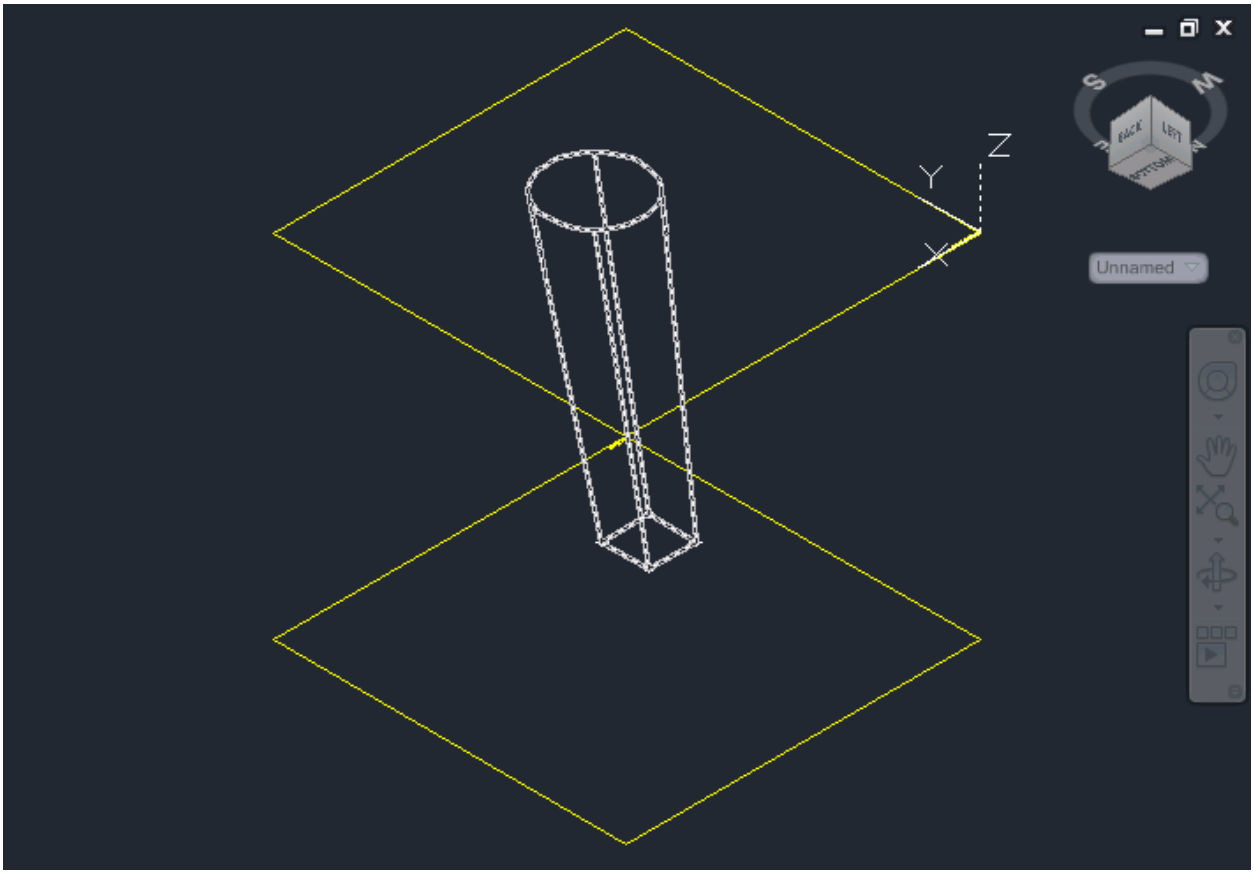
We need to discuss what to dimension and what to leave alone. When we later add a connector, like a rectangular duct connector, content builder will add width and height dimensions automatically after prompting for points. Since we do not want to over-constrain our part, we should NOT dimension the geometry that drives the connector shape. This will become apparent in our example section further below.

Connections

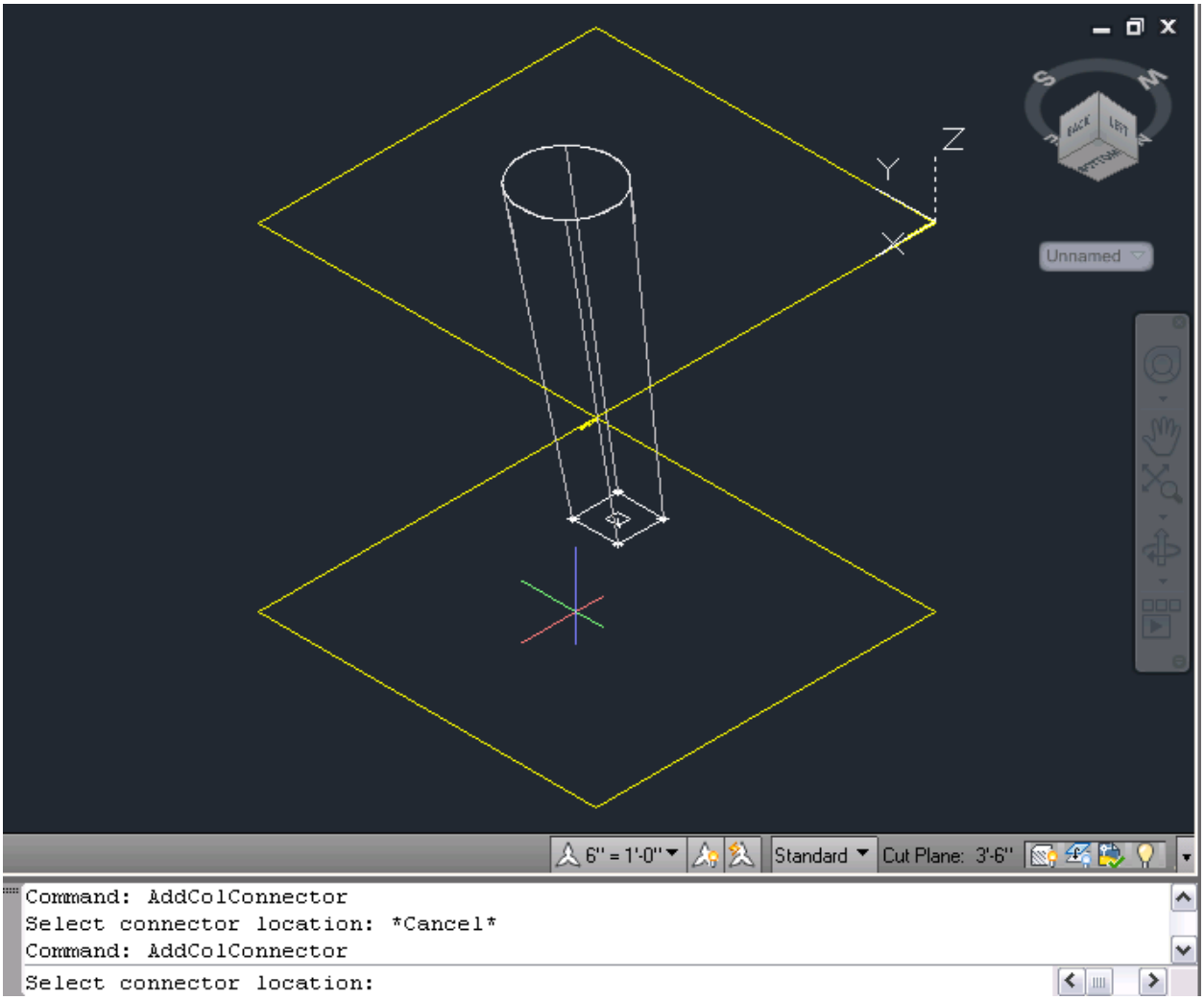
Let's discuss Connections.

Earlier, we made a rectangular and circular profile and added dimensions. We then added a Transition Modifier between them. We did this to demonstrate the concepts, and now we need to demonstrate connectors. Well, when we add connectors, it asks us for points that define the width and height or diameter. It will add those dimensions and we risk over-constraining the model if we have our own dimensions driving that geometry. For this demonstration, we will delete all dimensions from the two profiles.

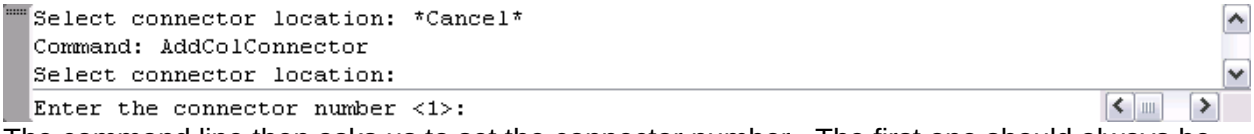
Before we add a connector, we don't want to be in Plan view. We are placing a connector on a 3D Body and need to see the complete body. Switch to a SE Isometric view. Right-click on Connections and select Add Connection.



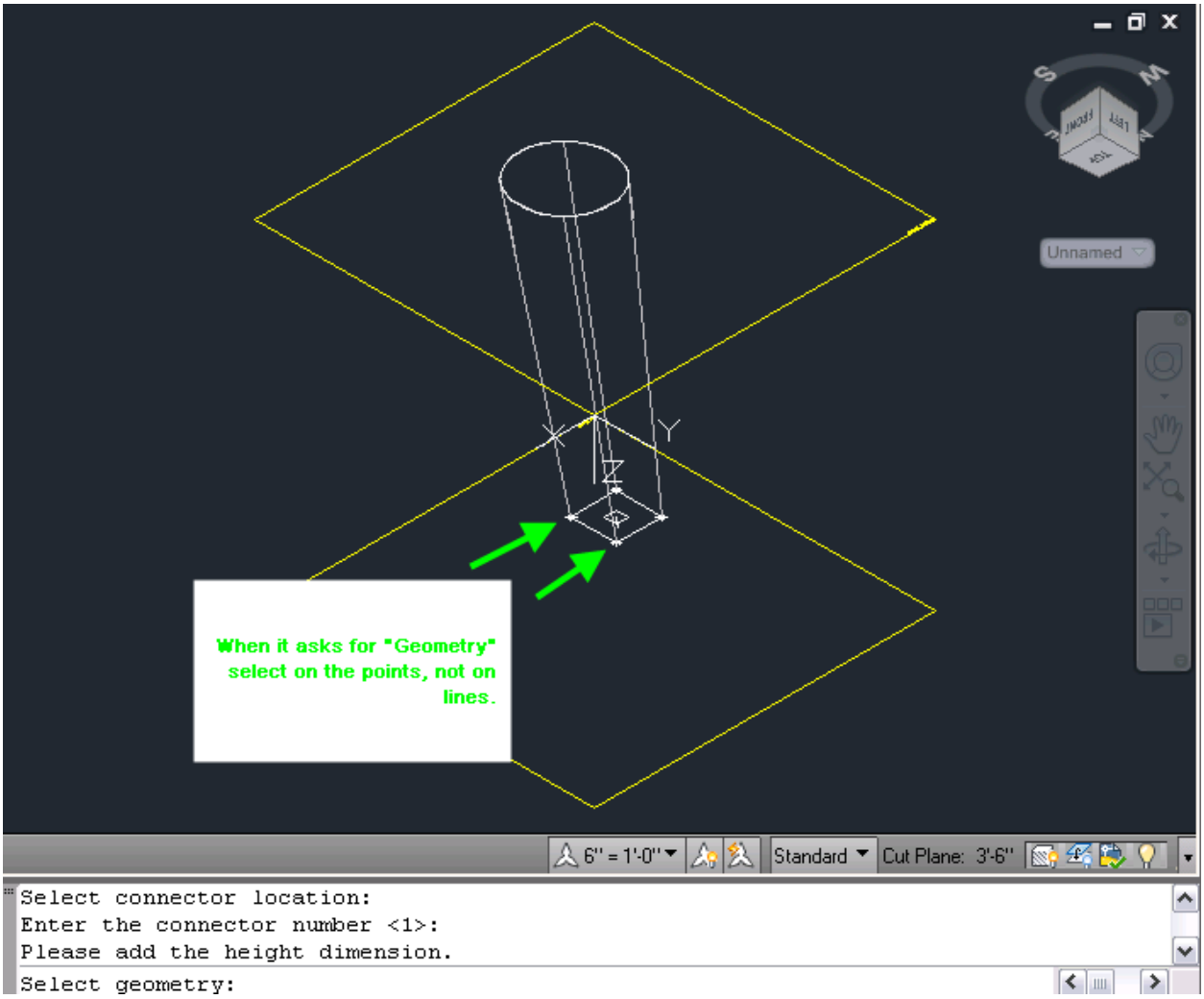
We will be adding a connector to the 3D Body at the rectangle end.



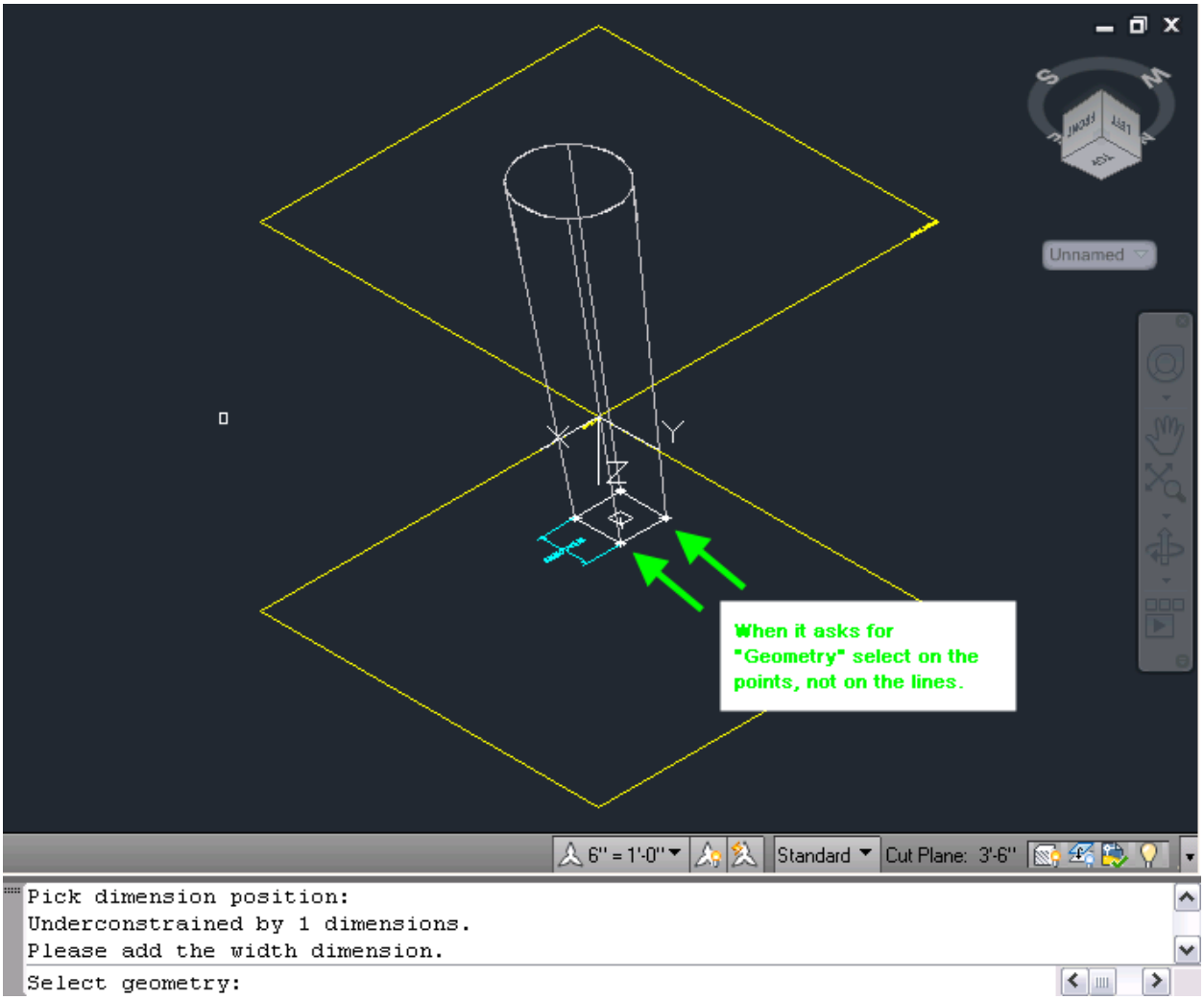
After we select Add Connector, the command line prompts us to select a location for the connector. As we hover our curser near the rectangle, we see a glyph, looks like a rectangle with an arrow pointing away from the 3D Body. This is very similar to adding a connector in the Block Based wizard. The arrow should always point in the direction that the duct or pipe approaches the part. It points away from the 3D Body.



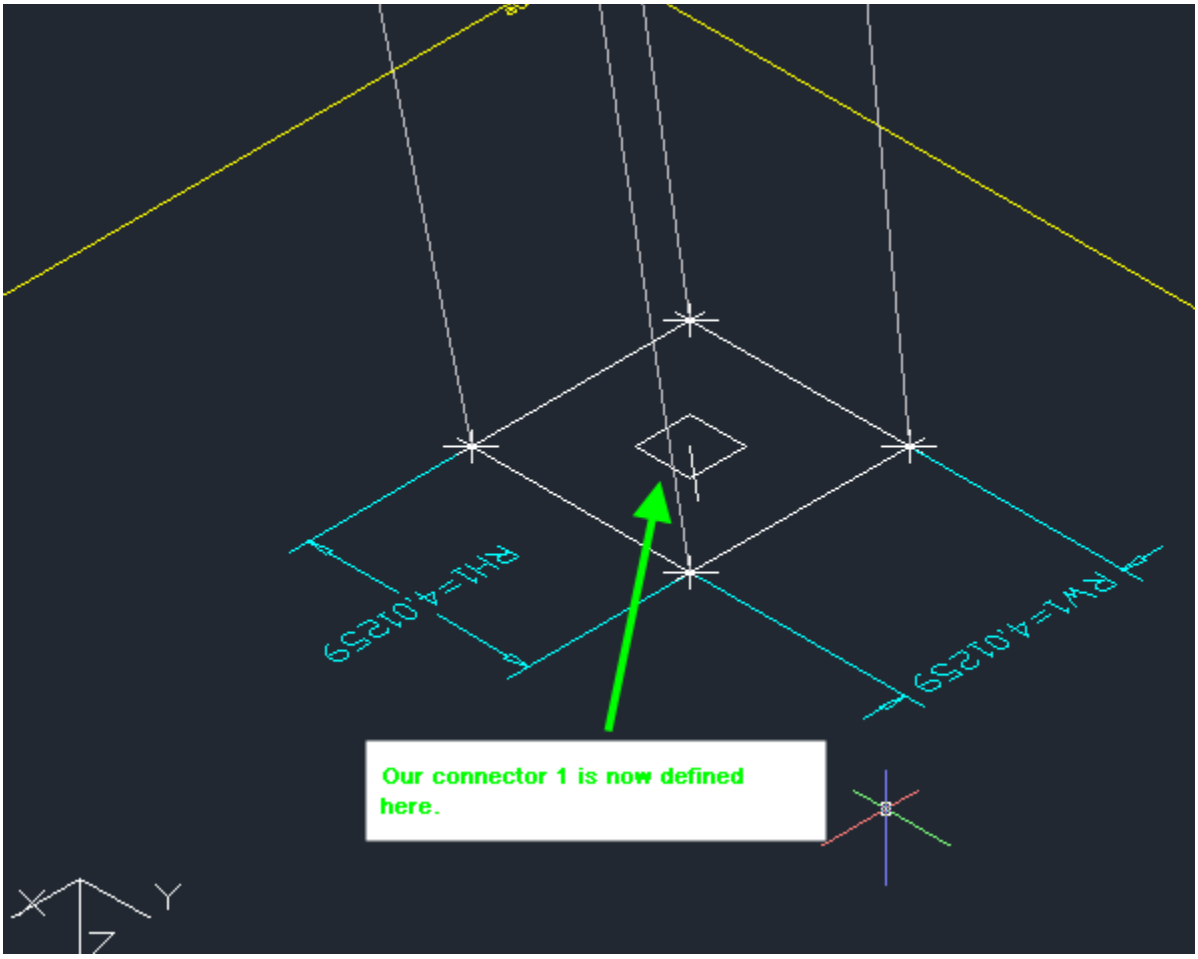
The command line then asks us to set the connector number. The first one should always be connector 1.



Specify the two points that define the height dimension for the connector. Then pick a point off to the side to place the dimension.

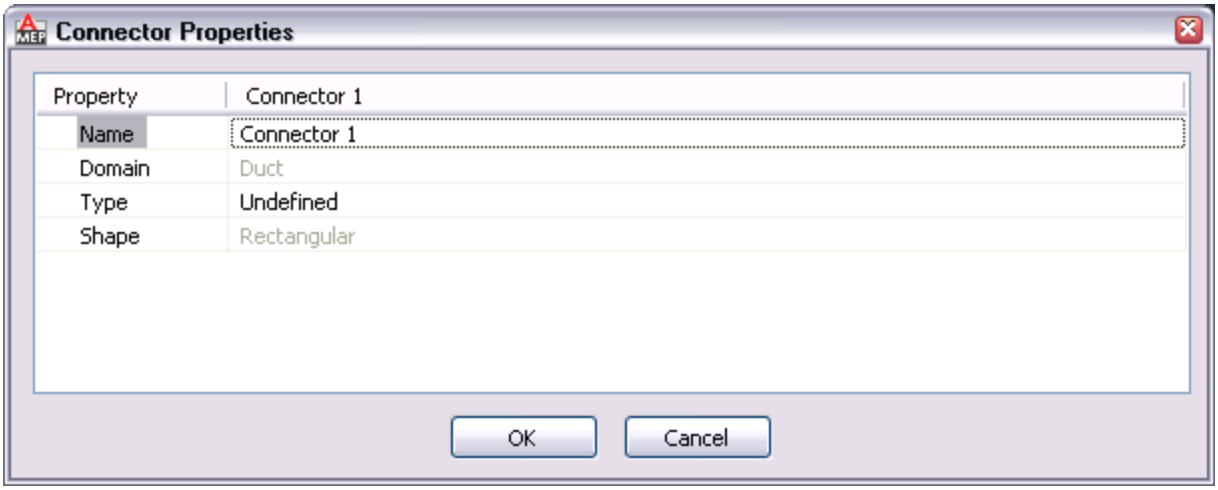


Specify the two points that define the width dimension for the connector. Then pick a point off to the side to place the dimension.

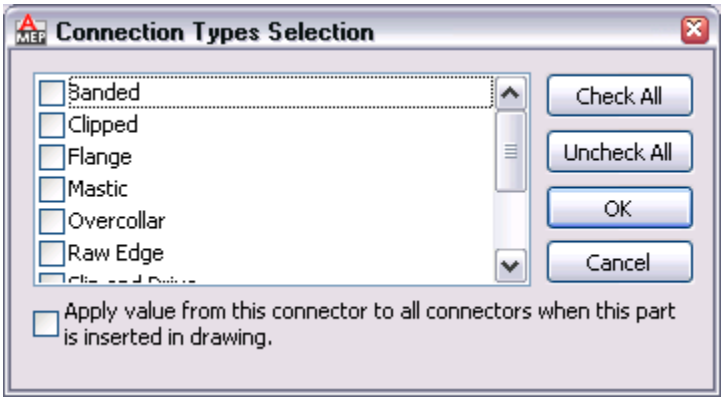


Notice that we now have a connector and two dimensions, RH1 and RW1. Maybe you have noticed these parameters in the DUCTFITTINGADD dialog for defining the connector width and height.

Now that we have a connector we can explore the connector edit dialog.



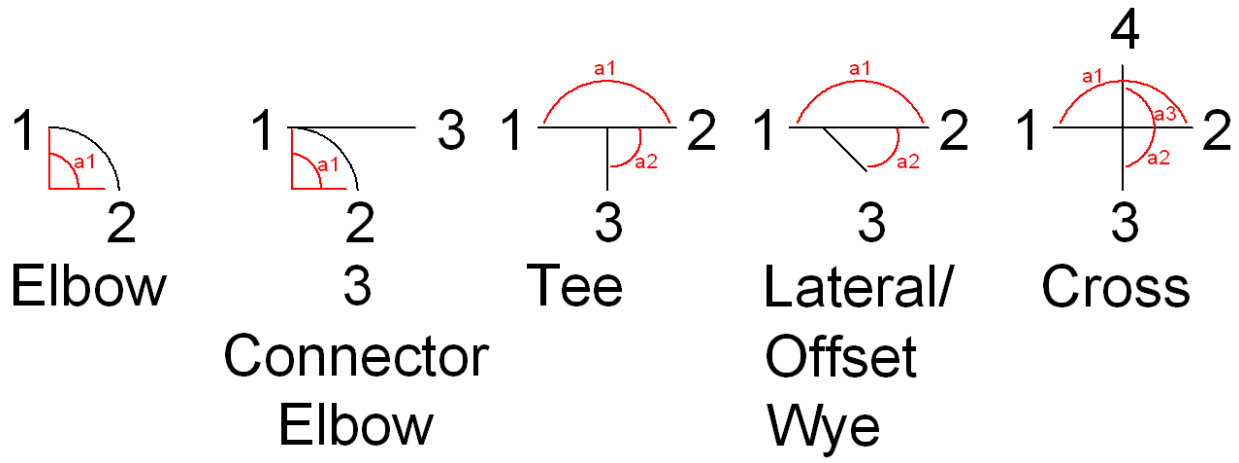
Here we can add a description in the connector name field.



Clicking on Undefined brings up this dialog. Leaving the checks all blank is fine and allows the connector to connect to any duct connection type. Same happens if we check all. But if we only check a few, we are limited to those connection types. The "Apply value from this connector to all connectors…" just helps us to keep all connectors on this part identical to each other in what they connect to.

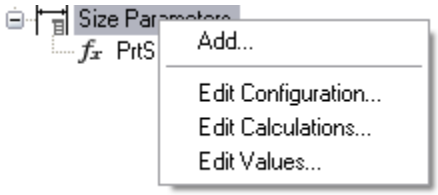
FITTING RULES

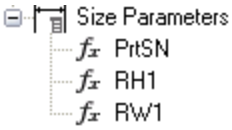
NOTE: For fittings, the fittings must be drawn a specific way or else the validation will fail. Content builder needs the connectors facing a specific direction in order to add them during AutoLayout. "AutoLayout requires that fittings are built with connectors in a specific order. In Addition, each part must have path angles (A1, A2, A3) defined in a specific order as shown. Male x Female Fittings require the Male Connector to be located at Connector 1.



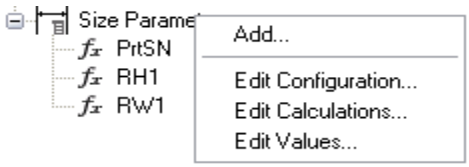
Size Parameters

Let's discuss Size Parameters.

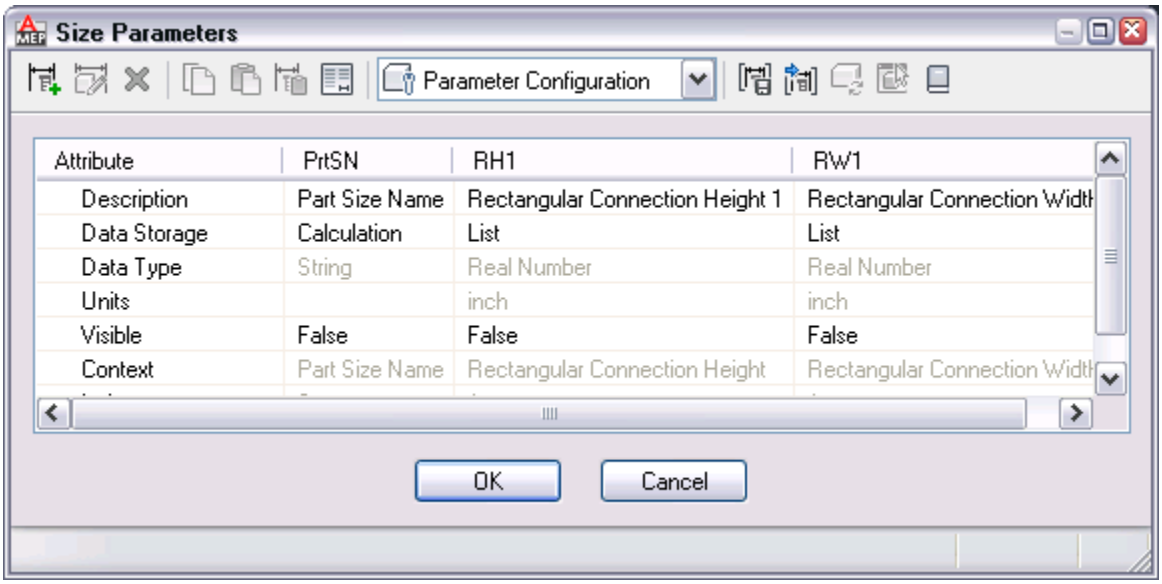




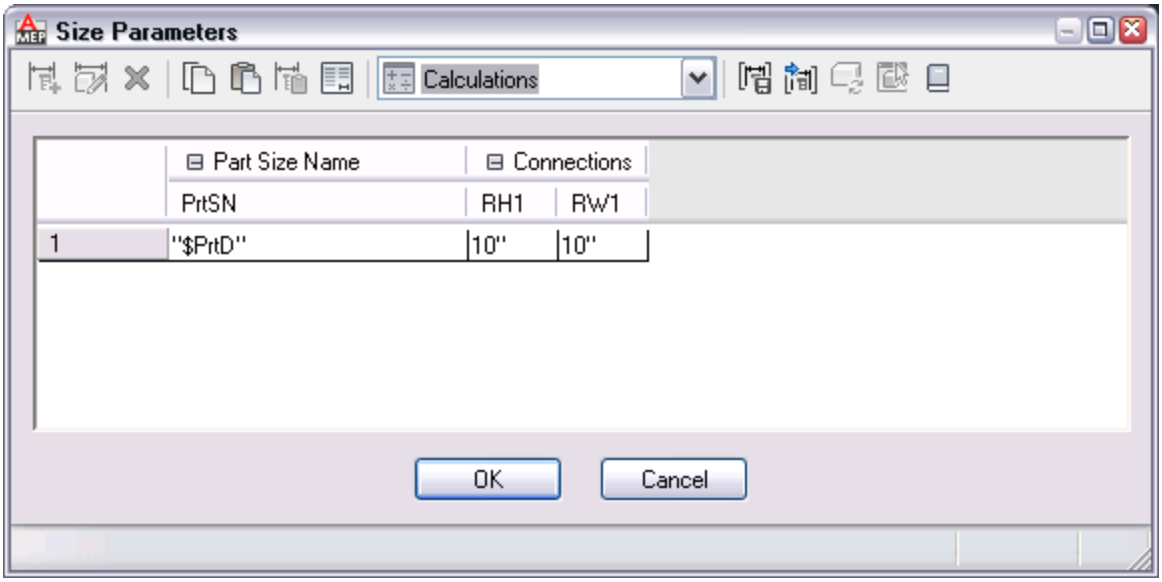
When a Model Parameter is created, it also creates a Size Parameter. Size Parameter formulas are different from the easier to understand formulas in Model Parameters. If a Model Parameter has a formula, it adds that formula to the equivalent Size Parameter and converts it to a formula that Size Parameters can understand. All Model Parameters that have values, are set to a "constant" type by default. If you want that variable to be driven by a list or table, you must change the configuration.



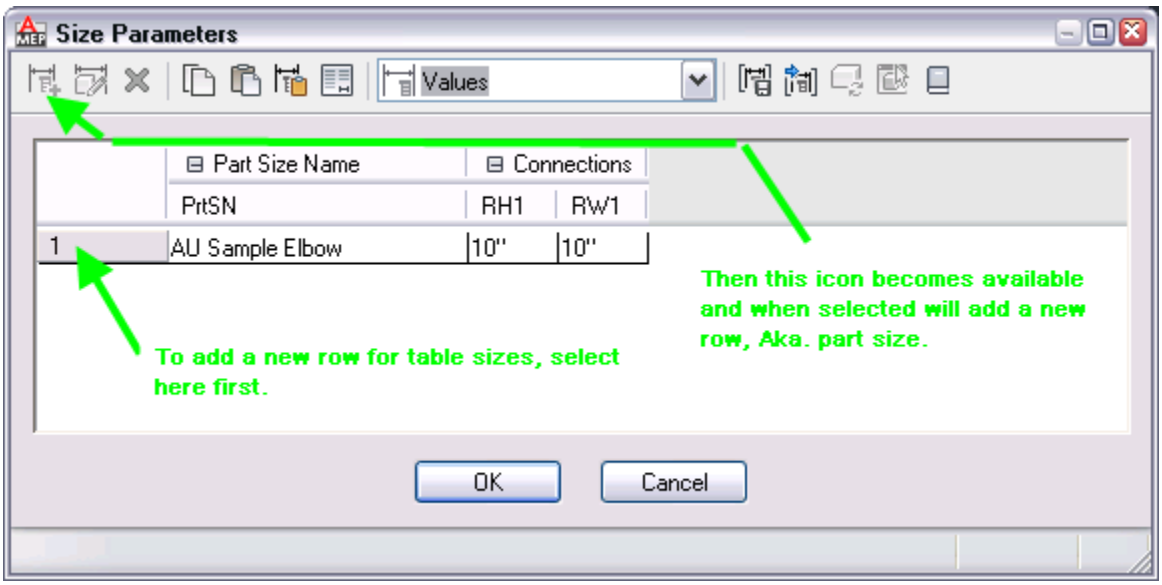
When we right-click on Size Parameters, these are the options.



This comes up when we click Edit Configuration. Here, you will want to set the "Data Storage." If a Size Parameter uses a formula, the Data Type says Calculation. Otherwise it has a value and can be a constant, a list or a table. A constant does not change between different sizes of your part. A List means that you get the same list of sizes regardless of the size of your part. A Table is more like a spreadsheet or a manufacturers cut sheet. For a specific size piece of equipment, dimension A = 12, dimension B = 12.5, dimension C = 2. For the next size piece of equipment, dimension A = 14, dimension B = 12.5, dimension C = 4. And each specific equipment size will have it's own set of values called a table. The other setting here that needs to be thought thru, is the "Visible" setting. Do you want this parameter visible in the PARTADD dialog? Any parameter can be visible. Only list and part size (table) parameters are editable in the PARTADD dialog.

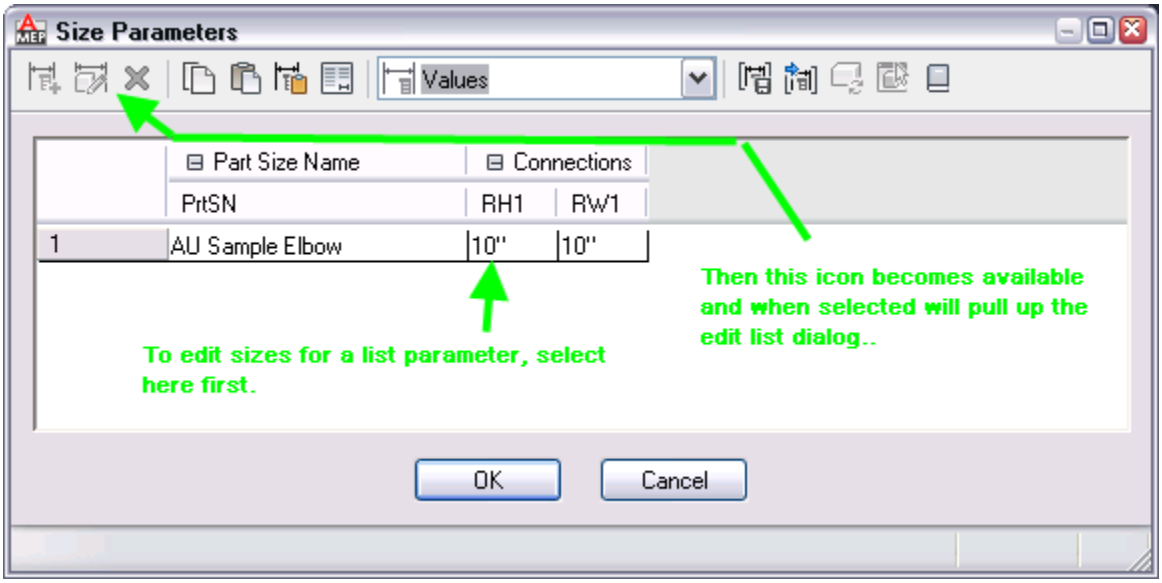


This comes up when we click on Edit Calculations. Don't try to edit the formulas here, they are much easier to edit in the Model Parameters.



This comes up when we click Edit Values. This is where we add our values from the manufacturer cut sheets. If our parameter is a list data type, we can build our list here. If we have some table parameters (must have more than one), we can add a new row for each part size, thus building up a table of parameters. Select inside the cell to edit a table value. Values that are calculated cannot be edited. We don't have any table or list types here or else the first two icons at the top would be available. The second icon in, pulls up the edit list dialog. It

becomes available after selecting a cell and then selecting the second icon in.



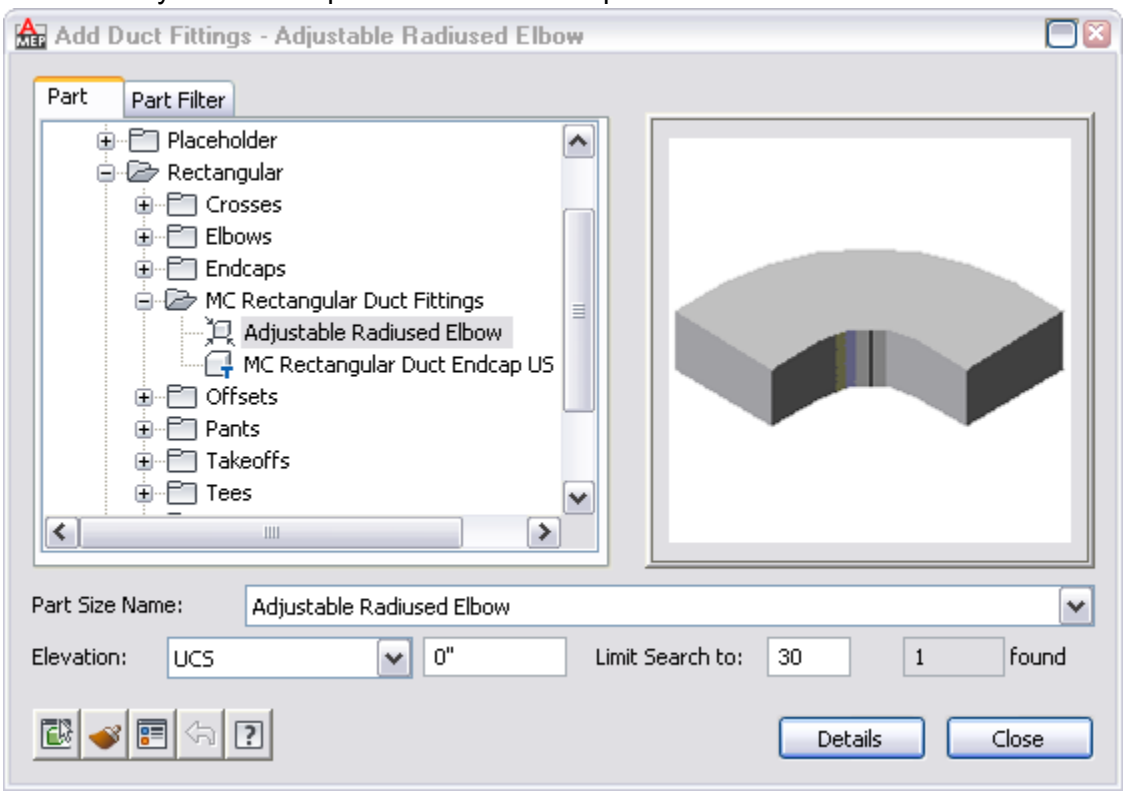
After adding and editing the values in the Size Parameters, we check the part validation one last time. We go to the Part Options and maybe remove the Hide Part Flag. Save the part and exit by clicking the X in the gray area above the top row of icons. Go add your part to model space. There are some instances where our part looks fine in content builder, and fails when inserted in model space. Our developers are working on these issues as we discover them.

Content Builder Is Easy!

You just need to know the rules of the road. If you have never seen an automobile, would you feel comfortable driving it? No. But let someone explain it to you; give yourself some experience and eventually you find that it becomes second nature. I'm going to repeat this throughout this whitepaper. Tell yourself this over and over and believe it.

Example – Duct Elbow

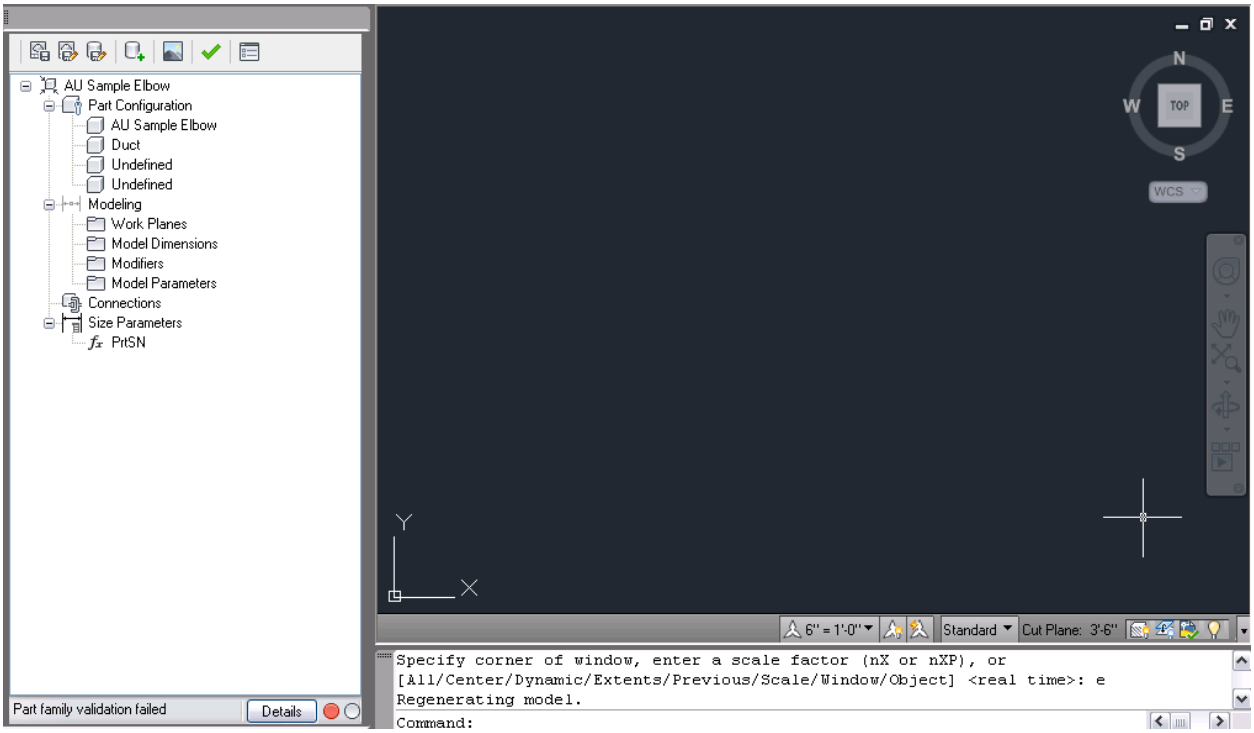
Let's put some of this together. Let's create an elbow like the one seen in the below image. We have already started our part named "AU Sample Elbow."



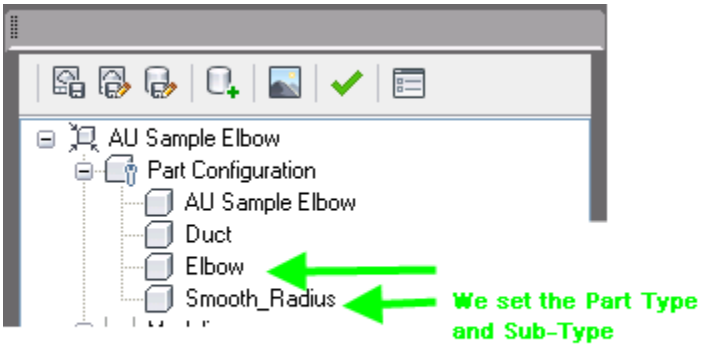
Each picture is going to be a step.

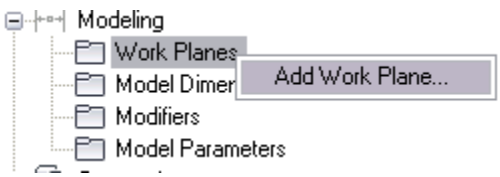
If you have been following along, your "AU Example Elbow" may be cluttered with geometry, dimensions, profiles and modifiers from the above sections. I'm going to delete the Top plane and any other planes, which will delete all objects created on the Top plane and pretty much reset our "AU Example Elbow" back to square one. So for our example here, we are starting at square one, inside Parametric Part Wizard.

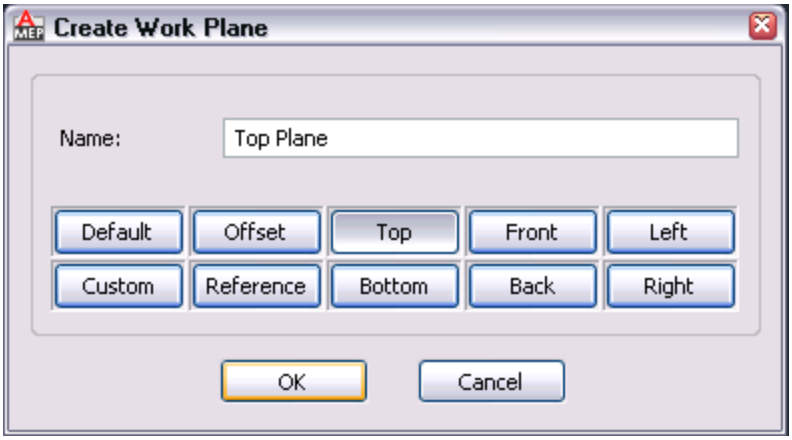
Our elbow seen in the image above, has an arc and two straight legs. We will be creating a rectangular profile and path lines. We will use the Add Path Modifier to generate the 3D Body.

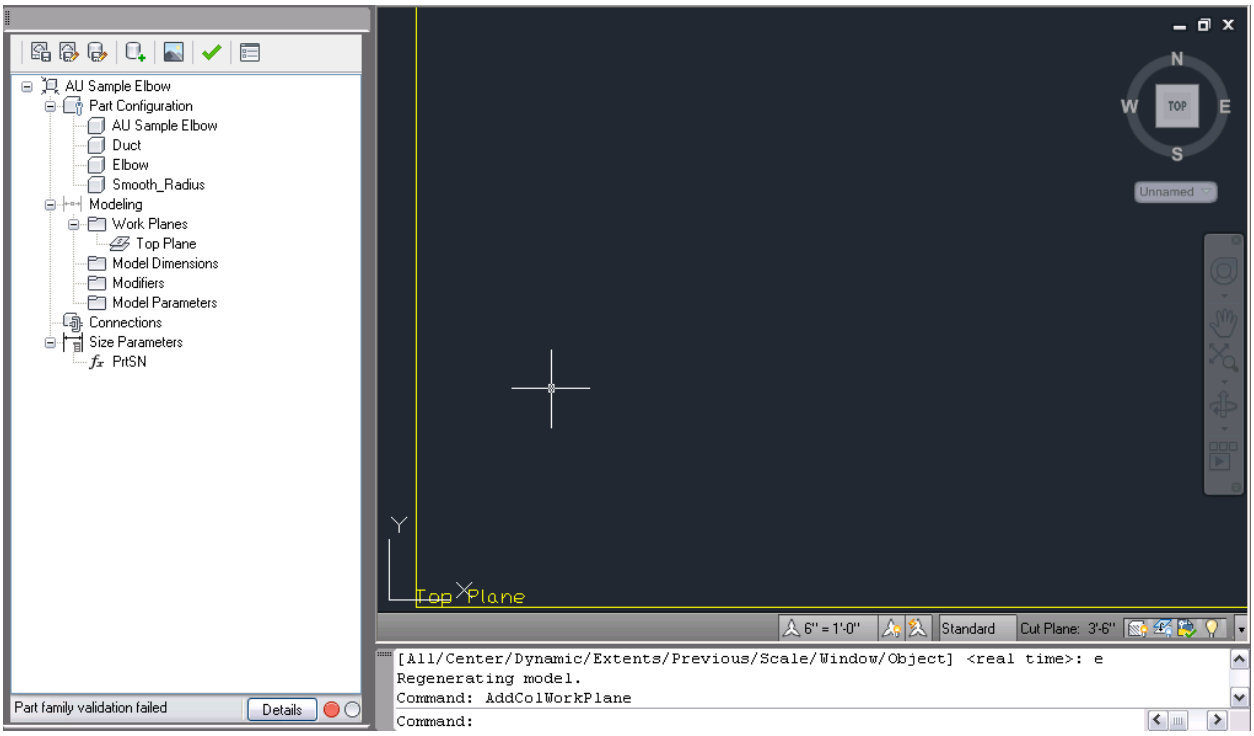


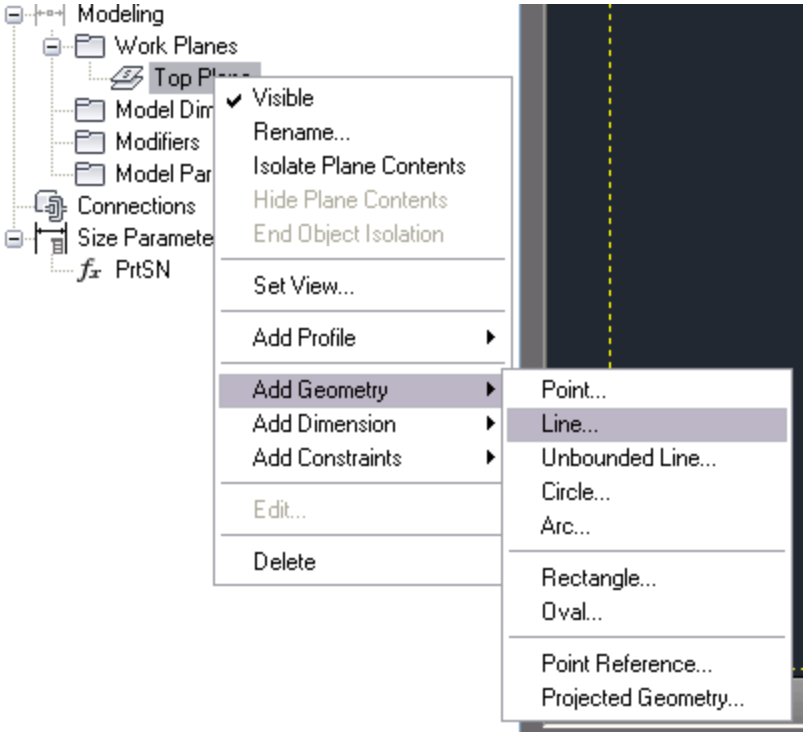
We are starting from scratch, as seen above.

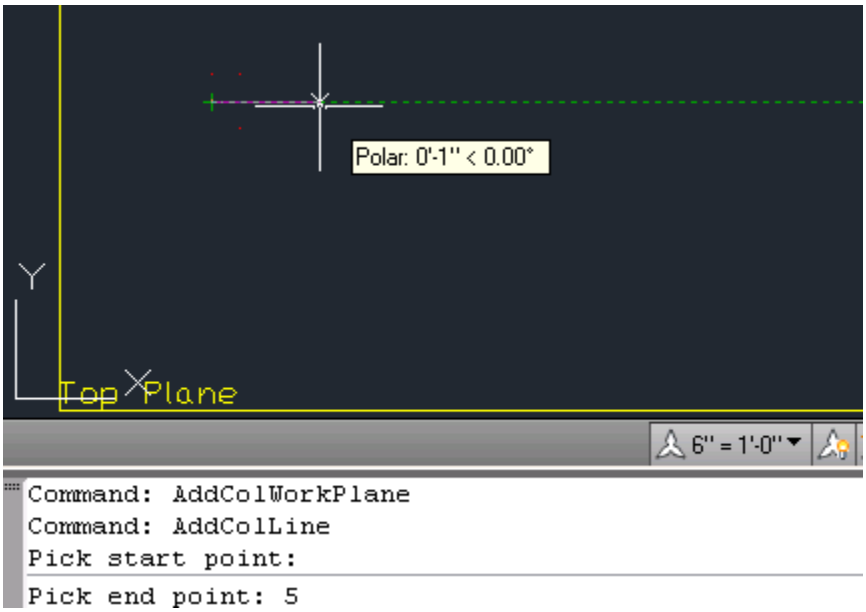


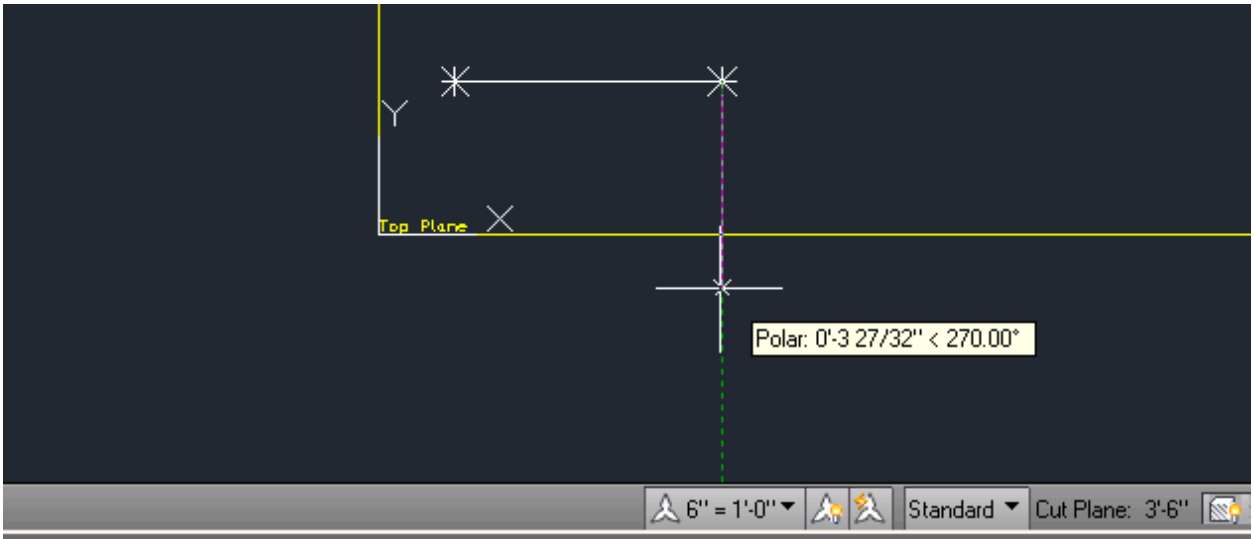



Create Work Plane
Name:
Top Plane
Default
Offset
Top
Front
Left
Custom
Reference
Bottom
Back
Right
OK
Cancel

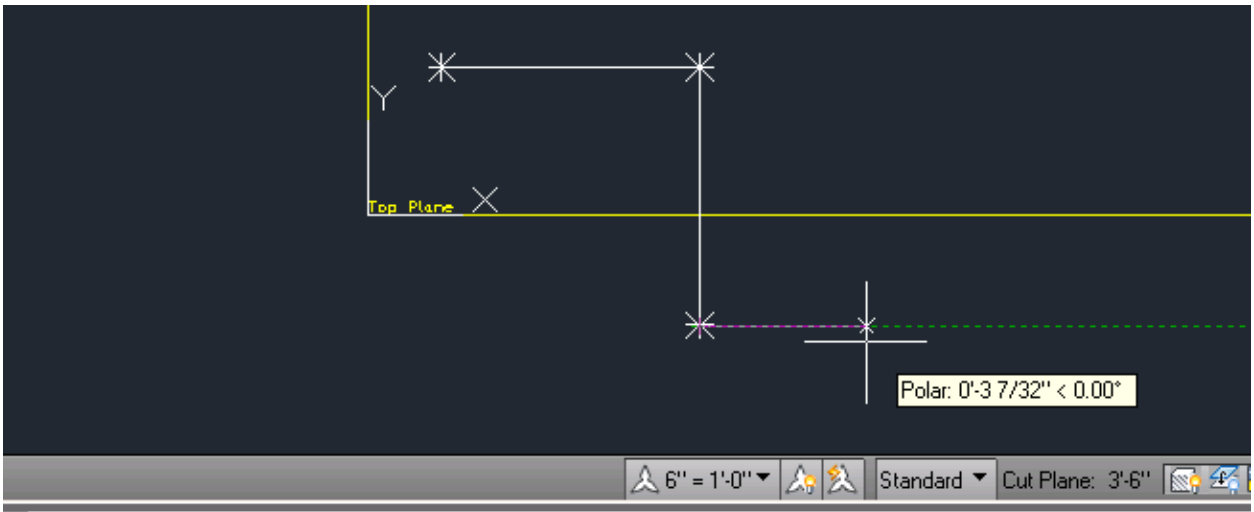

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Model Dimensions
Modifiers
Model Parameters
Connections
Size Parameters
PrtSN
Part family validation failed
Details
N
W
TOP
E
S
Unnamed
Top Plane
6" = 1'-0"
Standard
Cut Plane: 3'-6"
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: e
Regenerating model.
Command: AddColWorkPlane
Command:


Modeling
Work Planes
Top Plane
Model Dim
Modifiers
Model Par
Connections
Size Paramete
PrtSN
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Point...
Line...
Unbounded Line...
Circle...
Arc...
Rectangle...
Oval...
Point Reference...
Projected Geometry...

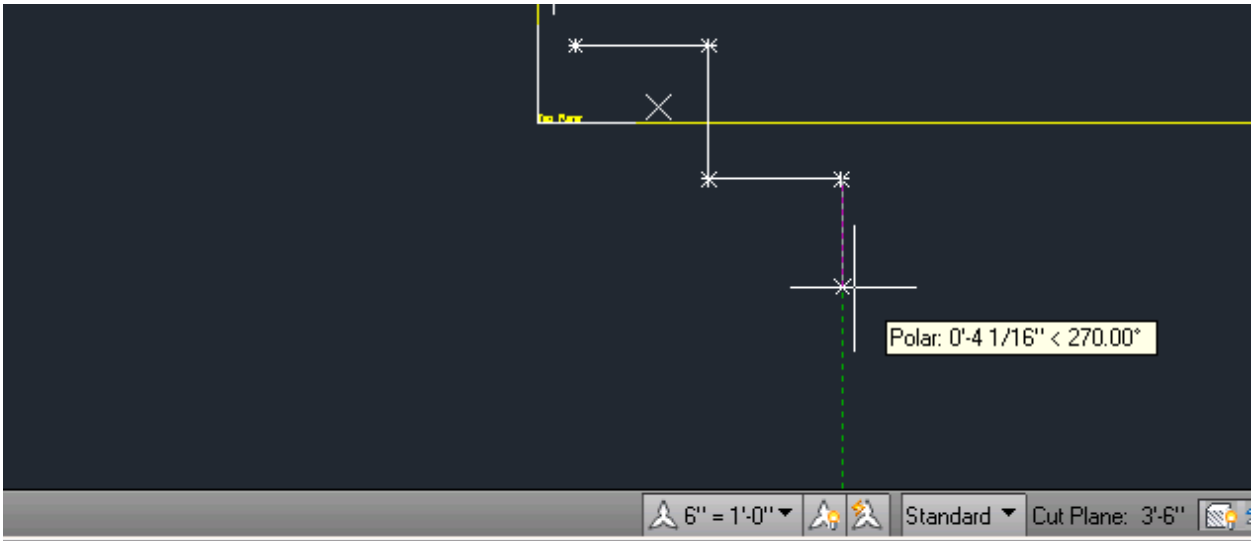

Polar: 0'-1" < 0.00°
Top
Plane
6" = 1'-0"
Command: AddColWorkPlane
Command: AddColLine
Pick start point:
Pick end point: 5

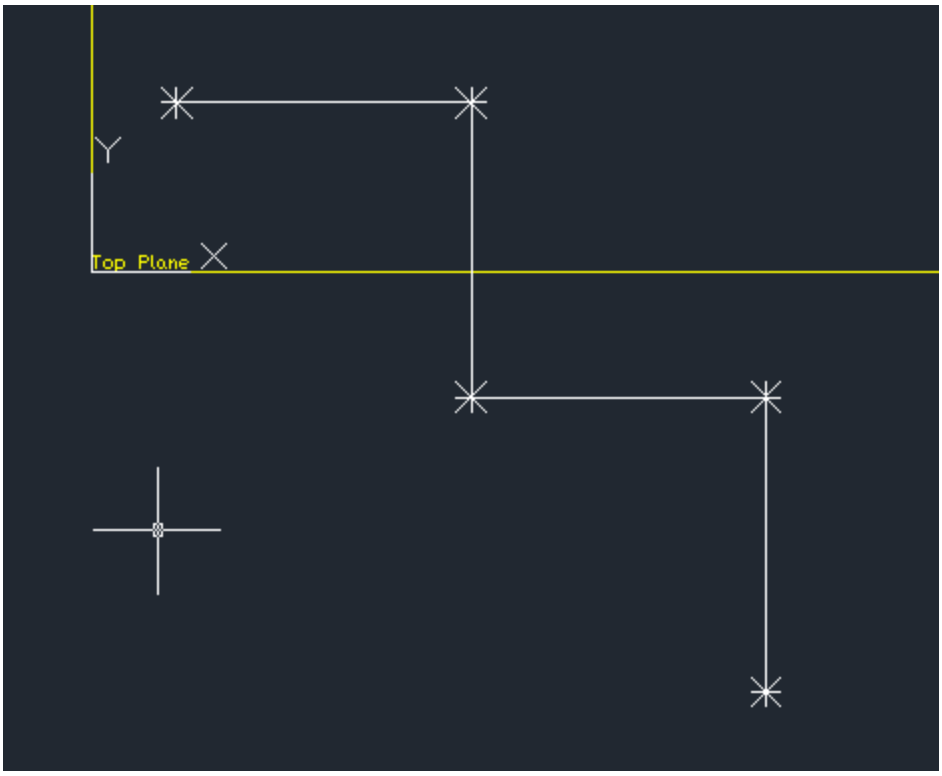


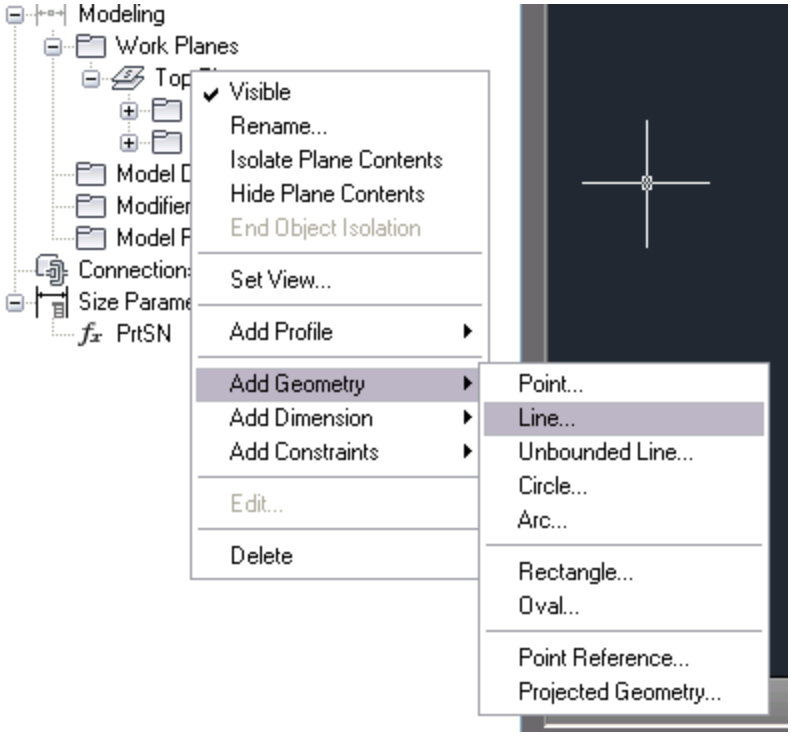

```
>>Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: .5x
Resuming ADDCOLLINE command.
Pick end point: 5
```

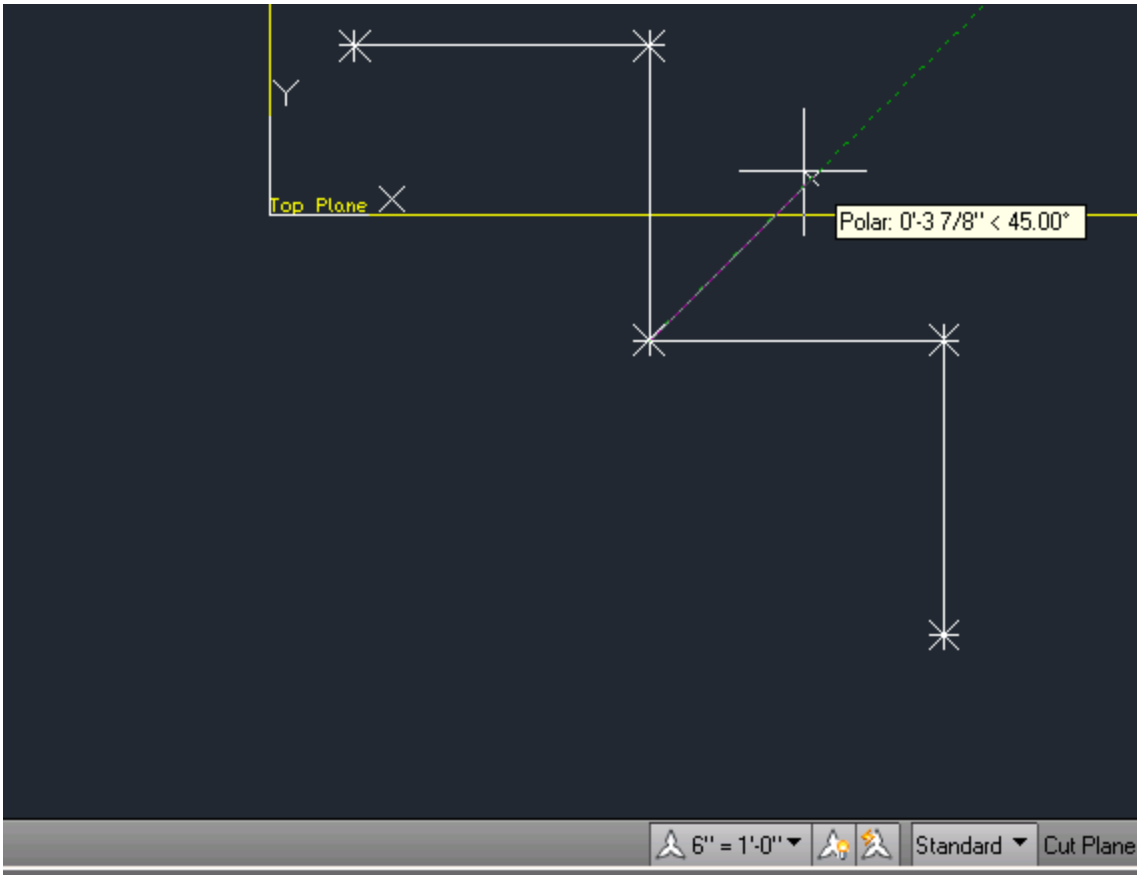



```
Resuming ADDCOLLINE command.
Pick end point: 5
Underconstrained by 3 dimensions.
Pick end point: 5
```

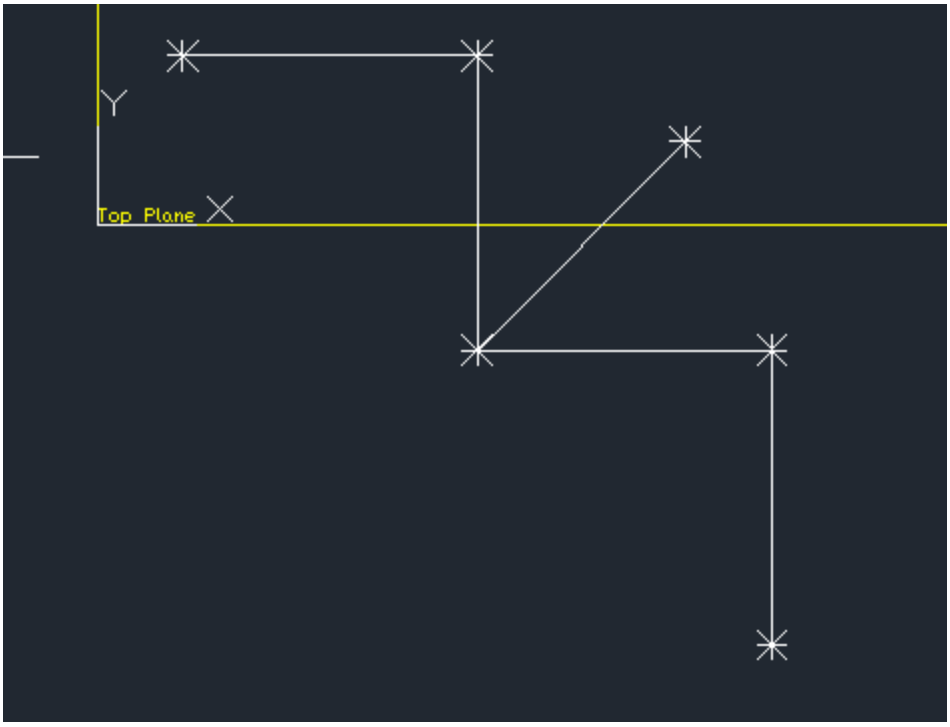

Polar: 0'-4 1/16" < 270.00°
6" = 1'-0"
Standard
Cut Plane: 3'-6"
>>Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: .5x
Resuming ADDCOLLINE command.
Pick end point: 5

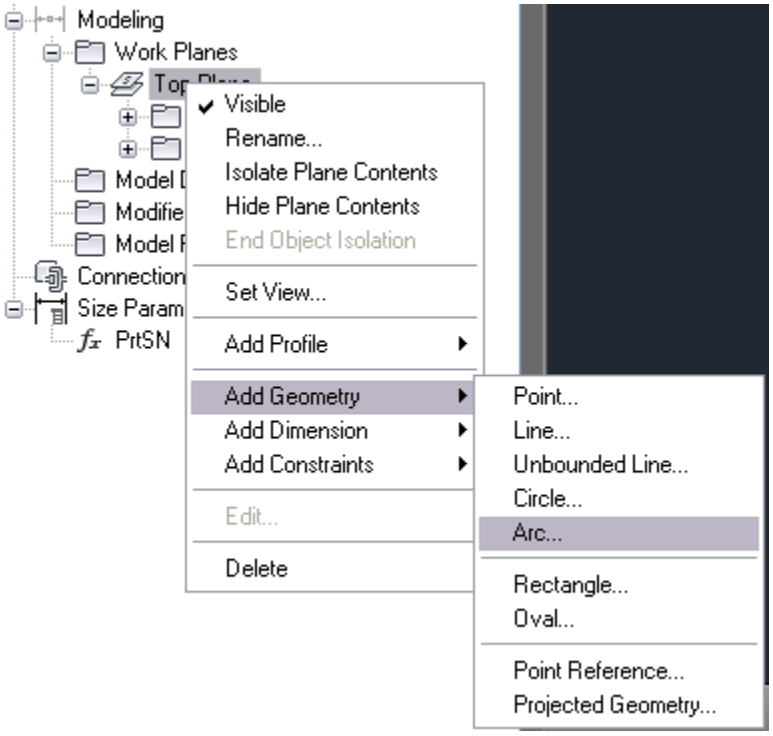

Top Plane

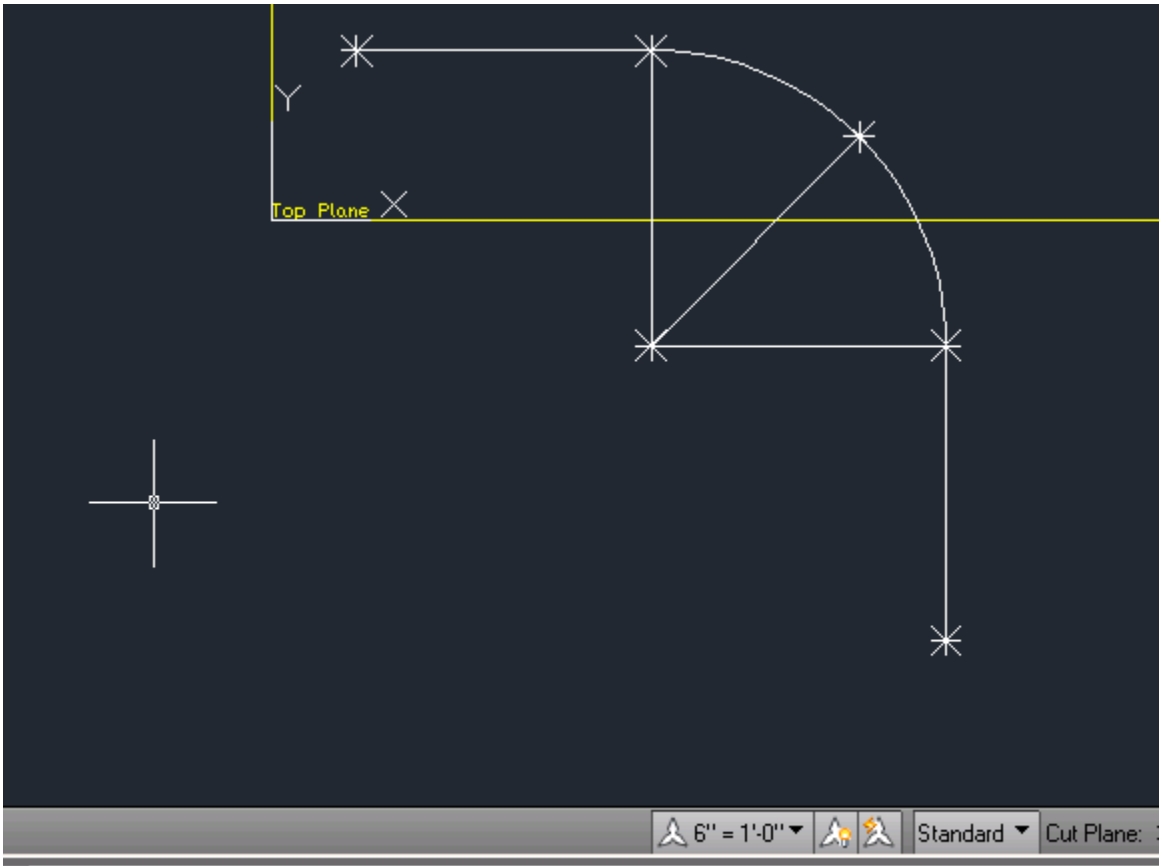

Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Point...
Line...
Unbounded Line...
Circle...
Arc...
Rectangle...
Oval...
Point Reference...
Projected Geometry...
Connections
Size Parame
PrtSN



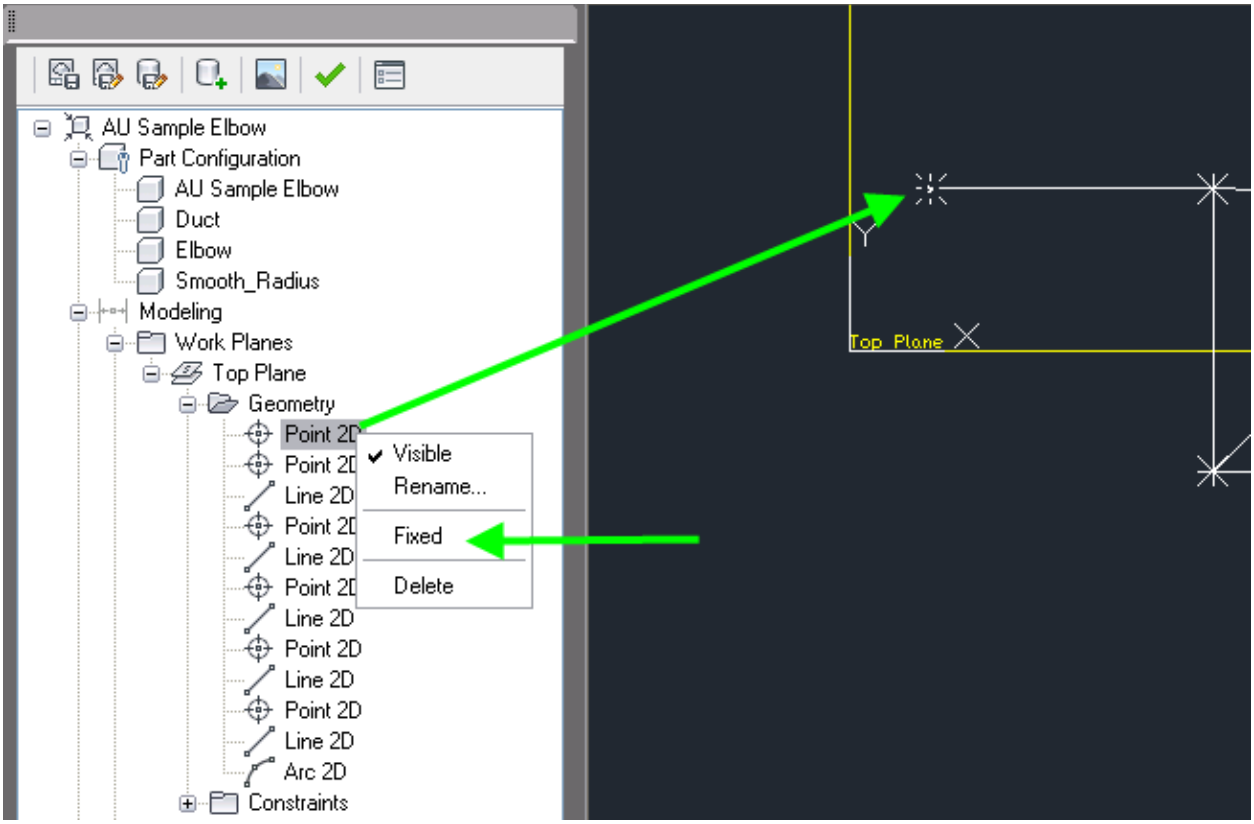

```
Specify first corner: Specify opposite corner:
Command: AddColLine
Pick start point:
Pick end point: 5
```

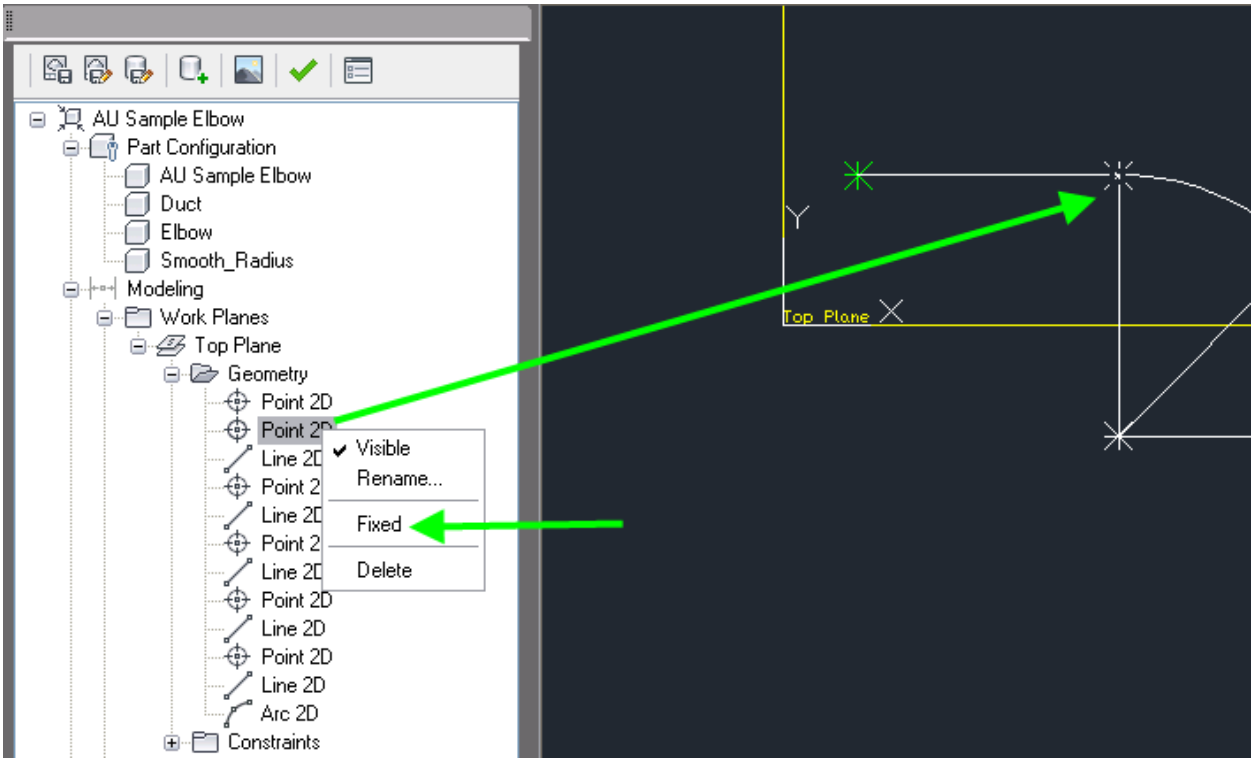

Top Plane

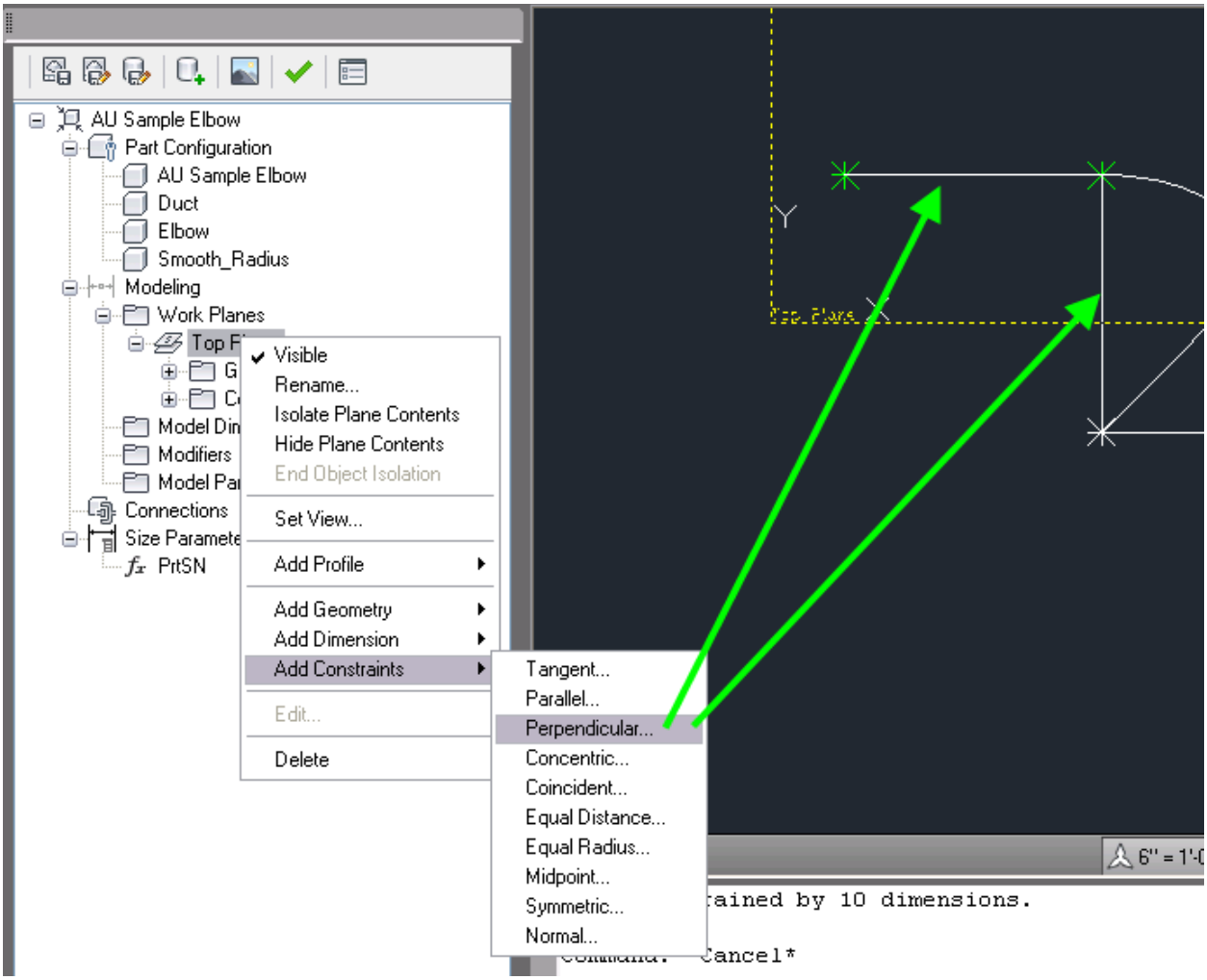

Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Point...
Line...
Unbounded Line...
Circle...
Arc...
Rectangle...
Oval...
Point Reference...
Projected Geometry...
Size Param
PrtSN

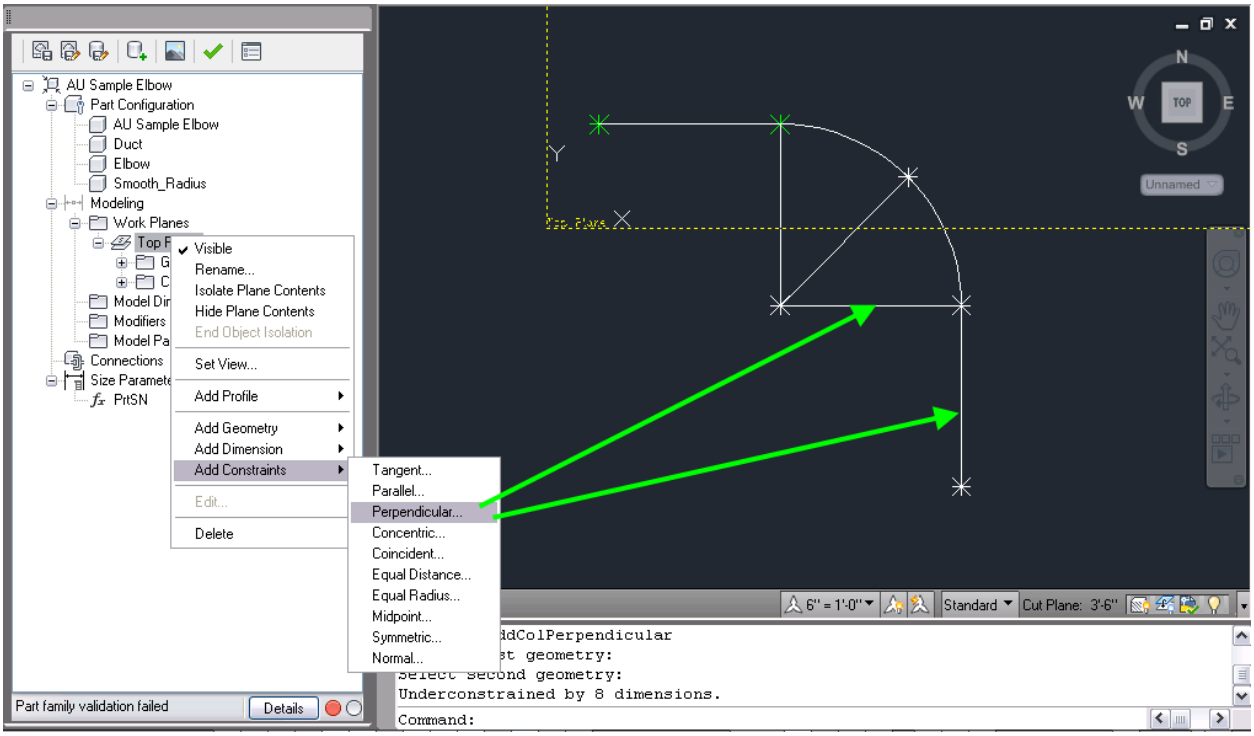


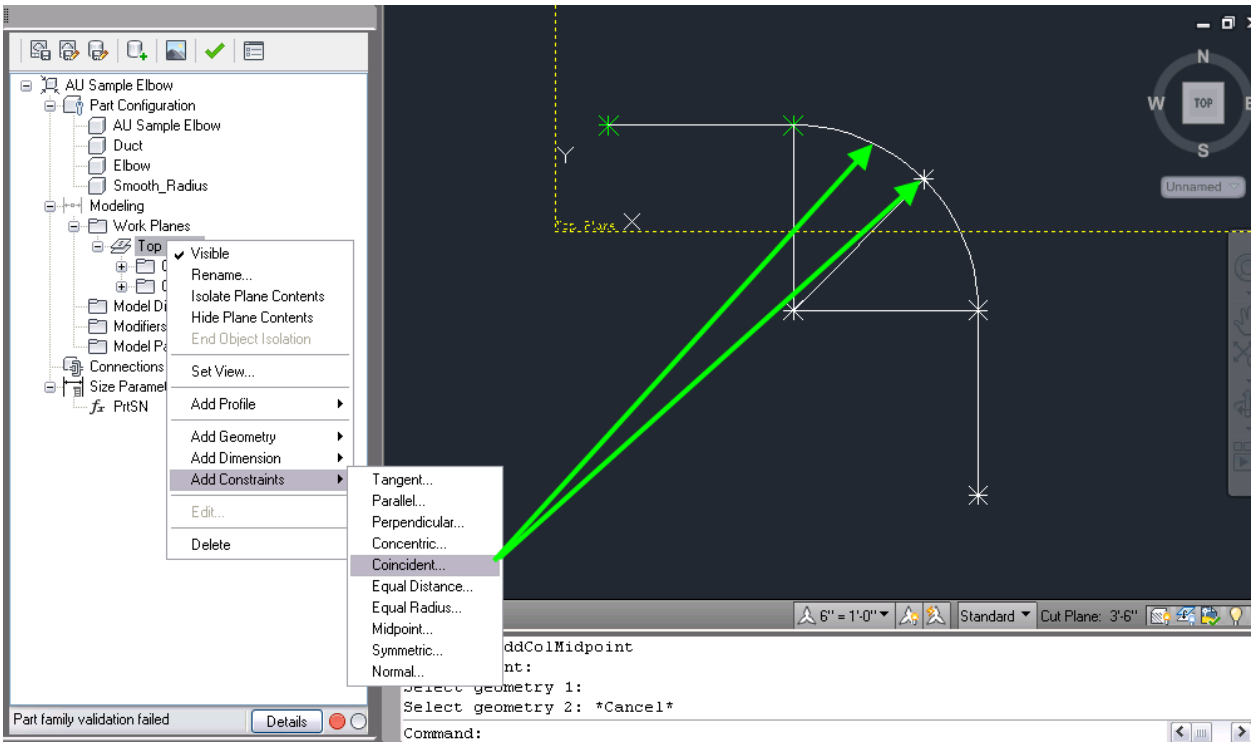

```
Pick start point:
Pick midpoint:
Pick end point:
```

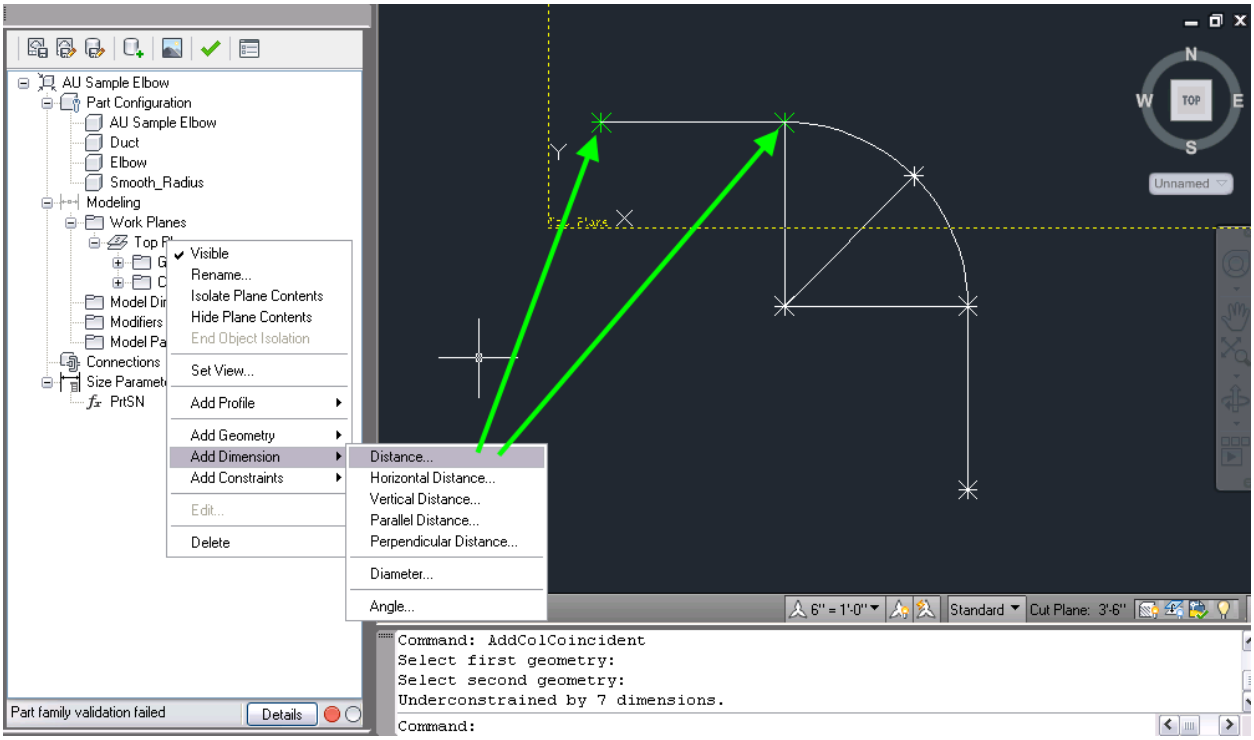

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Geometry
Point 2D
Point 2D
Line 2D
Point 2D
Line 2D
Point 2D
Line 2D
Point 2D
Line 2D
Point 2D
Line 2D
Arc 2D
Constraints
Visible
Rename...
Fixed
Delete
Top Plane

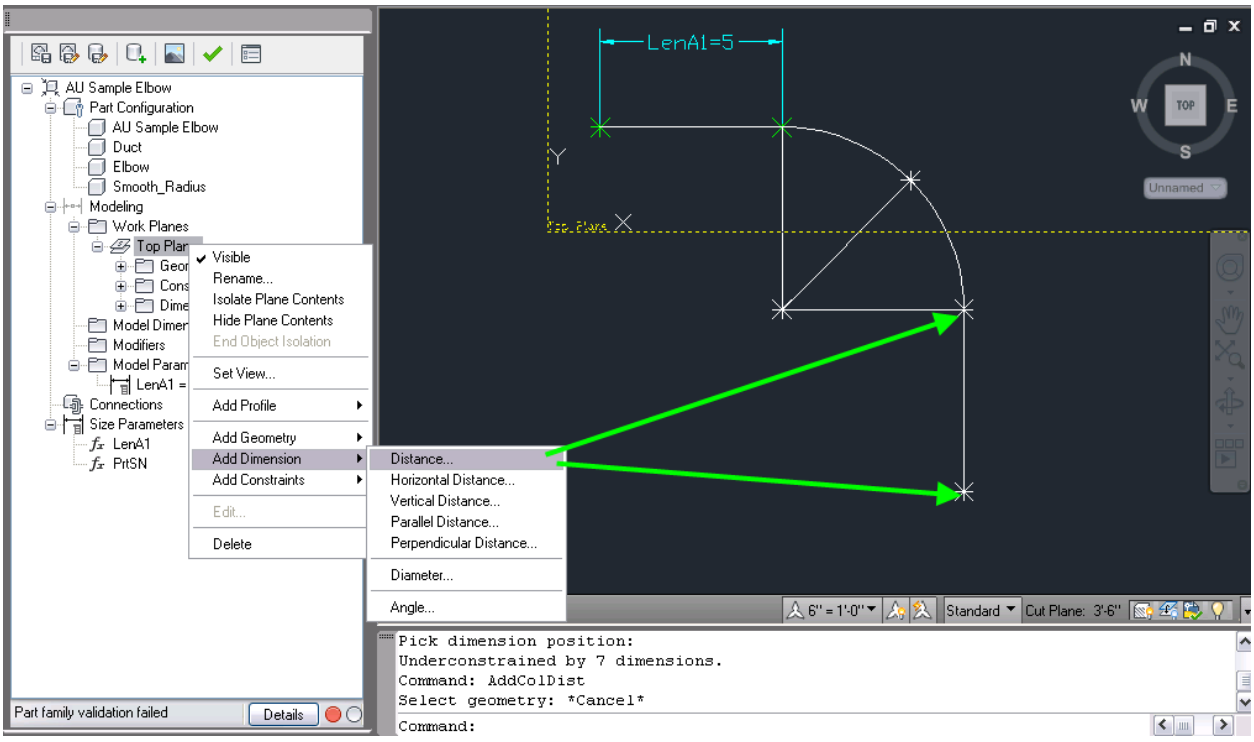

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Geometry
Point 2D
Point 2D
Line 2D
Point 2D
Line 2D
Point 2D
Line 2D
Point 2D
Line 2D
Point 2D
Line 2D
Arc 2D
Constraints
Visible
Rename...
Fixed
Delete
Top Plane

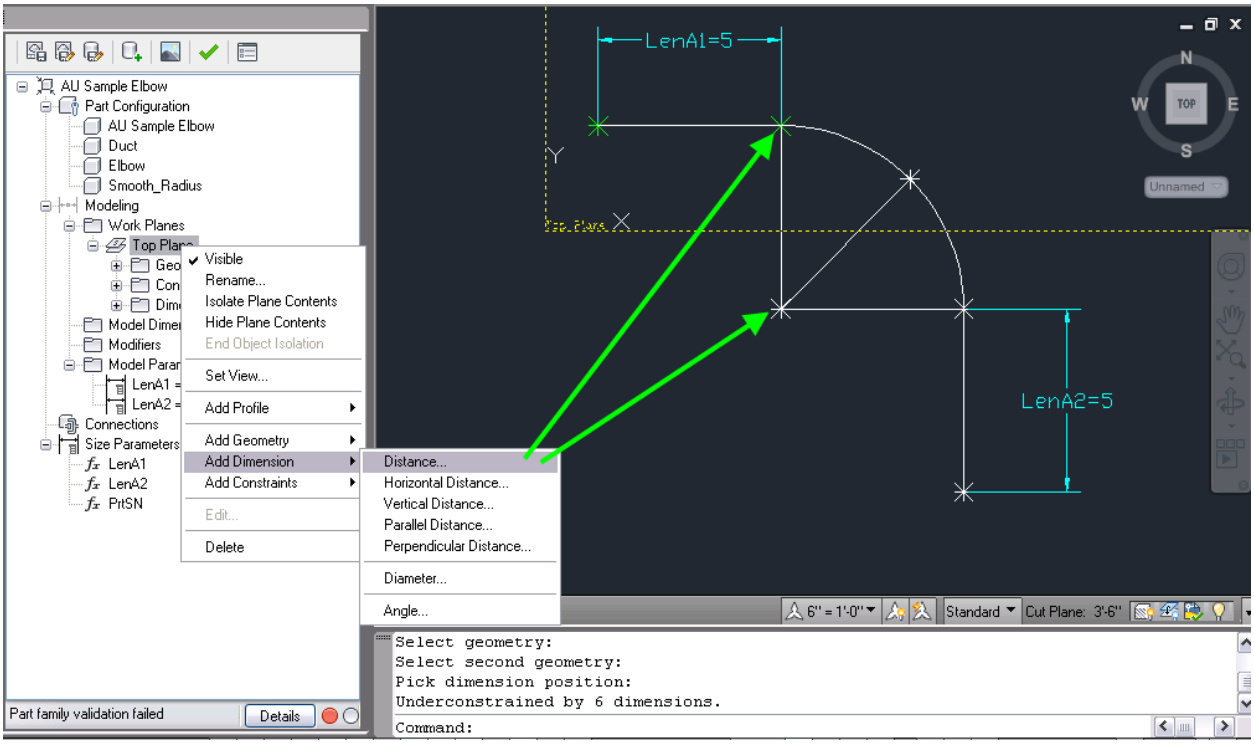

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Model Dim
Modifiers
Connections
PrtSN
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Tangent...
Parallel...
Perpendicular...
Concentric...
Coincident...
Equal Distance...
Equal Radius...
Midpoint...
Symmetric...
Normal...
ained by 10 dimensions.
Cancel*

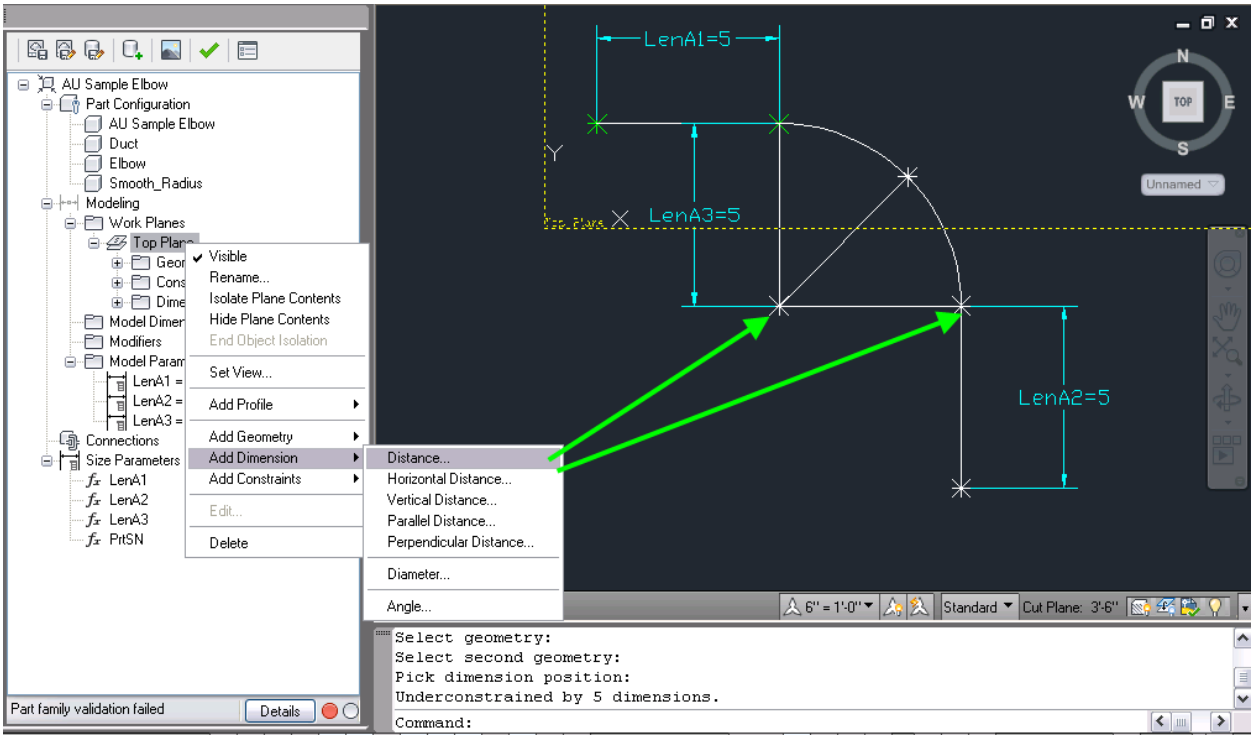

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Tangent...
Parallel...
Perpendicular...
Concentric...
Coincident...
Equal Distance...
Equal Radius...
Midpoint...
Symmetric...
Normal...
Connections
PrtSN
Part family validation failed
Details
Top Plane
N
W
TOP
E
S
Unnamed
6" = 1'-0"
Standard
Cut Plane: 3'-6"
dColPerpendicular
Underconstrained by 8 dimensions.
Command:

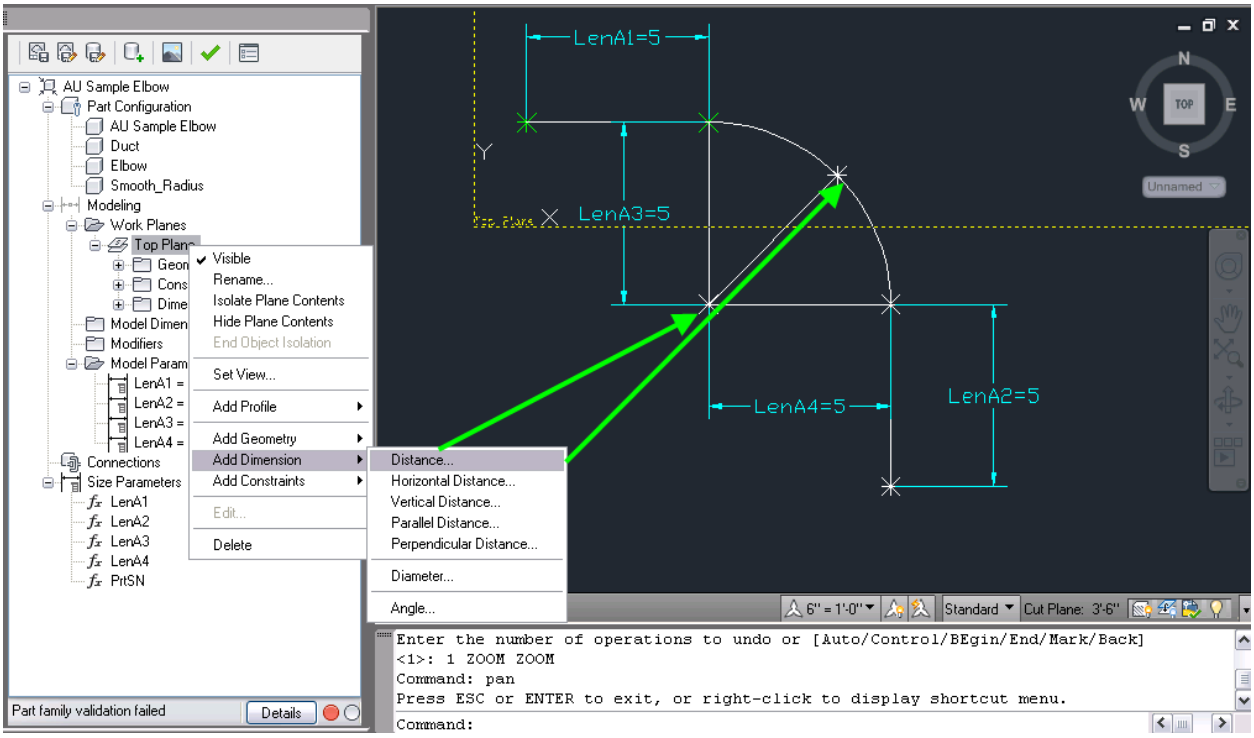

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Tangent...
Parallel...
Perpendicular...
Concentric...
Coincident...
Equal Distance...
Equal Radius...
Midpoint...
Symmetric...
Normal...
Connections
PrtSN
Part family validation failed
Details
Top Plane
N
W
TOP
S
Unnamed
6" = 1'-0"
Standard
Cut Plane: 3'-6"
dColMidpoint
Select geometry 2: *Cancel*
Command:

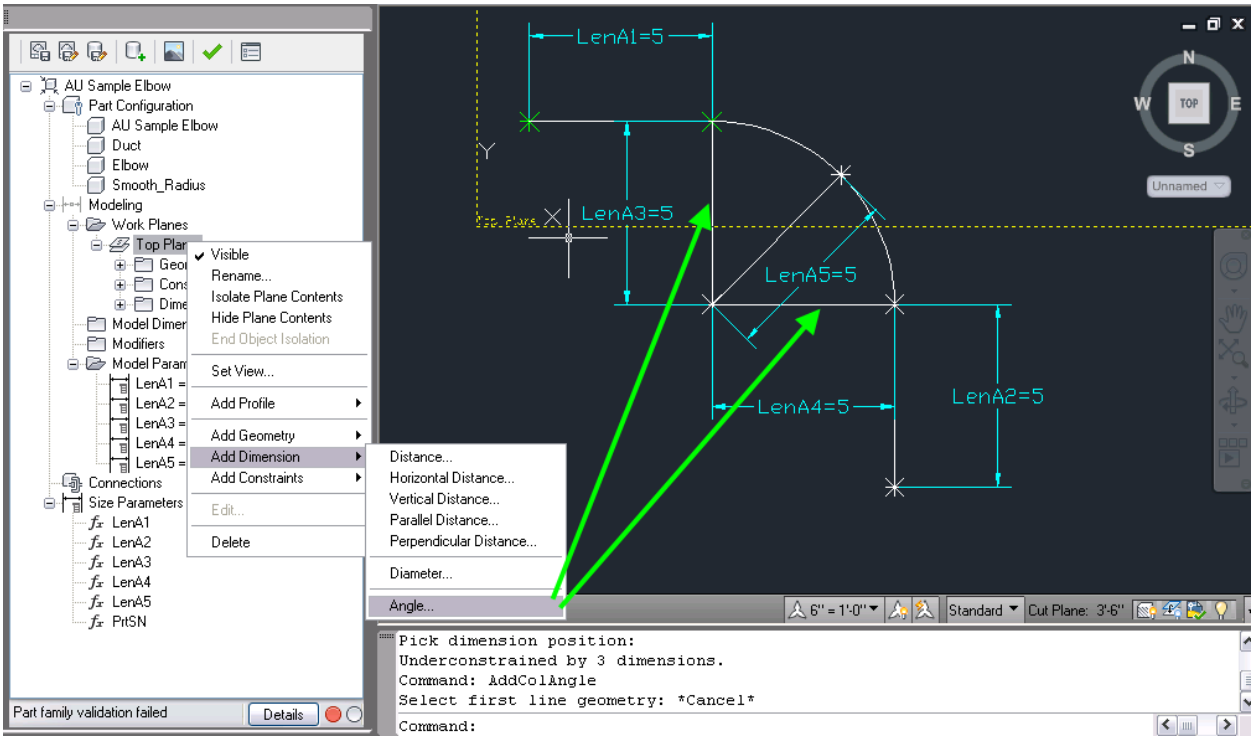

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Distance...
Horizontal Distance...
Vertical Distance...
Parallel Distance...
Perpendicular Distance...
Diameter...
Angle...
Connections
PrtSN
Part family validation failed
Details
Command: AddColCoincident
Select first geometry:
Select second geometry:
Underconstrained by 7 dimensions.
Command:

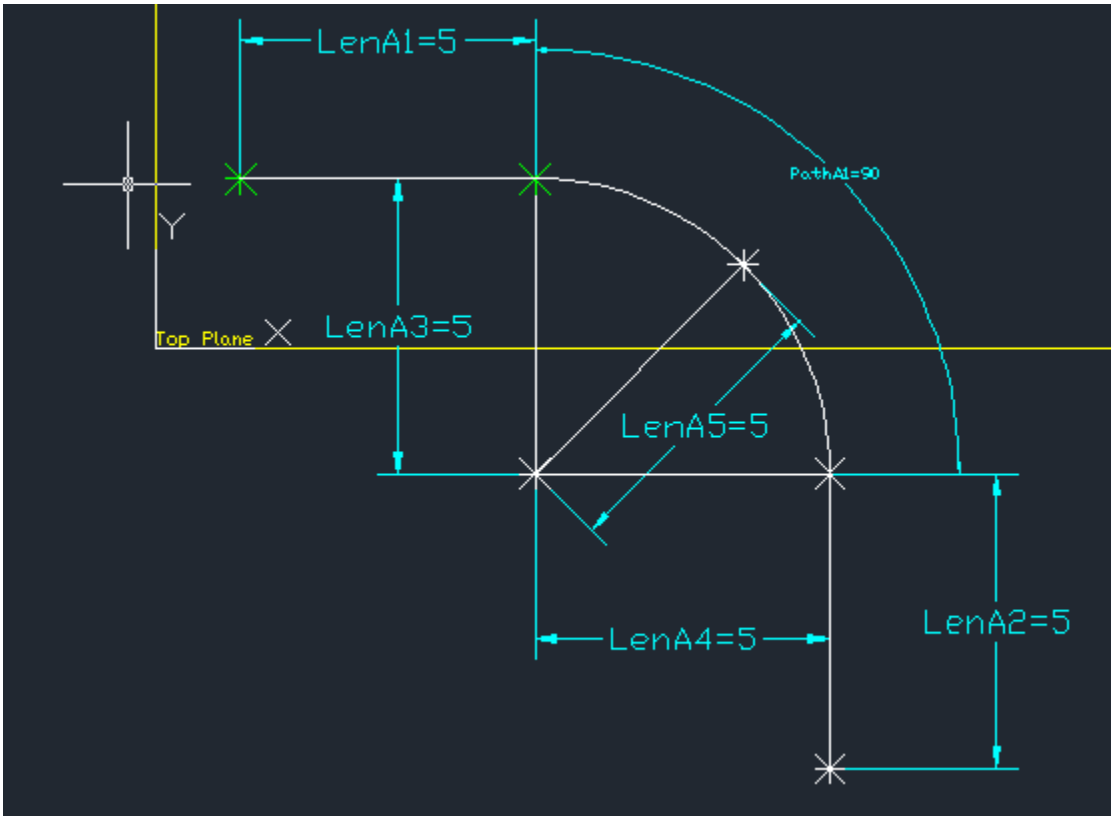

LenA1=5
AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Distance...
Horizontal Distance...
Vertical Distance...
Parallel Distance...
Perpendicular Distance...
Diameter...
Angle...
Connections
Size Parameters
LenA1
PrtSN
Part family validation failed
Details
Pick dimension position:
Underconstrained by 7 dimensions.
Command: AddColDist
Select geometry: *Cancel*
Command:

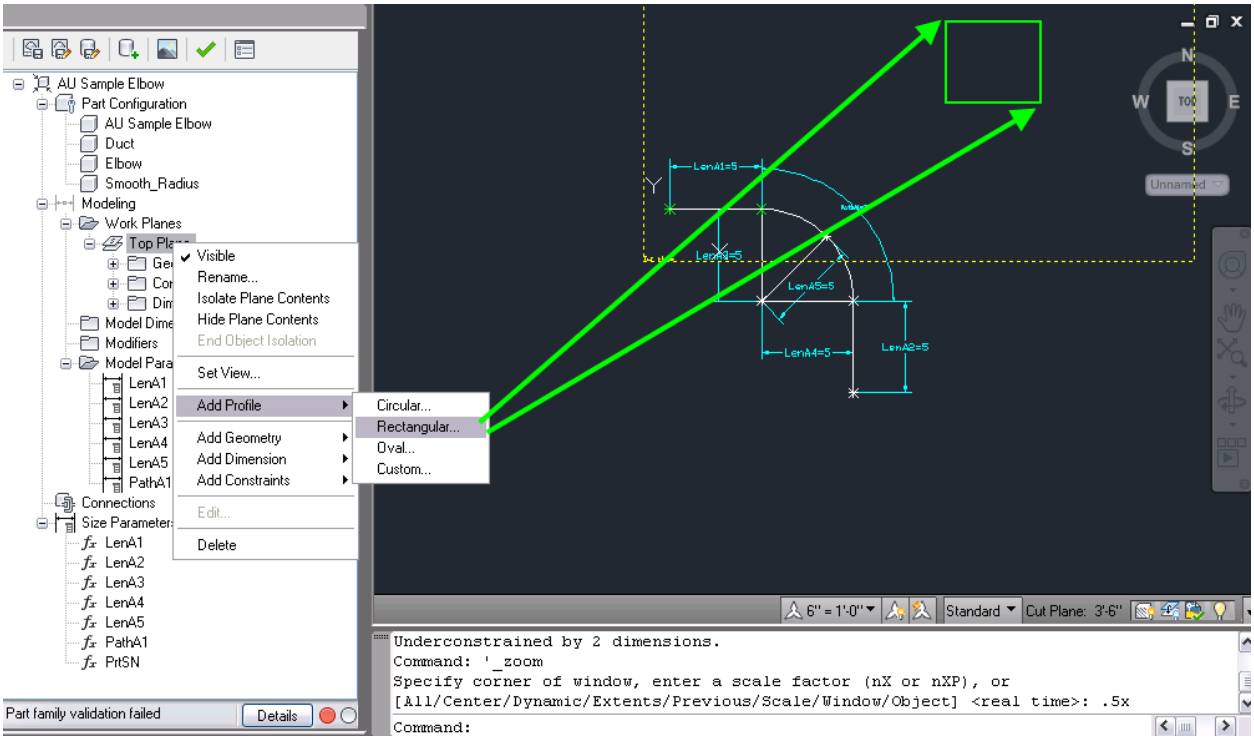

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Distance...
Horizontal Distance...
Vertical Distance...
Parallel Distance...
Perpendicular Distance...
Diameter...
Angle...
Model Dimensions
Modifiers
Model Parameters
LenA1 =
LenA2 =
Connections
Size Parameters
LenA1
LenA2
PrtSN
LenA1=5
LenA2=5
Top Plane
N
W
TOP
E
S
Unnamed
6" = 1'-0"
Standard
Cut Plane: 3'-6"
Select geometry:
Select second geometry:
Pick dimension position:
Underconstrained by 6 dimensions.
Command:
Part family validation failed
Details

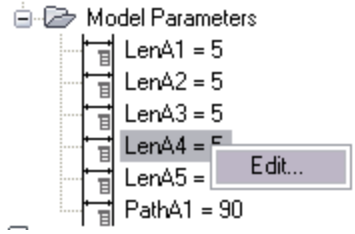

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Distance...
Horizontal Distance...
Vertical Distance...
Parallel Distance...
Perpendicular Distance...
Diameter...
Angle...
Model Dimensions
Modifiers
Model Parameters
LenA1 =
LenA2 =
LenA3 =
Connections
Size Parameters
LenA1
LenA2
LenA3
PrtSN
LenA1=5
LenA3=5
LenA2=5
Top Plane
N
W
TOP
E
S
Unnamed
6" = 1'-0"
Standard
Cut Plane: 3'-6"
Select geometry:
Select second geometry:
Pick dimension position:
Underconstrained by 5 dimensions.
Command:
Part family validation failed
Details

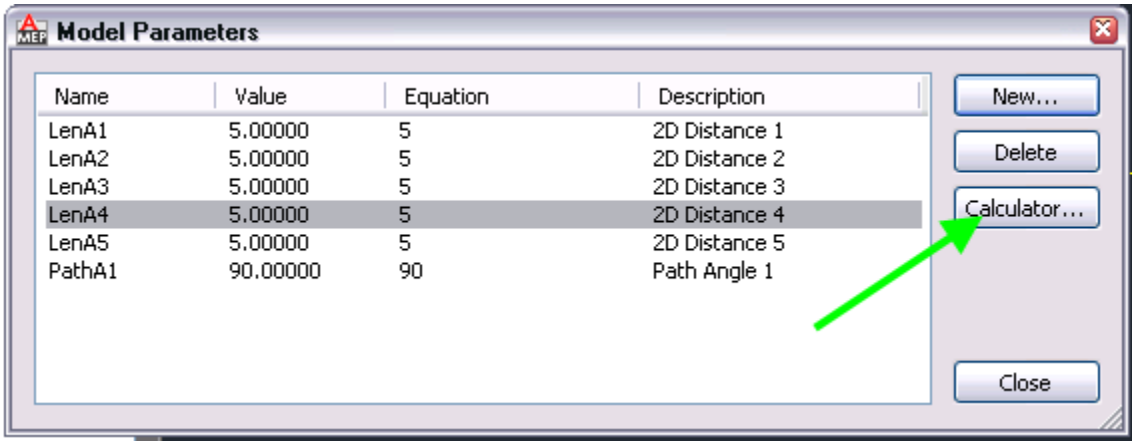

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Model Dimen
Modifiers
Model Param
LenA1 =
LenA2 =
LenA3 =
LenA4 =
Connections
Size Parameters
LenA1
LenA2
LenA3
LenA4
PrtSN
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Distance...
Horizontal Distance...
Vertical Distance...
Parallel Distance...
Perpendicular Distance...
Diameter...
Angle...
LenA1=5
LenA3=5
LenA4=5
LenA2=5
Part family validation failed
Details
Enter the number of operations to undo or [Auto/Control/BEgin/End/Mark/Back]
<1>: 1 ZOOM ZOOM
Command: pan
Press ESC or ENTER to exit, or right-click to display shortcut menu.
Command:

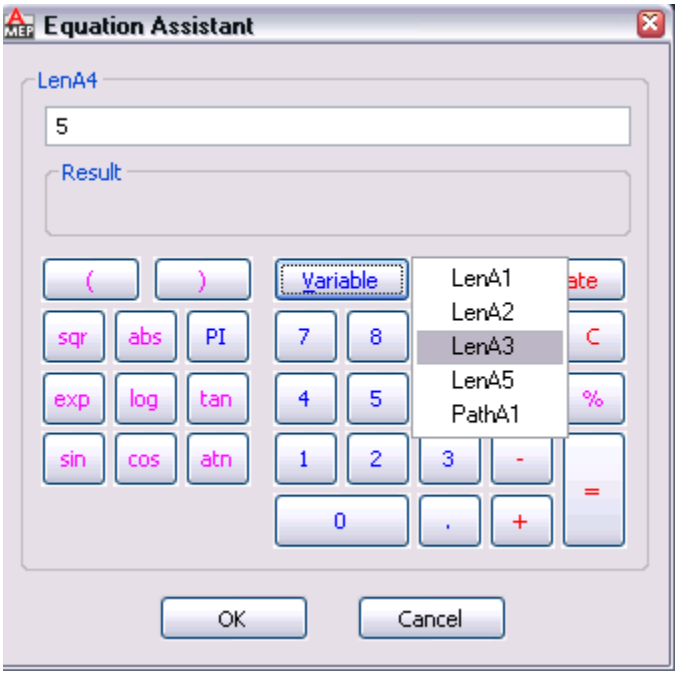

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Top Plane
Model Dimer
Modifiers
Model Param
LenA1 =
LenA2 =
LenA3 =
LenA4 =
LenA5 =
Connections
Size Parameters
LenA1
LenA2
LenA3
LenA4
LenA5
PrtSN
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Distance...
Horizontal Distance...
Vertical Distance...
Parallel Distance...
Perpendicular Distance...
Diameter...
Angle...
LenA1=5
LenA3=5
LenA5=5
LenA4=5
LenA2=5
Part family validation failed
Details
Pick dimension position:
Underconstrained by 3 dimensions.
Command: AddColAngle
Select first line geometry: *Cancel*
Command:

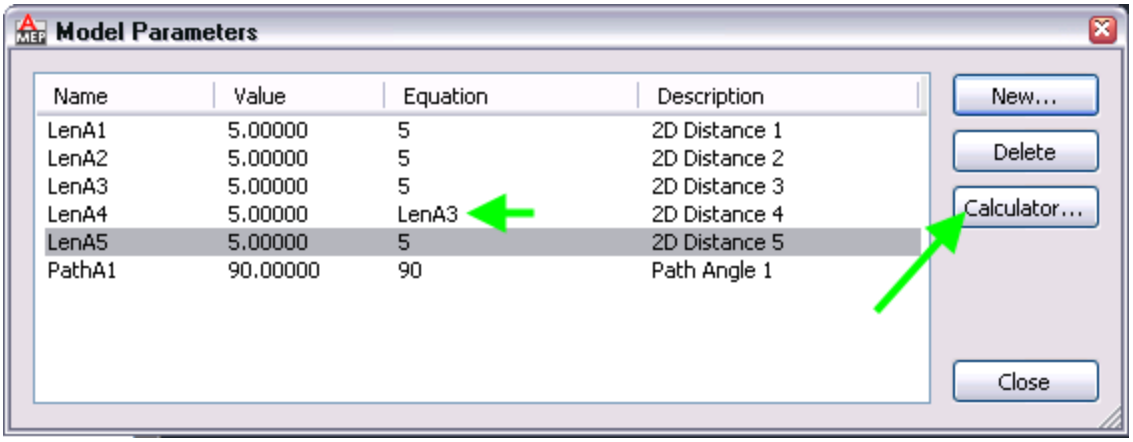

LenA1=5
PathA1=90
Top Plane
LenA3=5
LenA5=5
LenA4=5
LenA2=5

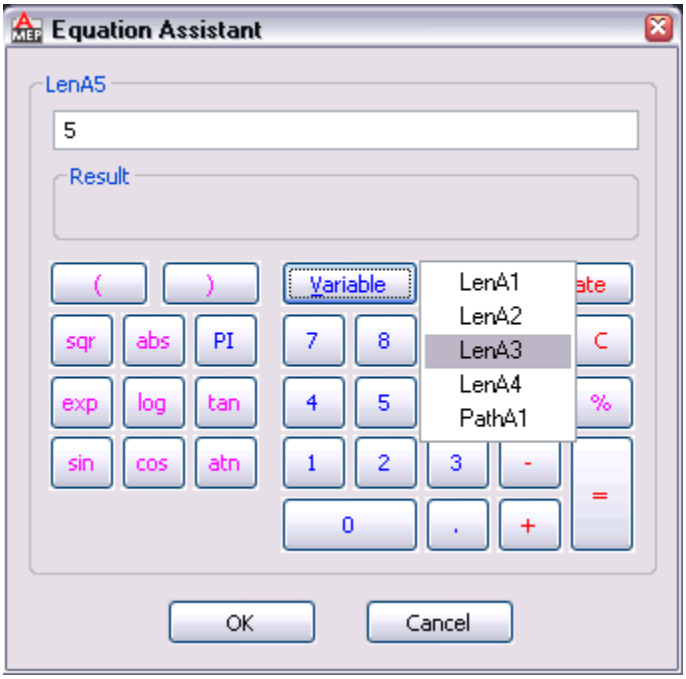

AU Sample Elbow
Part Configuration
AU Sample Elbow
Duct
Elbow
Smooth_Radius
Modeling
Work Planes
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Circular...
Rectangular...
Oval...
Custom...
Model Dime
Modifiers
Model Para
LenA1
LenA2
LenA3
LenA4
LenA5
PathA1
Connections
Size Parameter
LenA1
LenA2
LenA3
LenA4
LenA5
PathA1
PrtSN
Part family validation failed
Details
Standard
Cut Plane: 3'-6"
Underconstrained by 2 dimensions.
Command: '_zoom
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: .5x
Command:

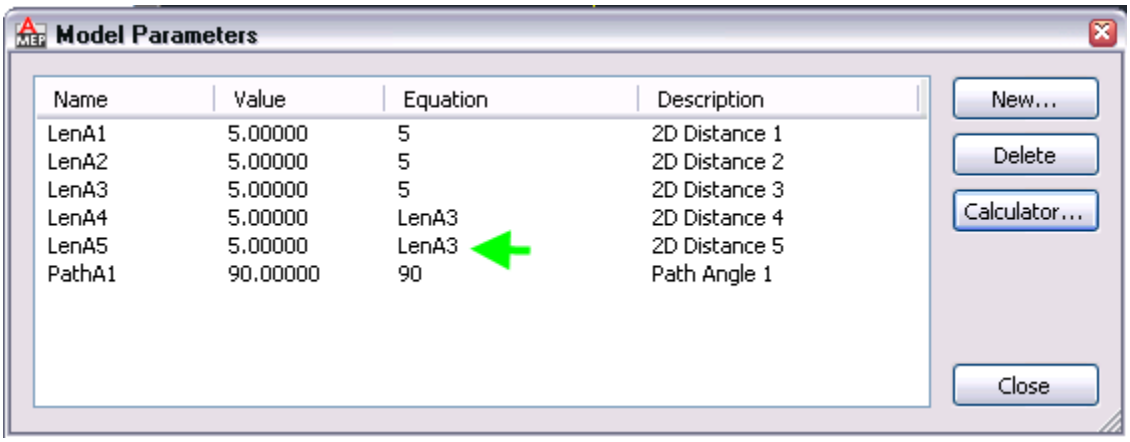

Model Parameters
LenA1 = 5
LenA2 = 5
LenA3 = 5
LenA4 =
LenA5 =
Edit...
PathA1 = 90

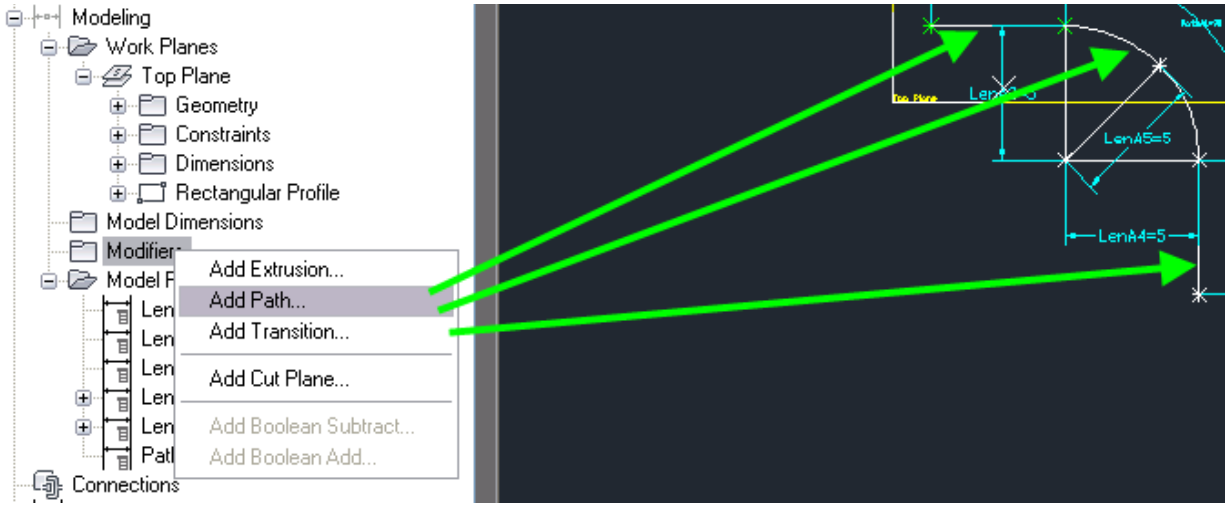

Model Parameters
Name
Value
Equation
Description
LenA1 5.00000 5 2D Distance 1
LenA2 5.00000 5 2D Distance 2
LenA3 5.00000 5 2D Distance 3
LenA4 5.00000 5 2D Distance 4
LenA5 5.00000 5 2D Distance 5
PathA1 90.00000 90 Path Angle 1
New...
Delete
Calculator...
Close

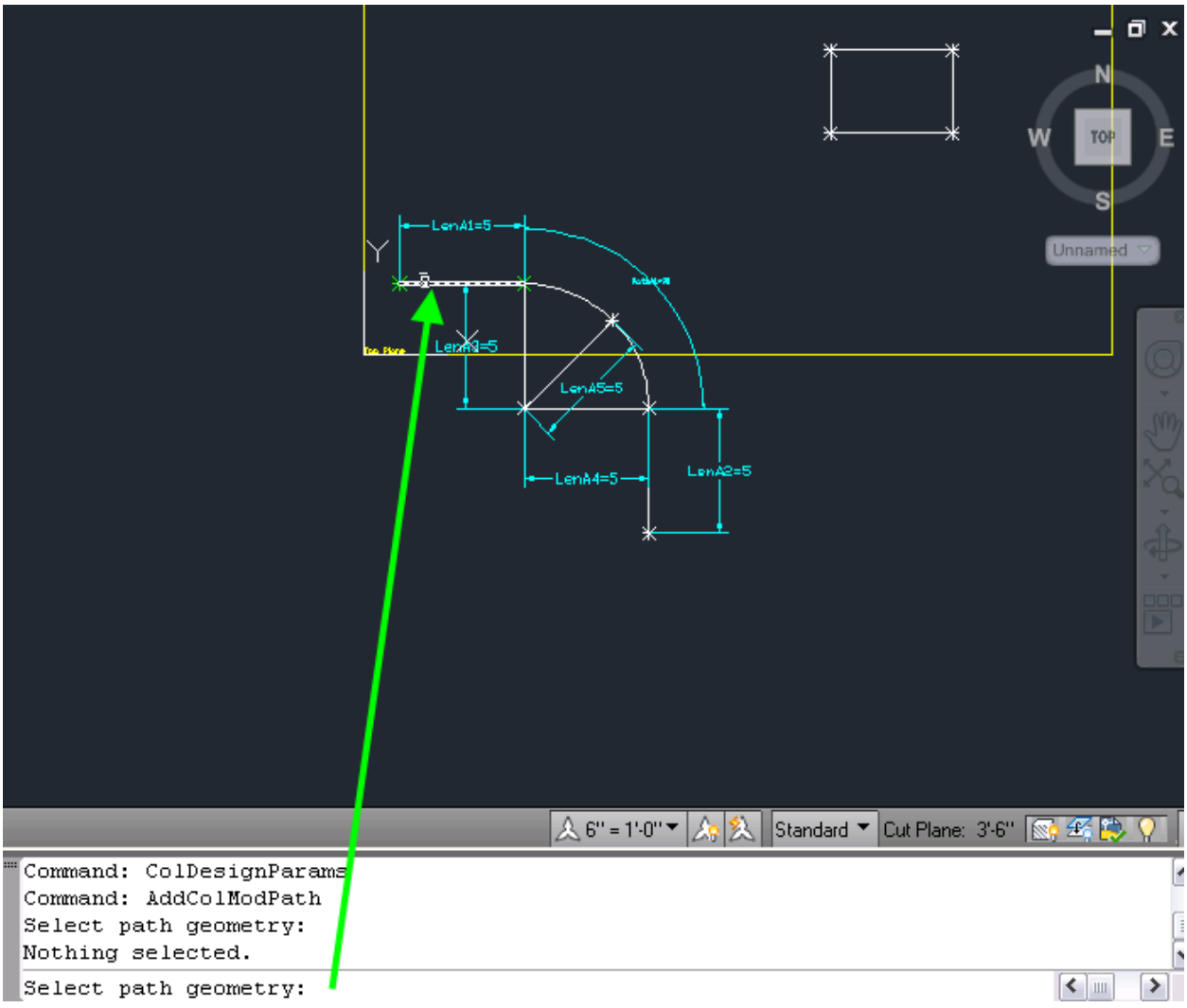

Equation Assistant
LenA4
5
Result
(
)
Variable
LenA1
LenA2
LenA3
LenA5
PathA1
sqr
abs
PI
exp
log
tan
sin
cos
atn
7
8
4
5
1
2
3
-
0
.
+
=
C
%
OK
Cancel

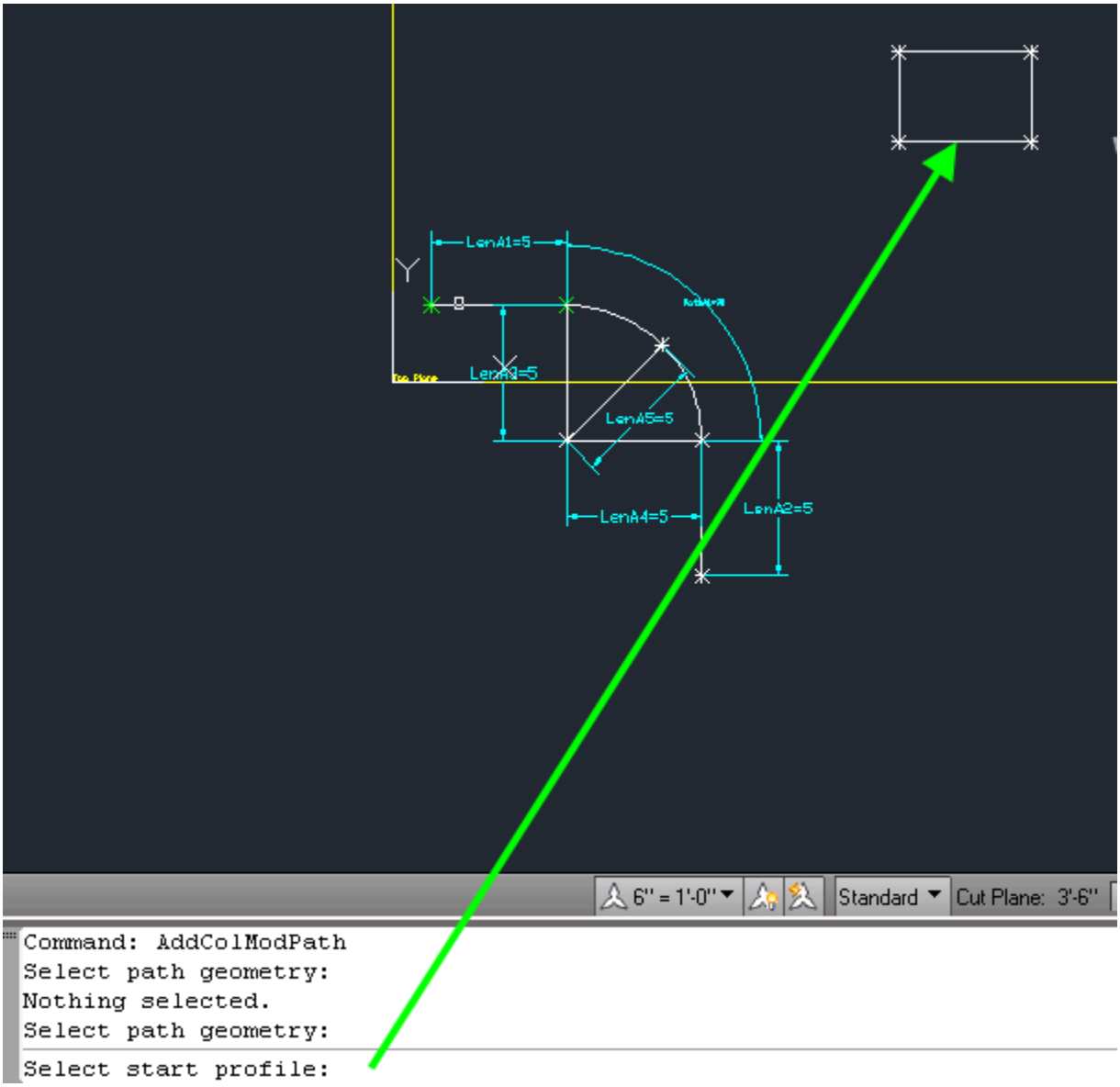

Model Parameters
Name
Value
Equation
Description
LenA1 5.00000 5 2D Distance 1
LenA2 5.00000 5 2D Distance 2
LenA3 5.00000 5 2D Distance 3
LenA4 5.00000 LenA3 2D Distance 4
LenA5 5.00000 5 2D Distance 5
PathA1 90.00000 90 Path Angle 1
New...
Delete
Calculator...
Close

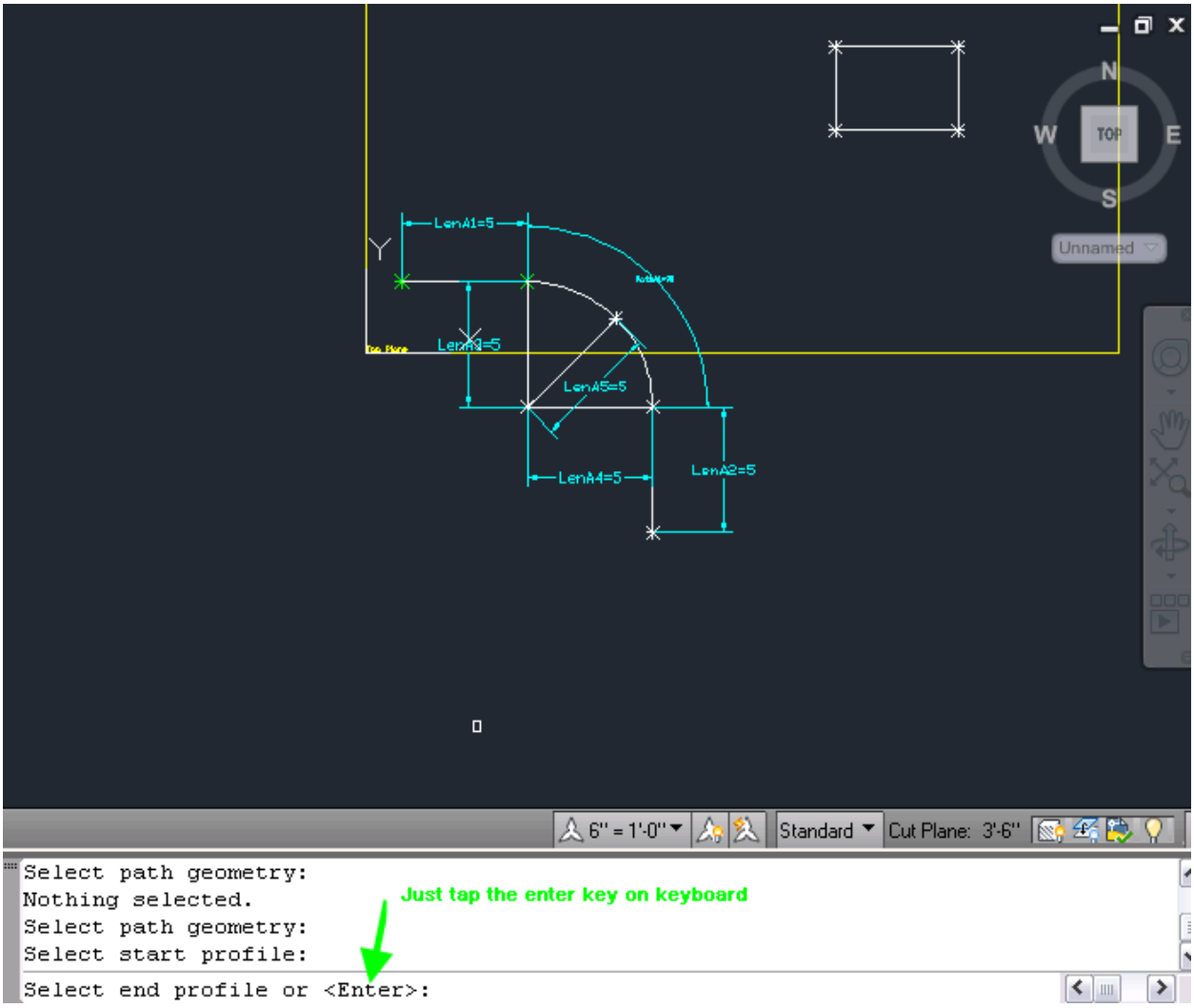

Equation Assistant
LenA5
5
Result
(
)
Variable
LenA1
LenA2
LenA3
LenA4
PathA1
sqr
abs
PI
7
8
C
exp
log
tan
4
5
%
sin
cos
atn
1
2
3
-
=
0
.
+
OK
Cancel

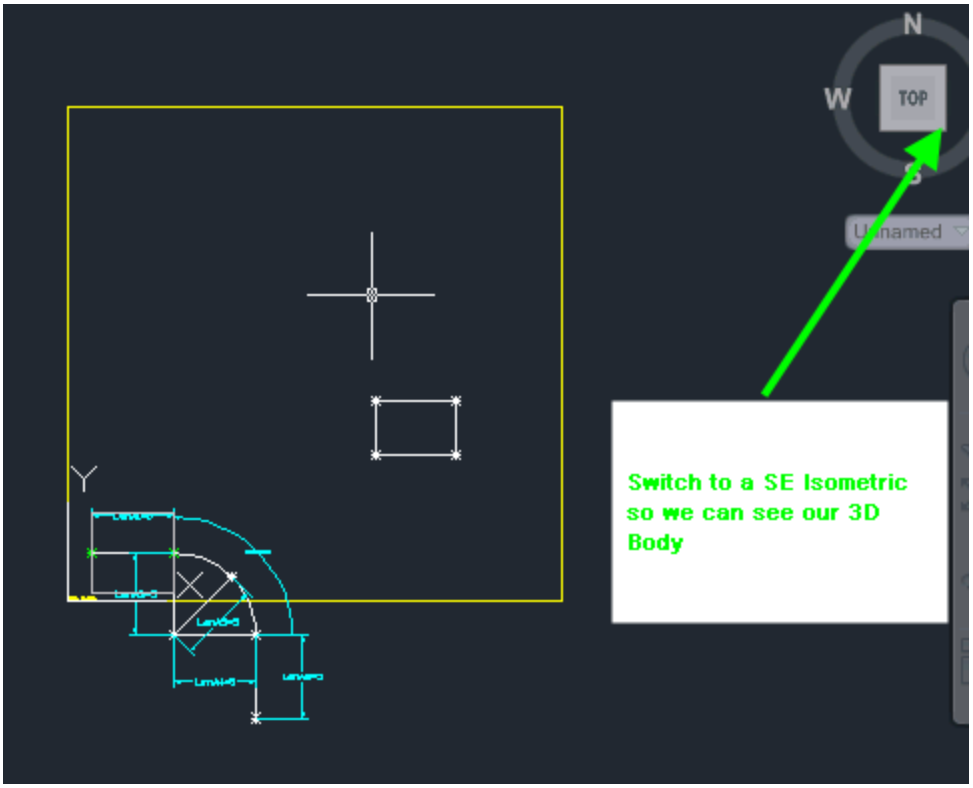

Model Parameters
Name
Value
Equation
Description
LenA1 5.00000 5 2D Distance 1
LenA2 5.00000 5 2D Distance 2
LenA3 5.00000 5 2D Distance 3
LenA4 5.00000 LenA3 2D Distance 4
LenA5 5.00000 LenA3 2D Distance 5
PathA1 90.00000 90 Path Angle 1
New...
Delete
Calculator...
Close

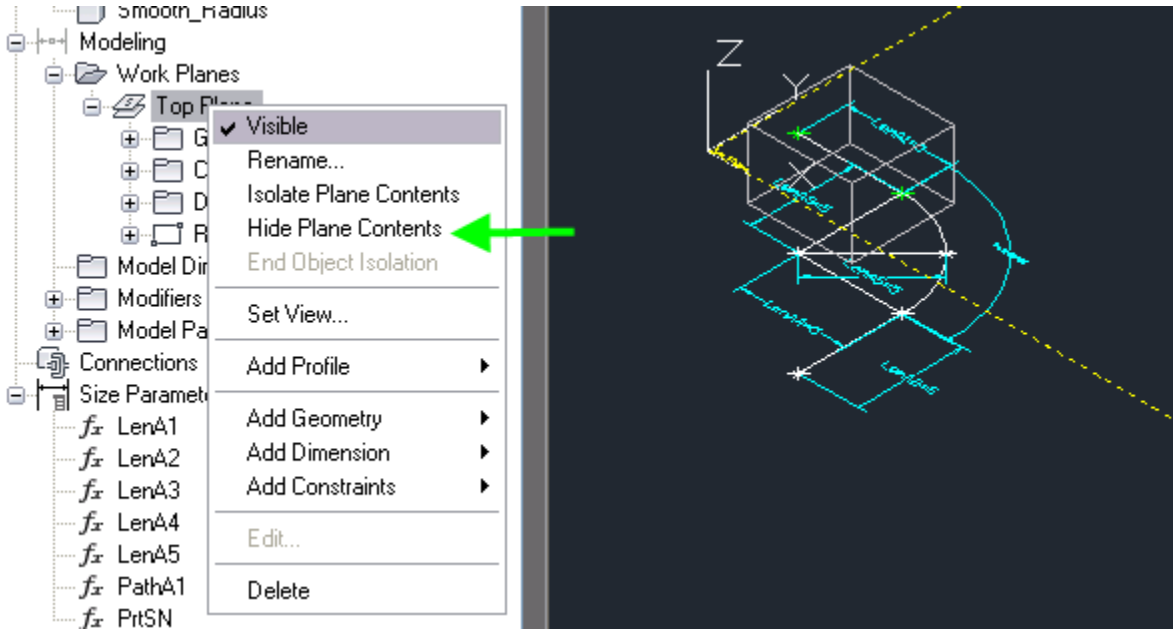

Modeling
Work Planes
Top Plane
Geometry
Constraints
Dimensions
Rectangular Profile
Model Dimensions
Modifier
Model P
Connections
Add Extrusion...
Add Path...
Add Transition...
Add Cut Plane...
Add Boolean Subtract...
Add Boolean Add...
LenA5=5
LenA4=5

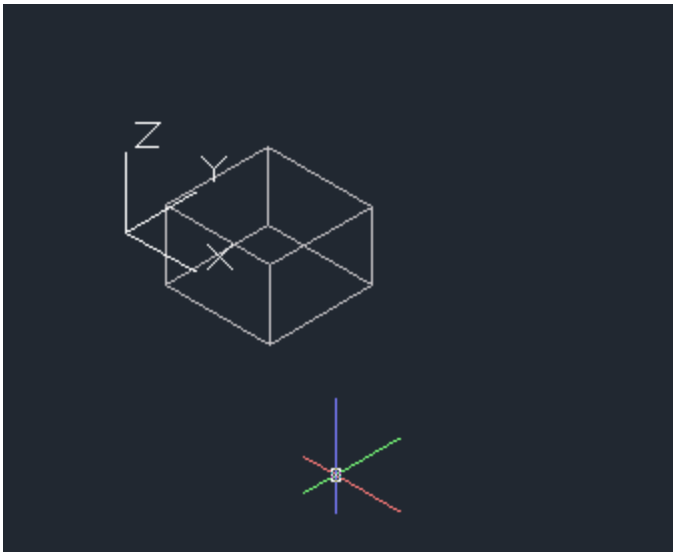

LenA1=5
LenA4=5
LenA5=5
LenA4=5
LenA2=5
N
W
TOP
E
S
Unnamed
6" = 1'-0"
Standard
Cut Plane: 3'-6"
Command: ColDesignParams
Command: AddColModPath
Select path geometry:
Nothing selected.
Select path geometry:

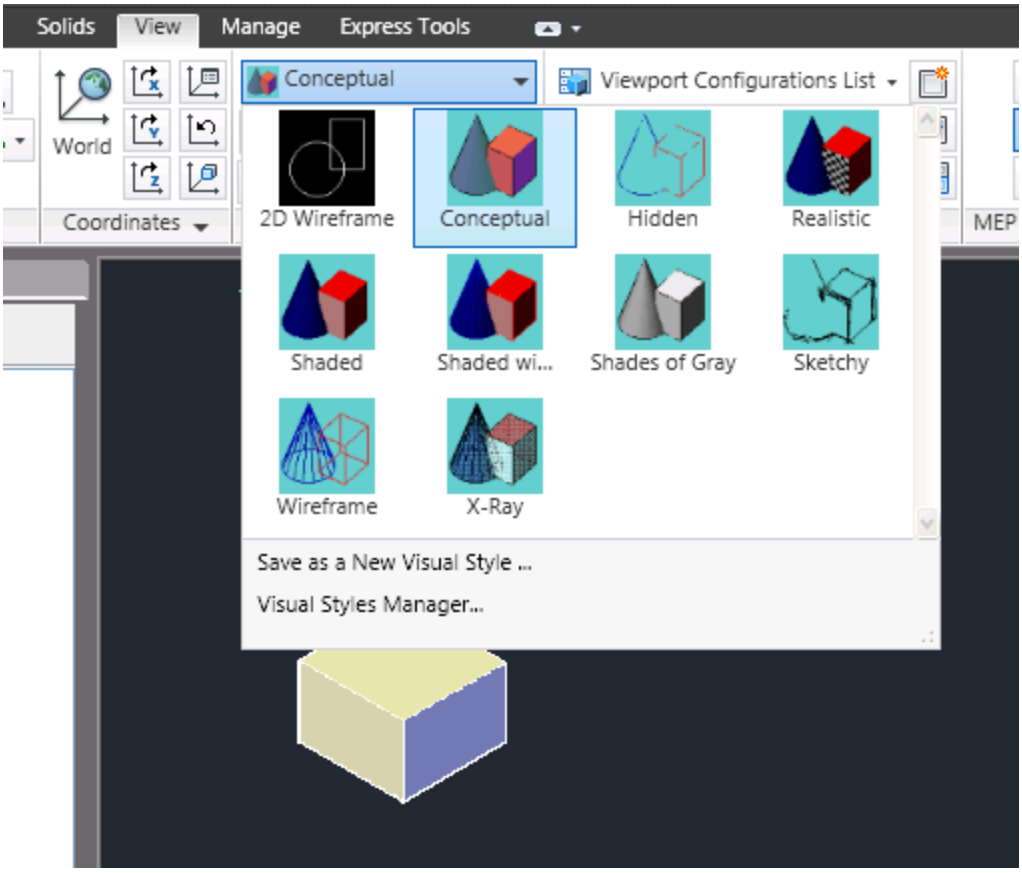

LenA1=5
LenA3=5
LenA5=5
LenA4=5
LenA2=5
6" = 1'-0"
Standard
Cut Plane: 3'-6"
Command: AddColModPath
Select path geometry:
Nothing selected.
Select path geometry:
Select start profile:

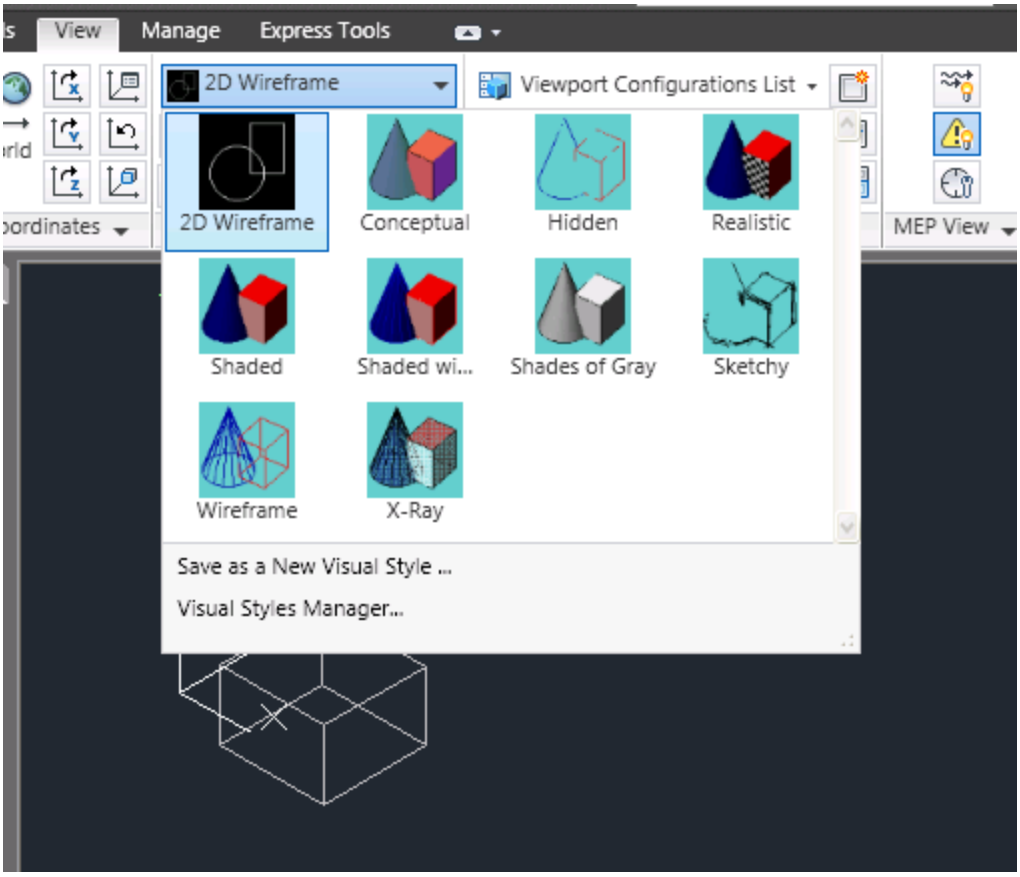

N
W
TOP
E
S
Unnamed
LenA1=5
LenA3=5
LenA5=5
LenA4=5
LenA2=5
6" = 1'-0"
Standard
Cut Plane: 3'-6"
Select path geometry:
Nothing selected.
Select path geometry:
Select start profile:
Select end profile or <Enter>:
Just tap the enter key on keyboard


N
W
TOP
S
Unnamed
Y
Switch to a SE Isometric so we can see our 3D Body

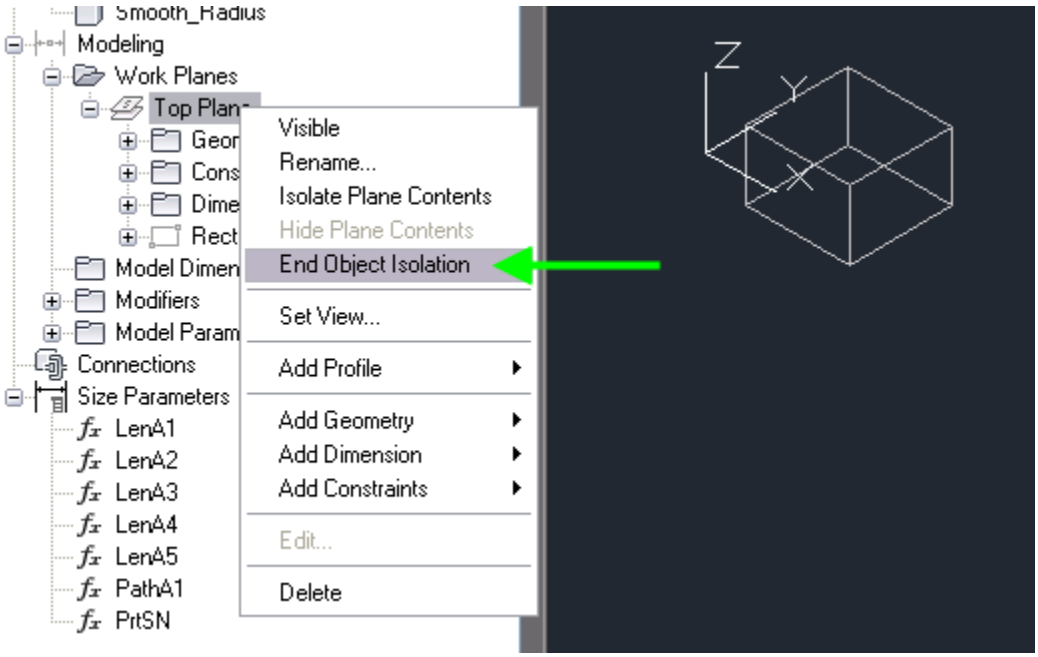

Smooth_Radius
Modeling
Work Planes
Top Plane
Visible
Rename...
Isolate Plane Contents
Hide Plane Contents
End Object Isolation
Set View...
Add Profile
Add Geometry
Add Dimension
Add Constraints
Edit...
Delete
Model Dir
Modifiers
Model Pa
Connections
Size Parameter
LenA1
LenA2
LenA3
LenA4
LenA5
PathA1
PrtSN
Z
Y

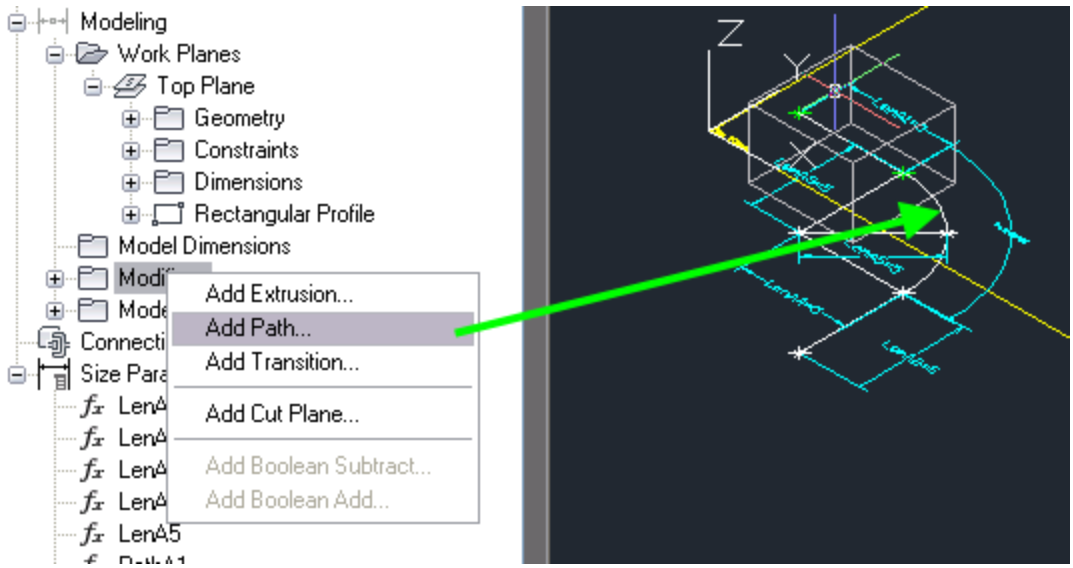

Z
Y
X

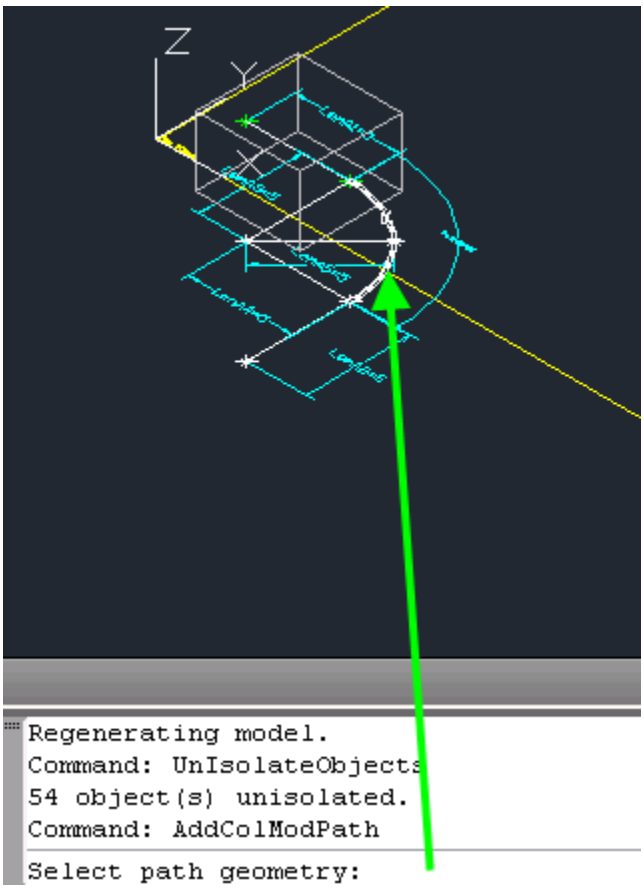

Solids
View
Manage
Express Tools
Conceptual
Viewport Configurations List
World
Coordinates
MEP
2D Wireframe
Conceptual
Hidden
Realistic
Shaded
Shaded wi...
Shades of Gray
Sketchy
Wireframe
X-Ray
Save as a New Visual Style ...
Visual Styles Manager...

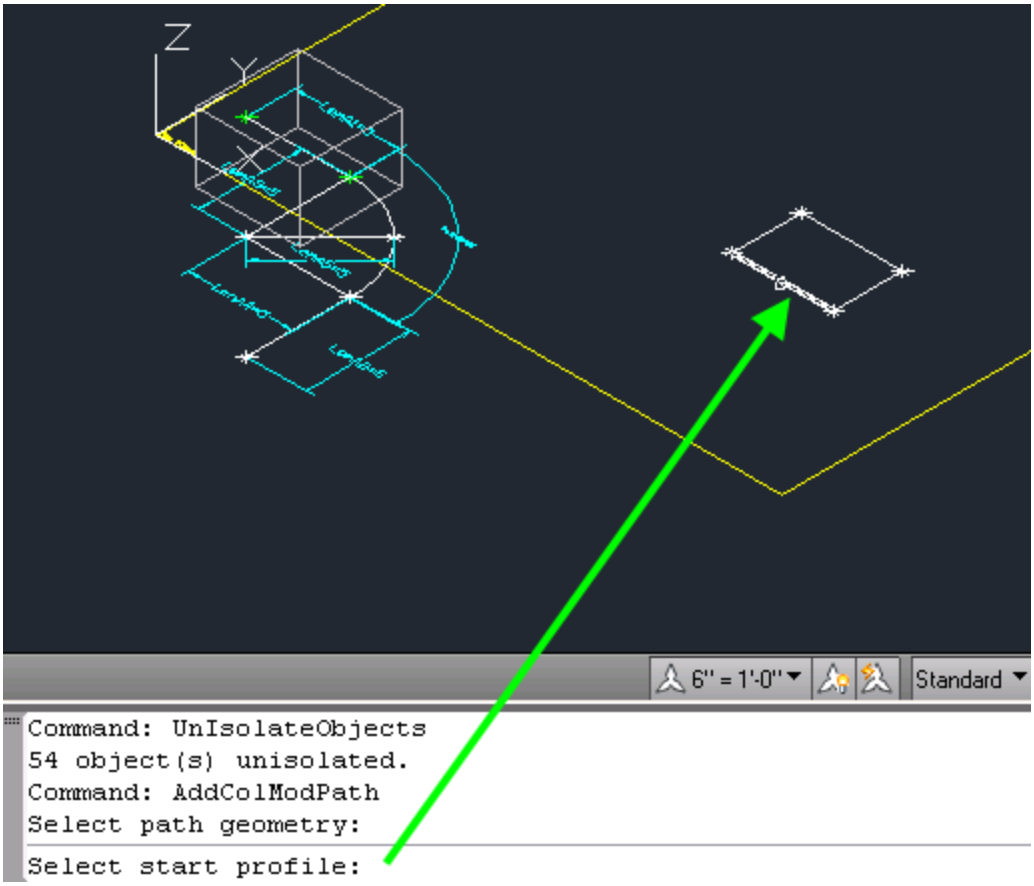


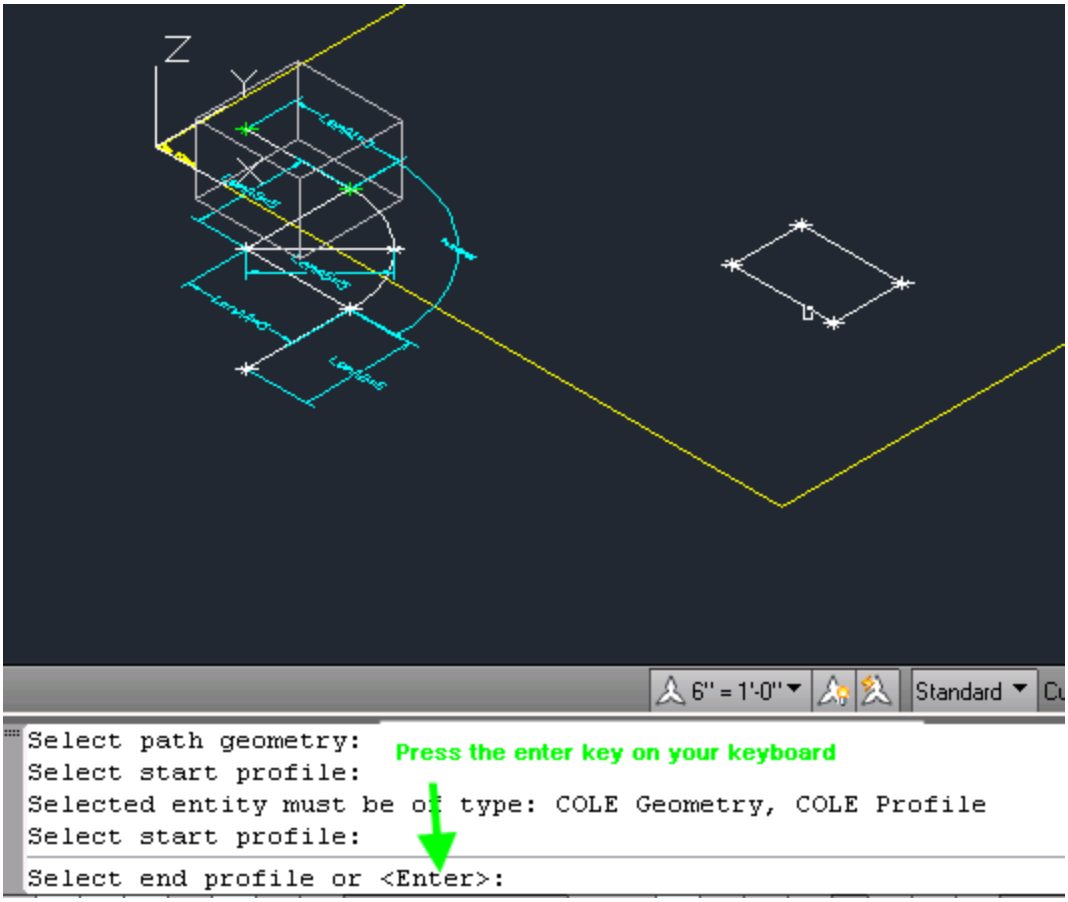
We need to go back to Wireframe, otherwise the dimensions and other objects will not appear.

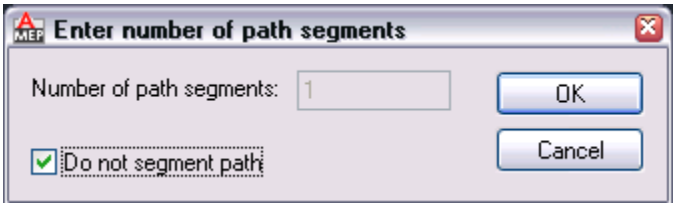



Modeling
Work Planes
Top Plane
Geometry
Constraints
Dimensions
Rectangular Profile
Model Dimensions
Add Extrusion...
Add Path...
Add Transition...
Add Cut Plane...
Add Boolean Subtract...
Add Boolean Add...
Z

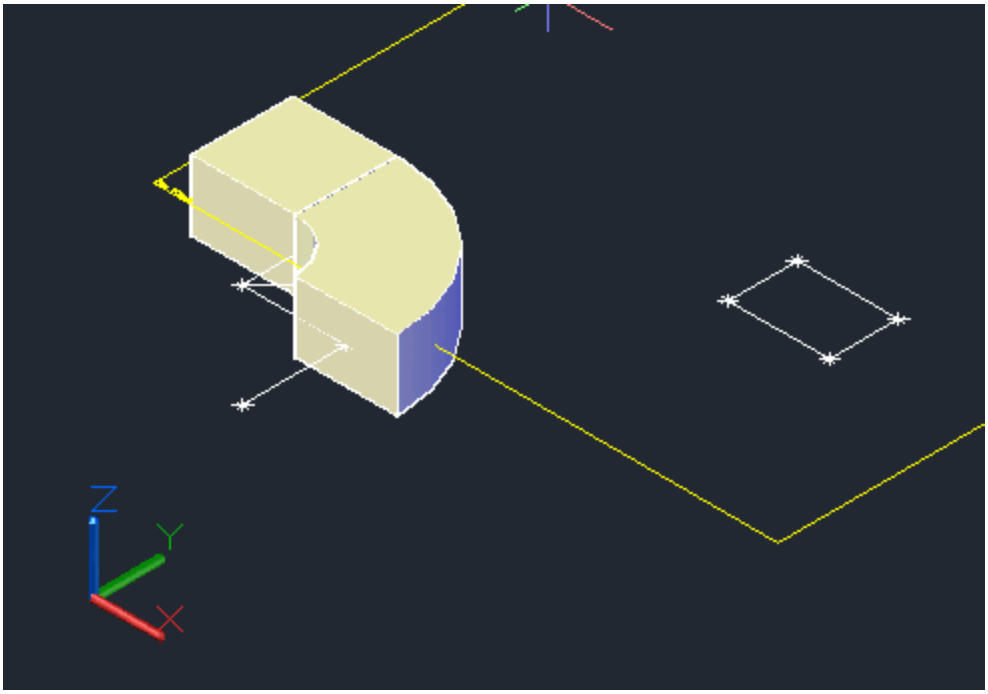

Z
Regenerating model.
Command: UnIsolateObjects
54 object(s) unisolated.
Command: AddColModPath
Select path geometry:

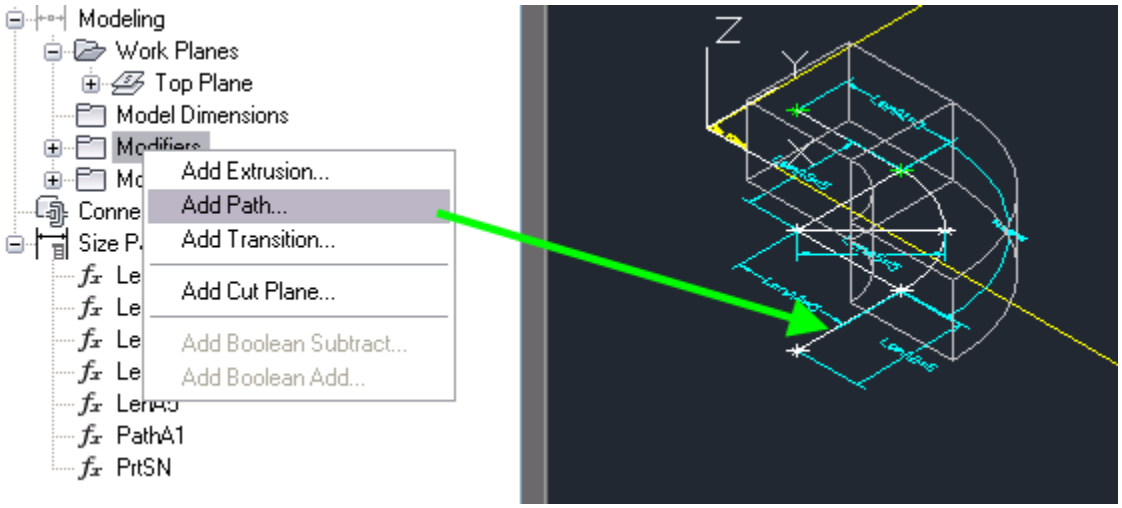

Z
6" = 1'-0"
Standard
Command: UnIsolateObjects
54 object(s) unisolated.
Command: AddColModPath
Select path geometry:
Select start profile:

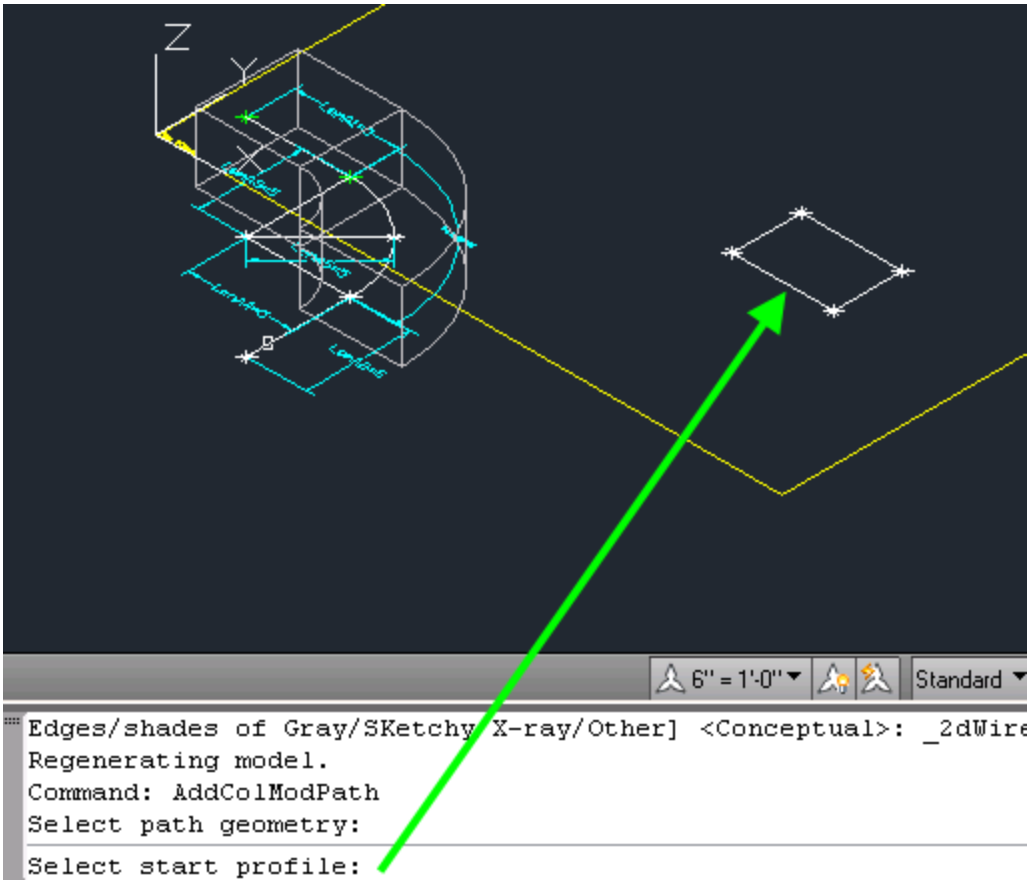


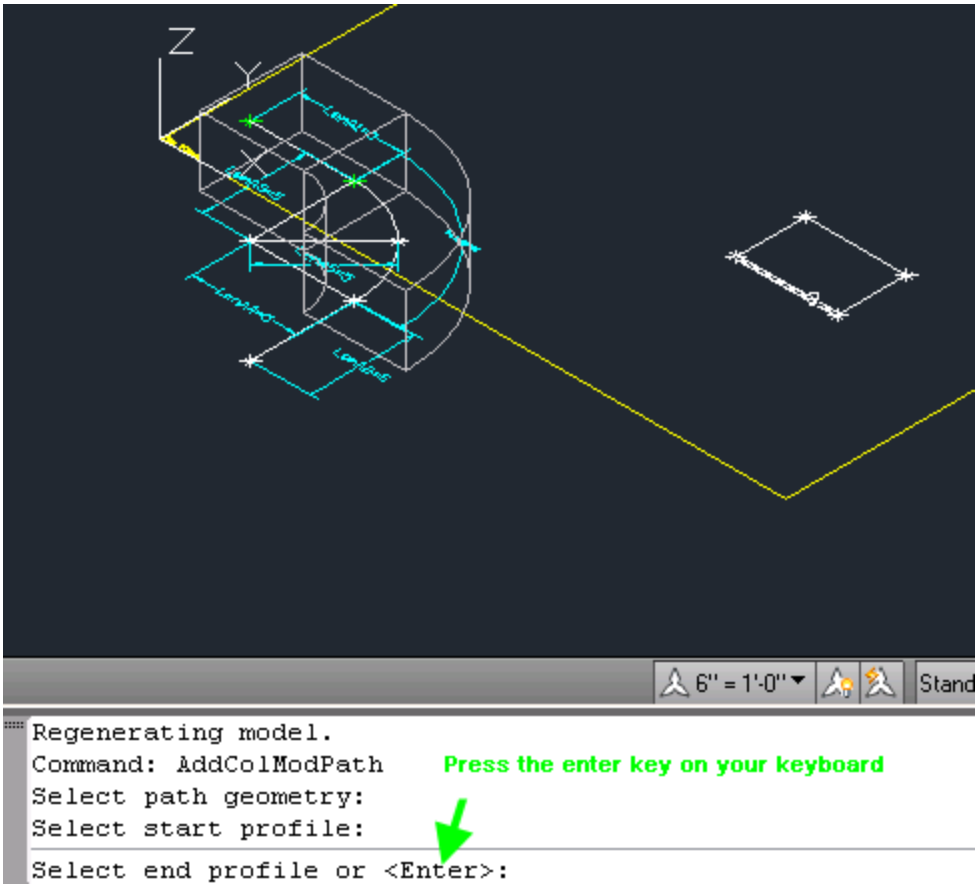


Tells the arc if the 3D Body is segmented or continuous.


Z
Y
X

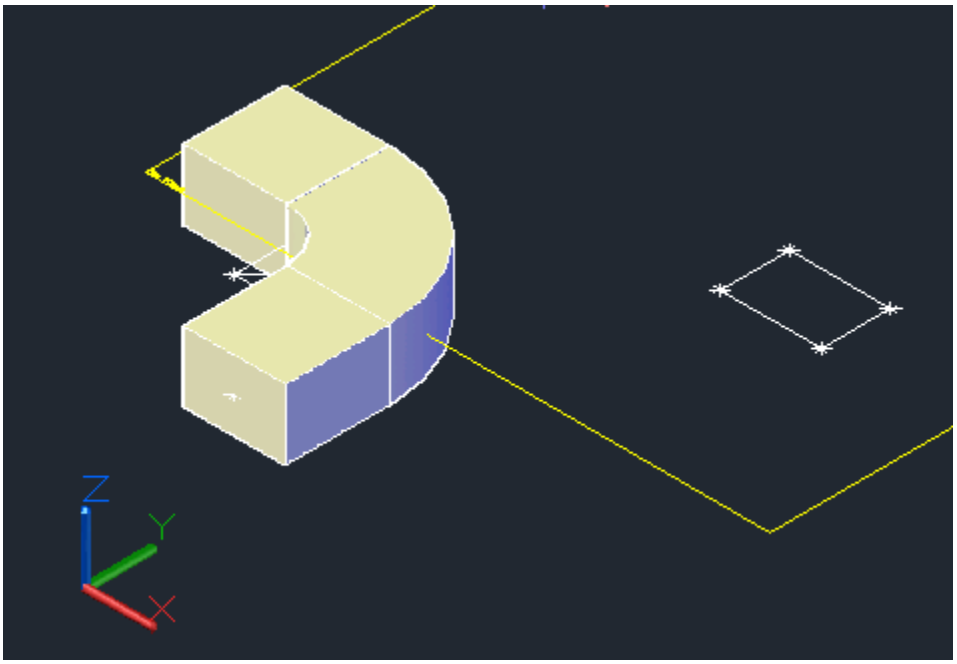

Modeling
Work Planes
Top Plane
Model Dimensions
Add Extrusion...
Add Path...
Add Transition...
Add Cut Plane...
Add Boolean Subtract...
Add Boolean Add...
PathA1
PrtSN
Z
Y


Z
Y
6" = 1'-0"
Standard
Edges/shades of Gray/SKetchy/X-ray/Other] <Conceptual>: _2dWire
Regenerating model.
Command: AddColModPath
Select path geometry:
Select start profile:

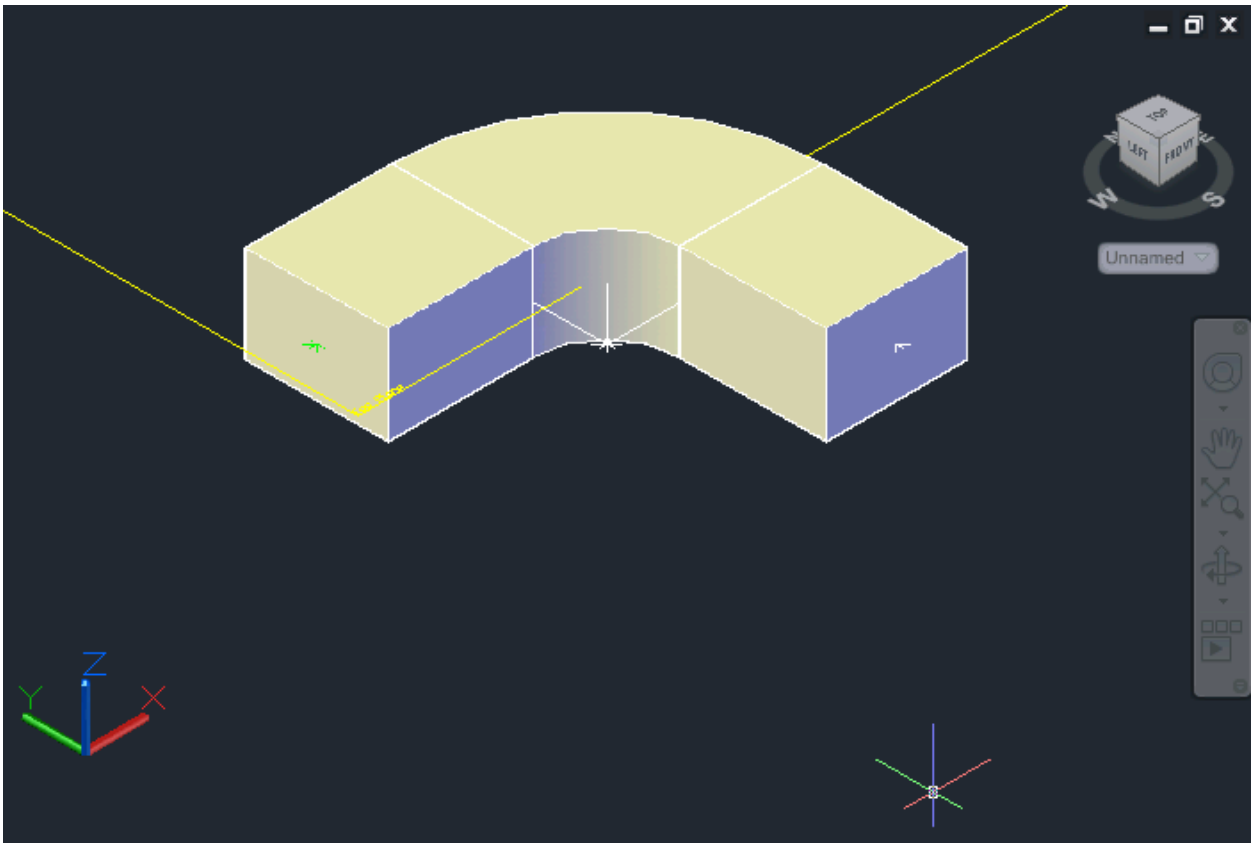


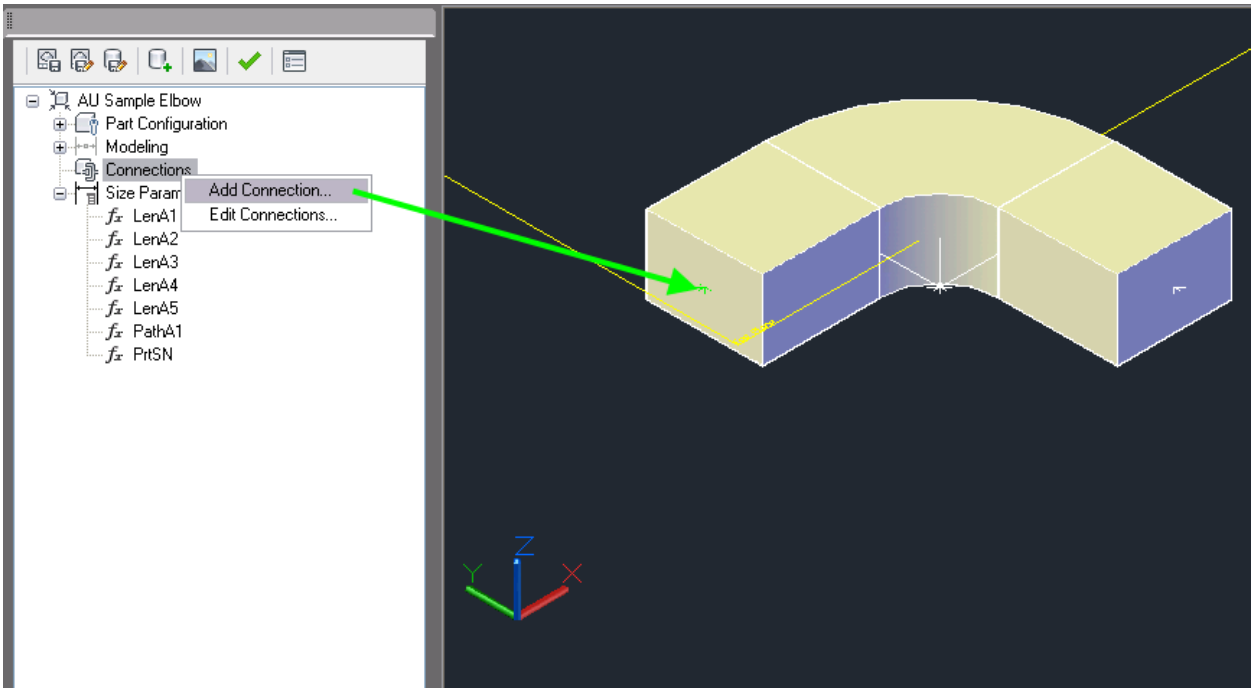

```
Regenerating model.
Command: AddColModPath
Select path geometry:
Select start profile:
Select end profile or <Enter>:
```

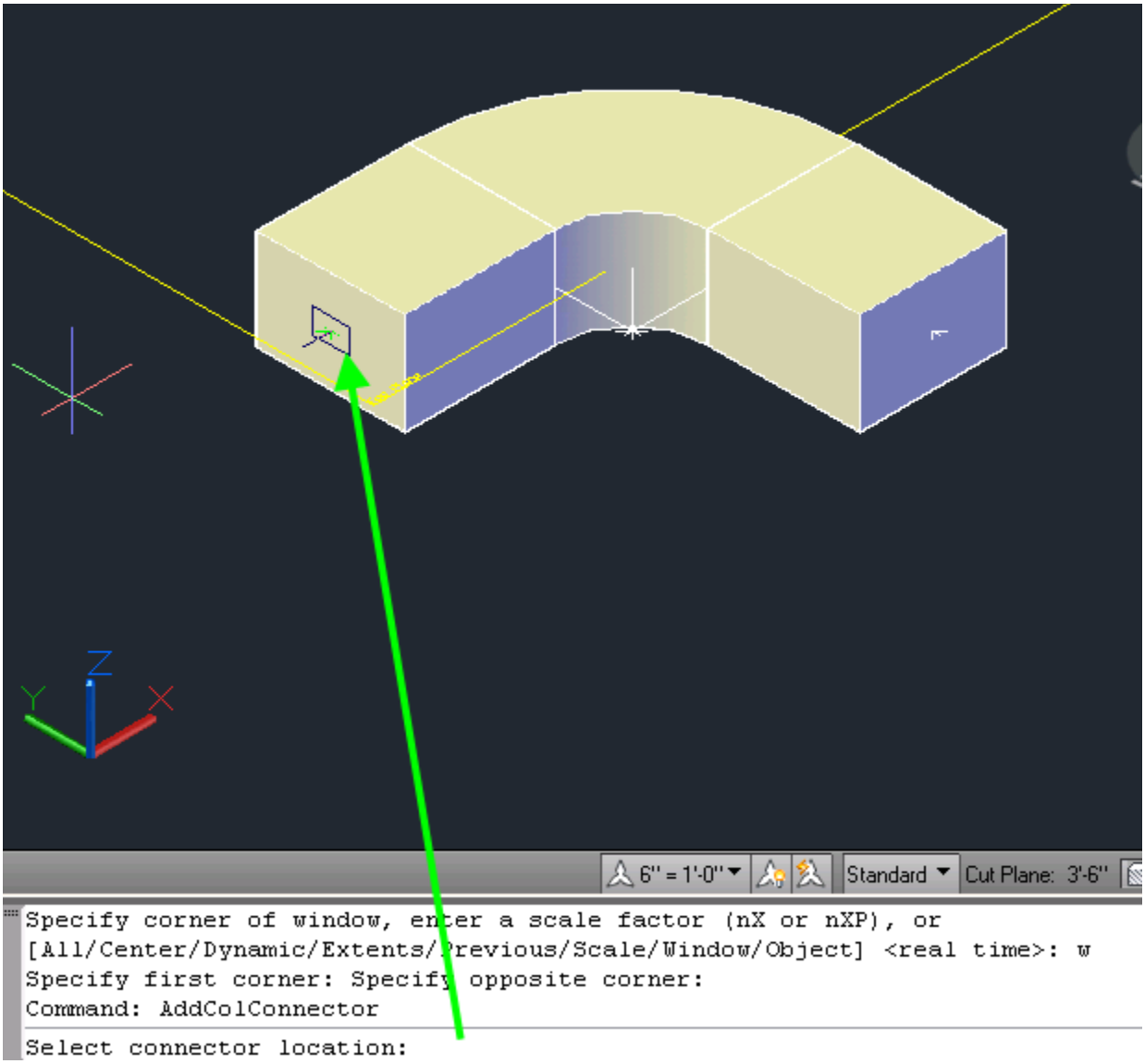
Press the enter key on your keyboard

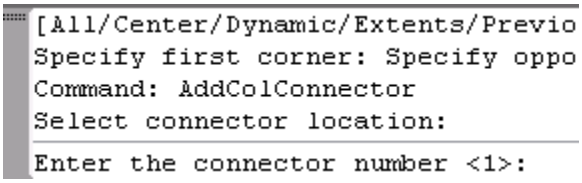


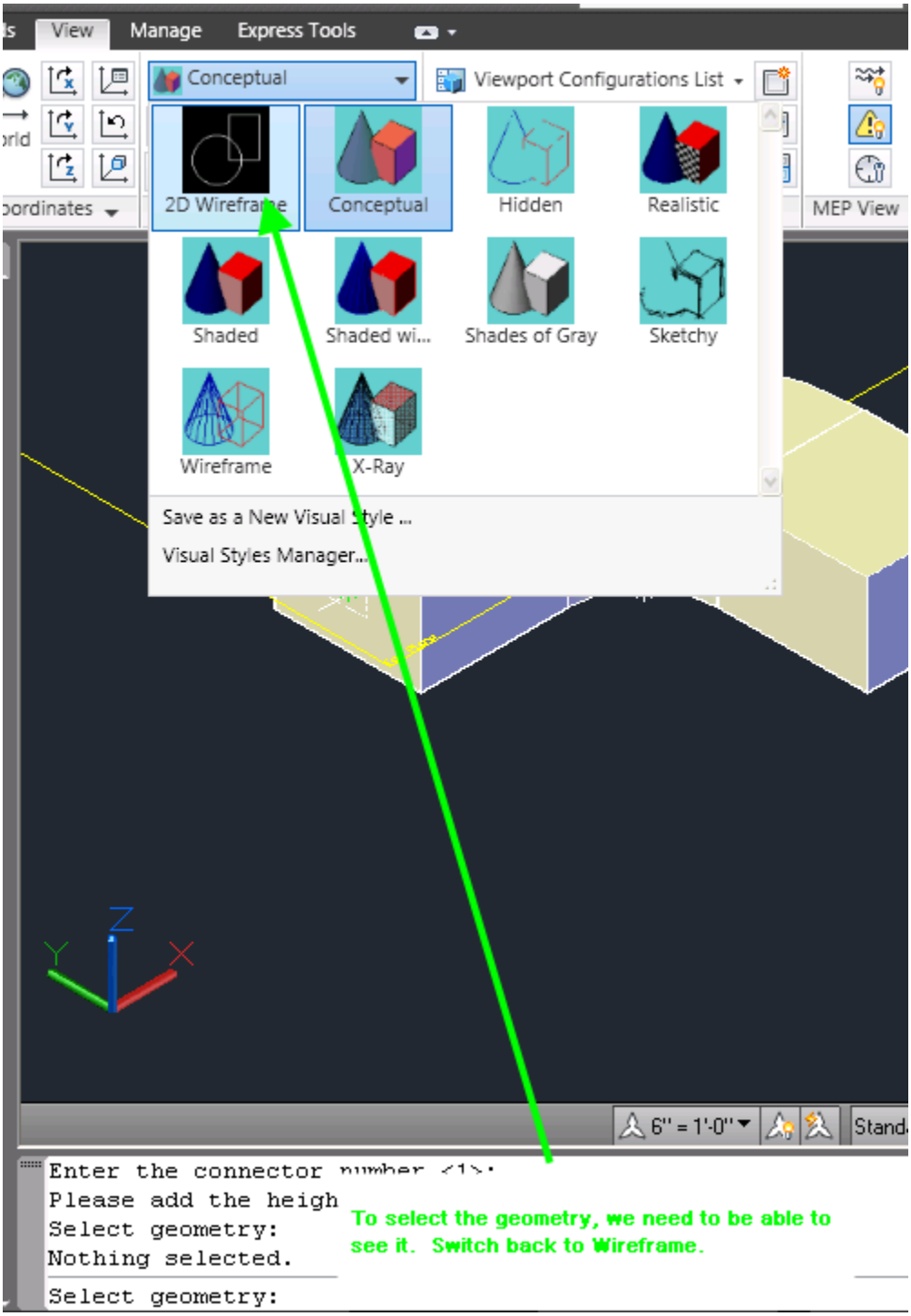

TOP
FRONT
RIGHT
S
E
Unnamed

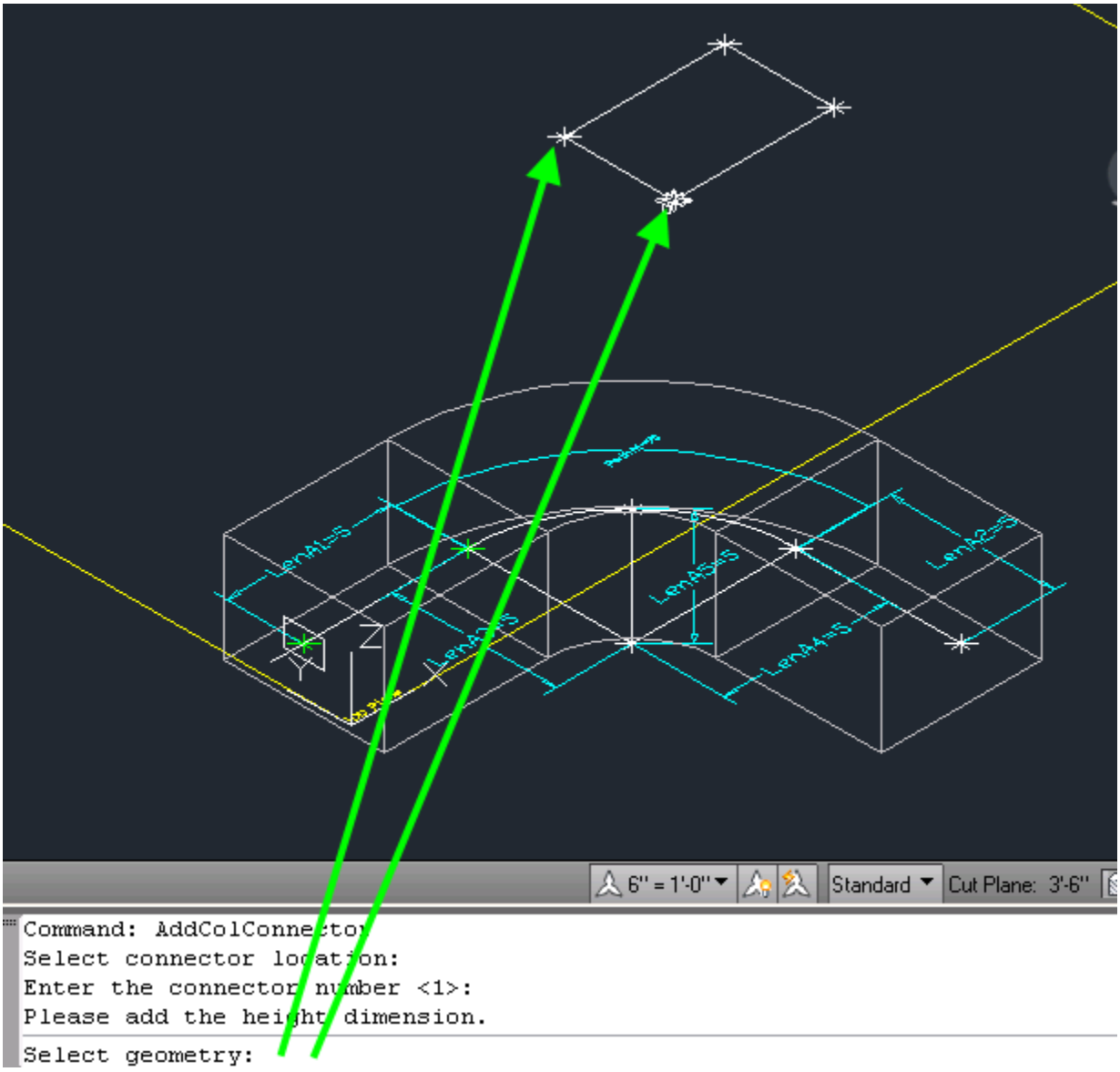

TOP
LEFT
FRONT
W
S
Unnamed
Z
Y
X

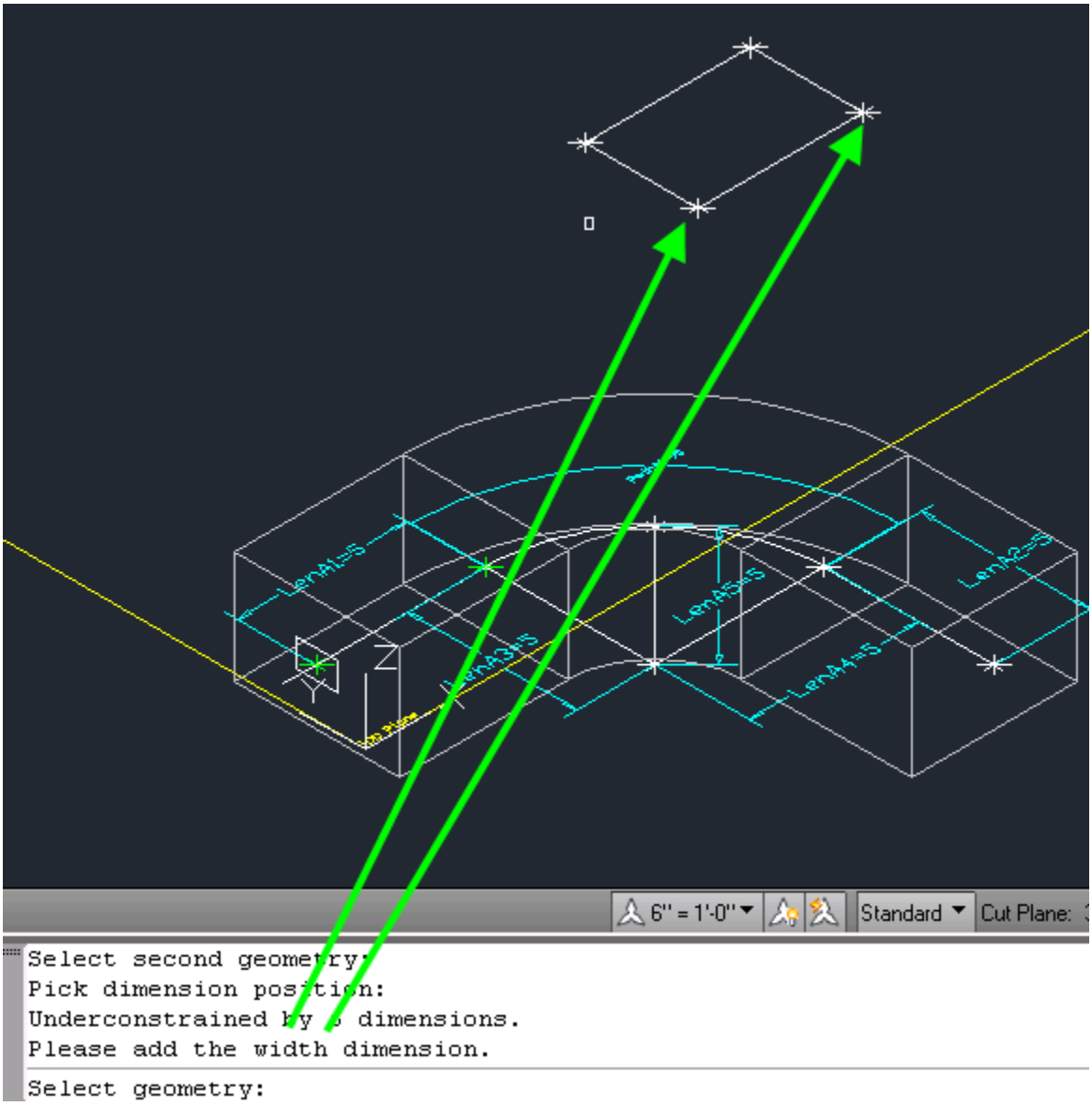

AU Sample Elbow
Part Configuration
Modeling
Connections
Size Param
Add Connection...
Edit Connections...
LenA1
LenA2
LenA3
LenA4
LenA5
PathA1
PrtSN
Z
Y
X

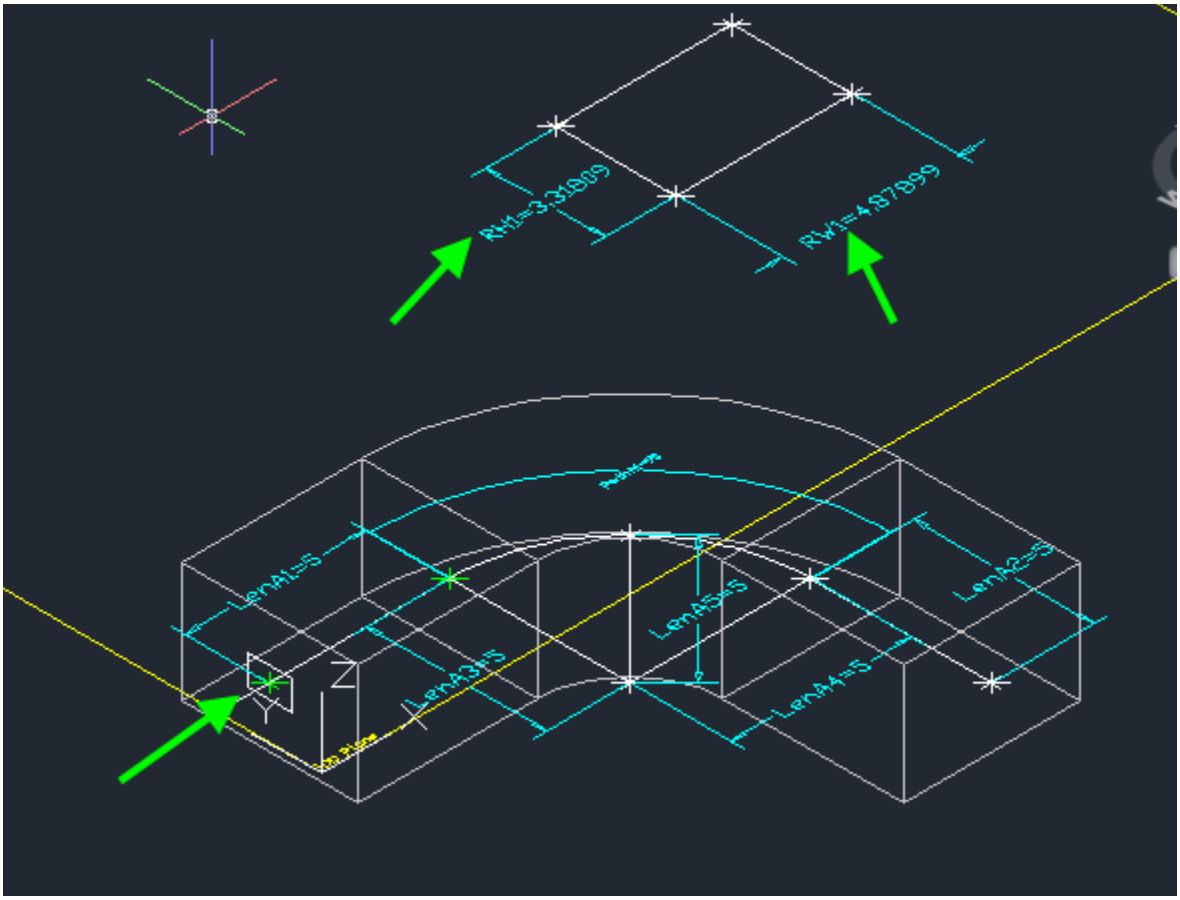

6" = 1'-0"
Standard
Cut Plane: 3'-6"
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: w
Specify first corner: Specify opposite corner:
Command: AddColConnector
Select connector location:

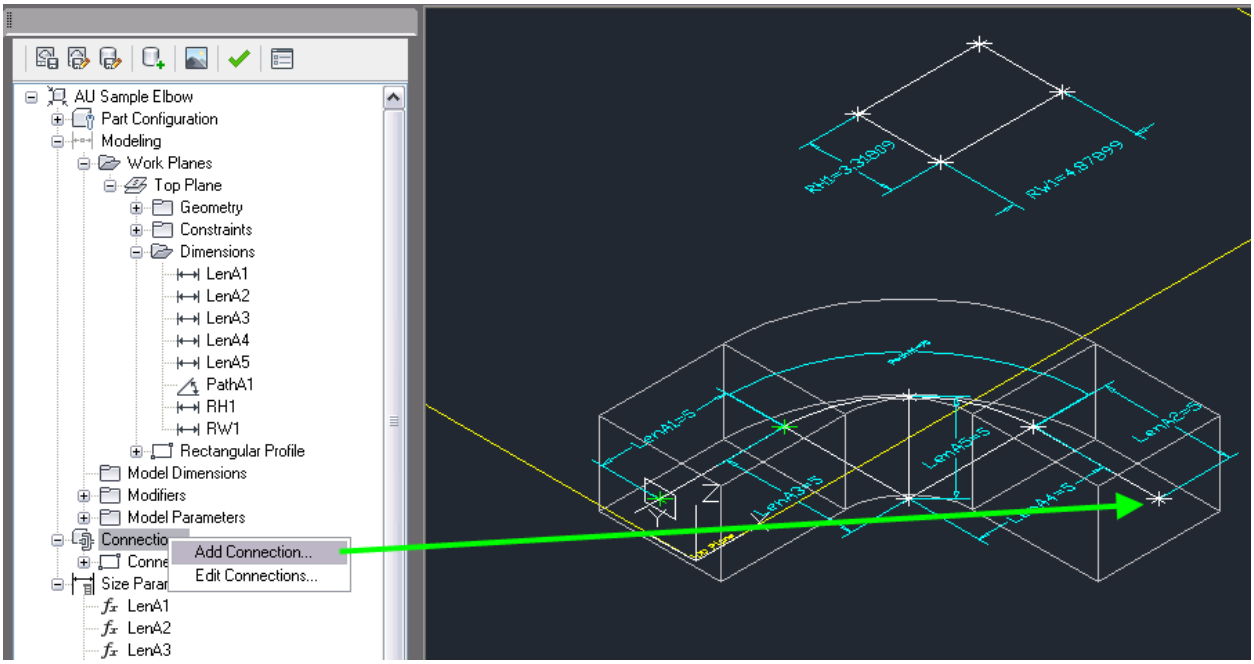

[All/Center/Dynamic/Extents/Previo
Specify first corner: Specify oppo
Command: AddColConnector
Select connector location:
Enter the connector number <1>:

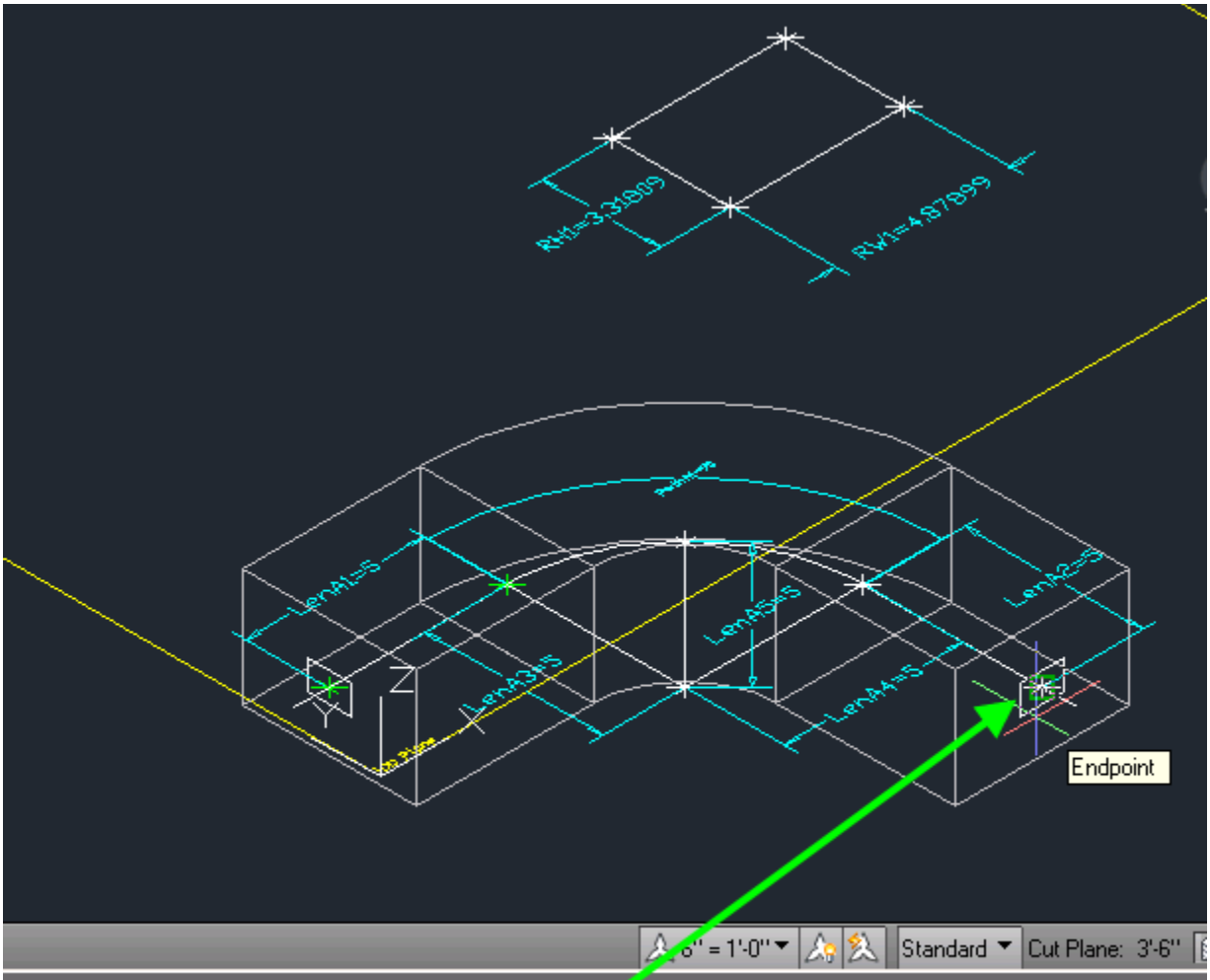

View
Manage
Express Tools
Conceptual
Viewport Configurations List
2D Wireframe
Conceptual
Hidden
Realistic
Shaded
Shaded wi...
Shades of Gray
Sketchy
Wireframe
X-Ray
Save as a New Visual Style ...
Visual Styles Manager...
MEP View
6" = 1'-0"
Enter the connector number <1>:
Please add the heigh
Select geometry:
Nothing selected.
Select geometry:
To select the geometry, we need to be able to see it. Switch back to Wireframe.


LenA1=S
LenA2=S
LenA3=S
LenA4=S
LenA5=S
6" = 1'-0"
Standard
Cut Plane: 3'-6"
Command: AddColConnector
Select connector location:
Enter the connector number <1>:
Please add the height dimension.
Select geometry:


LenA1=S
LenA3=S
LenA5=S
LenA2=S
LenA4=S
6" = 1'-0"
Standard
Cut Plane:
Select second geometry:
Pick dimension position:
Underconstrained by 5 dimensions.
Please add the width dimension.
Select geometry:

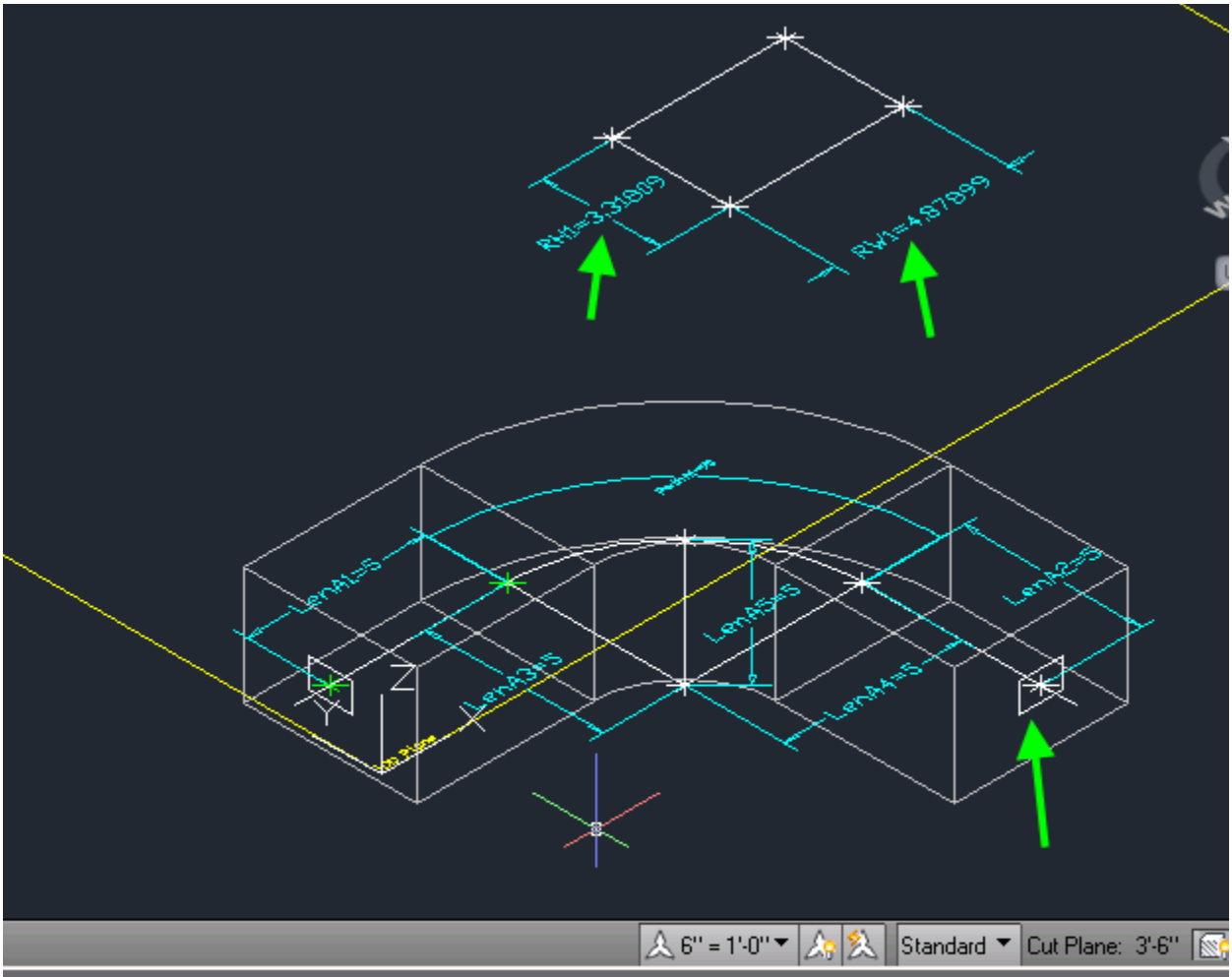

RH1=3.331809
RW1=4.487899
LenA1=5
LenA3=5
LenA5=5
LenA4=5
LenA2=5


AU Sample Elbow
Part Configuration
Modeling
Work Planes
Top Plane
Geometry
Constraints
Dimensions
LenA1
LenA2
LenA3
LenA4
LenA5
PathA1
RH1
RW1
Rectangular Profile
Model Dimensions
Modifiers
Model Parameters
Connectio
Add Connection...
Edit Connections...
Size Parar
LenA1
LenA2
LenA3
RH1=3.331809
RW1=4.487899

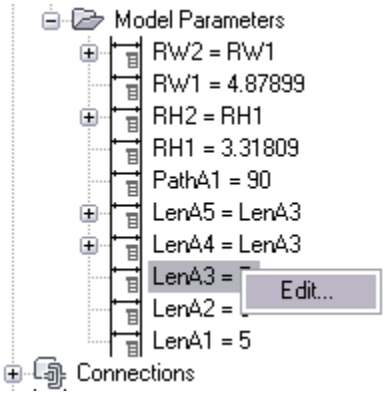


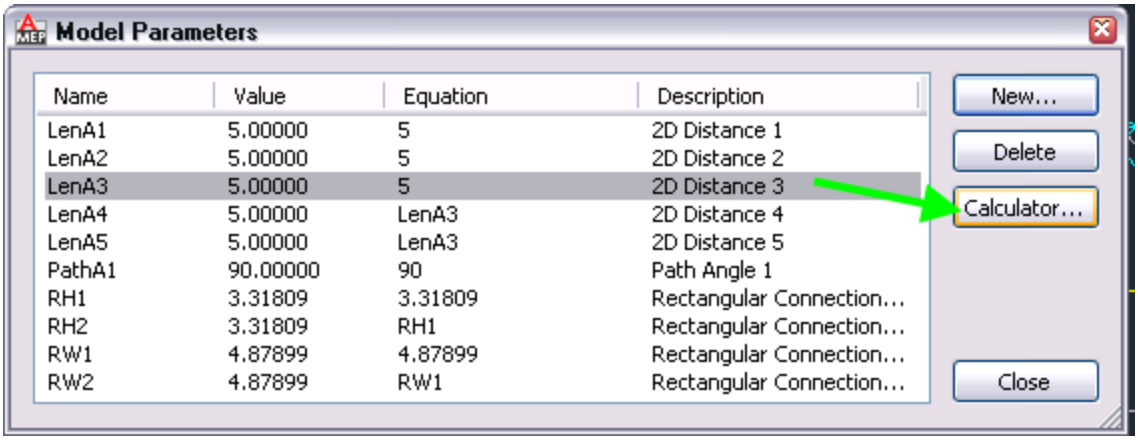

```
Select second geometry:
Pick dimension position:
Underconstrained by 5 dimensions.
Command: AddColConnector
Select connector location:
```

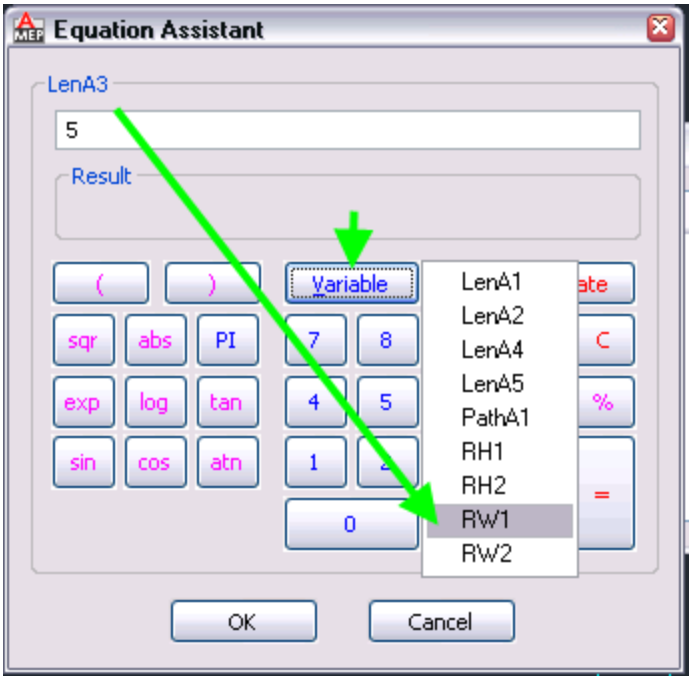
```
Pick dimension position:
Underconstrained by 5 dimensions.
Command: AddColConnector
Select connector location:
Enter the connector number <2>:
```



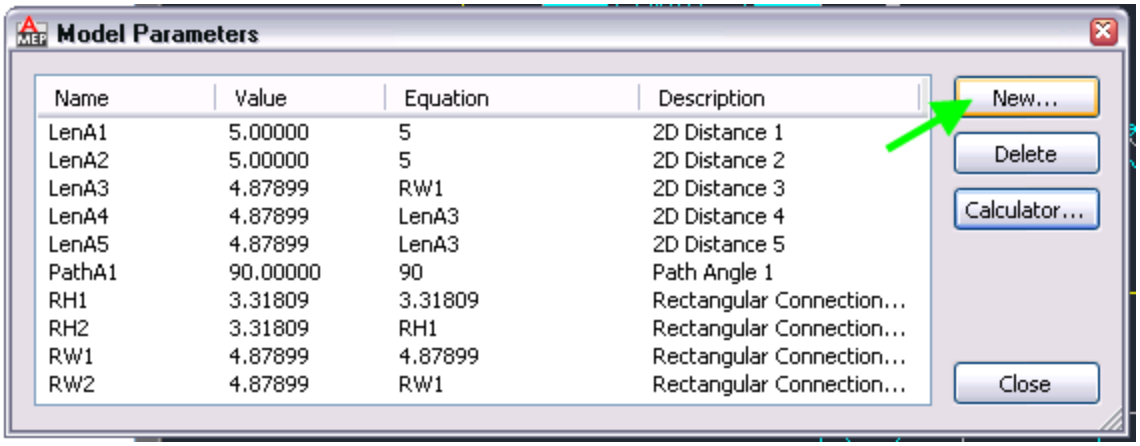

```
Underconstrained by 5 dimensions.
Command: AddColConnector
Select connector location:
Enter the connector number <2>:
Command:
```

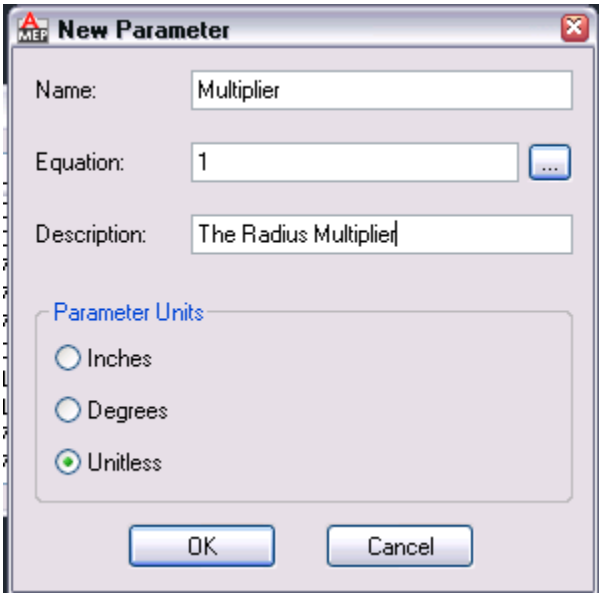


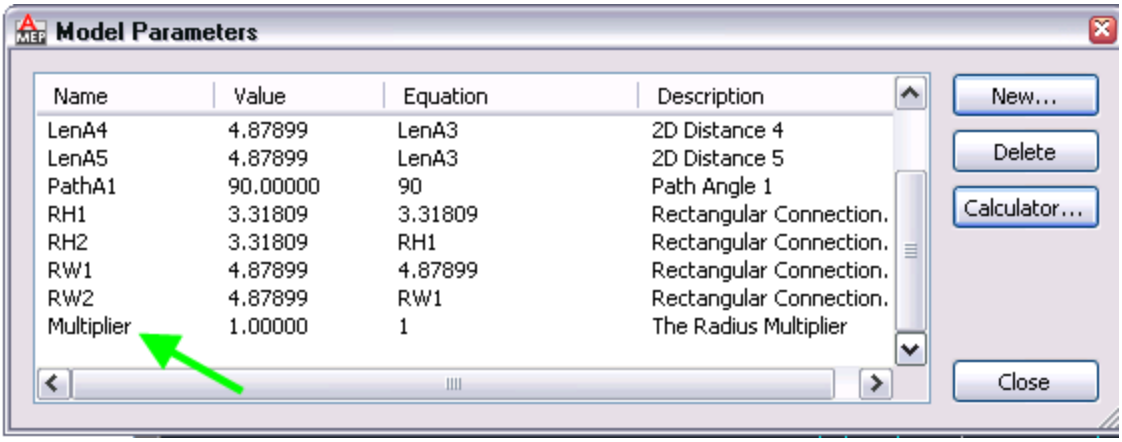

Model Parameters
Name
Value
Equation
Description
LenA1 5.00000 5 2D Distance 1
LenA2 5.00000 5 2D Distance 2
LenA3 5.00000 5 2D Distance 3
LenA4 5.00000 LenA3 2D Distance 4
LenA5 5.00000 LenA3 2D Distance 5
PathA1 90.00000 90 Path Angle 1
RH1 3.31809 3.31809 Rectangular Connection...
RH2 3.31809 RH1 Rectangular Connection...
RW1 4.87899 4.87899 Rectangular Connection...
RW2 4.87899 RW1 Rectangular Connection...
New...
Delete
Calculator...
Close

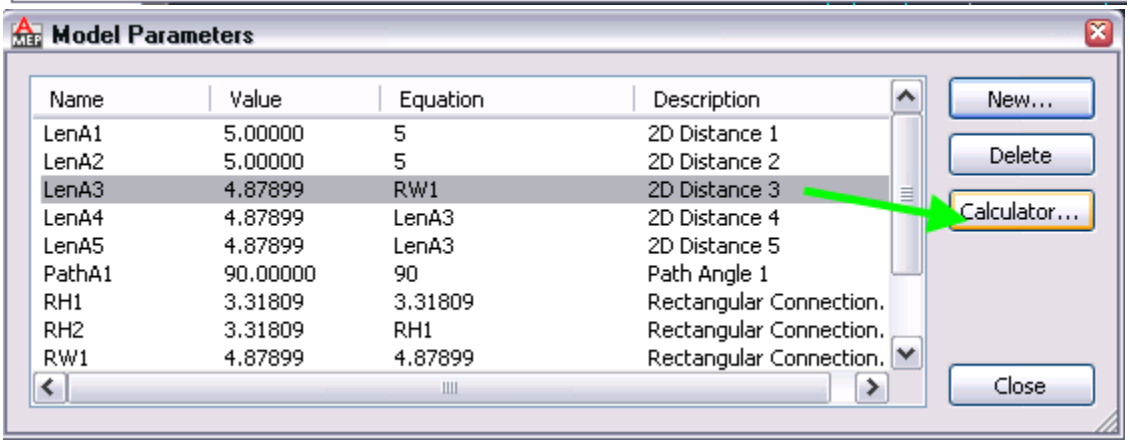

Equation Assistant
LenA3
5
Result
(
)
Variable
sqr
abs
PI
7
8
exp
log
tan
4
5
sin
cos
atn
1
0
LenA1
LenA2
LenA4
LenA5
PathA1
RH1
RH2
RW1
RW2
OK
Cancel

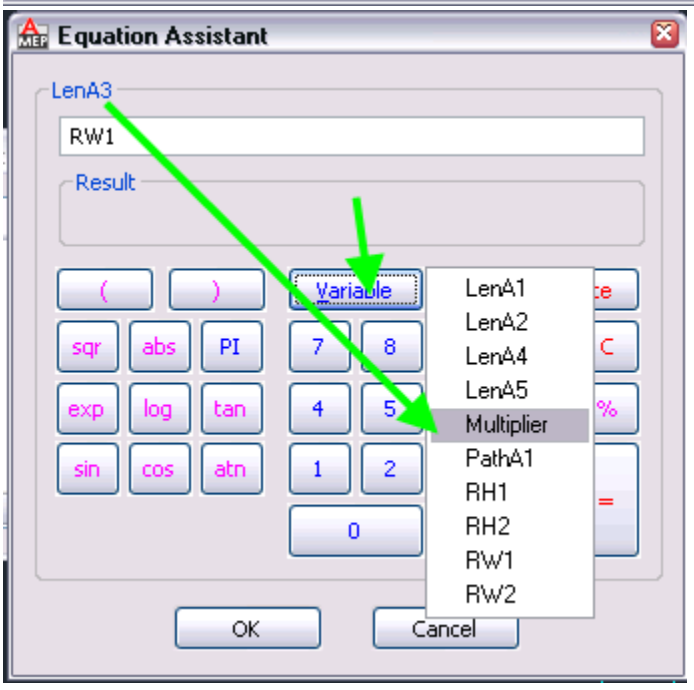
OR

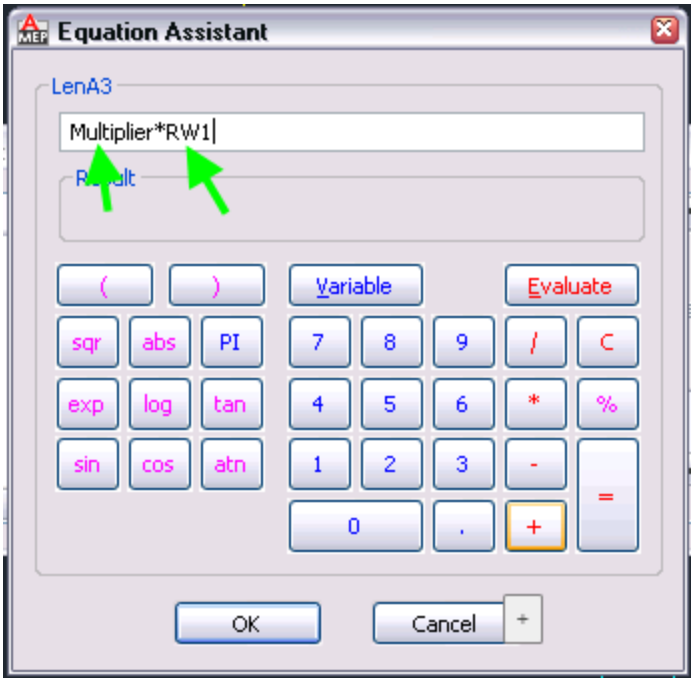

Model Parameters
Name
Value
Equation
Description
LenA1 5.00000 5 2D Distance 1
LenA2 5.00000 5 2D Distance 2
LenA3 4.87899 RW1 2D Distance 3
LenA4 4.87899 LenA3 2D Distance 4
LenA5 4.87899 LenA3 2D Distance 5
PathA1 90.00000 90 Path Angle 1
RH1 3.31809 3.31809 Rectangular Connection...
RH2 3.31809 RH1 Rectangular Connection...
RW1 4.87899 4.87899 Rectangular Connection...
RW2 4.87899 RW1 Rectangular Connection...
New...
Delete
Calculator...
Close

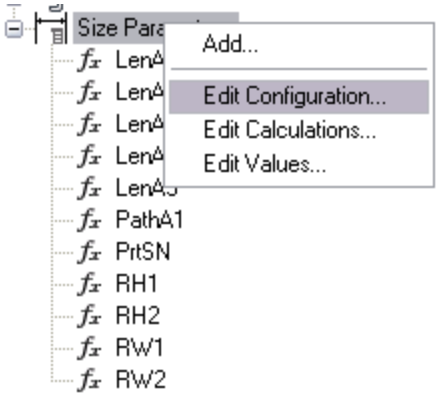

New Parameter
Name:
Multiplier
Equation:
1
...
Description:
The Radius Multiplier
Parameter Units
Inches
Degrees
Unitless
OK
Cancel

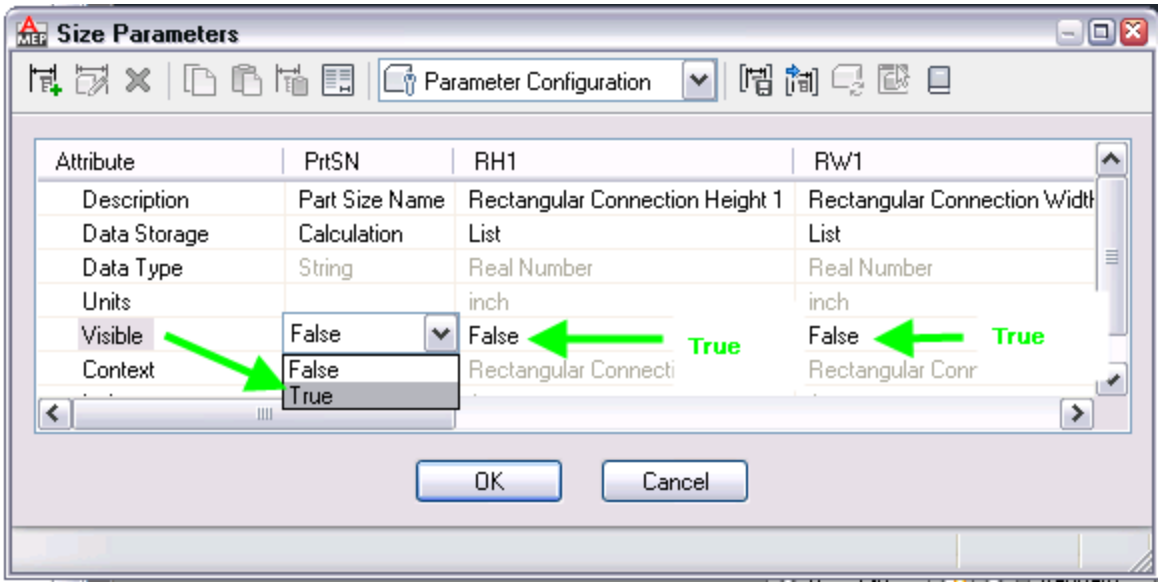

Model Parameters
Name
Value
Equation
Description
LenA4 4.87899 LenA3 2D Distance 4
LenA5 4.87899 LenA3 2D Distance 5
PathA1 90.00000 90 Path Angle 1
RH1 3.31809 3.31809 Rectangular Connection.
RH2 3.31809 RH1 Rectangular Connection.
RW1 4.87899 4.87899 Rectangular Connection.
RW2 4.87899 RW1 Rectangular Connection.
Multiplier 1.00000 1 The Radius Multiplier
New...
Delete
Calculator...
Close

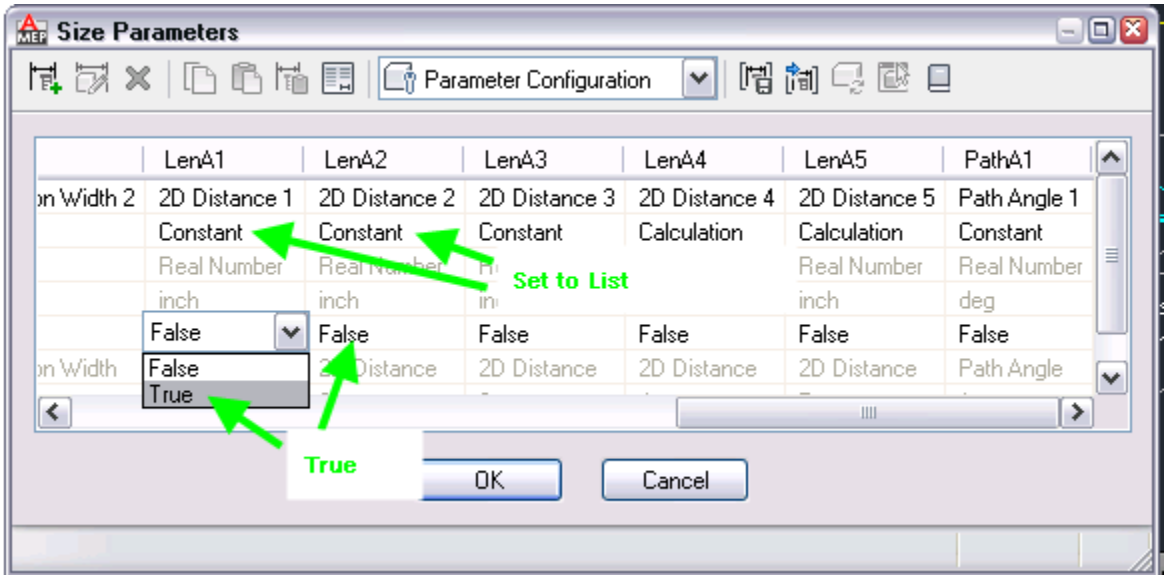

Model Parameters
Name
Value
Equation
Description
LenA1 5.00000 5 2D Distance 1
LenA2 5.00000 5 2D Distance 2
LenA3 4.87899 RW1 2D Distance 3
LenA4 4.87899 LenA3 2D Distance 4
LenA5 4.87899 LenA3 2D Distance 5
PathA1 90.00000 90 Path Angle 1
RH1 3.31809 3.31809 Rectangular Connection.
RH2 3.31809 RH1 Rectangular Connection.
RW1 4.87899 4.87899 Rectangular Connection.
New...
Delete
Calculator...
Close

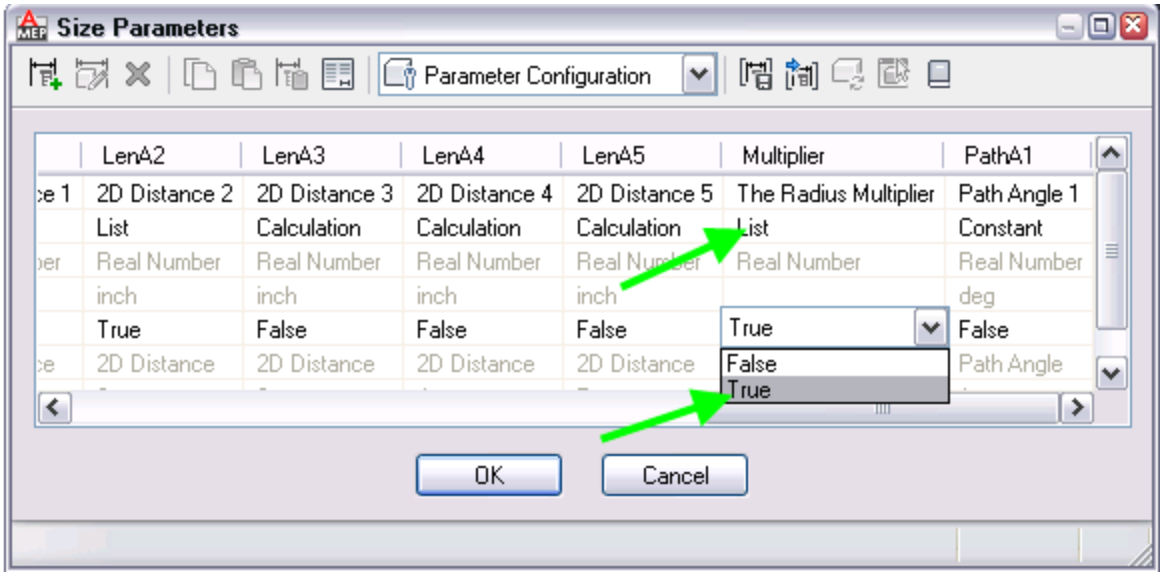

Equation Assistant
LenA3
RW1
Result
(
)
Variable
sqr
abs
PI
exp
log
tan
sin
cos
atn
7
8
4
5
1
2
0
%
=
LenA1
LenA2
LenA4
LenA5
Multiplier
PathA1
RH1
RH2
RW1
RW2
OK
Cancel

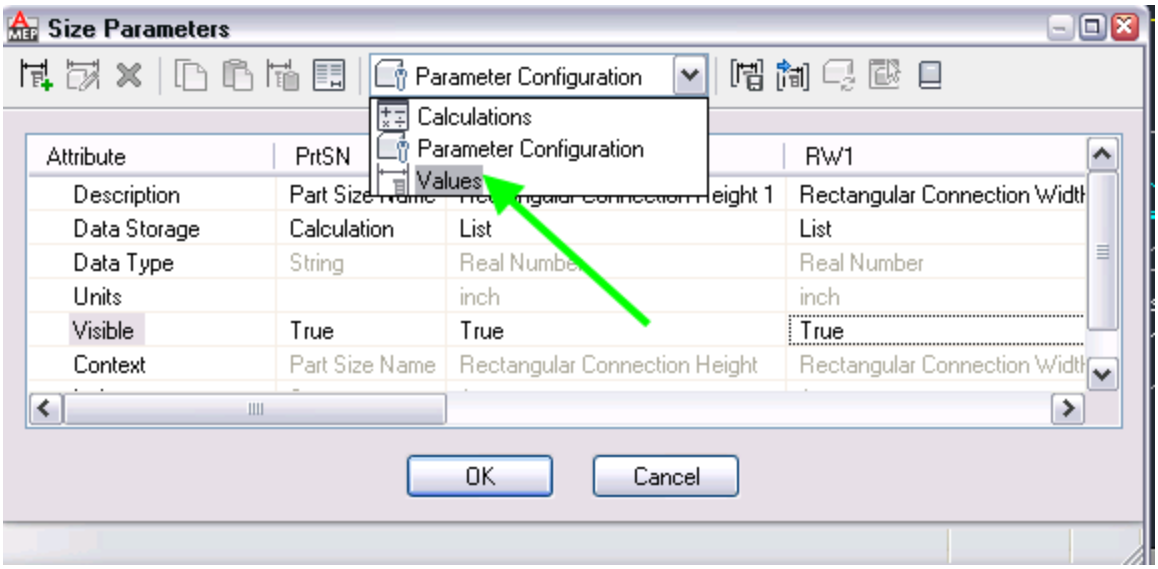

Equation Assistant
LenA3
Multiplier*RW1
Result
(
)
Variable
Evaluate
sqr
abs
PI
7
8
9
/
C
exp
log
tan
4
5
6
*
%
sin
cos
atn
1
2
3
-
=
0
.
+
OK
Cancel

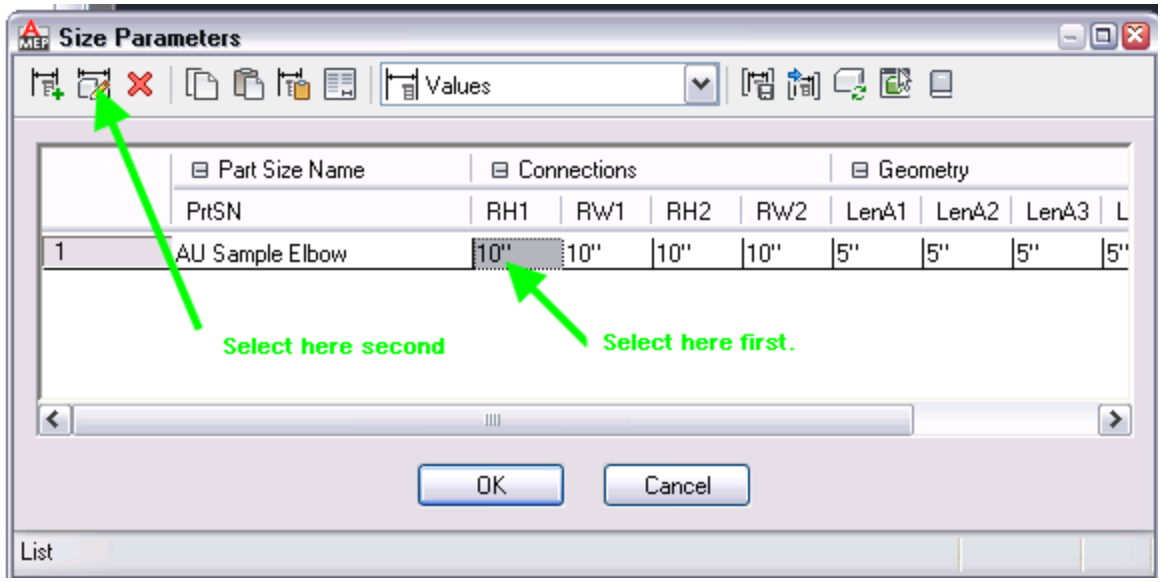

Add...
Edit Configuration...
Edit Calculations...
Edit Values...
PathA1
PrtSN
RH1
RH2
RW1
RW2

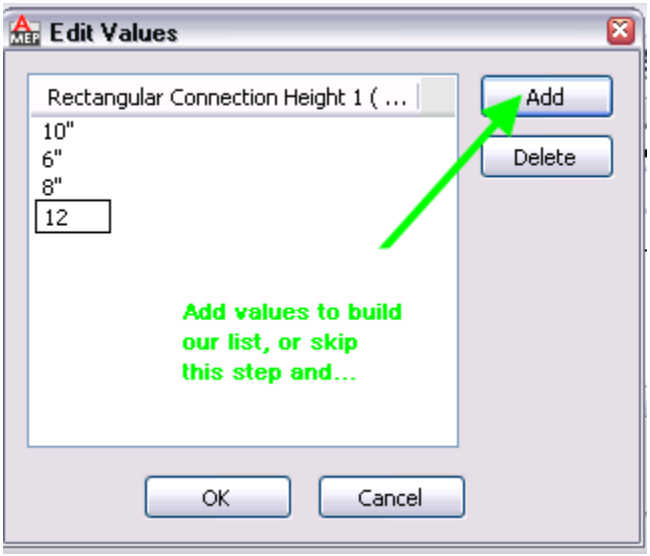

Size Parameters
Parameter Configuration
Attribute
PrtSN
RH1
RW1
Description
Part Size Name
Rectangular Connection Height 1
Rectangular Connection Width
Data Storage
Calculation
List
List
Data Type
String
Real Number
Real Number
Units
inch
inch
Visible
False
False
True
False
True
Context
False
True
Rectangular Connecti
Rectangular Conr
OK
Cancel

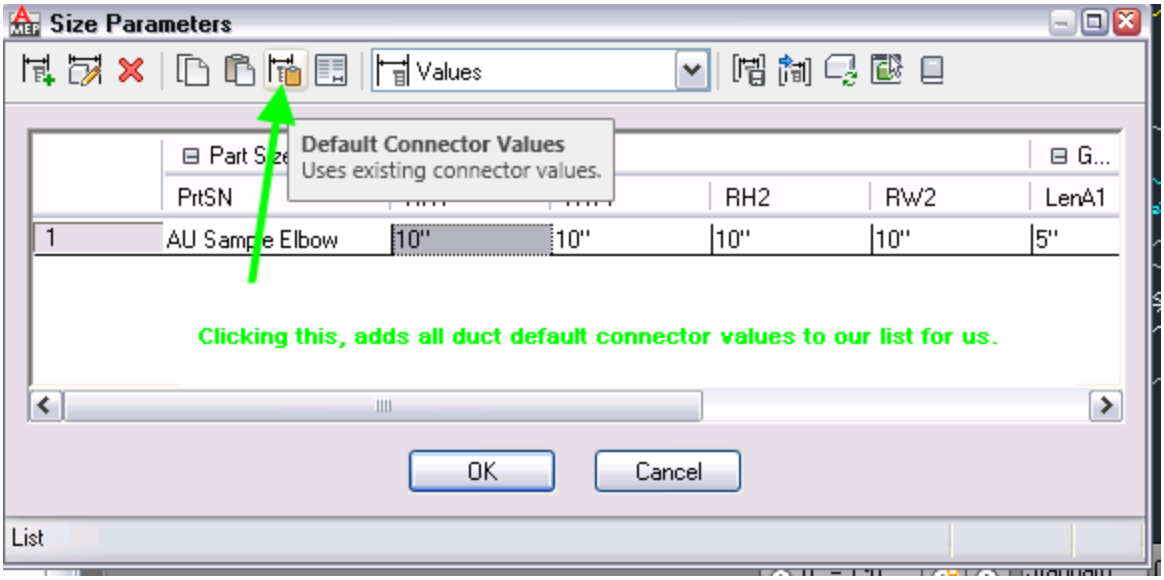

Size Parameters
Parameter Configuration
LenA1
LenA2
LenA3
LenA4
LenA5
PathA1
2D Distance 1
2D Distance 2
2D Distance 3
2D Distance 4
2D Distance 5
Path Angle 1
Constant
Constant
Constant
Calculation
Calculation
Constant
Real Number
Real Number
Real Number
Real Number
inch
inch
inch
deg
False
False
False
False
False
False
False
True
2D Distance
2D Distance
2D Distance
Path Angle
Set to List
True
OK
Cancel

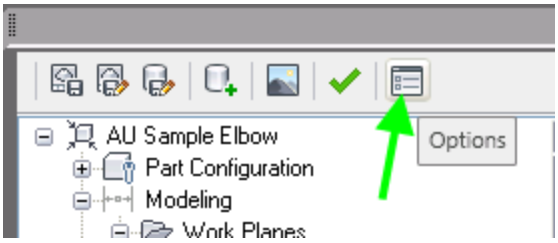

Size Parameters
Parameter Configuration
LenA2
LenA3
LenA4
LenA5
Multiplier
PathA1
2D Distance 2
2D Distance 3
2D Distance 4
2D Distance 5
The Radius Multiplier
Path Angle 1
List
Calculation
Calculation
Calculation
List
Constant
Real Number
Real Number
Real Number
Real Number
Real Number
Real Number
inch
inch
inch
inch
deg
True
False
False
False
True
False
2D Distance
2D Distance
2D Distance
2D Distance
Path Angle
False
True
OK
Cancel

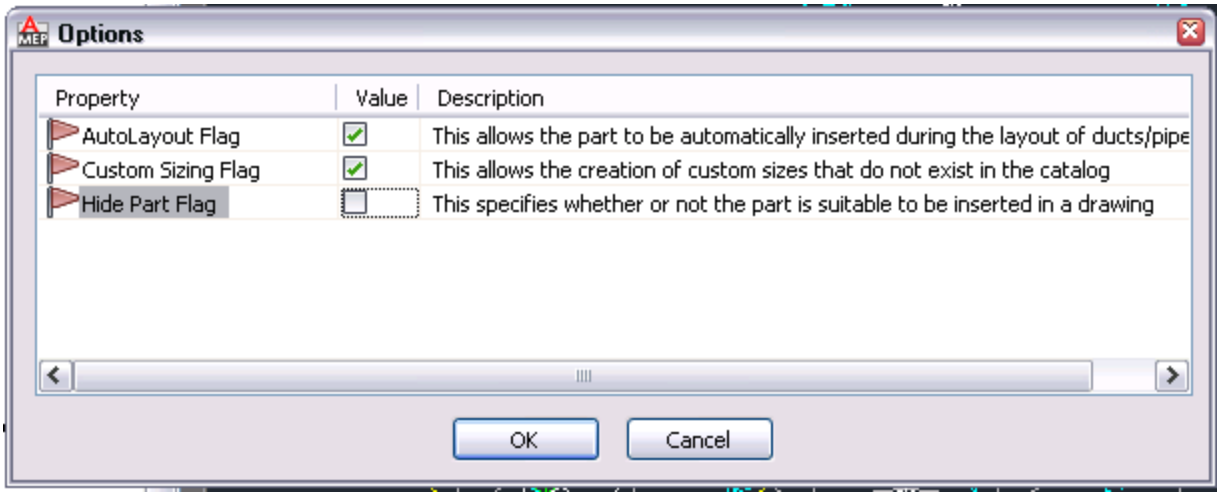

Size Parameters
Parameter Configuration
Calculations
Parameter Configuration
Values
Attribute
PrtSN
RW1
Description
Rectangular Connection Width
Data Storage
Calculation
List
List
Data Type
String
Real Number
Units
inch
inch
Visible
True
True
True
Context
Part Size Name
Rectangular Connection Height
Rectangular Connection Width
OK
Cancel

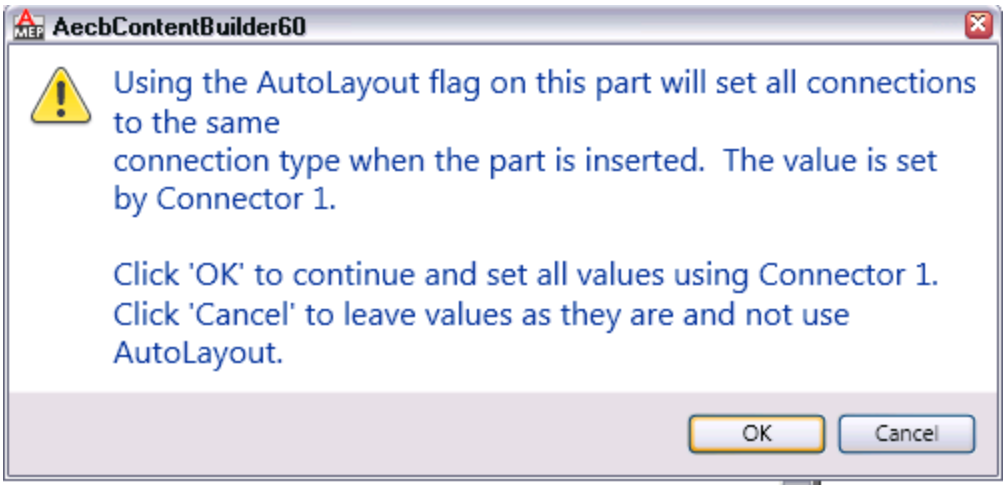

Size Parameters
Values
Part Size Name
Connections
Geometry
PrtSN
RH1
RW1
RH2
RW2
LenA1
LenA2
LenA3
1
AU Sample Elbow
10"
10"
10"
10"
5"
5"
5"
5"
Select here second
Select here first.
OK
Cancel
List

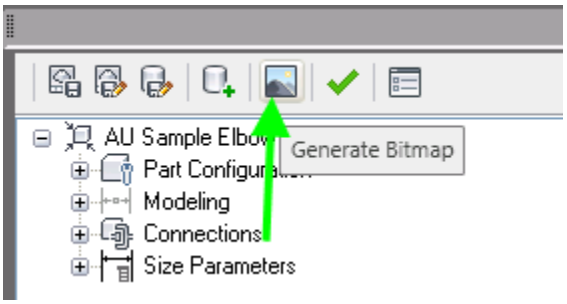

Edit Values
Rectangular Connection Height 1 (...
10"
6"
8"
12
Add
Delete
Add values to build our list, or skip this step and...
OK
Cancel

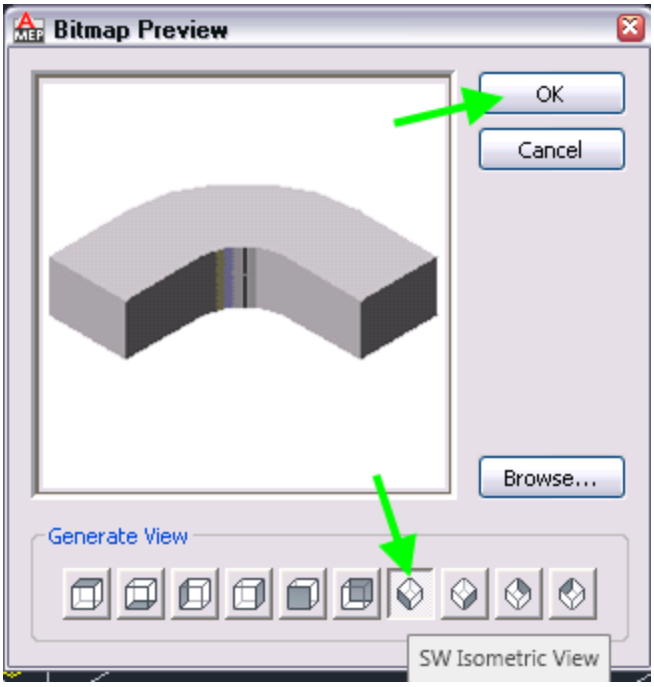

Size Parameters
Values
Default Connector Values
Uses existing connector values.
PrtSN
RH2
RW2
LenA1
1
AU Sample Elbow
10"
10"
10"
10"
5"
Clicking this, adds all duct default connector values to our list for us.
OK
Cancel
List

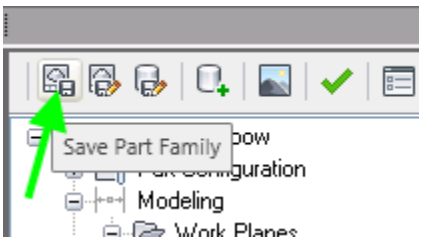

AU Sample Elbow
Part Configuration
Modeling
Options

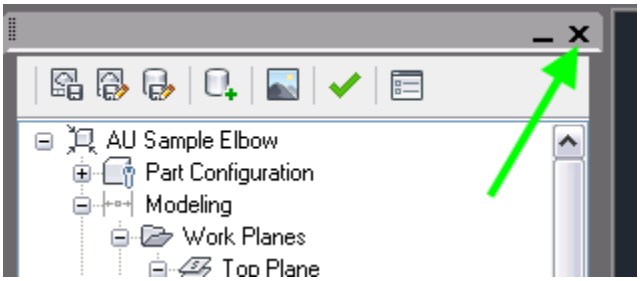

Options
Property
Value
Description
AutoLayout Flag
This allows the part to be automatically inserted during the layout of ducts/pipe
Custom Sizing Flag
This allows the creation of custom sizes that do not exist in the catalog
Hide Part Flag
This specifies whether or not the part is suitable to be inserted in a drawing
OK
Cancel

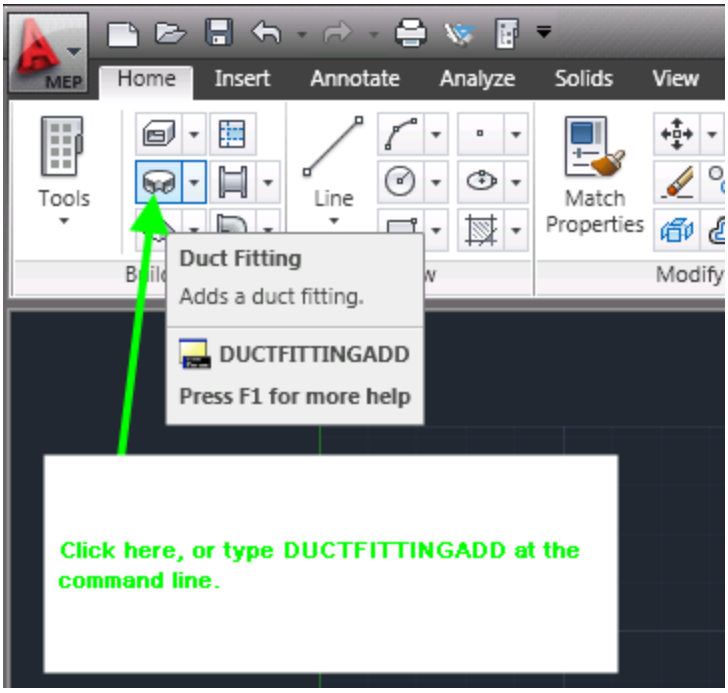

AecbContentBuilder60
Using the AutoLayout flag on this part will set all connections to the same
connection type when the part is inserted. The value is set by Connector 1.
Click 'OK' to continue and set all values using Connector 1.
Click 'Cancel' to leave values as they are and not use AutoLayout.
OK
Cancel

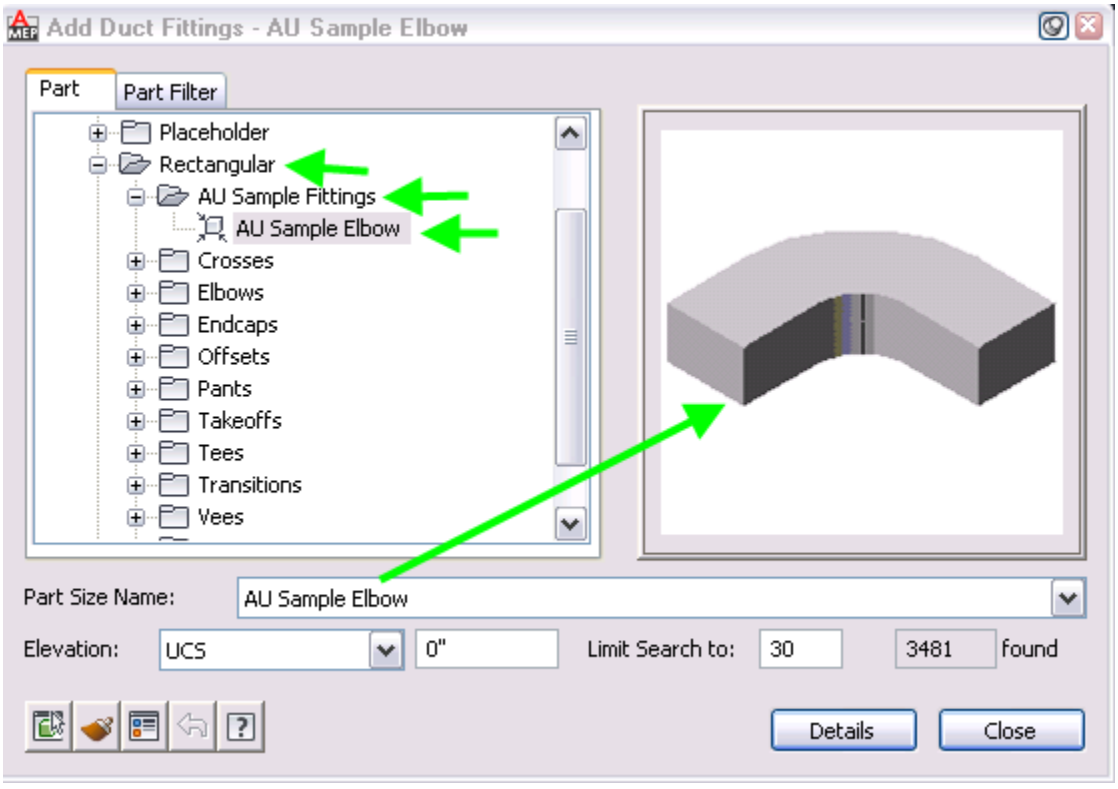

Generate Bitmap
Modeling
Connections
Size Parameters

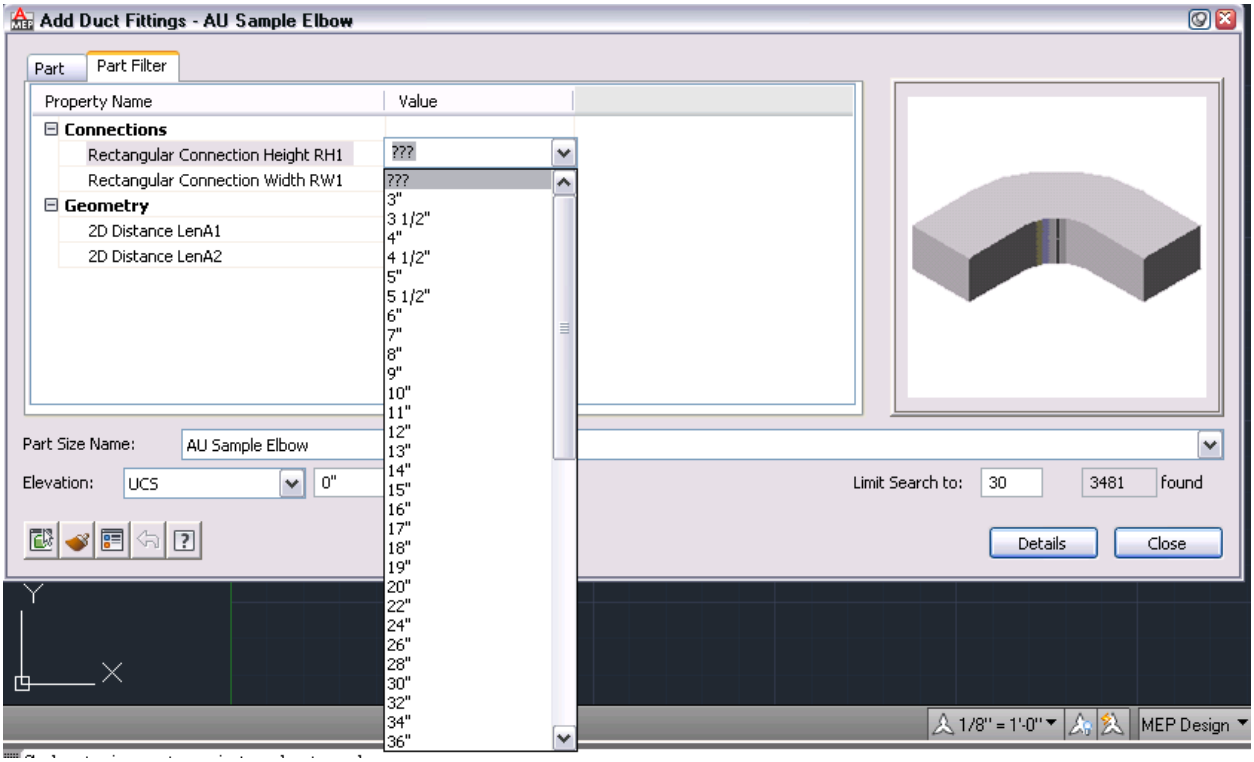

Bitmap Preview
OK
Cancel
Browse...
Generate View
SW Isometric View

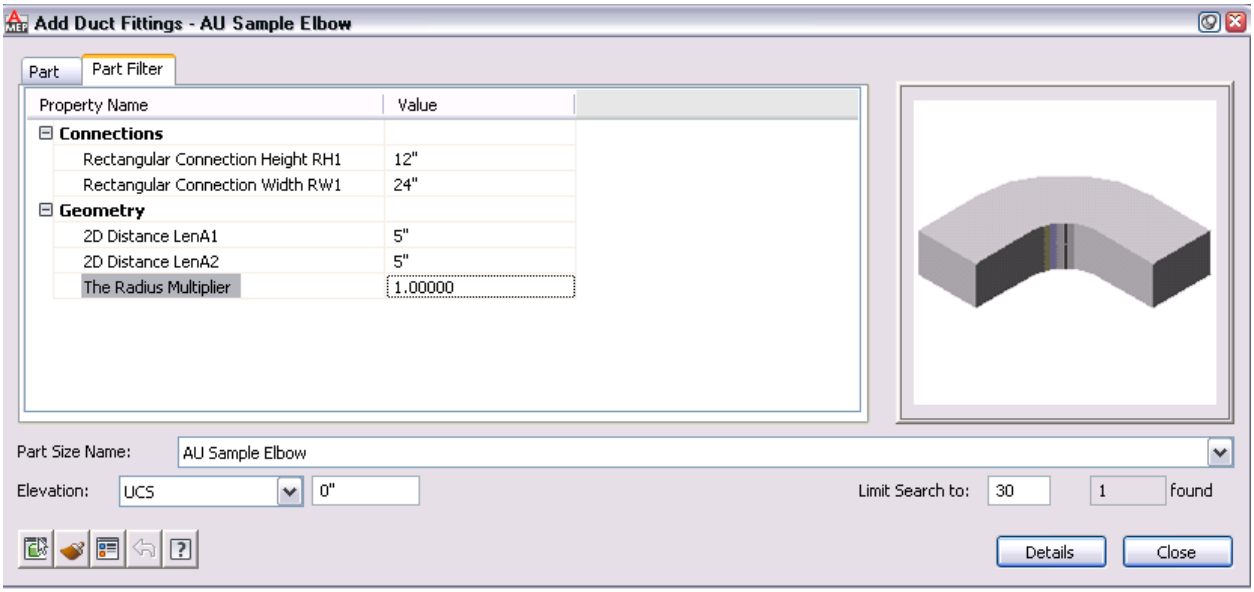

Save Part Family
Modeling

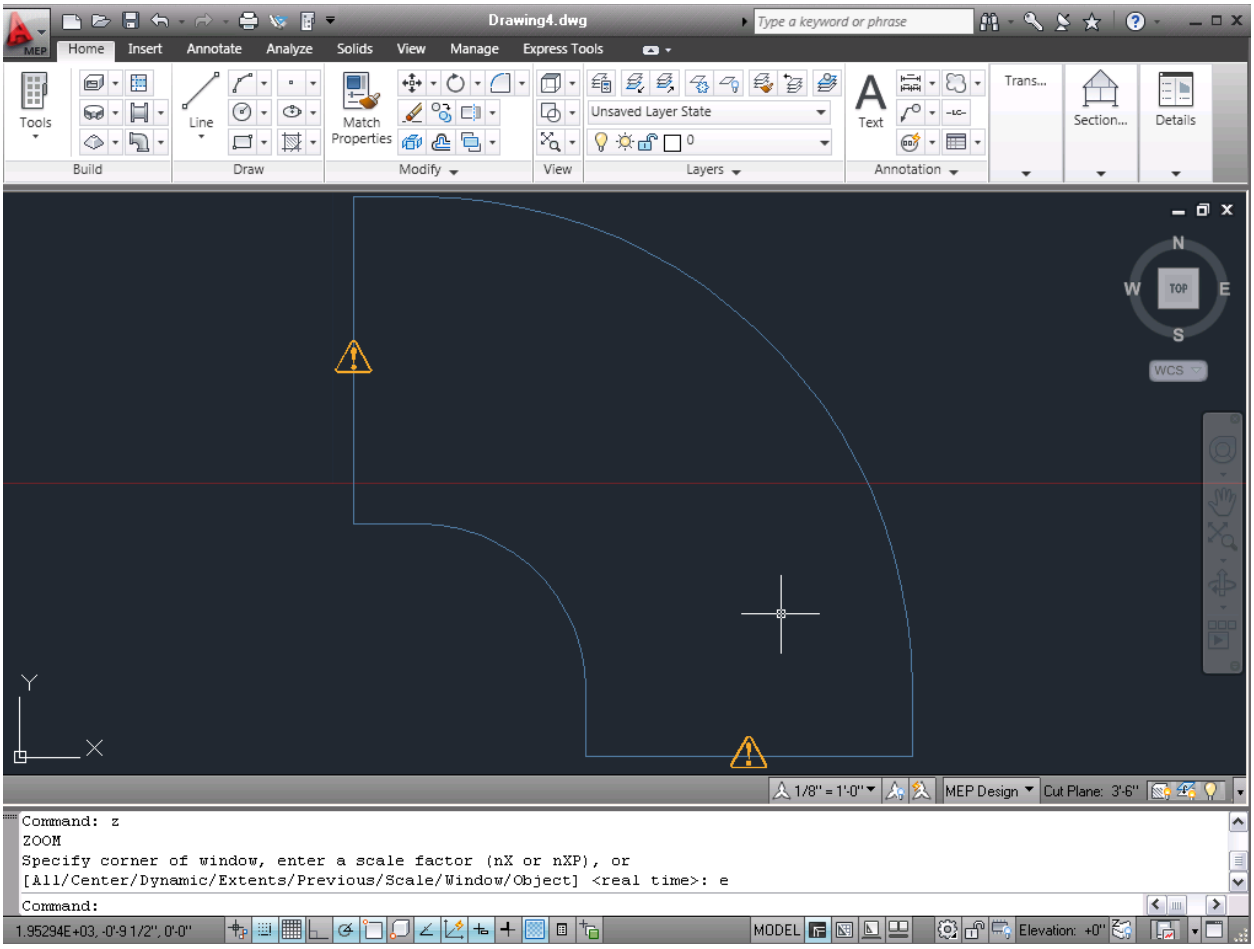

AU Sample Elbow
Part Configuration
Modeling
Work Planes
Top Plane


MEP
Home
Insert
Annotate
Analyze
Solids
View
Tools
Line
Match Properties
Modify
Duct Fitting
Adds a duct fitting.
DUCTFITTINGADD
Press F1 for more help
Click here, or type DUCTFITTINGADD at the command line.


Add Duct Fittings - AU Sample Elbow
Part
Part Filter
Placeholder
Rectangular
AU Sample Fittings
AU Sample Elbow
Crosses
Elbows
Endcaps
Offsets
Pants
Takeoffs
Tees
Transitions
Vees
Part Size Name:
AU Sample Elbow
Elevation:
UCS
0"
Limit Search to:
30
3481
found
Details
Close


Add Duct Fittings - AU Sample Elbow
Part
Part Filter
Property Name
Value
Connections
Rectangular Connection Height RH1
???
Rectangular Connection Width RW1
Geometry
2D Distance LenA1
2D Distance LenA2
???
3"
3 1/2"
4"
4 1/2"
5"
5 1/2"
6"
7"
8"
9"
10"
11"
12"
13"
14"
15"
16"
17"
18"
19"
20"
22"
24"
26"
28"
30"
32"
34"
36"
Part Size Name:
AU Sample Elbow
Elevation:
UCS
0"
Limit Search to:
30
3481
found
Details
Close
1/8" = 1'-0"
MEP Design


Add Duct Fittings - AU Sample Elbow
Part
Part Filter
Property Name
Value
Connections
Rectangular Connection Height RH1
12"
Rectangular Connection Width RW1
24"
Geometry
2D Distance LenA1
5"
2D Distance LenA2
5"
The Radius Multiplier
1.00000
Part Size Name:
AU Sample Elbow
Elevation:
UCS
0"
Limit Search to:
30
1
found
Details
Close



The FINAL product

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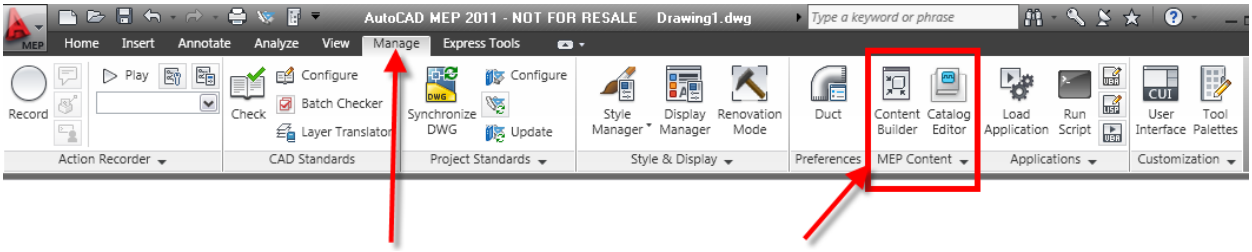
Appendix:

Steps to create your own company catalog:

1. Find the manage tab on the Ribbon.
2. Find the MEP Content group > Catalog Editor and launch it.



3. In the Catalog Editor dialog, File > New.
4. In the new Catalog dialog box, set the Part Domain. Is it a Cable Tray fitting catalog, Duct fitting, Conduit, Pipe, or MvPart catalog?
5. Give it a name and description.
6. Check the file path. It should be located near the other content catalogs, but not in the same directory. Example:
   C:\ProgramData\Autodesk\MEP 2011\enu\MEPContent\USI\MvParts
   It's ok to create a custom folder in this MvParts folder. Place your catalog in your custom folder. Do NOT put the catalog direct in the MvParts, but in your own folder here. Put your parts in sub-folders of your folder.
7. This catalog will not show up in any lists yet. We must add a part to the catalog first, and that can be tricky. We cannot use Windows Explorer to copy a part. That creates a duplicate and then every time you launch AutoCAD MEP, you will get an error requesting you regenerate your catalog. There is a right-way to duplicate a part without creating a duplicate. Back in AutoCAD MEP, find the manage tab on the Ribbon.
8. Find the MEP Content group > Content Builder and launch it.
9. In the Content Builder dialog, select the Part Domain. Is it a Cable Tray fitting catalog, Duct fitting, Conduit, Pipe, or MvPart catalog?
10. Select any parametric part. Then on the right, find the third button up from bottom called "Modify Part Size." Pick it.
11. Content Builder's Parametric Part Builder opens. On the left, you see a window pane which looks like a docked palette. At the top of the pane, are some icons. The first icon says save. The second in says "Save As." Click Save As.
12. It may pop up a message asking if you want to keep the part visible or make hidden. Say yes and keep it visible. We don't care anyway because we are only temporarily making a duplicate to get our new catalog to become visible.

13. The next dialog allows us to save the part family as a new part. Give it a new name. Suggestion: Name it “Temporary1” so we can find it easily later.
14. Under name it says part description. Granted this is a temporary part, but I’ll take this opportunity to tell you that this description is critical. This description must be edited. The description is what is seen when you go to add parts. So if you fail to edit this description, then in the add dialog boxes, you will see two parts with the same name. You will not know which is which. Just a good habit to edit the part description. Click ok to exit.
15. In the pane on the left, above the top row of icons is a gray bar. Hover your curser over this bar and you will see a –X appear. Click the X to exit Content Builder.
16. It will ask you to save changes to your new part. Say yes.
17. Shut down AutoCAD MEP.

When we use Content Builder’s “Save As” feature, it creates a new part and assigns that new part a GUID. That stands for “Globally Unique Identifier.” It is a string of 26 randomly generated characters and numbers. No two are alike, or so the theory goes. It is how the computer can tell one part from another and across different files. When we copy > paste a part manually from Windows Explorer or make a duplicate in our catalogs, we now have two parts with the same GUID. It defeats the purpose, the GUID is no longer unique. So when you launch AutoCAD MEP, it sees the duplicate and prompts us to regenerate our catalogs. It’s going to ask you to regenerate until you fix the duplicate. So if you must make a duplicate, don’t make copies from Windows Explorer. Use the “Save As” feature described above, which assigns a new GUID when it generates the new part.

18. Our part we created when we did the “Save As” is in the out of the box catalog next to the original part. We need to use Windows Explorer, so open it.
19. Browse in Windows Explorer to find the new duplicate part files. There will be a drawing file, a bmp and an xml. They will have the same name, just different dwg, bmp or xml extension. To locate the right folder, you can use Windows Explorer search feature to find “Temporary1.dwg.”
20. Once you find all three Temporary1 files together, select them. Right-click and Cut to clipboard.
21. Then browse to our new catalog, just a few folders higher in the folder tree. Create a sub-folder next to our catalog apc file. Call this folder “Temporary1.” Open the folder and Paste from clipboard. All three files should now be inside our new catalog.
22. Launch AutoCAD MEP.
23. At the command line, type “OP” to bring up the Options menu.
24. Switch to the tab “MEP Catalogs.”
25. What Part Domain was your catalog? Expand on the tree for your domain.
26. To the right, click the Add button.
27. A dialog box pops up allowing us to browse to our new catalog apc file. There is a shortcut. Observe the left side of the browse dialog. There are shortcut icons for History, Documents, Favorites, Desktop, etc. The first one at the top says “Content.”

That button takes you straight to the Content folders where these catalogs are stored. Find your apc file. Click the open button.

28. Ok out of Options menu.
29. Exit and relaunch AutoCAD MEP again.
30. This time, it will prompt to regenerate the catalogs of your Part Domain. Allow it to regenerate. Since there is but one part in your new catalog, it will flash by and may not even see it.
31. Now let's test the catalog. If your part domain is Duct, at the command line type in DUCTFITTINGADD. If Pipe, type in PIPEFITTINGADD. Same for Conduit, or CableTray or MvPart. Minimize all trees so we can see the root trees. Your new catalog should be next to the default out of the box catalog in the list. Add our Temporary1 part to model space.
32. You are now free to add new parts. This new catalog will show up when you open Content Builder and set it to that Part Domain. Create new parts in that Catalog. When you create a "New Chapter" inside Content Builder, it generates a sub-folder for you.
33. Once you have your own new parts in your catalog, feel free to delete the Temporary1 folder containing the Temporary1 files.
34. When a new release of AutoCAD MEP comes out, follow steps 22 thru 30 to add your custom catalog into the new release. No extra steps, unless you want to centralize or move those files. Just use Windows Explorer to move them. No copies.

Miscellaneous Thoughts on Line Geometry and Dimensions:

This concept of sequence needs to be on your mind before you layout a single line. It's not just dimensions or the order we declare points to be fixed. Let's say we draw a horizontal line. Then we continue off of one end, we add a vertical line. Let's finish and add a hypotenuse line to complete a right triangle. Let's dimension the length of the hypotenuse line. Let's change the dimension of the hypotenuse line. What do you think will happen? Will the triangle keep it's right-angle or become an obtuse triangle? It will stay as a right triangle. It moves two points, the whole vertical line shifts right allowing the hypotenuse line to lengthen. The vertical line lengthens too. We don't have any fixed points and even fewer set dimensions, so it's fair game to move more than one point, lengthen lines to maintain the shape.

What happens if prior to adding and flexing the dimension, we fix all three points? Then add the dimension to the hypotenuse and flex it? Right, Obtuse or something else? Something else. The dimension value changes but nothing moves or flexes. It's broken and over constrained.

What if we fixed the two bottom points, but not the upper point? Then draw the hypotenuse dimension. Then draw a vertical dimension. Flex the hypotenuse

dimension? We get an obtuse triangle. The two dimensions were declared by the user. The fact that we drew two perpendicular lines, is not a declaration we want them to stay perpendicular. It will try to keep them perpendicular if it can, but when something has to give

What if we draw the three lines? Fix the two bottom points. Add the hypotenuse dimension. Next we manually add a constraint to hold the horizontal and vertical lines perpendicular. Then flex the dimension. What happens? The upper point moves up. It stays as a right-triangle. However, the angle of the hypotenuse line changes as it grows.

Content builder try's to maintain shape. If you have parallel lines or perpendicular lines, it will try to maintain that unless a rule (a dimension or constraint) forces it's hand. Dimensions and manually placed constraints take precedence over auto-generated constraints, regardless of the sequence of creation. Content builder will try to maintain shape unless a rule forces its hand