

TABLE OF CONTENTS

Introduction	1
Manual Format	2
Tubing and Piping Design	3
Pull-down Menus	4
Edit	4
Insert	7
Tools	9
Analyze	12
Tubing Design Workbench	13
Basic Tubing	17
Routing a Run	18
Select/Query Line ID	18
Route a Run	22
Tubing Display Options	51
Route Tools	61
Create an Offset Route	61
Route From a Spline	73
Modifying Runs	79
Definition of a Run	81
Line ID Management	113
Part Placement	129
Place Tubing Part	129
Place Tubing Part	129
Class Browser	133
Placing Multiple Parts	141
Insulation	148
Loose Parts Management	155
Bill of Materials	161
Creating Tubing Connections	167
Part Connections	183
Connecting Parts	183
Disconnecting Parts	194
Offset Segment Connection	201
Manipulating in the Tree	209
Creating a Tubing Spool	213
Transfer Elements	219
Copy/Paste 3D Systems	223

Model Management	237
Break Run	237
Transfer	240
Validate Run Turns	242
Adjust Run Extremity	249
Resize/Respec Part	258
Flip Part Position	263
Move/Rotate Part on the Run	266
 Flexible Tube Routing	 289
Offset from Surface	295
Routing Through Predefined Points	301
FLEX Algorithm	305
 Bundles	 309
Follow Tube	316
Manage Flexible Bundle	327
Manage Local Slack	330
 Analysis	 333
Manage Flows	336
Measuring 3D Systems	339
Analyze Networks	348
Analyze Item	350
Validation Report	352
Report Generation	360
 Piping Design Introduction	 365
Pull Down Menus	365
Insert	365
Piping Design Workbench	367
 Piping	 369
 Creating a PCF	 385
 Problems	 391
Problem #1.0	391
Problem #2.0	393
Problem #3.0	395
Problem #4.0	400

Appendix A	403
Equipment & Systems - General	403
Display	404
Design Criteria	405
Standards	407
Piping Design	408
Appendix B	409
Creating a Tubing Part	409
Appendix C	435
Creating a Schematic	435

Introduction

CATIA Version 5 Tubing and Piping Design

Upon completion of this course, the student should have a full understanding of the following topics:

- Creating runs
- Placing tubing parts
- Managing tubing connections
- Moving tubing systems in the tree
- Working with flexible tubes
- Working with flexible bundles
- Modifying tubes
- Analyzing tubing systems
- Creating tubing reports

Manual Format

It is important to understand the format of the manual in order to use it most effectively. This manual is designed to be used along with an instructor. However, you will need to do a lot of reading as well, in order to fully understand CATIA Version 5. The exercises in this book will list steps for you to complete, along with explanations that inform you of what you have just done and of what you are getting ready to do. The actual steps are in bold type and the information that follows the steps is for your benefit. Anything that appears in *italics* refers to a message CATIA provides—this includes information in pull-down menus, pop-up windows and other messages.

An example of a step and its explanation is shown below (note: normally the lines will not be there):

Select a location to the right of the origin. This specifies the other end point of the line. You will continue specifying locations in order to complete your profile. It should appear similar to the diagram shown below.

As you can see, the desired action blends in with the text except that it appears in bold. The information following the step explains what that step accomplished and where you are going next. It is important to read this information in order to better your understanding of CATIA Version 5.

Also, you will find that the exercises build upon themselves. Later exercises often assume you know how to do certain steps which have been covered earlier in the course. If you did not quite pick up what you needed to know from an exercise, you will probably want to review it several times before moving onto more advanced sections. The advanced sections assume that you have a good understanding of the previous sections. Therefore, fewer steps will be provided. Eventually, you are expected to be able to create tubing-related elements without any steps being provided.

Tubing and Piping Design

The Equipment and Systems solution demonstrates how models can quickly become complex in CATIA. Within this solution there are rules and checks that are automatically performed. These checks will ensure that parts fit correctly, or that appropriate bend radii are being used. They can also automatically add parts, such as washers and gaskets. The idea is not just to create compatibility within a tubing system, but to create a tubing design that will work within a global system. For example, pipes are used to deliver water to a home. The water will need to go into the water heater and then into the washing machine to wash clothes. The water heater will be a system all to itself, with cold water coming in and hot water leaving. The water heater will also need to create heat. The heat can be created with gas or electricity, but an energy circuit will need to be defined. How the water heater works might be designed by an outside company, but the systems going up to it will still need to be designed. Once the water is heated, it will then need to go into the washing machine; a flexible tube will need to be designed. With the water in the machine, it will then need to be disposed of; a disposal system will need to be designed. To design the water system, it could be necessary to use six or more different workbenches (Space Reservation, Piping, Equipment reservation, Electrical Part Design, Electrical Harness Assembly, Tubing, Weld Design). Before using any of the Equipment and Systems workbenches, it will be helpful to have a good understanding of Part Design, Generative Shape Design, and Assembly Design, and to also have a working knowledge of the Catalog Editor.

This book is limited to the Tubing and Piping portions of the Equipment and Systems solution.

The Tubing and Piping workbenches have many similarities. The icons look nearly the same, and the process used to create a system is practically identical. There is also some overlap between the two for work that needs to be done. For these reasons, they are usually taught at the same time. With the workbenches closely related in design and result, mastering Tubing will make learning Piping very easy.

Working with Tubing and Piping will involve a lot of rules. The rules will be created at different levels. Some of the rules can be worked around, while others are created by an administrator and cannot be changed. These rules are meant to speed the design process by narrowing the options available and limiting the amount of rework necessary. You may find the rules to be cumbersome at first, but as you get used to what they are doing, they can become a useful aide. In this class, no rules will be created. The rules that are used are default CATIA.

Route Tools

There are many ways to create runs. Determining the best method will be based upon the type of geometry available. When tubing, there will often be a return line created. Instead of routing back through a series of points, it may be easier to offset the run. The offset can then be modified, if necessary. It is also possible to route from a spline, allowing more control over the route that the run will follow. Polylines can be used for this, as well. If the wireframe element is modified, the run can be updated to show changes in the geometry.

Open the **Offset Runs** document located in the *Offset Runs* directory.

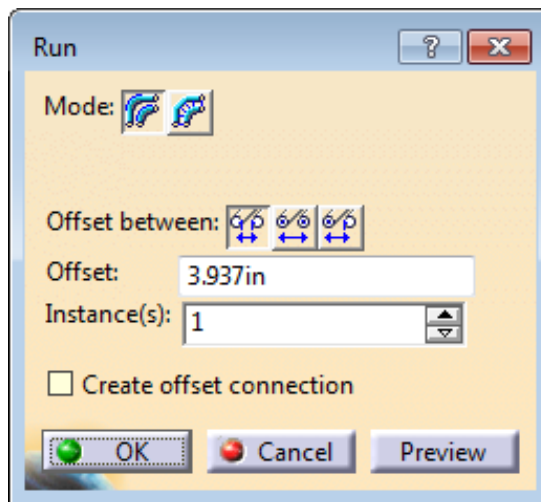
Create an Offset Route

An offset route duplicates a run in a specified direction. Once offset, the run can be modified like any other run, or a connection to the original run can be created.

Select the **Create an Offset Route** icon.  It has a black arrowhead beside it. Selecting the arrowhead displays a sub-toolbar with additional, related icons.



The *Run* window appears.



Mode



Constant Clearance

Maintains the turn Radius of the master tube



Constant Radius

Maintains the Clearance between tubes

Offset Between

Types of offset measurement



Outside Edge to...

The offset distance is measured between the outside edges



Centerline to...

The offset distance is measured between the centerlines



Centerline to...

The offset distance is measured from the centerline of the first run to the outside of the second run

Offset

Specifies the distance between the runs

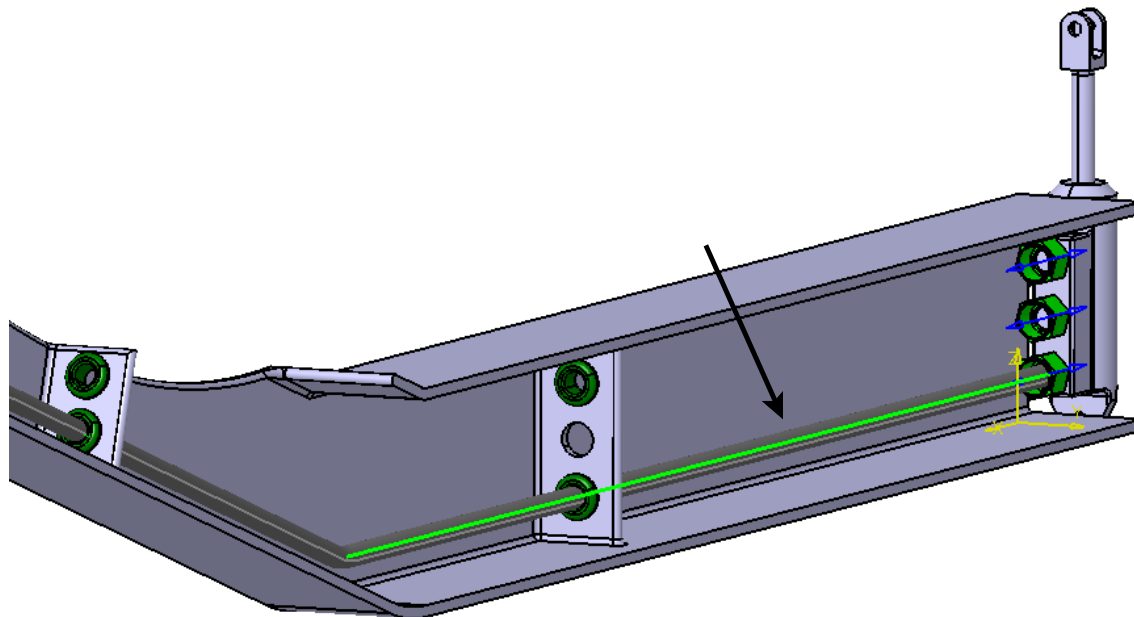
Instance(s)

Specifies the number of offsets to be created


Create Offset Connection

Links the offset runs to the original run

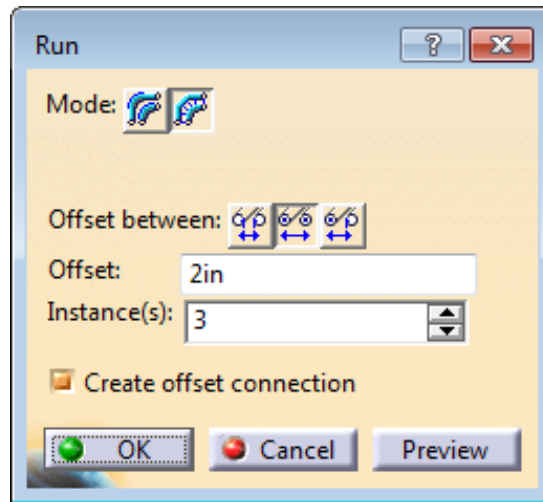
Select the run segment shown below. The compass snaps to the run. The offset will align with the positive *W* direction. Negative values can be entered, if desired.



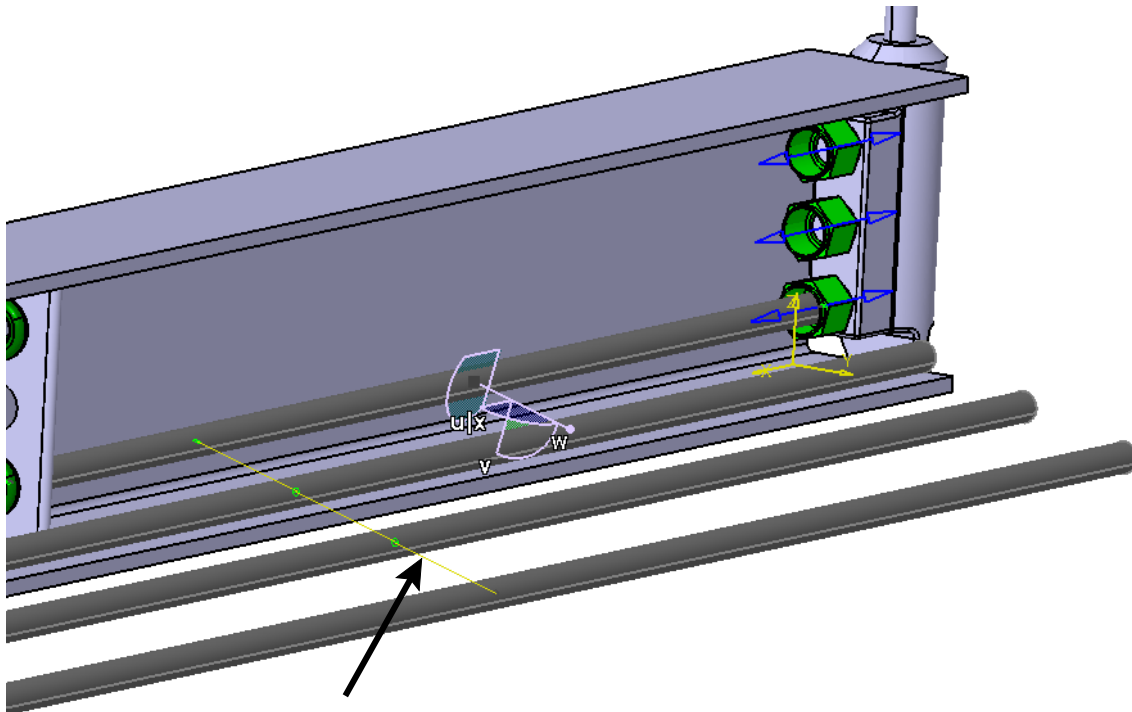
Select the **Constant Clearance** icon.  This will force the runs to keep the same distance everywhere, including corners.

Select the **Centerline to Centerline** icon.  This will force the spacing applied to be applied between the centerlines of the two runs.

Enter an *Offset* value of 2.0 and change the *Instance(s)* field to 3.0, then ensure the *Create offset connection* checkbox is turned on. The window should look the same as below.



Select *Preview*. The product appears as shown.



The runs are created. Each run has a master-slave connection to the one before it. The connections are the green and red symbols. With the *Create offset connection* option selected, when the original run is modified, the others will follow.

Press the third mouse button on the compass and select *Edit*. The *Parameters for Compass Manipulation* window appears.

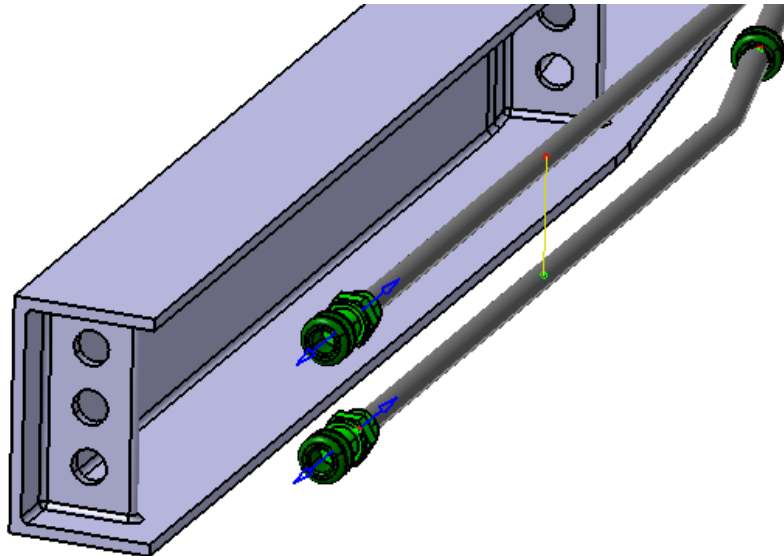
Change the *Along X* field under the *Angle* column to zero and select *Apply*. The compass now matches the directions of the global axis. You could also have selected the *Use Local Axis System* option from the contextual menu after pressing the third mouse button on the compass. This would have automatically forced the compass axes to be parallel with the global axes.

Select *Close* in the *Parameters for Compass Manipulation* window, then select *Preview* in the *Run* window. The runs change orientation.


Change the *Offset* to 2.5 and *Instance(s)* to 1.0, then select *OK*. There will be a single run offset 2.5 inches directly above the original run.

Change the *Frame_Offset* parameter in the *C frame* part document from 9.0 to 5.0. You will need to double select on the parameter twice to edit it.

Double-select on the *Offset Runs* product in the tree to make it active. You are taken back to the Tubing Design workbench. Notice that the runs no longer pass through the holes.



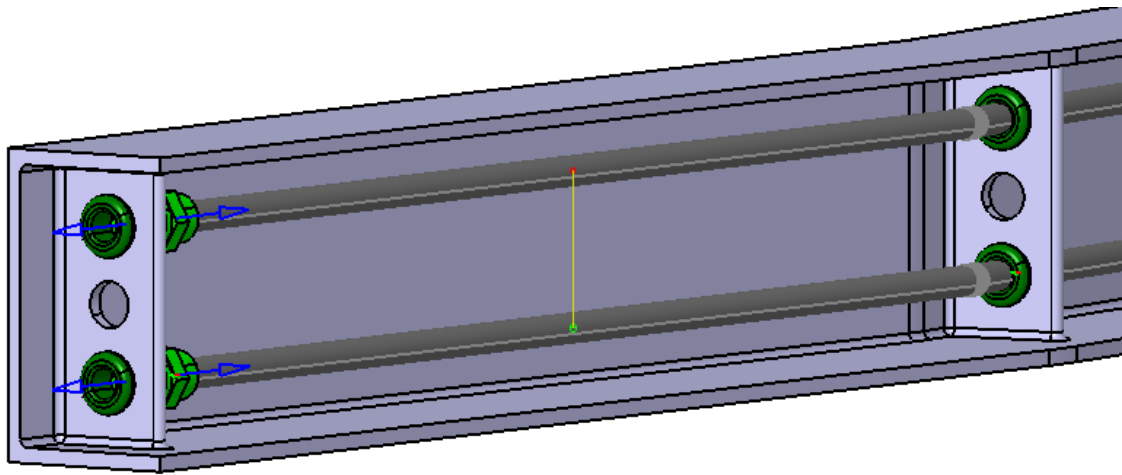
However, the runs do still go through the grommets. The problem is that the constraints locating the grommets are not up to date. You can verify the constraints by opening the specification tree.

Select the **Force Update** icon.  The constraints are now up to date, but the runs may not be. Unfortunately, CATIA gives no indication that the runs might need to be updated.

If the runs do not update with the grommets, select the **Force Update** icon again.

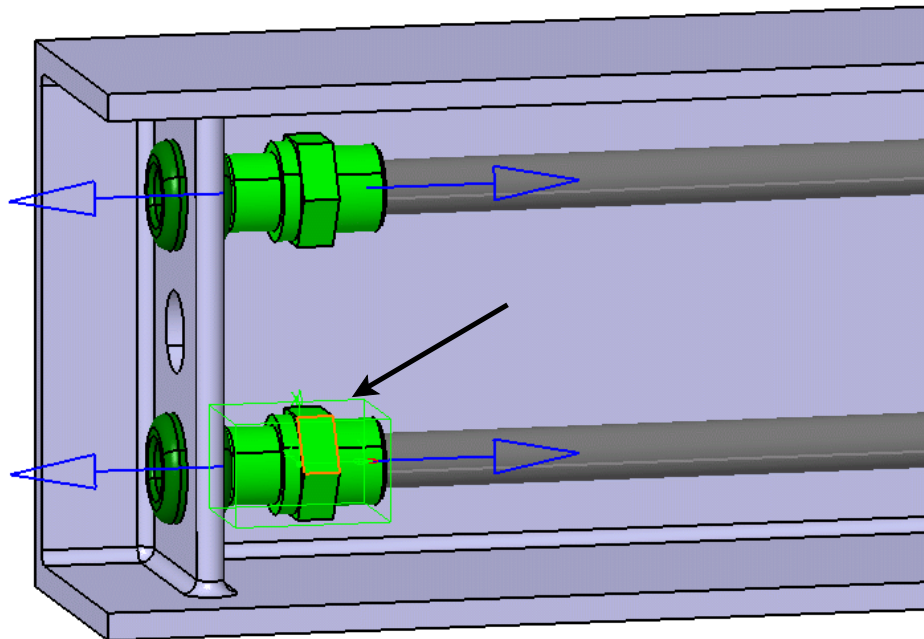


The runs should now be in the proper position with the grommets.

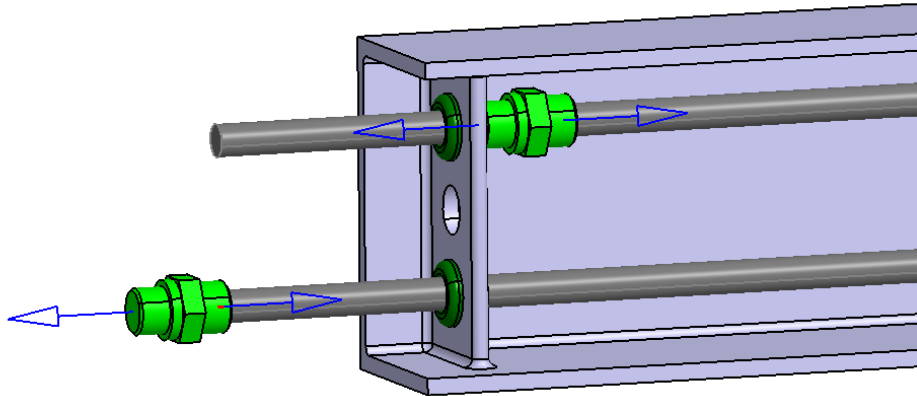



The first run was created with respect to the tubing connections on the grommets. The second run was created with respect to the first run.


Select the **union** shown below. The green manipulation box appears around it.



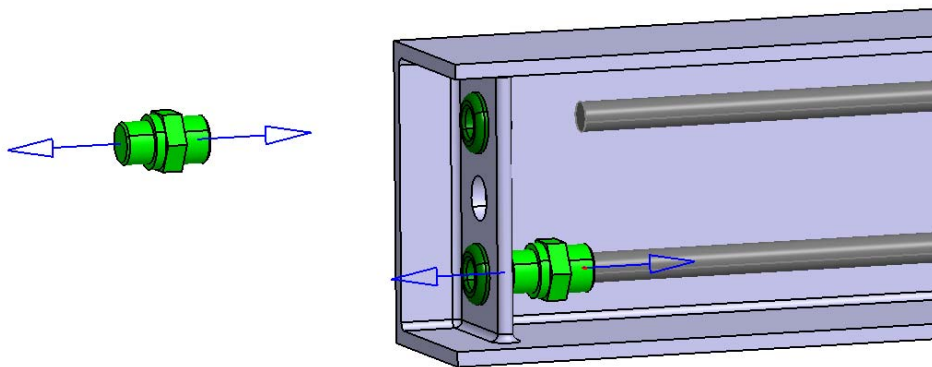
Drag this connector to the left about 6.0 inches. The connector will move, regardless of the constraints that are put on it. The run will follow the connector and will automatically update. The second run will move without its connector because it is not attached to the connector; it is only associated with the first run. This can be a problem later on when doing analysis.



Select the **Force Update** icon.  The connector will move back to its correct location.

Select the **Force Update** icon again to update the runs, if necessary. 

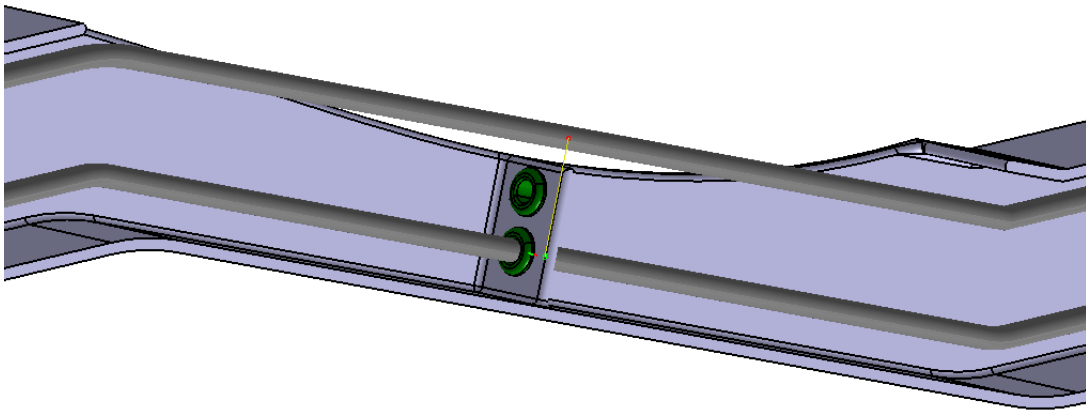
Select the upper union and drag it to the left. The run does not follow.



Select the **Force Update** Icon.  The connector returns to its correct location.

For the run to update with the union, it must be connected. Creating connections will be covered later.

Now take a look at the middle of the frame.



The run does not pass through the grommet here. The segments will be adjusted rather than the entire run. Adjusting a segment can be done in several ways. This time, you will offset it instead of dragging it to a location or placing it via the node edit table.


Delete the offset run, then select the **Offset Run icon.**  The same run will be created, but without an offset connection.

Turn off the *Create offset connection* checkbox. The new run will no longer be connected to the first run in any way. By using the **Connect** and **Disconnect** icons, connections between runs can be managed on individual segment sections. These options will be investigated later.

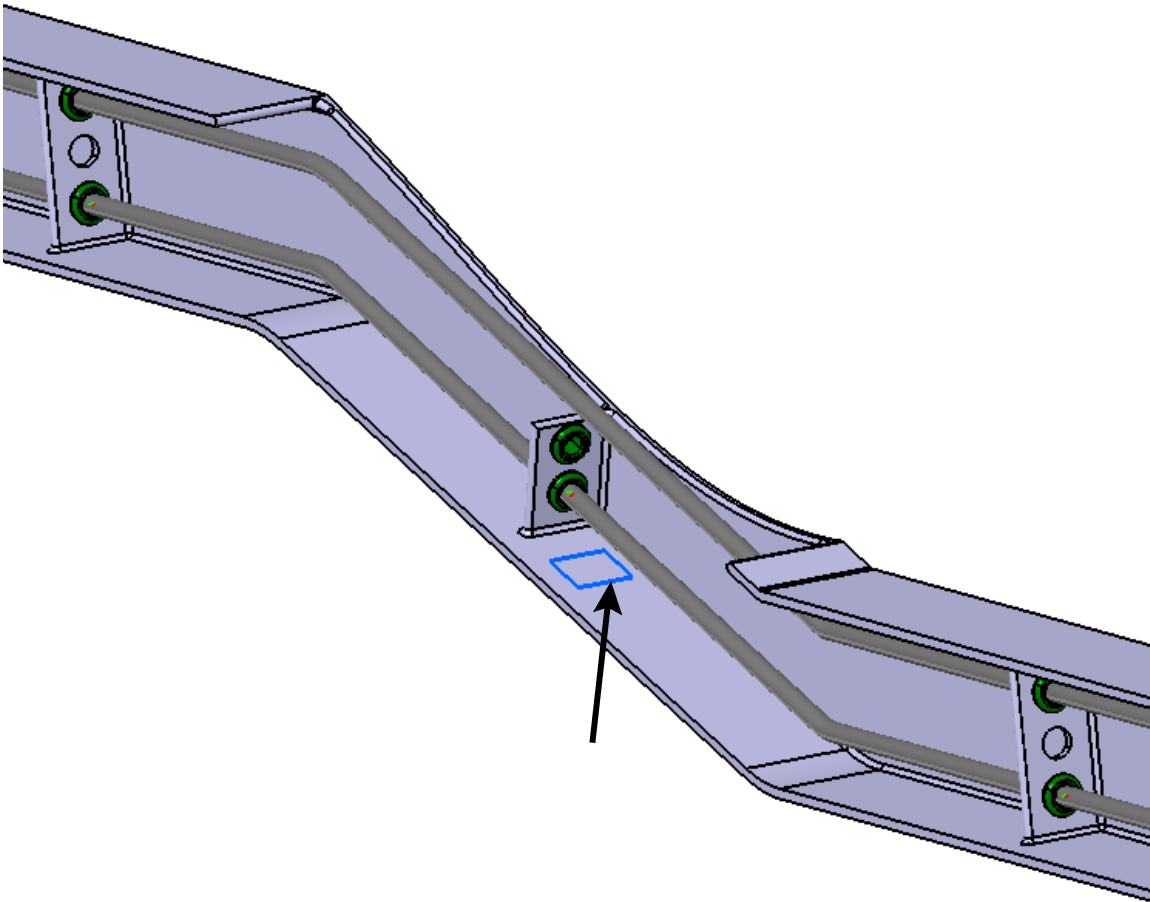
Select the same run segment as before. The compass snaps to it.

Rotate the compass to point in the Z direction. This can be done in the *Parameters for Compass Manipulation* window.

Select *OK*. The run will be created with no master/slave connections.

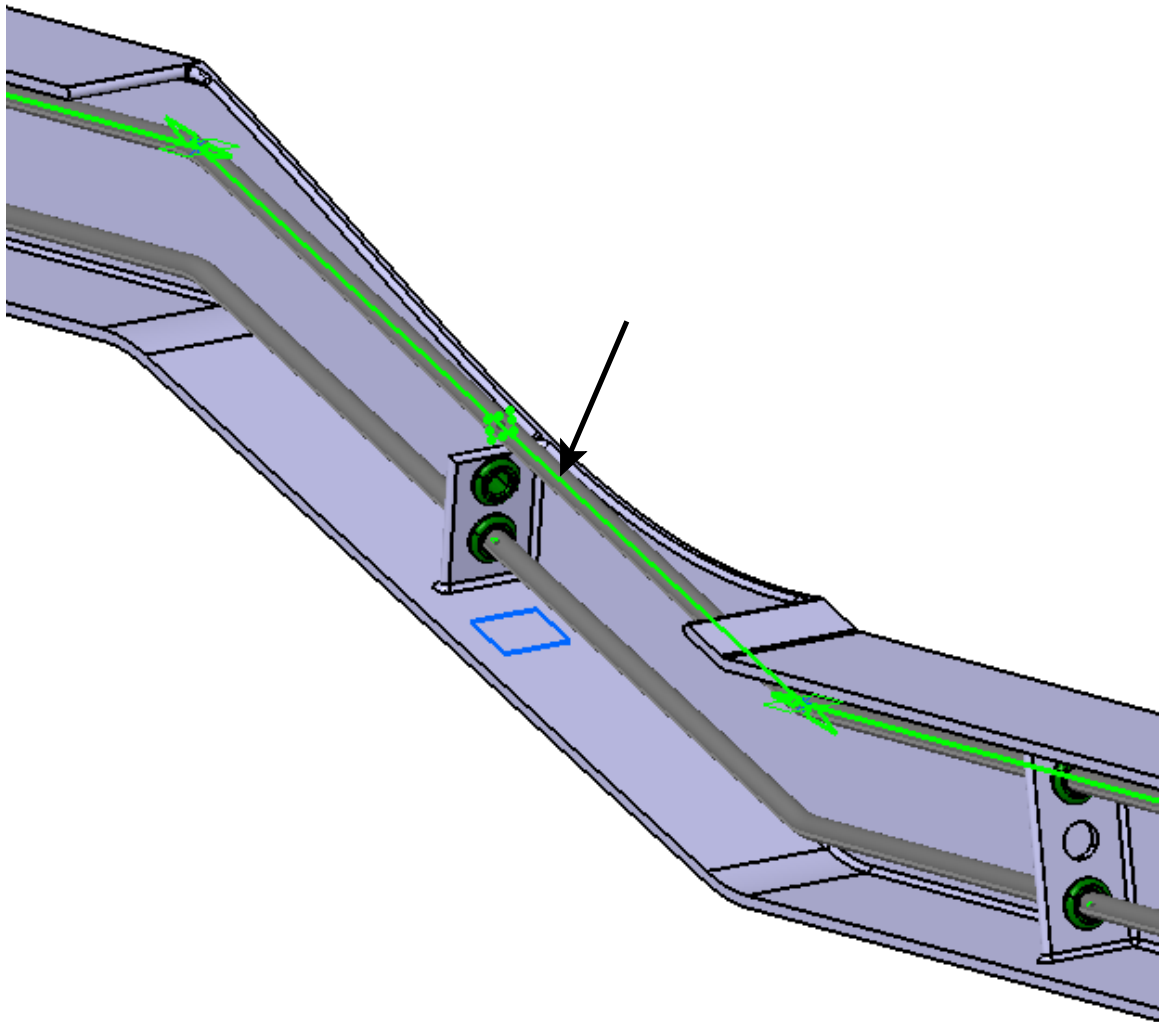
Select the **Offset Plane icon.**  It is located at the bottom of the CATIA window in the General Environment Tools toolbar.

Select the top face of the frame where shown. This places the reference plane on that surface. It is from there that the offset will be measured. A blue plane appears on the frame as shown.

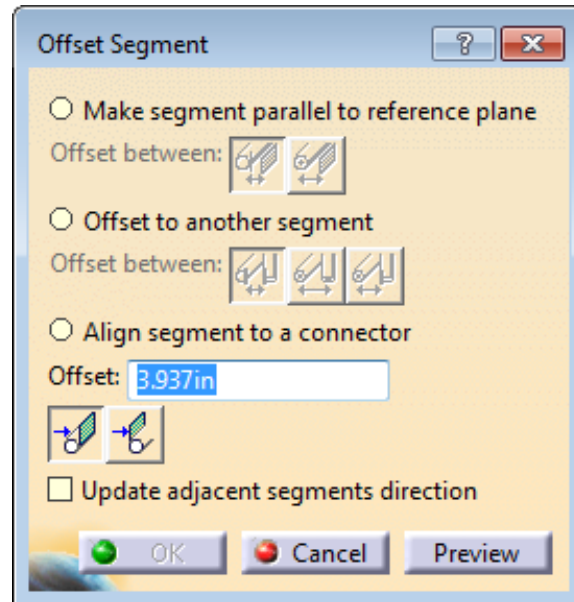


Press the third mouse button on the offset run and select *RunXXXX* object from the contextual menu, then select *Definition*. To offset the segment, the properties of the run need to be open. The *Definition* window appears.


Press the third mouse button on the angled run segment where shown.



Select **Offset segment** from the bottom of the contextual menu. The *Offset Segment* window appears.



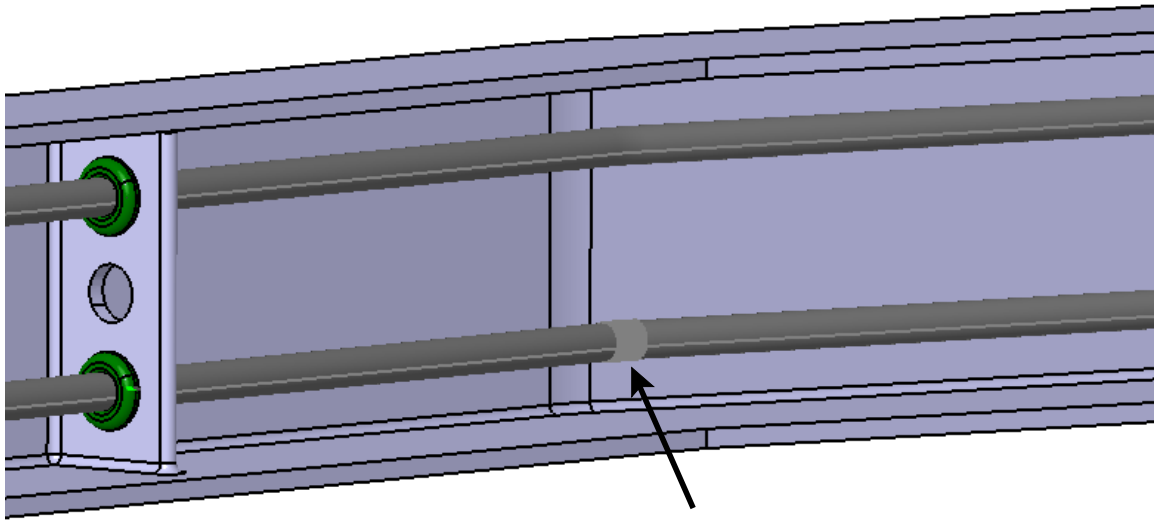
In this window, a segment can be offset from the reference plane. It can also be aligned with a connector, or offset to another segment.

Select the **Make segment parallel to reference plane** option and choose the **Centerline to Reference Plane** icon.  This specifies that the offset value will be measured between the blue reference plane and the centerline of the run segment.

Change the Offset to 2.25 and select Preview. A green dashed line appears where the run will move to. There are no measurement tools inside the definition window. If a measurement is necessary, the Measure Tools toolbar must be used. Fortunately, it can be accessed while the *Offset Segment* window is open.

Select OK twice. The run passes through all of the grommets.

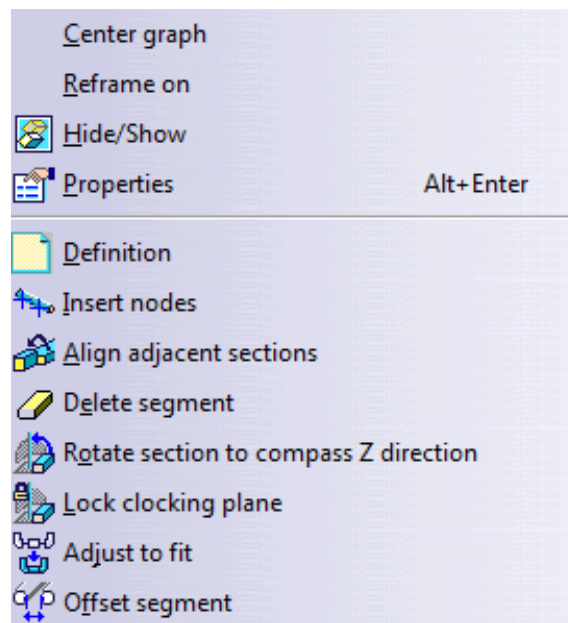
Look at the runs wherever a turn is made. Some of the corners have a different appearance.



At first, this may look like a graphical representation issue. In actuality, the segments are twisted. They can be fixed by aligning adjacent sections.

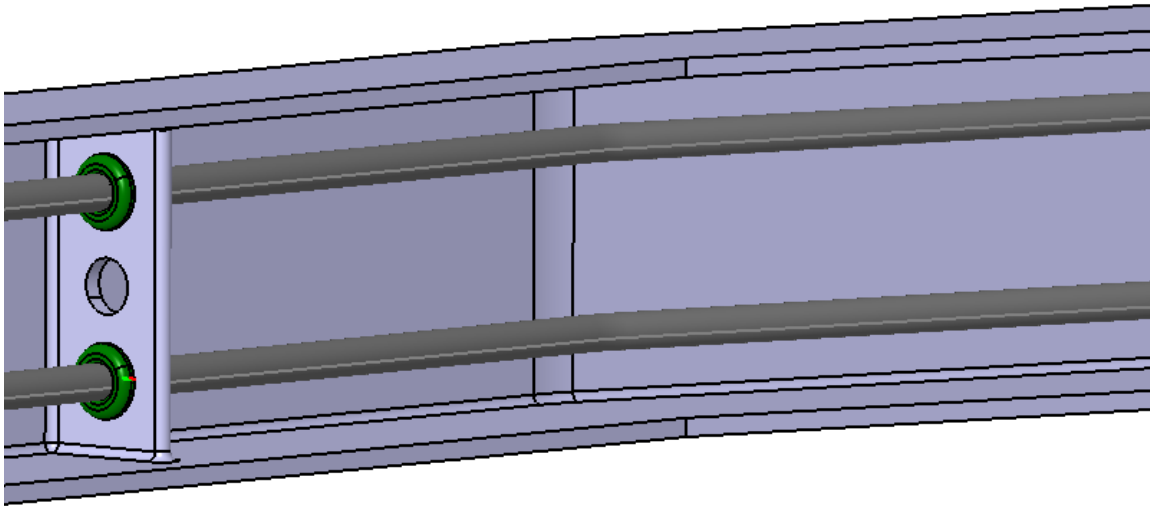
Press the third mouse button on the lower run and choose *Run-XXXX object*, then *Definition*. The *Definition* window appears.

Press the third mouse button on one of the green segment lines. A contextual menu appears for that segment.



Select *Align adjacent sections*. This option adjusts segments so that they line up. It is similar to using closing points for a multi-section surface. All the turns for the run should now be smooth.

Select *OK*.



Save and close the document.

Creating Tubing Connections

Not all parts utilized while tubing will have tubing properties. If the tubing properties are not a priority, using assembly constraints to put connectors in place works fine. However, you may want to have a logical connection or a quicker way to position your tubing parts. To do this, you will need to define one or more connection points on those parts.

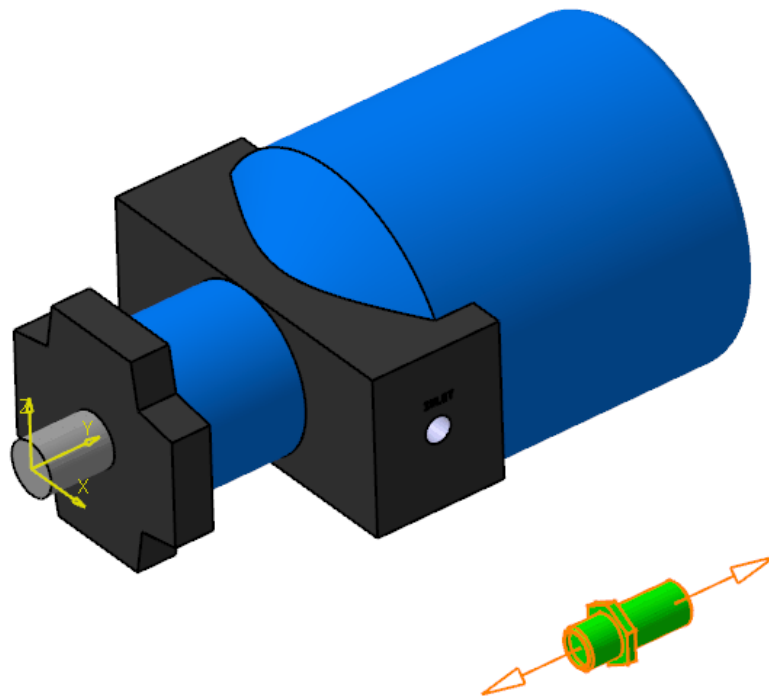
Open the Creating Tubing Connections document from the *Creating Tubing Connections* directory. The *Pump.1* part instance in the assembly is just a regular part with no special properties.

Set the line ID to *1/2in*, then select the Place Tubing Part icon.  The *Place Tubing Part* window appears.

Select *Tube Function* for the *Function Type*. A point needs to be selected for part placement. A union will be placed in the inlet hole, but there is currently no point there to select.

Select a point in space and change the *Part Type* to *Union*. The *Part Selection* window appears.

Select the *UNION-BULKHEAD-FFSM-08*. It is the first option in the list.



A part is inserted at the point you selected.

Select *Close*.

Select the **black arrowhead** beside the **Build Tubing Part** icon. A sub-toolbar with related icons appears.



A sub-toolbar



Select the **Build Connector** icon from the sub-toolbar. The *Manage Connectors* window appears.



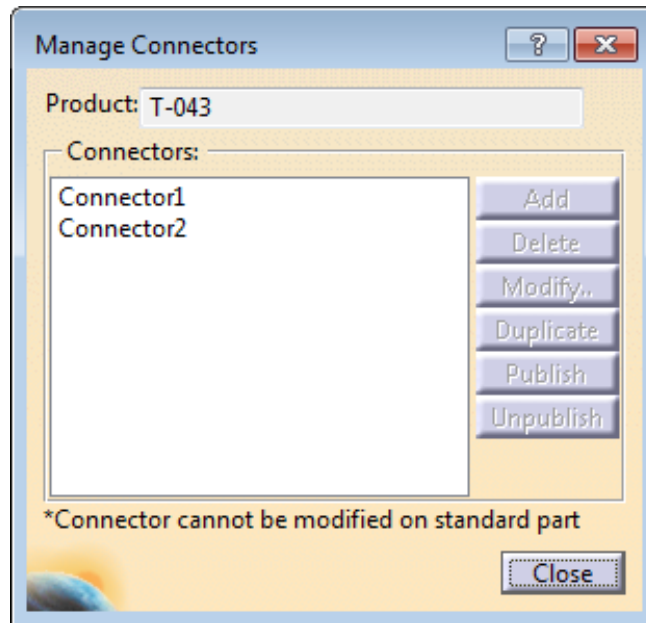
The *Manage Connectors* window appears.



<i>Product</i>	Specifies the product that the connectors will be assigned to
<i>Connectors</i>	Displays the connectors that exist on the product
<i>Add</i>	Creates a connector
<i>Delete</i>	Removes a connector
<i>Modify</i>	Modifies an existing connector
<i>Duplicate</i>	Duplicates a connector
<i>Publish</i>	Publishes a connector
<i>Unpublish</i>	Removes the publication from a connector

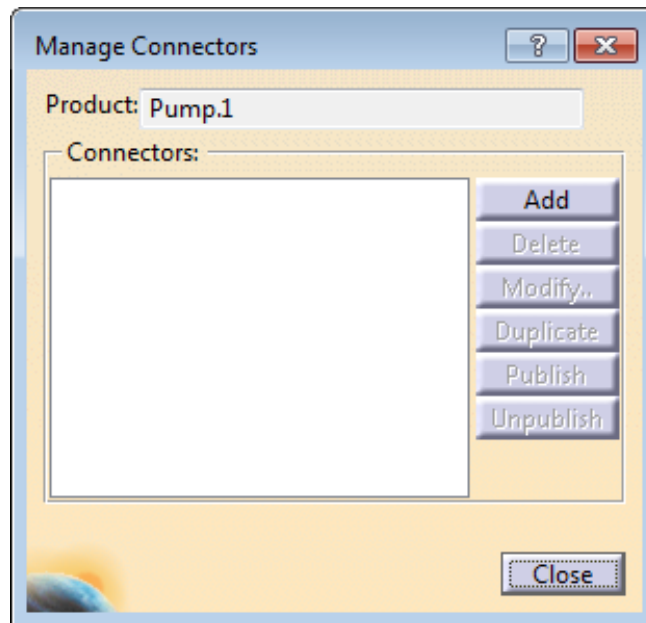
When you select this icon, you will be adding information. CATIA needs a place to store this information, and, by default, the product you are working in will be selected as the storage location. Storing connectors at the product level works, but it is a better idea to add connectors on the part instance. Parts that are inserted through tubing will often already have connectors on them.

Select the union either graphically or from the tree.



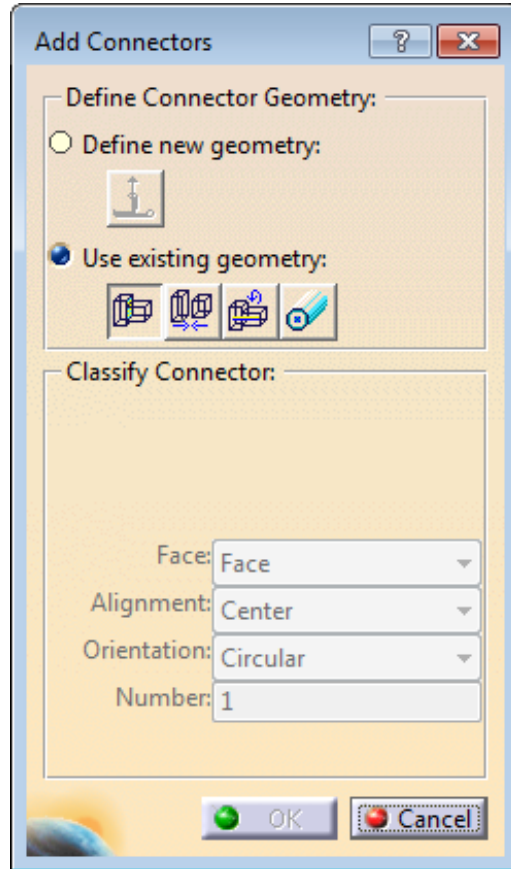
This part already has connectors associated with it. The note at the bottom of the window is important. These connectors are unable to be modified. When using a standard part, you cannot add, remove, or modify the connections that were created by the librarian.

Select *Pump.1*.



This part has no connections since it is not a tubing part. However, connections can be added.

Select the **Add** button. The *Add Connectors* window will appear.



Define Connector Geometry

Define new geometry

Allows a connector to be placed at a location specified by the user

Use existing geometry



Select Face Connector...

Defines the face that another part will mate up to



Select Alignment...

Defines the centerline that another part will align with



Select Orientation...

Defines a face or plane that another part will be parallel or normal to




Select Datum Point

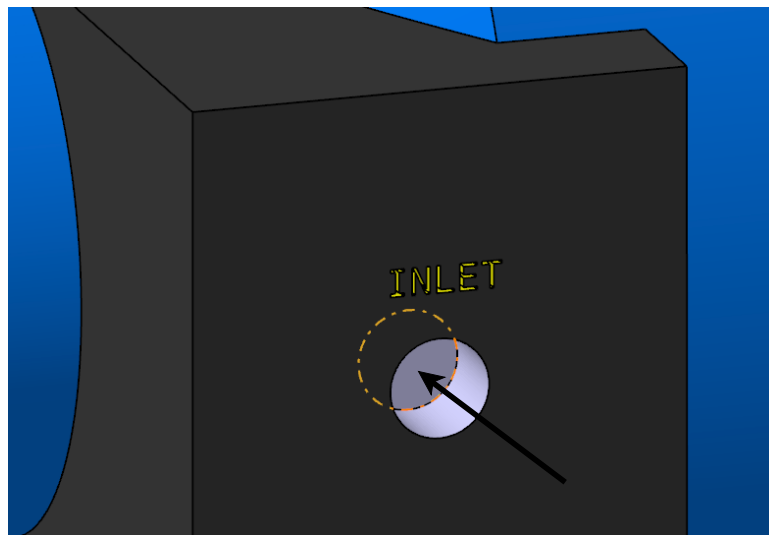
Used for flexible tubes or as a pivot point

Classify Connector

<i>Face</i>	Specifies whether the run will stop (<i>Face</i>) or continue through (<i>Hole</i>) at this connection
<i>Alignment</i>	Defines what portion of the run will go through the connection
<i>Orientation</i>	Defines the orientation of the connector
<i>Circular</i>	Specifies any orientation is allowed
<i>Rectangular</i>	Specifies the orientation must match the shape
<i>Up</i>	Specifies a definite “upward” direction that will force an explicit orientation
<i>Number</i>	Specifies the number of connections on the part

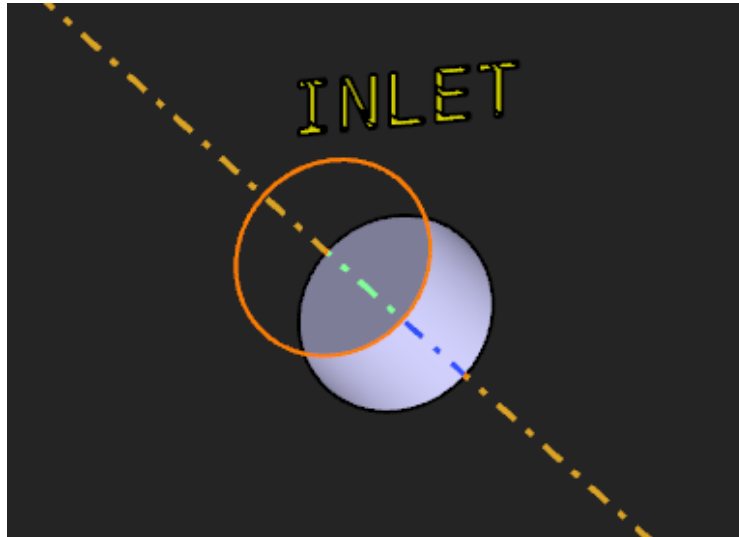
Select the ***Use existing geometry option***. The Select Face Connector Geometry icon should be selected.  This defines which face the connecting part will mate up to.


Select the **back of the inlet hole**. It will stay selected throughout the creation process.



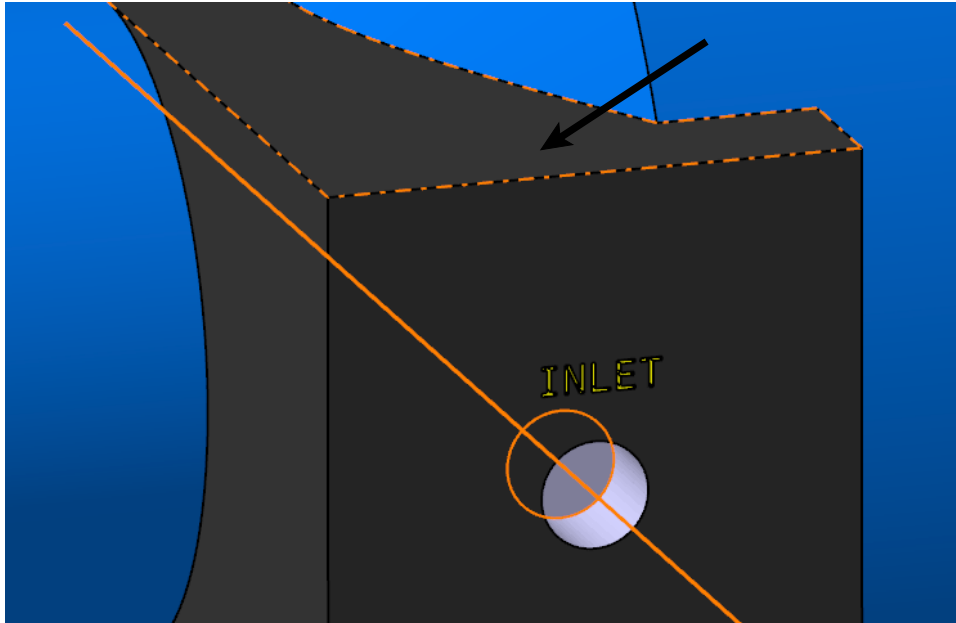
Click the **Select Alignment Connector Geometry** icon. 

Select the centerline of the inlet hole. This is done by clicking inside the hole, on the cylindrical face.



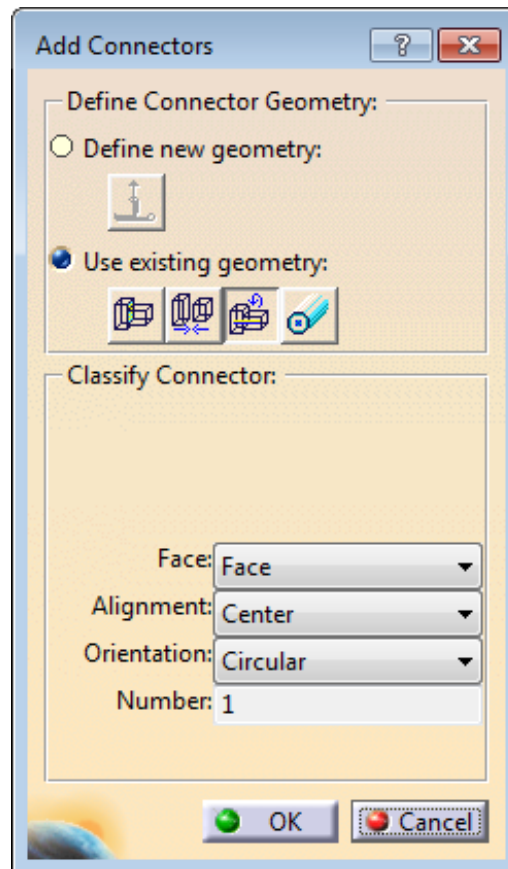
Click the **Select Orientation Connector Geometry** icon.  This is the last piece of required information. Defining an orientation will keep the connector from spinning in its location.

Select the face shown below.

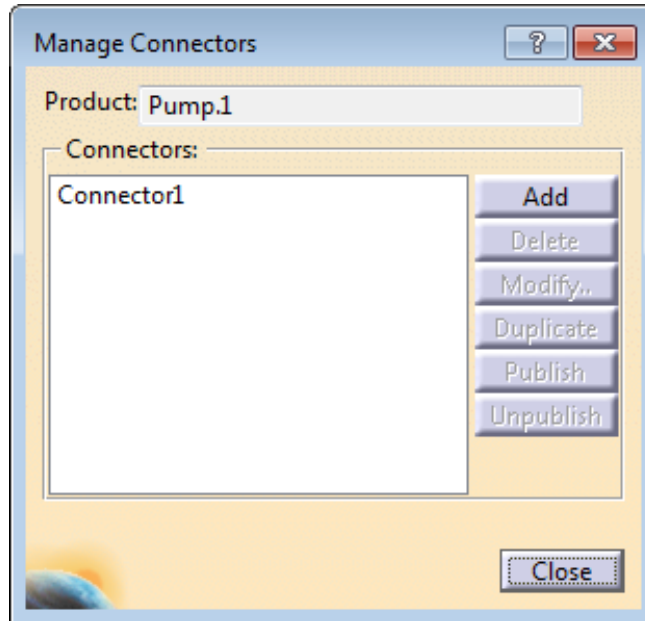


With the top of the block selected, the definition of the connector is set. Now the *Classify Connector* options are available to define the type of connection that will be made. There are two types of *Face* connections: *Face* and *Hole*. If you are routing a run through a point and want it to continue through the location (like a support or a clamp), you would want a *Hole* connector. Since you are putting a connector at this point and the run will actually end, you should define it as a *Face* connector. The alignment can be changed so that the tube will not go through the center of the hole. This is typically used with rectangular connectors, or if there are two connectors through the same hole. The *Alignment* of this connector will stay as *Center*. *Orientation* is used to define how the connector is to be oriented.

Set the *Classify Connector* options as shown here.



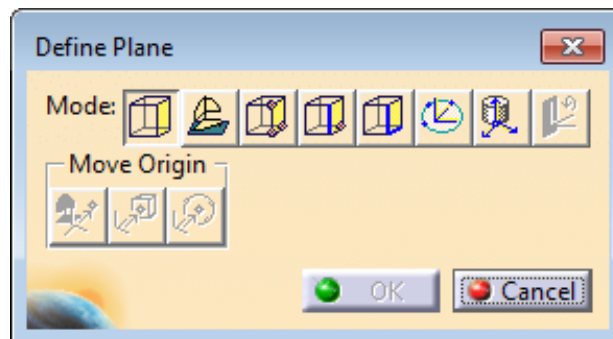
Select **OK**. The *Manage Connectors* window reappears, and the new connector is listed.



Select the **Add** button again. The *Add Connectors* window appears. You will create a second connector using the other option.

Select the **Define new geometry** option. The *Select Plane to Define Connector Geometry* icon becomes available.

Click on the **Select Plane to Define Connector Geometry** icon.  The *Define Plane* window appears.



Mode









Define Plane

Defines a plane by selecting a face







Define Plane Using...

Defines a plane by positioning the compass

	Define 3 Point Plane	Defines a plane based on three selected points
	Define Line-Point Plane	Defines a plane by selecting a point and a line or edge
	Define Line-Line Plane	Defines a plane by selecting two lines
	Define Plane at Center...	Defines a plane by selecting three points along a circular edge
	Define Plane at Prod...	Defines a plane at the origin of the product
	Define Orientation	Changes the orientation of the connector

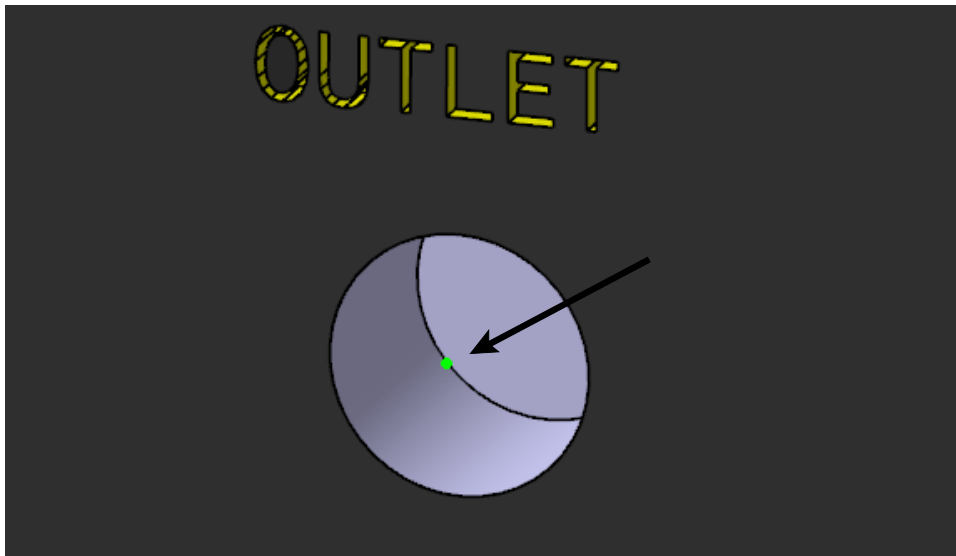
Move Origin

	Define Origin at Plane...	Defines the origin using the compass or a plane
	Define Origin at Point...	Defines the origin at a point or on a surface
	Define Origin at...	Defines the origin at the center of a circle created by selecting three points

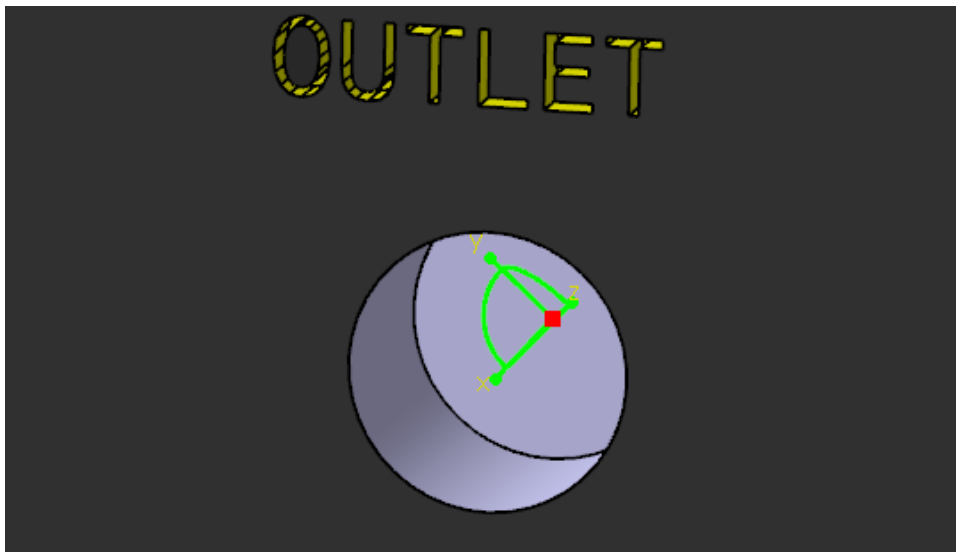
Select the **Define Plane at the Center of Circle icon**.  As you move your mouse around, you should see that there are some places which are unable to be selected. When the location is selectable, a green dot appears and the cursor changes from an arrow to a hand.

Rotate the model around so you can see the outlet hole.

Select on the edge inside of the outlet hole, as shown below. This is the first point of a three point circle that will position the plane.



Select a second and third point in a clockwise fashion around the hole. When you select the third point, an axis should appear. It may not be rotated exactly the same as shown here, since it is based on your three point selection.

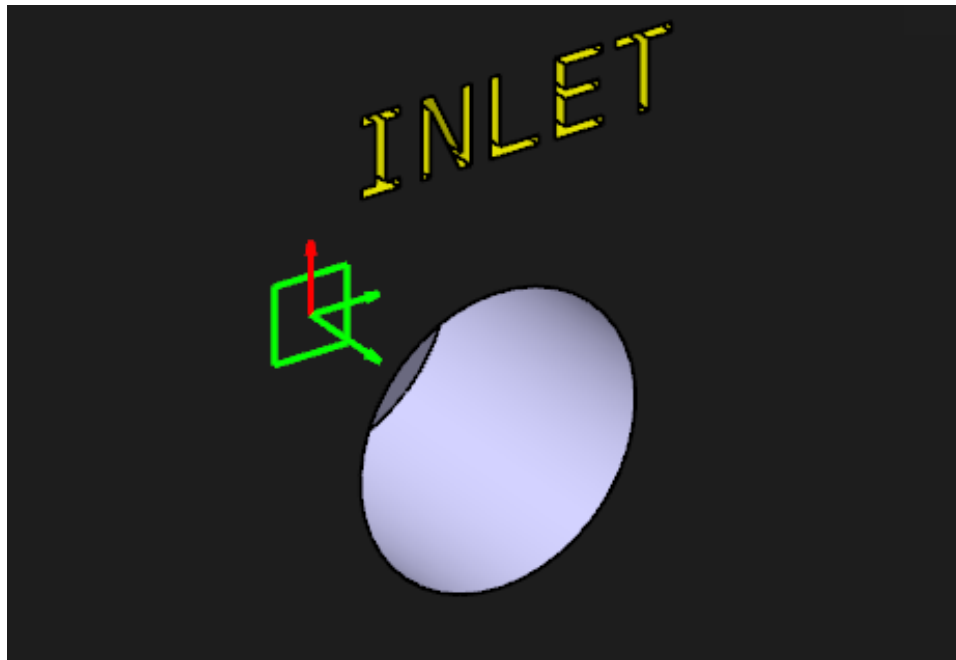


The z-direction is pointing into the part. This could be a problem.

Select **OK**. The *Add Connectors* window appears. Since this is the same type of connector as the first one created, the *Classify Connector* settings should be fine.

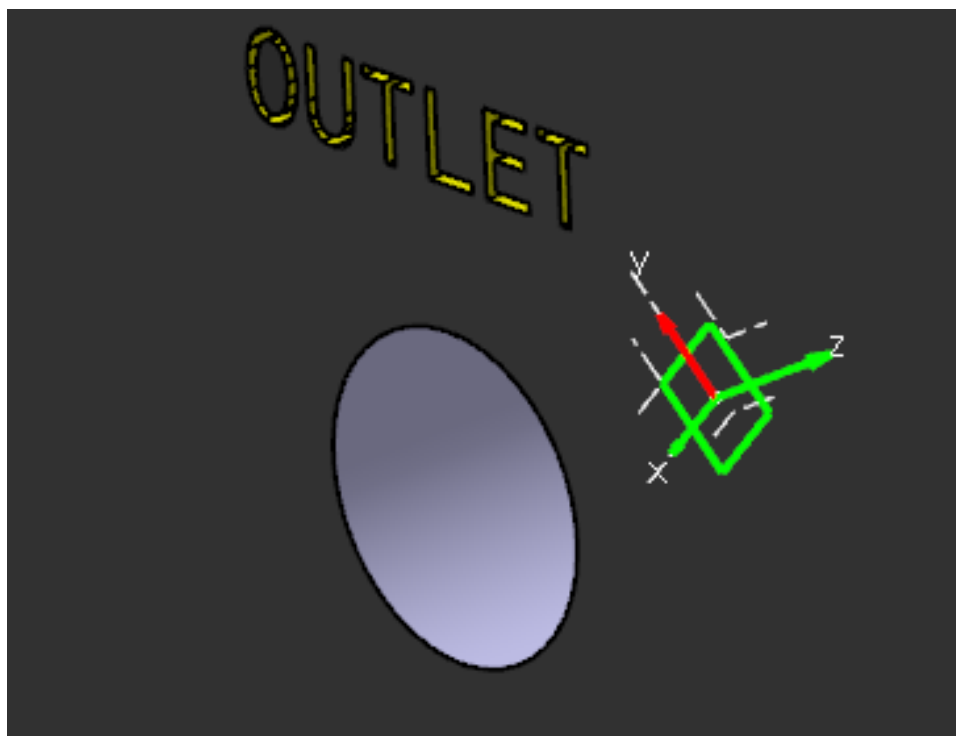
Select **OK**. The *Manage Connectors* window reappears with *Connector1* and *Connector2* listed.

Select **Connector1** from the window. Since it is on the other side you may need to rotate your view. It should appear as shown.



The plane is at the back of the hole, since this was the selected face. The origin point is also on this face. There is a green arrow pointing outward along the centerline of the hole based on the *Alignment* selection that was made. The red arrow pointing upward is the orientation arrow. When this connector is used, the element it connects to will attach to this geometry.

Select **Connector2** from the window. Rotate your model to see the connector, if necessary.



The arrow for alignment and the red arrow for orientation are pointing in different directions than the first connector. The orientation of the second connector will be roughly in the direction of the second point selected for creation. When creating a connector using this option, the right hand rule is applied. If you want the connector to face outward instead of inward, the three points should be selected counterclockwise. The direction that the connector should be facing depends on what you are connecting to. Typically, however, the z-direction will point away from the solid.

Select the *Modify* button. The *Modify Connectors* window appears. You will now adjust *Connector2* so that it points in the correct direction.

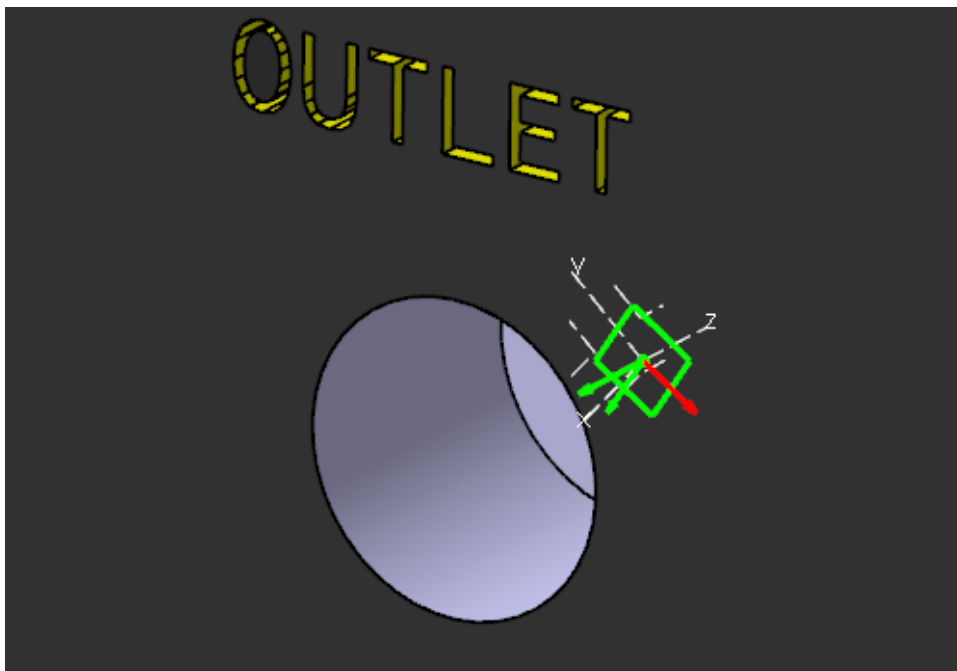
The axis is the problem and will need to be redefined. Notice that the *Use existing geometry* option is greyed out. Once the connector geometry has been defined, it can only be changed via the *Use existing geometry* option.

Click on the *Select Plane to Define Connector Geometry* icon.  The *Define Plane* window appears.

Select the *Define Plane at Center of Circle* icon. 

Select a location along the bottom of the hole as before. The last time this connection was created, the points were selected in a clockwise manner. The right hand rule was applied and the axis pointed into the solid. This time, you will select the points in a counter-clockwise order, which will point the axis in the opposite direction.


Select two more points in a counter-clockwise order around the bottom edge of the hole, then select *OK*. The *Modify Connectors* window reappears. The axis should look similar to below.



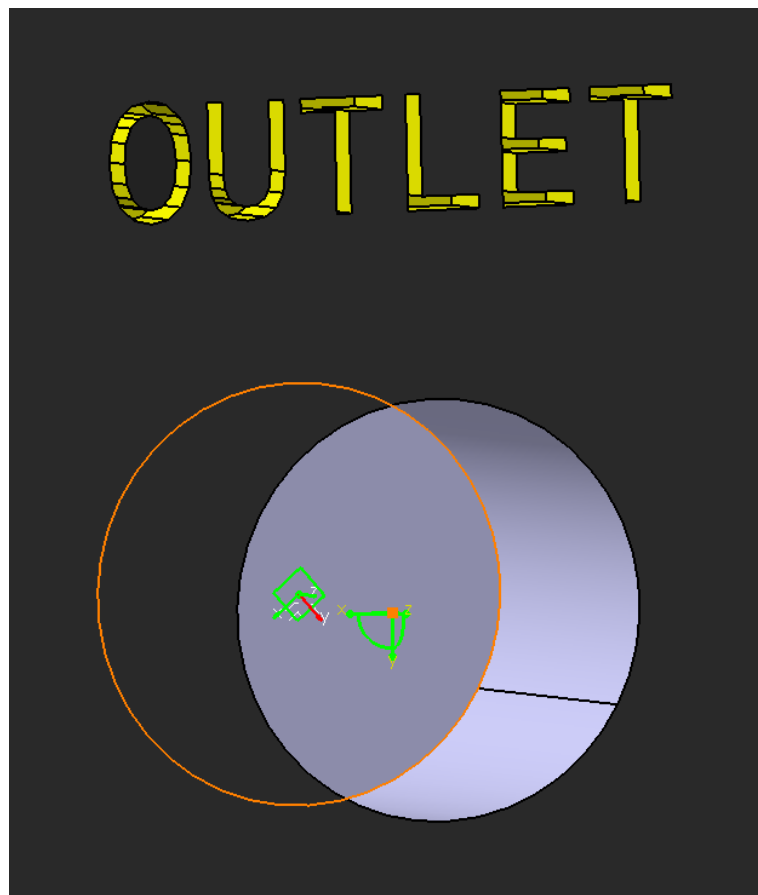
Select Connector2. The z-direction of the connector is now pointing away from the solid. However, the orientation is pointing down and to the right. If the orientation is a priority, then you will want to make sure when picking the three points that your second point is in the direction desired for orientation. It would be more accurate if the point were created at a specific location rather than selected arbitrarily, as in this example. You will use another method of defining the connector's orientation.


Select the *Modify* button again. The *Modify Connectors* window appears.

Click the **Select Plane to Define Connector Geometry icon.**  The *Define Plane* window appears.

Select the **Define Plane icon.**  This will enable you to select a plane or a planar face for the connector location.

Select anywhere on the back face of the Outlet hole as shown. A green axis system will appear with the xy plane positioned on the Outlet hole's back face. Notice the z axis is already pointing away from the solid.



Click on the **Define Origin at Center of Circle icon.** It is located in the *Move Origin* section of the *Define Plane* window.  This will force the axis origin to move to the center of the hole.

Select three points along the inside edge of the Outlet hole the same as you did before.

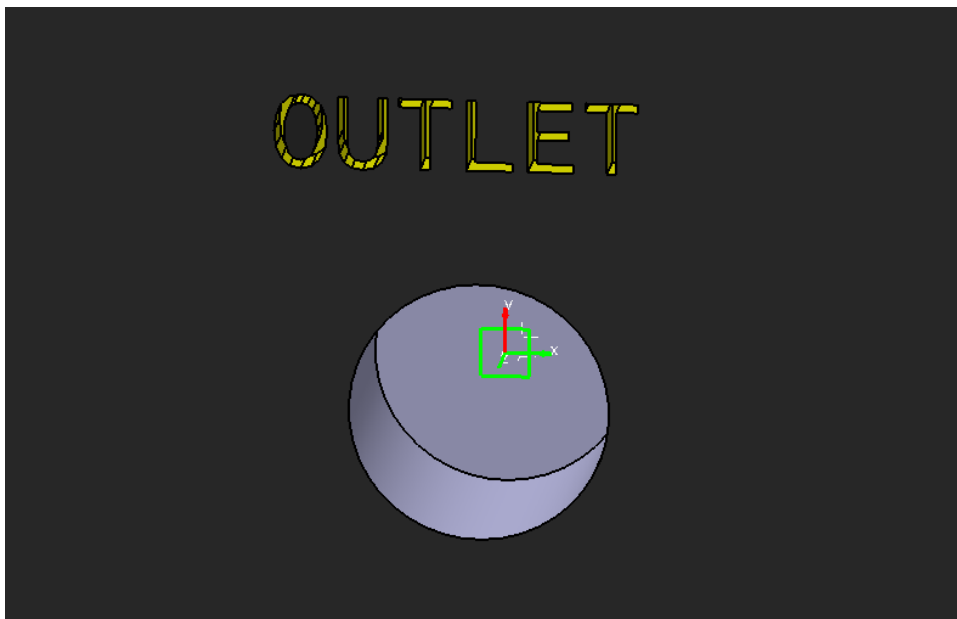
It does not matter if you select the points clockwise or counter-clockwise this time.

The local axis should have snapped to the center of the Outlet hole's back face. In this case, the orientation of the connector will match the negative z direction of the global axis, although you may not see the direction of the red arrow change until you select *Okay* to the *Define Plane* window. If you need the red arrow to match the orientation of the first connector, you can drag the xy plane arc of the local axis.

Rotate the local axis by clicking and dragging on the xy plane's arc until the y direction is pointing toward the top of the pump. The compass will rotate in 90 degree increments, so you do not need to worry about being exact. Keep dragging until it snaps to the correct location. You may need to wait several seconds for the axis to update in the graphical area.

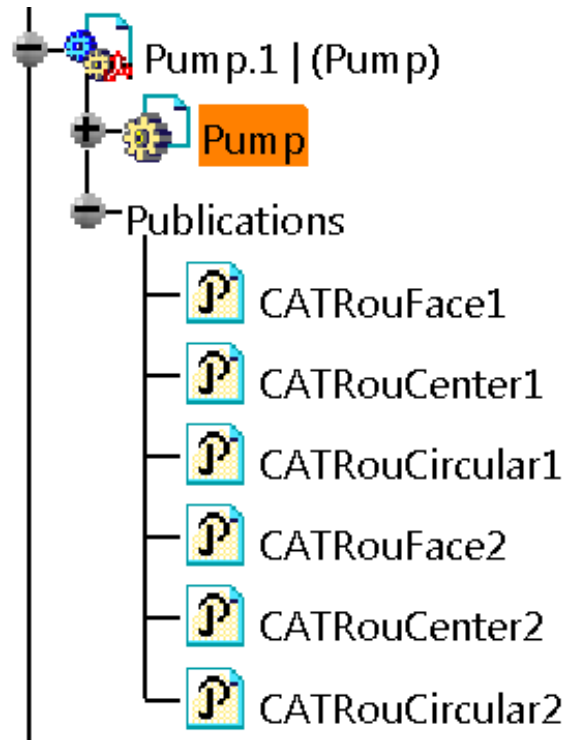
Select *OK* to both the *Define Plane* and *Modify Connectors* windows. The *Manage Connectors* window reappears.

Select *Connector2* from the list. The xy plane should be on the back face of the Outlet hole, the origin point at the center of that same face, the z axis pointing outward along the centerline of the hole, and the red arrow pointing upward. Your Outlet connector should appear as shown here.



Select *Close* in the *Manage Connectors* window.

In the specification tree, there is now a *Publications* branch under the *Pump.1* instance.



These publications are the classifications placed on the connectors, and will appear whenever a connector is created.

Save and close the document.

Piping

In Piping Design, routing is still accomplished by creating a run, then placing a pipe in it. However, there will be no flexible pipes, and the catalogs will have different parts. Piping will also have different piping lines and routing rules than Tubing Design. Pipe, when placed inside a run, will not have a part body, which is different from tubes. When switching to any workbench in *Equipment and Systems*, it is a good idea to verify that the correct project and discipline are being used.


Open the Piping document from the *Piping Exercise* directory.

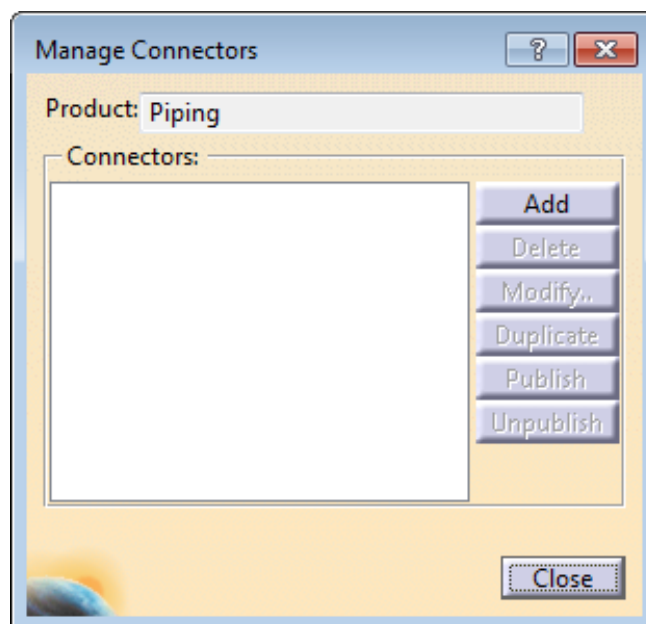
Switch to the Piping Design workbench. It is located in the *Equipment and Systems* solution.

The boiler part in this product has already been assigned electrical properties. If you look closely at the part icon in the specification tree, it has a different symbol than normal. In its properties, you will find that the part is a piece of electrical equipment. Expanding the tree for this part reveals that it has two electrical connections for bundle connection points. With the electrical properties defined, it is ready to have tubing or piping connections assigned.

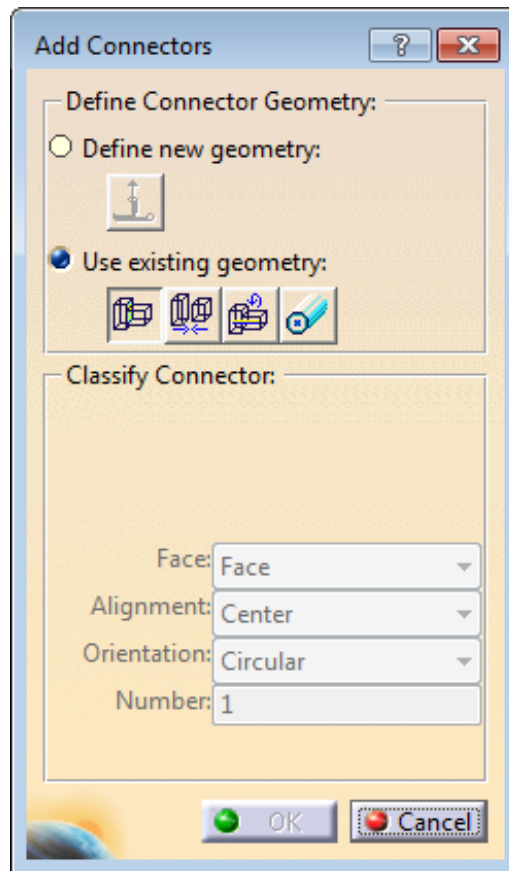
Select the black arrowhead beside the Build Piping Part icon.  The following sub-toolbar appears.




Select the Build Connector icon from the sub-toolbar.  The *Manage Connectors* window appears. It is the same as in Tubing Design.





Select **Boiler.1** from the specification tree, then select **Add**. The *Add Connectors* window will appear. When building connectors for either Tubing or Piping, the process is the same. The connector created can be used in both workbenches.



Select the **Use Existing Geometry** radio button. You will create a connector at each axis system, for a total of five connectors when finished.

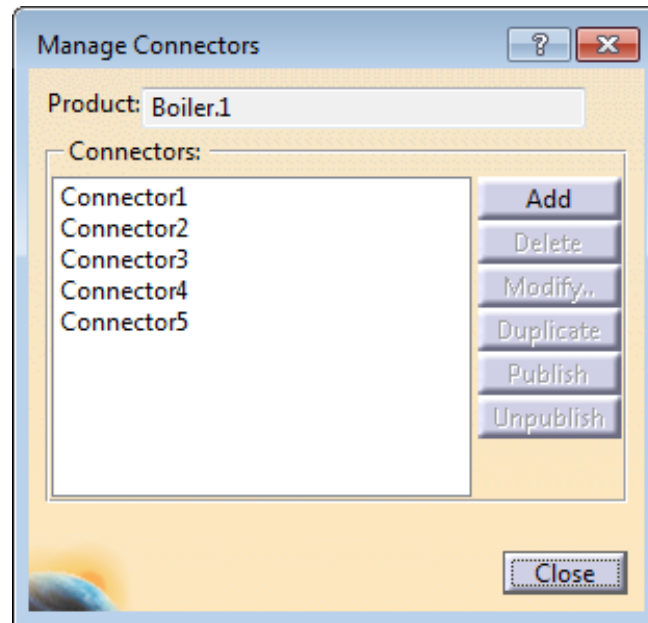
Ensure the **Select Face Connector Geometry** icon is chosen and pick the xy plane of *Axis System.1*.  This will define the support for the connector.

Click on the **Select Alignment Connector Geometry** icon and choose the z axis of *Axis System.1*.  This will define the direction of the connector.

Click on the **Select Orientation Connector Geometry** icon and choose the yz plane of *Axis System.1*.  This will define the orientation of the connector.

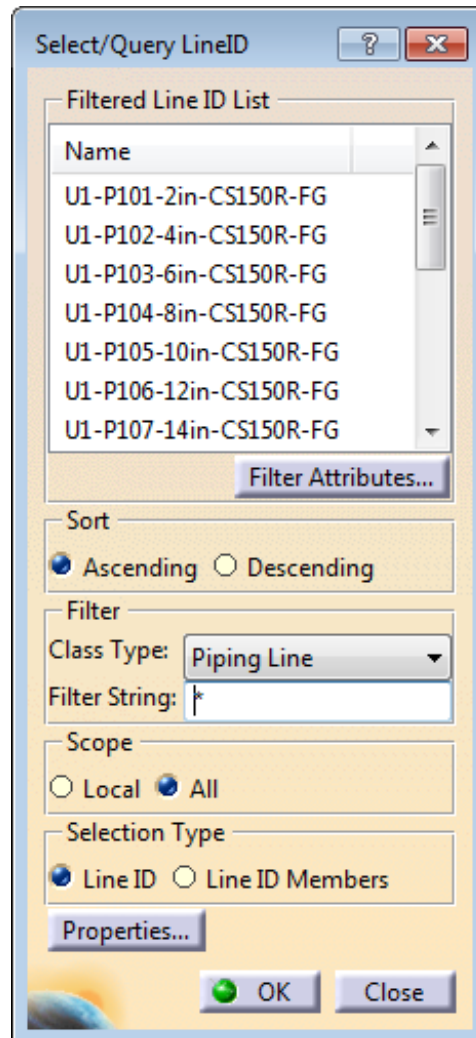
Select **OK** to the *Add Connectors* window and repeat this process for the other four axis systems.

When finished, the *Manage Connectors* window should appear as shown.




Close the *Manage Connectors* window. You will use one of the connectors to place some piping parts.

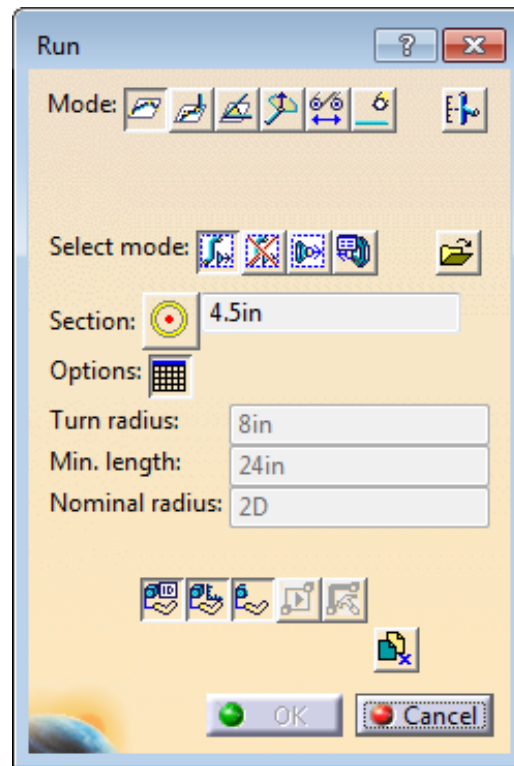
Click the **Select/Query Line ID** icon. 



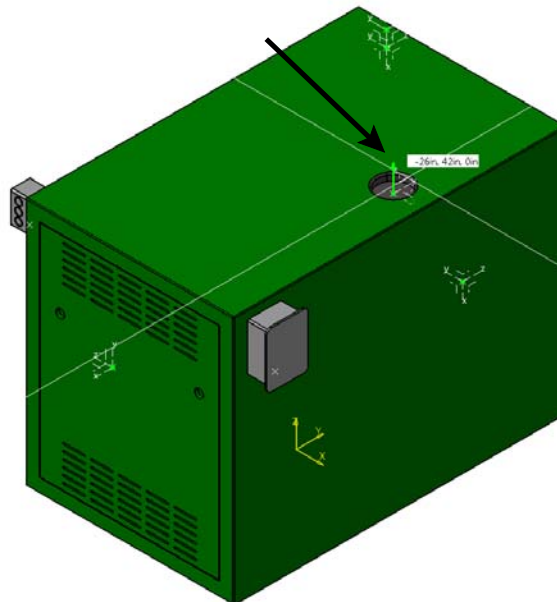
This window has the same options that are in Tubing Design.

Select the 4in line ID from the list and click *OK*. This will set the runs to have a 4in diameter.

Select the **Route a Run** icon.  The *Run* window appears. It has the same options as in Tubing Design, also.

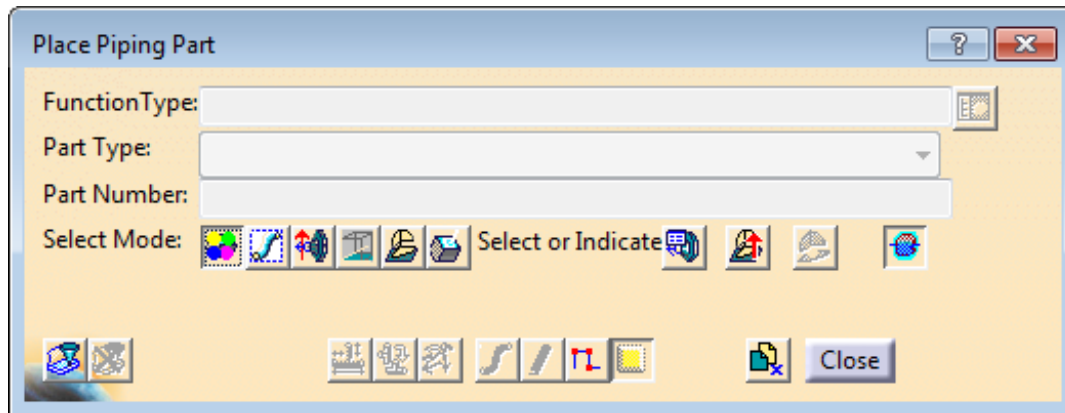


Select the connector on top of the Boiler.




Using the *Directional* mode, create a straight run up to the 58in mark. It should be around 24 inches long.

Select the **Place Piping Part** icon.  The *Place Piping Part* window appears.



Turn off the **Select Spec/NonSpec Part Placement** icon, then select in space. 
Turning off the icon changes the list of parts available. Clicking in space makes the Open Class Browser icon selectable.

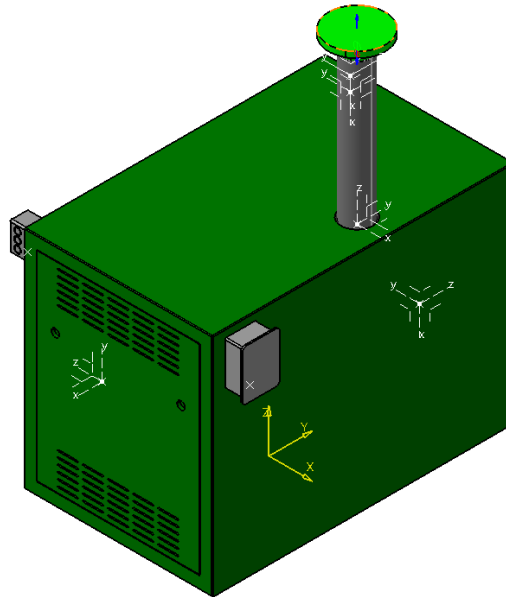
Set the *Function Type* to *Pipe Function* from the *Class Browser* window and select the top of the run.  This will locate the part that is going to be placed. A green dot appears where you selected.

Set the *Part Type* to *Flange*, then select the first part from the list in the *Part Selection* window. The part is placed.

Select the **Flip the Part** icon in the *Place Piping Part* window, then select *Close*.



The model should look the same as here. Hide the axis systems and the geometrical set in *Boiler.1* if desired.




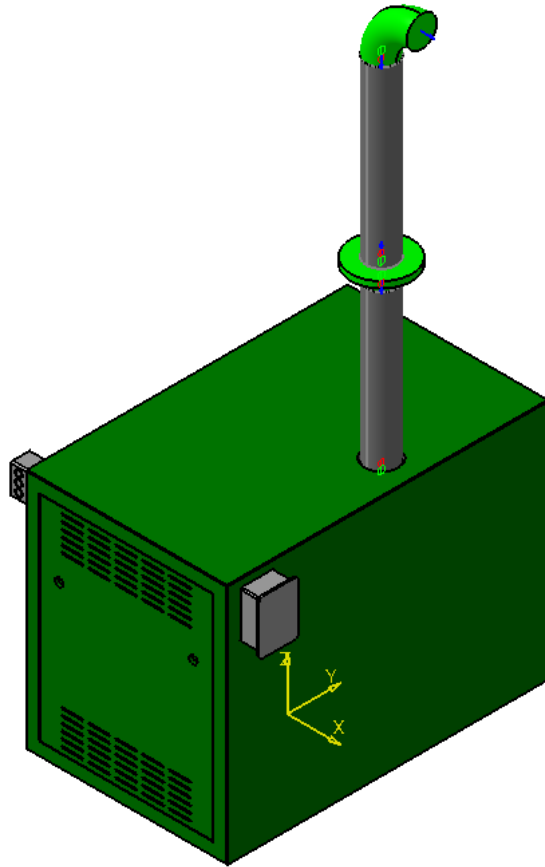
Create a run that extends up from the flange to the 88in mark.



Select the **Place Tubing Part** icon and place an *EL-90-WM-BW-LR-4in* elbow at the open end of the new run. The elbow needs to be rotated.

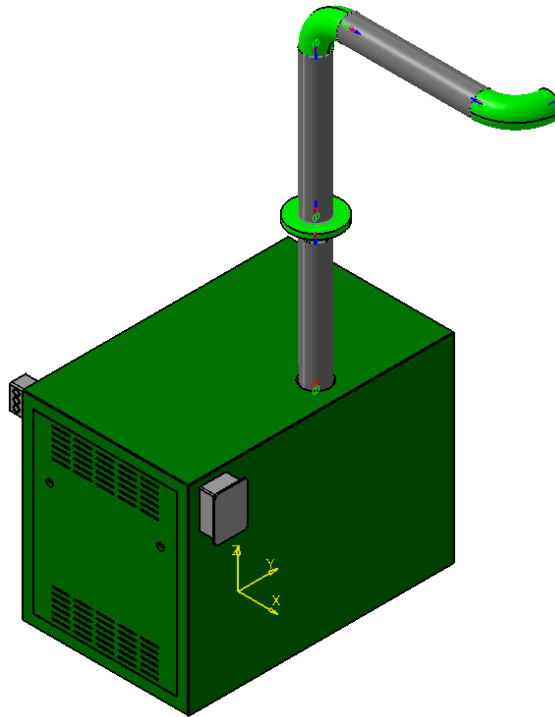


Select the **Move/Rotate the Part** icon within the *Place Piping Part* window, then rotate the elbow as shown.  The axis systems and points have been hidden here.




Create a run that leaves the elbow and goes to the 40in mark. Another run is placed.

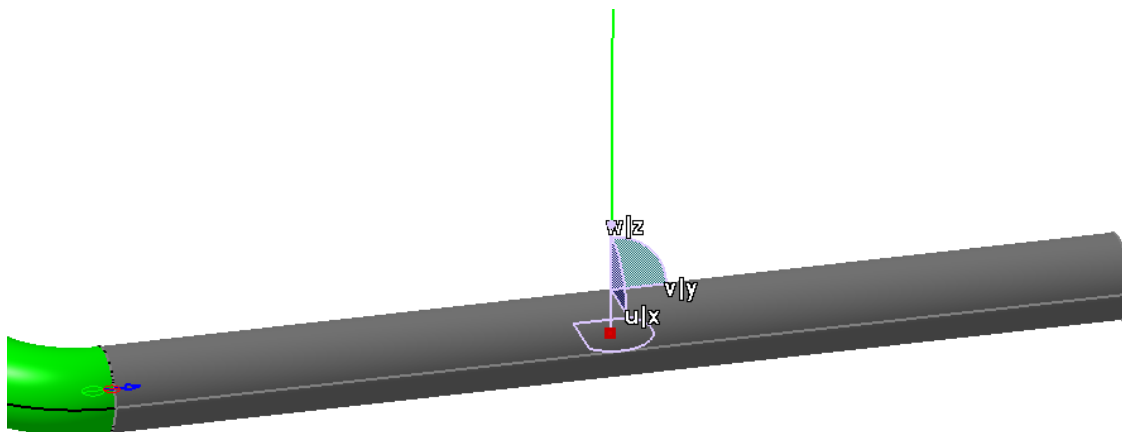
Place the same type of elbow as before at the end of the new run and rotate it as shown.



Create another run off of the new elbow extending to 64in. For the last section, a tee part will be placed. Placing a tee in Piping is the same as it is for tubing. The runs must be perpendicular. For this tee, the run will extend in the positive z direction.

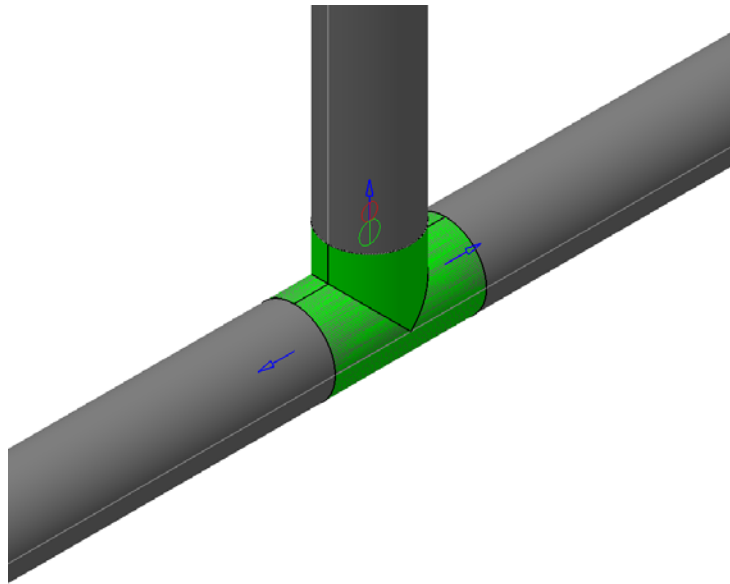
Select the **Route a Run** icon, then select on the new run somewhere close to the middle.  The compass snaps to that location.

Press the third mouse button on the compass and choose *Use Local Axis System* from the contextual menu. This is an easy way to align the compass with the part it is currently attached to. Your compass should appear as shown below.



Create a run to the 130in mark in the positive z direction. A new run is placed perpendicular to the previously created run.

Place a tee part at the intersection. Remember, tees are located in the *Branch Function* section, which is found under *Piping Part Function*. There is only one tee that will match the rules for these runs.



You may have noticed that there is only one master-slave connection on the tee, and that the tee is interfering with the bottom run. This will not be a problem, since you can break the bottom run with the tee.


Select the black arrowhead beside the **Create an Offset Route** icon.  The following sub-toolbar appears.

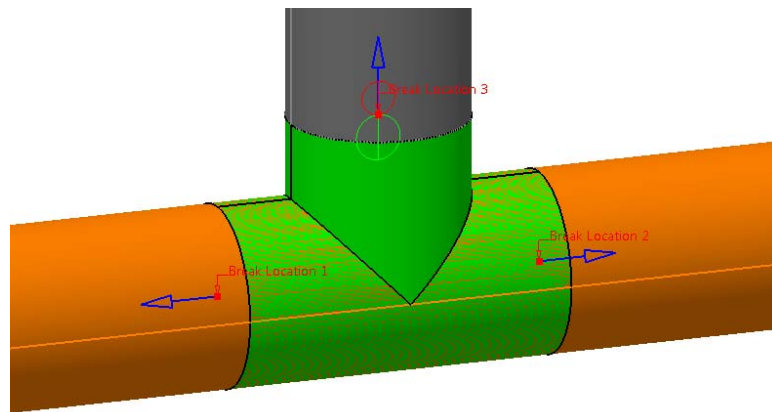


Select the **Break an Existing Run Into Two Runs** icon from the sub-toolbar. 

The *Break Run* window appears.

Click on the **Select a Run's Segment to Break** icon within in the window, then select the run that is interfering with the tee.  The run highlights orange.


Click the **Select a Part to Define Break Location** icon within the window and choose the tee part.  The run splits into two, and the *Break Location* points in the graphical area are visible.

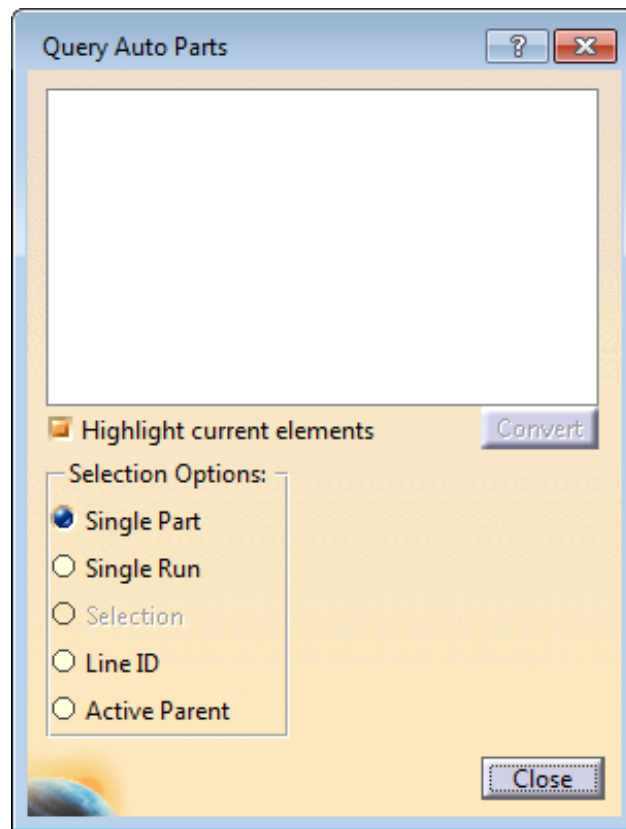


Select **OK**. There are now see three master-slave connections related to the tee part.

Place a **PIPE-BW-4in-STD** pipe in the first run you created. Pipes have a light bluish-gray color.

Place another pipe in the second run and select **Close**. This one attaches to the flange. When placed, a flange is added to the document to mate with the pre-existing flange. Tubing and piping can have rules written for automatic part placement. These can be confusing, so you will investigate the automatic part placement feature.

Select on the **Query Auto-Parts icon**.  It is located in the General Environment Tools toolbar under the sub-toolbar of the Analyze Item icon. The *Query Auto Parts* window appears.



Highlight Current Elements

Highlights the parts that have been auto-placed

Convert

Converts auto parts into non auto parts

Selection Options

Single Part

Selects the auto parts for placed parts

Single Run

Selects the auto parts for selected runs

Selection

Selects the auto parts for preselected parts

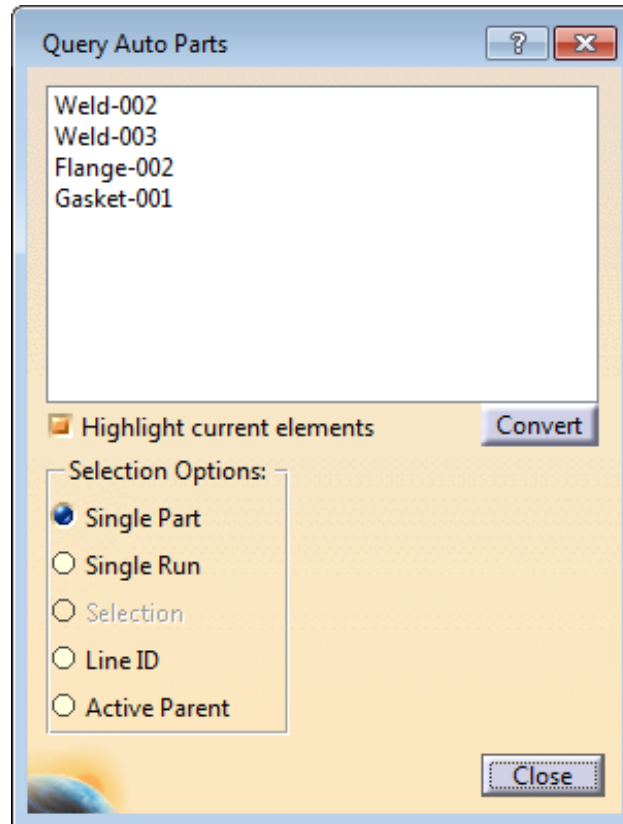
Line ID

Selects the auto parts for line IDs

Active Parent

Selects the auto parts under the active parent

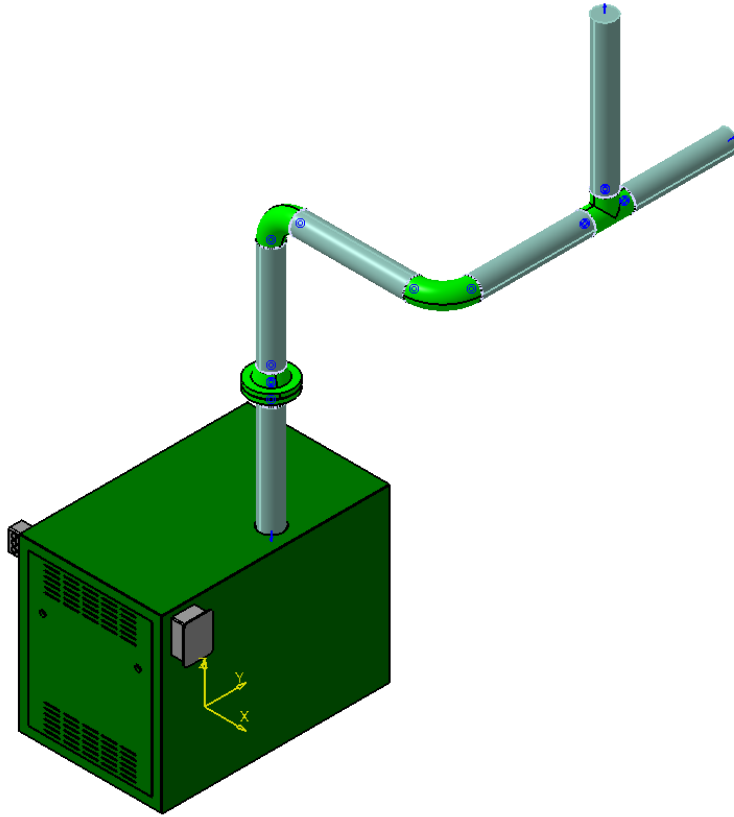
Select the pipe that was just placed.



By placing a pipe up to a flange, another flange was added. A gasket exists between the two flanges that attaches the piping parts to the pipe with welds at each end. These are the parts that were auto placed. It is a good idea to check the auto parts to make sure the correct parts have been added. When a part is deleted, if it has any auto parts associated with it, they will be deleted as well, unless they are converted. Once the part is converted, it does not show up in this window, and will stay in the model, even if the original part is deleted.

Close the *Query Auto Parts* window.

Place a pipe in all the runs using the same type as before. Here is how the model appears with the runs hidden.



Place a new product in the specification tree named **Piping Parts**. This can be done by pressing the third mouse button on the *Piping* product in the specification tree, then selecting *Components*, then *New Product*.

Select the **Transfer Elements** icon.  The piping parts are going to be transferred to the new product. The *Transfer elements* window appears.

Click on the **Select Line IDs to Transfer** icon.  The parts will be filtered by their line ID.

Select the **4in** line ID from the *Selection list* window and select *OK*. All the piping parts are selected and the *Transfer elements* window reappears.

Select the **Existing document** radio button in the *Transfer elements* window, then select **Piping Parts.1** from the specification tree. The *File Name* field is greyed out, but it displays the name of the chosen product.

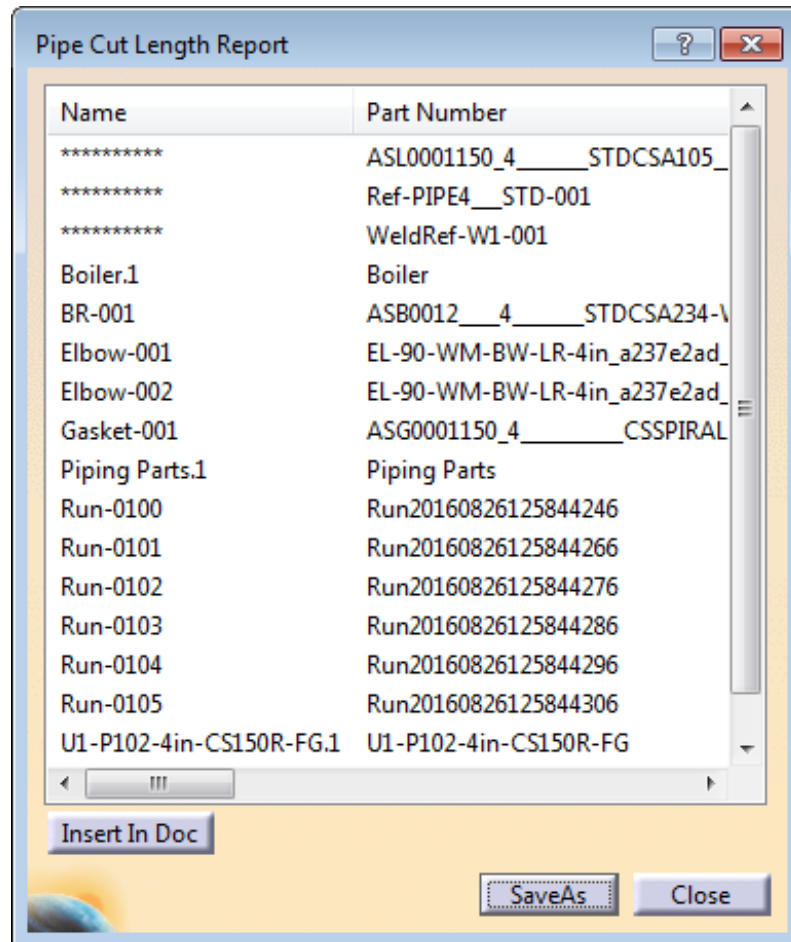
Select *OK*. All of the piping parts are now inside the *Piping Parts* product, leaving only *Boiler.1* at the top level.

Select the **Tools** pull-down menu and choose **Report**, then choose **Generate**. You will generate a report for the piping system you created.

Select the **Browse** icon. 

Open *PipingCutLengthList.xml* from the *Piping Exercise* directory. This is a predefined report.

In your *Generate a report* window, make sure that the *All objects in Document* option is selected, then choose *OK*. A report is generated.



Select *Save As* and save the report as an *.htm* file in your area.

Close the *Generate a Report* window.

Navigate to your area and open the file. It should appear similar to what is shown here.

The screenshot shows a web browser window titled 'Pipe Cut Length Report - Windows Internet Explorer'. The address bar shows the file path 'C:\Users\reneel\Desktop\New folder\Pipe Report.htm'. The main content area displays a table titled 'Pipe Cut Length Report' with the following data:

Name	Part Number	Part Name	Nominal Size	Material Category	Material Code	Parent Piping Line Name	Pipe Length	Quantity
*****	ASL0001150_4 STDCSA105	FLANGE WELDNECK	4in	Carbon steel	A105	U1-P102-4in-CS150R-FG	*****	2
*****	Ref-PIPE4 -001	PIPE	4in	Carbon steel	ASTM_A_53	U1-P102-4in-CS150R-FG	147.375in	6
*****	WeldRef.W1-001	*****	4in	*****	*****	*****	*****	9
Boiler-1	Boiler	*****	*****	*****	*****	*****	*****	1
BR-017	ASB0012_4 STDCSA234-WPB	TEE	4in	Carbon steel	A234-WPB	U1-P102-4in-CS150R-FG	*****	1
Elbow-029	EL-90-WM-BW-L_a237e2ad_93b8_54009f12_c9d	ELBOW 90 LONG RADIUS	4in	Carbon steel	A234-WPB	U1-P102-4in-CS150R-FG	*****	1
Elbow-030	EL-90-WM-BW-L_a237e2ad_93b8_5400a010_f49	ELBOW 90 LONG RADIUS	4in	Carbon steel	A234-WPB	U1-P102-4in-CS150R-FG	*****	1
Gasket-019	ASG0001150_4 CSSPIRALW_CS	GASKET RF	4in	Carbon steel	SPIRALW_CS	U1-P102-4in-CS150R-FG	*****	1
Piping Parts.1	Piping Parts	*****	*****	*****	*****	*****	*****	1
Run-1502	Run20140829110734630	*****	*****	*****	*****	*****	*****	1
Run-1503	Run20140829110734640	*****	*****	*****	*****	*****	*****	1
Run-1504	Run20140829110734650	*****	*****	*****	*****	*****	*****	1
Run-1505	Run20140829110734620	*****	*****	*****	*****	*****	*****	1
Run-1506	Run20140829110734670	*****	*****	*****	*****	*****	*****	1
U1-P102-4in-CS150R-FG.1	U1-P102-4in-CS150R-FG	*****	4in	*****	*****	*****	*****	2
*****	*****	*****	*****	*****	*****	*****	*****	*****

Save and close the model.