



Going With the Flow with Inventor® Tube & Pipe

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MD5386

This class will be a presentation of tips, tricks and workarounds for Inventor Tube & Pipe module. A powerful design tool built into the Inventor landscape, Tube & Pipe is the bread & butter of many designers. We will explore some basics of getting started, as well as some secrets and workarounds for successfully routing and populating pipe runs in this environment. The class will focus on rigid piping, but tips will be applicable to flexible hose and tubing design as well.

Learning Objectives

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

- Overview of Authoring & Publishing Tube & Pipe components.
- Basics of Tube & Pipe styles and template.
- Constraining pipe routes to surrounding geometry.
- Productivity tips & tricks.

About the Speaker

Chris is the CAD Department Supervisor at Powell Fabrication & Manufacturing, Inc. in St. Louis MI. He has been working as a mechanical designer and drafter for the last 20 plus years, using Autodesk Products for the majority of that time. For the last several years he has been the Lead CAD Technician and Vault Administrator at Powell Fab, integrating it with Autodesk Inventor, Autocad and Autocad Electrical. He has been implementing changes to the company's document management procedures using Vault Professional as the main system for lifecycle and revision control, and Engineering Change Orders to control all drawing changes. Recently he was promoted to the position of CAD Supervisor. 2 years ago Chris was recognized as an Autodesk Expert Elite for his activity on the Autodesk Discussion Forums, social media and for his blog Cad Tips, Tricks & Workarounds. CADNotes.com and Design & Motion.net have also published some of Chris' articles.



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Blog: [CAD Tips, Tricks & Workarounds](#)



Let's dive right in,... shall we?

1. Overview of Authoring & Publishing Tube & Pipe Components.

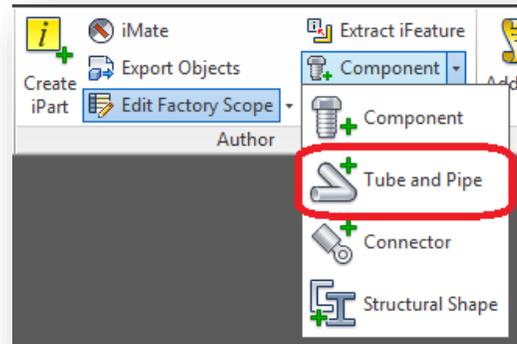
In this section we will discuss how to author your own custom Tube & pipe fittings and conduit, to be published to your Content Center for use with Inventor® Tube & Pipe.

Authoring Tube & Pipe Fittings

Authoring is the method of preparing your parts for use with Inventor Tube & Pipe. Parts and iParts need to be authored to determine the connection points, end treatments, size etc. that Tube & Pipe will use to create and populate your Pipe Runs. Additional information on how to author Tube & Pipe fittings can be found in the Inventor® 2015 Help files at: [Tube & Pipe Authoring](#)

Authoring Standard Fittings.....

Users can author their own custom pipe fittings from an Inventor iPart, or from individual part files. The authoring tool can be found in Inventor by navigating to the Manage tab on the ribbon, and the Author panel. By default the Component authoring tool is present on the ribbon, but several other tools can be found by clicking on the down arrow next to the tool.



When authoring Tube & Pipe components select this tool and enter the necessary information into the dialog box that opens. The information needed will vary by the type of the fitting being authored, but will include:

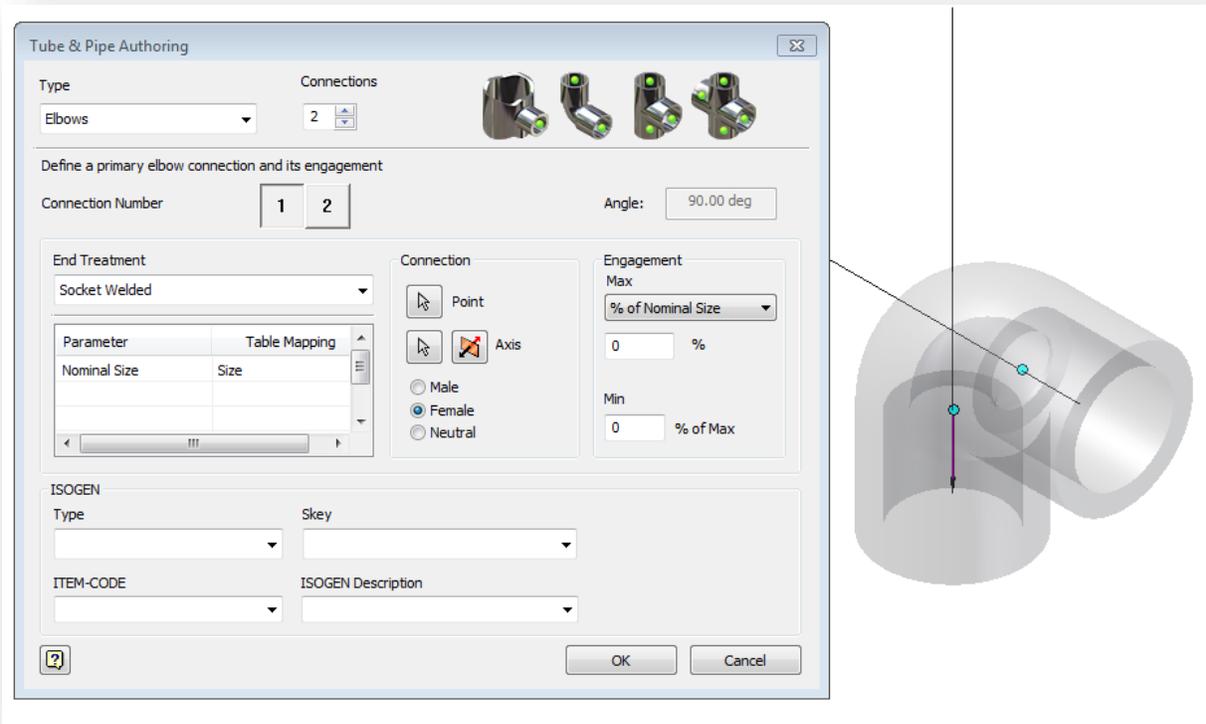
- Part Type
- Number of Connections
- End Treatments per Connection
- Parameter Mappings From the iPart table
- Connection Point & Axis
- Gender & Engagement of Each Connection



Tip: If you are authoring a fitting that either does not fit into one of the types listed, or which you want to publish to a custom Content Center folder, choose *Other* from the “Type” pull down. This allows you to specify where it goes when the part is published.



Tip: Only part files may be authored for Tube & Pipe. Assemblies must first be derived to a part before they can be authored.



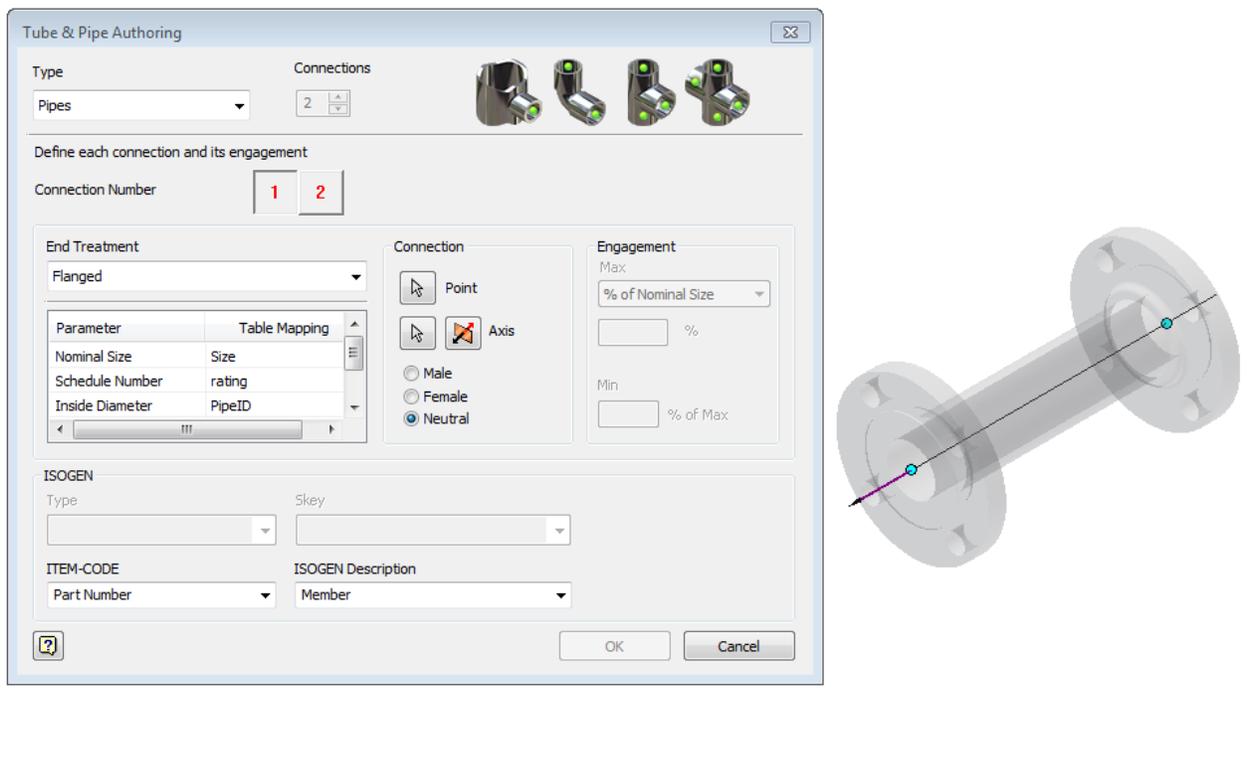
Engagements can be used to specify the depth at which a fitting is connected to a pipe conduit in your routes; for example, to specify thread engagement on threaded pipe fittings. The gap between pipe and fittings in welded systems is determined in the Tube & Pipe styles and will be covered in a later section.

Authoring pipe conduit...

Authoring Pipe conduit is done in essentially the same way as authoring fittings, but can only be done on iParts, not standard Inventor parts. The iPart being authored requires a bit of preparation first. The iPart table must contain the following parameters for Tube & Pipe to recognize it as a conduit part:

- Nominal Size
- Schedule Number
- Inside Diameter
- Outside Diameter
- Pipe Length

In the image below, these parameters will be required in the parameter mapping section in the middle of the dialog box. Pipe length will only be required on the first connection point of the conduit.



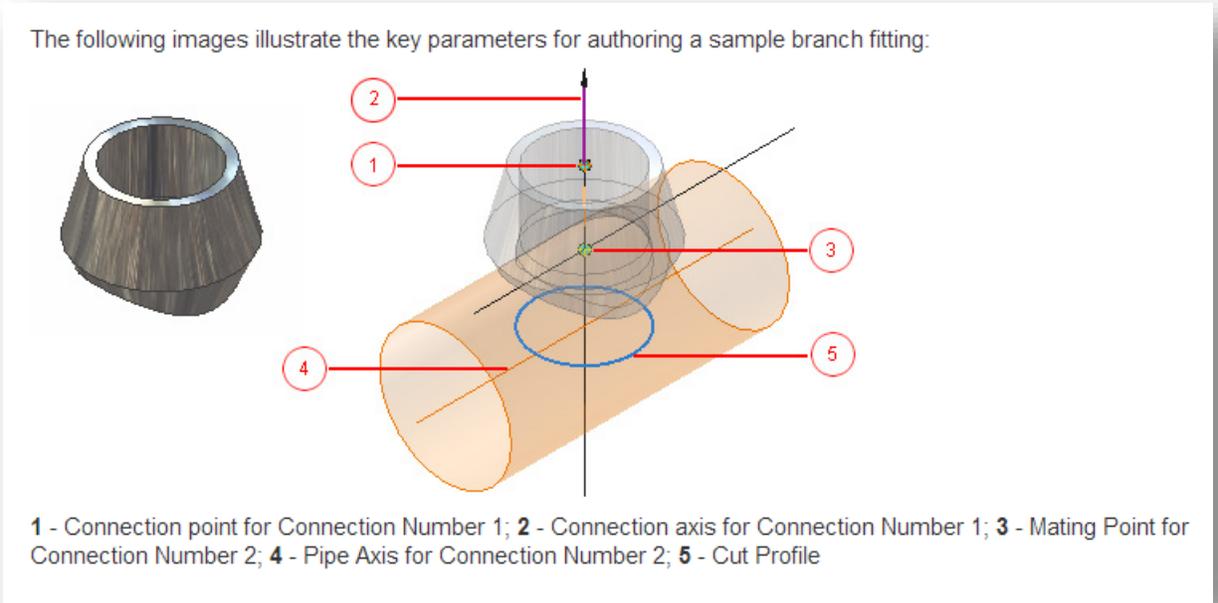
Tip: Notice in the image above that the pipe spool is a flanged design. As long as all parameters are present the pipe can be whatever shape is needed.

Authoring branch fittings....

Branch fittings, such as weld-olet and thread-olet type fittings also have a specific set of parameters that are needed, as well as some part features that must be present in order to correctly mate the fitting to the conduit and cut a hole at the branch intersection. Needed parameters are:

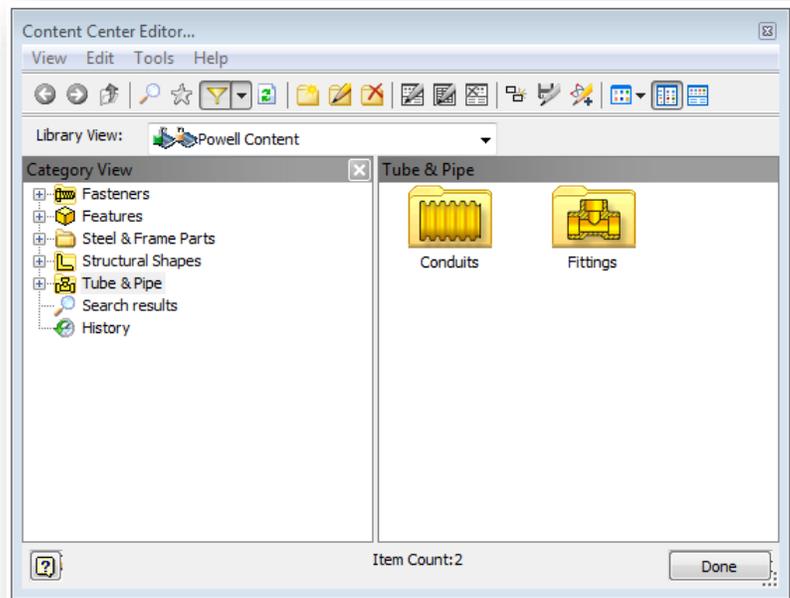
- Min Nominal Size – The pipe size of the branch fitting.
- Max Nominal Size – The pipe size of the conduit being connected to.

Also required on the iPart are a work axis at the run diameter of the mating pipe, and a sketch for the cutout of the branch fitting, also on the same axis, as shown below. This image is taken from the Inventor 2015 Help files on authoring branch fittings.



Publishing Fittings to the Content Center

Once your fittings are authored, they can be published to the Content Center for easy access. Tube & Pipe allows you to select any authored fitting for insertion in a pipe run, even if it is not a Content Center part. The benefit of the Content Center is that it allows for easy sorting and access to your components while routing pipe runs. The Content Center is broken up into categories and families. Categories are major groupings of components such as Fasteners, Structural Shapes and Tube & Pipe. These categories are divided into more specific categories, for example in Tube & Pipe; Elbows, Tees, Flanges etc. These then contain individual families, which may be iParts or individual parts that have been published. Content Center allows for the creation of as many custom categories as you need to organize your custom content in the way that works best for you.



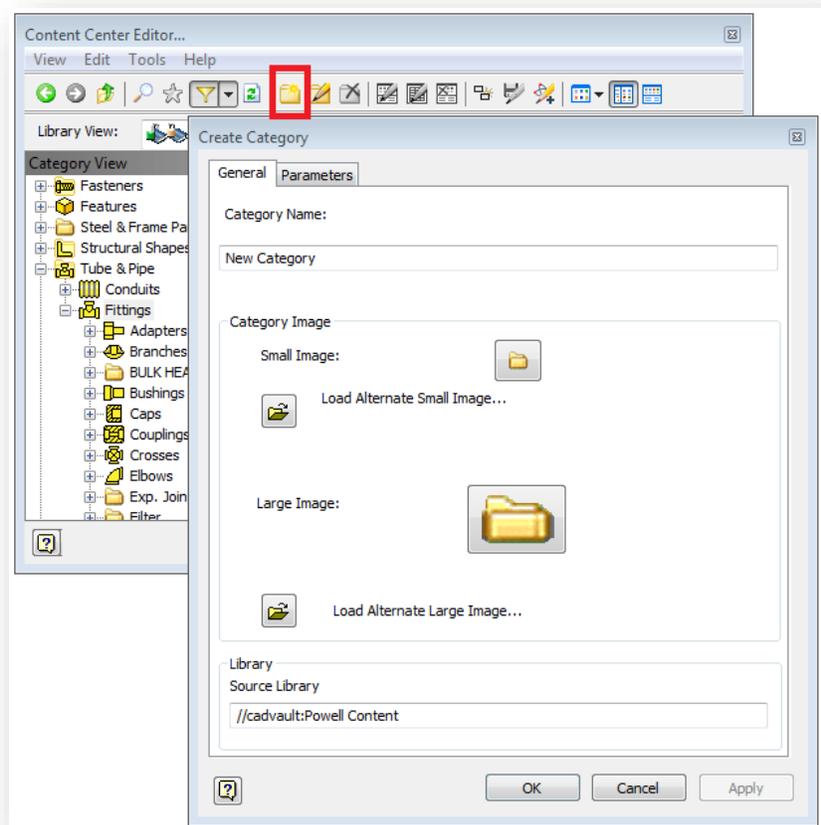
Before parts can be published to the Content Center, you must first make certain that you have a Read/Write library; whether you are using Vault or local content. For more information on creating libraries, see [this section](#) in the Inventor 2015 Help files, or [this section](#) in the Vault 2015 help files.

Note that libraries cannot be added during the publish process. Nor can categories be added to the library during the process. Any custom categories you wish to publish to must be created prior to publishing the parts.

Creating Custom Families in the Content Center.....

Creating a custom category in the Content Center is fairly easy, once you are certain that you have access to a read/write library. The Content Center editor is found on the Manage tab in Inventor, in the Content Center panel. From the editor, select the button for Create Category and fill in the necessary fields.

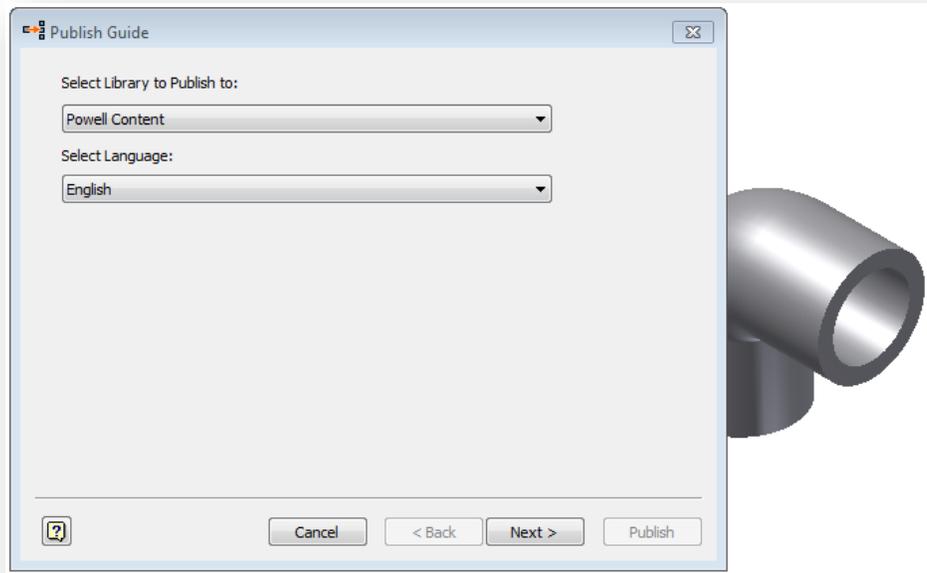
Assign a unique category name, and either accept the default folder images or, if you have an image stored on your computer or network, you may select an alternate. The source library specifies to which library, if you have more than one, the category will be saved to.



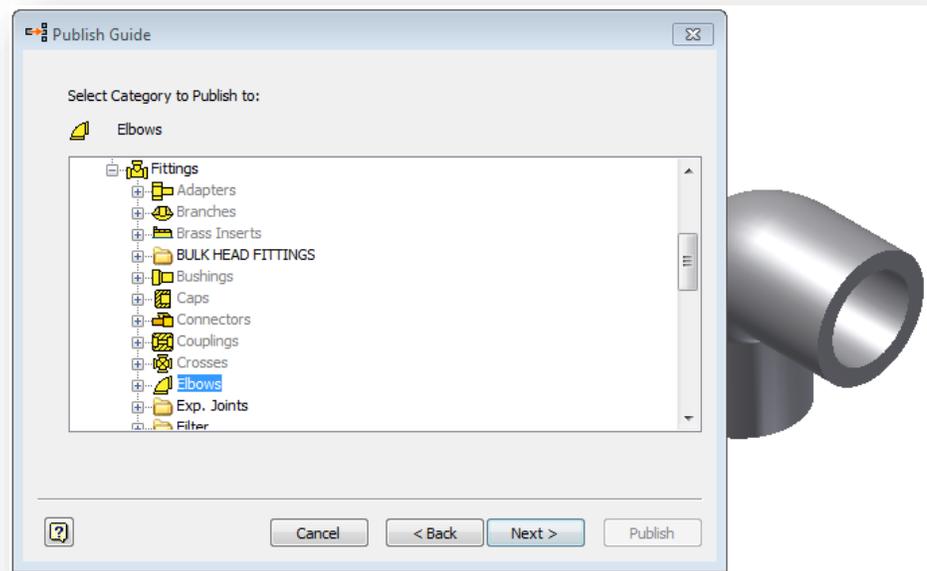
Publish the components....

Once the library and categories are prepared, it's time to publish your custom fittings. Publish is found near the Content Center editor button on the Manage tab in Inventor. There are several screens to work through to ensure all of the proper information is entered.

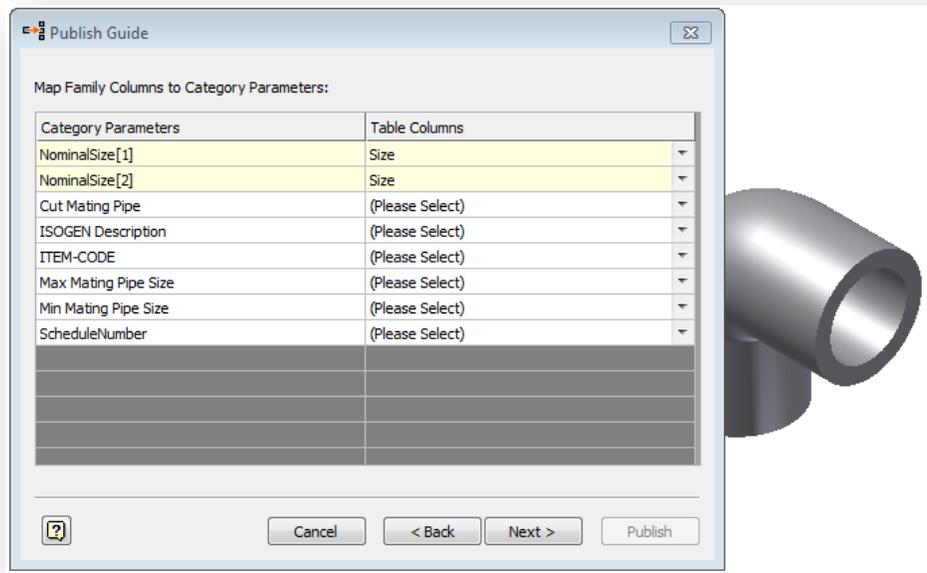
The first screen asks you to specify a library to publish the part to. If you have more than one, you can select from a pull down. Next it asks for a language to publish the part in.



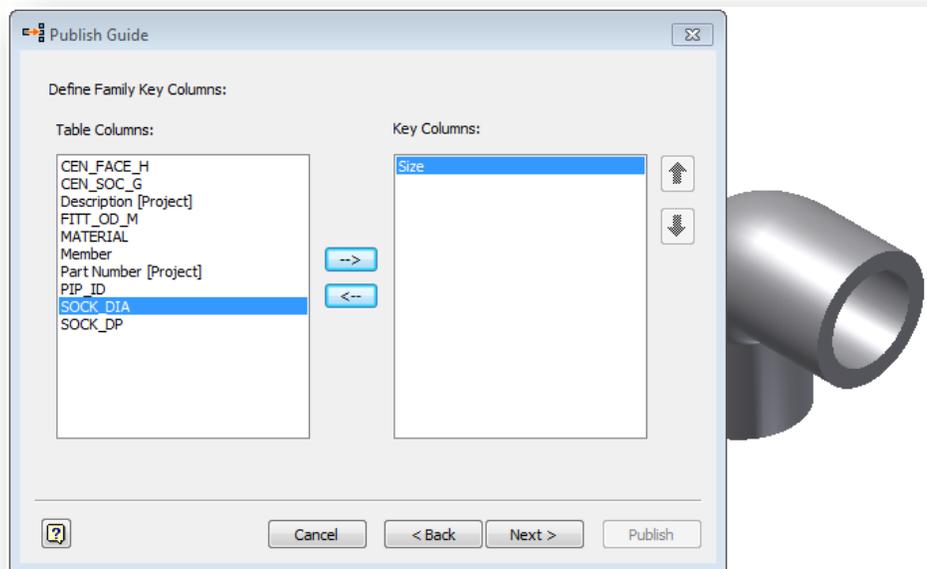
The second screen asks you to accept the default category, if a default was used during the authoring stage. If you selected “other” during authoring, this is where you would specify which category to publish the part to.



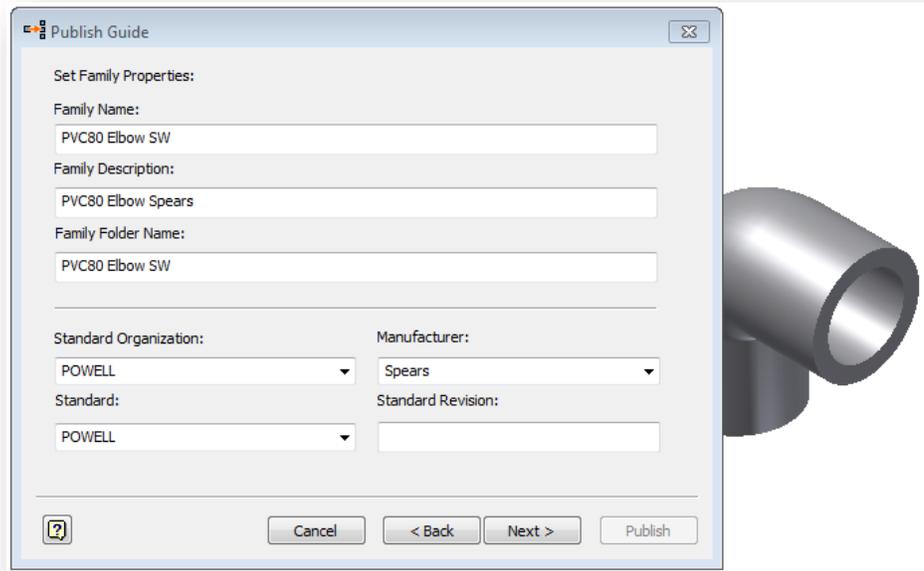
Next is a screen for mapping parameters. Most of these may be done for you already, as these would have been set during authoring. Required parameters will be highlighted.



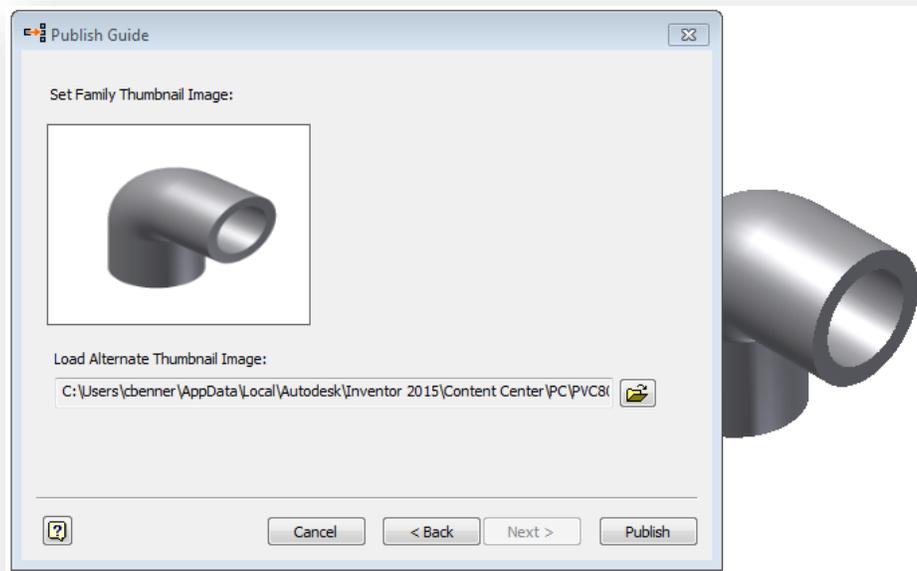
The next screen asks you to define key columns from the list of family table columns. You may have as many key columns as are needed to define a specific part from an iPart table that may contain many choices such as material, pressure ratings, size etc. When a published part is placed in Inventor, you will be presented with a screen allowing you to choose from among these keys to get the specific part you want.



Next is a screen asking you to assign a family name. This must be unique. The family description may be whatever you want, but family folder name will copy the family name. The Standard and Standard Organization fields **MUST** be filled in if you plan to use the part in the definition of a Tube & Pipe style, as these are used for filtering the fittings used in these styles. Custom standards and Organizations can be used here, as well as the defaults in the pull down list.



The final screen gives you the option of accepting the information entered and publishing the part, or going back and making changes.



Working With Family Table Columns.....

Once a custom component has been published to the Content Center, you should look at the structure and content of the family table in the Content Center editor. Depending on how the part was authored and published, columns may not contain exactly the data that you would like. There is also the opportunity to map family table columns to iproperties on the Inventor part when it is generated. This can be essential for Bills of Materials or for Vault properties. [This page](#) in the Inventor 2015 Help section gives more information and terminology for editing Content Center family table columns. We will not dive too deep into this topic as the Content Center is a subject all its own.

Copying Standard CC Families to Custom.....

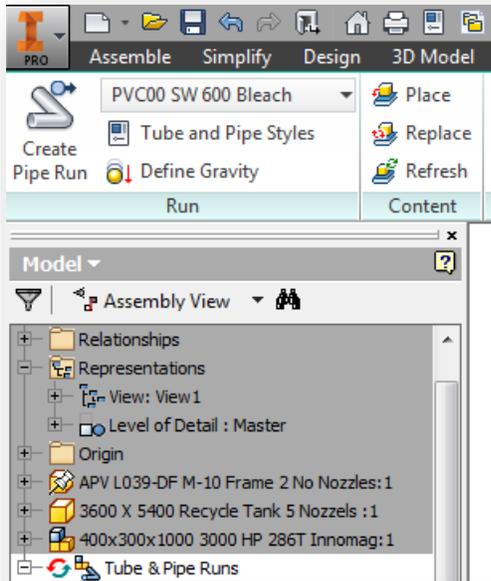
The Content Center out of the box contains over 750,000 parts and covers 18 international standards. That said, it likely does not include everything you could ever need for your designs, nor does it contain all of the property information you may need for your Bills of Materials. Standard Content Center families [can be copied to a custom library](#), either with or without a link to the original family. Once the family has been added to your library you can change whatever is needed in the family table, such as part numbers or descriptions, to customize the part to your needs. This is also a good way to copy standard parts where the geometry does not change, but material may be different. For example, PVC fittings copied to a new family and changed to CPVC. The fittings are identical other than the material.

2. Basics of Tube & Pipe Styles and Template.**Tube & Pipe styles.*****Piping Runs.iam.....***

Before you can begin creating pipe runs, there are a few more items that need to be set up. Foremost among these are your Tube & Pipe styles. As you know, Inventor uses template files to create new Part, Assembly or Drawing files. Tube & Pipe is no different, and has its own template file. This is not as obvious, and unless you have read every page of the help files, it may be easily overlooked. The template file used by Inventor Tube & Pipe is called ***Piping Runs.iam***. It is installed with Inventor in the Design Data folder, inside of a sub-folder called Tube & Pipe. In this folder there are also sub-folders containing English unit and Metric unit versions of the file. Any new Tube & Pipe styles should be created in this file and saved there. When you create your first pipe run and enter the Tube & Pipe environment, this file is used to create the Master Run Assembly. We will talk more about this in a later section.

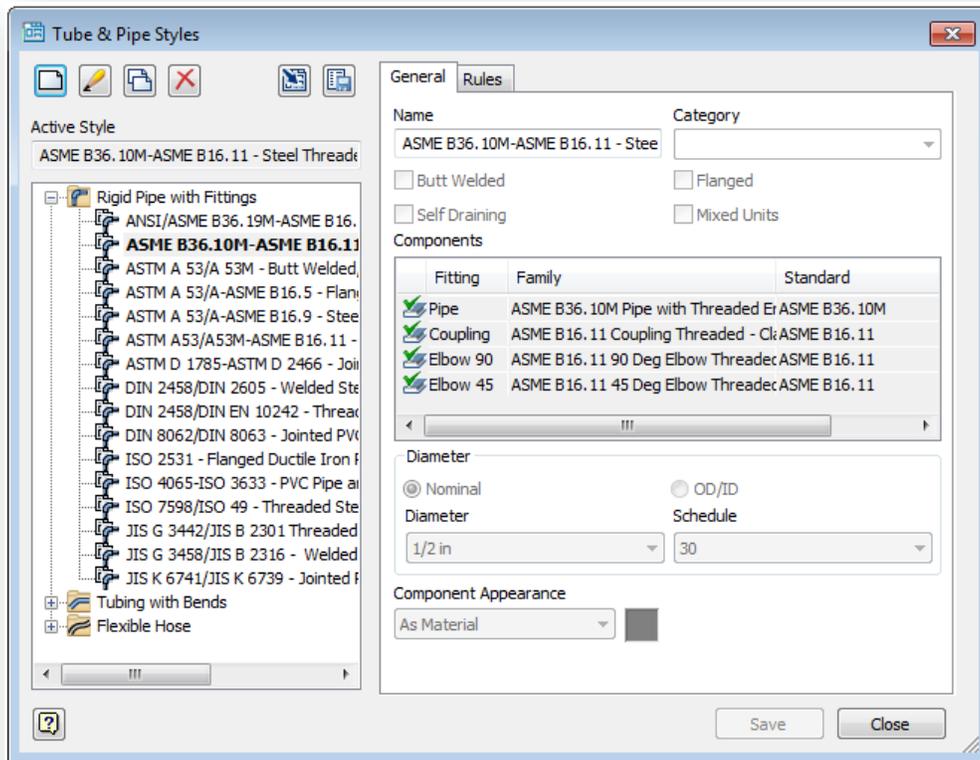
Tube & Pipe Styles.....

All tube & pipe runs are created using Tube & Pipe styles. These are essentially a set of rules and fitting mappings that Inventor uses to populate the routes you lay out in your designs.



There are several base styles available out of the box, and you may create as many of your own as are needed. The styles define nominal size, pipe conduit and essential fittings, end treatments, and rules such as minimum and maximum spool length, and increment. There should be a style created for each combination of these items, and for each size.

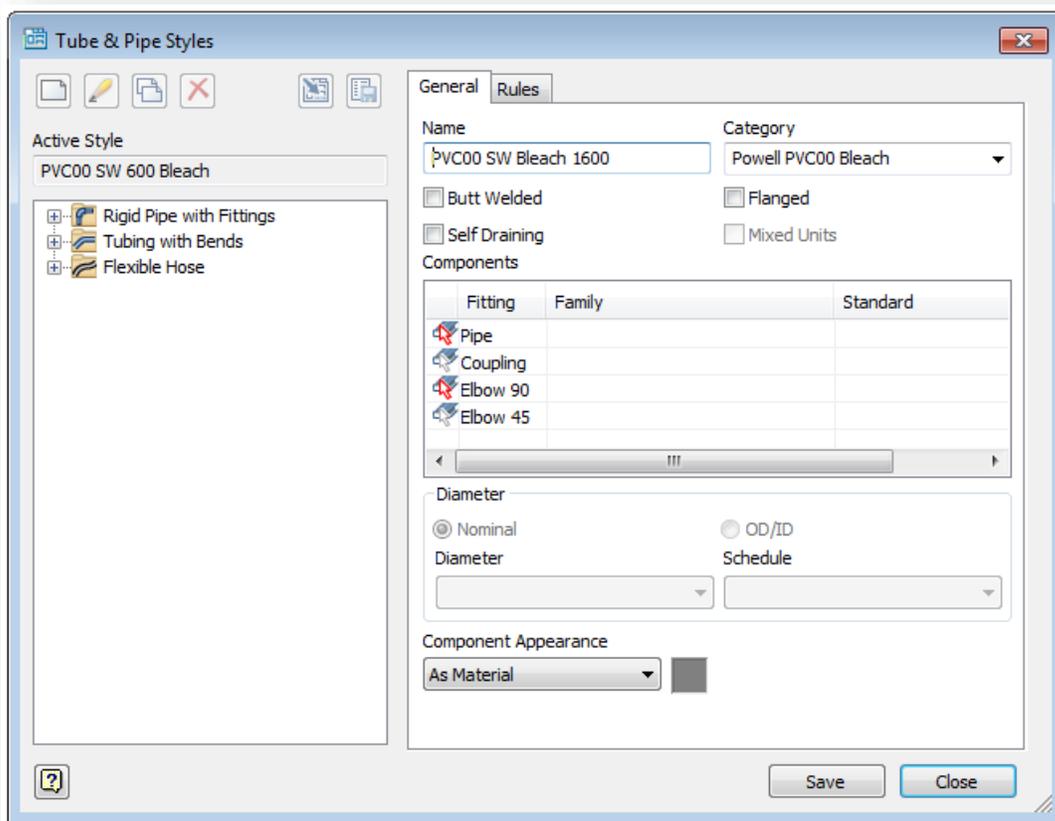
Think of these as your individual pipe specs, which may specify the same things. Create these styles in the tube & pipe template file, as mentioned above, unless you need them to only be available in the current design. Tube & Pipe styles are accessed on the Manage tab when inside of an individual pipe run, or on the Tube & Pipe tab when at the Master Run Assembly level.



The toolbar across the top gives you the ability to create and edit your own custom styles. When creating a new style, select New to start fresh, or Copy to use an existing style as a starting point. Creating new categories within the browser is a simple matter of giving your new style a unique category name. Once that style is saved the new category is also created, and more styles may be added to it.



Once you have selected New or Copy, the new style will be created under whatever category your currently active style is in. The active style is distinguished by its name being in **bold** text. If you use the Copy command, the new style will appear directly below the style you copied from, and will include **_Copy (1)** at the end of the name. You can simply edit that out and name the style whatever you want.



Your newly created style opens in a dialog box similar to the one above, waiting for you to fill in the information. Give the Style a name and a Category name, or accept the current category name. Categories may be created for things such as different materials, pressures, fluid codes... however you want to differentiate your pipe styles. Each style within a category may be

named however you would like; suggestions would be to include the size, material or spec, and perhaps end treatment. But this is up to you or your company standards.

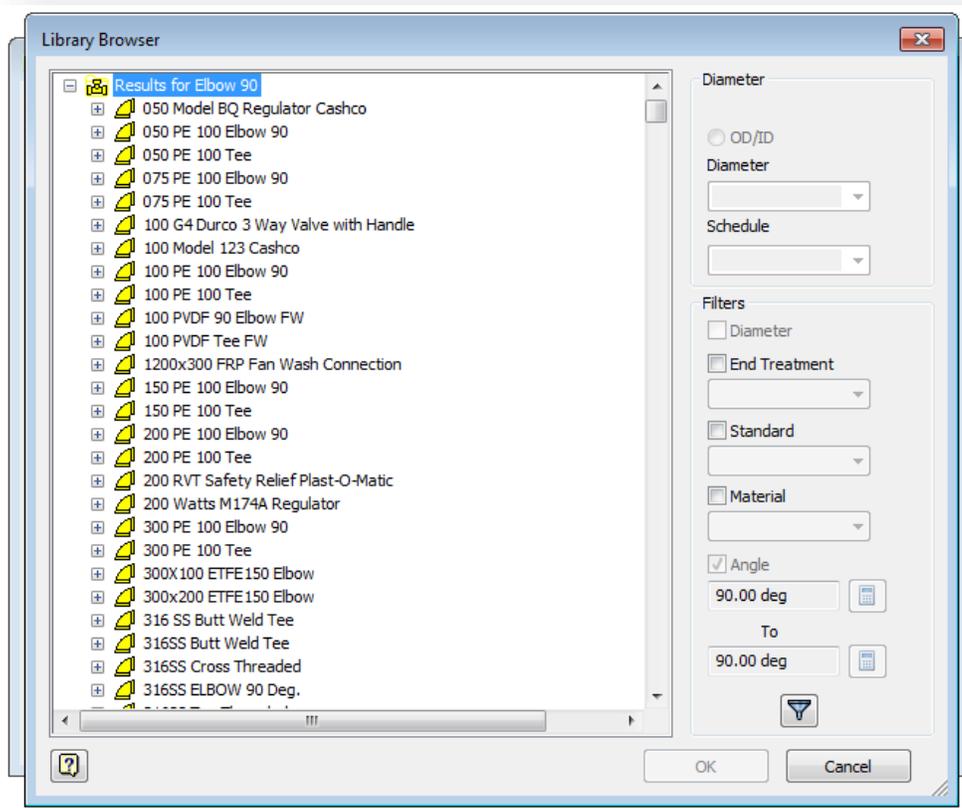
Once you have a style and category name, you need to specify the base fittings for this style. Decide whether this is going to be a Flanged or Butt Weld style. I will not be talking about Self Draining or Mixed Unit styles, since (frankly) I know very little about them.



Tip: If your style is going to include or allow any flanged valves, fittings or instruments check the Flanged box. This will make it so that the gaskets and mating flanges are inserted automatically anytime you place a fitting or instrument that has flanges on it. Butt Weld allows for another setting which I will get to in a while.

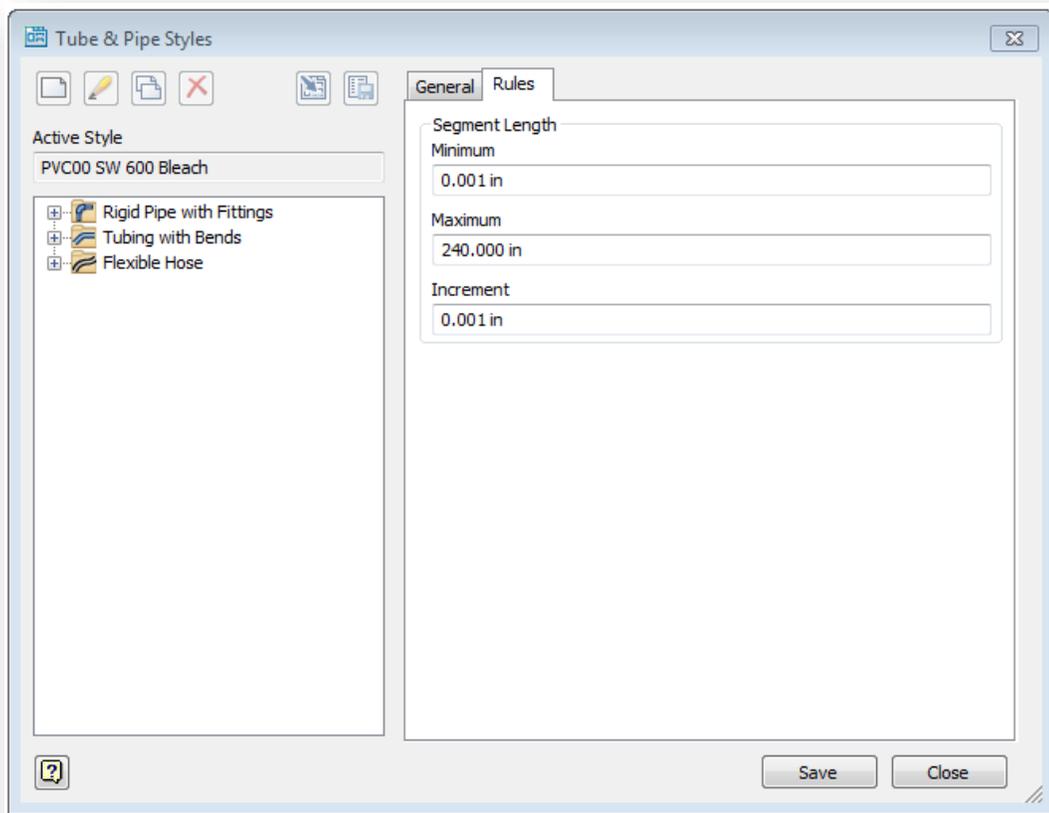
You must specify fittings to be automatically placed in this style, based on which boxes you have checked above. If you have checked nothing, you will need to specify: Pipe, Coupling, Elbow 90 and Elbow 45. For Flanged: Pipe, Elbow 90, Elbow 45, Flange and Gasket. If only Butt Weld is checked: Pipe, Elbow 90 and Elbow 45. This assumes that you have these fittings defined and in your Content Center already. To create, Author and Publish these please see Autodesk® Inventor's Help documentation. Perhaps I'll write up a walk through for these next.

When you double click inside one of the empty fields under the fittings, a Library Browser opens which has already narrowed your choices to the type of fitting you are adding. You can further narrow the results by adding filters and selecting the filter button at the bottom (looks like a funnel).



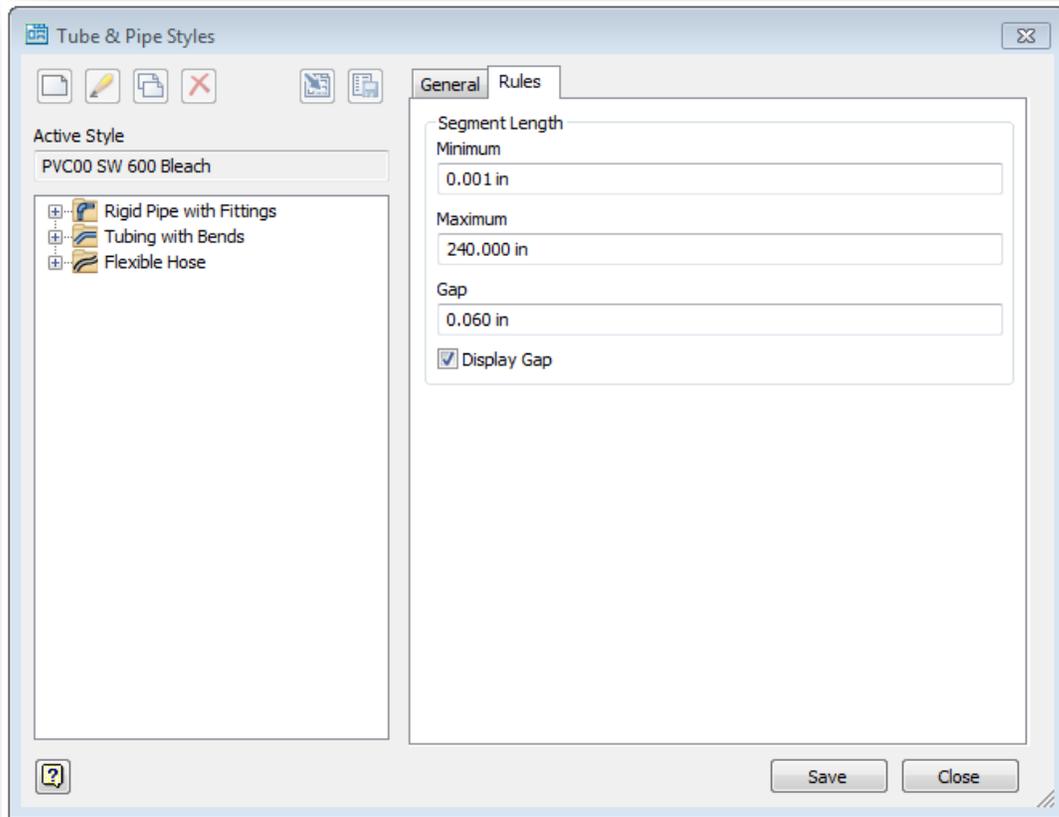
Once you have selected the fittings for each line required by your style, you're halfway there. You must also decide whether to set the size by Nominal Diameter & Schedule (as in rigid pipe), or by specifying OD & ID. You may also choose a color other than the default for this pipe style. Note that colors will only apply to the pipe and fittings generated by your routes. Any fittings added in line later will come in as the model color.

Now that this section of the style is filled in, select the Rules tab to complete the final steps. You must specify a Minimum, Maximum and Increment length for your pipe spools. These can be whatever fits your specs.



 **Tip:** For Minimum I almost always go with .001". This allows me to create extremely small gaps between fittings if I need to. With the piping I run, I sometimes need to have 2 small branch fittings in the same spot 180 degrees apart. This setting allows me to have 2 route nodes that are .001" apart, which rounds out to nothing on a fabrication drawing. You will know what is going to work best for your situation. Maximum is usually dictated by the longest length of tube or pipe that you can buy. In my case this is 240" or 20'. Increment specifies the increment by which your pipe lengths can grow. For example, you can specify that you always want your pipe lengths to be within the nearest 1/2". For me, I again set this to .001 so my pipes can be whatever length they need based on equipment layouts.

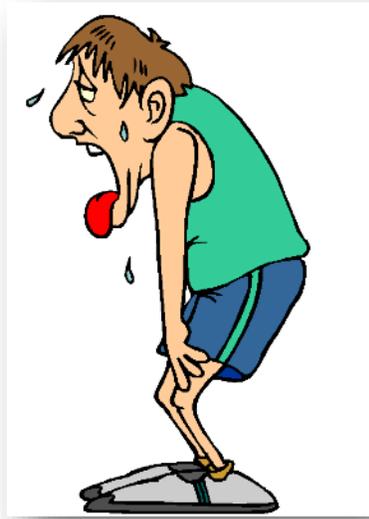
If you selected a Butt Weld style on the general Tab, the Rules page will look like this:



Here you can specify a gap between welded pipe and fittings, based on your welding specs. This will ensure proper cut lengths in your BOM.

When you have set everything in your style to what you want or need, hit Save and your new style will be created. If you have given it a new Category name, the new category will also be created. Now that you have one style done for this new category, you can copy it as many times as needed for the various sizes, and simply change the size setting and name as needed. In this way you can quickly create an entire family of tube & pipe styles.

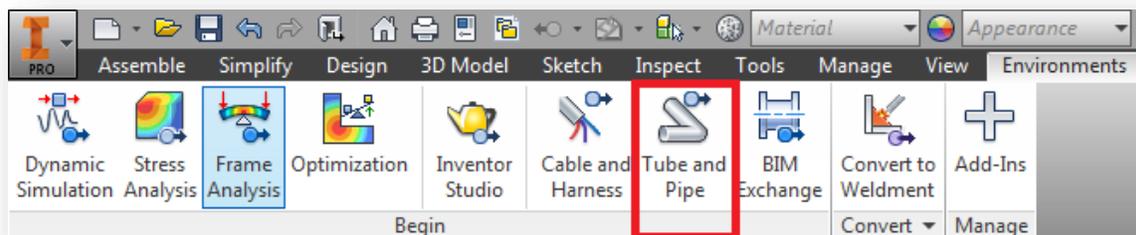
The Export & Import buttons on the tool bar can be used for just what it implies. Tube and pipe styles, or categories can be exported to a .xml file. If you feel comfortable with doing so these can be edited outside of Inventor and then imported back in with the changes. This can be used to quickly copy a category, save it for a different fluid code and change the color in the .xml file (using find & replace). I don't recommend this for the faint hearted or those not at least somewhat familiar with .xml files. This Export/Import feature also allows you to copy styles between machines that do not share a common server. Simply export them and put onto some portable media, and Import onto the machine that needs them.



3. Constraining Pipe Routes to Surrounding Geometry.

Entering the Tube & Pipe Environment.

Now that you have authored and published fitting libraries, and created Tube & Pipe styles, it is time to design some piping. In Inventor Tube & Pipe, your piping designs will be broken down into a Master Run Assembly, followed by individual Pipe Runs. Each Run may contain as many Routes as are needed to define the system. To use the Tube & Pipe tools, you must be inside an assembly file. From the Environments tab on the ribbon, select the Tube & Pipe tool.



The first time you use Tube & Pipe in any assembly, it needs to create the Master Run Assembly, called by default *Tube & Pipe Runs.iam*. This file is created using the template file *Piping Runs.iam* that we discussed in the previous section. A dialog box will open, prompting you for a name and location of both the Master Run and your first Pipe Run. You can accept the defaults, or rename both to suit your needs, as well as relocate them to a different path.



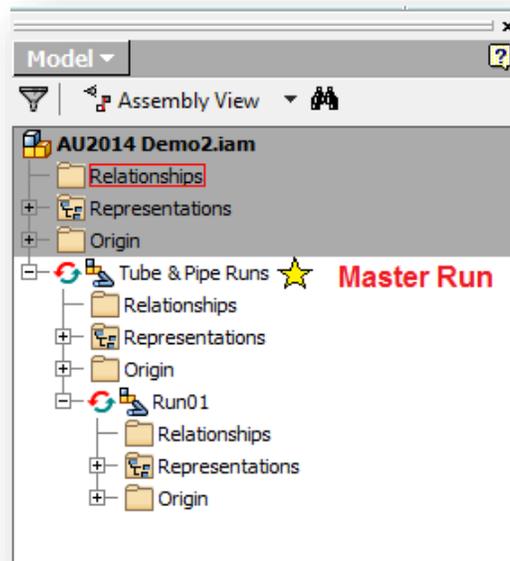
Tip: The Run name defaults to Run01, 02 etc. The location for Run File path also contains this name. If you change the Run File name, the location name will not update unless you click in the location box shown below.

Tube & Pipe Settings.....

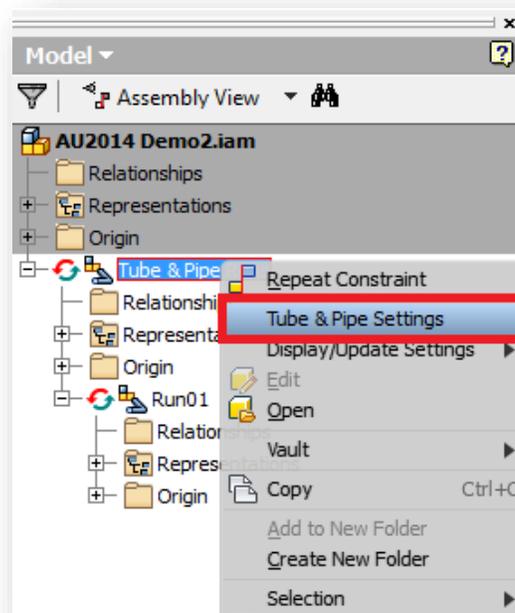


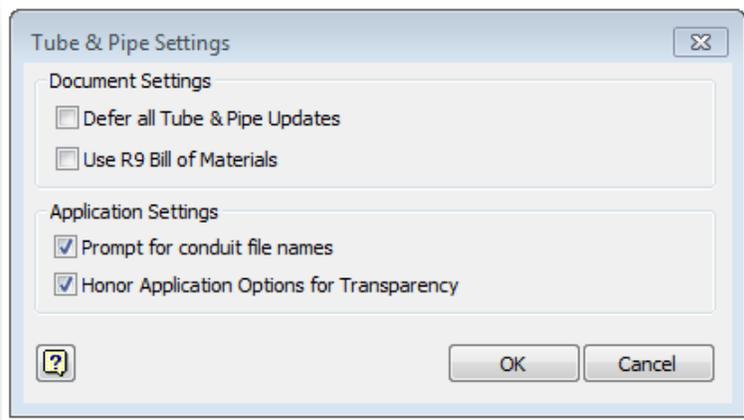
Tip: This section will be started off with a tip, one that I believe is fairly important. Tube & Pipe runs are generally, but not always, designed to be connected on at least one end to a piece of equipment. Some are connected on both ends. This may make it appear that the pipe run is fully constrained, but in reality it is not. It is possible, at either the Master Run level, or the parent assembly level, to inadvertently grab and drag a pipe run to a new location, which can severely damage any routes you have completed. To avoid this, here is what you need to do. Once the Master Run and your first pipe run have been created, return to the Top Level assembly. Place Flush Constraints between the Top level assembly and the Master Run assembly, on all three Origin Planes. This essentially grounds the Master Run assembly to its parent. Then, for each pipe run you create, follow the same procedure at the Master Run assembly level. Constrain each pipe run to the Master Run, origin plane to origin plane. This will prevent you from being able to drag a pipe run off of its connection points.

Note: Do not follow this for pipe runs that do not have connections to equipment, as it will prevent you from being able to use constraints to position it as needed.



The Tube & Pipe Master Run assembly contains a couple of settings you need to know about. Once you have created the Master Run and your first pipe run, return to the Master Run level of your Assembly, and right click on Tube & Pipe Runs. Select Tube & Pipe Settings.





Defer all Tube & Pipe Updates will do just what it says. Changes made to the assembly which have an effect on Tube & Pipe runs, will not update the Tube & Pipe runs until you use the Update tool on the Manage tab. This can save some time in regenerating the assembly when changes are made, but it can also cause some confusion from not being able to see pipe run updates right away.

Use R9 Bill of Materials will force all assemblies to use the Inventor Release 9 format for BOM's.

Under the Application Settings are the two that may be the most important settings.

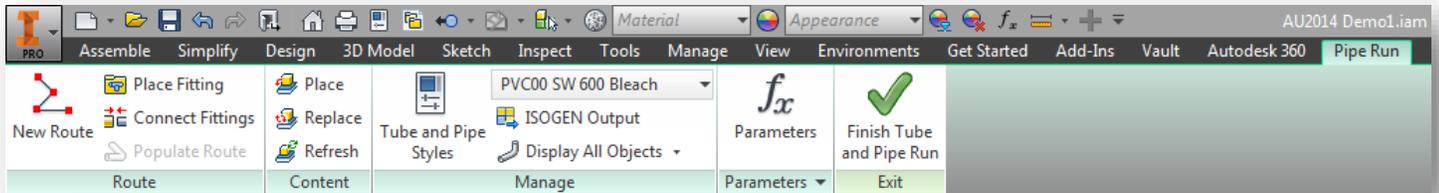
Prompt for conduit file names will force Inventor, on a save or update, to ask you for a file name for any pipe conduit spools being created. By default, Tube & Pipe will generate its own unique file names for spools, with no input needed from you. Check this if you want to have more control over how your spool files are named.



Tip: Checking this box may come in handy if you plan to use your designs over for other projects. By adding some common text to your spool file names, such as a project or job number, it makes it much easier to use a “find & replace” function in Copy Design or Design Assistant when copying a design later to a new project.

Honor Application Options for Transparency. This option forces Tube & Pipe to use the same transparency options that you have set for all assemblies in your Application Options. Prior to Inventor 2015, Tube & Pipe sub-assemblies did not follow transparency settings. When editing a Tube & Pipe run, the entire model would become opaque. Thanks to the relentless suggesting of your instructor, however, this option was finally added in 2015. This gives the user the choice to have all Tube & Pipe runs remain opaque, or only the currently active one.

When you create or edit a pipe run in Inventor Tube & Pipe, you are taken to the Pipe Run tab on the ribbon. This tab only exists when you are in the piping environment and contains special tools needed for routing pipe and adding fittings.



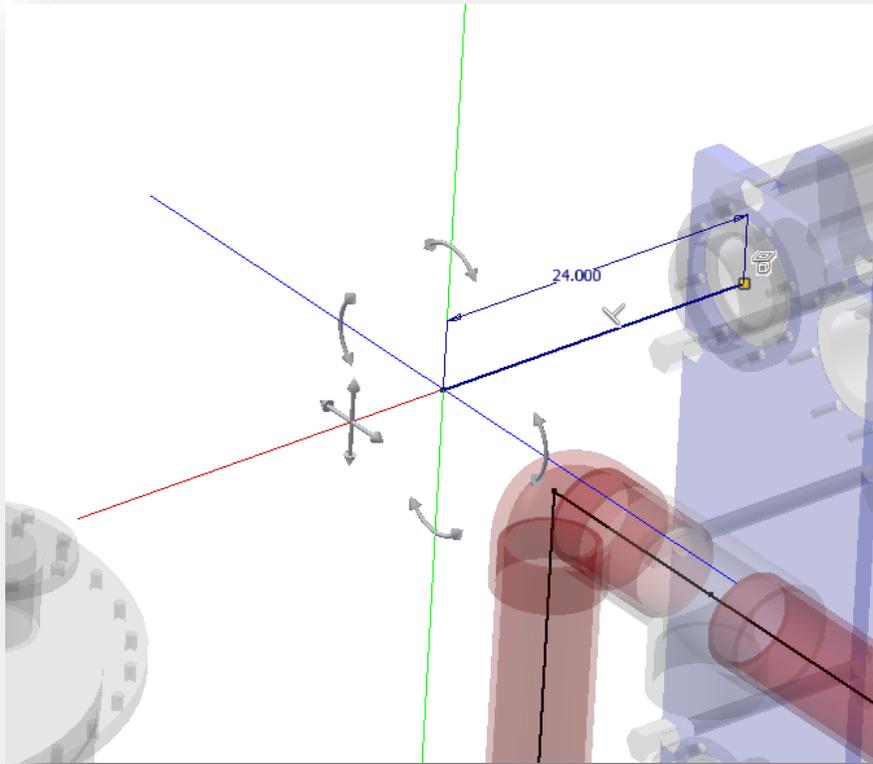
From this tab you can begin a new route within a run, place fittings from either the Content Center or from your file storage, create and modify Tube & Pipe styles, add or manage parameters, and select the current Tube & Pipe style. The Place tool under the Content panel will allow you to select fittings from your Content Center library to be placed into your pipe run. The Place Fitting tool under the Route panel allows you to navigate to a specific fitting that is in your file structure. These fittings must be authored, but do not need to be published to be used.

Route Basics

A newly created pipe run will probably need a route or two. Some people begin by placing a few fittings at the ends of their run, perhaps where the run connects to a piece of equipment. But the basics of routing need to be covered, I think, before anything. A pipe route is essentially a 3D Sketch. It follows the same rules as 3D sketches, and needs to be constrained as such. We're going to cover just a few of the basics and some routing tips. For a more detailed overview of constructing pipe routes in Inventor, see this [Help section](#).

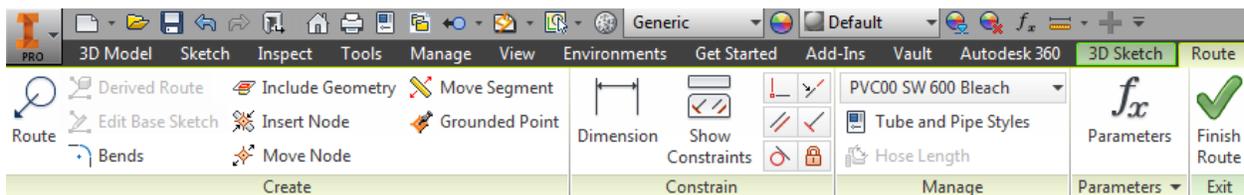
When you begin a new route, using the New Route tool on the above ribbon, you are prompted to begin selecting geometry for your route. Begin by selecting a point on a piece of equipment or on a previously placed fitting. After your first selection the [3D Orthogonal Route Tool](#) becomes active. This gives you several tools to use to determine the direction, angle, rotation etc. for your next route point. Right clicking on the curved arrows gives you the ability to rotate that point of your route to any specified angle. Right clicking along any of the axes will give you the ability to specify a length in that direction (You don't need to right click on any of the line directions, just stop your mouse over a line or geometry and start typing a dimension). The (4) arrows at the end of the tool will bend your route in that direction, 45° (the angle arrows will show up only if you have specified a 45° elbow in your style). You can increase the referenced geometry line or orthogonal tool under your mouse pointer by pressing "+" or "-" numeric keys. Using these tools you can quickly lay out a route between fittings or other equipment in your layouts.

Auto Routes are created by Tube & Pipe when valid geometry is selected at the end points of a proposed route. Inventor Tube & Pipe will generate the segments and points needed to complete a route between the selected points. Since there may be more than one valid solution, you can toggle through them using the familiar “Select Other” tool, accepting the route that best fits your design intent.



Orthogonal Route Tool

When you begin a route, the ribbon changes to a route menu, which looks similar to a 2D sketch menu. Some familiar tools exist on this menu, including Dimensions, Constraints and Include Geometry. These three tools will give you the flexibility to create a route that will grow, shrink, stretch and update as needed as your design changes.

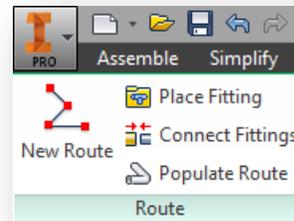




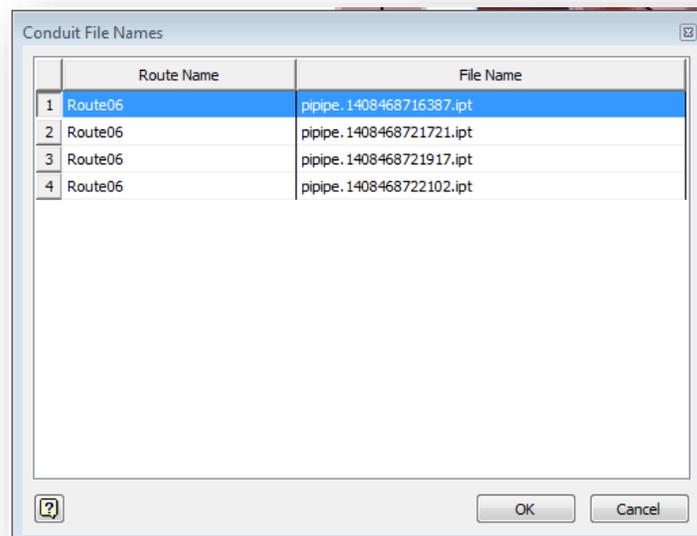
Tip: Do not over constrain or over dimension your routes. As you are routing, Inventor will assume constraints between segments. These may not be what you intend, or may not update as things change. Use the Include Geometry to bring in work planes or edges, and use coincident, collinear or perpendicular constraints to lock your sketch to these references. Always leave as many segments as possible un-dimensioned, relying instead on these geometric constraints to lock it down (for example do a perpendicular constraint between every other route segment, or add collinear and parallel between them). This will allow your route to easily update if the equipment it is attached to, or an adjacent route need to change.

Treating a pipe route as a 3D sketch and using strategically placed dimensions, work features and constraints, you should be able to create strong flexible routes that will easily adjust to changes in your design intent.

Once the route is completed, it is time to populate it with pipe conduit and fittings. Select Populate Route from the Route panel of the Pipe Run tab on the ribbon. The route will be populated by the fittings and conduit you specified in the Tube & Pipe style used for that route.

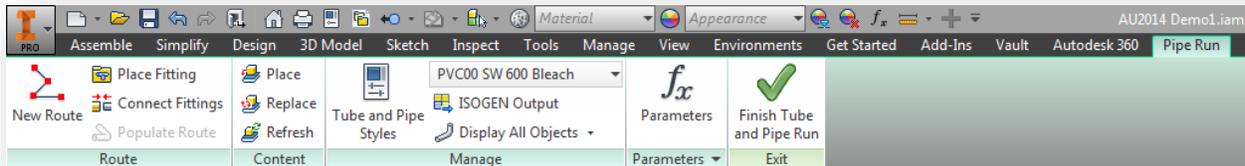


If you specified in your Tube & Pipe Settings that you want to prompt for conduit file names, this is where you will be given a dialog box containing all new pipe segments being created as a result of populating this route. You can either accept the default file names given or change them as appropriate.

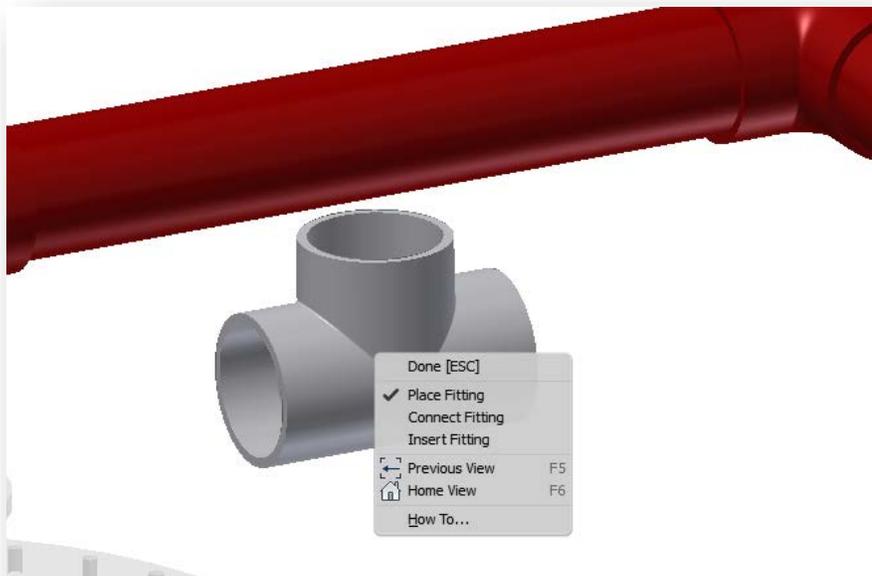


Placing Pipe Fittings.....

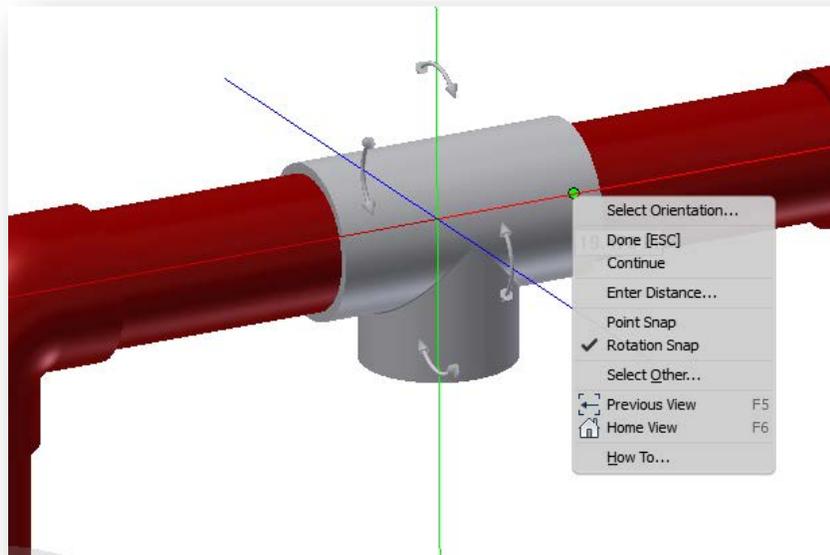
We've now got a route created and populated with the base fittings specified in the Tube & Pipe style. It's time to add any other fittings, valves or instruments that are needed to complete the pipe run design. With a pipe run open for editing, select either the Place or Place Fitting tool from the ribbon.



 **Tip:** When you are working within a pipe run, use the Place Fitting tool to quickly place Content Center fittings without having to navigate the Content Center menus. Simply locate a fitting that is already in your design; whether in the current run by selecting it from the screen, or in a different run by finding it in your model browser. You can also use SHIFT+Right Click and select Part Priority, then select any fitting visible on the screen, regardless of which pipe run it is in. With the fitting preselected, grab the Place Fitting tool (this can also be found in the right click context menu). The fitting you have preselected pops up for placement in your run. **NOTE:** Never, ever, ever copy and paste fittings within a pipe run, or from one to another. This will cause problems, such as the inability to delete the copied fitting should you need to.

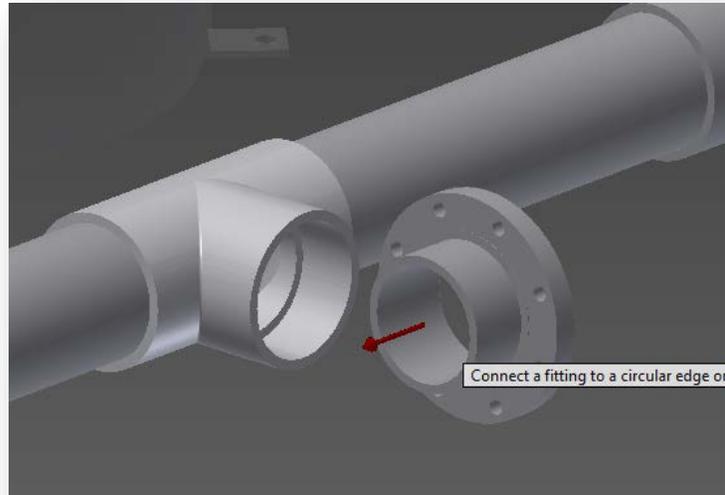


Once you have selected a fitting using either method, and you are ready to place it in your pipe run, you are faced with three choices: Place, Connect or Insert. The Place option will drop the fitting on the screen at a selected location, either in the route or simply on the screen for later use. If placed in the route, you will then be able to specify a location and orientation for the fitting.

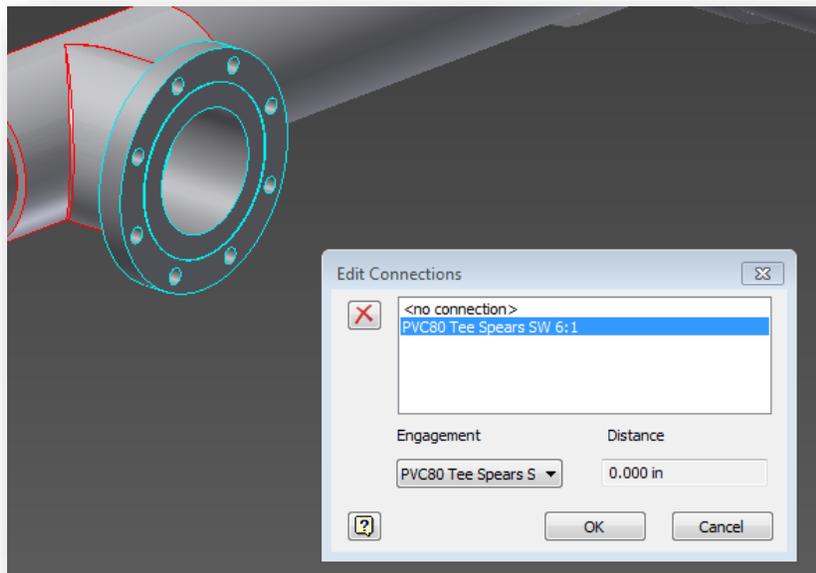


As with the Orthogonal Route tool, the Place fitting option gives you rotational arrows and axes to work with. Right clicking on the rotational arrows allows you to rotate the fitting to a specified angle in the direction of the arrowhead you selected. Multiple rotations can be done to get the correct angle, and angles can be modified later by right clicking the fitting and using Edit Fitting Orientation. You can specify a distance in either direction by right clicking on the axes and selecting Enter Distance, or by simply picking a point and typing a numeric distance. The insertion point of any fitting placed in this way is added to your route as a node. These nodes can be moved later by editing the route and modifying the way it is dimensioned or constrained. Route nodes can even be dimensioned off of work features imported from outside geometry. For example a specific fitting can be located at a specific dimension from a frame member by including geometry on the face of that frame member while in the route editing tools.

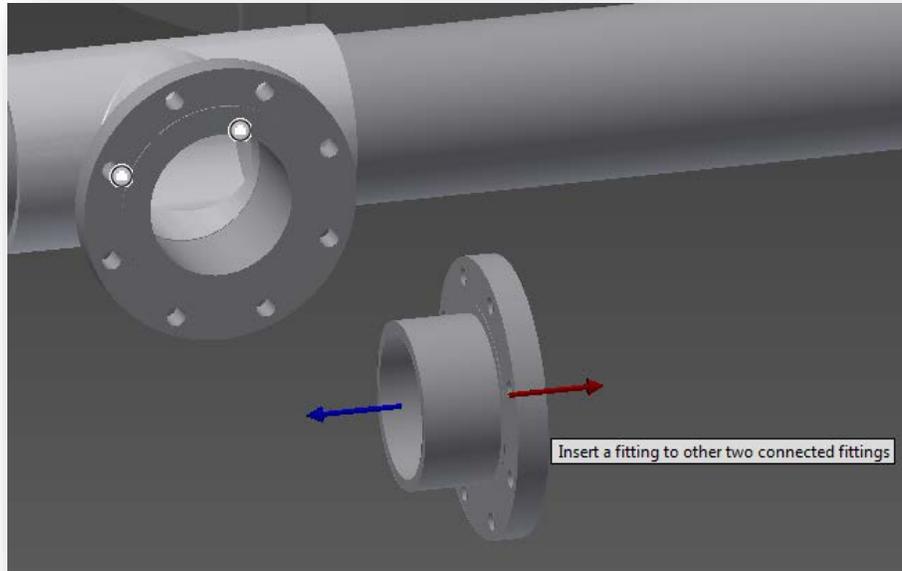
The Connect option when placing fittings does exactly what it sounds like. It connects the fitting being placed to another fitting. What you need to remember here is that the fitting cannot be placed inline on a route. It requires at least one open connection, such as another fitting or the open end of a pipe run.



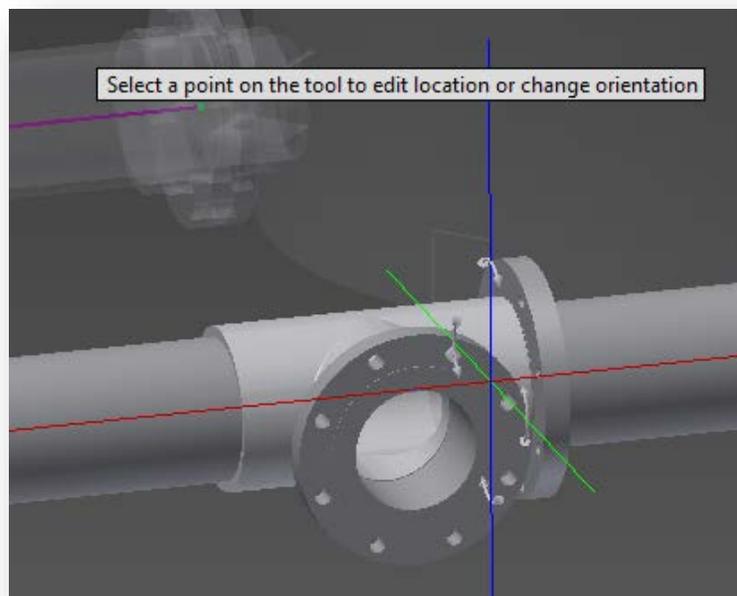
Depending on the number of connections on the fitting being placed, you can toggle between them using the space bar on your keyboard, until you reach the connection you wish to use. Then simply hover your mouse near the open connection point on your pipe or the mating fitting, and Inventor will snap the fitting into place. You will again see the familiar rotation arrows once the fitting has connected. Existing fitting connections may be modified by selecting a fitting, right clicking, and choosing “Edit Fitting Connections”. From this screen you may choose any established connections on that fitting and edit or delete them.



The final option for placing fittings is Insert. This option will place the selected fitting between two other existing fittings, or between a fitting and pipe. The fitting will have red and blue arrows at the connection points, with the red arrow signifying the insert orientation of the fitting.



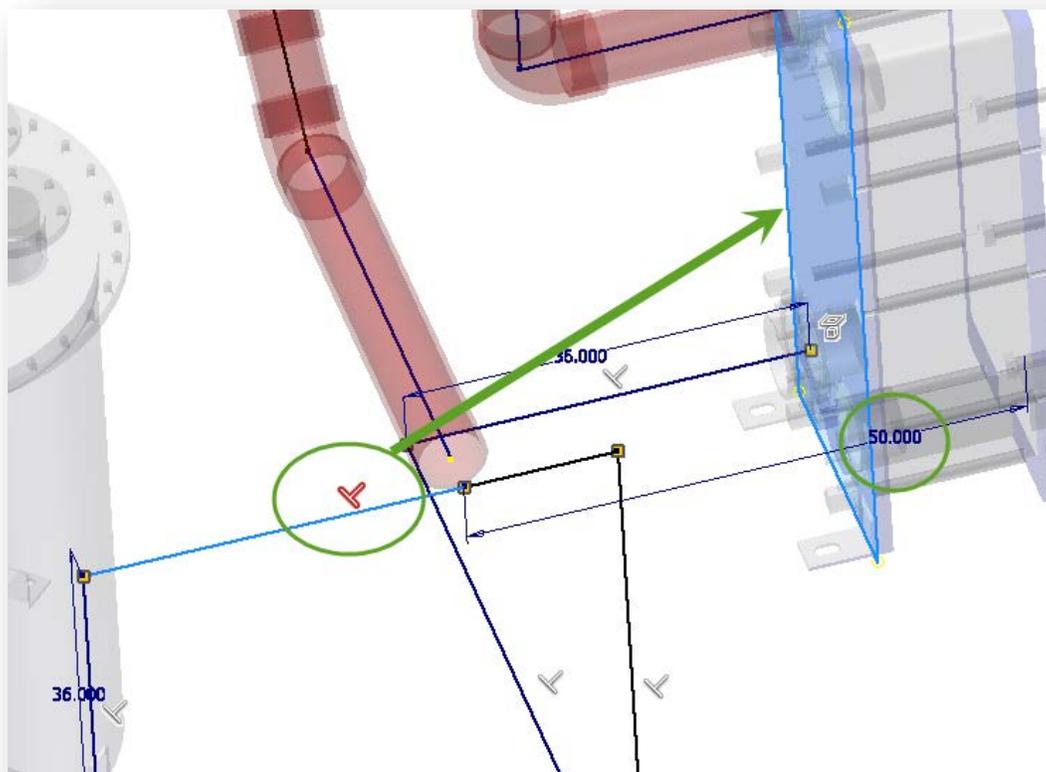
In the example above, In order to insert this flange into the tee, I would space bar to toggle the red connection point and hover near the right side of the tee. Inventor will snap the fitting into place, offer the rotation arrows, and the pipe will shrink or grow to accommodate the new fitting into whatever constraints are on the route.



Including Geometry in Routes.....

In your pipe routes it is possible and sometimes preferable to use geometry elsewhere in your design to constrain your routes. I have mentioned I do not like Auto Routes. They never seem to update properly when the design changes. I prefer to place my own constraints and dimensions on my pipe routes, to give me more control over how they behave when surrounding geometry changes or moves. In the image below, I converted an Auto Route to a sketch, removed all dimensions and constraints except for those with the 36.000 dimensions on them. These I left constrained to the connection points and simply changed the length.

On the Route tab of the Ribbon is a tool called Include Geometry. This is similar to “Project Geometry” in the Inventor part sketch mode. Using this tool, I included several surfaces from the surrounding equipment, bringing them into my route sketch as work planes. These can now be used to place constraints or dimensions in your route.



In this example I have highlighted that the perpendicular constraint and the 50.000 dimension are both referenced to a work plane brought in from the face of the heat exchanger in this design. By doing this, and leaving the first segment of this pipe leg not dimensioned, I have established that if the heat exchanger moved forward or back, the route will grow or shrink to accommodate it. It is important to note that once a route is fully constrained and populated, if fittings are added to the route later, those constraints and dimensions may change. It is a good idea to always double check your routes, and replace any constraints or dimensions that may have changed as you add fittings to your routes.

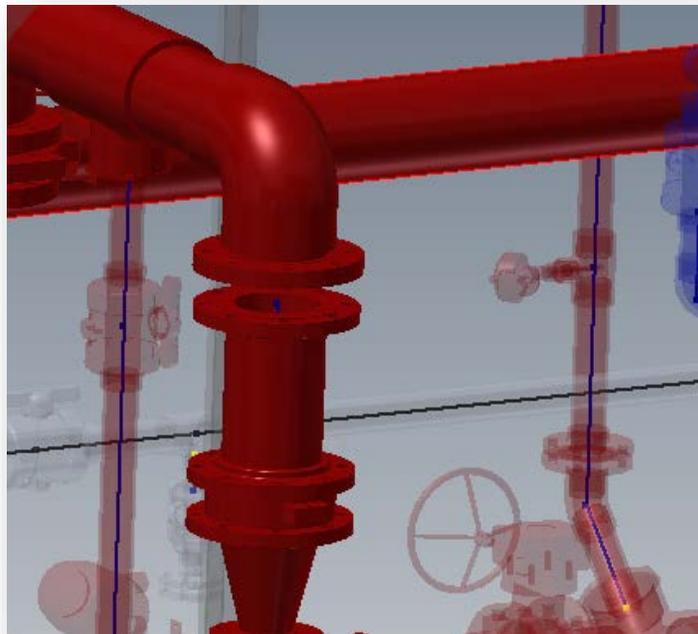
4. Productivity Tips & Tricks

This section will share a few tips I have picked up while using Inventor Tube & Pipe. As with most CAD software, these tips are most likely not the only correct way to perform these tasks, but hopefully you will find them helpful.

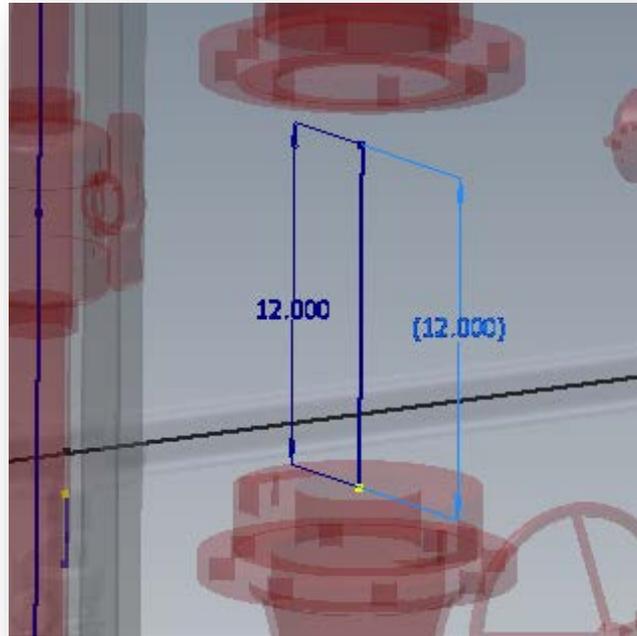
Closing a Gap in a Pipe Run

From time to time as you are designing with Tube & Pipe, a situation may come along where you have done everything right, but somehow you find there is a gap in your pipeline. I have seen this happen under a wide range of circumstances, and by the time you see it, it is usually too late to understand how it happened in the first place. One example I use to demonstrate this, is where two pipelines of different specs come together. One of the fittings is placed using assembly constraints, not tube & pipe, in order to ensure proper alignment with equipment.

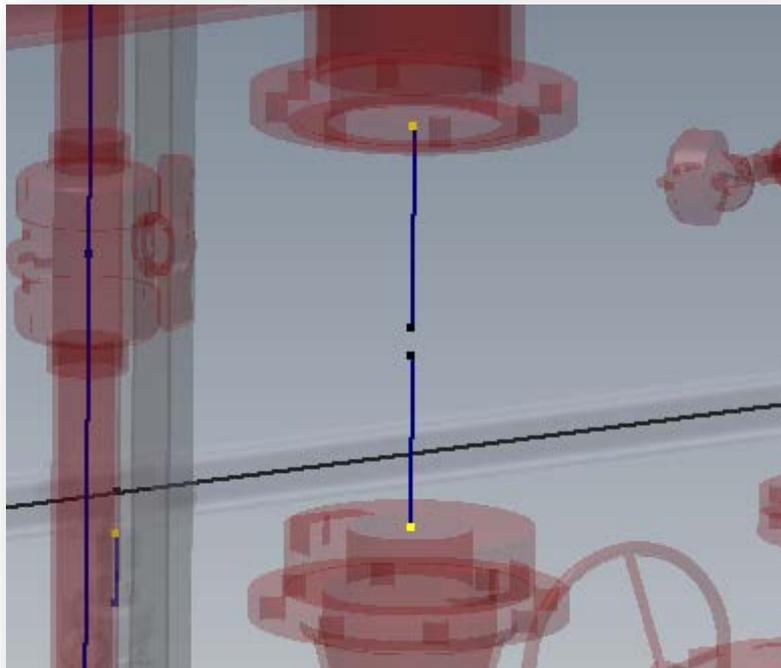
At the junction of these two pipelines, is a valve. This valve needs to be removed, as the design criteria changed. However, because of the way the two lines were constructed, the route cannot heal itself when the fitting is removed, and I end up with a gap between the two lines. The first image shows the line as it was routed, followed by an image showing the line with the valve removed. You can see the remaining gap where the pipeline did not heal itself.



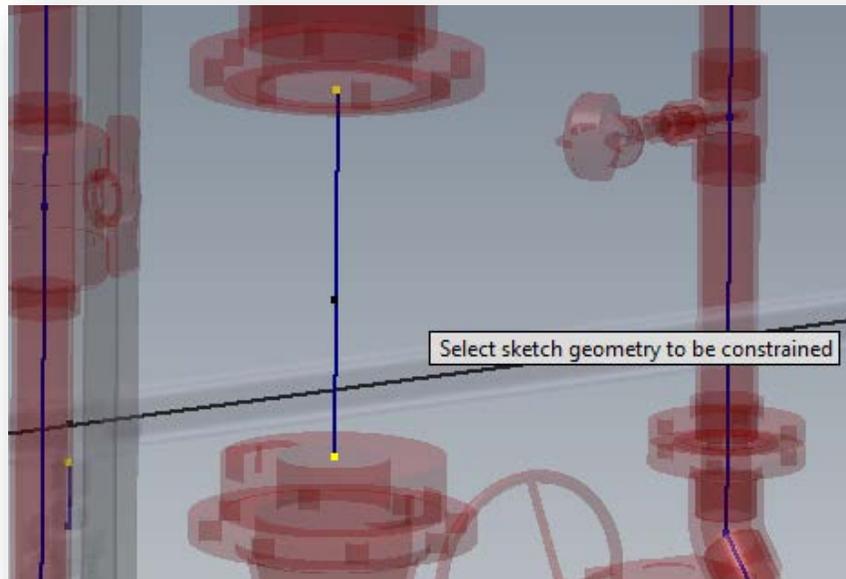
To repair this, regardless of the cause, find the route that contains the gap and edit it.



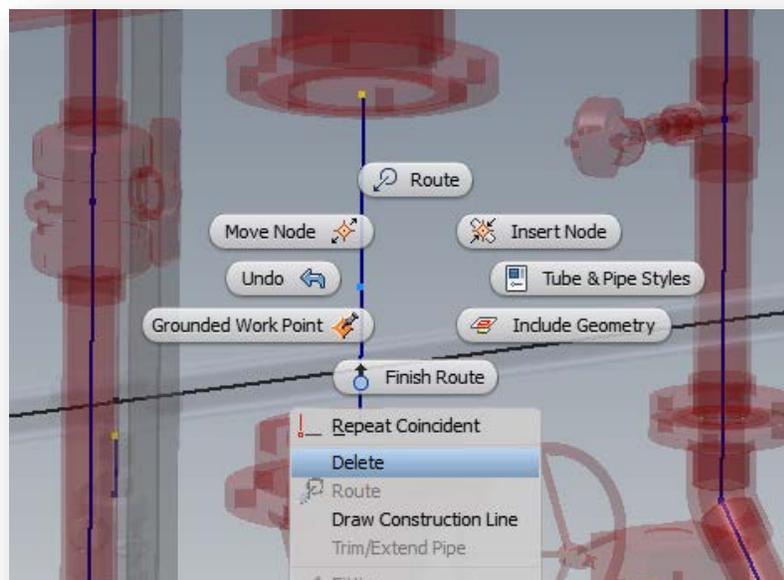
Once in the route editor, I change the dimension of the line to shorten it. Be aware of your minimum pipe length settings. In this case I made it 6" long. The next step is to add a route segment from the top flange, downward toward the end point of the lower segment, but not connecting. Then strip all dimensions off of the two segments.

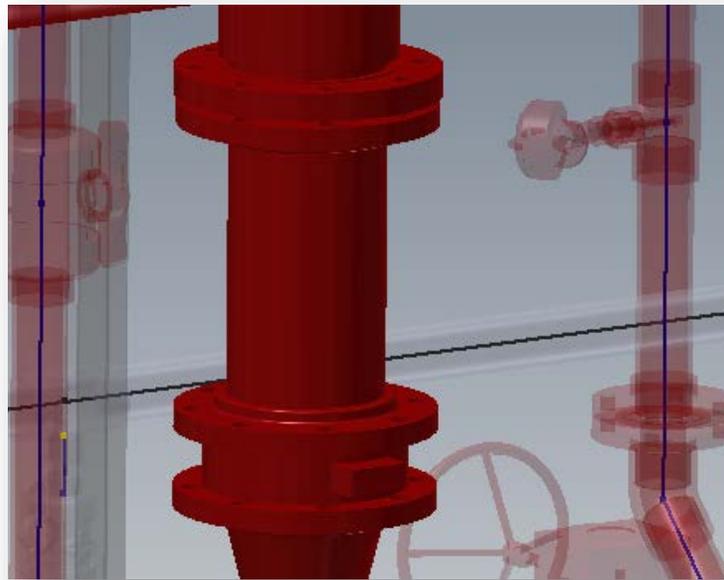


Next, from the route menu, place a coincident constraint connecting the two endpoints of the segments, so that you have a single segment with a node point where the two ends meet.



Finally, select the node point, and from the context menu, delete it. Tube & Pipe will automatically place a reference dimension on the newly created single segment, and you're ready to finish. Once you return the route repopulates and the gap is filled in.





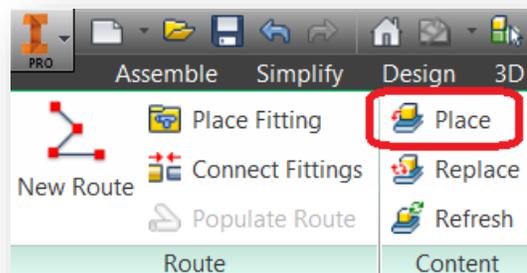
Again, the causes of this type of challenge can vary, and may sneak up on you just when you think everything is working fine. But the solution is always the same.

This technique is detailed in this short YouTube video:

[Close Gaps in Pipe Run](#)

Quick Placement of Pipe Fittings

The primary tool used to add fittings to a pipe route, is the “Place” tool. This is found on the Pipe Runs tab of the ribbon.



This tool takes you to the Content Center where you must navigate your folders until you find the fitting you are looking for, and then select the correct instance from the Family table. If the

fitting you need is already at use, anywhere in your assembly, there is a much quicker way to insert another instance of it. First, and I cannot stress this enough, never, ever ever use Copy and Paste in a Tube & Pipe assembly. Ever. This can cause serious problems later, such as not being able to delete any extra fittings that you pasted. Instead, use the tool found next to Place on the same ribbon tab: Place Fitting.

This tool, selected directly from the ribbon, will take you to your file paths, local or network, instead of the Content Center. It is what you would use to insert a fitting that has been authored, but not published. However, if you can see the fitting you need on your screen, or even in your model browser, simply highlight the fitting to pre-select it, and then use the Place Fitting tool. This will instantly place an instance of the fitting you selected, in the size you selected, into your currently active model. To select fittings on the screen that are not part of the currently active assembly, Shift-Right Click on the screen and set your selection priority to Part. For pipe designs where you tend to use a lot of the same fittings, valves or instruments, this is a nice quick way of adding them to your design, while avoiding the extra mouse clicks of navigating your Content Center

Here is a link to a very short video demonstration of the Place Fitting quick placement.

[Quick Placement of Fittings](#)

Aligning fittings across Pipe Routes

When routing pipes, you will eventually come across this situation. You have 2 pipelines, included in your design as separate pipe runs. You need to have a pipe run from one to the other. Inventor Tube & Pipe currently does not have any sort of automatic branching capability. In other software, if a route is sketched from one pipe run and intersects a route in a different pipe run, the software will automatically insert a branch fitting, based on the current pipe specs. The new route will then populate and connect to the branch fitting.

In Inventor, to accomplish this same design feat, you must insert your branch fittings first, in both pipe runs, and then route between them. So the question becomes, how do you make certain they are properly aligned and perpendicular to your pipe runs? This tip will include some of the “Include Geometry” that we looked at in the section on Auto Routes.

What I generally do, once I have decided where I need to make this connection, is look for some geometry in the area, perhaps on my equipment or on the machine frame. Origin planes of surrounding parts, or surfaces of your frame members etc, are good examples. Once I have found a flat surface running perpendicular to my crossing pipeline, I edit both pipe runs to add the branch fittings, such as a tee. Then one at a time, I edit the route that the tee has been placed into. Once in the route editing tool, I use Include Geometry to project the surface I have previously chosen, bringing it into my sketch as a work plane.

Once included, I delete the dimensions on the branch fitting that were placed by the program, and add a dimension from the branch fitting to the work plane. I repeat this step in the route on the other pipe run, and when both runs have regenerated, the branch fittings are perfectly inline. Now I can create a route running between the two pipe runs, and it will be perpendicular to both. It is not required that they be in the same elevation, as long as you are able to fit an elbow or two in the line to accommodate any height differences.

This technique can also be used for lining up end fittings from different pipe runs, such as aligning tie points along the end of a machine frame. The following video demonstrates the technique being used for the crossover as described above.

[Aligning Branches](#)

Options for Size Changes within a Pipe Run

Currently in Tube & Pipe, there is no easy way to changed pipe sizes in the middle of a pipe run. Placing a reducer inline does nothing to change the size of the pipe downstream. There are basically two options for dealing with this. The first option, given a scenario like the one pictured below, is to break up the pipe run. This requires a separate route for each line size in your pipe run.



Essentially you would first create a route for the smaller diameter, specifying a length for that leg. Then populate the route, and add the reducing fitting to the end. A second route for the larger diameter would then be created starting from the outlet of the reducing fitting.

The other method for handling this is to run the entire pipeline at one size, insert the reducer inline, and then manually change the size of all pipes and fittings on the smaller side of the reducer. This will work, but it must be noted that any further editing of that pipeline can cause the pipe that you changed the size of, to revert back to the size specified in the route.

Help Links for Tubing and Flexible Hose

[Flexible Hose Routes](#)

[Bent Tube Routes](#)

[Tube & Pipe Help Home](#)

[YouTube Search Results – Inventor Tube & Pipe](#)