Smart 3D

Piping Tutorial

PROCESS, POWER & MARINE

Version 2014 R1

Monday, August 04, 2014

TSP3D-TP-100107A

PART OF HEXAGON

INTERGRAPH®

Copyright

Copyright © 2010-2014 Intergraph® Corporation. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2014. All Rights Reserved.

Portions of the user interface copyright 2012-2014 Telerik AD.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
300 Intergraph Way
Huntsville, AL 35813

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee’s internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee’s internal, non-commercial. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.
b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www.bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, http://www.treas.gov/offices/enforcement/ofac/, and the U.S. Department of State Debarred List, http://www.pmddtc.state.gov/compliance/debar.html.
c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.
d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, PDS, SmartPlant, FrameWorks, I-Sketch, SmartMarine, IntelliShip, ISOGEN, SmartSketch, SPOOLGEN, SupportManager, SupportModeler, Sapphire, and Intergraph Smart are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2013, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

Contents

Session 1

Piping: An Overview

Objective

By the end of this session, you will be able to:

- Identify the tasks that can be performed in the Piping environment.

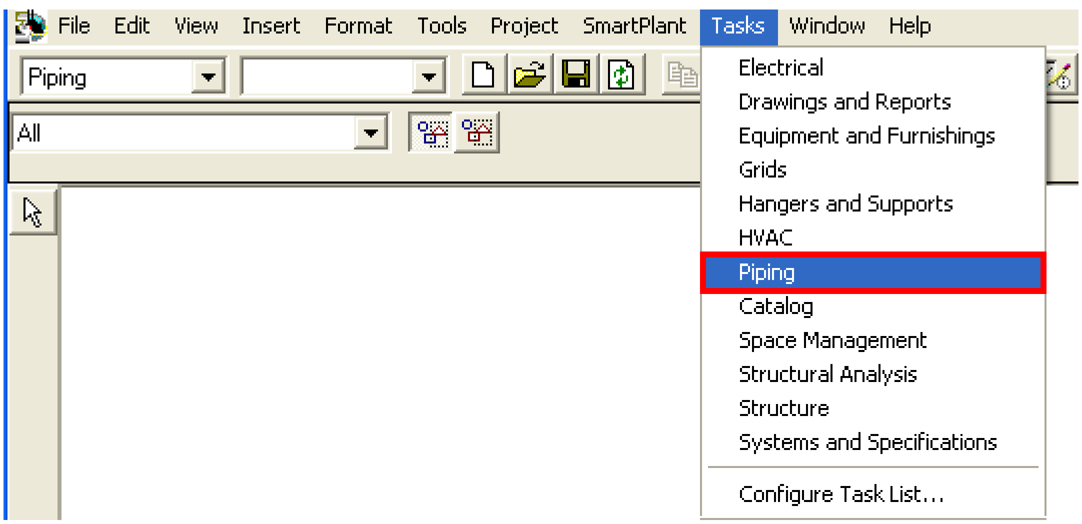
Prerequisite Session

- S3D Overview

Overview

You use the **Piping** task of Smart 3D to model distributed pipelines in your model. By using the **Piping** task, you can create a fully rendered three-dimensional model of various pipelines. The **Piping** task also helps insert piping components and instruments during design and then create spools to fabricate all the piping components as a unit.

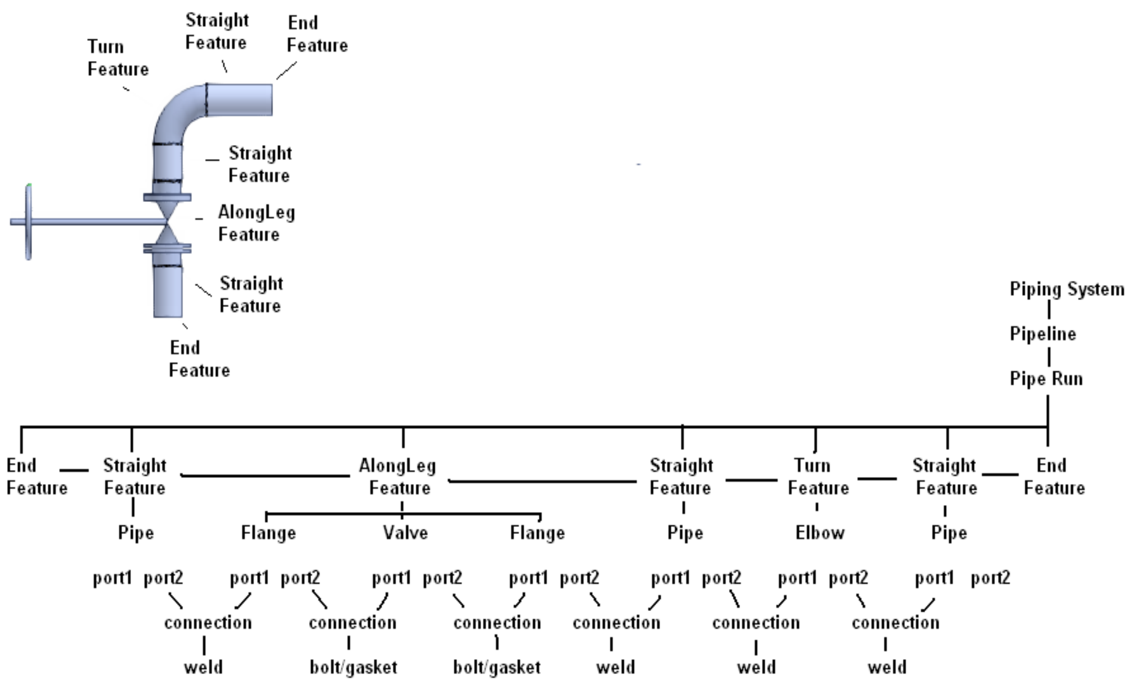
To access the **Piping** task in Smart 3D, click the **Task > Piping** command.



Pipeline modeling is aided by the piping specification, which limits and automates the selection of piping parts. Within a particular pipeline service, the specification author decides the allowed parts and their associated requirements that can be used in the service. Limiting the selection of parts through the use of a piping specification helps eliminate the need to make decisions related to the applicability, cost, procurement, and safety of the parts within pipeline services.

Pipe part selection is further aided by project rules defined in the reference data. The rules relate to the use of particular type of parts in design situations.

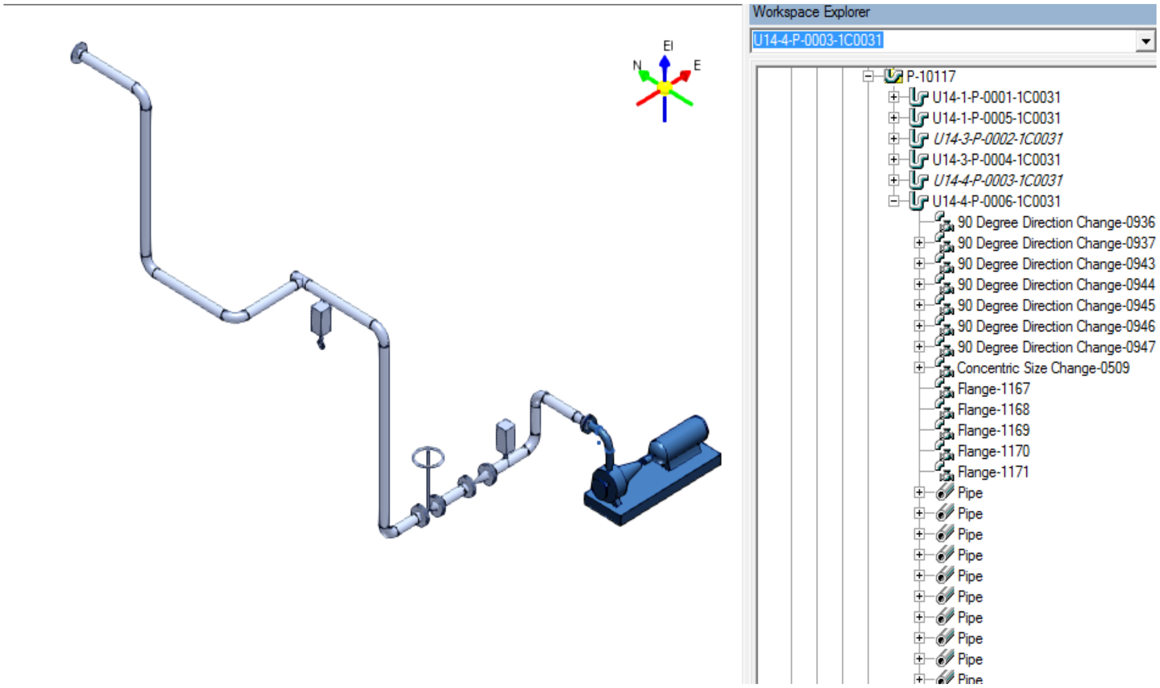
Before starting work on the piping task, you need to be familiar with the piping feature model and all the objects required to route a piping system. The figure below shows a piping feature model and the relationship among the features, which represent a section of the piping system.



A piping feature model generally consists of the following components:

- **Piping System:** A piping system is a way of organizing pipelines within the system hierarchy. You can base the piping system on the area where the pipelines are located or the fluid that the pipelines carry.
- **Pipeline System:** A pipeline system is a way of organizing pipe runs within the system hierarchy and controlling the specifications that can be used within that system. If a pipeline system exists in a model, you can route the pipe runs and arrange them as children in the system hierarchy.
- **Pipe Run:** A pipe run is a connected series of the pipe features that normally have the same nominal piping diameter (NPD) and flow direction. All the pipe runs in a model are governed by the same piping specifications. All the pipe features belong to a pipe run. One or more pipe runs together form a pipeline.
- **Pipe Feature:** A pipe feature is a logical collection of parts driven by the pipe specification. While routing a pipe run, you can place features on the pipe; these features define high-level design information. The software automatically selects the specific parts based on the pipe specification of the pipe run. Features are not displayed in the workspace explorer because of their ability to own several parts.

- **Piping Part:** A piping part is a physical component that comprises a feature and is generally selected by the software. The figure below shows some examples of pipe parts that represent a section of a piping system. The highlighted portion in the figure shows a section of the workspace explorer containing the hierarchy of the pipe parts.



- **Tap:** A tap is a small diameter port, generally screwed or socket welded on a piping component. A tap is added to a component after the component is placed in the model rather than during symbol creation. Taps are generally added to piping components to facilitate venting, drainage and connection of components such as instruments.
- **Spool:** A spool is a collection of pipe parts and welds that represent a section of the pipe to be fabricated as a unit.
- **Connection:** A connection is an object generated by the software when two objects are connected. This object generates connection parts that are not displayed graphically in the model. Examples of connection parts are bolts, nuts, washers, and gaskets.
- **Clamp:** A clamp is a connection object generated by the software when a pipe part creates a clamped connection.
- **Weld:** A weld is a connection object generated by the software when two piping parts are joined to create a welded connection.
- **Equipment:** The equipment may have one or more pipe nozzles through which connections to piping systems are made. Using the Equipment task, you can directly model equipment such as pumps, towers, and tanks in the model. You can also place the pieces of equipment from the catalog if they are available in the project.
- **Piping Nozzle:** A piping nozzle is the connection point between pipe parts and pieces of equipment. It contains the definition of the connection such as schedule thickness, end preparation, pressure rating, and size.

The Piping environment enables you to perform various tasks on the piping objects available in the **Locate Filter** drop-down list. You will now learn about the common tasks that you can perform in the **Piping** task.

Common Tasks in Piping

When in the Piping task, you can perform the following tasks by using the commands available on the vertical toolbar.

Select - Used to select objects in the model.

Route Pipe - Places pipe runs in the model.

Insert Split - Places a welded split or a take-down joint in the selected pipe.

Insert Component - Places valves, tees, and other in-line components in the selected pipe.

Insert Tap - Places a tap in the selected component.

Generate Spools - Generates piping spools for the piping in the model.

Create Penetration Spools - Generates penetration spools for pipes that have penetration plates placed in the model.

Sequence Objects - Renames the components of a pipeline or spool so that they are in a sequential order.

Group Pipe Parts - Automatically creates, names, and assigns piping objects to a Work Breakdown Structure (WBS) item.

Route Flex Pipe - Places flexible piping in the model.

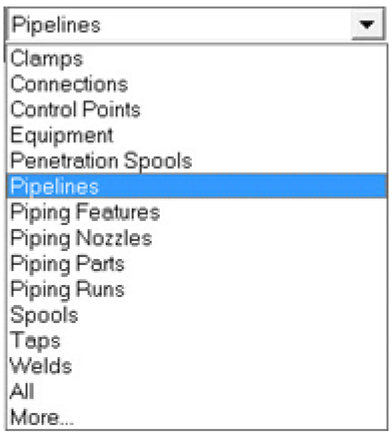
Check Manufacturability - Analyzes pipes in the current workspace and reports the objects that will be difficult or impossible to manufacture.

Following are the tasks that can be performed in the Piping environment:

- **Create and route a pipe run:** You can create a new pipe run and a branch from a pipe run, extend an existing pipe run, and route a pipe run to or from nozzles and features by using the **Route Pipe** command on the vertical toolbar.
- **Insert splits:** You can insert splits to define the limit points for insulation. You can use splits to divide a pipe run into sections by placing a set of flanges, a union component, or a clamped fitting by using the **Insert Split** command on the vertical toolbar.
- **Insert pipe components and custom instruments or specialty items:** You can insert pipe components to create sophisticated pipe layouts that divide, branch, and convey fluids between equipment. While adding pipe components, Smart 3D might add the necessary mating parts. The **Insert Component** command adds inline components, reducing components, and other components to a pipe run.

- **Add taps on pipe components:** You can use the Insert Tap command to add taps to standard components such as caps, instruments, orifice flanges, and valves.
- **Generate spool assemblies:** You can create spool assemblies by applying a set of rules that breaks the pipeline into several pipe sections. You can then use these assemblies to create a detailed spool drawing. Spools can be created by using the **Generate Spools** command.
- **Order the welds on pipelines or spools sequentially:** You can select the rule in which the welds are sequenced on pipelines or spools by using the **Sequence Objects** command.
- **Create, name, and assign piping objects to a WBS item:** You can select rules from a set of rules that control the grouping behavior and creation of WBS items. WBS groupings can be used to drive the creation of isometric drawings by using the **Group Pipe Parts** command.

The objects with which you work in the Piping task can be located in a model by using the Locate Filter drop-down list.



SESSION 2

Routing Pipes

Objective

By the end of this session, you will be able to:

- Identify the default properties of pipe runs in a system.
- Route a pipe.

Prerequisite Session

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)

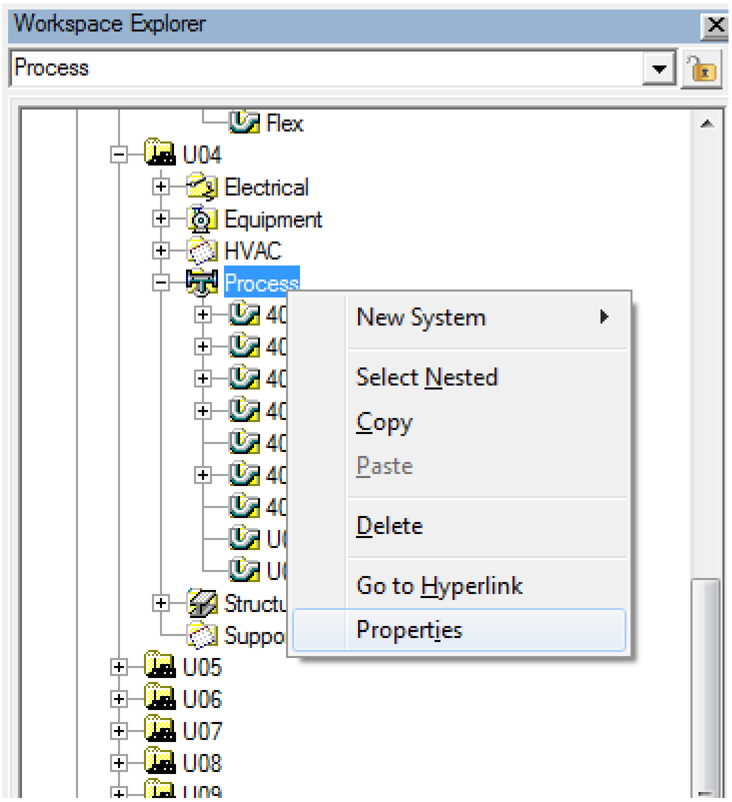
Overview

In Smart 3D, when you route a pipe and create a new pipe run in a model, you need to set the default or common properties of the new pipe run at the system level. Default properties for the new pipe run can be defined from any one of the following:

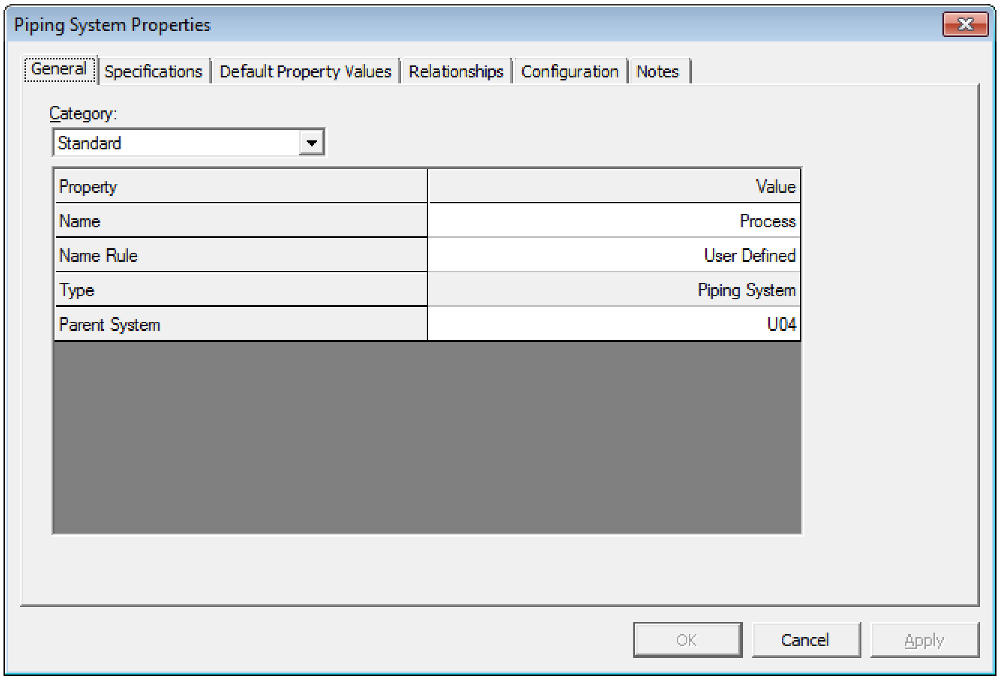
- The pipe run to which you are connecting
- A P&ID in an integrated environment
- The default properties of the parent piping system defined in the **Systems and Specifications** task of Smart 3D.

If these properties are defined in a piping system, then the entire pipe run modeled under piping system will inherit these properties.

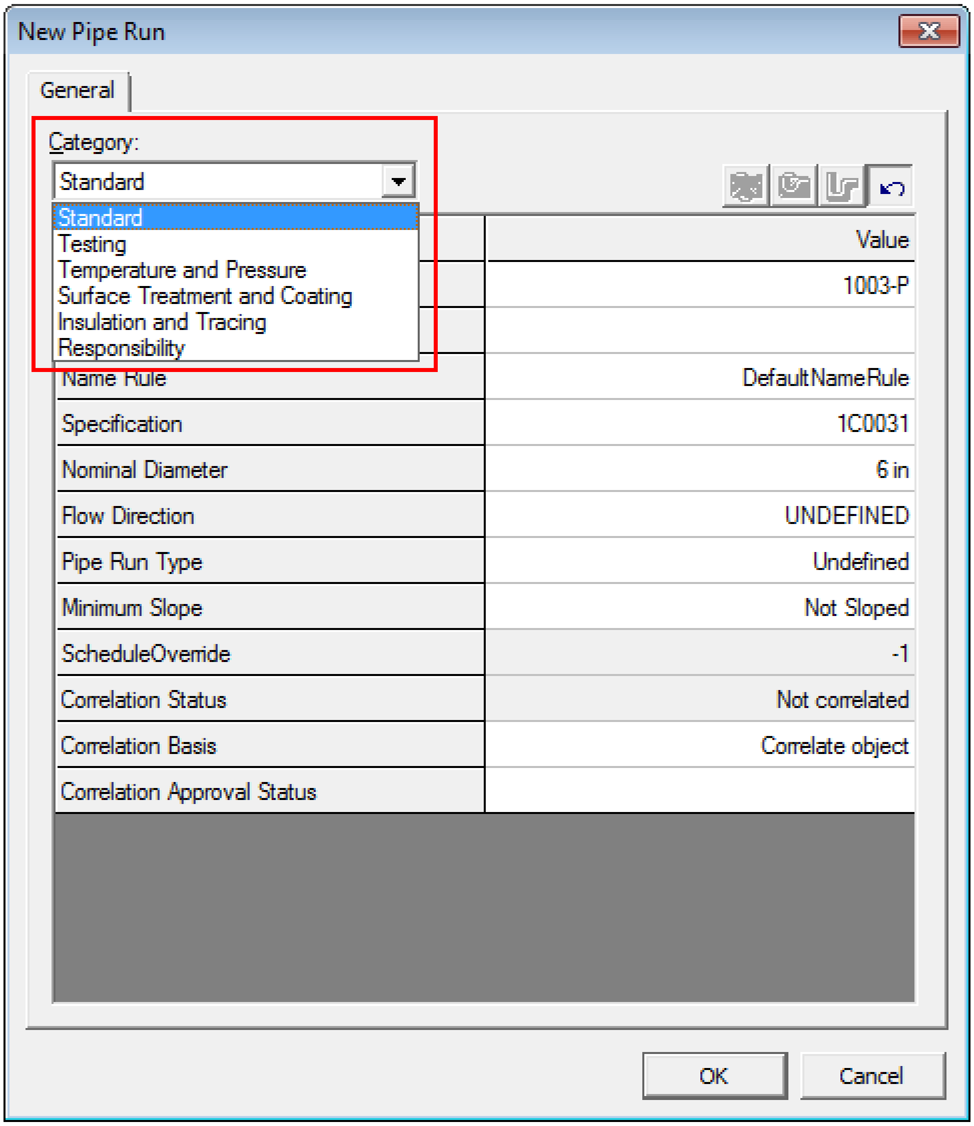
To access the properties, you right-click the **Process** system in the **Workspace Explorer** and select the **Properties** command, as shown below.



The **Piping System Properties** dialog box appears where the default properties of a pipe run are defined.



The **New Pipe Run** dialog box appears when the user creates a new pipe run. You can view and make changes to the default properties of a pipe run by using the **New Pipe Run** dialog box.



You can select the category for which you want to define values by using the Category drop-down list in the New Pipe Run dialog box, as shown above.

Pipe run properties are divided into different categories as follows:

- Standard
- Testing
- Temperature and Pressure
- Surface Treatment and Coating
- Insulation and Tracing
- Responsibility

New Pipe Run Dialog Box

Creates a new pipe run in your model using properties that you specify. You can access the **New Pipe Run** dialog box by going to the Piping task, selecting the **Route Pipe** command, and then selecting **<New Pipe Run>** from the **Run** option on the ribbon bar.

Defaults for the new pipe run can be inherited from the pipe run you are connecting to, from a P&ID in an integrated environment, or from the grandparent piping system default properties values as defined in the Systems and Specifications task.

Pipe run properties are divided into several different categories: **Standard**, **Testing**, **Temperature and Pressure**, **Surface Treatment and Coating**, **Insulation and Tracing**, and **Responsibility**. You can select the category that you want to define values for by using the **Category** option.

Property Defaults Source

These four commands indicate and control the default properties for the new pipe run. If you click a command that is active, the default properties of that system or object are used. Default properties from the design basis have the highest priority.

Piping System

Uses the default properties on the parent piping system.

Pipeline System

Uses the default properties on the parent pipeline system.

Pipe Run

Uses the default properties from an existing pipe run.

Last Used

Uses the last saved properties for the pipe run.

Standard

Pipeline

Select the pipeline system to which you want the pipe run to belong. The last pipeline system that you selected is the default. Select **More** to display all defined pipeline systems. You can create a new pipeline in the Systems and Specifications task.

Name

Displays the name of the pipe run. The pipe run name is based on the **Name Rule** selection. If you specify a name in this box, the software automatically sets the **Name Rule** box to **User Defined**.

Name Rule

Specify the naming rule to name this pipe run. You can select one of the listed rules or specify the pipe run name yourself in the **Name** box.

Specification

Select the pipe specification that you want to control this pipe run. Only those pipe specifications that are allowed in the pipeline that you selected appear. You assign pipe specifications to pipeline systems in the System and Specifications task. You define pipe specifications in the reference data. See the *Piping Reference Data Guide* for more information on defining pipe specifications.

Nominal Diameter

Select the nominal pipe diameter (NPD) for the pipe run. The pipe specification controls the available NPDs in this list. If you select an equipment nozzle as the starting point of your pipe run, the software automatically uses the NPD of the nozzle as the NPD of the pipe run. You can select the NPD units to display on a session file basis using the **Tools** > **Options** command on the **Units of Measure** tab.

Flow Direction

Select the flow direction for the pipe run. If you select an equipment nozzle as the starting point of your pipe run, the software automatically uses the flow direction of the nozzle as the flow direction of the pipe run.

Pipe Run Type

Specifies the type of pipe run that is routed. Select **Core Pipe Run** to route a core pipe run. Select **Jacket Pipe Run** to route a jacketed pipe run. The default value is **Undefined** for a non-jacketed pipe run. This property is used to differentiate a core pipe run and a jacketed pipe run.

Minimum Slope

Displays the slope for the pipe run. You can specify the slope as a ratio, a percentage, or in degrees.

- As a ratio, if a run drops 1/4" for every foot of horizontal distance, specify the slope as **1/4"/1'-0"**.
- As a percentage, if a run drops 1" for every 10" of horizontal distance, specify the slope as **10%**.
- In degrees, a five-degree slope can be specified as **5deg**.

Slope Direction

Displays the direction type of the pipe run slope. If it is set to **Single**, the pipe run slopes either up or down. You can change this to **Multiple** if needed. If it is set to **Multiple**, the pipe run can slope both up and down. You cannot change this setting back to **Single** once **Multiple** is selected.

Schedule Override

Displays the thickness override for parts in the pipe run. If the piping specification rules do not allow you to override components, then the software disables this option.

Correlation Status

Displays whether the pipe run has been correlated to a pipe run in a P&ID.

Correlation Basis

Displays if the pipe run is required to be correlated to a P&ID pipe run. Select **Correlate object** if the pipe run has a correlating pipe run in a P&ID. Select **No correlation is required** if the pipe run does not have a correlating pipe run in a P&ID.

Correlation Approval Status

Displays if the component is approved with discrepancies in the three-dimensional data compared with design basis data. Select **Topology mismatch approved** if the topology mismatch of components can be ignored. Select **Data and Topology mismatches approved** if the data and topology mismatches of components can be ignored. Select **None** if you do not want to approve a mismatch.

Testing

Testing Requirements

Displays the type of non-destructive testing required for welds. To change the options on the list, edit the **Testing Type** select list in Catalog.

Testing Type

Displays the type of non-destructive weld testing for the welds. To change the options on the list, edit the **Testing Type** select list in Catalog.

Testing Percentage

Displays the percentage of the welds on this pipe run that are to be tested. This option is available only if you are viewing pipe run properties.

Temperature and Pressure

NOTE If service limits have been defined, then you must define one set of pressure/temperature values (**Design Maximum Temperature** and **Design Maximum Pressure**, for example) before you can place the object. The software does not prevent you from defining testing temperatures and pressures that exceed the defined service limits. Temperature and pressure values are controlled by a Project option. If the Project option is **On**, and you should key in the temperature and pressure value for all the pipe runs. If the Project option is **Off**; you will receive a warning, "The temperature and pressure values are ignored."

Design Maximum Temperature

Displays the maximum design temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit or **C** for Celsius for example).

Design Minimum Temperature

Displays the minimum design temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit or **C** for Celsius for example).

Design Maximum Pressure

Displays the maximum design pressure. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Operating Maximum Temperature

Displays the maximum operating temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius for example).

Operating Minimum Temperature

Displays the minimum operating temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit or **C** for Celsius for example).

Operating Maximum Pressure

Displays the maximum operating pressure. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Testing Maximum Temperature

Displays the maximum testing temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit or **C** for Celsius for example).

Testing Minimum Temperature

Displays the minimum testing temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius for example).

Testing Maximum Pressure

Displays the maximum testing pressure. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Design Minimum Pressure

Displays the minimum design pressure. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Operating Minimum Pressure

Displays the minimum operating pressure. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Testing Minimum Pressure

Displays the minimum testing pressure. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Surface Treatment and Coating

Interior Surface Treatment Requirement

Displays the interior treatment requirement for the object. To change the options on the list, edit the **Interior Surface Treatment** select list in Catalog.

Interior Surface Treatment Type

Displays the interior treatment type for the object. To change the options on the list, edit the **Interior Surface Treatment** select list in Catalog.

Exterior Surface Treatment Requirement

Displays the exterior treatment requirement for the object. To change the options on the list, edit the **Exterior Surface Treatment** select list in Catalog.

Exterior Surface Treatment Type

Displays the exterior treatment type for the object. To change the options on the list, edit the **Exterior Surface Treatment** select list in Catalog.

Cleaning Requirement

Displays the cleaning requirement for the object. To change the options on the list, edit the **Cleaning Requirement** select list in Catalog.

Steamout Requirement

Displays the steamout requirement for the object. To change the options on the list, edit the **Steamout Requirement** select list in Catalog.

Steamout Pressure

Displays the steamout pressure for the object. Include the pressure unit of measure (**psi** for pounds per square inch for example).

Steamout Temperature

Displays the steamout temperature for the object. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius for example).

Auxiliary Treatment Requirement

Displays the auxiliary treatment requirement. To change the options on the list, edit the **Auxiliary Treatment** select list in Catalog.

Auxiliary Treatment Type

Displays the auxiliary treatment type for the object. To change the options on the list, edit the **Auxiliary Treatment** select list in Catalog.

Interior Coating Requirement

Displays the interior coating requirement for the object. To change the options on the list, edit the **Coating Type** sheet in the AllCodeLists.xls workbook in the reference data.

Interior Coating Type

Displays the type of interior coating for the object. To change the options on the list, edit the **Coating Type** sheet in the AllCodeLists.xls workbook in the reference data.

Interior Coating Area

Displays the area of the interior coating for the object.

Exterior Coating Requirement

Displays the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Displays the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Area

Displays the area of the coating for the object.

Coating Color

Displays the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Insulation and Tracing

Insulation Specification

Displays the insulation specification for the object. If **Inherited Properties or Spec**  displays in the box, the insulation specification for the object is inherited from the pipe run to which the object belongs.

Defines which insulation specifications are available for which systems in System and Specifications. Insulation specifications are defined in the reference data.

- **Not Insulated** disables the remaining insulation properties.
- **User Defined** indicates that you can specify the remaining insulation properties using custom settings.
- **More** displays a dialog box so that you can select an insulation specification from the reference data. The software then automatically enters information from that insulation specification.

Insulation Purpose

Displays the purpose of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Purpose** box. However, if **Insulation Specification** is manually defined, you can select the purpose of the insulation in this box. If **Insulation Specification** is a selected reference data insulation specification, the software reads the purpose from the insulation specification from the reference data and displays it here.

To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Material

Displays the material of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Material** box. However, if **Insulation Specification** is manually defined, you can select the insulation material to use. If **Insulation Specification** is a selected reference data insulation specification, the software reads the material from the insulation specification as defined in the reference data and displays it here.

To change the options on the list, edit the **Insulation Material** select list in Catalog.

Insulation Thickness

Displays the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data. If the value in the **Insulation Specification** field is a selected reference data insulation specification, the software reads the thickness from the insulation specification from the reference data and displays it here.

Insulation Temperature

Displays the temperature. Include the temperature unit of measure (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius for example) when specifying this value.

Heat Tracing Requirement

Indicates whether or not the equipment is heat-traced. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Type

Displays the type of heat-tracing. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium

Displays the heat-tracing medium to apply to the run. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium Temperature

Displays the temperature of the heat-tracing medium. Include the unit of measurement of temperature (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius, for example).

Responsibility

Cleaning Responsibility

Displays the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Displays the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Displays the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Displays the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Displays the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Displays the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

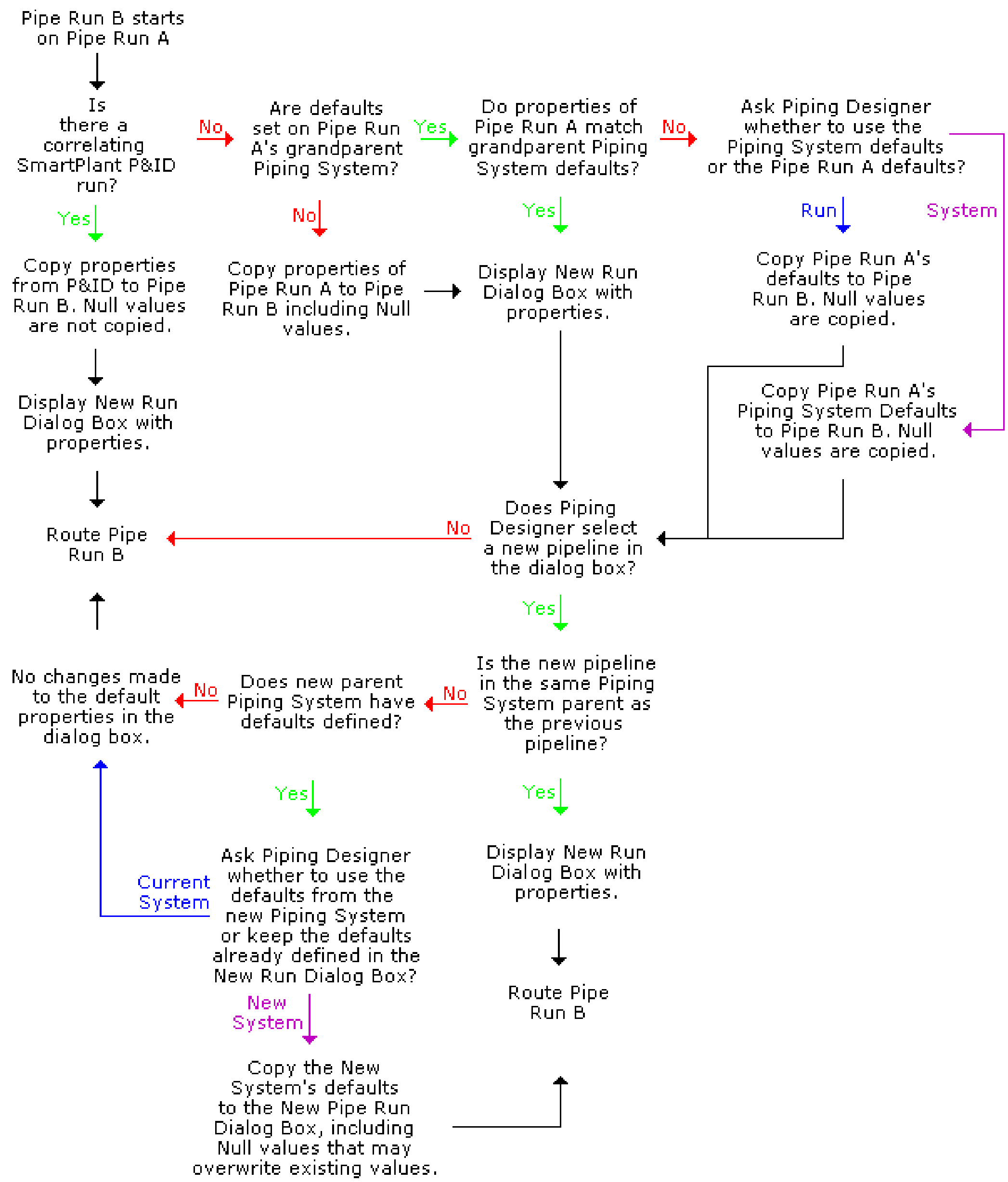
Supply Responsibility

Displays the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Displays the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

How the Software Evaluates Defaults



Defaults Tab (Route Pipe Settings Dialog Box)

The **Defaults** tab is only displayed when you are actively routing a pipe run and you click the **Settings** option on the ribbon bar.

Minimum Straight Length - Displays the minimum allowed length of straight pipe.

Straight Option - Select the option code associated with the pipe that you want to use while routing. Only those option codes defined in the controlling piping specification appear.

Turn Option - Select the option code associated with the turn component that you want to use while routing. Only those option codes defined in the controlling piping specification and shared by all of the short codes that can be placed automatically *while* routing appear. Only the options available for all short codes available during routing are listed. If pipe spec writer only added a second choice for 90 deg direction change, then route would not display it in the drop down list.

Branch Type - Select the option code for the branch type that you want to use while routing.

Automatic Angle Lock On - Specifies if the Angle lock should automatically lock when you type a new angle value into the **Angle** box.

Bend Radius Multiplier - Specify the bend radius multiplier for bends. This overrides the default bend radius multiplier defined in the piping specification. This field is not available unless the **Pipe Bend Radius By User Option** is set to **Pipe Bend Radius By User Is Enabled** in the Catalog task. You can find the option in Catalog under the **Piping > Piping Specifications > Plant Options** node in the tree view.

Reference 3D Tab

Displays Reference 3D object properties. You can modify these properties as needed.

NOTE This tab is available only when you select a Reference 3D object.

Port ID

Displays the name of the Reference 3D object port.

Nominal Size

Enter the nominal size of the port.

NPD Unit Type

Enter the units for the **NPD** column. For example, type **mm** (millimeters) or **in** (inches).

Termination Class

Specifies the termination class for the nozzle.

Termination Sub Class

Specifies the termination sub-class for the nozzle.

End Preparation

Enter the end preparation code for the part port. Select a valid code from the drop-down list.

End Practice

Specifies the end practice for the nozzle.

End Standard

Enter the end standard code for the part port. Select a valid code from the drop-down list.

Rating Practice

Specifies the rating practice for the nozzle.

Pressure Rating

Enter the pressure rating code for the part port. Select a valid code from the drop-down list.

Schedule Practice

Specifies the schedule practice for the nozzle.

Schedule Thickness

Enter the schedule thickness short description or code for the part port. Select a valid code or short description from the drop-down list.

Steps for Deleting Existing Pipelines

This session will cover the procedure for routing pipelines in Smart 3D.

Before going through this procedure and the subsequent Smart 3D Piping sessions, define your workspace to include all objects in the SP3Dtrain model database:

1. Start the software by selecting **Start > Programs > Intergraph Smart 3D > Smart 3D**.
2. In the **New** dialog box, select the EnglishUnits or MetricUnits template, and then click **OK**.
3. Select the **File** menu, and select the **Define Workspace** command.
4. In the **Filter** drop-down list of the **Define Workspace** dialog box, select the **More…** option.
5. In the **Select Filter** dialog box, select **All** under **Plant Filters** and click **OK**.
6. Select the **View > Fit** command.

   NOTE The next steps are only necessary to perform if you did not go through the instructions to delete equipment and piping objects in the Smart 3D Equipment Sessions. Otherwise, perform these steps to delete existing piping objects from the workspace before starting the session.

   Use the filter mechanism to select the existing modeled objects.

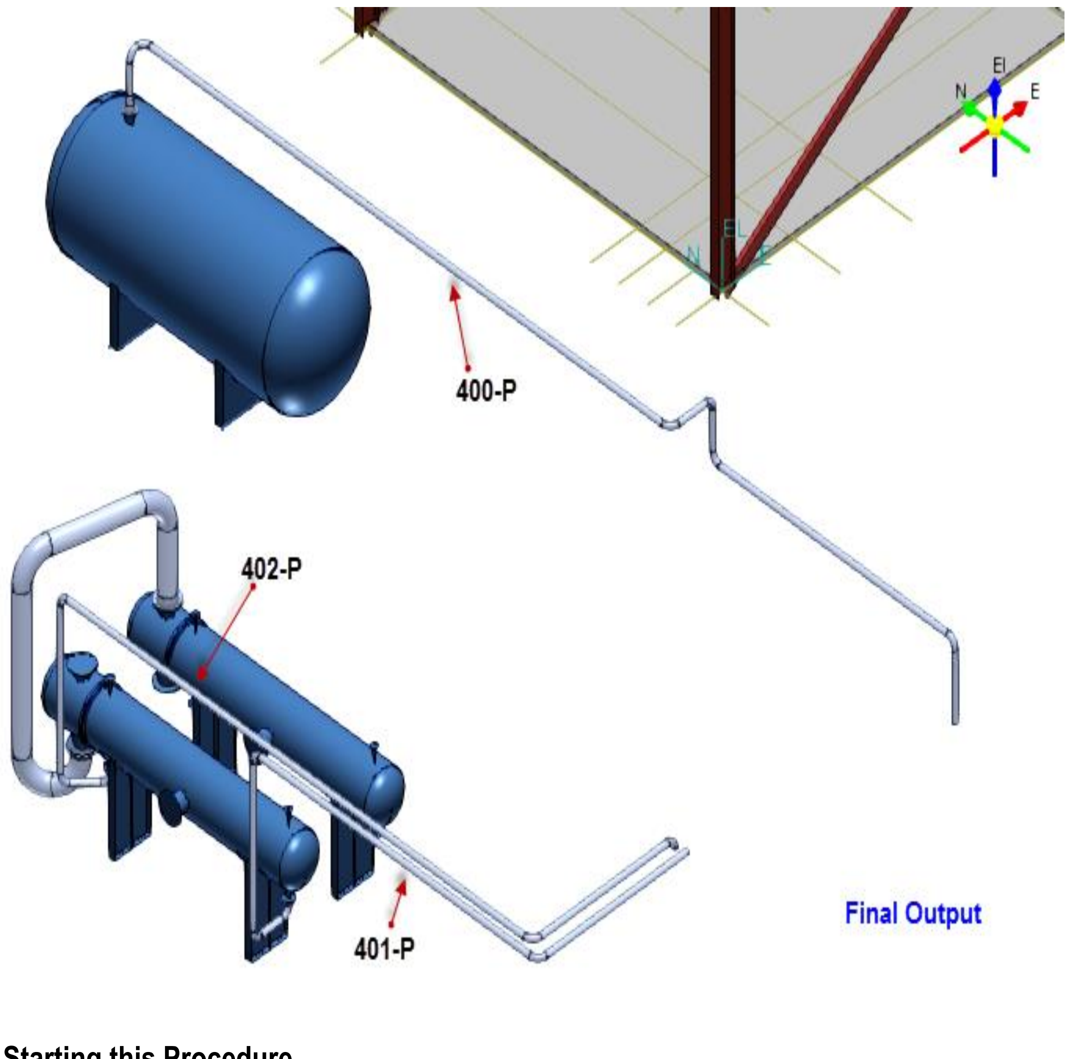
7. Select the **Tools > Select by Filter** command to open the **Select Filter** dialog box.
8. Select and expand the **For Instructors Only** folder.
9. Select the **Piping Tutorial Session - Select and Delete** filter, and click **OK**.
10. Click Delete to delete the selected objects.

Steps for Basic Pipe Routing

Exercise Objective

In this exercise you will be routing pipelines from the mentioned below equipment nozzles to their completion in a plant by using SmartSketch, Work Plane Control, and Length Control tools in Unit U04 of your workspace. After routing, the pipelines should resemble this.

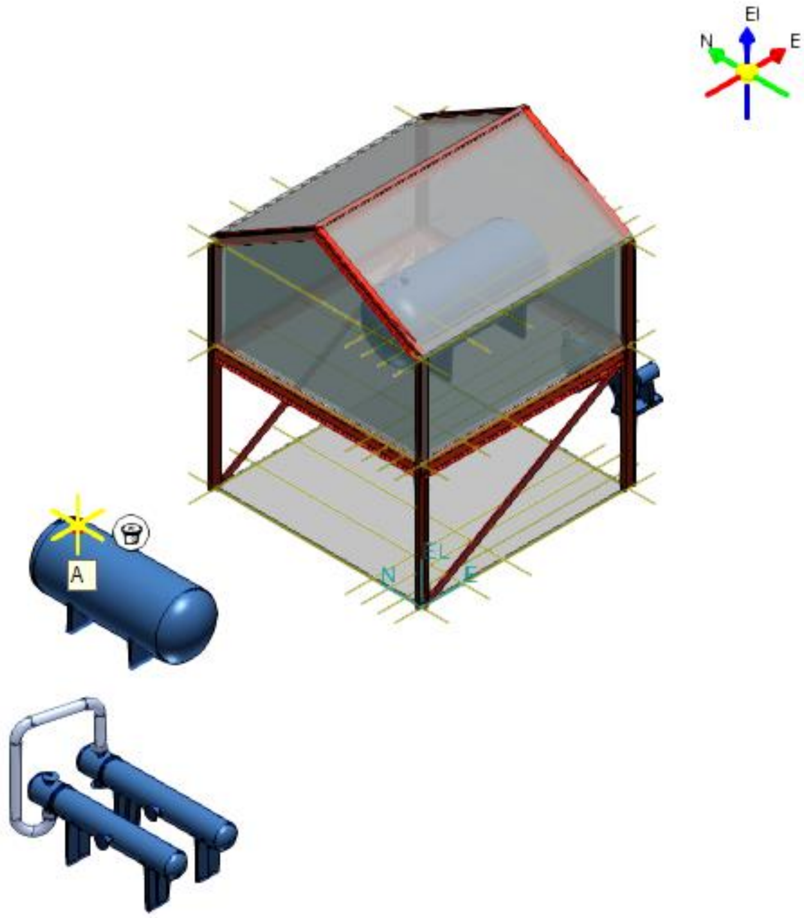
A pipeline 400-P from the equipment nozzle 40V-101/A
A pipeline 401-P from the equipment nozzle 40E-101A/E1
A pipeline 402-P from the equipment nozzle 40E-101A/E2



Before Starting this Procedure

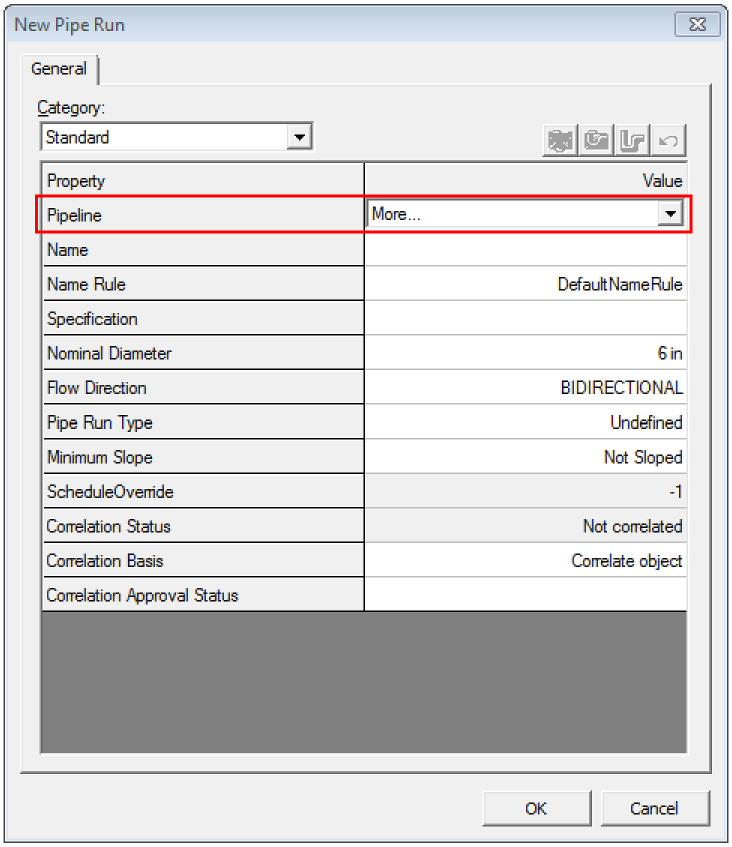
- Define your workspace to display unit **U04** and coordinate system **U04 CS**. In your training plant, select **U04** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the active permission group is set to **Piping**.

1. Click Route Pipe  on the vertical toolbar.
2. Select the nozzle 40V-101/A as the starting point, as shown below. The starting location can be an existing pipe run, a nozzle, a point in space or a piping component.



*The **New Pipe Run** dialog box appears.*

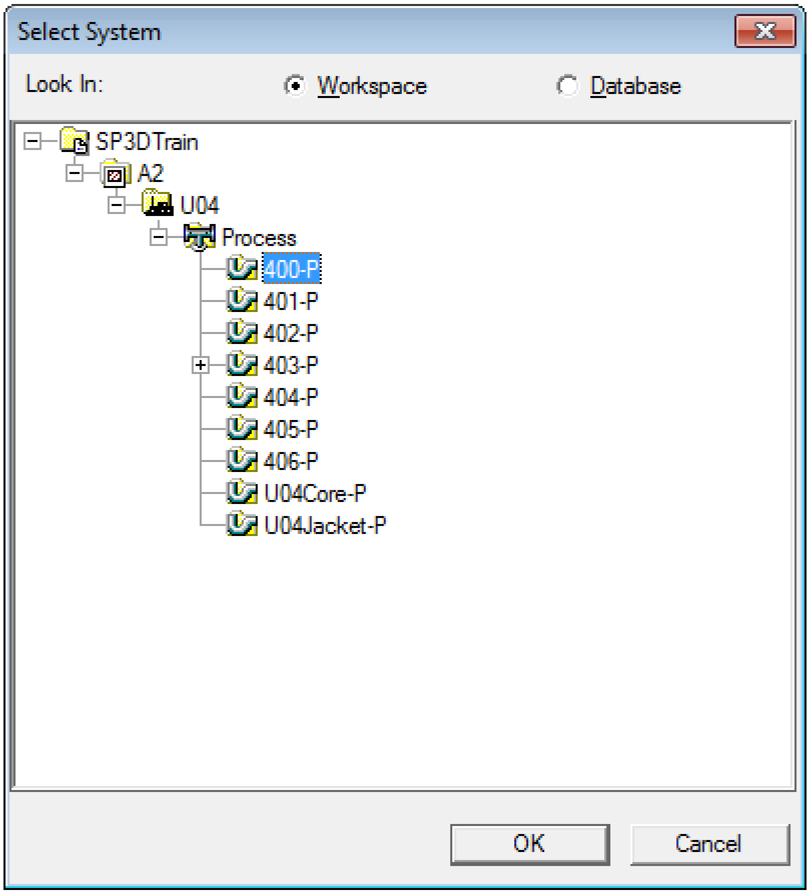
3. Select the **More…** option in the **Pipeline** drop-down list in the dialog box.



*The **Select System** dialog box appears.*

4. Here, you will select a pipeline system where the piping you are going to route will be located. The pipeline system is not just a hierarchy containing object; it also dictates the specifications or defaults that are available for routing.

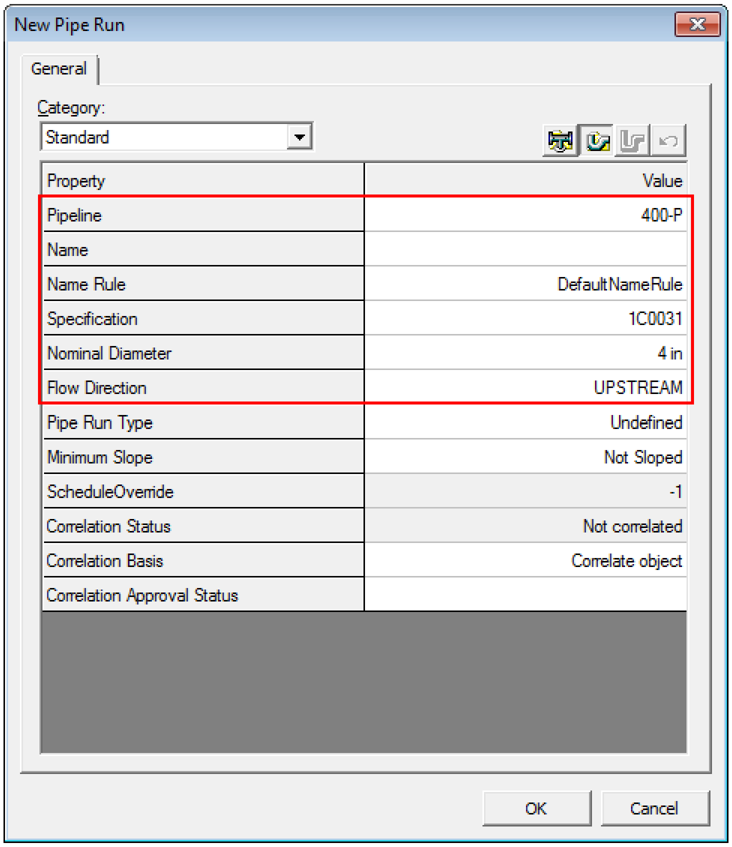
Expand the system folder hierarchy **A2>U04>Process** and select the pipeline **400-P** in the **Select System** dialog box. Then, click **OK**.



Smart 3D populates the properties of a pipe run with the specifications defined at the parent system level as follows:

- Specification for the pipe run - Only those pipe run specifications that are allowed on this system are displayed. If pipe specifications are set as a default property on the piping system level, then this can be inherited from the piping system. In the current scenario, the specification is set to **1C0031**.
- Nominal diameter for this pipe run - The pipe specification controls the available NPDs in the list. If you select the equipment nozzle as the starting point of your pipe run, SP3D automatically uses the NPD of the nozzle as the NPD of the pipe run. In the current scenario, the nominal diameter is set to **4 in**. The user does have an option of changing the size.
- Flow direction - If you select an equipment nozzle as the starting point of your pipe run, SP3D automatically uses the **Flow Direction** of the nozzle as the **Flow Direction** of the Pipe Run. In the current scenario, the flow direction is set to **UPSTREAM**.

Under the **Standard** category, **Pipeline, Naming Rule**, **Specification**, **Nominal Diameter**, and **Flow Direction** are the required values.



Depending on the product settings, you might have to switch to **Temperature and Pressure** category of properties and define the design maximum temperature and pressure.

5. Select the **Temperature and Pressure** category in the **Category** drop-down list in the **New Pipe Run** dialog box and verify that the software has transferred the temperature and pressure default values from the parent system. In this case, the transferred temperature and pressure default values are from the piping system.

At the top right of the **New Pipe Run** dialog box, there are four buttons. These four commands indicate and control the default properties for the new pipe run. If you click a command that is active, the default properties of that system or object are used. Default properties from the design basis have the highest priority.

Piping System

Uses the default properties on the parent piping system.

Pipeline System

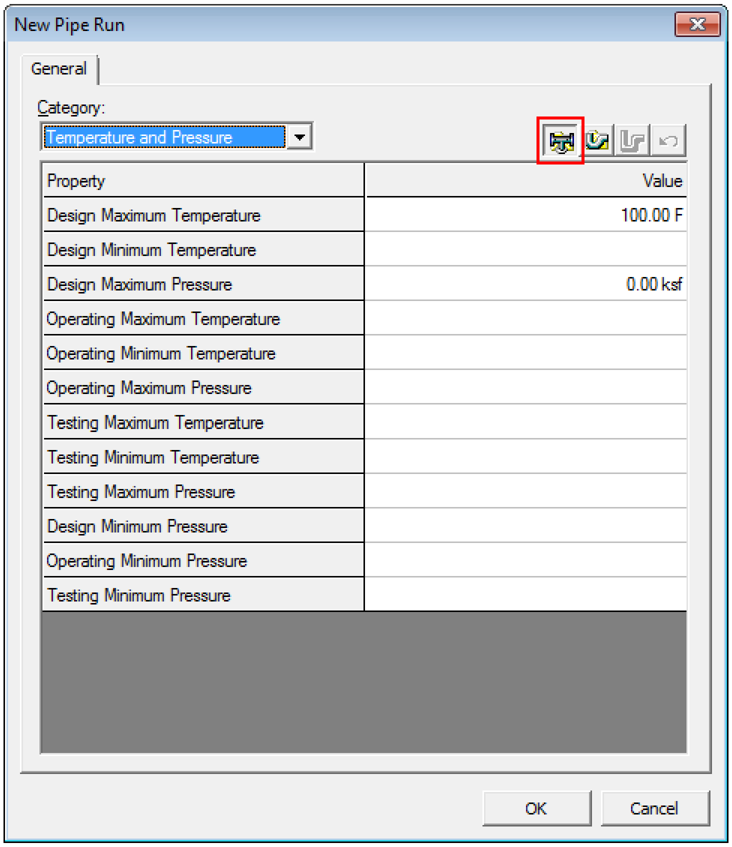
Uses the default properties on the parent pipeline system.

Pipe Run

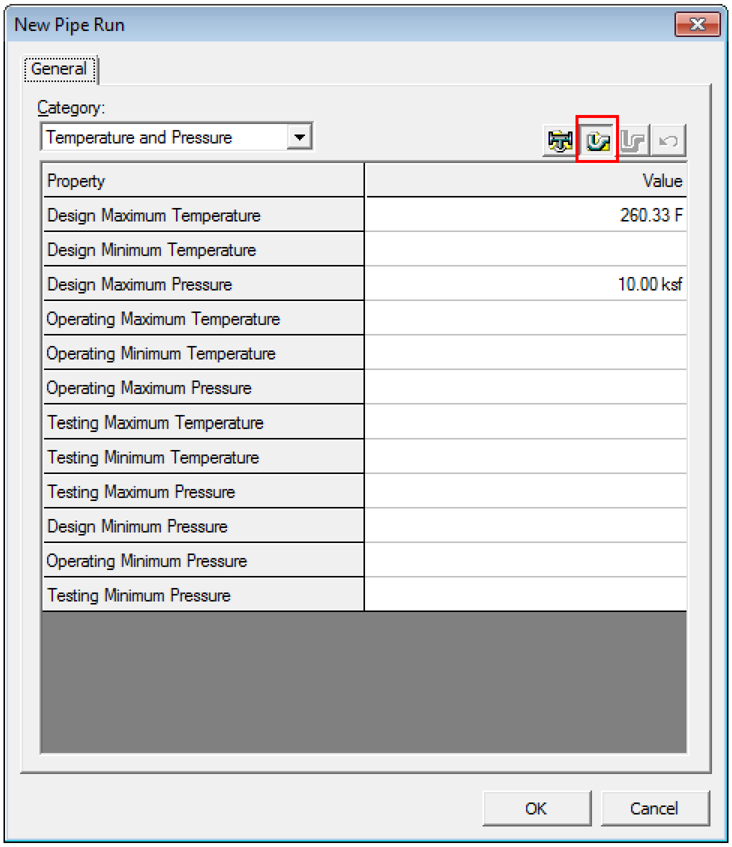
Uses the default properties from an existing pipe run.

Last Used

Uses the last saved properties for the pipe run.



1. To inherit the default properties from the pipeline rather than the piping system. Click on the **Pipeline System** icon. Verify that the Temperature and Pressure values have changed in the **New Pipe Run** dialog.

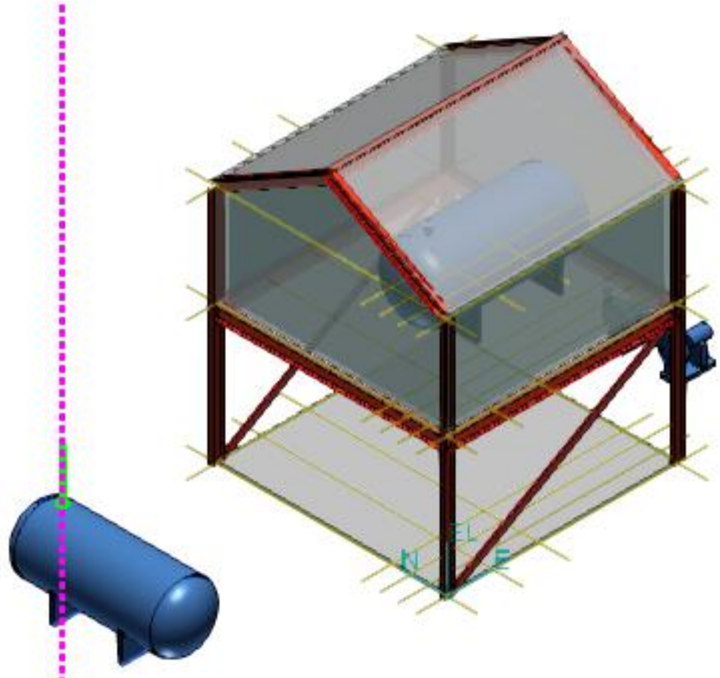


2. Click back on the Piping System button to change these values back. Click **OK**.

   SP3D displays a warning Temperature/Pressure note on a message box.

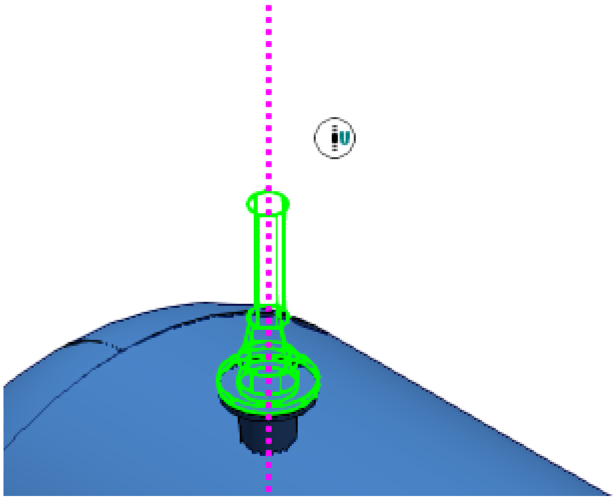
3. Check the **Do not show this message again** option and, click **OK**.

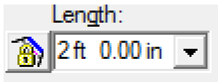
An outline of a pipe appears in the graphic view. Smart 3D locks the angle at 0 deg. As a result, you can only route the pipe in the upward direction.



4. Key in **2 ft** in the **Length** drop-down list of the Route Pipe ribbon. This will constrain the length of the pipe to 2 ft.

5. Position the cursor on top of the pipe. SmartSketch will display a U glyph. The SmartSketch glyph U indicates that the projection for the routing of a pipe is going in the vertical direction (the Up-Down Plane).



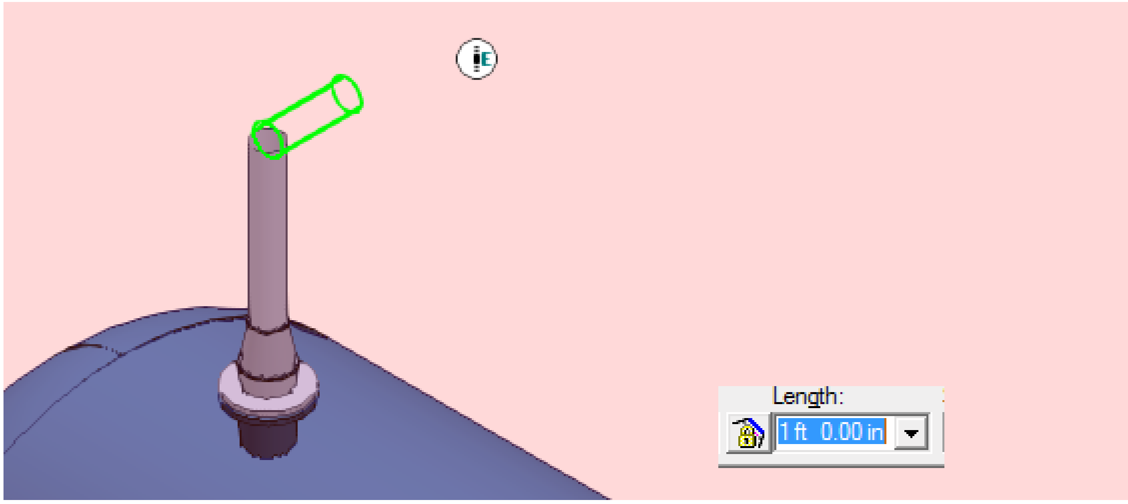


6. Left click in the graphic view to accept the placement of the pipe.

   You have now successfully placed the first pipe segment for this pipe run by using the SmartSketch glyphs to control the direction. Now you will route the next segment.

7. Key in **1 ft** in the **Length** drop-down list on the **Route Pipe** ribbon.

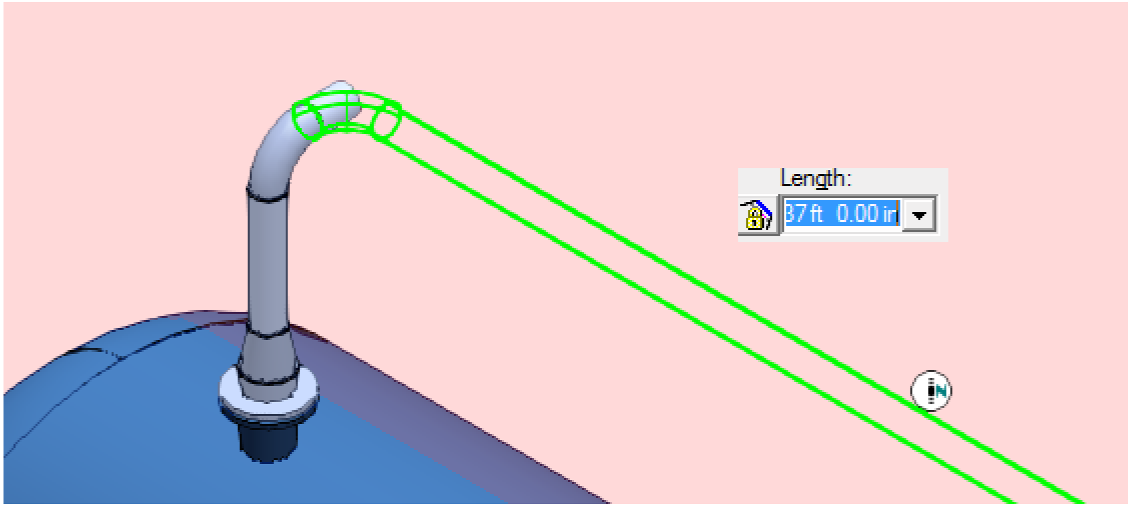
8. Position the cursor in the east direction until SmartSketch displays the E glyph, as shown below, which indicates that you are in the East-West plane.



9. Click in the graphic view to accept the placement of this pipe.

10. Key in **37 ft** in the **Length** drop-down list on the **Route Pipe** ribbon.

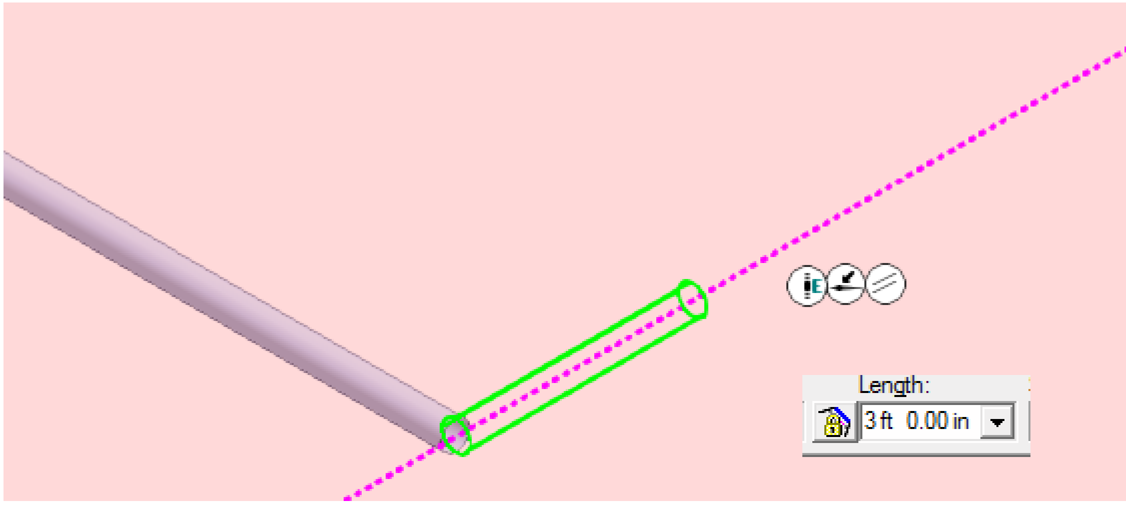
11. Position the cursor in the south direction until SmartSketch displays the N glyph, as shown below, which depicts that you are in North-South plane.



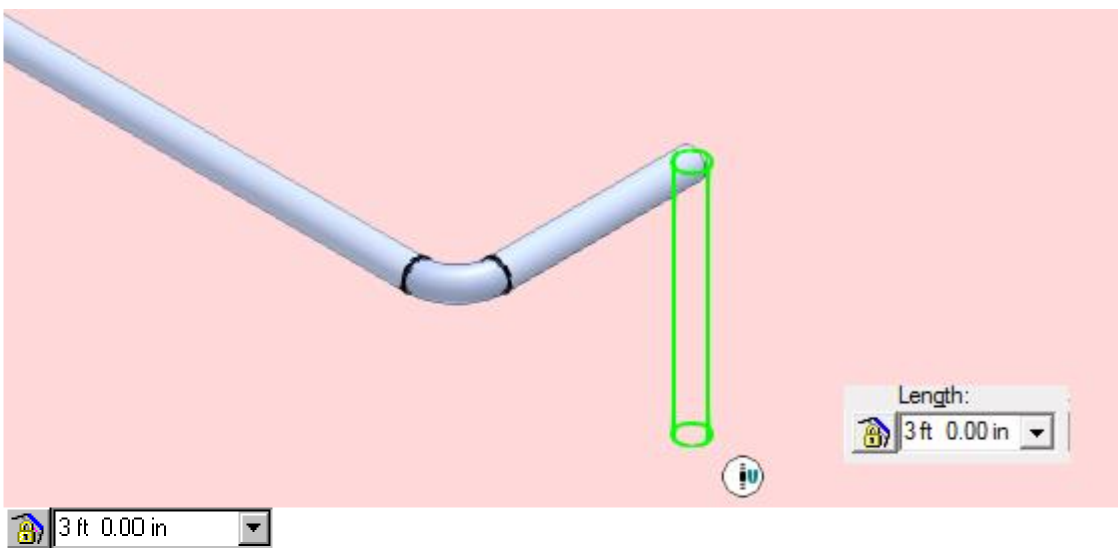
12. Click in the graphic view to accept the placement of this pipe.

    TIP While in the Route Pipe command, you can use any view command to zoom in and zoom out. You can right-click to go back to the placement mode.

13. Key in **3 ft** in the **Length** drop-down list on the **Route Pipe** ribbon.
14. Position the cursor in the east direction until SmartSketch displays the E glyph.

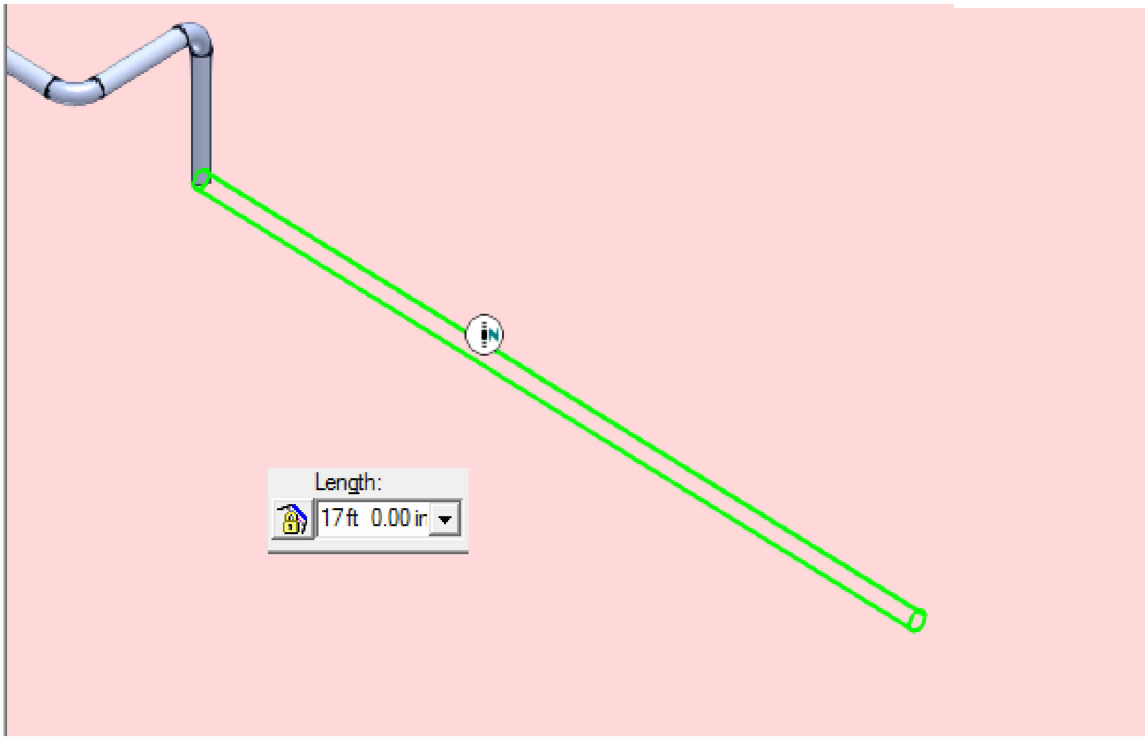


15. Click in the graphic view to accept the placement of this pipe.
16. Key in **3 ft** in the **Length** drop-down list on the **Route Pipe** ribbon.
17. Position the cursor down until SmartSketch displays the U glyph.

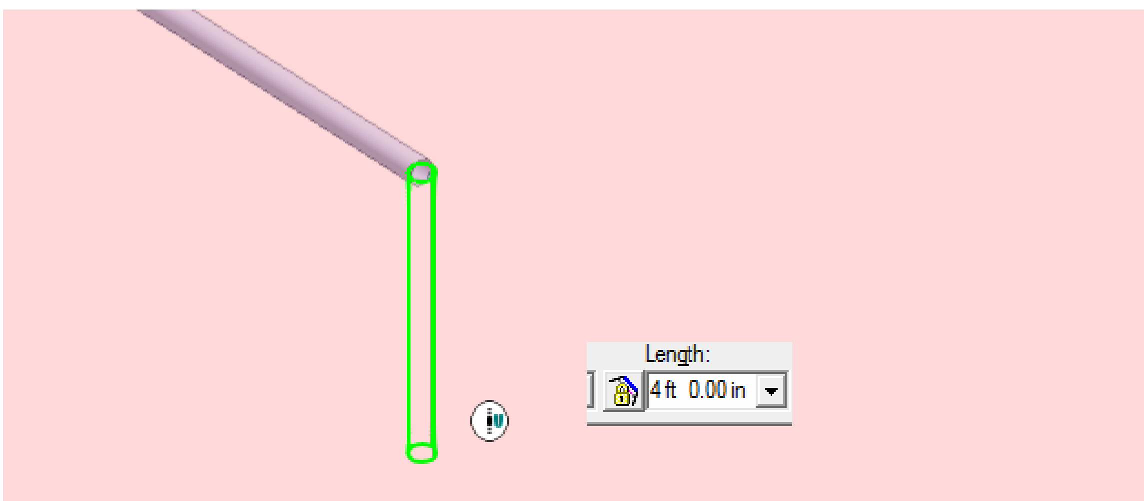


18. Click in the graphic view to accept the placement of this pipe.

19. Key in **17 ft** in the **Length** drop-down list on the **Route Pipe** ribbon and position the cursor in the south direction until SmartSketch displays the N glyph.

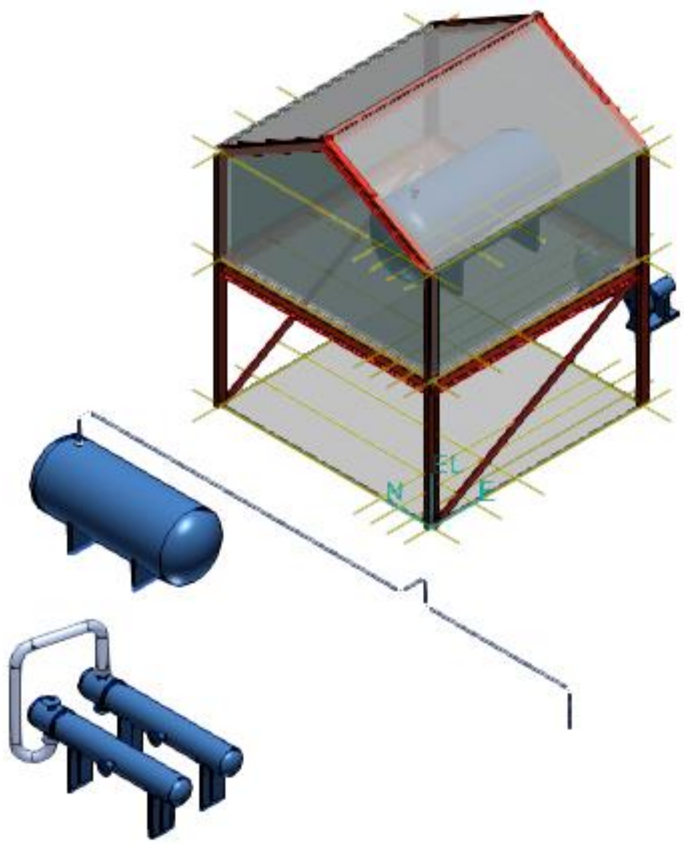


20. Click in the graphic view to accept the placement of this pipe.
21. Key in **4 ft** in the **Length** drop-down list on the **Route Pipe** ribbon and position the cursor down until SmartSketch displays the U glyph.

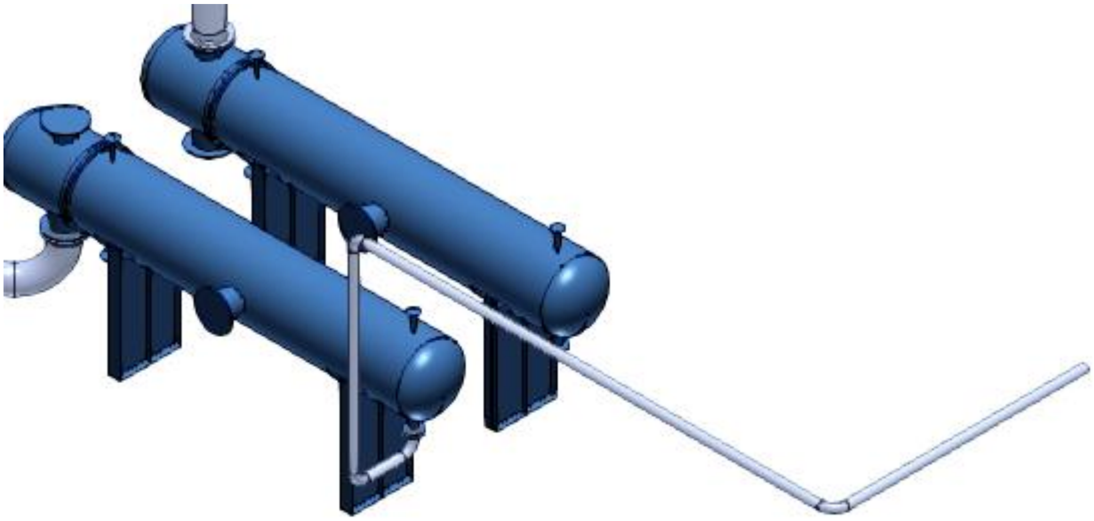


22. Click in the graphic view to accept the placement of this pipe.

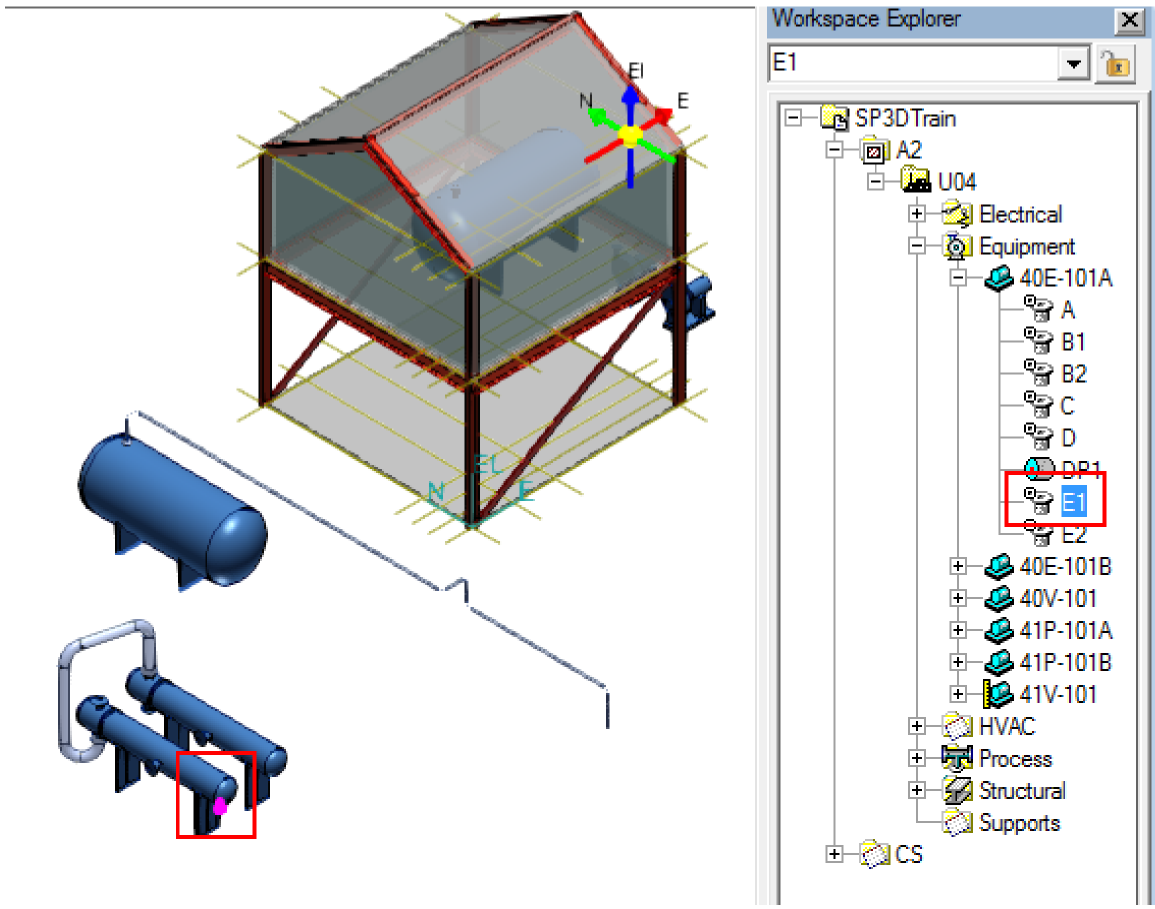
23. Right-click in the graphic view to terminate the **Route Pipe** command. The routed pipeline 400-P should resemble below.



Now, route a pipeline **401-P** from the equipment/nozzle **40E-101A/E1** to its completion by using the Working Plane Control, Angle Control and, the Length Control tools, as shown below.



24. Use the **Workspace Explorer** to locate the nozzle **40E-101A/E1**.



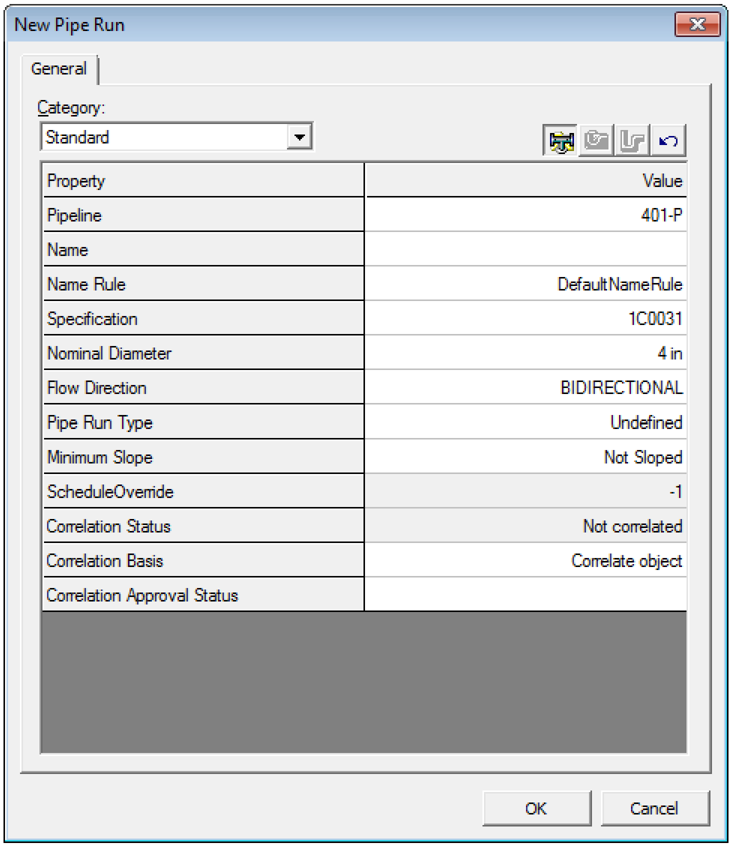
25. Click **Route Pipe**  on the vertical toolbar.

26. Click the equipment nozzle **40E-101A/E1**.

    The New Pipe Run dialog box appears.

27. Set the following parameters in the **New Pipe Run** dialog box, as shown, and click **OK**.

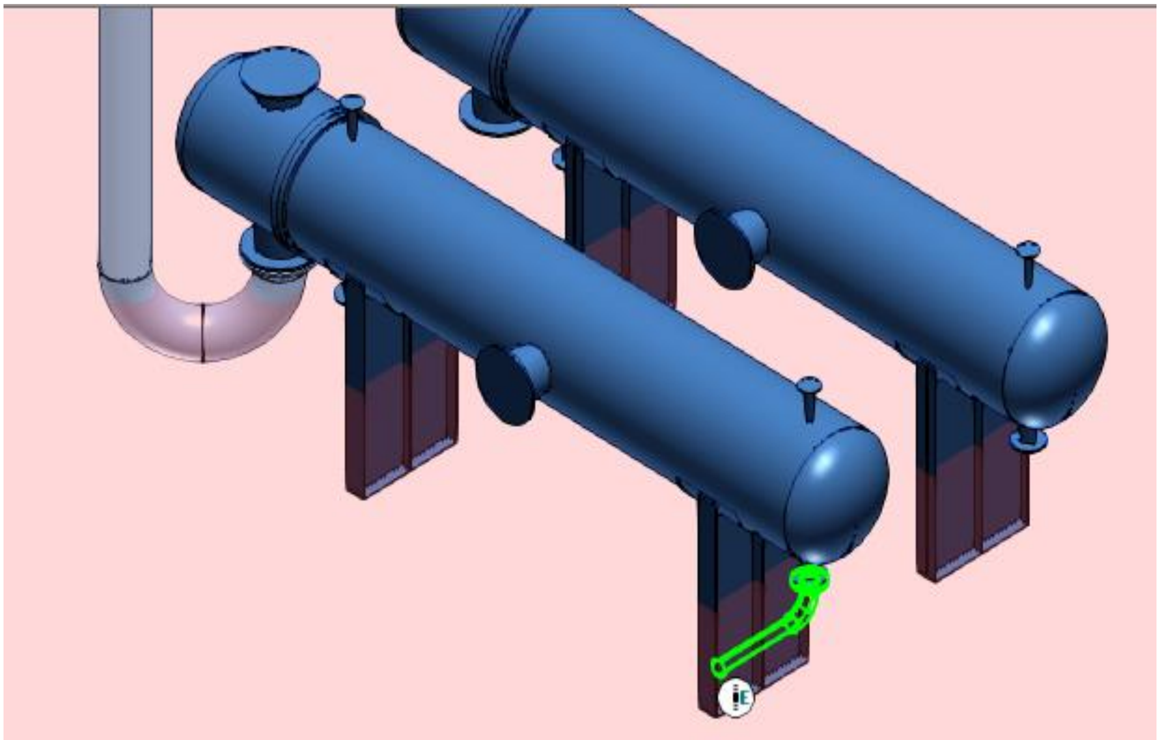
Pipeline: 401-P
Name Rule: DefaultNameRule
Specifications: 1C0031
Nominal Diameter: 4 in
Flow Direction: BIDIRECTIONAL
Minimum Slope: Not Sloped



An outline of a pipe will appear in the graphic view.

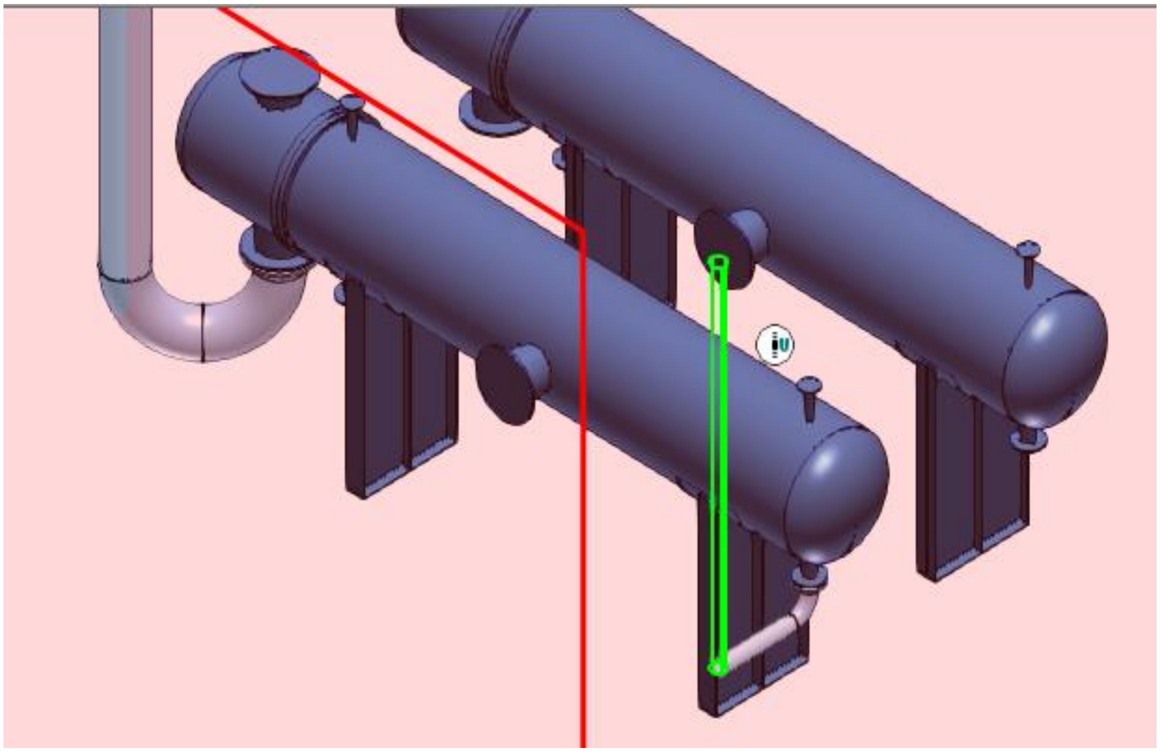
28. On the **Route Pipe** ribbon, select 90 deg in the **Angle** drop-down list and key in **2 ft 6 in** in the **Length** drop-down list.

29. Position the cursor in the west direction and use SmartSketch to locate the E glyph and click in the graphic view to place the pipe.

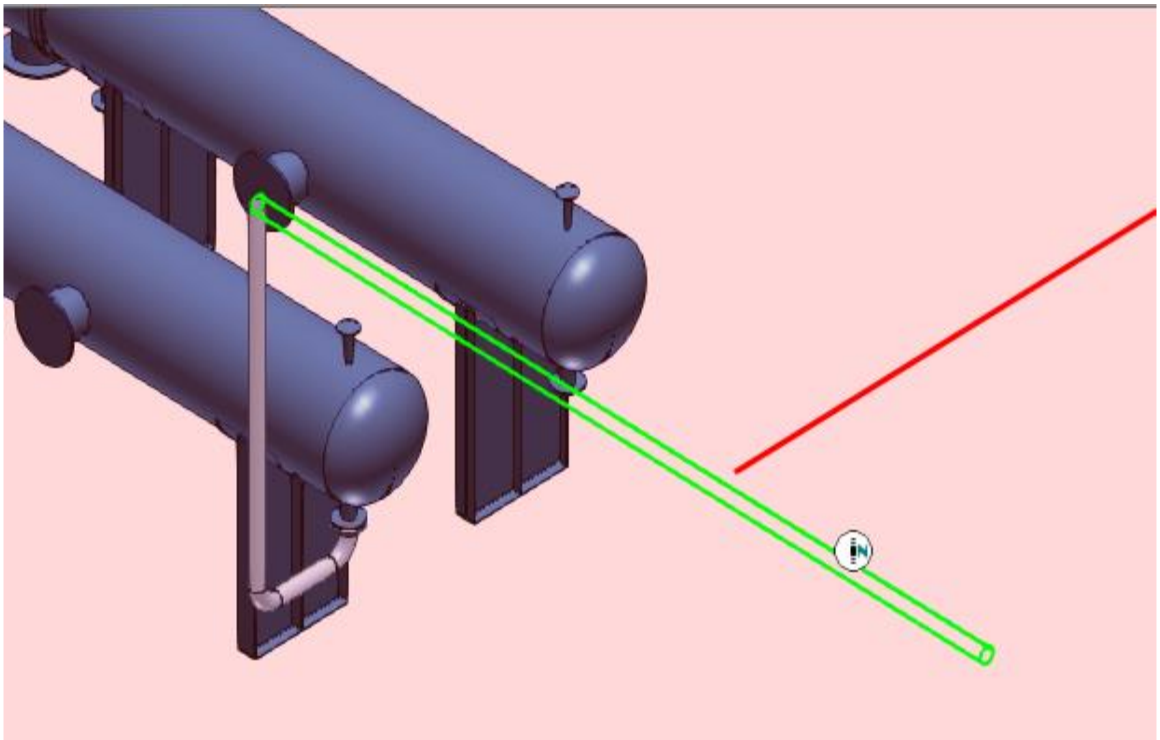


30. Now, on the **Route Pipe** ribbon, key in **9 ft** in the **Length** drop-down list and use SmartSketch to locate the U glyph.

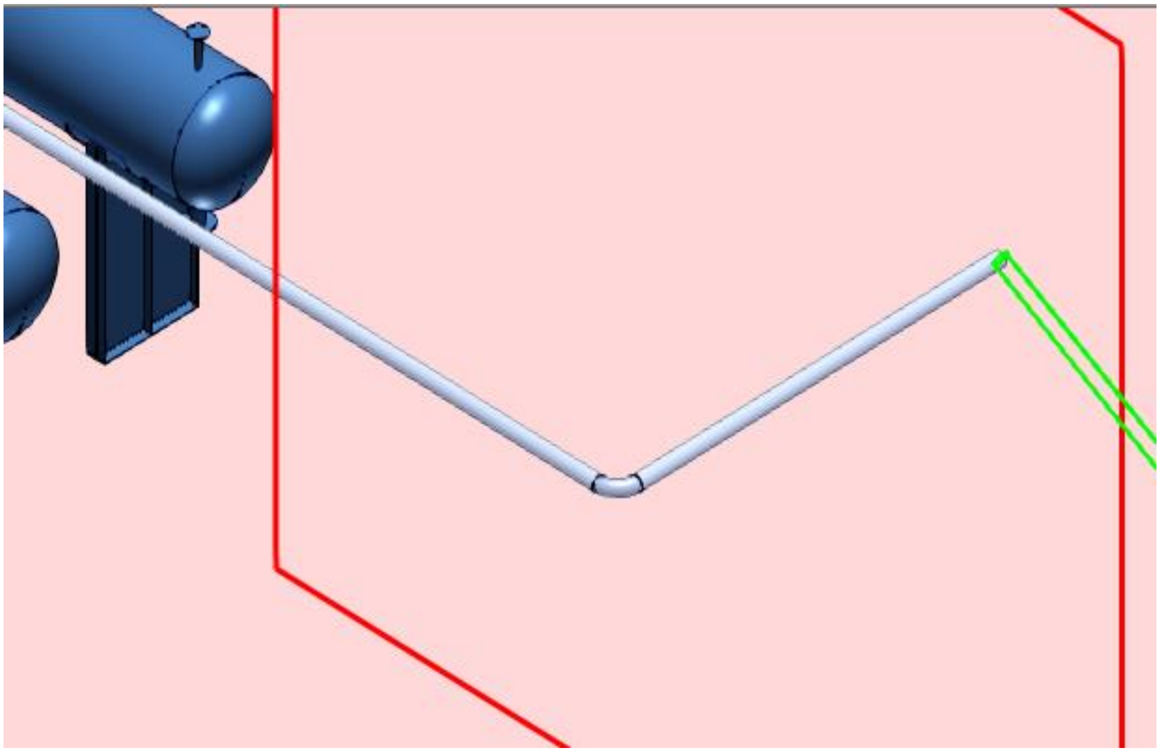
31. Click in the graphic view to place the pipe, as shown.



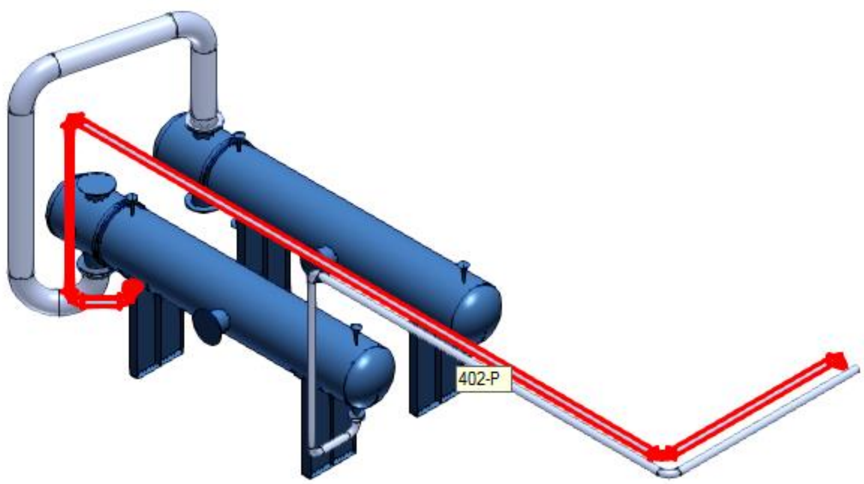
32. Now, on the **Route Pipe** ribbon, key in **20 ft** in the **Length** drop-down list and use SmartSketch to locate the N glyph.
33. Click in the graphic view to place the pipe, as shown.



34. Now, on the **Route Pipe** ribbon, key in **10 ft 6 in** in the **Length** drop-down list and use SmartSketch to locate the E glyph.
35. Click in the graphic view to place the pipe.
36. Right-click the graphic view to terminate the **Route Pipe** command.

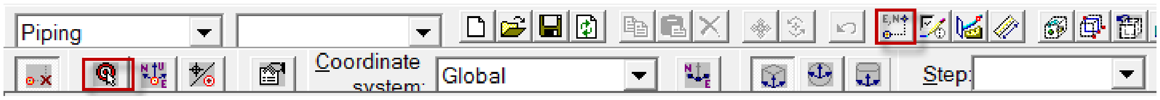


Now, route a pipeline 402-P from the equipment/nozzle 40E-101A/E2 to its completion by using PinPoint, Relative Tracking, Working Plane Control, Angle Control, and Length Control tools, as shown.



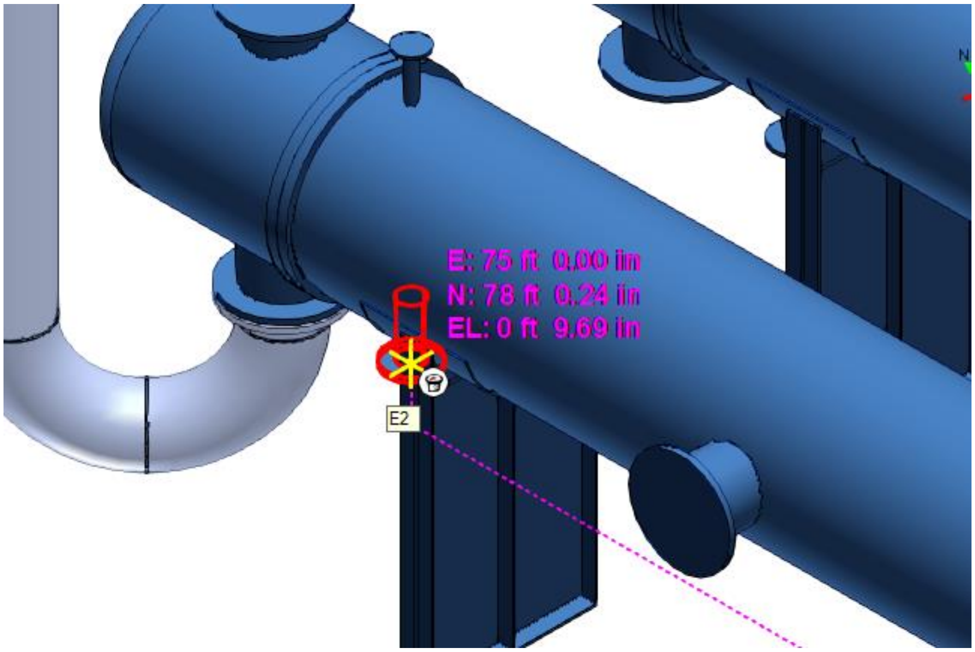
37. Use the **Workspace Explorer** to locate the nozzle **40E-101A/E2**.

38. Click **Pinpoint** on the Common toolbar, then click **Reposition Target** on the **PinPoint** ribbon.



39. Select nozzle E2 to reposition the target.

40. Click **Relative Tracking** on the pinpoint ribbon. Next, click **Route Pipe** on the vertical toolbar and select the equipment nozzle E2, as shown.



*The **New Pipe Run** dialog box appears.*

41. Set the following parameters in the dialog box and click **OK**.

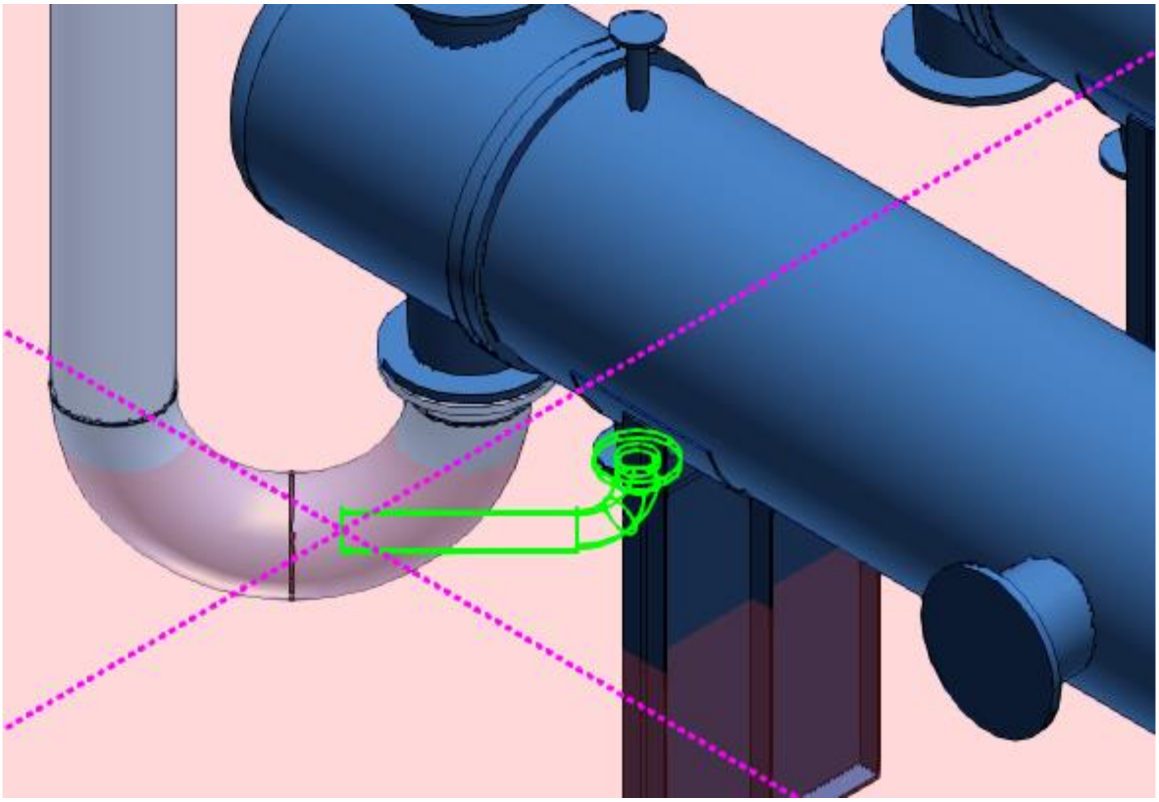
Pipeline: 402-P
Name Rule: DefaultNameRule
Specifications: 1C0031
Nominal Diameter: 4 in
Flow Direction: BIDIRECTIONAL
Minimum Slope: Not Sloped
Correlation Bases: Correlate object

42. Select **90 deg** in the **Angle** drop-down list on the **Route Pipe** ribbon. Make sure the plane is set to **No Plane**

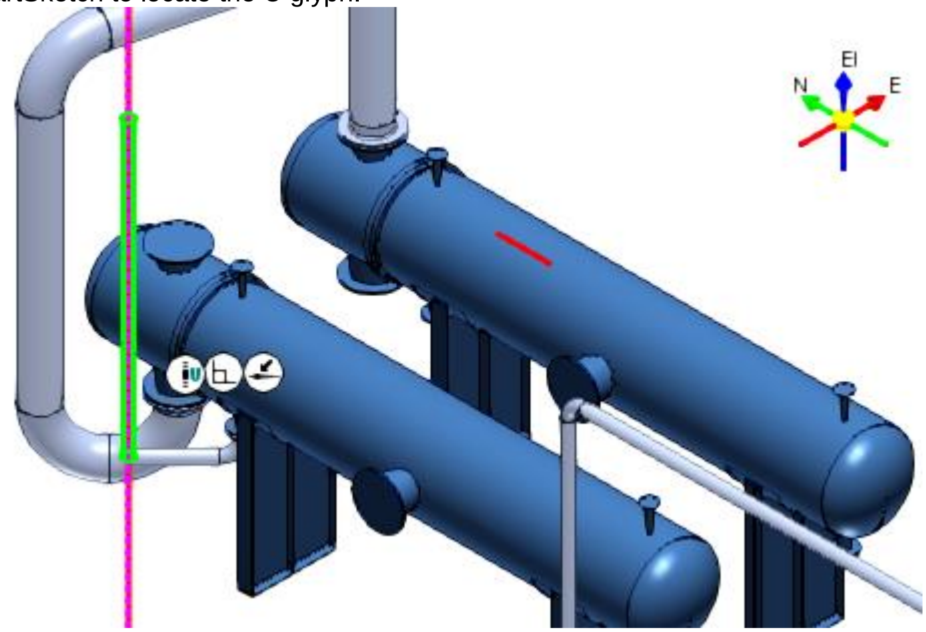
TIP You cannot key in anything in the angle field when the route plane is set to No Plane. You can only select 0 deg or 90 deg.

43. On the **PinPoint** ribbon, key in **-0 ft 21in** for easting E and **0 ft 21 in** for northing N. These coordinates will show the parameters as -1 ft 9.00 in and 1 ft 9.00 in, respectively.

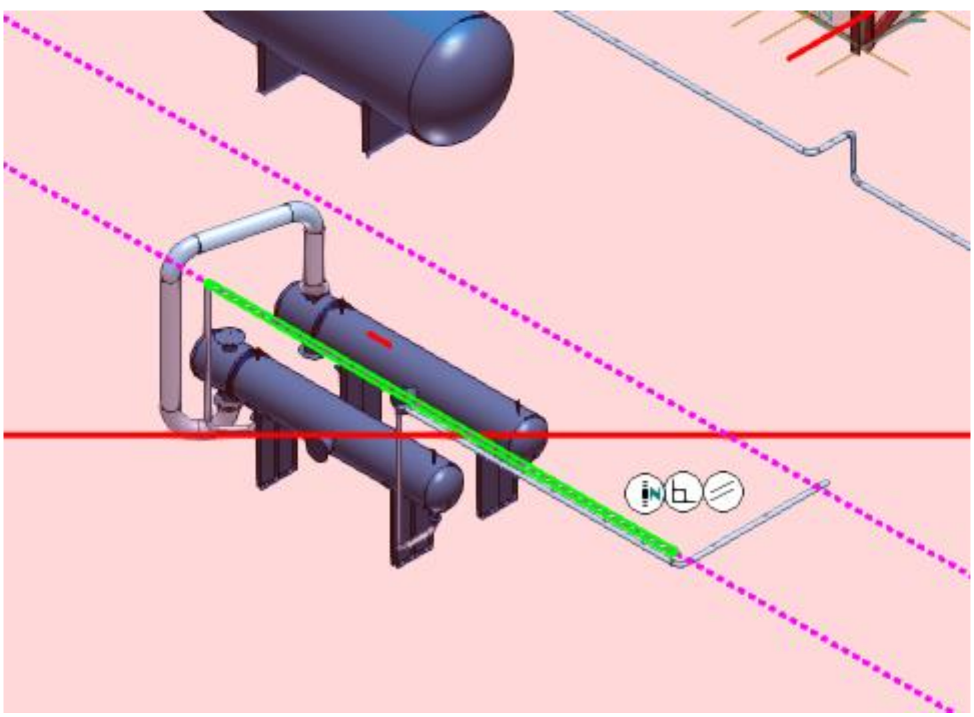
44. Notice the Pinpoint target is automatically placed when you identify the equipment nozzle E2. Select the **Zoom Area** command to ensure target has been placed correctly and then left-click in the graphic view to place the pipe.



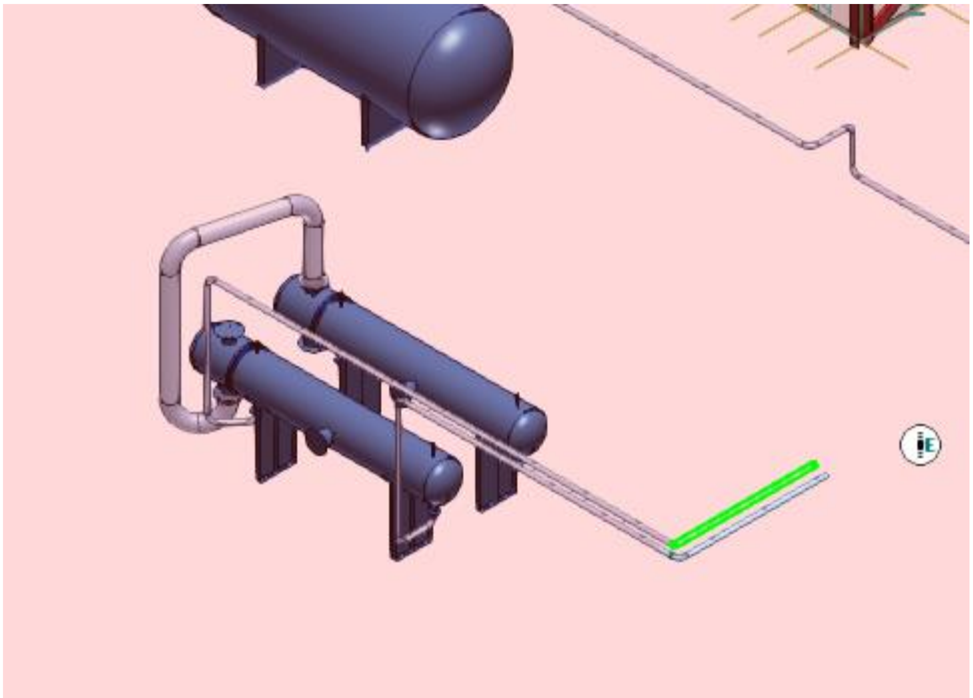
45. Click in the graphic view to place the pipe.
46. Now, on the **Route Pipe** ribbon, key in **9 ft in** the **Length** drop-down list and use SmartSketch to locate the U glyph.



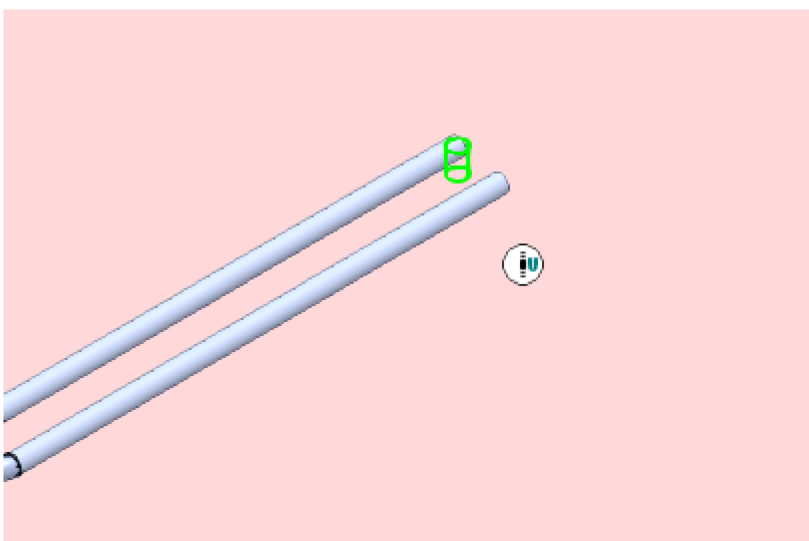
47. Click in the graphic view to place the pipe.
48. Now, on the **Route Pipe** ribbon, key in **33 ft 2 in** in the **Length** drop-down list and use SmartSketch to locate the N glyph.
49. Click in the graphic view to place the pipe.



50. Now, on the **Route Pipe** ribbon, key in **10 ft in** the Length drop-down list and use SmartSketch to locate the E glyph.
51. Click in the graphic view to place the pipe.



52. Now, on the **Route Pipe** ribbon, key in **0 ft 6in** in the **Length** drop-down list and use SmartSketch to locate the U glyph.
53. Click in the graphic view to place the pipe.



54. Right-click in the graphic view to terminate the **Route Pipe** command.

For information about routing pipes, refer to the **Route Pipe** topic in the *Piping User's Guide*.

S E S S I O N 3

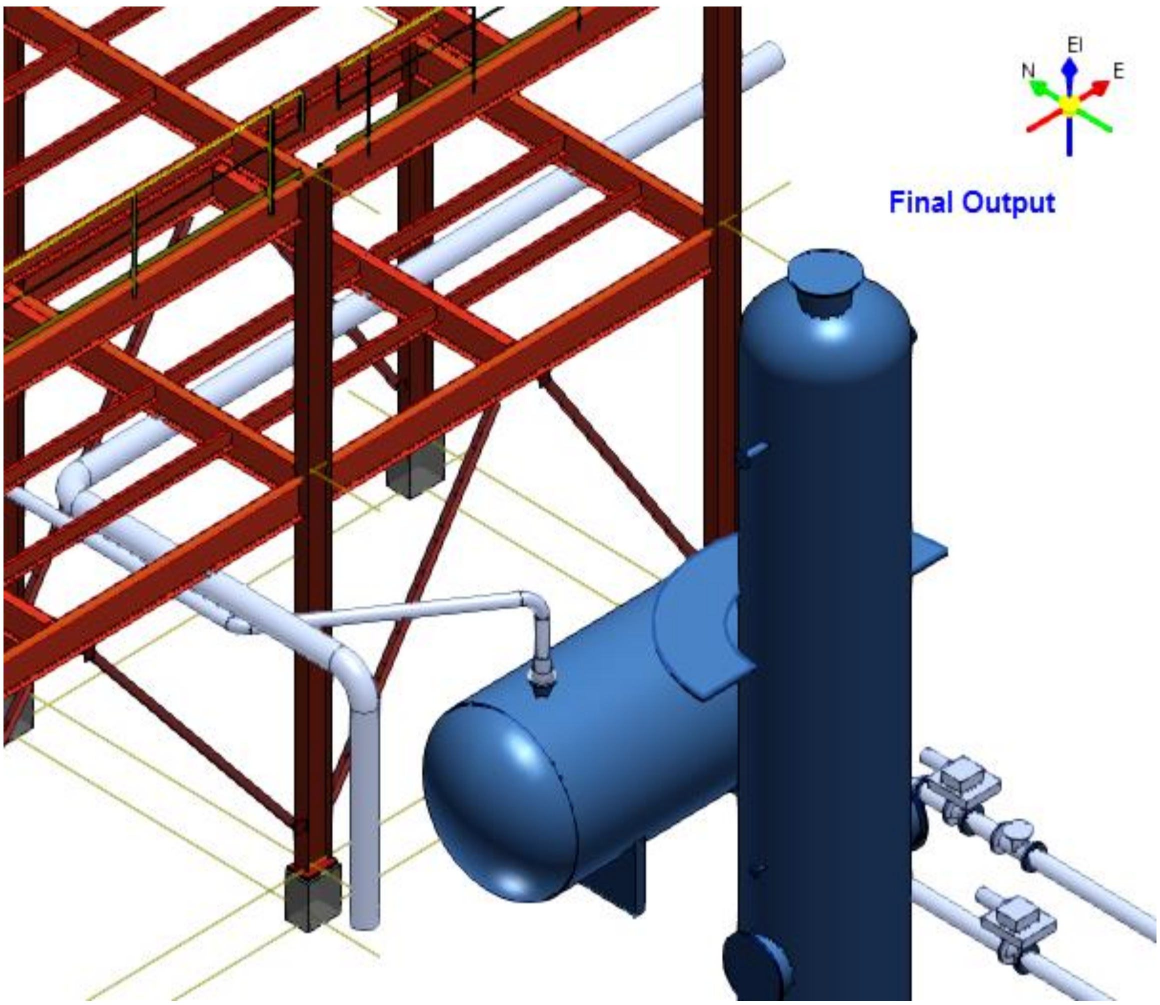
Routing Pipe Bends

Objective

By the end of this session, you will be able to:

- Route Pipe Bend and change default bend radius multiplier during placement.
- Change bend radius multiplier of a previously pipe bend.

After you are finished, the routed pipe bends should resemble the figure below:

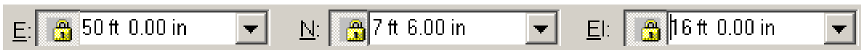


Before Starting this Procedure

- Define your workspace to display unit **U02** and coordinate system **U02 CS**. In your training plant, select **U02** form **Plant Filters > Training Filters** in the **Select Filter** dialog box.

- Make sure you are in the Piping task and that the active permission group is set to **Piping**.

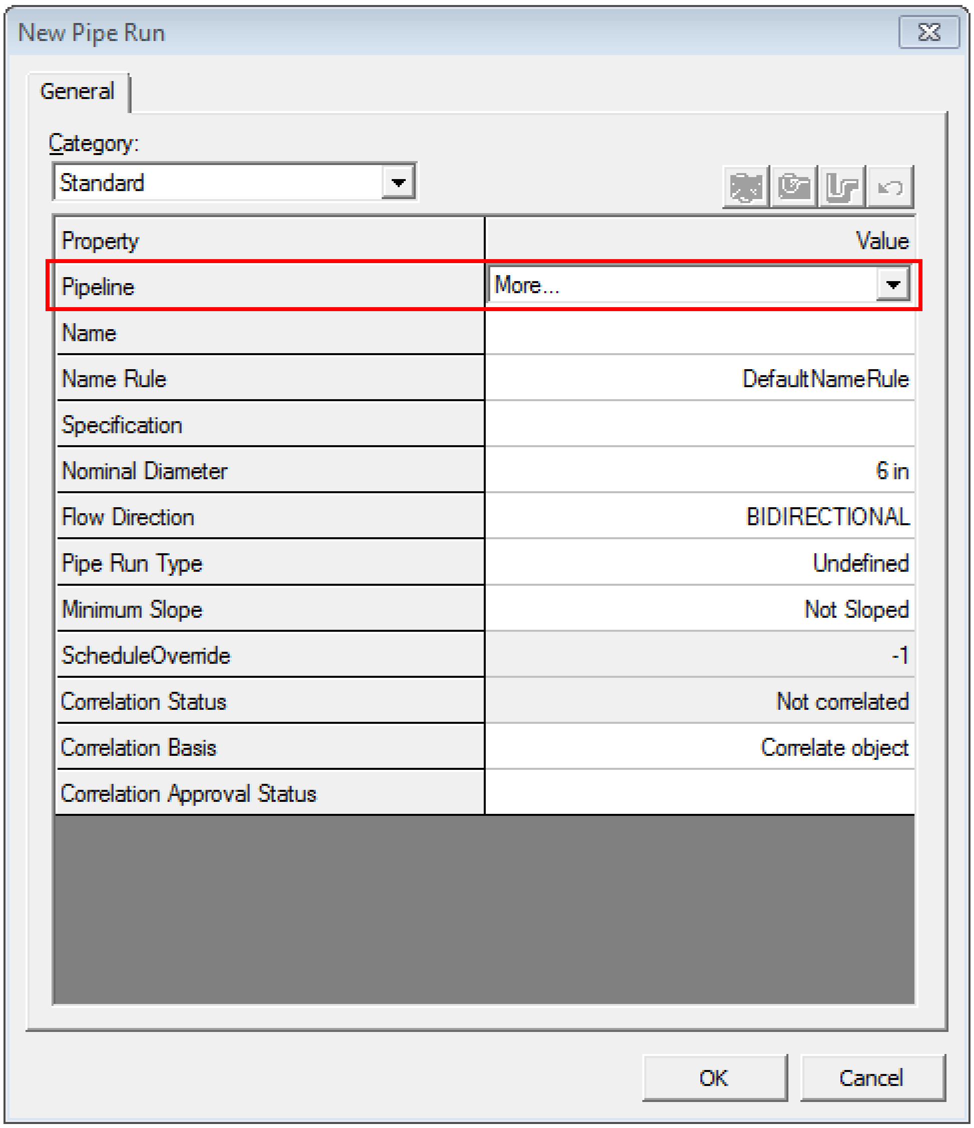
1. Activate the **Pinpoint** ribbon by selecting **Tools > Pinpoint**.
2. Set the active coordinate system to **U02 CS** and select the **Set Target to Origin** option on the **Pinpoint** ribbon.
3. Click **Route Pipe** on the vertical toolbar.
4. On the **Pin-Point** ribbon, key in **50** for easting **E** and **7 ft 6in** for northing **N** and **16 ft** for **EL** elevation.



5. Click in the graphic view to accept placement.

   The New Pipe Run dialog box appears.

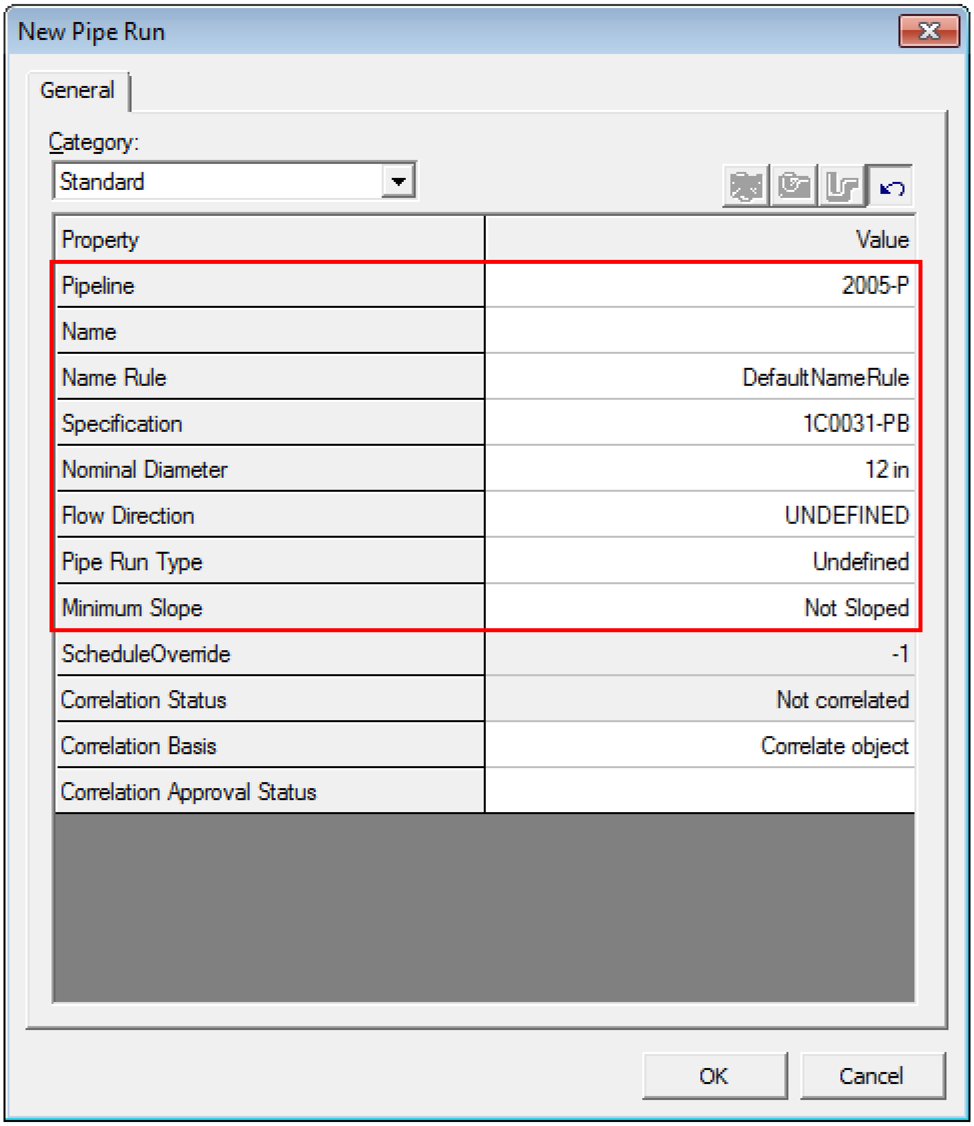
6. Select the **More…** option in the **Pipeline** drop-down list in the dialog box.



The Select System dialog box appears.

7. Here, you will select a pipeline system where the piping you are going to route will be located. The pipeline system is not just a hierarchy containing object; it also dictates the specifications or defaults that are available for routing.
8. Expand the system folder hierarchy **A2 > U02 > Process**, and then select the pipeline **2005-P** in the **Select System** dialog box, as shown in above.
9. Click **OK**.
10. Set the following parameter in the New Pipe Run dialog box as shown below, and click **OK**:

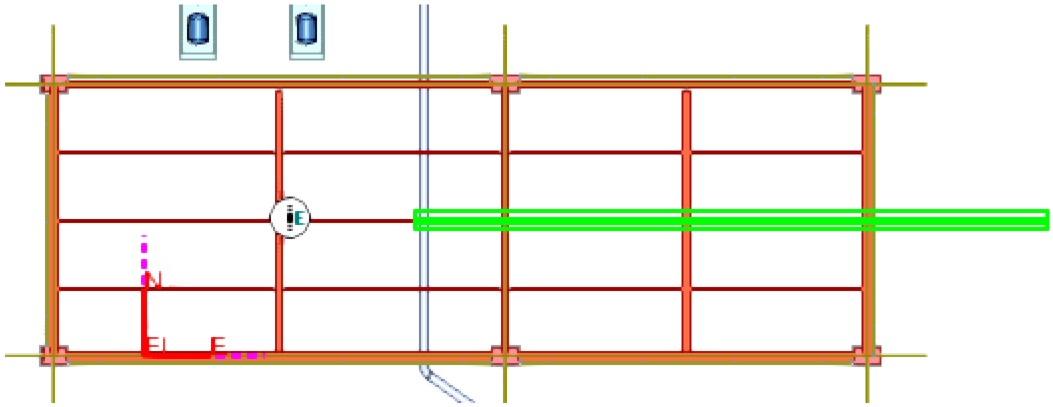
Pipeline: 2005-P
Name Rule: DefaultNameRule
Specifications: 1C0031-PB
Nominal Diameter: 12 in
Flow Direction: UNDEFINED
Minimum Slope: Not Sloped



An outline of a pipe appears in the graphic view.

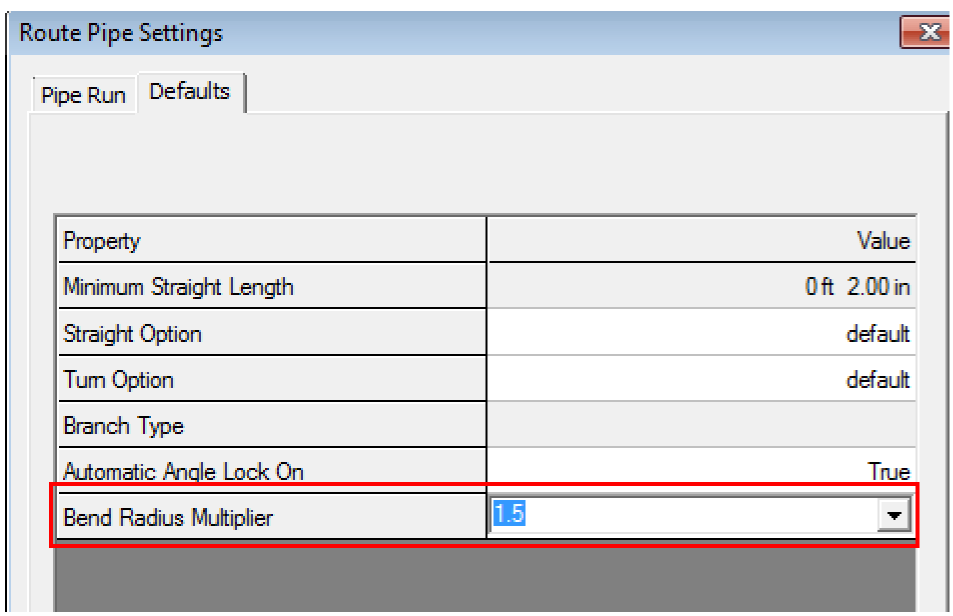
11. On the Route Pipe ribbon, key in **35 ft** in the **Length** drop-down list.

12. Position the cursor in the west direction and use Smart Sketch to locate the E glyph, and click in the graphic view to place the pipe.

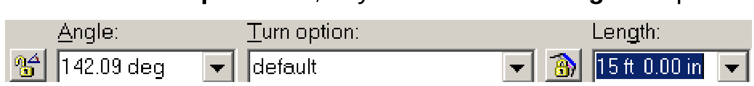


13. Before placing next pipe segment, activate the **Properties Dialog** box on the ribbon bar, as shown below.

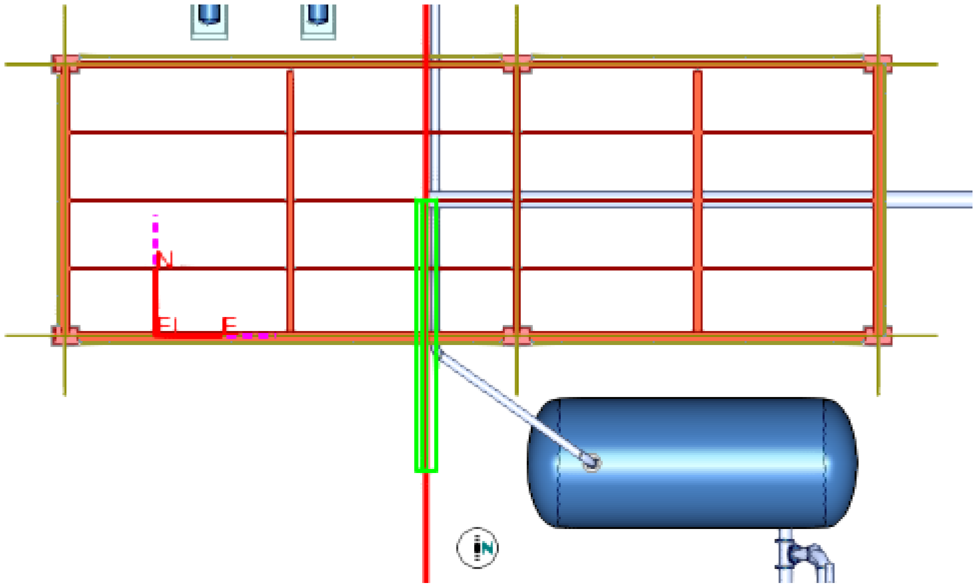
14. On the **Defaults Tab** of the **Route Pipe Settings** dialog box, change the **Bend Radius Multiplier** to **1.5**, and then click **OK**.



15. On the **Route Pipe** ribbon, key in **15 ft** in the **Length** drop-down list.

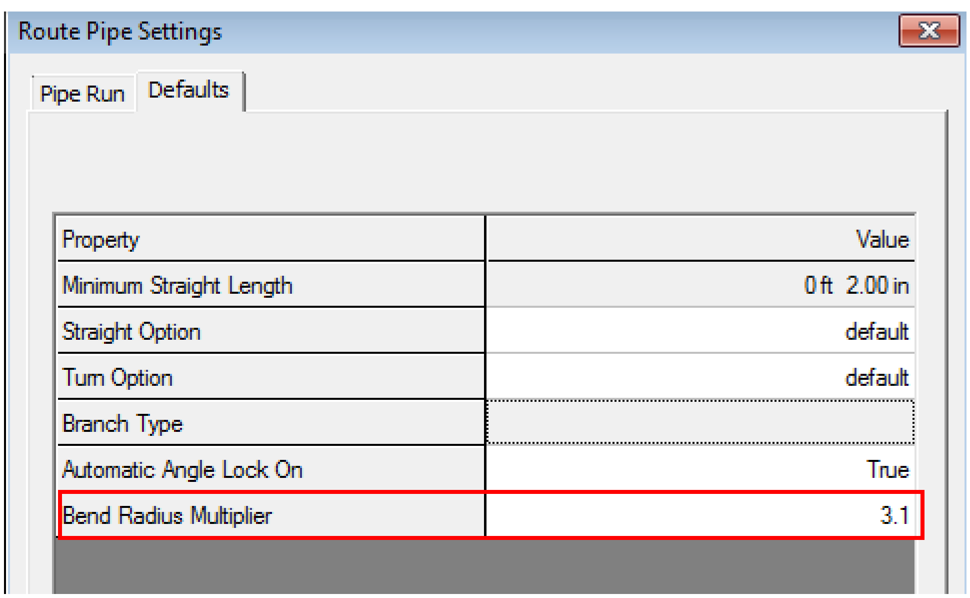


16. Position the cursor in the South direction and use SmartSketch to locate the N glyph, and click in the graphic view to place the pipe.

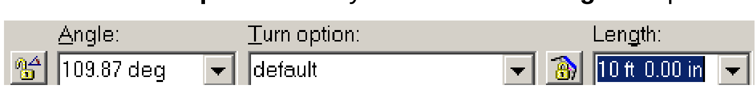


17. Before placing next pipe segment, activate the **Properties Dialog** box on the ribbon bar, as shown below.

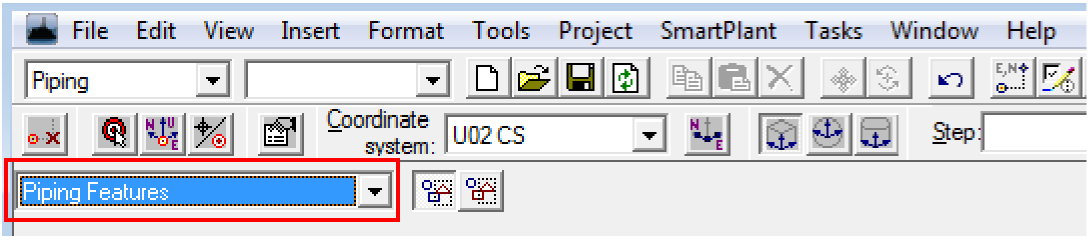
18. On the **Defaults Tab** of the **Route Pipe Settings** dialog box change the **Bend Radius Multiplier** to **3.1**, and then click **OK**.



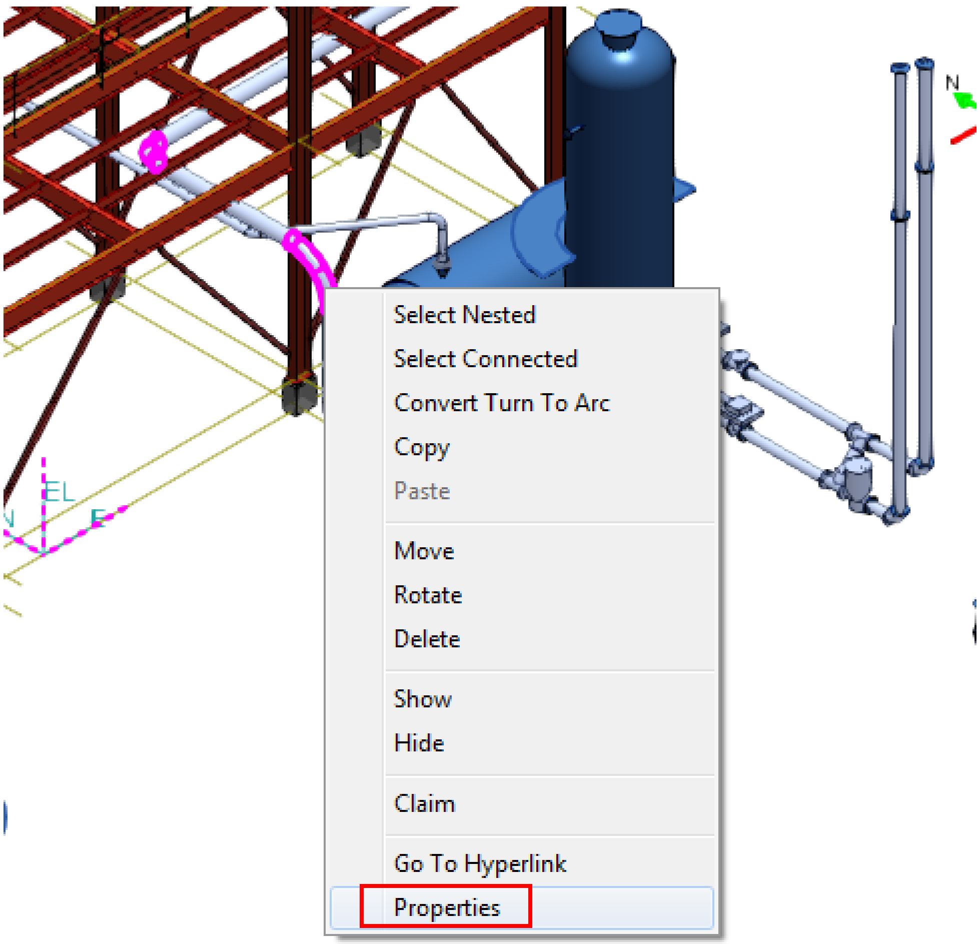
19. On the **Route Pipe** ribbon key in **10 ft** in the **Length** drop-down list



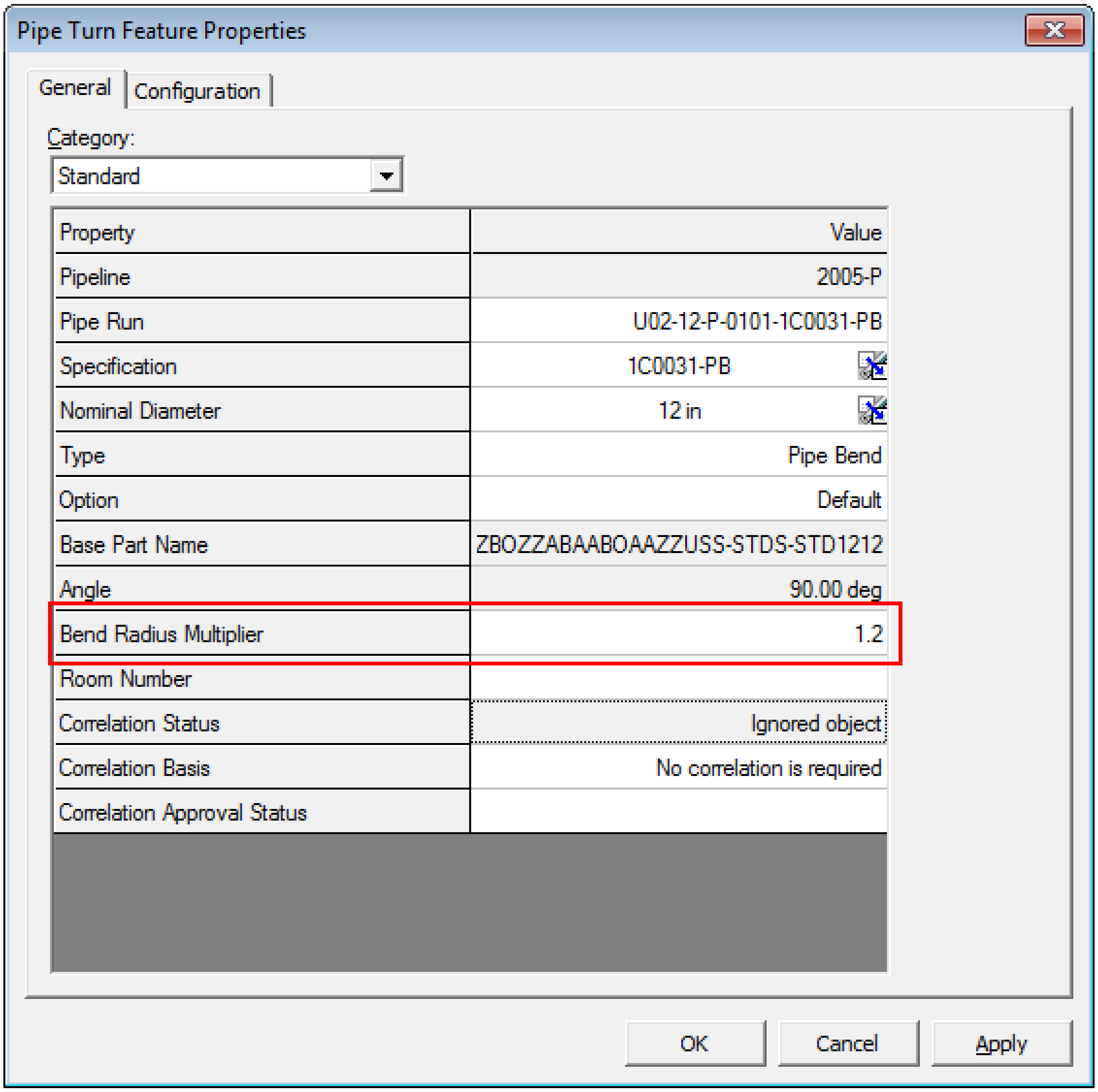
20. Position the cursor in the downward direction and use SmartSketch to locate the U glyph, and then click in the graphic view to place the pipe.
21. Right mouse click to exit the **Route Pipe** command.
22. Review the **Locate Filter** to verify that is set to **Piping Features**.



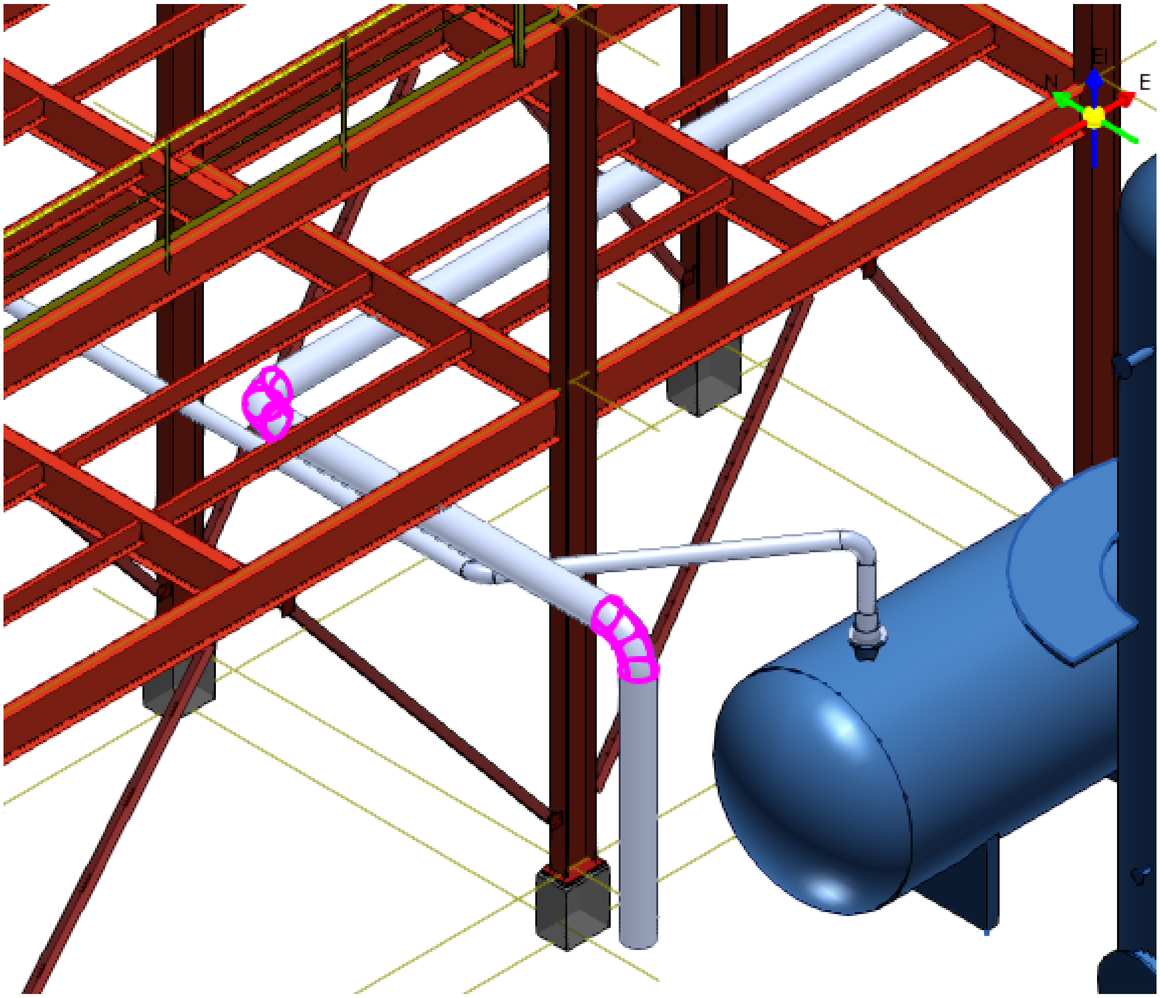
23. Press and hold the CTRL key, and then select both pipe bends in pipeline 2005-P.
24. Right-click on one of the pipe bends, and then select **Properties**.



25. In the **Pipe Turn Feature Properties** dialog box, change the **Bend Radius Multiplier** to **1.2**.



26. The final result should look like this.



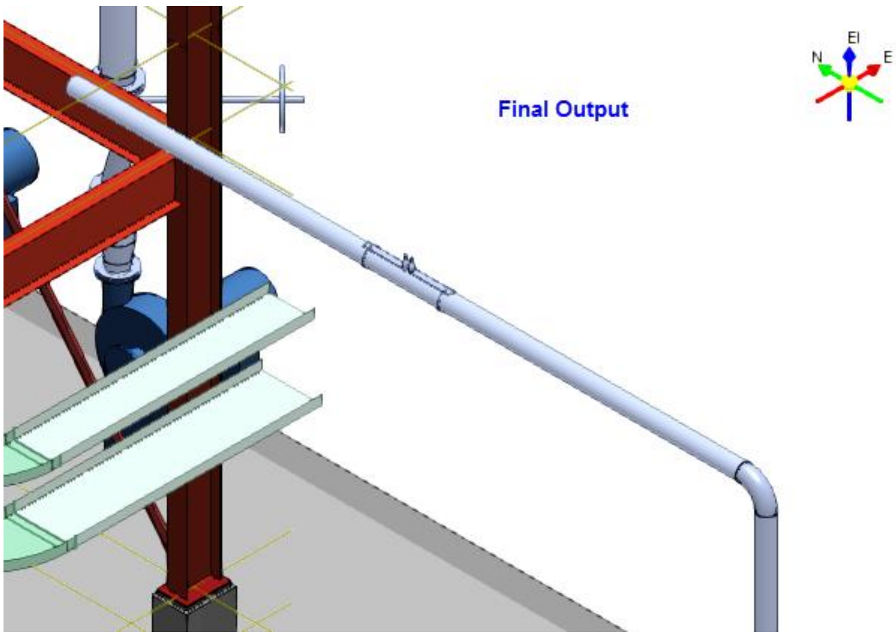
SESSION 4

Routing Pipe Using Cardinal Point References

Objective

By the end of this session, you will be able to:

- Route the pipeline 1003-P from E-102 (a kettle type heat exchanger) using cardinal points as a reference.
- Include and exclude the Insulation Thickness.



Before Starting this Procedure

- Define your workspace to display **Unit U01** and coordinate system **U01 CS**. In your training plant, select **U01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Click **Route Pipe**  on the vertical toolbar, and then click the piping nozzle **N2** of **E-102**.

   A New Pipe Run dialog box appears.

2. Key-in the following parameters in the **General Tab** under following categories.

   Standard Category

Pipeline: 1003-P
Specification: 1C0031
Nominal Diameter: 6 inch

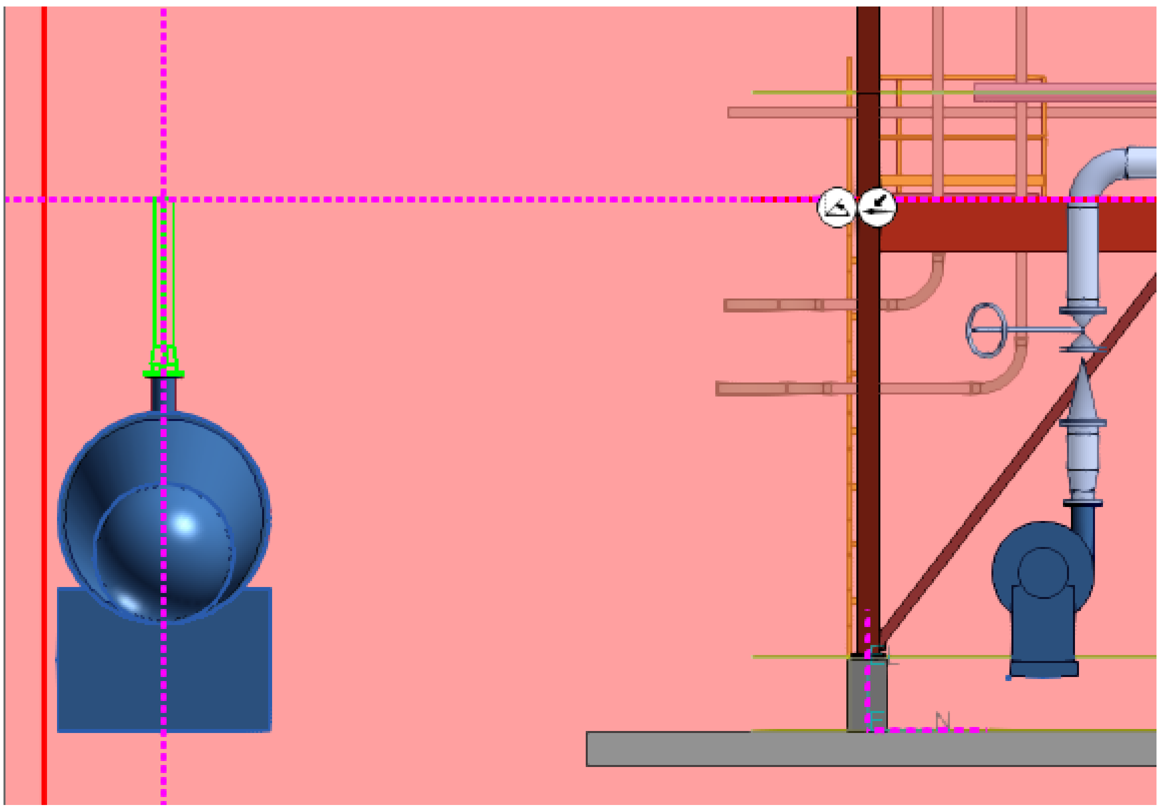
Insulation & Tracing Category

Insulation Specification: User defined
Insulation Purpose: Heat conservation
Insulation Material: MW - Mineral Wool
Insulation Thickness: 0 feet 3inch

3. Click **OK** to close the dialog box.

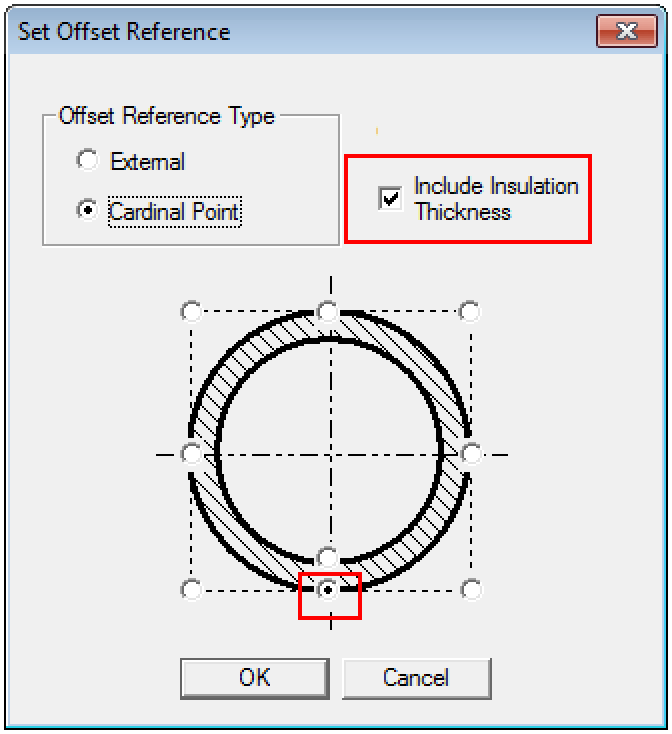
   The outline of the new pipe run appears on the screen.

4. Change the view to **Looking West** from **Common Views**.
5. Make sure that the **Plane** constraint is set to **Elevation Plane: North-South**
6. Move the cursor upwards and then towards the first column until the **Point on Curve** glyph appears, and then click in the graphical view to place the pipe as shown.

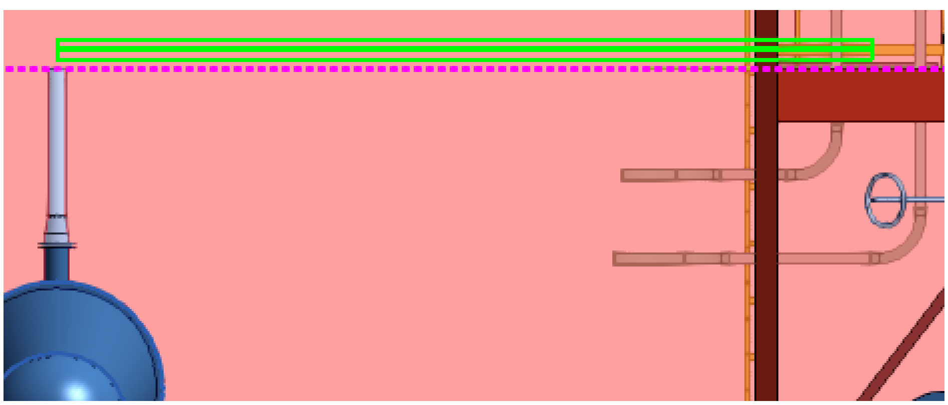


7. Under **Offset** on the **Route Pipe** ribbon, select the **Set Offset Reference** option to set the options for reference offsets while routing a pipe run. Set the offset reference type as **Cardinal Point** and select the BOP.

8. Check the **Include Insulation Thickness** option.



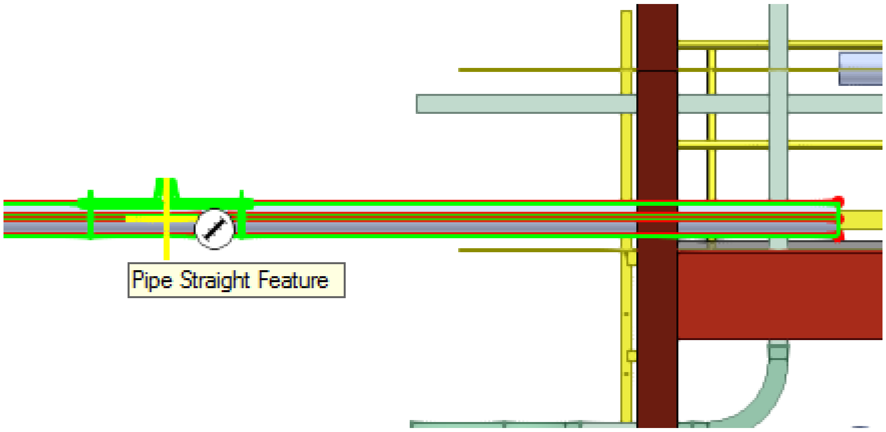
9. Key-in **23 ft** in the **Length** box and move the cursor towards the pipe rack until the North glyph appears.
10. Click in the graphical view to place the pipe. Right-click to terminate the **Route Pipe** command.



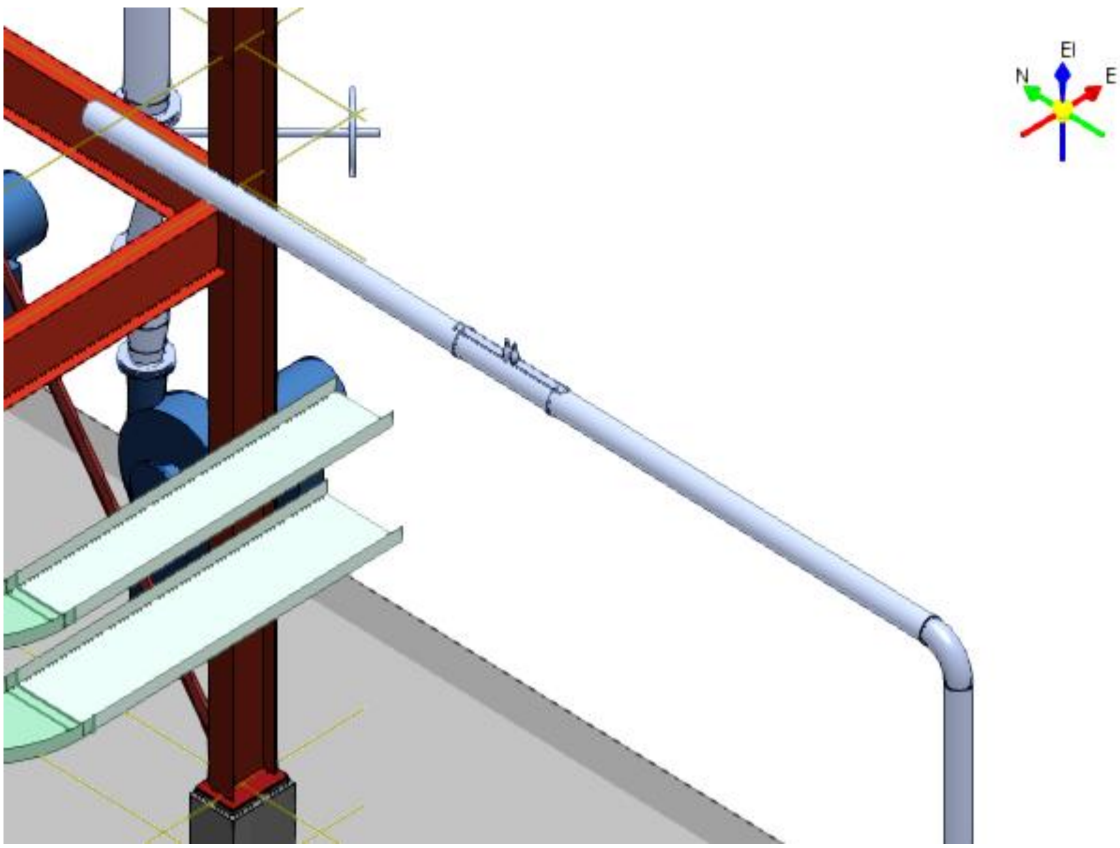
11. Notice that the pipe executes an offset of 3 inches (Insulation Thickness) while routing.
12. Select **Insert Component** on the vertical toolbar, and the select the midpoint of the pipe that was just placed.

    The Type drop-down list on the Insert Component ribbon opens and displays the list of components that you place on the pipe.

13. Select the **Clamp-on Sensors** from the list. An outline of the Component then appears in the Graphical view.



14. Click **Finish** to place the component. Note that this is a surface mounted component. The component uses the surface of the placed pipe as a reference unlike the components that have been placed up to this point.



SESSION 5

Inserting Components in a Pipe Run

Objective

By the end of this session, you will be able to:

- Place valves, bends, tee-type branches, olet-type branches, reducer components, and other components by using the **Insert Component** command.
- Place a branching component on a skew line by using the point along option.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- Piping: An Overview
- Routing Pipes

Overview

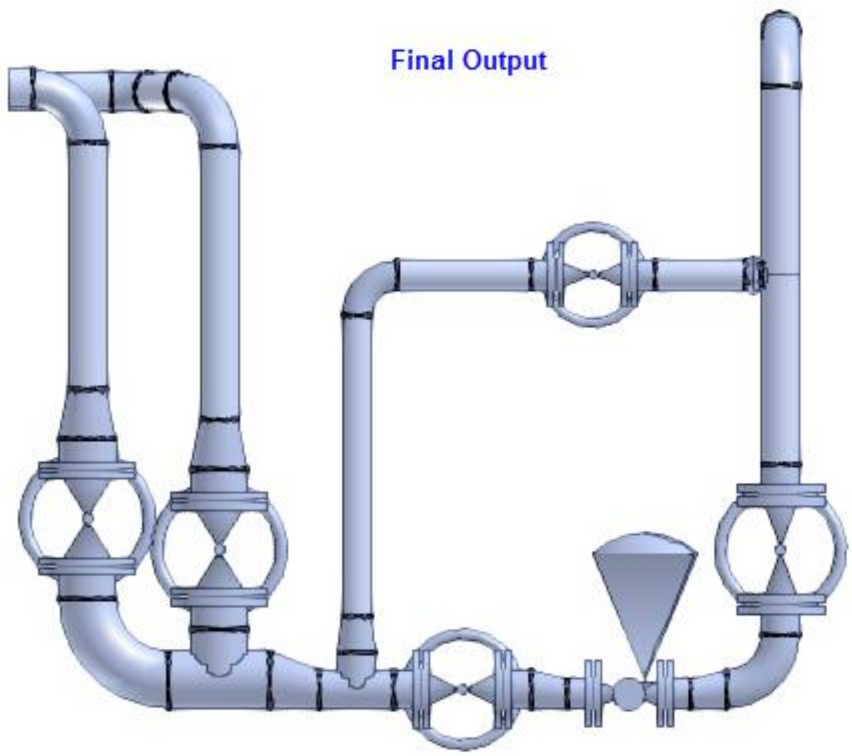
The **Insert Component** command adds piping, instrument, and specialty components to pipe runs. This command can be used any time during the design process. You can add components either during or after the routing of a pipe, or one after another, for fitting-to-fitting layout.

Placement operations are specification-driven. SP3D uses the piping specifications, the nominal diameter of the selected pipe run, and the active placement point to create a list of valid component types. For example, if the active placement point is not at the end of a pipe run or at an equipment nozzle, turn components are not included in the list of available component types. During insertion, the command cuts the pipe, when necessary, and inserts a base in-line component and the mating and connection parts required to connect the inserted part to the adjacent objects.

When inserting components such as Default branch, Default Turn, and Default Reducer SP3D enables you to place a pre-defined default of that component. Upon placement of the Default component the system gathers definition information from branch Table for the Default Branch Components and in the Piping Specification for the Default Turn and Default Reducer components.

Insert Component Command

You use the Insert Component command to place components such as valves, steam traps, strainers, tees, turns, fire protection items, and other piping items in a model.



You can also use the **Insert Component** command to select a component from a P&ID to export it to your model if any one of the following is true:

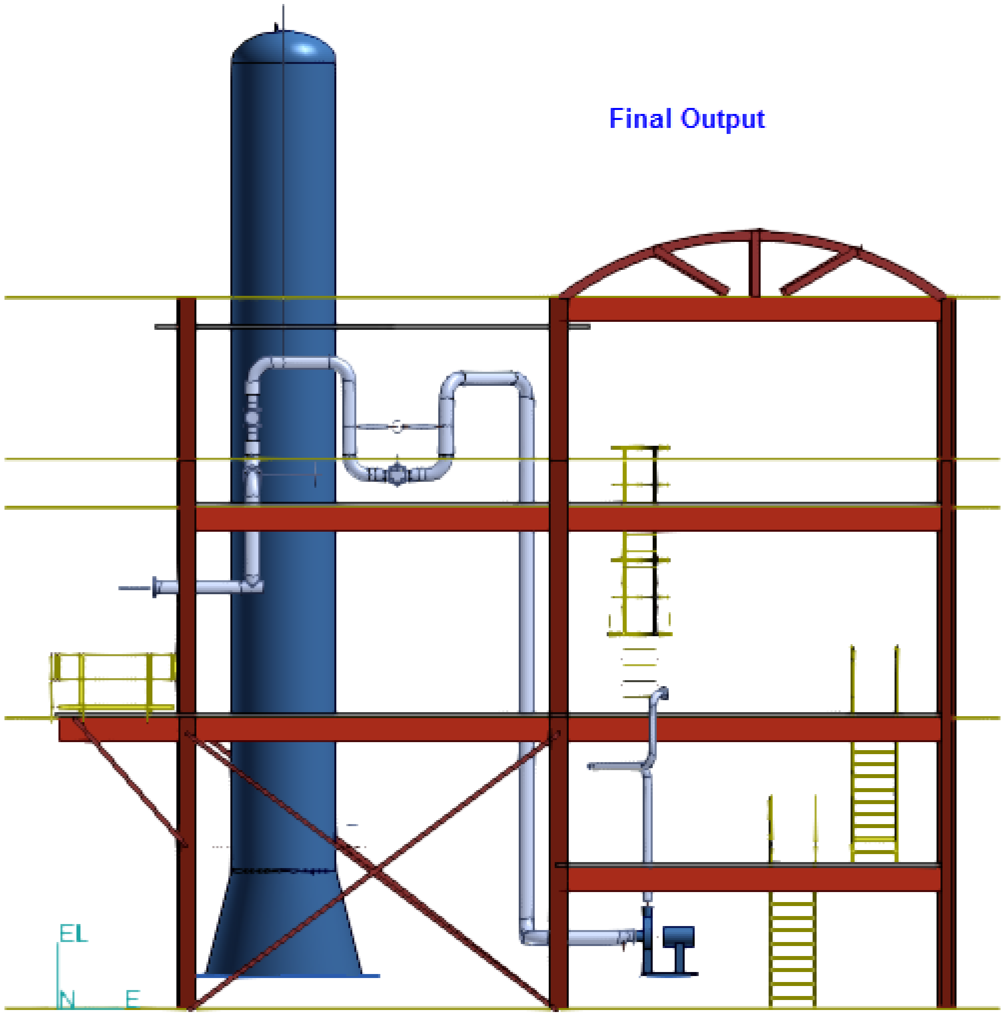
- Design basis data has been made available from the P&ID.
- Selected equipment nozzle is correlated with the P&ID.
- Run of the selected feature or port is correlated with the P&ID.

Inserting Components While Routing a Pipe Run

Objective

By the end of this session, you will be able to:

- In this exercise you will be routing two pipelines **300-W** and **301-W** from the **Pump P-101** or suction nozzle to the top of the tower in **Unit U03** of your workspace using the **Route Pipe** command. Then, use the **Insert Component** command to place components at various locations. The routed pipe section after inserting the components will look like this:



Before Starting this Procedure

- Define your workspace to display **Unit U03** and coordinate system **U03 CS**. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

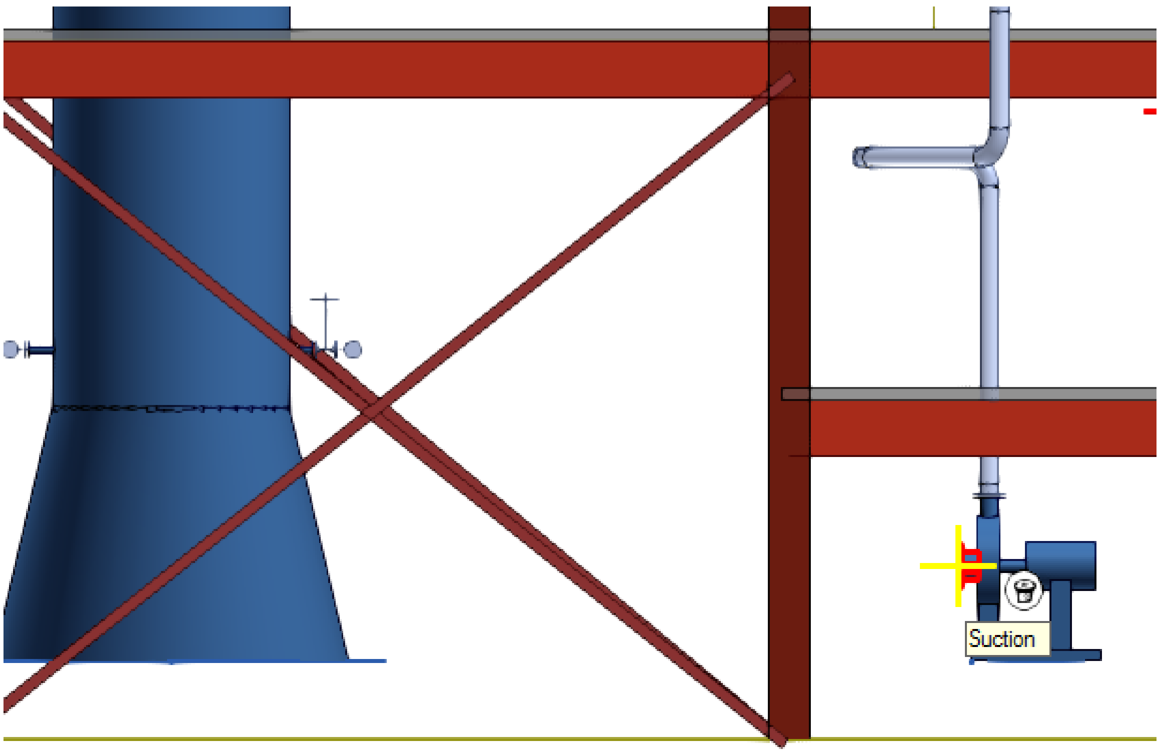
Inserting a Flange at an Equipment Nozzle

1. Activate **PinPoint** on the **Common** toolbar, and then click **Relative Tracking** on the **PinPoint** ribbon.

2. Change the view to **Looking North** using the **Common Views**  button on the toolbar. This will enable you to get a better view of Pump P-101.

3. Click **Insert Component**  on the vertical toolbar.

4. Use the SmartSketch options to locate the suction nozzle of **Pump P-101**, and click the nozzle.



The New Pipe Run dialog box appears.

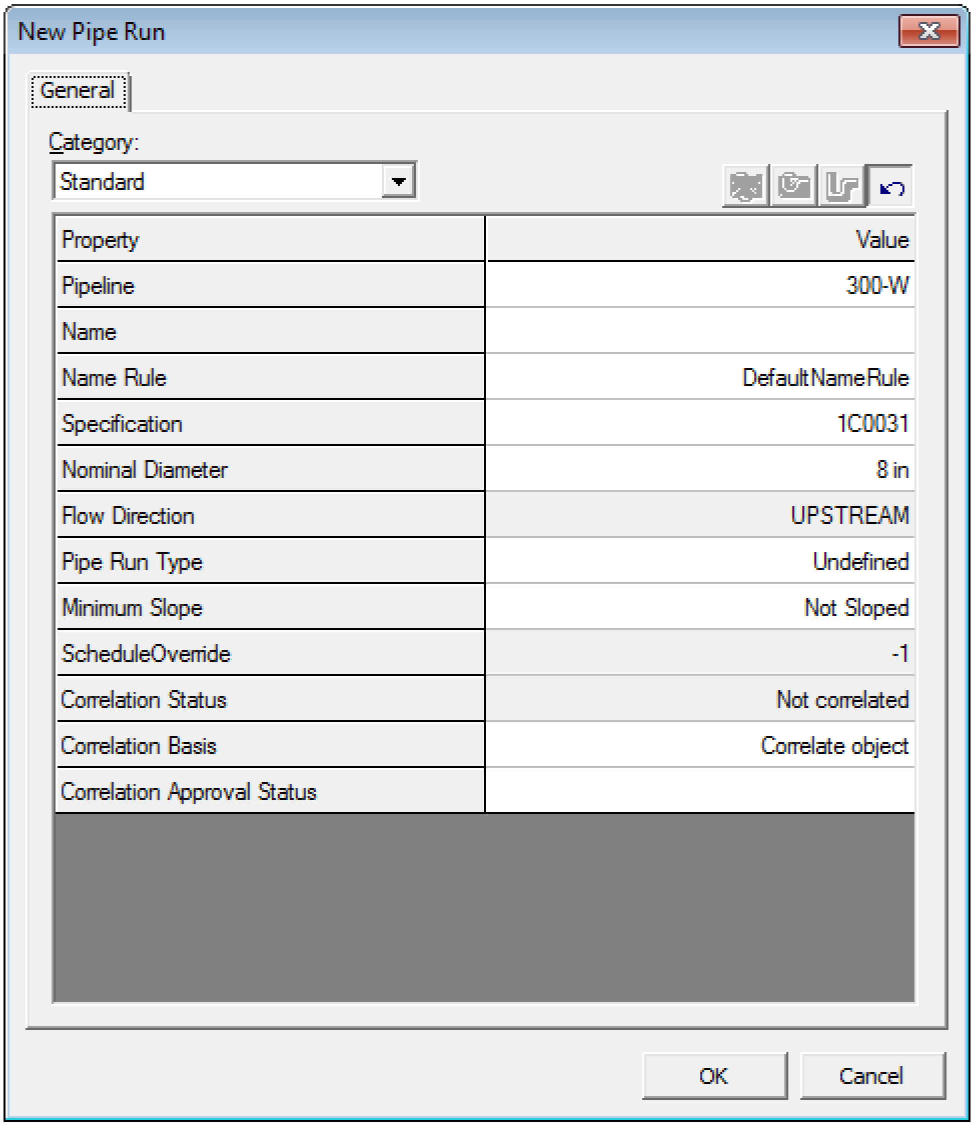
5. Select 300-W in the Pipeline field.

   The system selects the other parameters by using the piping specification.

6. Ensure that the New Pipe Run dialog box displays the following parameters, and click **OK**:

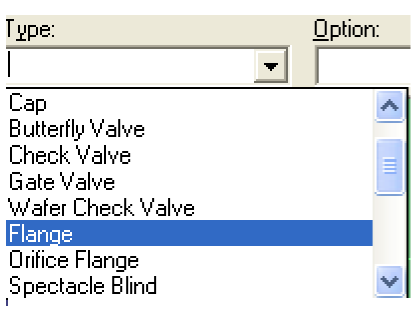
   Pipeline: 300-W
   Name Rule: DefaultNameRule
   Specification: 1C0031
   Nominal Diameter: 8 in
   Flow Direction: UPSTREAM
   Minimum Slope: Not Sloped

Schedule Override: <undefined value>
Correlation Basis: Correlate object



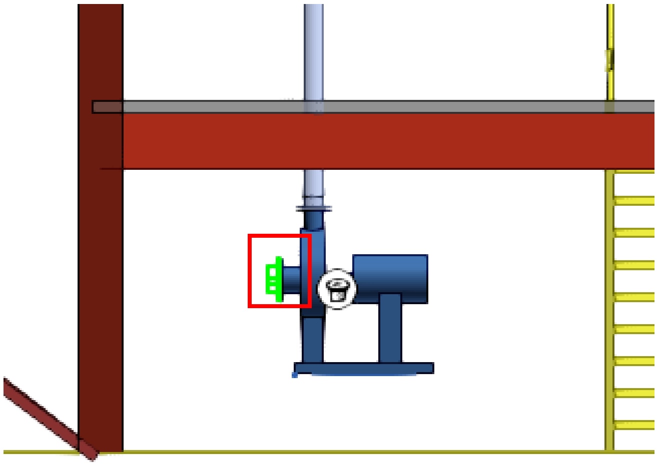
*In the **Insert Component** ribbon, the **Type** drop-down list displays a list of components that you can choose to place in your model.*

7. Select **Flange** from the list.

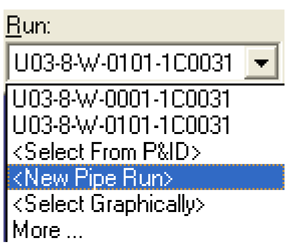


Placement operations are specification-driven. Smart 3D uses the pipe specification, the nominal diameter of the selected pipe run, and the active placement point to create a list of valid component types.

You will now see an outline of a flange on the suction nozzle of Pump P-101.



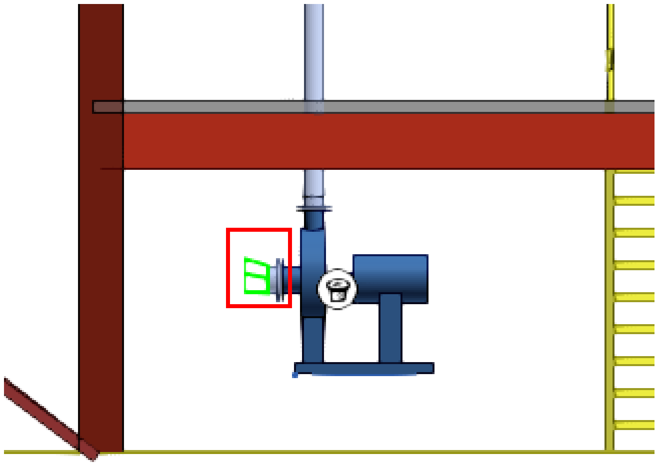
8. Click **Finish** on the **Insert Component** ribbon to place the flange.
9. Now, select the **Eccentric Size Change** option in the **Type** drop-down list on the **Insert Component** ribbon.
10. You are required to place a new pipe run so select the **New Pipe Run** option in the **Run** drop-down list on the **Insert Component** ribbon.



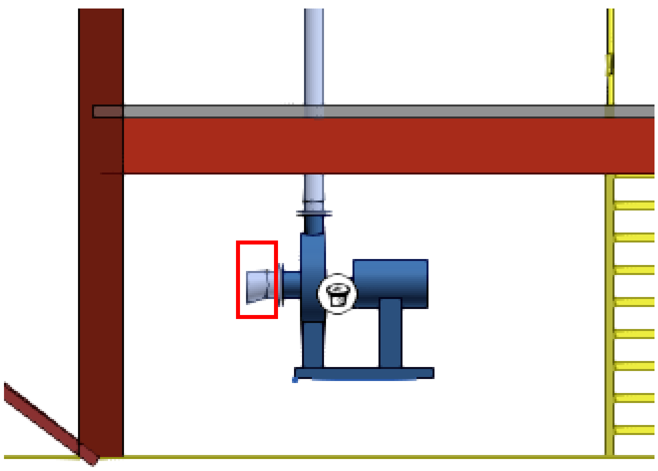
The New Pipe Run dialog box appears.

11. Change the **Nominal Diameter** to **10 in**, and then click **OK**.

The outline of an eccentric reducer appears in the graphic view.

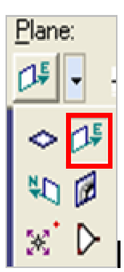


12. The flat side of the eccentric reducer points downwards. In the **Angle** drop-down list on the ribbon, type **180 deg** to rotate the eccentric reducer. Then, click **Finish** on the ribbon to place the reducer.



Basic Routing Techniques using PinPoint and Offset Method

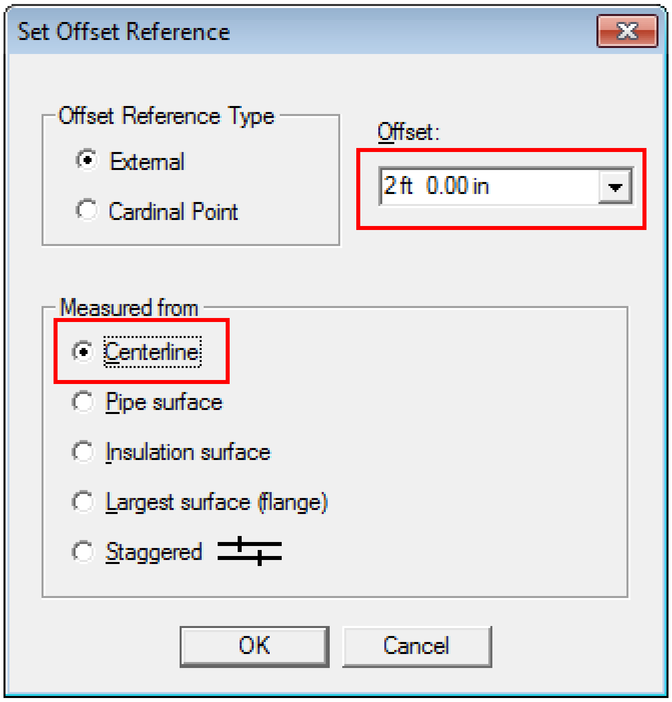
1. Click Route Pipe on the vertical toolbar.
2. Select the **Elevation Plane: East-West** option in the **Plane** drop-down list on the **Route Pipe** ribbon.



3. Under **Offset** on the **Route Pipe** ribbon, select the **Set Offset Reference** option to set the options for reference offsets while routing a pipe run.

   The Set Offset Reference dialog box displays.

4. Set the offset reference by **Centerline** and key in **2 ft** in the **Offset** drop-down list in the **Set Offset Reference** dialog box.

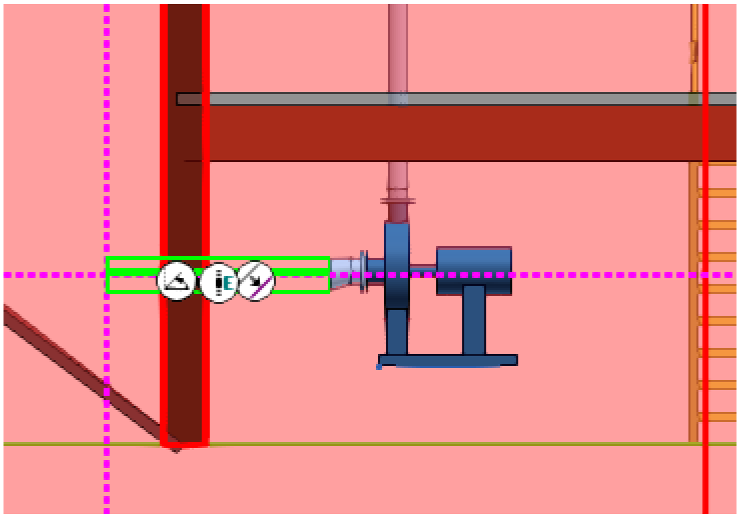


TIP There are two types of offset references:

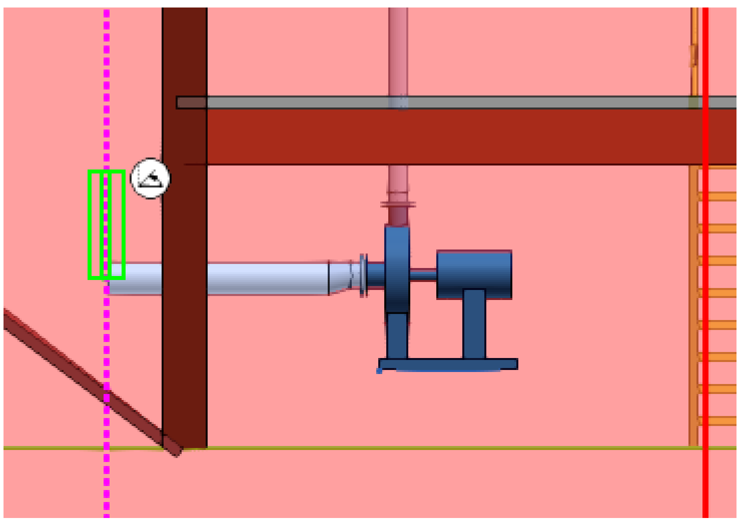
- **External** - Route a pipe a specified distance from another object, such as another pipe running parallel to the one you are placing.
- **Cardinal Point** - Route a pipe by the top, sides, bottom, or invert elevation of the pipe instead of the pipe centerline. You must use **Tools > Pinpoint** and lock the plane when using this reference type.

5. Move the cursor over the column shown below until the offset glyph appears and move slowly away from the column until perpendicular projection line is displayed.

The system will display a projection line that indicates 2 ft offset from the column.

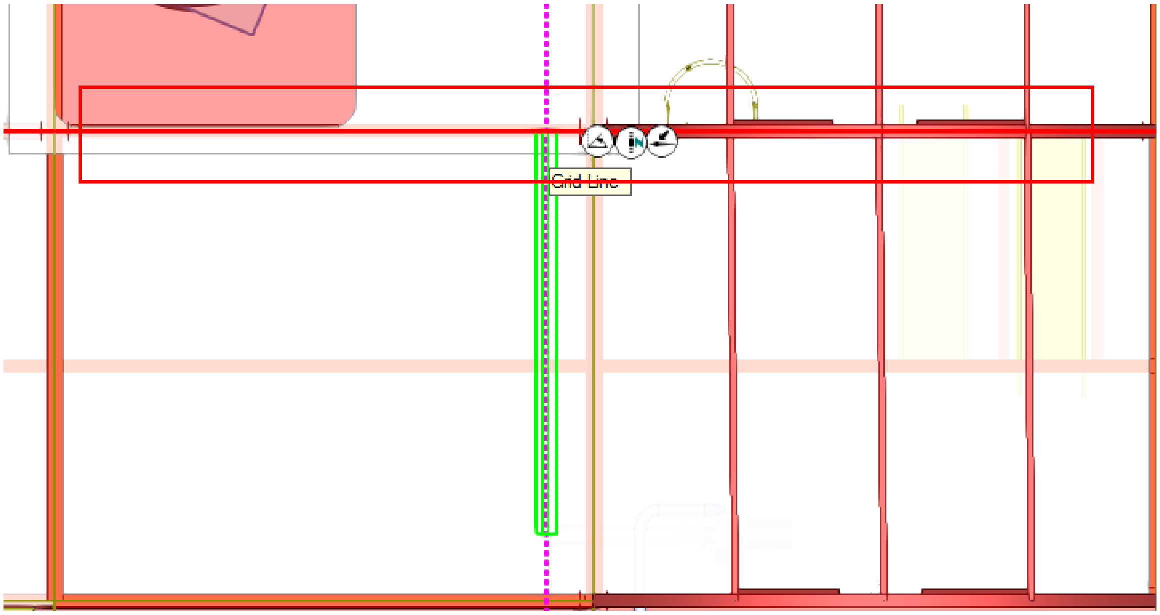


6. Click in the graphic view to place the pipe.

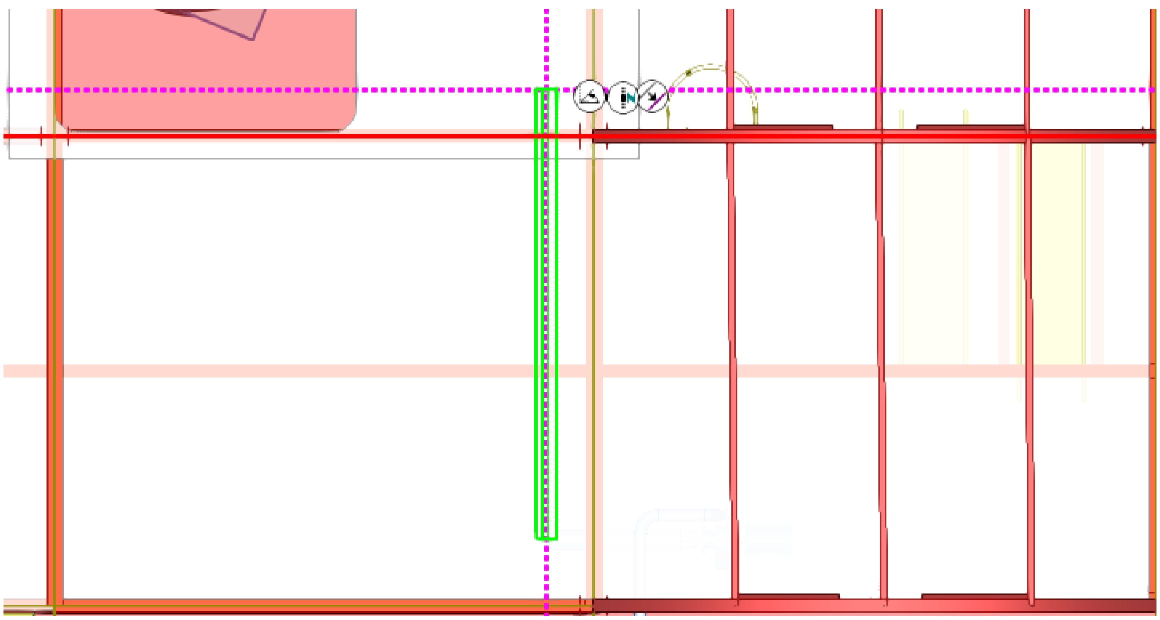


7. You need to route the next part of the pipe in the north direction. Use the **Common Views** button to change the view to **Looking Plan**. Then, select the **Plan Plane** option in the **Plane** drop-down list on the **Route Pipe** ribbon.

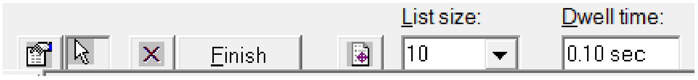
8. Now move the cursor in the north direction. You will see an outline of the pipe. Move the cursor to the grid line highlighted to add it to SmartSketch.



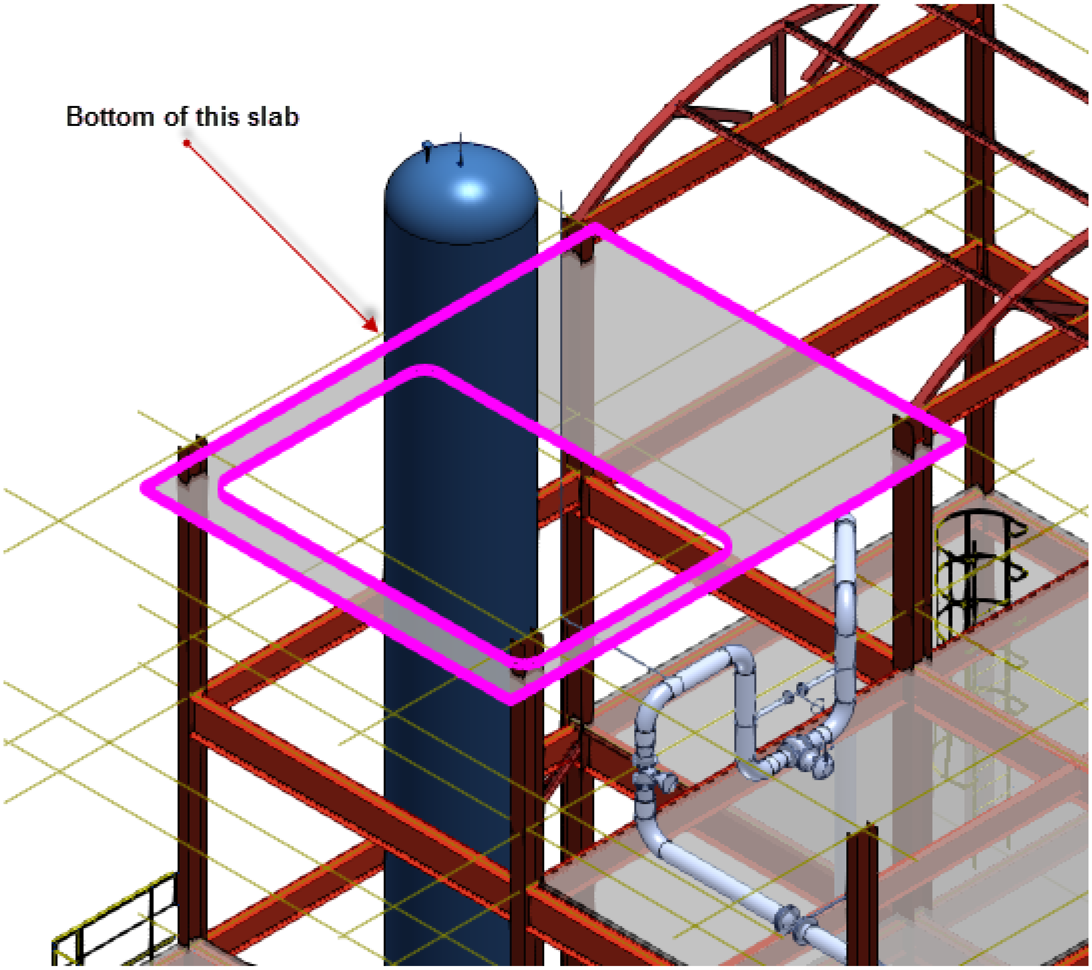
9. Locate the projection line displayed by the system at **2 ft** offset from the grid line and click in the graphic view to place the pipe.



10. Now, you need to route the next segment of the pipe run to the roof of the model. Click **Add to SmartSketch List** on the **Common** toolbar.
11. Some objects might already be in the list and might be highlighted. Clear the list by selecting the red "X" on the SmartSketch List ribbon bar.

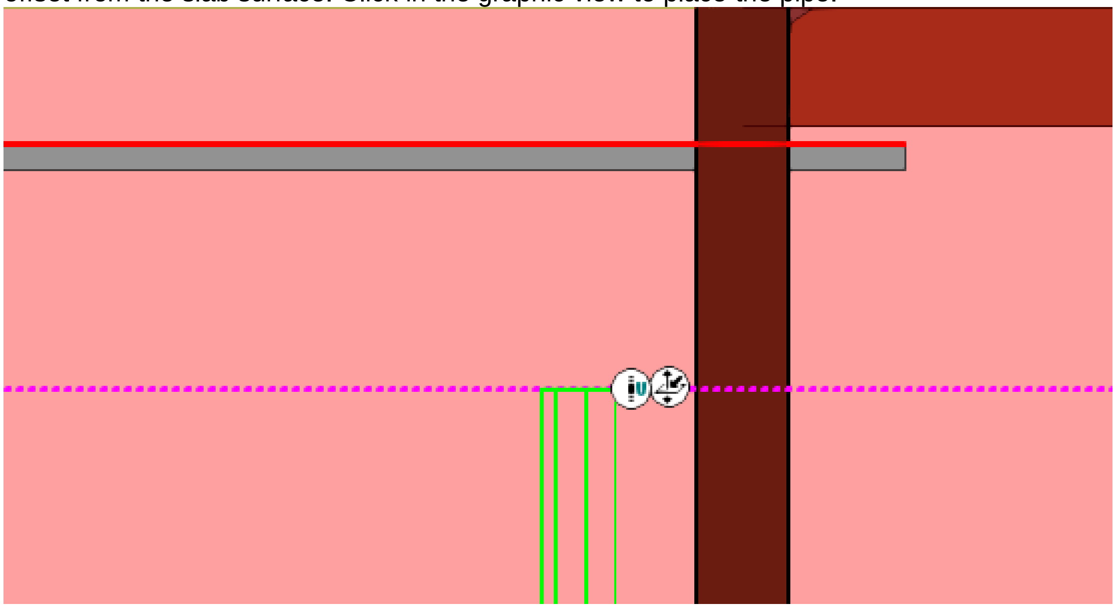


12. Select the bottom surface of the slab to add it to the list.

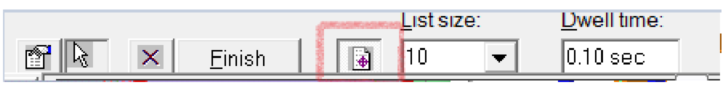


13. Click **Locate from List Only**
14. Click **Finish** to close the SmartSketch list ribbon.
15. Click the **Plane** drop-down list on the ribbon and select **Elevation Plane: East-West** to route the pipe.
16. Key in **3 ft** in the **Offset** drop-down list.

17. Use the **Common Views** button to change the view to **Looking North**.
18. Now, move the cursor upwards and locate the projection line displayed by the system at 3 ft offset from the slab surface. Click in the graphic view to place the pipe.

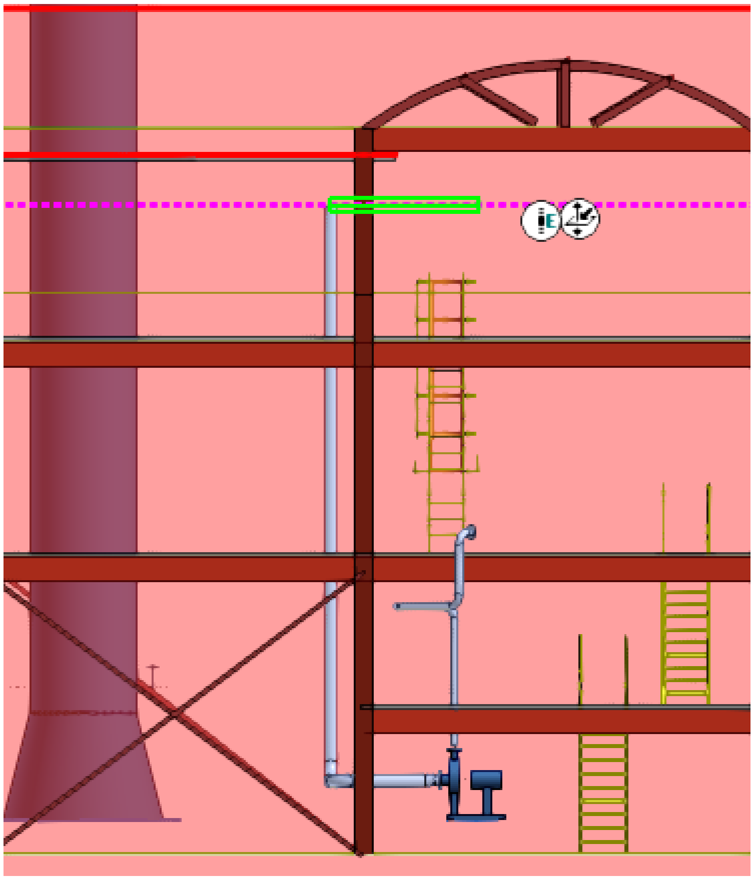


19. Right-click to cancel the route piping command.
20. Go back to the **SmartSkecth List Ribbon**, and turn off the list option.



21. Use the **Reposition Target** command (from the Pinpoint ribbon bar) to place the target at the end of the pipe that you just routed.

22. Use the **Route Pipe** command, and re-connect to the end of the piping that you just completed (and where the target is currently locate).



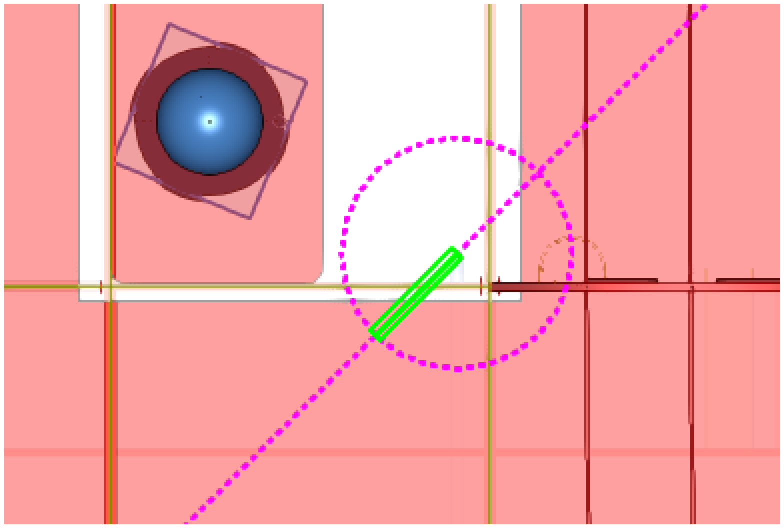
23. Now, you will route the next segment of the pipe in the west-south direction by using **PinPoint**. Use the **Common Views** button to change the view to **Looking Plan**.

24. Click the **Spherical Coordinates**  option on the **PinPoint** ribbon.

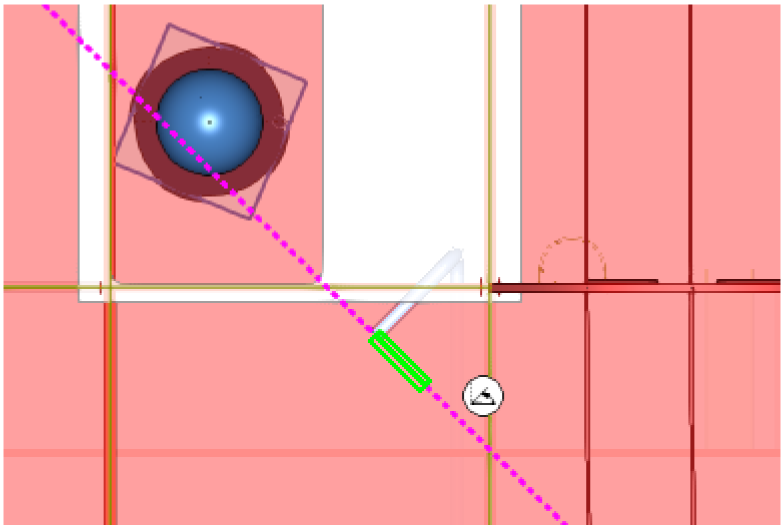
25. Select the **Plan-Plane** option in the **Plane** drop-down list on the **Route Pipe** ribbon.

26. On the **PinPoint** ribbon, key in **7 ft** in the **Distance** drop-down list and **45 deg** in the **Horizontal** drop-down list.

The system displays the constraints within which the pipe can be placed.

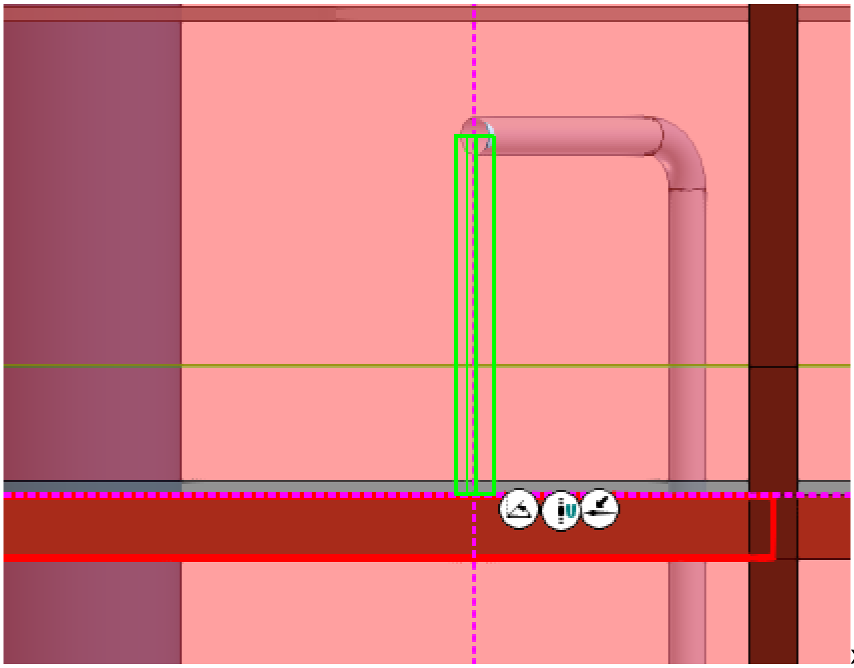


27. Click in the graphic view to place the pipe. The view of your model should resemble this.



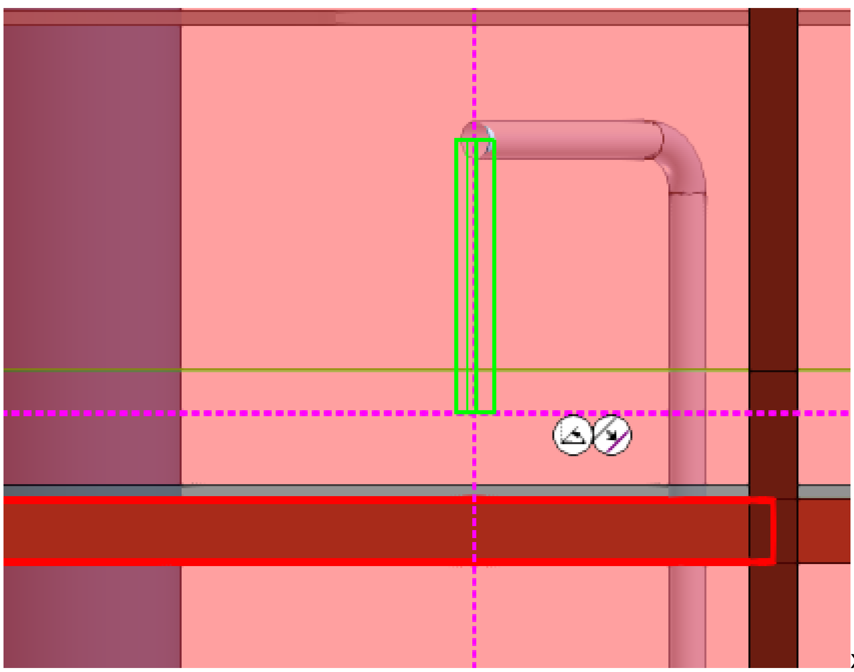
28. Now, use the **Common Views** button to change the view to **Looking North**.
29. Select the **Elevation Plane: East-West** option in the **Plane** drop-down list to route the pipe.
30. Key in **2 ft** in the **Offset** drop-down list.

31. Move the cursor to Beam as shown below to add it to SmartSketch.



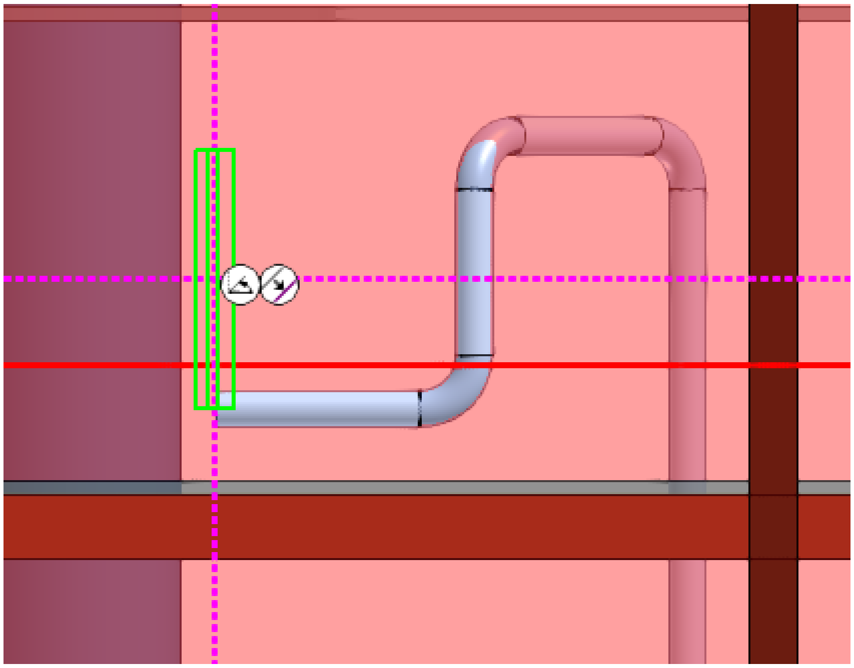
x

32. Now move the cursor upwards to locate the projection line displayed by the system at **2 ft** offset from **Beam** as shown above. Click in the graphic view to place the pipe.

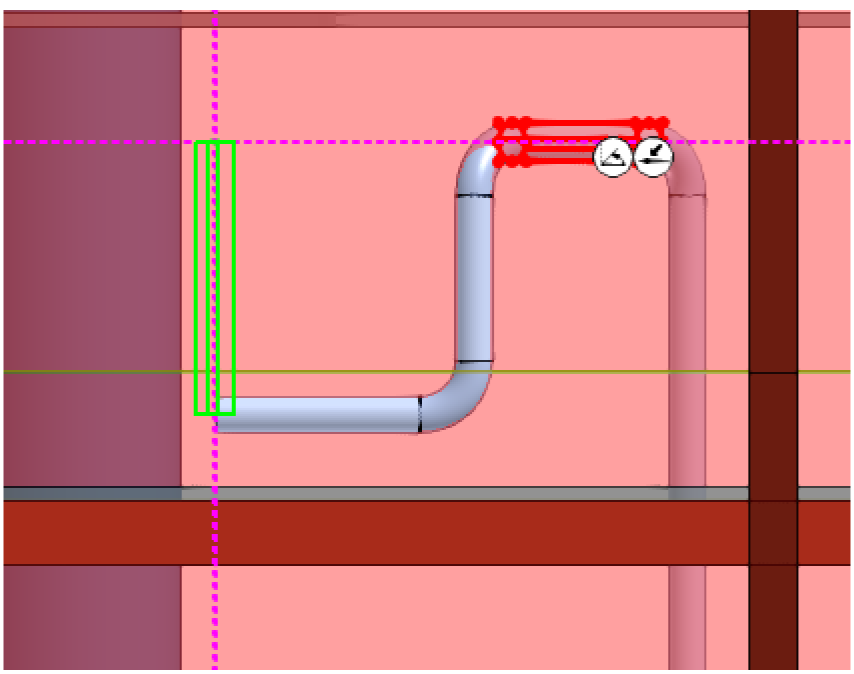


x``x

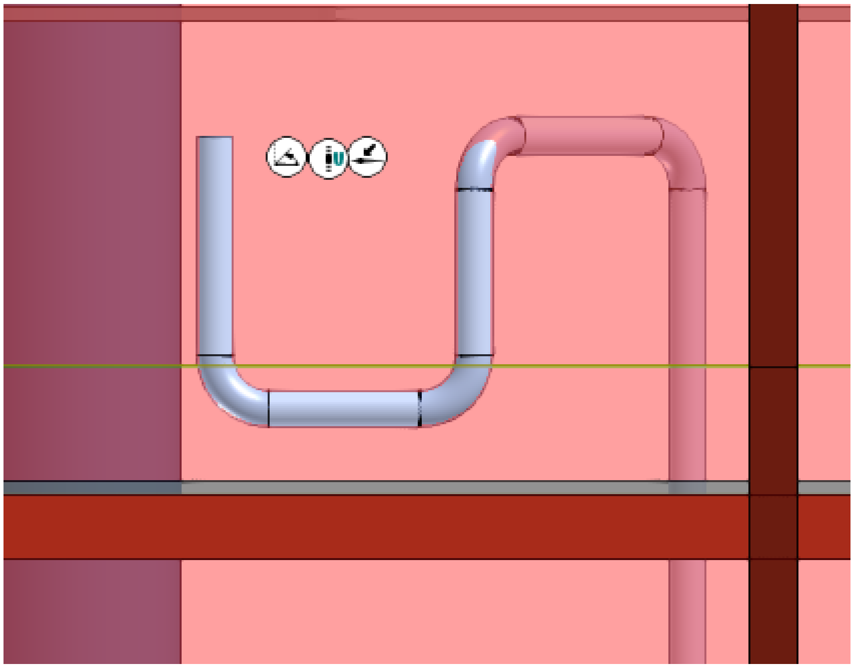
33. Now, you will route the next segment of the pipe in the west direction. On the ribbon, key in **6 ft** in the **Length** drop-down list and click in the graphic view to place the pipe. Unlock the **Lock Length** control on the **Route Pipe** ribbon.



34. Now, you will route the next segment of the pipe upwards. Use the **Pipe Straight Feature** highlighted to find the intersection point.

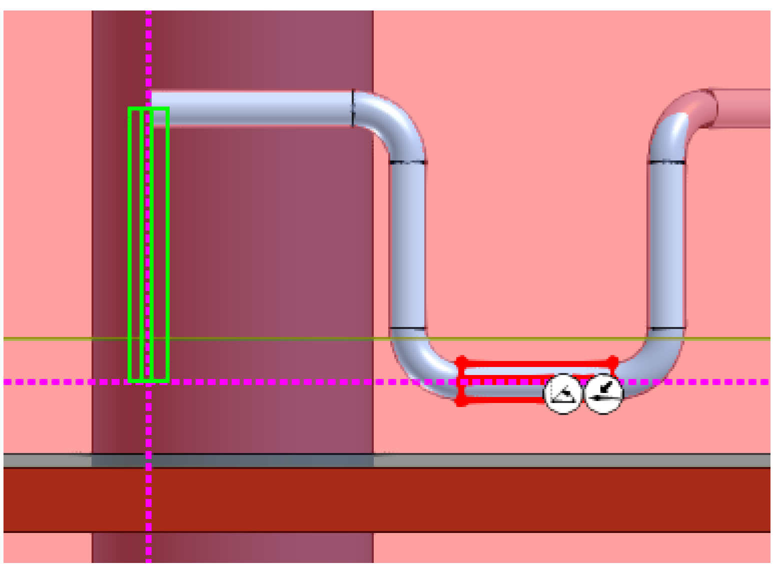


35. Click in the graphic view to place the pipe.



36. Now, key in **6 ft** in the **Length** drop-down list and move your cursor to the west direction. Click in the graphic view to place the pipe. Unlock the **Lock Length** control on the **Route Pipe** ribbon.

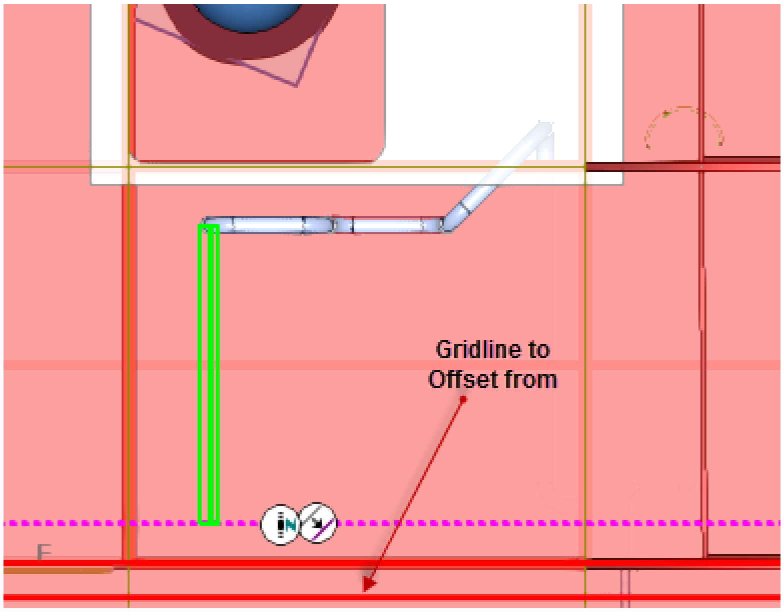
37. Move your cursor downwards and use the **Pipe Straight Feature** highlighted to locate the intersection point. Click in the graphic view to place the pipe. The view of your model should resemble this.



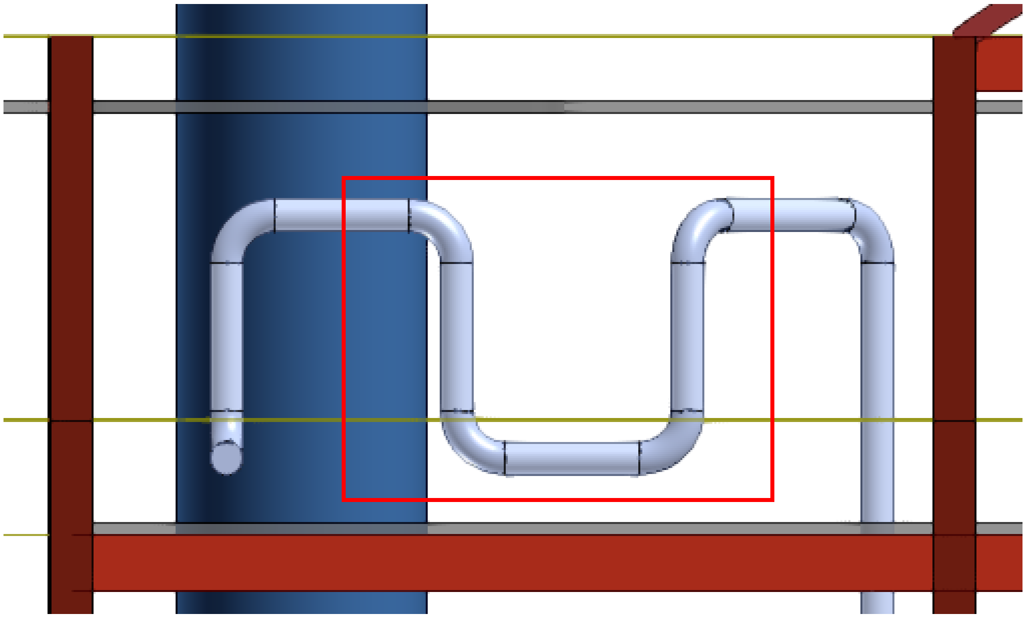
38. Now, you will route the next segment of the pipe in the south direction. Use the **Common Views** button to change the view to **Looking Plan**.

39. Click the **Plane** drop-down list on the ribbon and select **Plan Plane** to route the pipe.

40. Move the cursor to the grid line highlighted to add it to the SmartSketch list. Then, move the cursor upwards to locate the projection line displayed by the system at 2 ft offset from the grid line.
41. Click in the graphic view to place the pipe.



42. Right-click in the graphic view to terminate the **Route Pipe** command.
43. Now, place a 3 inch NPD bypass above the main pipe in between the pipes. Use the **Common Views** button to change the view to **Looking North**.



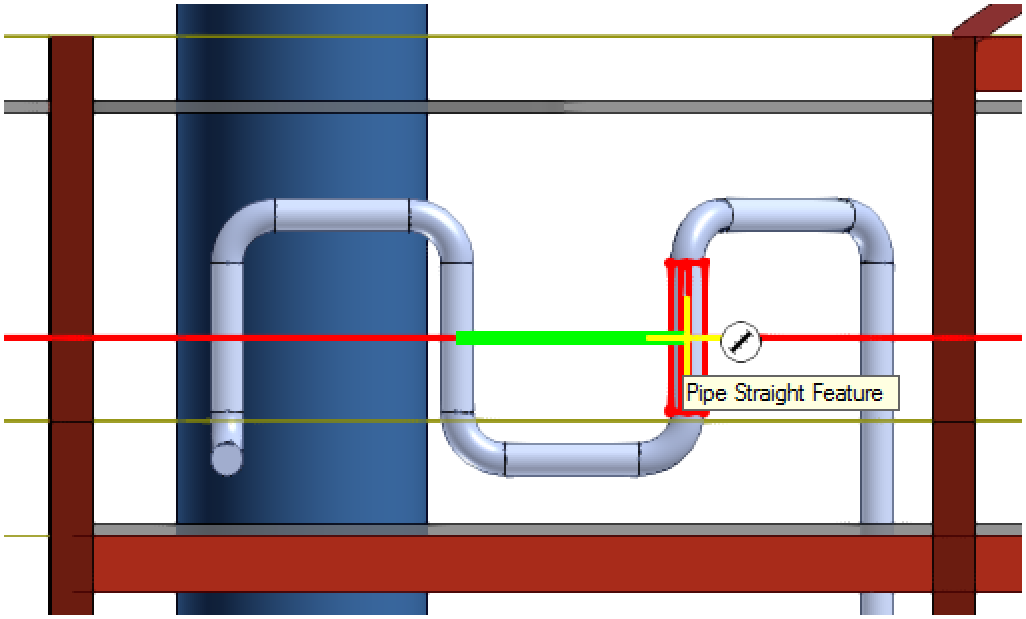
44. Click **Route Pipe** on the vertical toolbar. Use SmartSketch to locate the midpoint of the first pipe and click it.

45. Click the **Plane** drop-down list on the ribbon and select **No Plane** to route the pipe.

    The New Pipe Run dialog box appears.

46. Change the **Nominal Diameter** to **3in** and click **OK**.

47. In the graphic view, move the cursor to the next pipe and use SmartSketch to find the intersection point. Click the intersection point to place the pipe as shown.



48. Now, you will route a pipe from the **Nozzle C** at the top of the tower **T-101**.

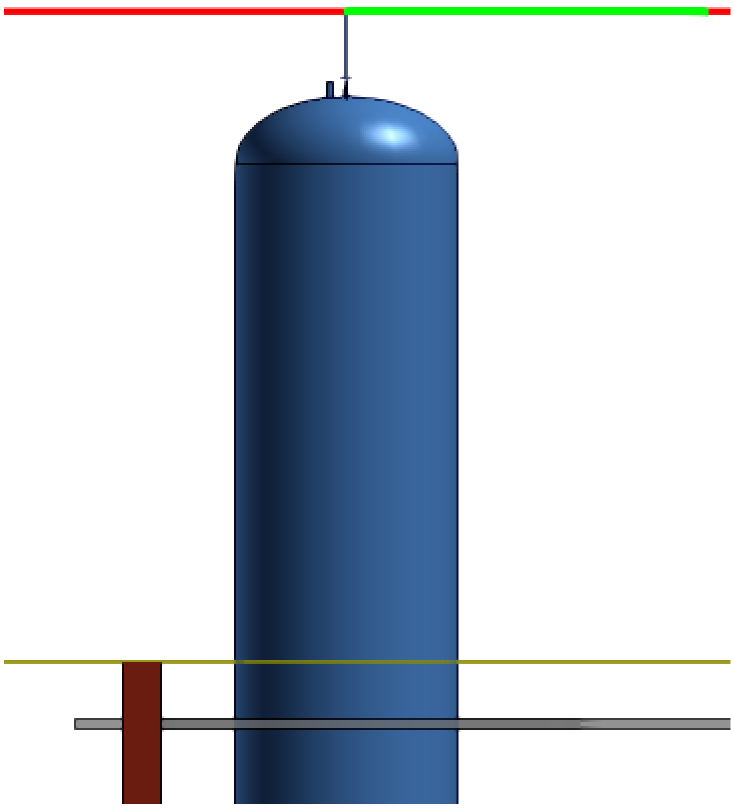
49. Click the **Relative Tracking** button in the **PinPoint** toolbar.

50. Click **Route Pipe** on the vertical toolbar and then click the nozzle at the top of the tower **T-101**.

*The **New Pipe Run** dialog box appears.*

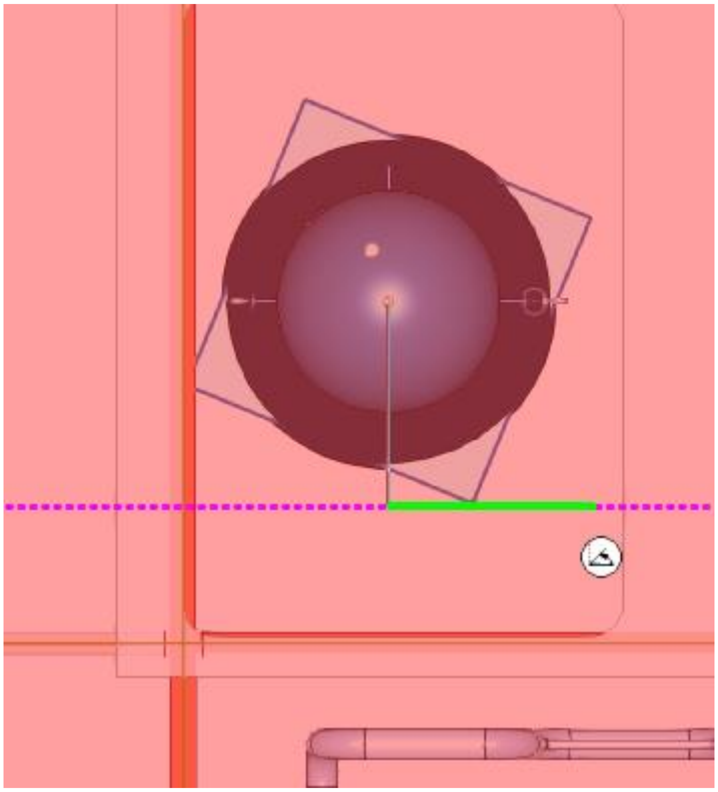
51. Change the **Pipeline** to **301-W** and click **OK**.
52. On the **PinPoint** ribbon, click the **Rectangular Coordinates**  option and key in **2 ft** in the **El** drop-down list.

53. Click in the graphic view to place the pipe.



54. Now, you will route the next segment of the pipe in the south direction. Use the **Common Views** button to change the view to **Looking Plan**.
55. Click the **Plane** drop-down list on the ribbon and select **Plan Plane** to route the pipe.

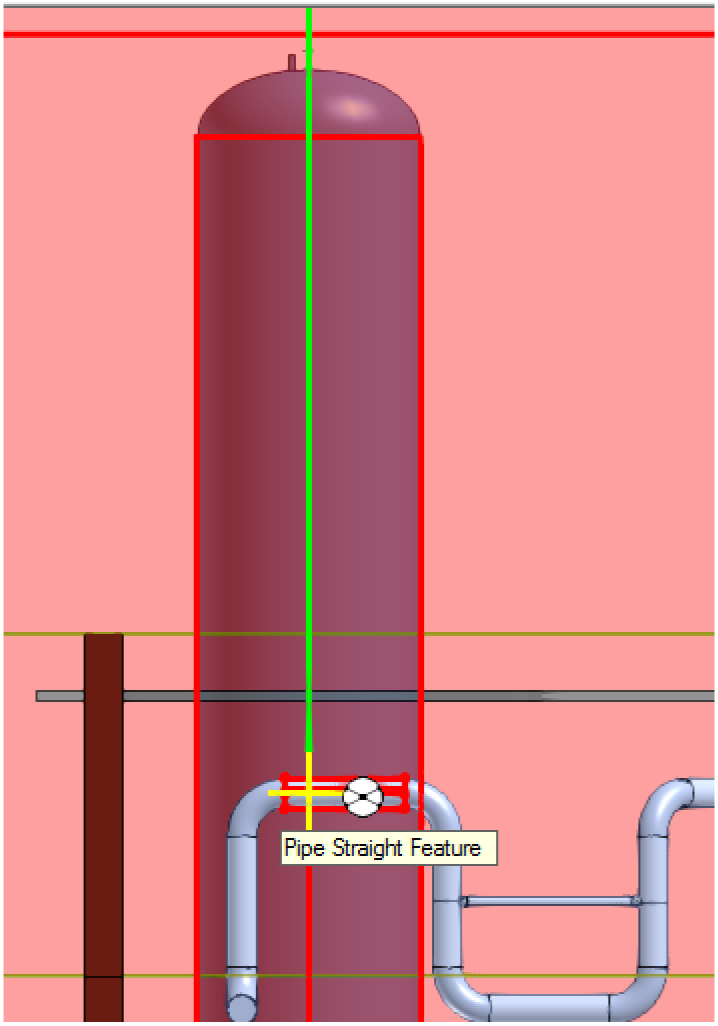
56. In the **Route Pipe** ribbon, key in **6 ft** in the **Length** drop-down list. Move the cursor straight downwards and click in the graphic view to place the pipe.



57. Now, use the **Common Views** button to change the view to **Looking North**.

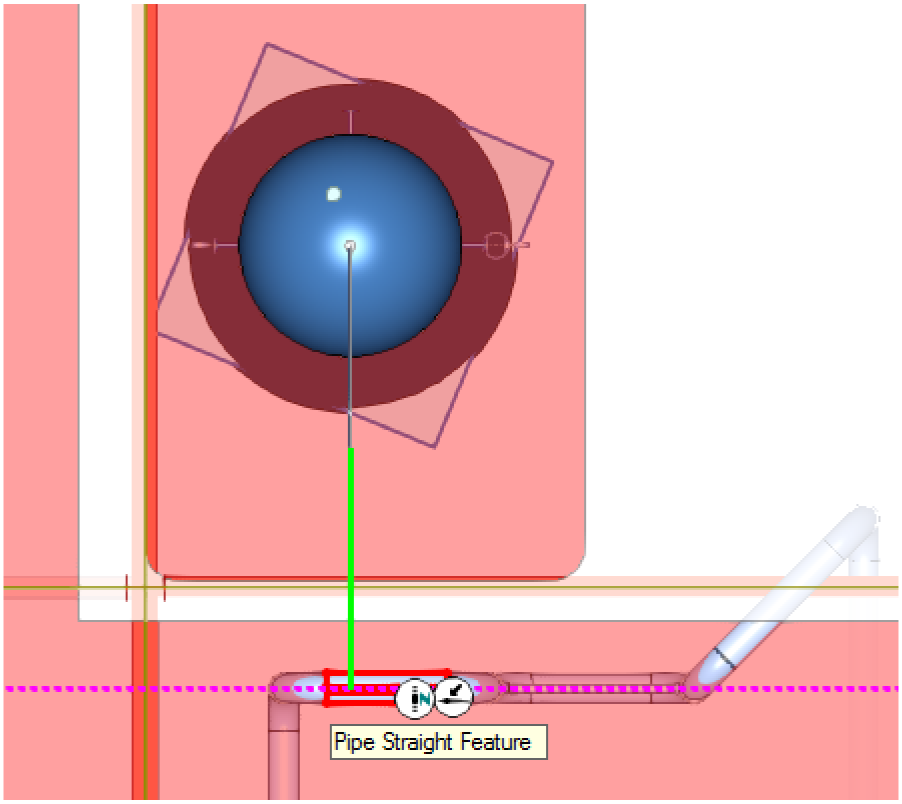
58. Click the **Plane** drop-down list on the ribbon and select **Elevation Plane: East-West** to route the pipe. Unlock the **Lock Length** control on the **Route Pipe** ribbon.

59. Move the cursor downwards to the already placed pipe and use SmartSketch to locate the intersection point. Click in the graphic view to place the pipe.

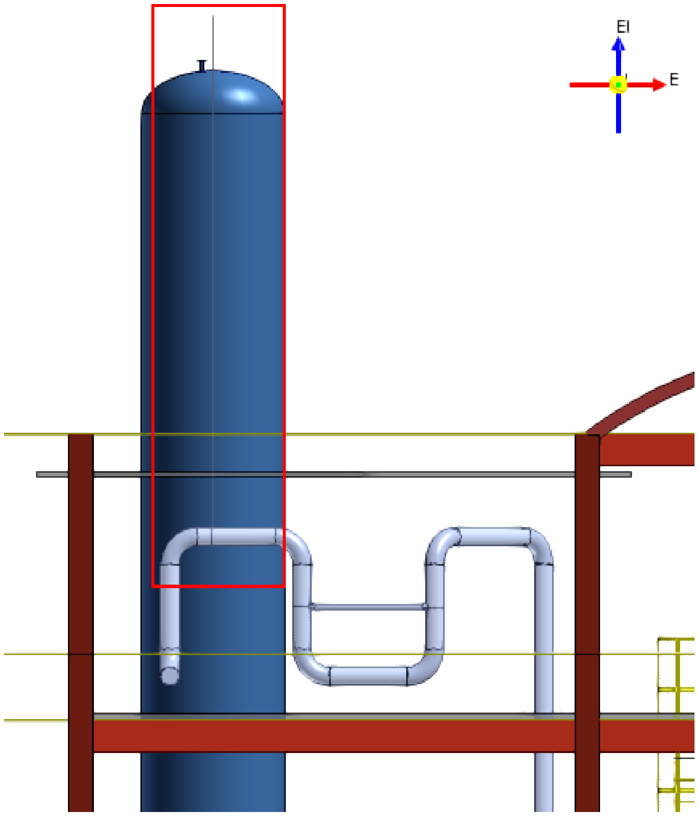


60. Now, use the **Common Views** button to change the view to **Looking Plan**.
61. Click the **Plane** drop-down list on the ribbon and select **Plan Plane** to route the pipe.

62. Move the cursor downwards to the already placed pipe and use SmartSketch to locate the intersection point. Click in the graphic view to place the pipe.

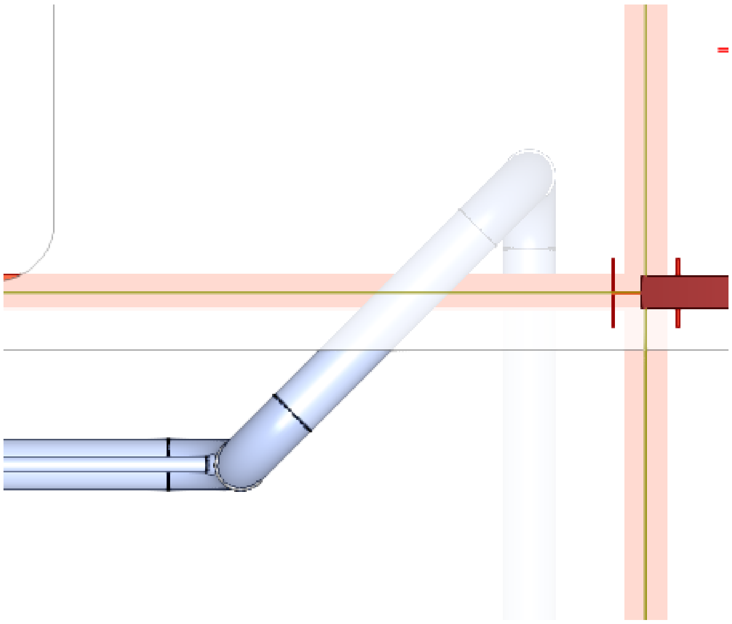


The view of your model should resemble this.

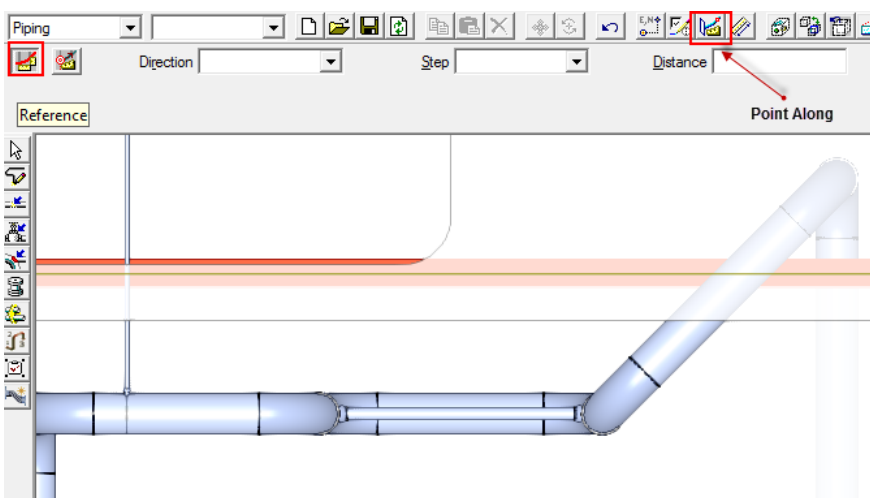


Inserting a Component by Point Along on a Skew Line

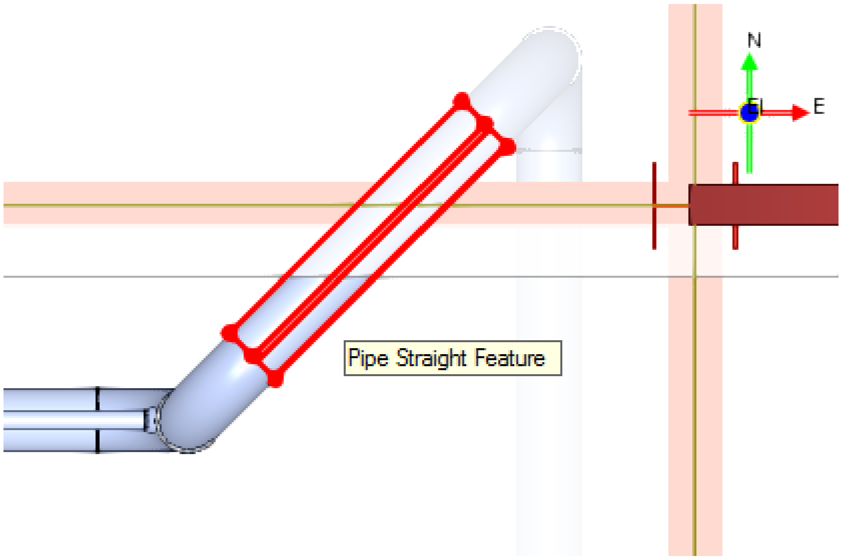
1. Bring the skew line on pipeline 300-W into plan plane view similar to below.



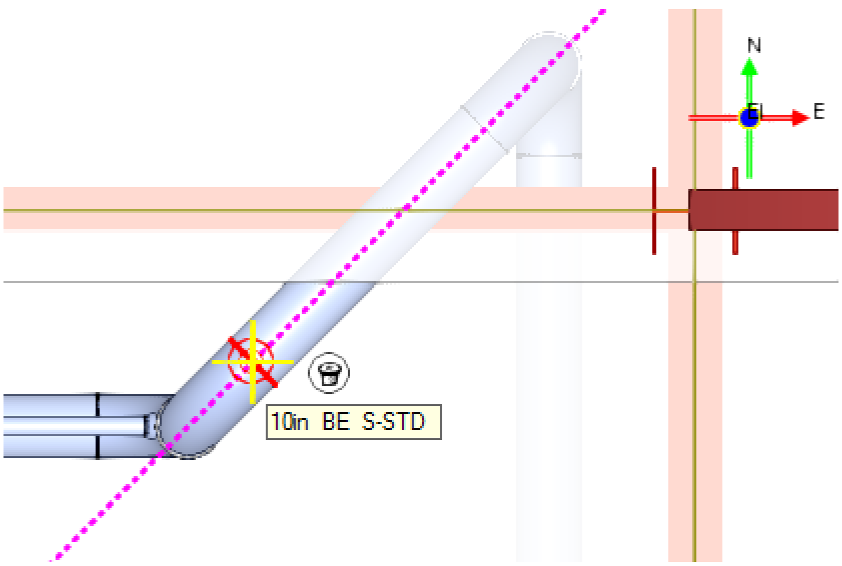
2. Click **Point Along**  on the Common ribbon, and then click the reference button on the **Point Along** ribbon.



3. Select **Along object** in the Direction dropdown menu. Left-click on the skew line in the graphic display.



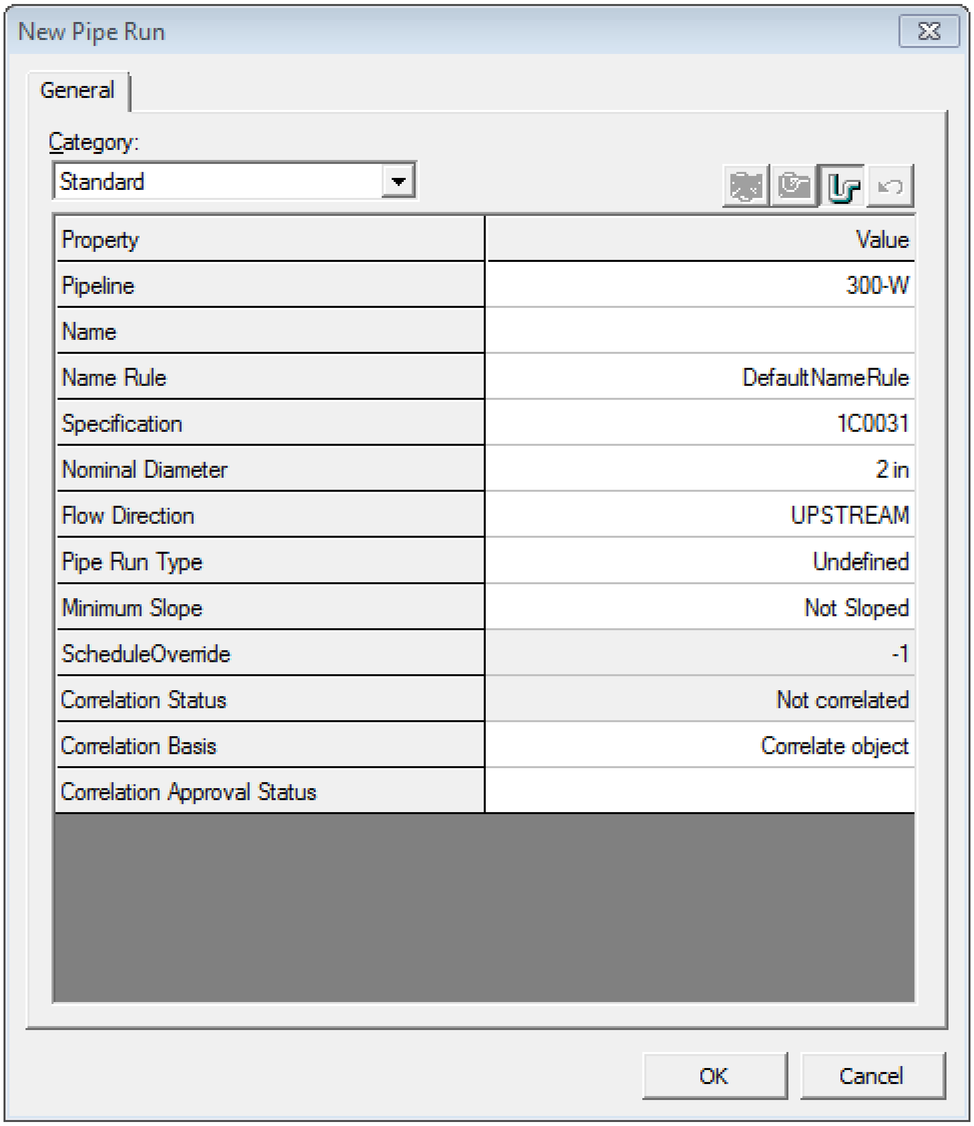
4. Reposition the target on the end of the straight feature.



5. Key in **3in** the **Distance** field.
6. Now click **Insert Component** on the vertical toolbar, and select the pipe on which to insert the component.

   The Type Drop-Down list displays a list of all the components.
7. Select **Weldolet** from the list.
8. Select the **<New Pipe Run>** option in the **Run** drop-down list on the **Route Pipe** ribbon.
9. Change the following parameters in the **New Pipe Run** dialog box and click **OK**.

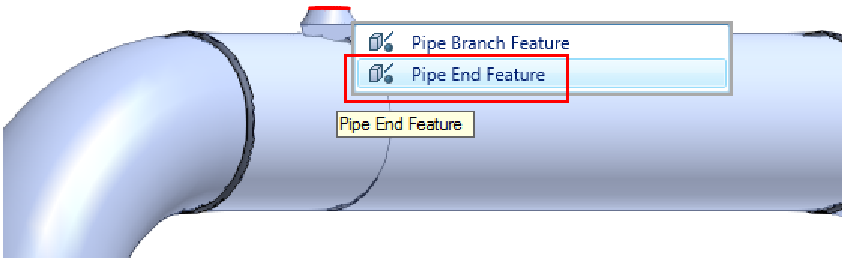
Nominal Diameter: 2 in.
Flow Direction: Upstream



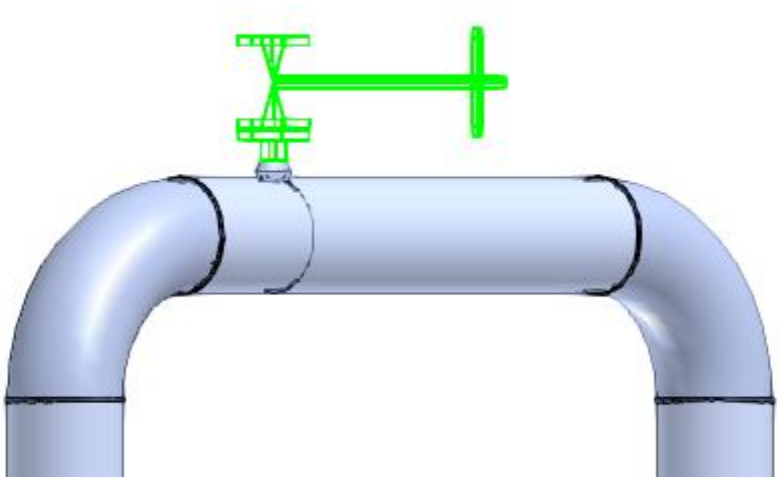
An outline of the Weldolet appears in the graphical view.

10. Click the **Enter Insertion Point**. Next, key in **1 ft** in the **Step** ribbon. Run the cursor along the pipe feature to notice that the distance is shown only in increments of 1 ft.
11. Left-Click in the graphic display and click **Finish**.

12. Select the weldolet end feature and click the **Insert Component** button on the Vertical Toolbar.



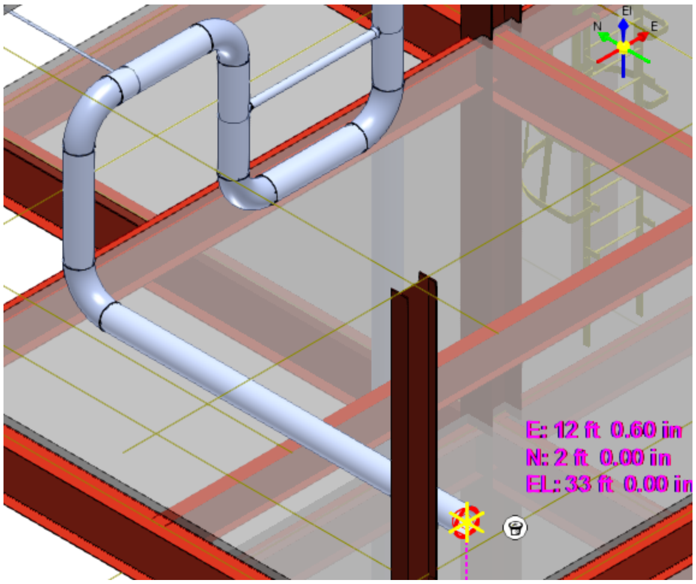
13. Select the **Instrument Root Valve** from the **Type** drop-down list. An outline of the Instrument Root Valve on the weldolet appears as shown.



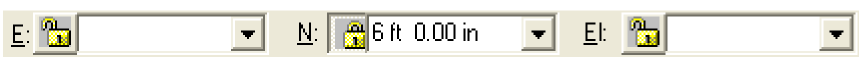
14. Click **Finish** to place the valve.

Inserting a Gate Valve a Known Distance from a Pipe End

1. Locate the free end of the pipeline 300-W, as shown. Click the **Reposition Target**  option on the **PinPoint** ribbon and put the target at the free end of the pipe.

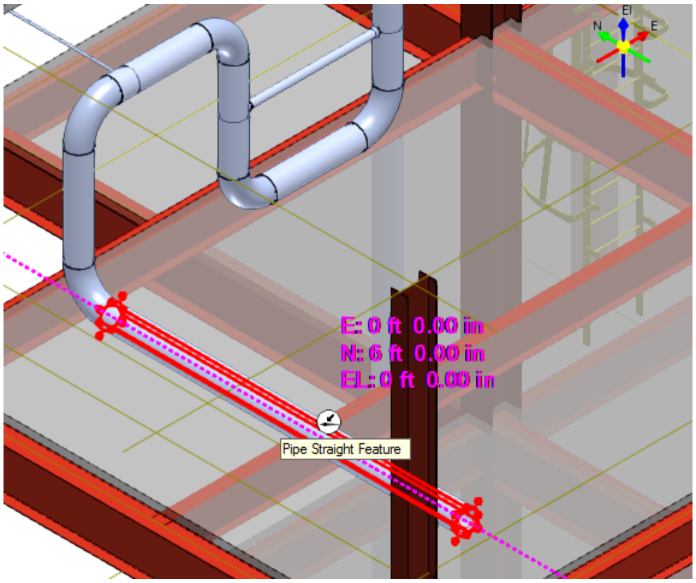


2. Use PinPoint to define the distance of 6 ft from the target. Because the pipe runs along the north-south direction, type **6 ft** in the **North** field on the **PinPoint** ribbon.



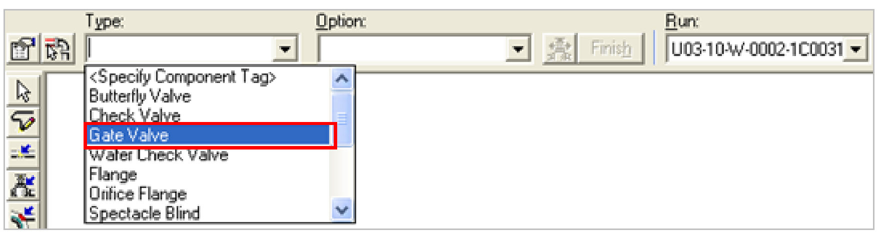
3. Click **Insert Component** on the vertical toolbar.

4. Now, position the cursor along the **Pipe Straight Feature**, as shown, and click to define the active placement point.



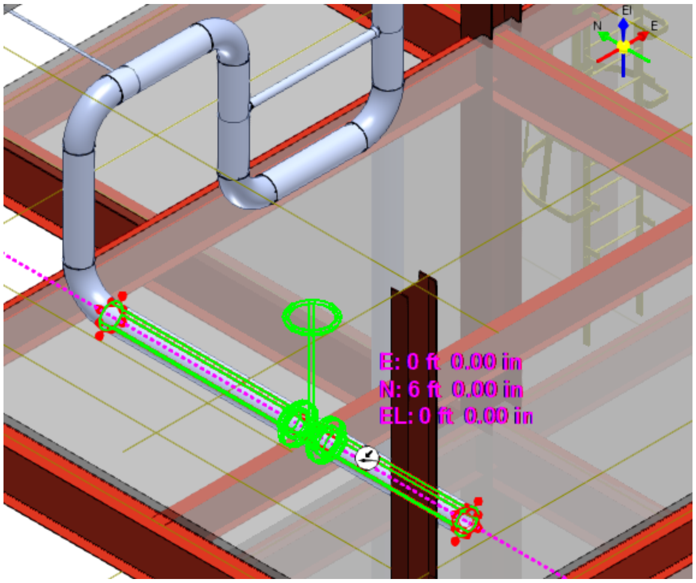
PinPoint constraints your cursor movement on an infinite north plane at 6 feet from the target. Therefore, you just need to provide a projection line along the Pipe Straight Feature so that the software finds the intersection point between the plane and the line.

5. In the **Type** drop-down list of the **Insert Component** ribbon, select the **Gate Valve** option.



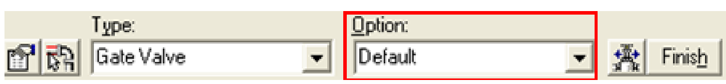
6. You will now see an outline of a valve (base part) with mating flanges at the active placement point, as shown.

If you select a component type that has an end preparation type or size that does not match the end preparation type or size of the adjacent component, SP3D uses the mating selection rule to select a mating component to be installed between the two incompatible ends. For example, if you choose a flanged valve, SP3D will choose flanges to be used between the pipes and the valve.

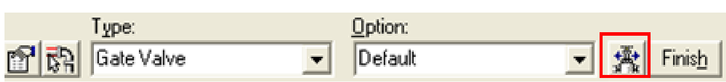


Notice that Smart 3D automatically selects the **Default** option in the **Option** drop-down list on the ribbon. The option selected in the Option drop-down list instructs Smart 3D to retrieve the primary commodity item, a secondary commodity item, or any other special option commodity item as defined in the piping specification.

If no default option is specified in the piping specification, Smart 3D prompts you to select an option in the **Option** drop-down list.



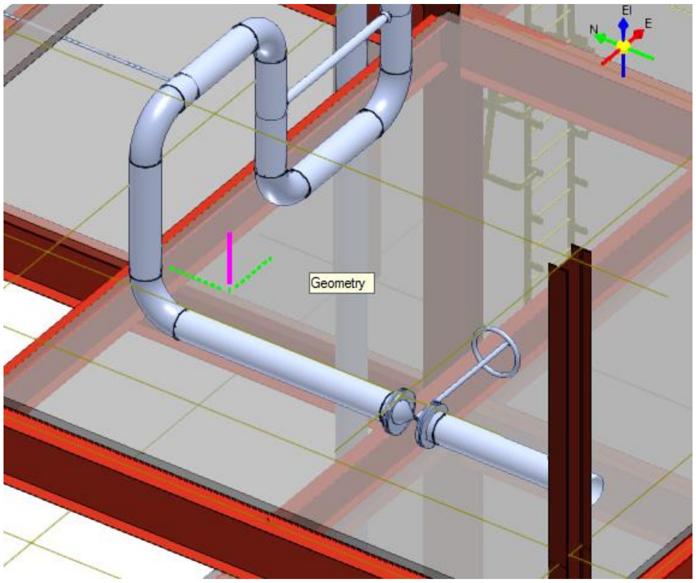
You can use the **Insert Point** step to redefine the component position that you are placing along the straight feature that you selected. The component is shown in dynamic display and moves with the mouse when in this step.



7. The **Angle** drop-down list on the ribbon displays the current rotation angle of the component. Key in **90 deg** in the **Angle** drop-down list to rotate the valve 90 deg about the axis of the feature path, as shown.

8. Click **Finish** on the **Insert Component** ribbon to place the valve.
9. Right-click in the graphic view to terminate the **Insert Component** command.

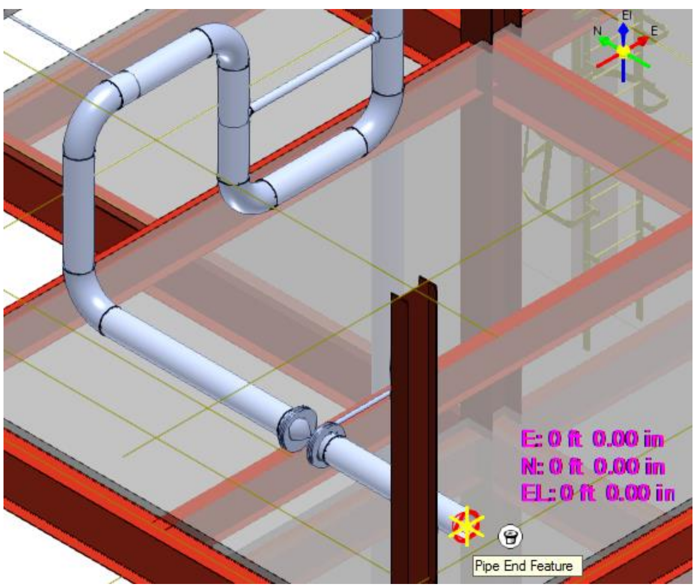
You have now placed a gate valve in your model.



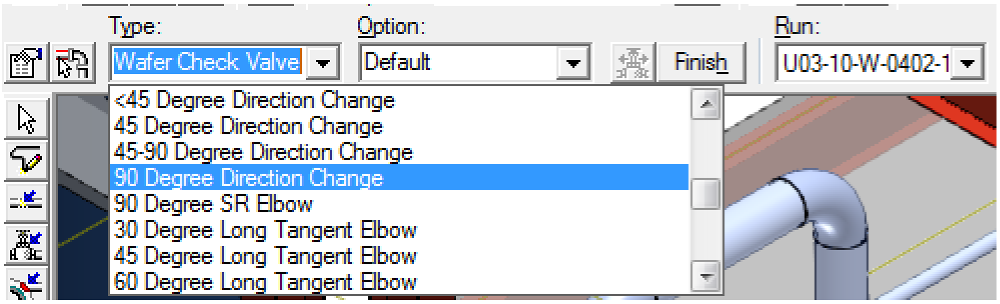
Inserting a 90 Degree Elbow at a Pipe End

Now, place a 90 degree elbow at the end of the pipe. Perform the following steps to place the elbow:

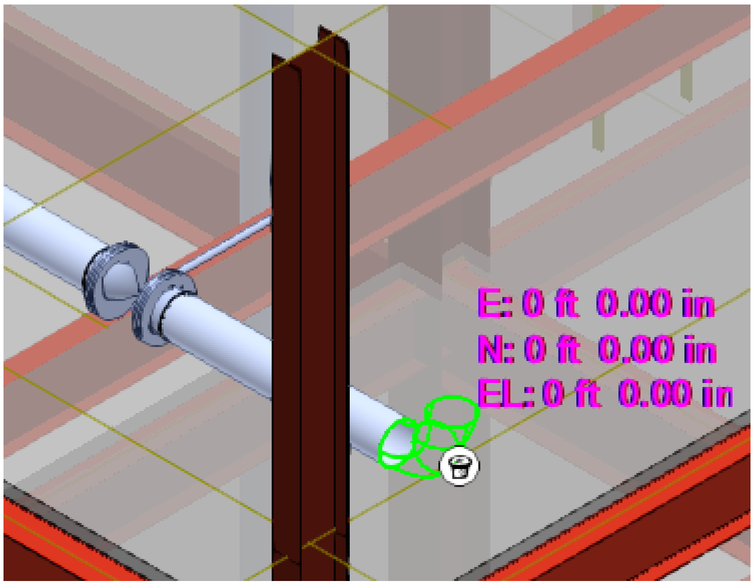
1. Click **Insert Component** and then select **Pipe End Feature** located at the free end of pipeline 300-W.



2. Select the 90 Degree Direction Change option in the Type drop-down list on the ribbon.

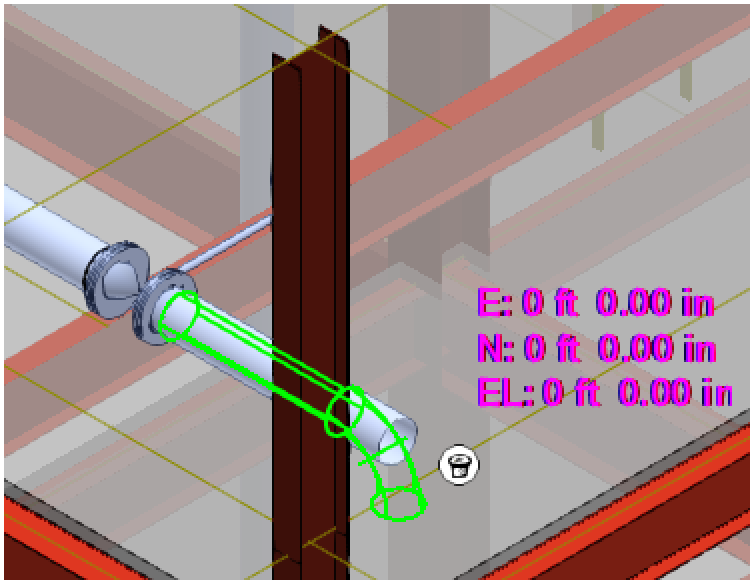


3. You will now see the outline of an elbow at the active placement point, as shown.

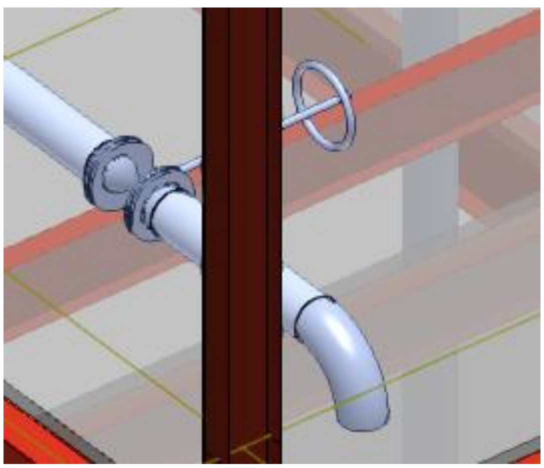


4. Key in 180 deg in the Angle drop-down list to rotate the elbow. Select Origin in the Reference position drop-down list on the ribbon to place the elbow by its origin.

The Reference position option allows you to position the component by a particular port, by its origin, or by the port of a solver-generated mating part at the active placement point.



5. Click **Finish** on the Insert Component ribbon to place the elbow.



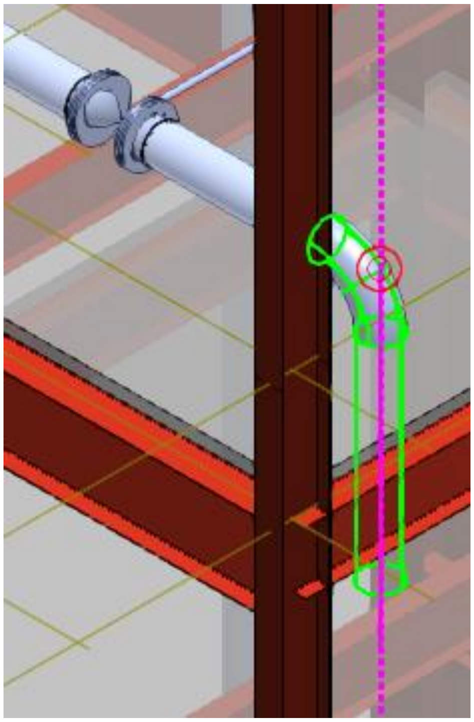
Inserting a Tee at a Pipe End

Now, place a pipe at the end of the elbow and then place a tee at the end of this pipe. Perform the following steps to place a piece of pipe and then a full equal size tee:

1. Click the Route Pipe button on the vertical toolbar. Select the free end of the elbow to start routing.

   If you do not terminate the Insert Component command, Smart 3D should start routing from the open port of the elbow.

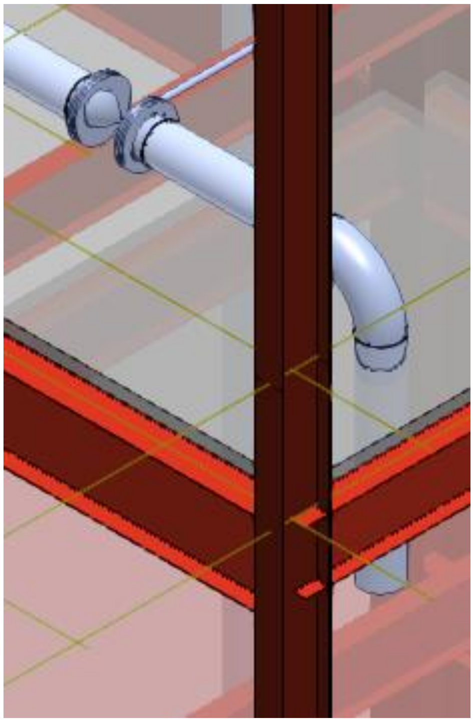
   TIP While in the **Insert Component** command, activate the **Route Pipe** command to place a **Pipe Straight Feature** at the current active route location, as shown.

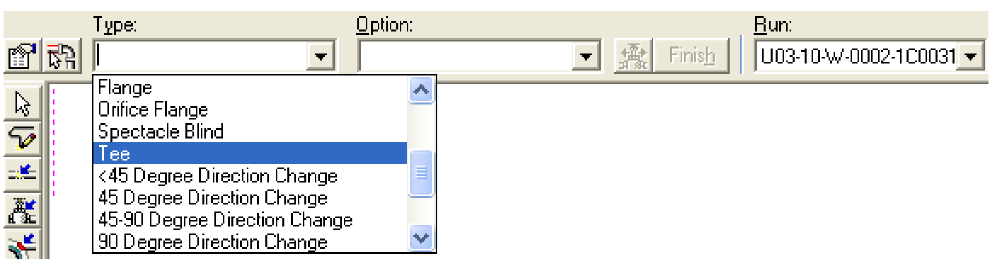
2. On the **Route Pipe** ribbon, key in **5 ft** in the **Length** field. Click the **Plane** drop-down list on the ribbon and select **No Plane** to route the pipe.

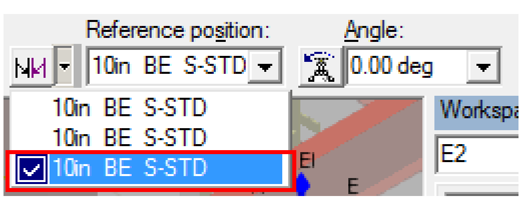
3. Click in the graphic view to place the pipe.



4. Now, click **Insert Component** on the vertical toolbar.
5. Select the **Pipe End Feature** of the pipe that you routed.
6. Select the **Tee** option in the **Type** drop-down list on the ribbon.



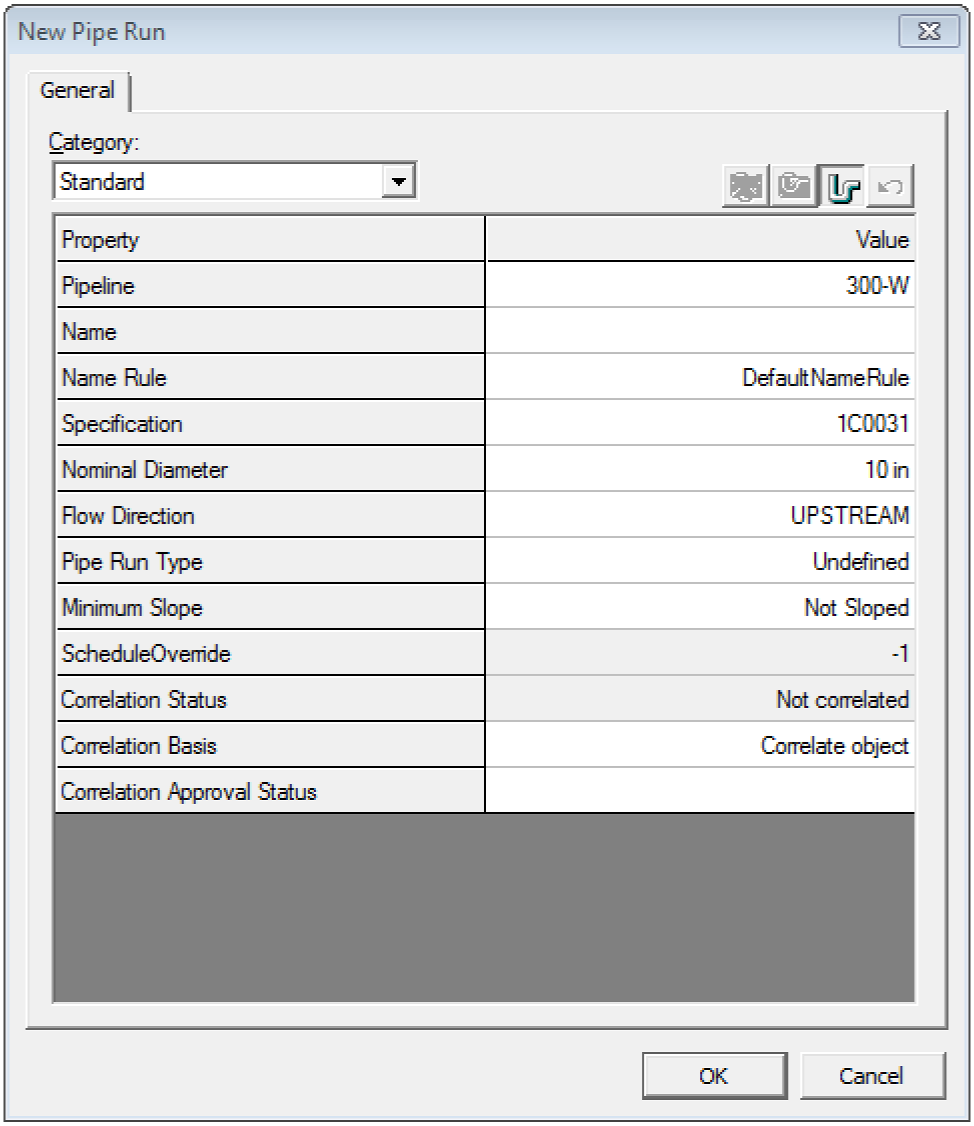
7. On the ribbon, just before **Reference** position, there is a small drop-down arrow called the **Flip** drop-down list, where you can select the port to be used for the placement of a component. Click this arrow and select the third port, **10in BE S-STD**, as shown.



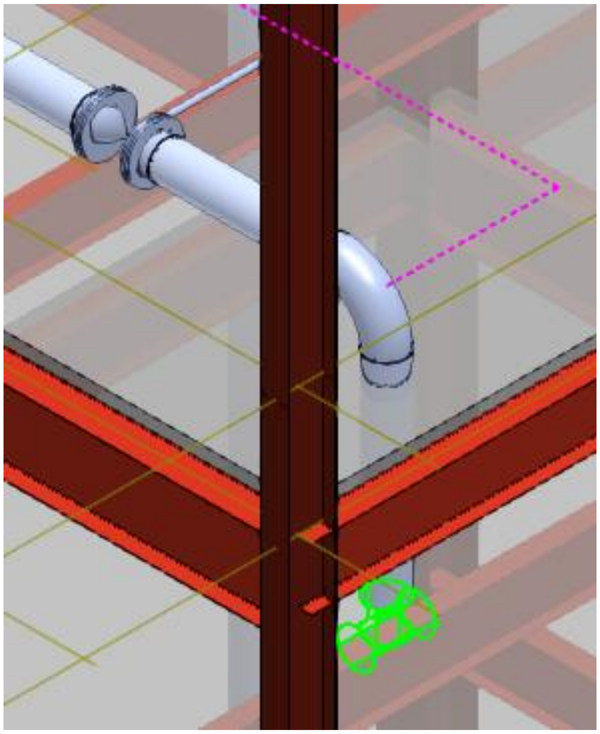
8. Select the **<New Pipe Run>** option in the **Run** drop-down list on the **Route Pipe** ribbon.

   The New Pipe Run dialog box appears.

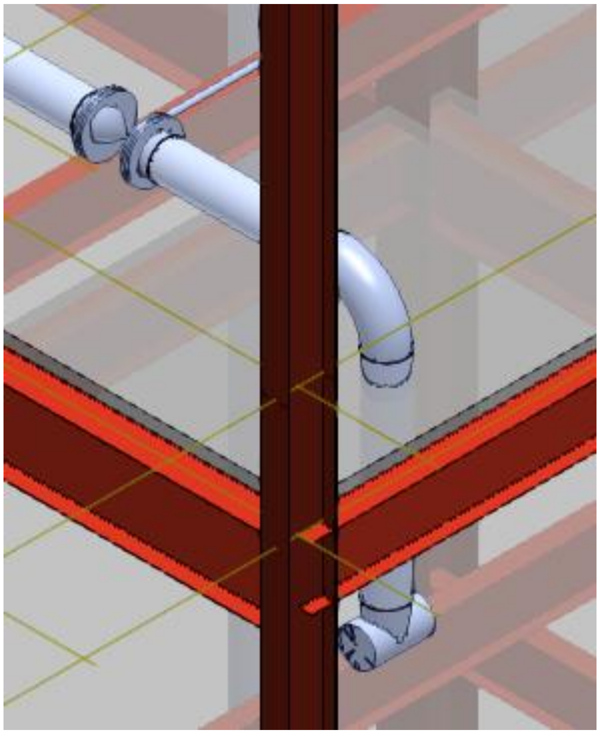
9. Click **OK** to accept the default values of the new pipe run.



You will now see the outline of a tee at the active placement point.

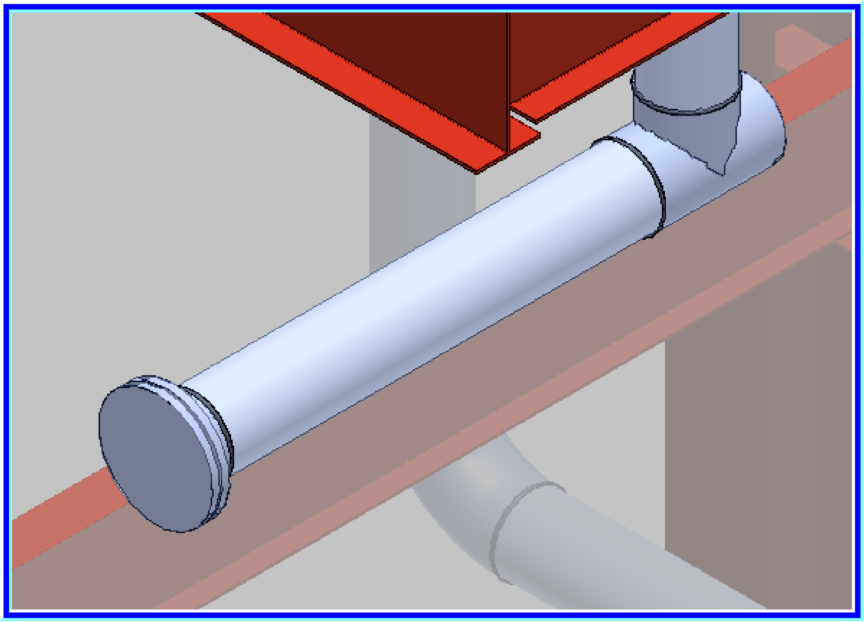


10. Click **Finish** on the ribbon to accept the placement of the tee.

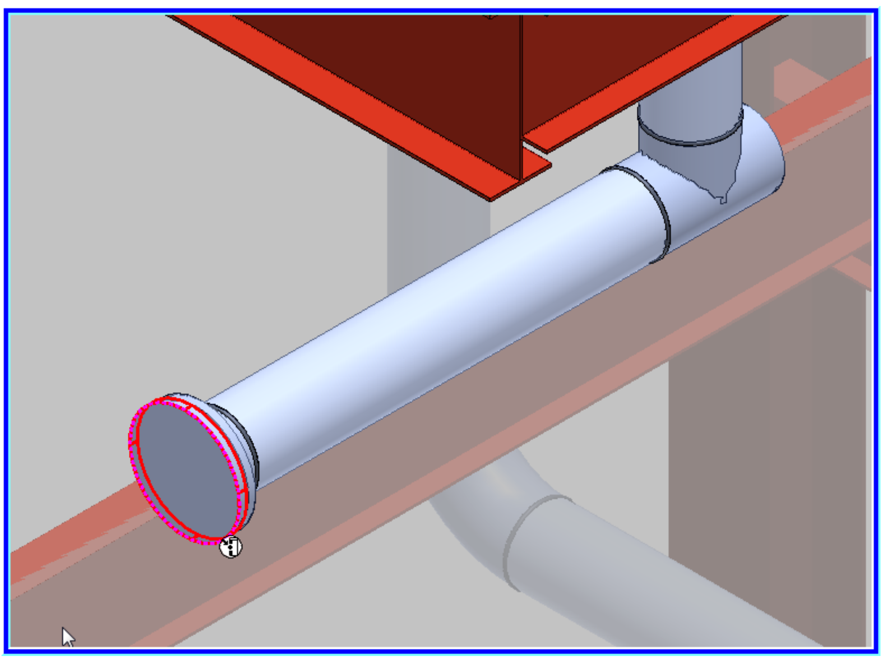


Next, we will see the capability of placing a branch component on another component (other than pipe)

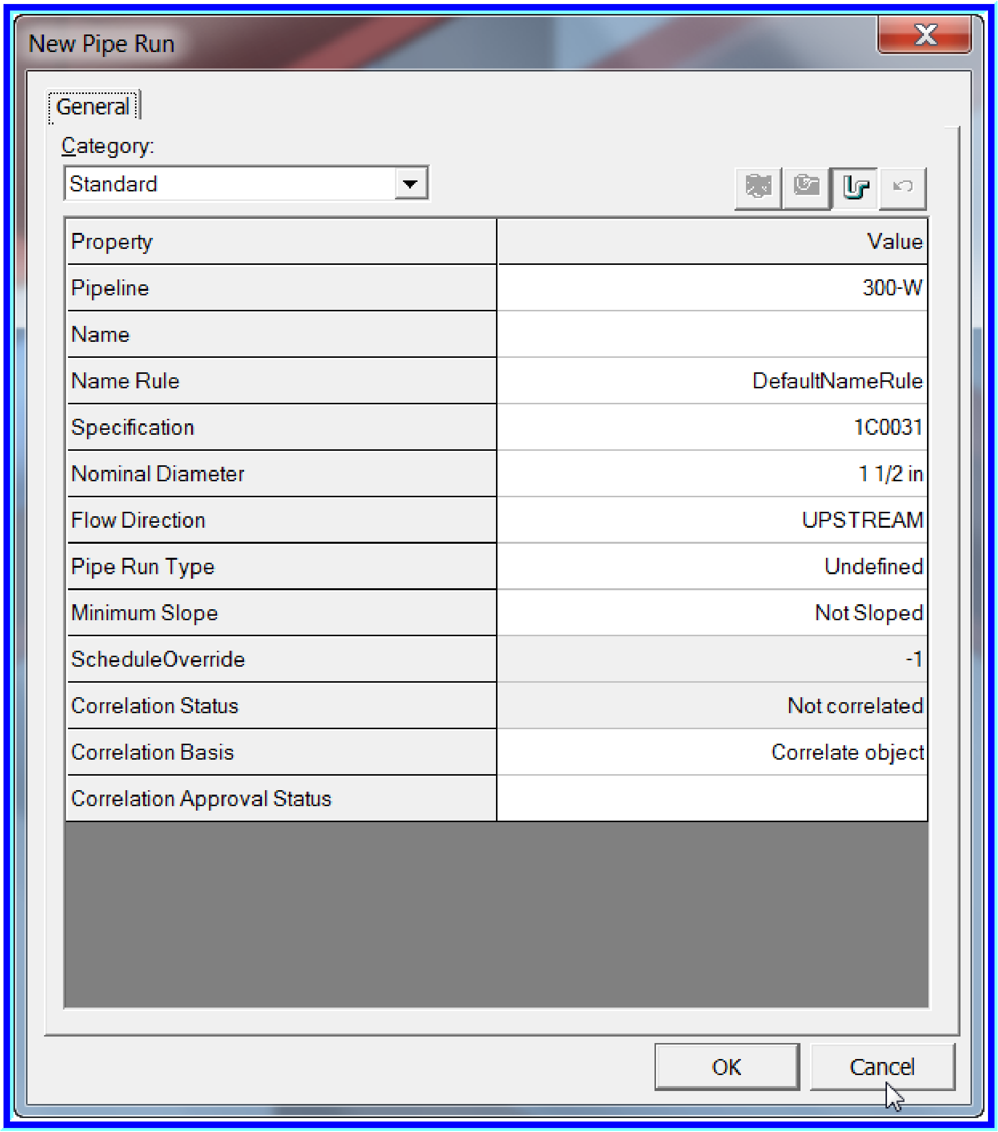
1. Connect to the tee that you just place and then route 5’ of pipe, place a flange, and then place a blind flange.

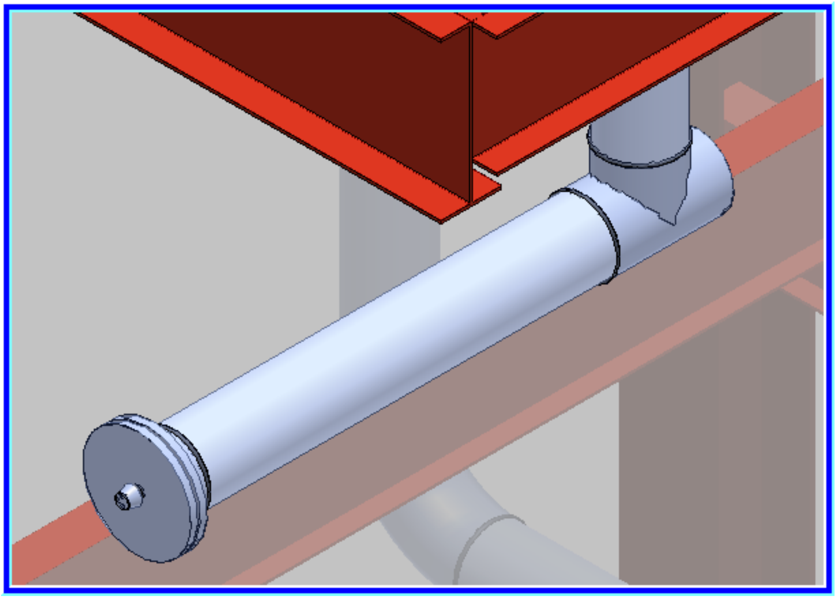


2. Select the **Insert Component** command, and choose the blind flange as the parent.

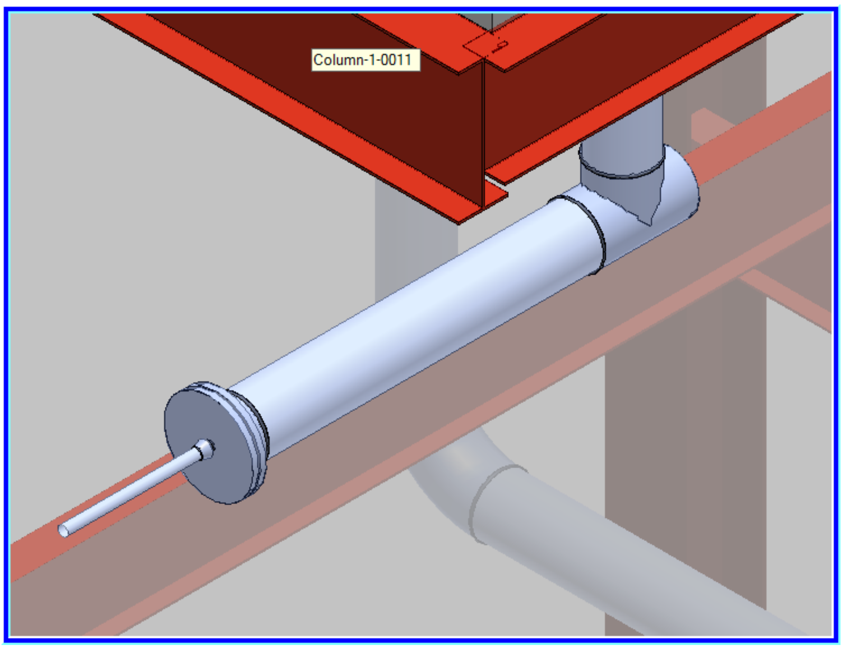


3. Select "Weldolet" from the type and specify the following information:





4. Connect to the weldolet, and then place a 2' section of pipe

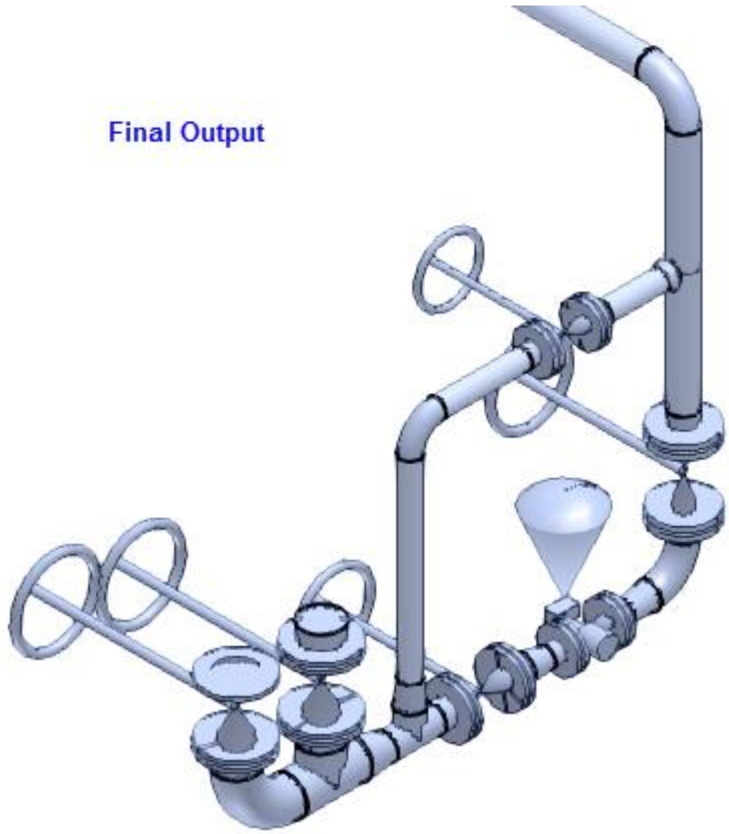


Creating a Control Valve Station

Objective

By the end of this session, you will be able to:

- Create a control valve station by using **Insert Component** on the pipeline **400-P** in **Unit U04** of your workspace. After inserting the components, the control valve station will look like this.

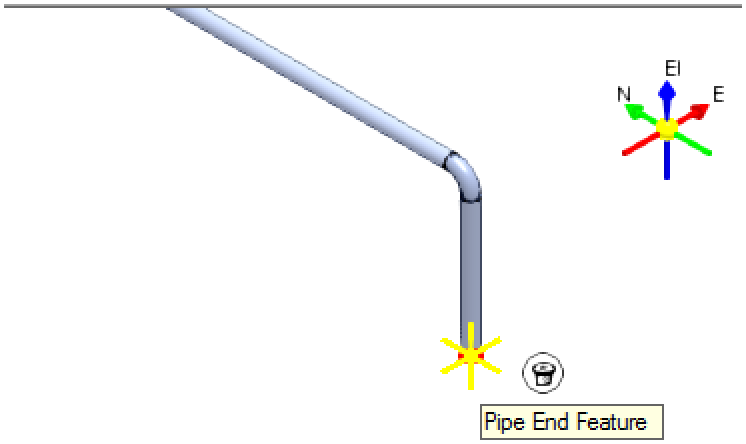


Before Starting this Procedure

- Define your workspace to display **Unit U04** and coordinate system **U04 CS**. In your training plant, select **U04** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Click **Insert Component**  on the vertical toolbar.

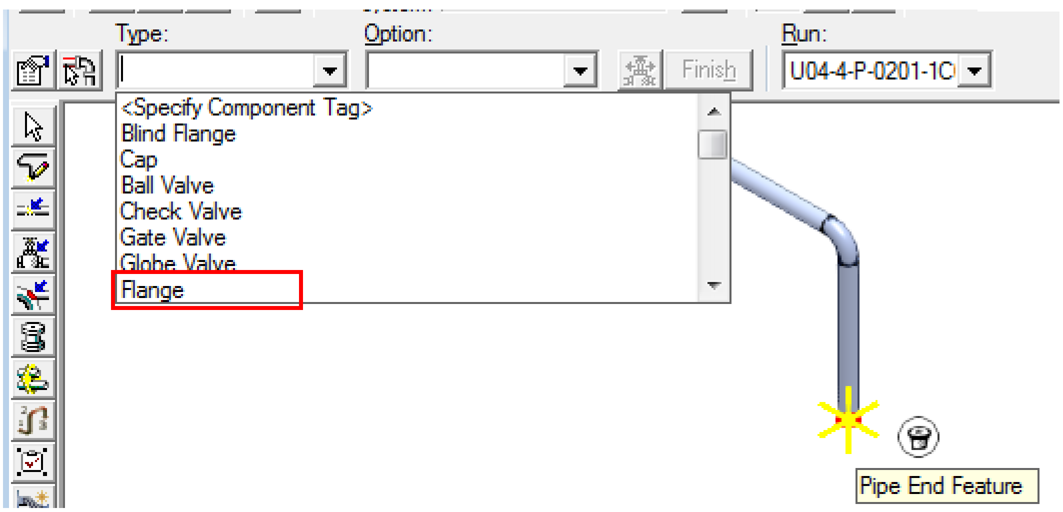
2. Select the **Pipe End Feature** of the pipeline **400-P** by using the SmartSketch options of Smart 3D.



The Insert Component ribbon appears.

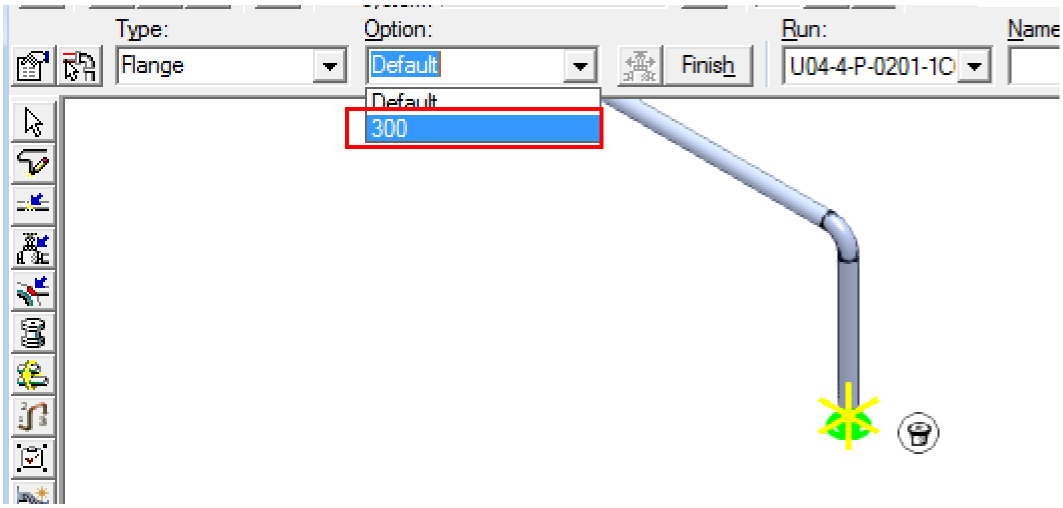
The Type drop-down list on the Insert Component ribbon displays a list of components that you can choose to place in the pipeline 400-P.

3. Select **Flange** option in the **Type** drop-down list, as shown.

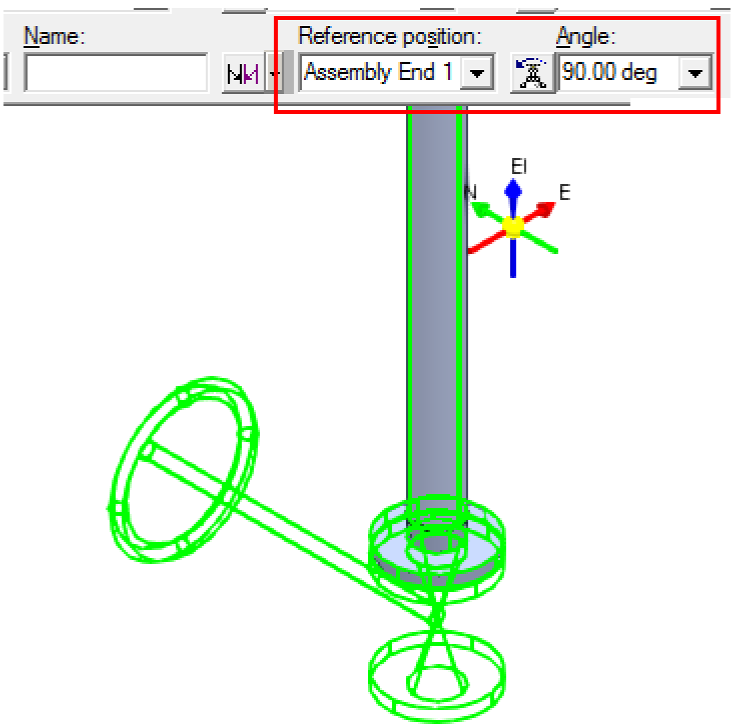


An outline of a flange at the end of the pipeline will appear in the graphic view.

4. Select **300** option in the commodity **Option** drop-down list and click **Finish** on the **Insert Component** ribbon to place the selected flange.



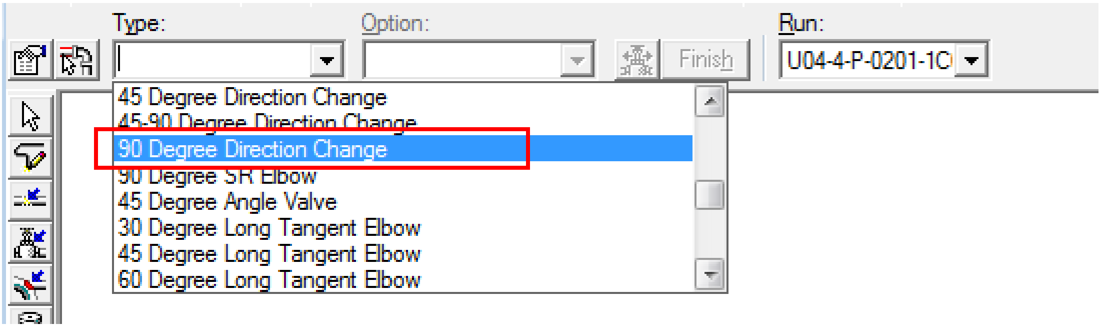
5. Select the **Gate Valve** option in the **Type** drop-down list on the **Insert Component** ribbon.
6. On the **Insert Component** ribbon, select the **Assembly End 1** option in the **Reference** position drop-down list to redefine the placement point. Then, key in **90 deg** in the **Angle** drop-down list to rotate the valve so that the operator is facing the north direction, as shown.



An outline of the gate valve will appear in the graphic view.

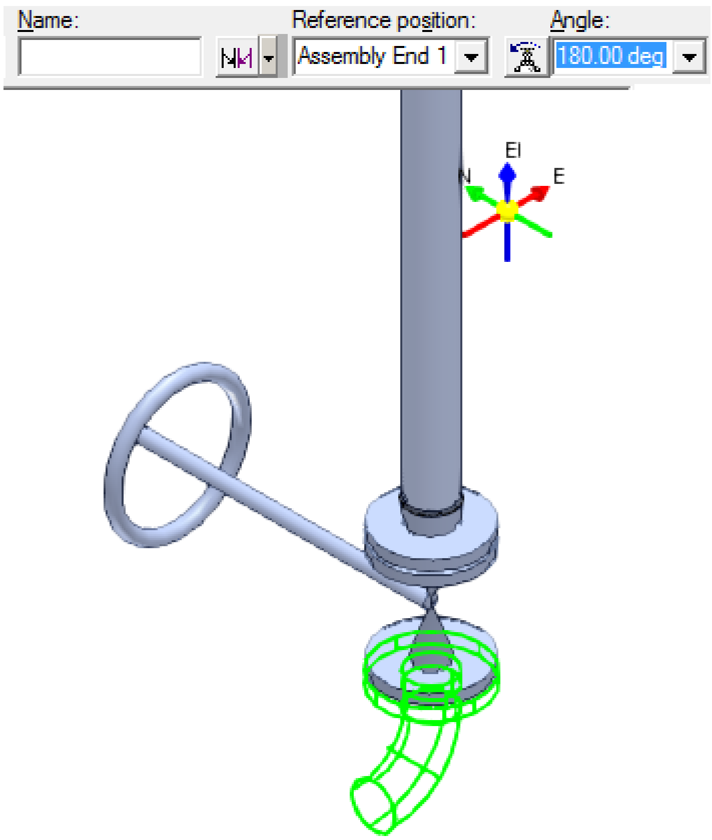
7. Click **Finish** to place the gate valve.

8. Select the **90 Degree Direction Change** option in the **Type** drop-down list on the **Insert Component** ribbon.



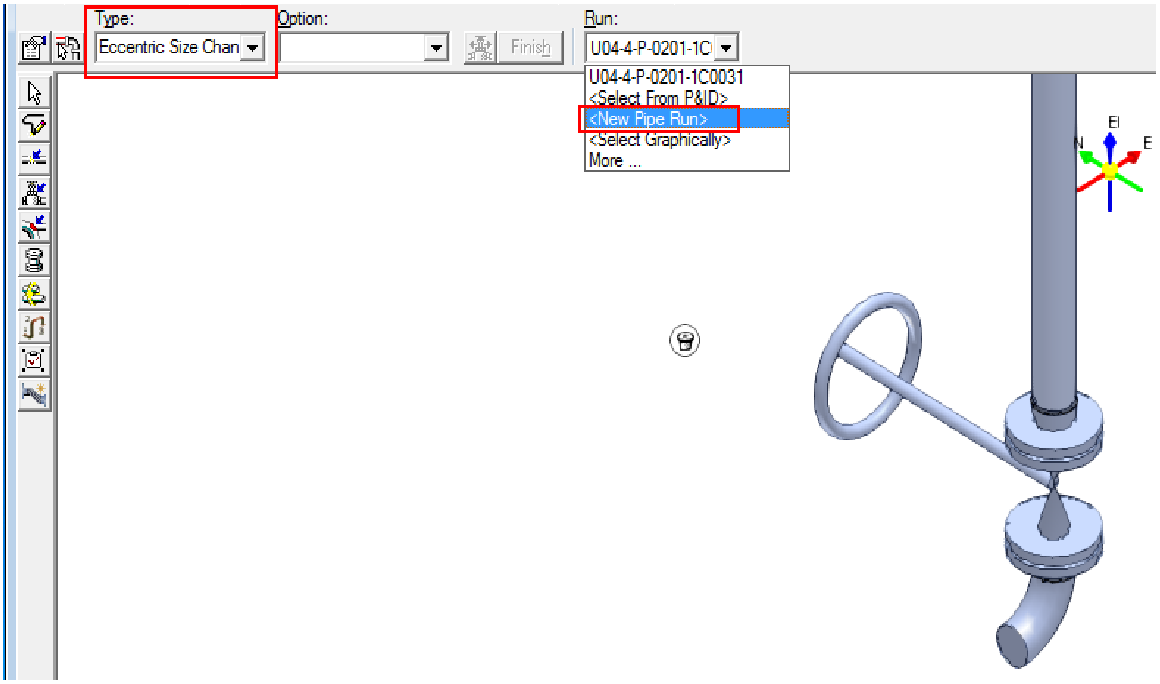
An outline of the elbow and mating flange will appear in the graphic view.

9. Key in **180 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the elbow so that the open port faces the west direction, as shown.



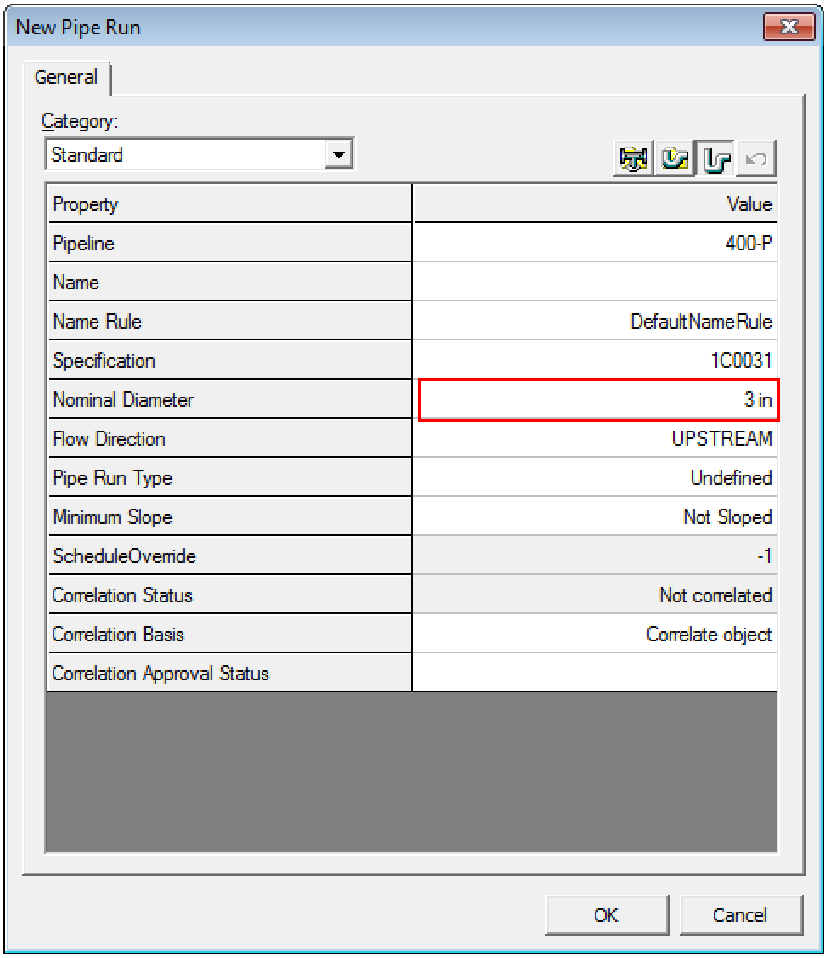
10. Click **Finish** to place the 90-degree elbow.
11. Select the **Eccentric Size Change** option in the **Type** drop-down list on the Insert Component ribbon.

12. Select the **<New Pipe Run>** option in the **Run** drop-down list on the **Insert Component** ribbon.



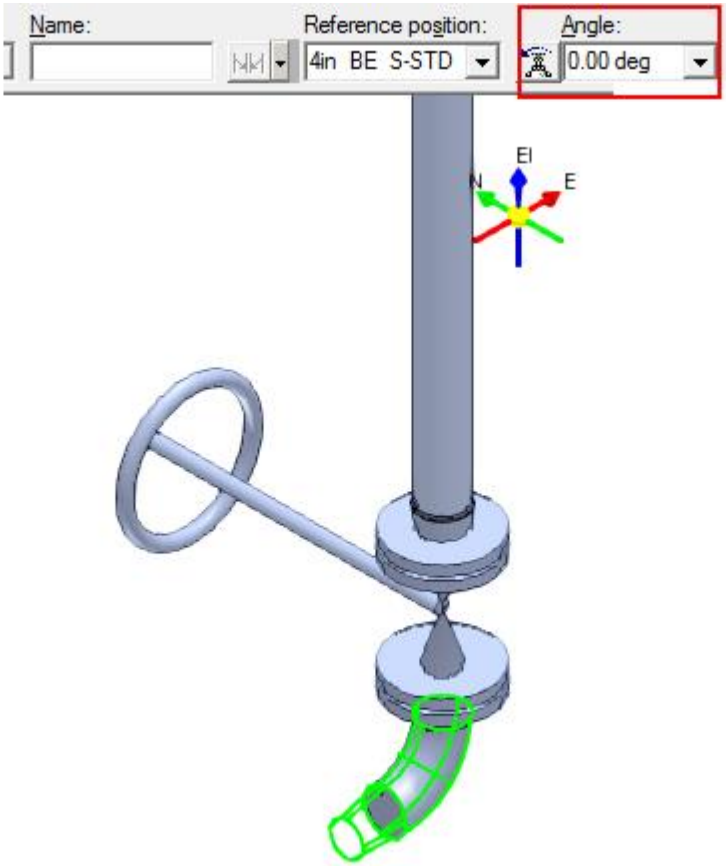
The New Pipe Run dialog box appears.

13. Select **3** in for the **Nominal Diameter** option and click **OK** on the New Pipe Run dialog box to accept the other default values of the new pipe run.



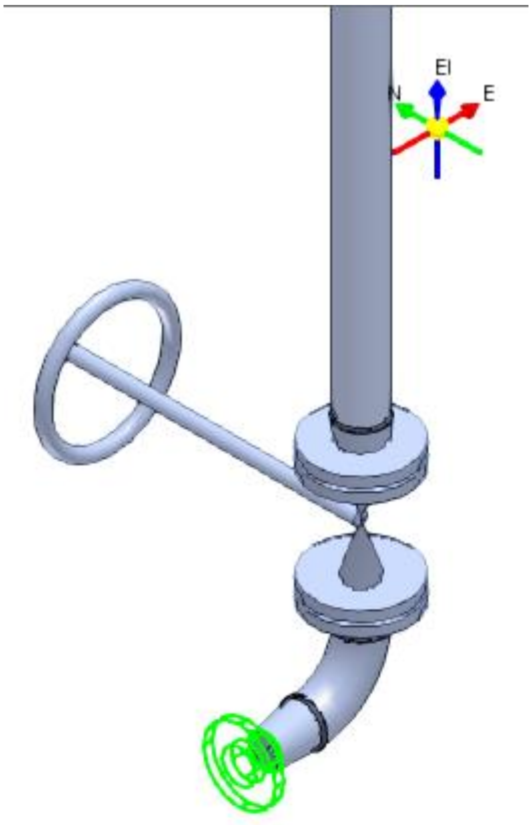
An outline of the eccentric reducer will appear in the graphic view.

14. Key in **0 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the eccentric reducer so that the flat side is facing the bottom.

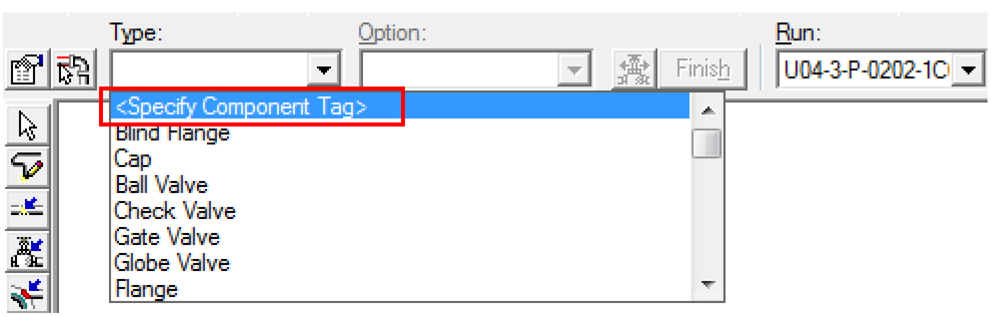


15. Click **Finish** to place the eccentric reducer.
16. Select the **Flange** option in the **Type** drop-down list on the **Insert Component** ribbon.

    An outline of the flange will appear in the graphic view, as shown.

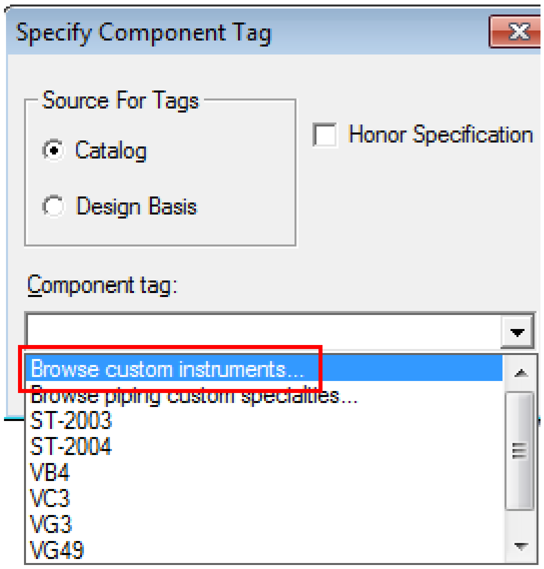


17. Click **Finish** to place the flange.
18. Select the **<Specific Component Tag>** option in the **Type** drop-down list, as shown.



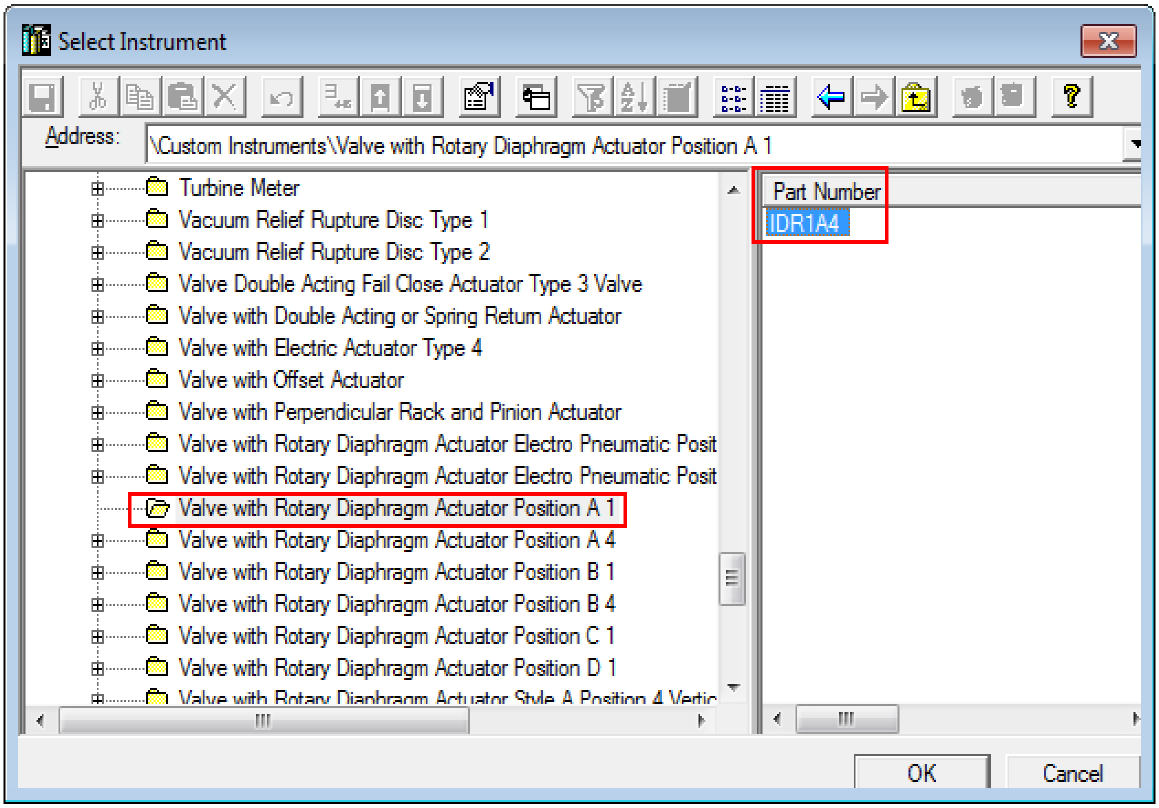
The Specific Component Tag dialog box appears.

19. Select the **Browse Custom Instruments…** option to select the instrument from the catalog.



The Select Instrument dialog box appears. This dialog box displays a list of the available custom instruments that you can insert in the pipe run. These custom instruments are already defined in the catalog.

20. Expand **Custom Instruments > Valve with Rotary Diaphragm Actuator Position A1** and select the part number **IDR1A4** in the right pane. Then, click **OK** to close the **Select Instrument** dialog box.



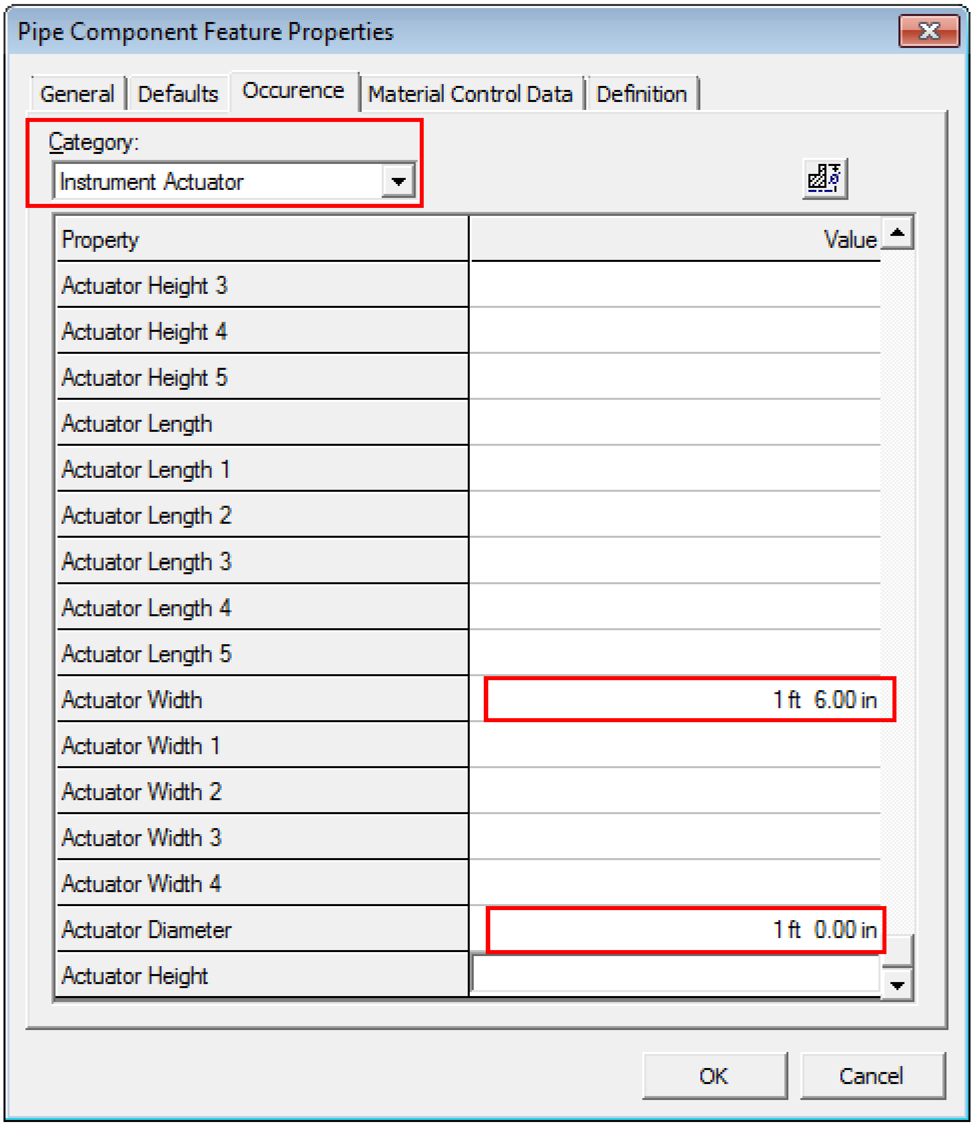
An outline of the instrument will appear in the graphic view.

21. Click the **Properties** option on the **Insert Component** ribbon to change the properties of the valve actuator.

    The Pipe Component Feature Properties dialog box appears.

22. Select the **Instrument Actuator** category under the **Occurrence** tab on the **Pipe Component Feature Properties** dialog box to change the dimensions of the valve actuator as follows:

Actuator Width: 1 ft 6 in
Actuator Diameter: 1 ft



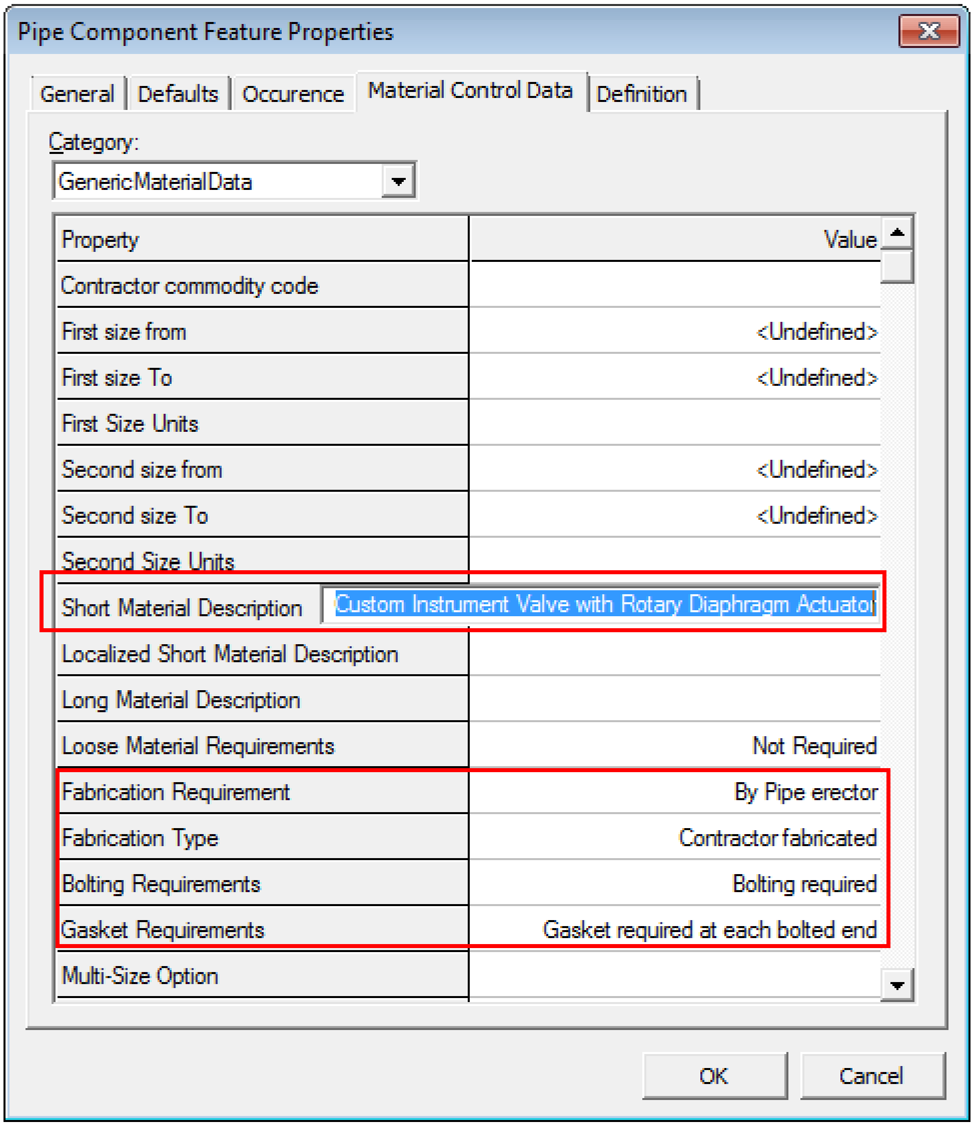
23. To define the fabrication and construction requirements for the valve, select the **Fabrication and Construction** category on the **Pipe Component Feature Properties** dialog box and set the following specifications:

    Fabrication Requirement: By Pipe erector
    Fabrication Type: Contractor fabricated
    Construction Requirement: New
    Construction Type: New

24. Select the **Standard** option in the **Category** drop-down list in the **Pipe Component Feature Properties** dialog box and then, key in **CVD-201** as the tag number in the **Name** field for this instrument.

25. Select the **Material Control Data** tab and then select the **GenericMaterialData** option under the **Category** drop-down list. Change the material properties for the instrument valve and set the following specifications:

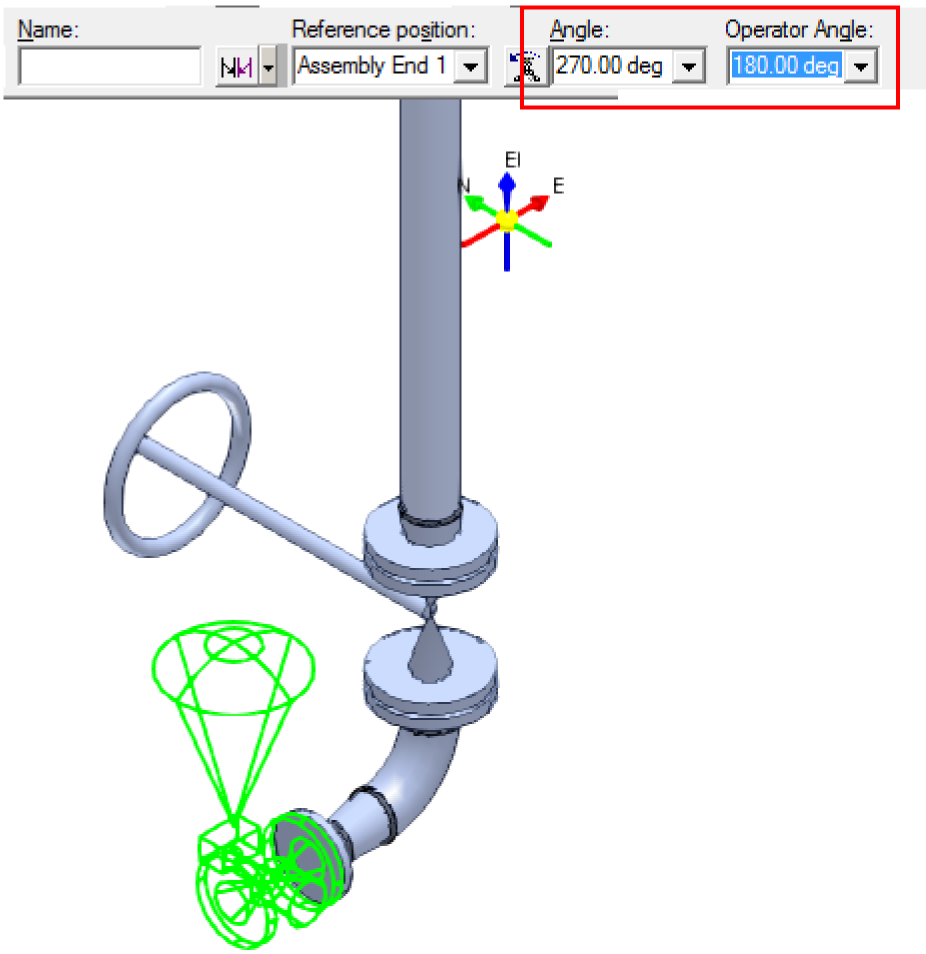
    Short Material Description: Custom Instrument Valve with Rotary Diaphragm Actuator
    Fabrication Requirement: By Pipe erector
    Fabrication Type: Contractor fabricated
    Bolting Requirements: bolting required
    Gasket Requirements: Gasket required at each bolted end



26. Click **OK** to close the **Pipe Component Feature Properties** dialog box.

    An outline of the instrument will appear in the graphic view.

27. Key in **270 deg** in the **Angle** drop-down list and **180 deg** in the **Operator Angle** drop-down list to rotate the instrument and the actuator, respectively, on the Insert Component ribbon.



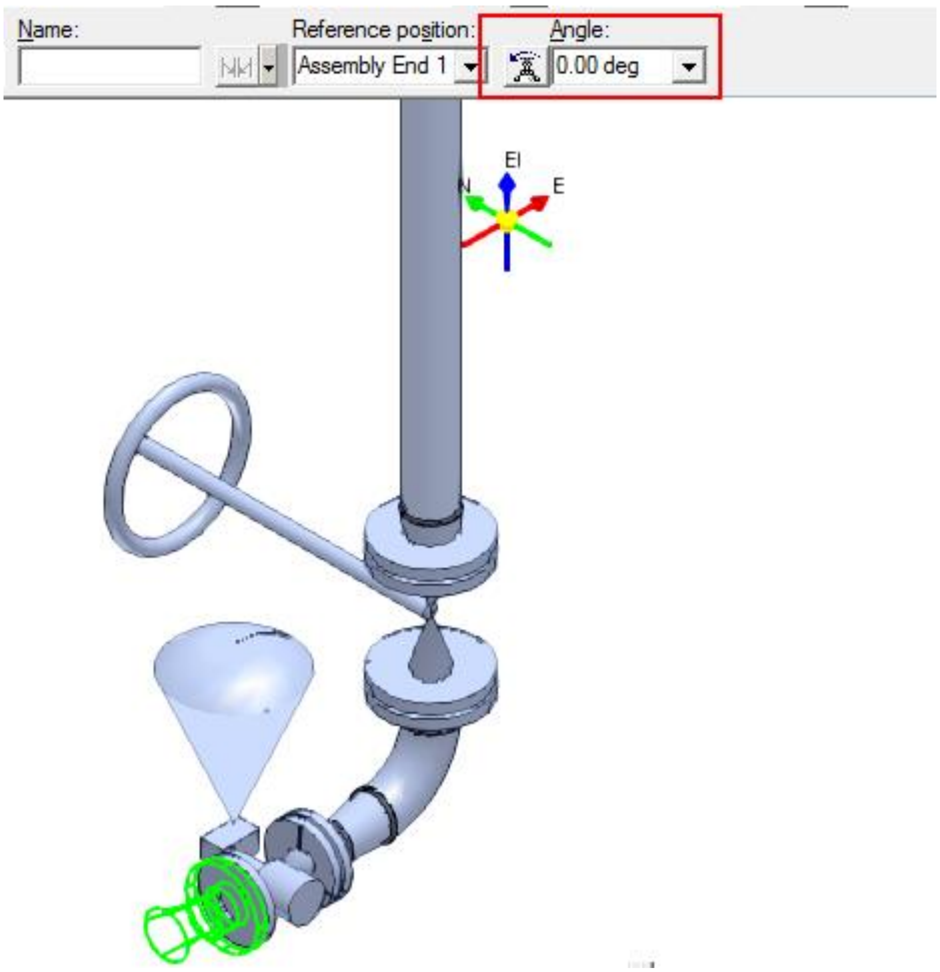
28. Click **Finish** to place the instrument.

29. Select the **Eccentric Size Change** option in the **Type** drop-down list and the **<New Pipe Run>** option in the **Run** drop-down list on the **Insert Component** ribbon.

    The New Pipe Run dialog appears.

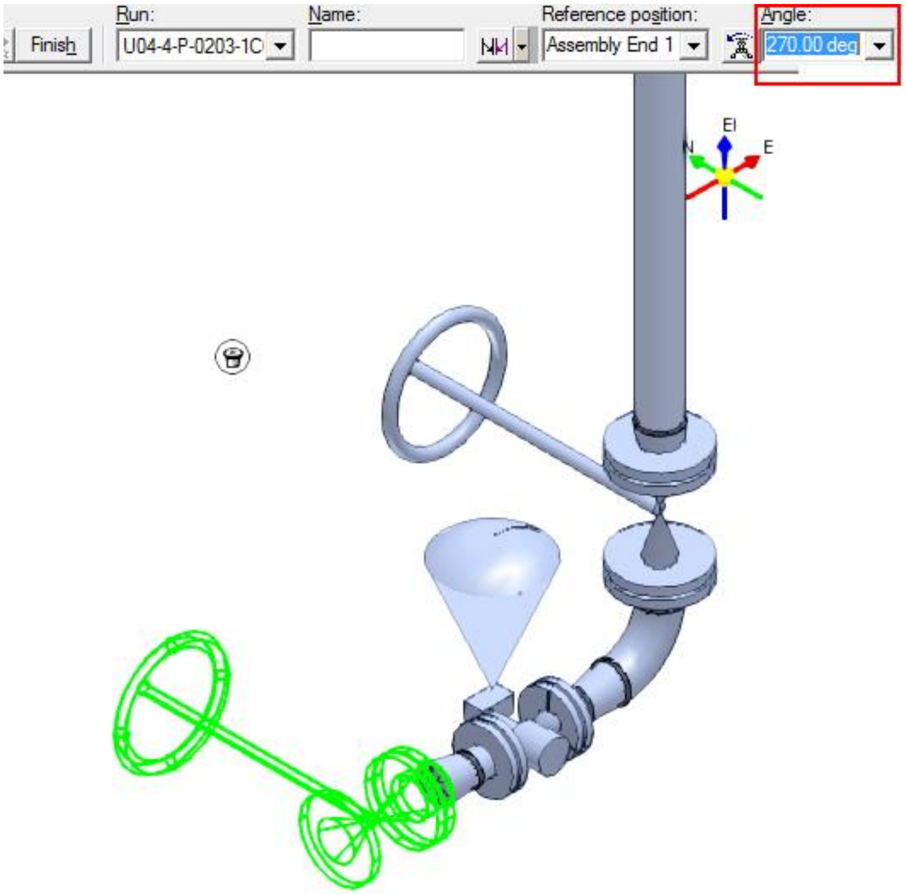
30. Make sure **Nominal Diameter** is set to **4 in** and click **OK** to accept the other default values of the new pipe run.

An outline of the eccentric reducer and mating flange will appear in the graphic view, as shown.



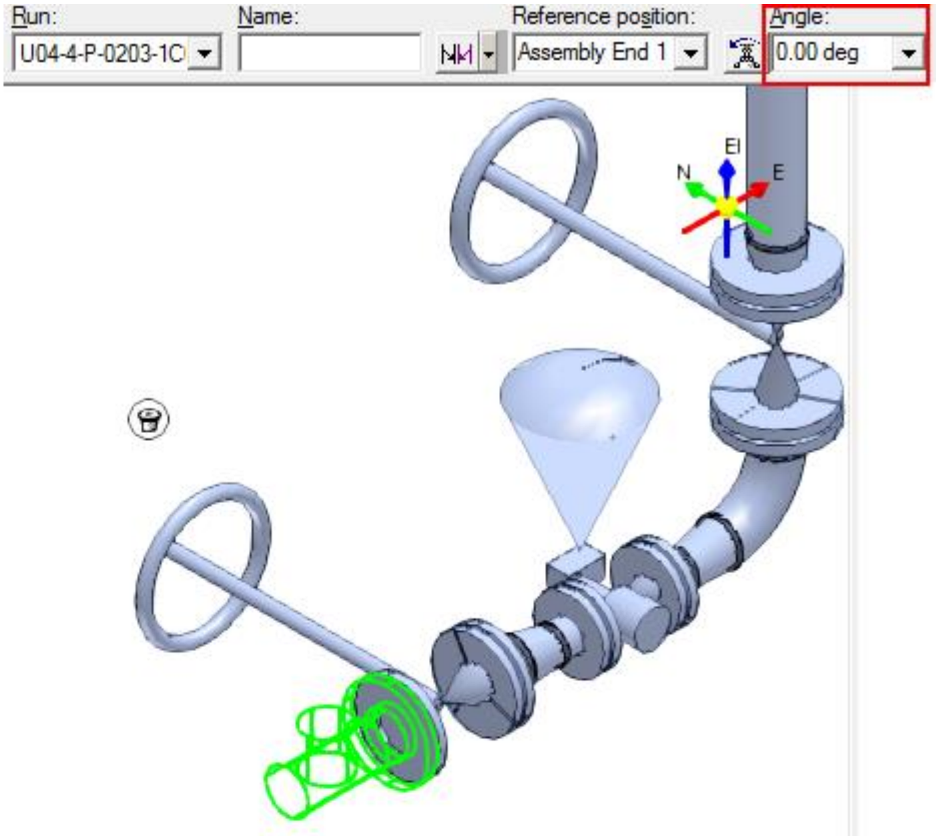
31. Key in **0 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the eccentric reducer so that the flat side is towards the bottom.
32. Click **Finish** to place the eccentric reducer.

33. Select the **Gate Valve** option in the **Type** drop-down list and key in **270 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the gate valve so that the operator is facing the north direction, as shown.



34. Click **Finish** to place the gate valve.

35. Select the **Tee** option in the **Type** drop-down list and key in **0 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the tee so that port 3 is pointing upwards, as shown.

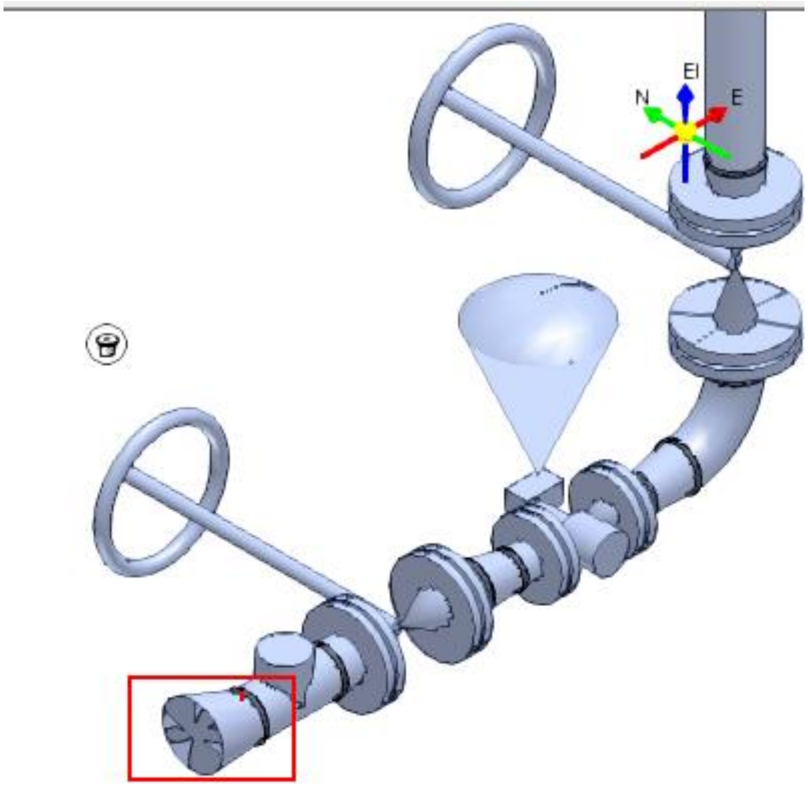


36. Click **Finish** to place the tee.
37. Select the **Eccentric Size Change** option in the **Type** drop-down list on the **Insert Component** ribbon/
38. Select the **<New Pipe Run>** option in the **Run** drop-down list on the **Insert Component** ribbon.

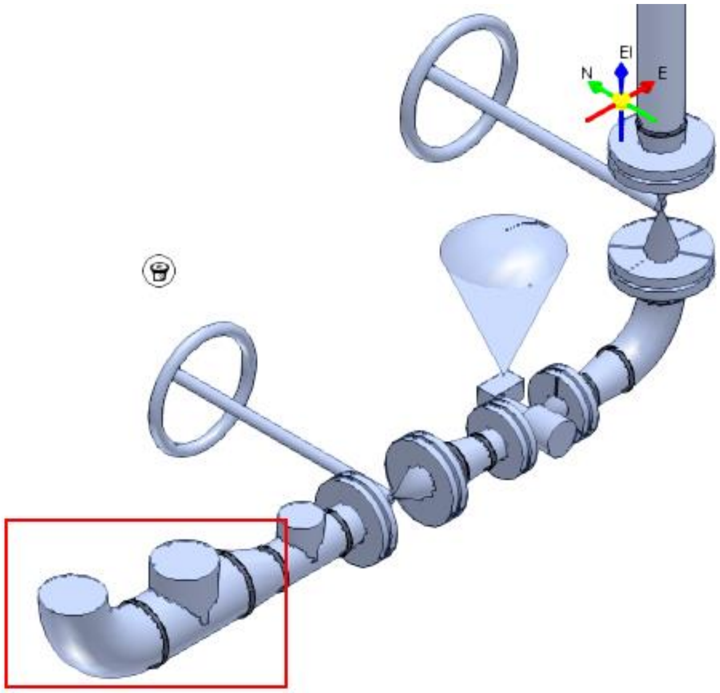
    The New Pipe Run dialog box appears.
39. Make sure **Nominal Diameter** is set to **6 in** and click **OK** on the **New Pipe Run** dialog box to accept the other default values of the new pipe run.

    An outline of the eccentric reducer will appear in the graphic view.

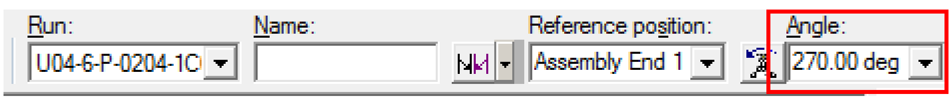
40. Click **Finish** to place the eccentric reducer.

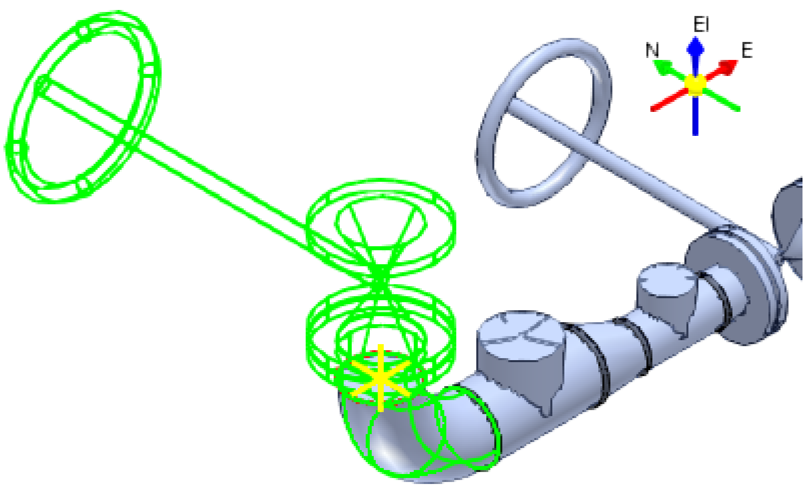


41. Now insert a **Tee** and a **90 Degree Direction Change**, as shown, by selecting them in the **Type** drop-down list.



42. Now insert a gate valve and rotate it by 270 deg so that the gate valve operator is facing the north direction, as shown.

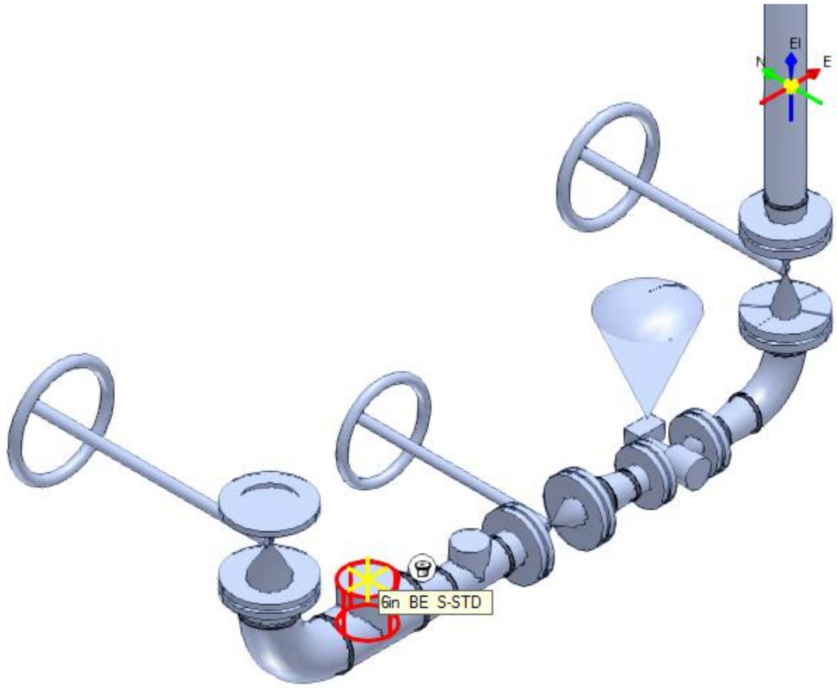




43. Click **Finish**. Right-click to terminate the **Insert Component** command.

44. Click **Insert Component**  again on the vertical toolbar.

45. Select **port 3** of the **6 inch** tee, as shown, by using the SmartSketch options of Smart 3D.

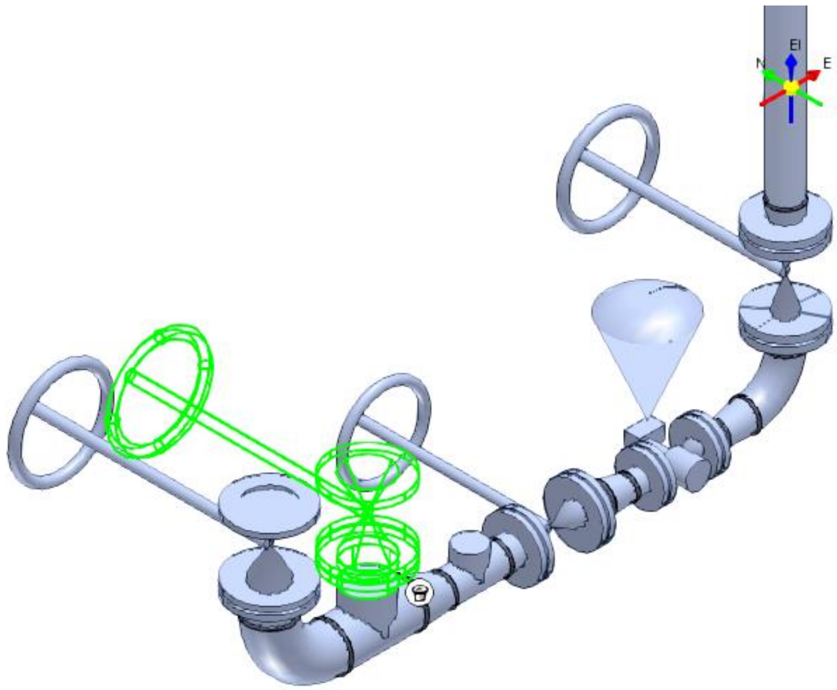


The New Pipe Run dialog box appears.

46. Make sure **Nominal Diameter** is set to **6 in** and click **OK** on the **New Pipe Run** dialog box to accept the other default values of the new pipe run.
47. Select the **Gate Valve** option in the **Type** drop-down list on the **Insert Component** ribbon.

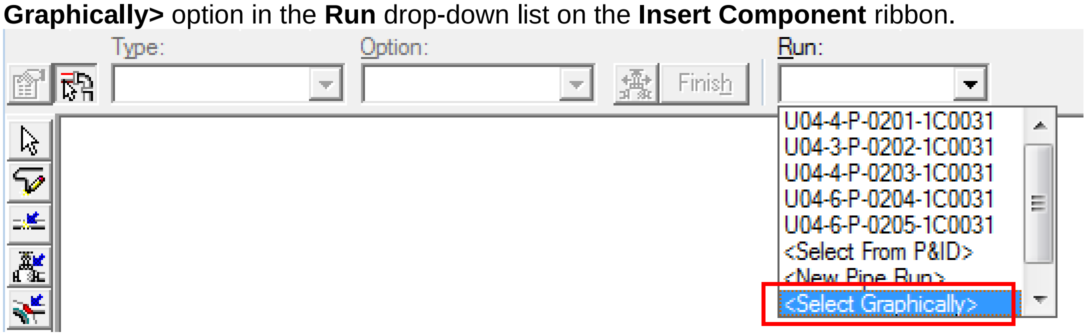
    An outline of the gate valve and the mating flange will appear in the graphic view.

48. Key in **270 deg** in the **Angle** drop-down list to rotate the gate valve so that the valve operator is facing north, as shown.

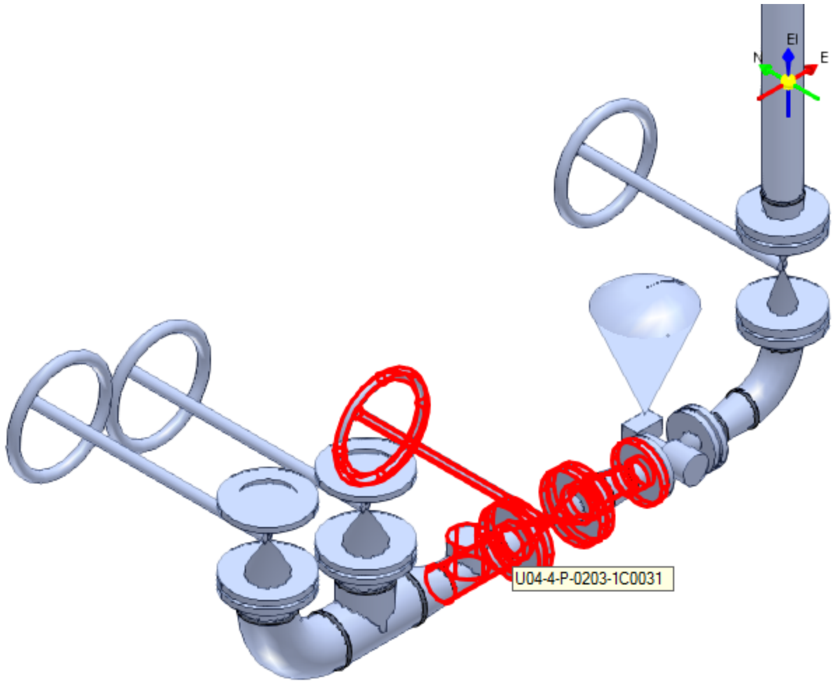


49. Click **Finish** on the **Insert Component** ribbon to place the gate valve and right-click to terminate the **Insert Component** command.

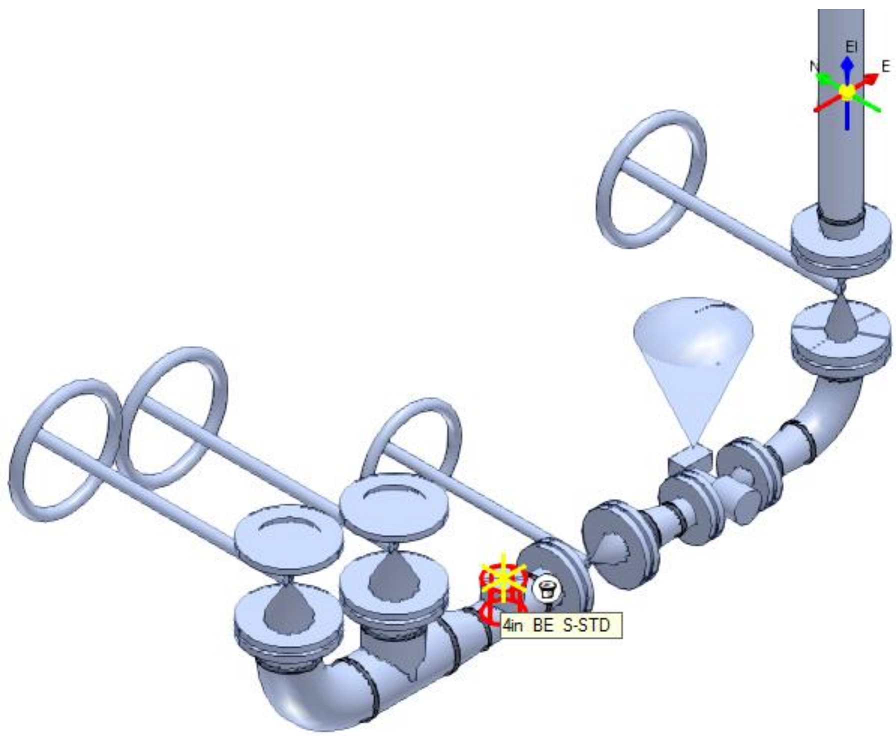
50. Click the **Insert Component** button again on the vertical toolbar and select the **<Select Graphically>** option in the **Run** drop-down list on the **Insert Component** ribbon.



51. Select the pipe run to which the tee belongs, as shown.



52. Select **port 3** of the **4 inch** tee in the graphic view by using the SmartSketch options of Smart 3D, as shown.

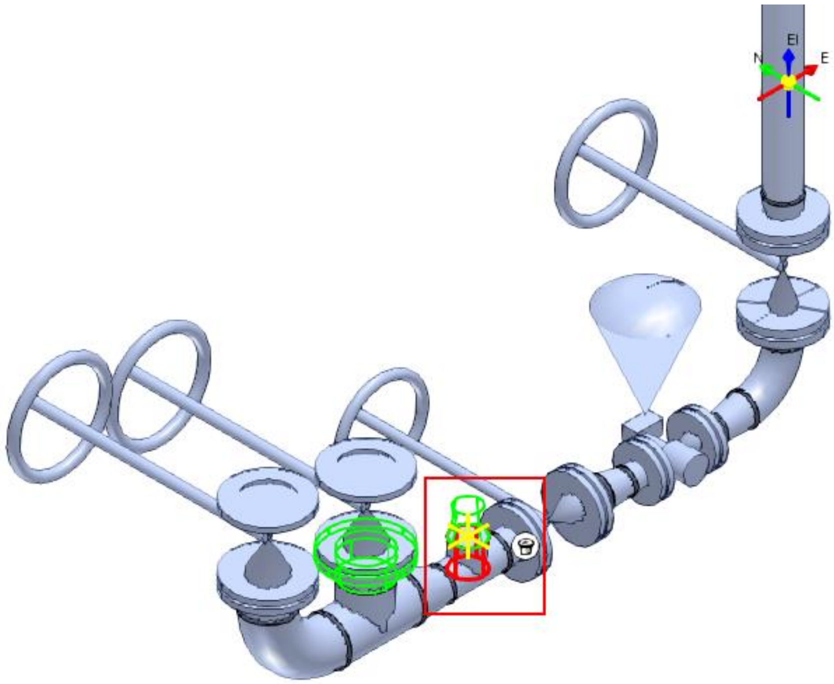


53. Select the **Concentric Size Change** option in the **Type** drop-down list and the **<New Pipe Run>** option in the **Run** drop-down list on the **Insert Component** ribbon.

    The New Pipe Run dialog box appears.

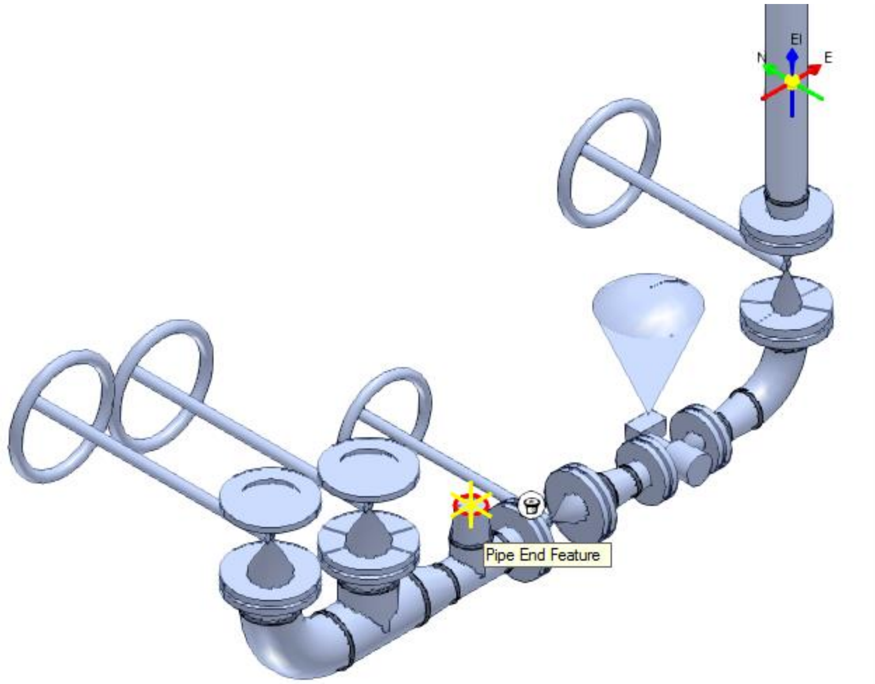
54. Make sure **Nominal Diameter** is set to **3 in** and click **OK** on the **New Pipe Run** dialog box to accept the other default values of the new pipe run.

    An outlined concentric reducer will appear in the graphic view, as shown.

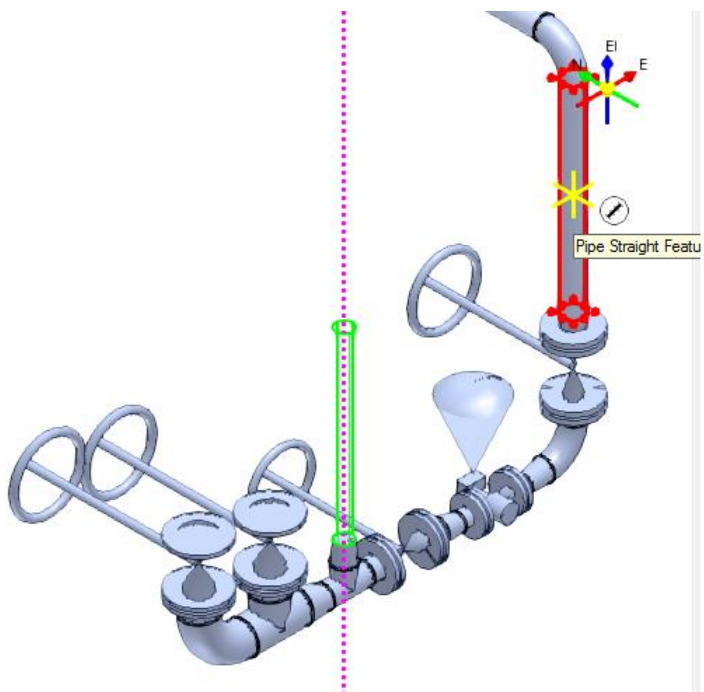


55. Click **Finish** to place the concentric reducer and right-click to terminate the **Insert Component** command.

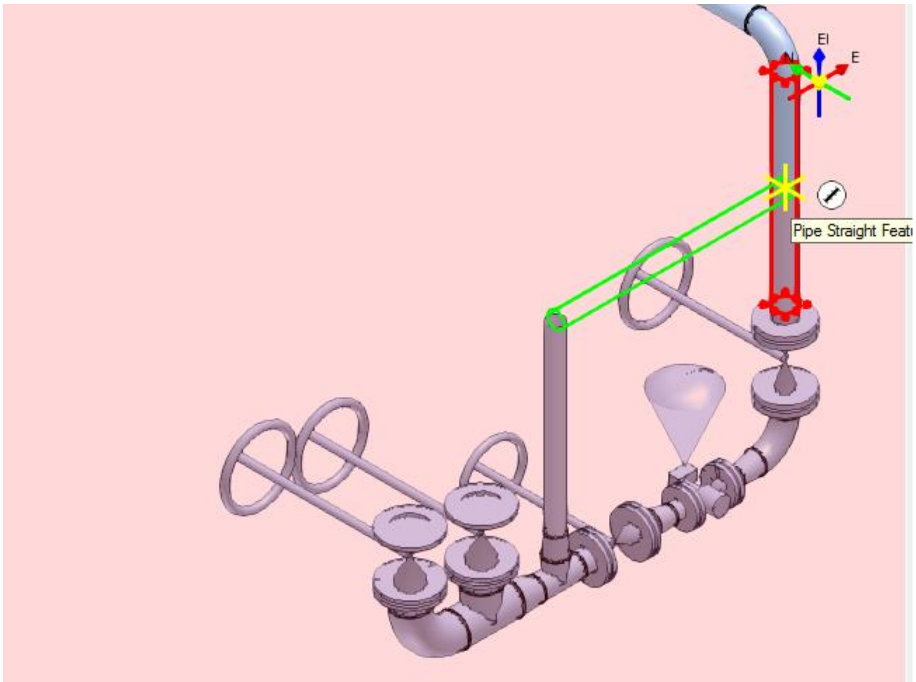
56. Click **Route Pipe** on the vertical toolbar and locate the end of the concentric reducer, as shown.



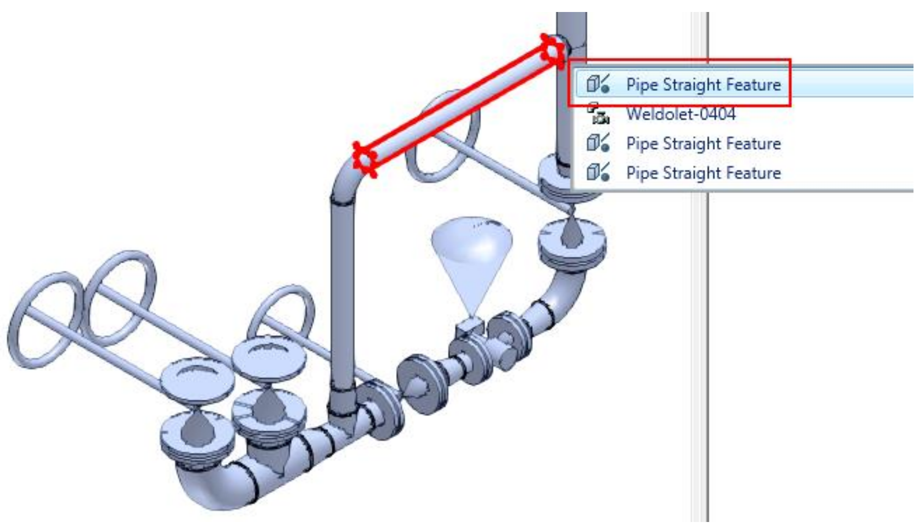
57. Route a **Pipe Straight Feature** to the same elevation as the mid-point of the vertical straight feature, as shown.



58. Make a 90-degree turn and route into the vertical straight feature to create a by pass line, as shown.



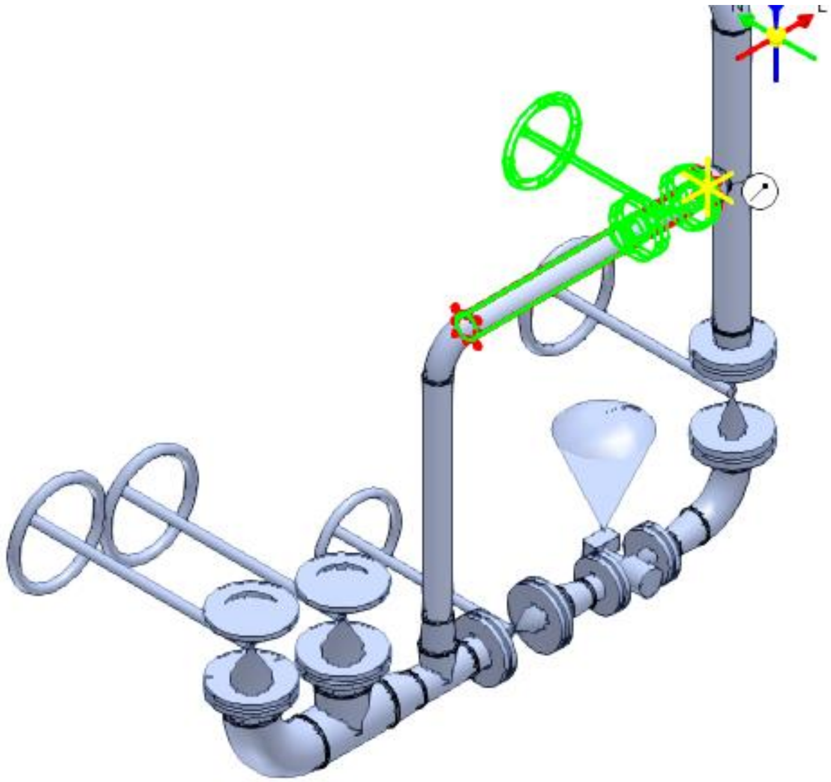
59. Click Insert Component again on the vertical toolbar.
60. Specify the end of the by-pass line by using the SmartSketch options of Smart 3D, as shown.



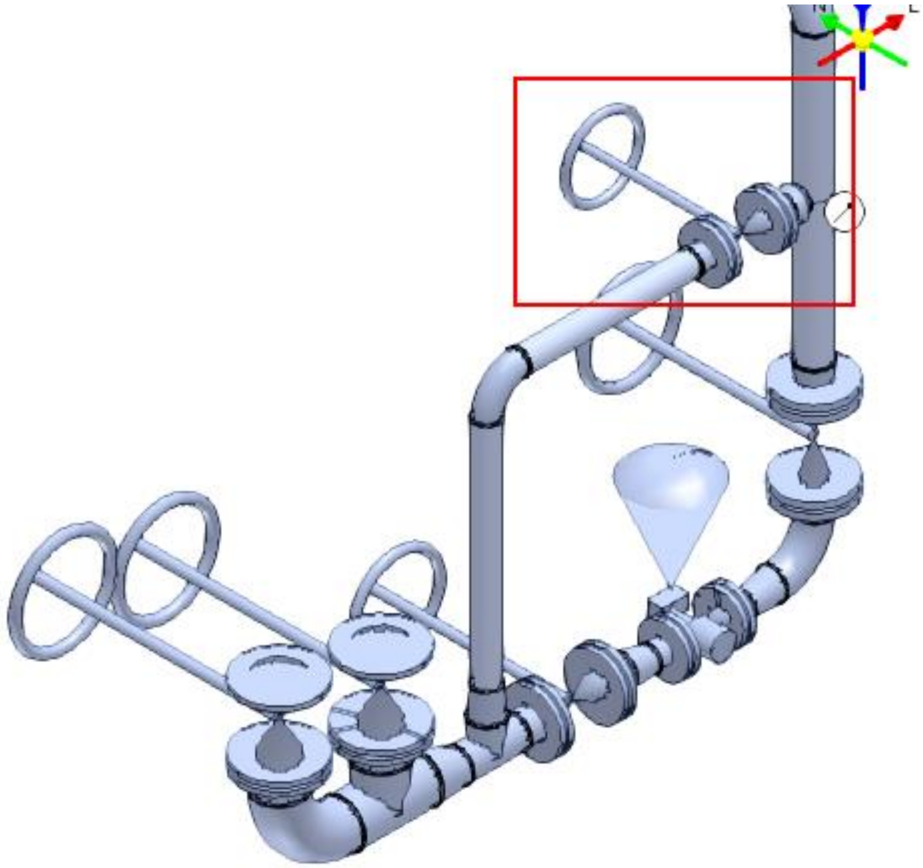
61. Select the **Gate Valve** option in the **Type** drop-down list on the **Insert Component** ribbon.

    An outline of the gate valve and the mating flange will appear in the graphic view.

62. Key in **90 deg** in the **Angle** drop-down list to rotate the gate valve so that the valve operator faces north, as shown.



63. Click **Finish** to place the gate valve, as shown.



For more information related to inserting components in pipe runs, refer to the topic Inserting Components in the user guide **PipingUsersGuide.pdf**.

SESSION 6

Routing Jacketed Piping

Objective

By the end of this session, you will be able to:

- Route core and jacketed piping and place jacket connection components

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

Overview

Smart 3D allows for the capability to route core and the related jacket piping. When routing a new pipe run, there is a **Pipe Run Type** option that allows you to declare that you want to route the core or jacket. Subsequently, a second run is then routed and related to the previous core or jacket run. To facilitate the placement of a jacket, special components are needed (such as a 6x6x8 multi size flange). The primary size is for the core connectivity, where the 8in size is for connecting the jacket pipe run.

Placing Jacketed Piping

Objective

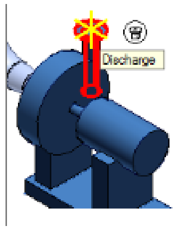
Route a core pipe run as well as the related jacket pipe run.

Before Starting this Procedure

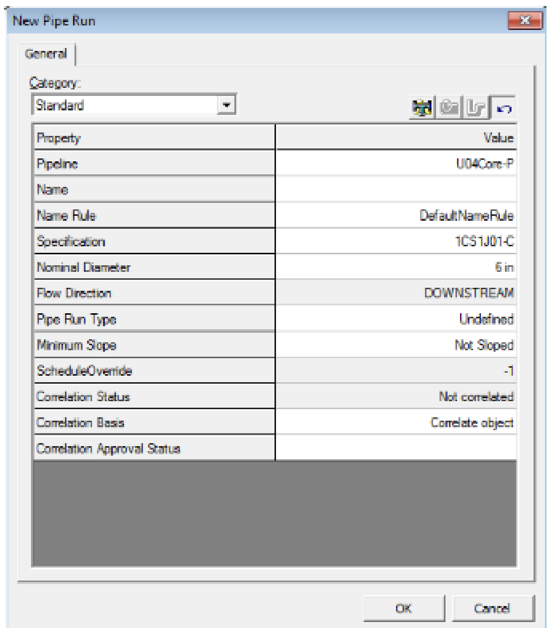
- Define your workspace to display **Unit U03** and coordinate system **U03 CS**. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Click **Zoom Area** and zoom in to the area around equipment **41P-101A**.

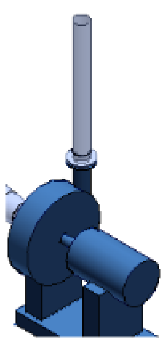
2. Click **Route Pipe**, and identify the discharge nozzle as the starting point.



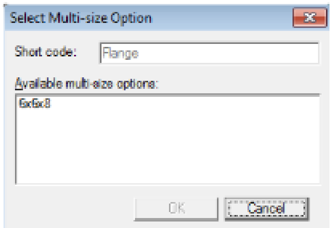
3. Enter the following run properties and accept them.

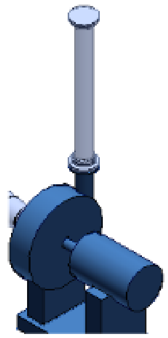


4. Key in a distance of **4.5ft** for the length of the first segment of core piping.

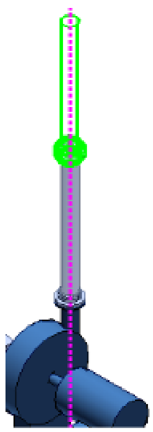


5. Click **Insert Component** and insert a **Flange** with the 6x6x8 multi-size option.

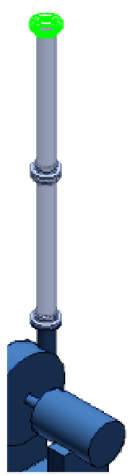




6. Continue routing and place another 4.5ft of core piping.



7. Finally, insert a third 6x6x8 flange (this completes the placement of the core pipe run).

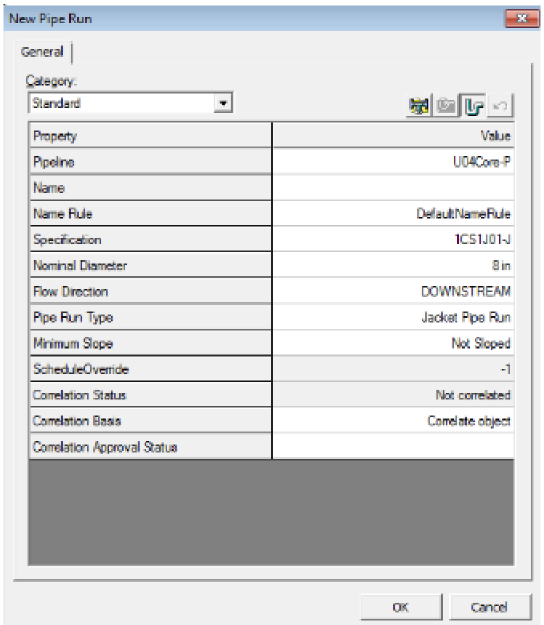


8. Now select each flange part (one at a time) and then change the **Mating Part Flanges** to **Base Part Flanges**.

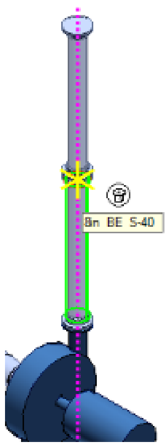
Next we will route the jacket for the core

9. Choose the route pipe command and identify the 8in port on the 6x6x8 core flange that is connected to the nozzle as the starting point for the jacket run.

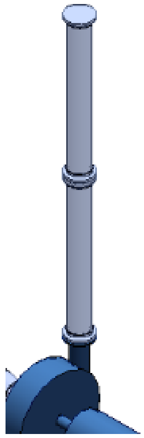
10. Enter the following route properties, and accept them.



11. Connect to the 8in port of the next flange to place the first jacket segment.

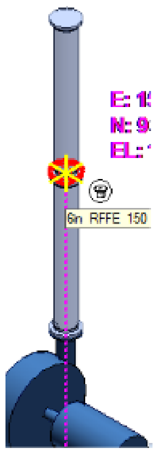


12. Place the second jacket in the same manner by connecting pipe between the two remaining flanges to complete the jacketed portion.



Finally, weldolets and smallbore piping will be modeled as jumpers to connect the jacketed piping across the flanged connection.

13. Route a 1" line to act as a jumper. Start about 1 foot above the flange pair, and then finish approximately 1 foot below the flange pair (connecting to the jacket).



14. Click **Route Pipe**  and start a branch off of the jacket (1 foot above the flange pair), using the following run information:

15. Route 1' to the East, then 2 feet down, and then back to the West to tie into the jacket on the other side of the flange.

SESSION 7

Placing Instruments

Objective

By the end of this session, you will be able to:

- Place an instrument in a pipeline.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)
- *Routing Pipes from P&ID* (on page 221)

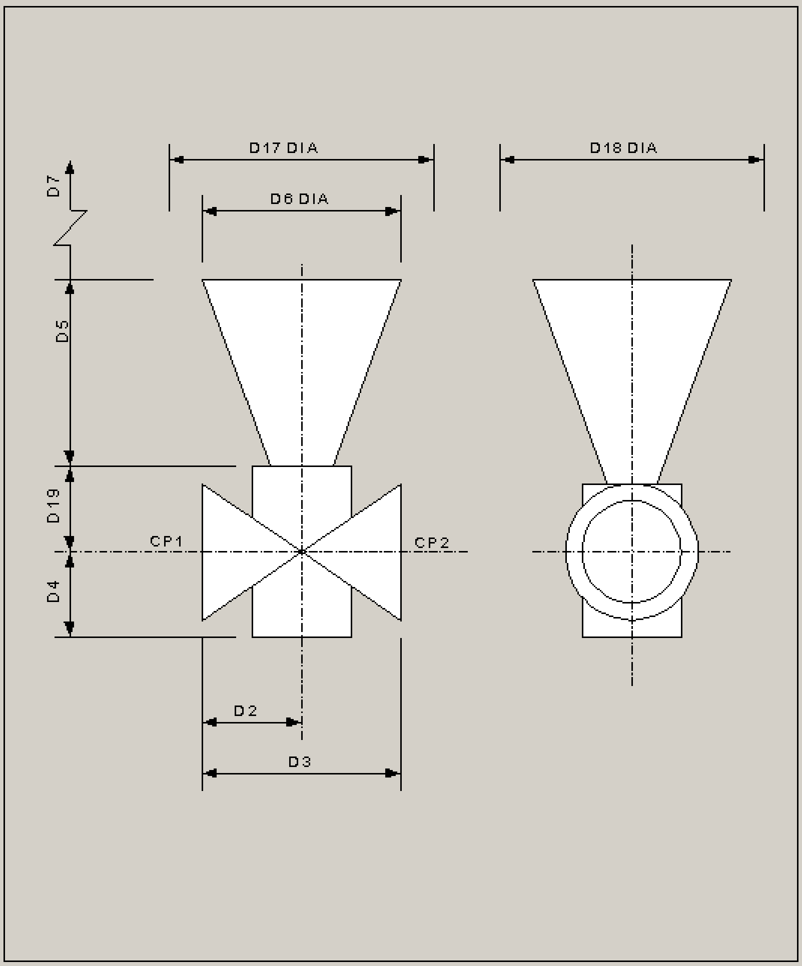
Overview

After placing a pipe, you might need to place instruments such as thermocouples and control valves at various places in the pipe run, to produce a desired condition in a controlled medium. For example, you might want to place components in an exchanger pipe that control the temperature of a process fluid heated by hot oil.

In SP3D, you can place either stock instruments or custom instruments depending on your requirement. The stock instruments are specific or typical instruments that are predefined in the reference data. These stock instruments are usually identified by using the tag numbers. The custom instruments are placed by selecting from a predefined hierarchy in the Catalog or from a PI&D file or from a SmartPlant Instrumentation Dimensional Datasheets (DDP). These instruments are typically driven by parameters and therefore their size, variable characteristics, and dimensions can be retrieved DDP or defined interactively at placement time or changed after placement in the model.

SmartPlant Instrumentation can produce dimensional data sheet from a selected instrument tag number. This data sheet includes general tag information, dimensional data of the specific group, and the graphic schematic of the instrument and its dimensions.

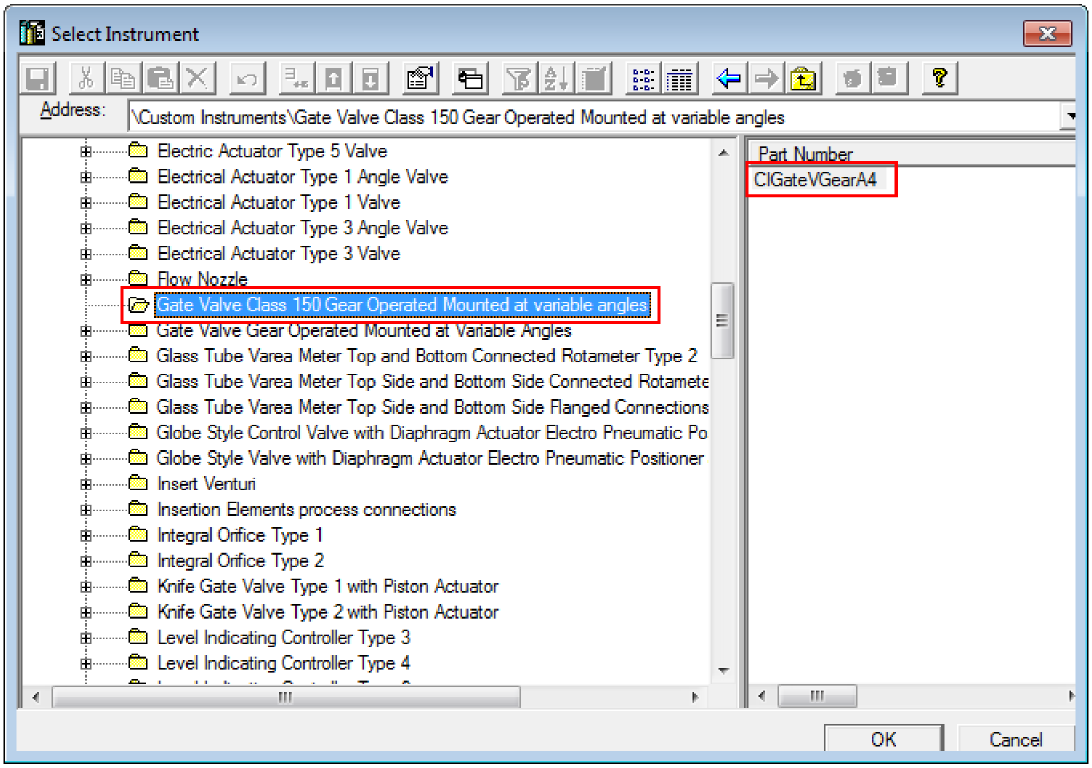
Description Line 1	Valve Linear Diaphragm Actuator	Group:	ISSDS		Tag Number:	
Description Line 2		Status:		Status date: 10/21/2007	FV -311	
Spec Document			Inlet	Outlet	Process Conn. #3	Process Conn. #4
Manufacturer		Size	8	8		
Model		Size UOM	in	in		
P&ID	PID1001	Class	150	150		
Piping Design Area		End Prep.	RF	RF		
Line Number	P-202-4"-1C0031-	Design Code	ASME-B16.5	ASME-B16.5		
Equipment		Pipe Sched.				
		Weight	UOM	Dry	Full	No.Connect Points=D1



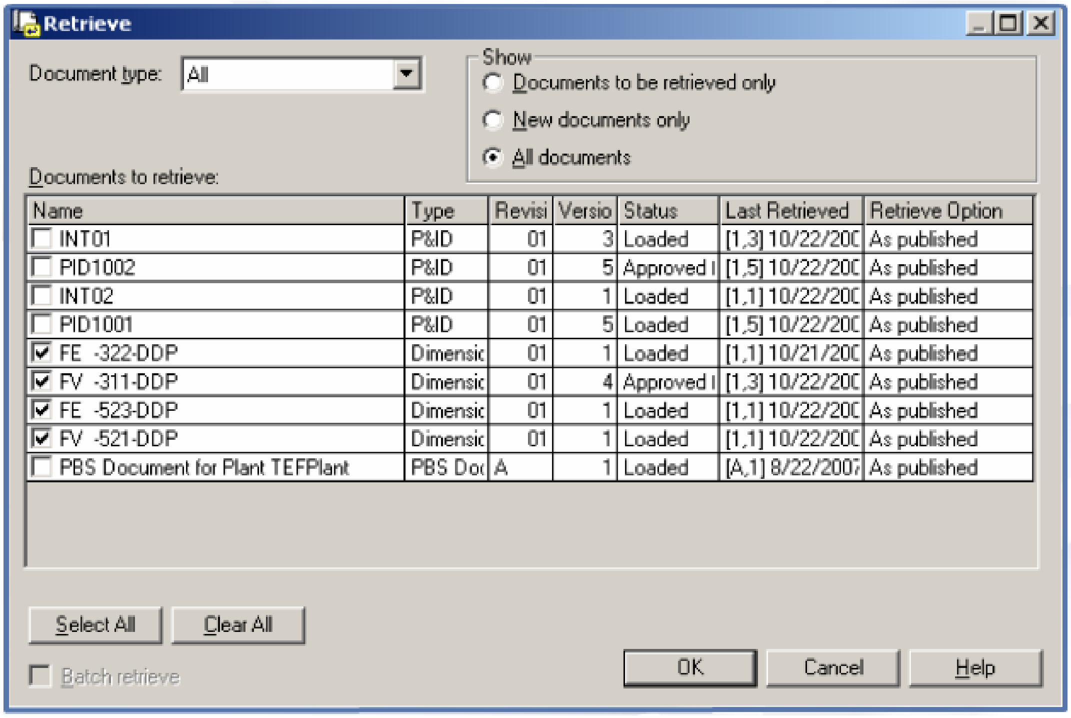
D2	6
D3	12
D4	2
D5	10
D6	12
D7	10
D17	16
D18	18
D19	8

01	rh	10/21/2007		DIMENSIONAL DATASHEET	INTERGRAPH
					Sheet 1 of 1
No.	By	Date	Revision	CAD Code: PDS	Doc. No.: FV -311-DDP Rev.: 01

You use **Insert Component**  on the vertical toolbar to place stock instruments and custom instruments in a pipe run. The custom instruments that you can choose for a particular model are listed in the **Select Instrument** dialog box.



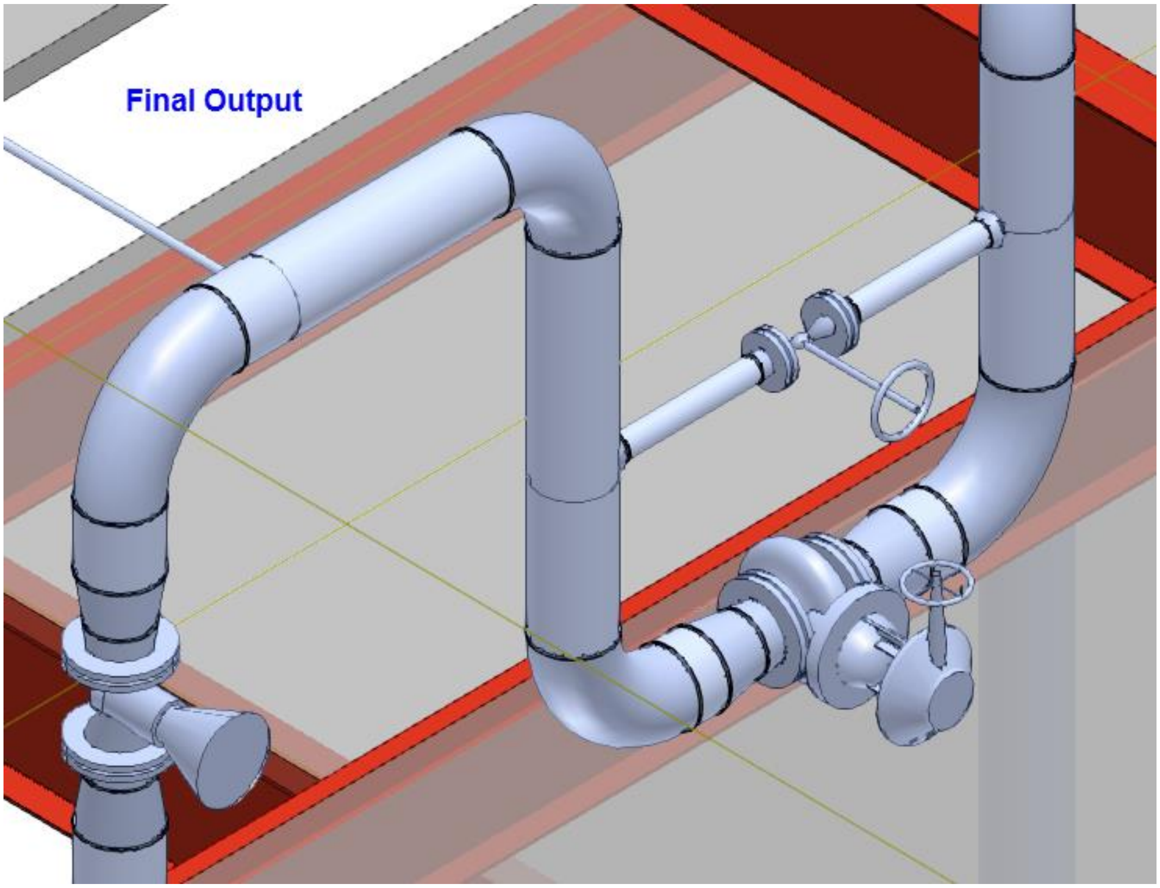
You can place the new correlated instruments on a pipe run by using the DDP supplied by the SmartPlant Instrumentation. To retrieve these design basis data you can use the SmartPlant > Retrieve command to open a dialog box that assists you in retrieving the applicable documents.



Placing Instruments in a Pipeline

Objective

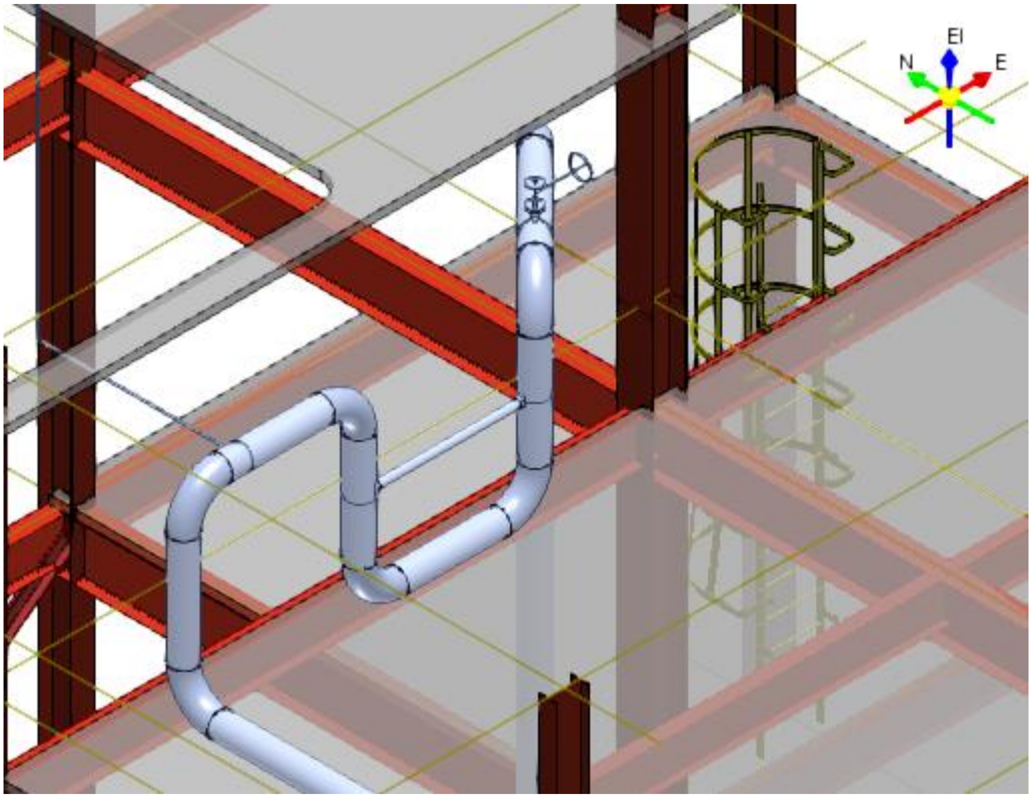
In this exercise you will be manually inserting the instruments **ClGateVGearA4**, **VL2** on Catalog Basis and custom instrument **FV-311** on Design Basis from DDP along the pipeline **300-W** in **Unit U03** of your workspace. The view of your model after placing the instruments should resemble the highlighted area in .



Before Starting this Procedure

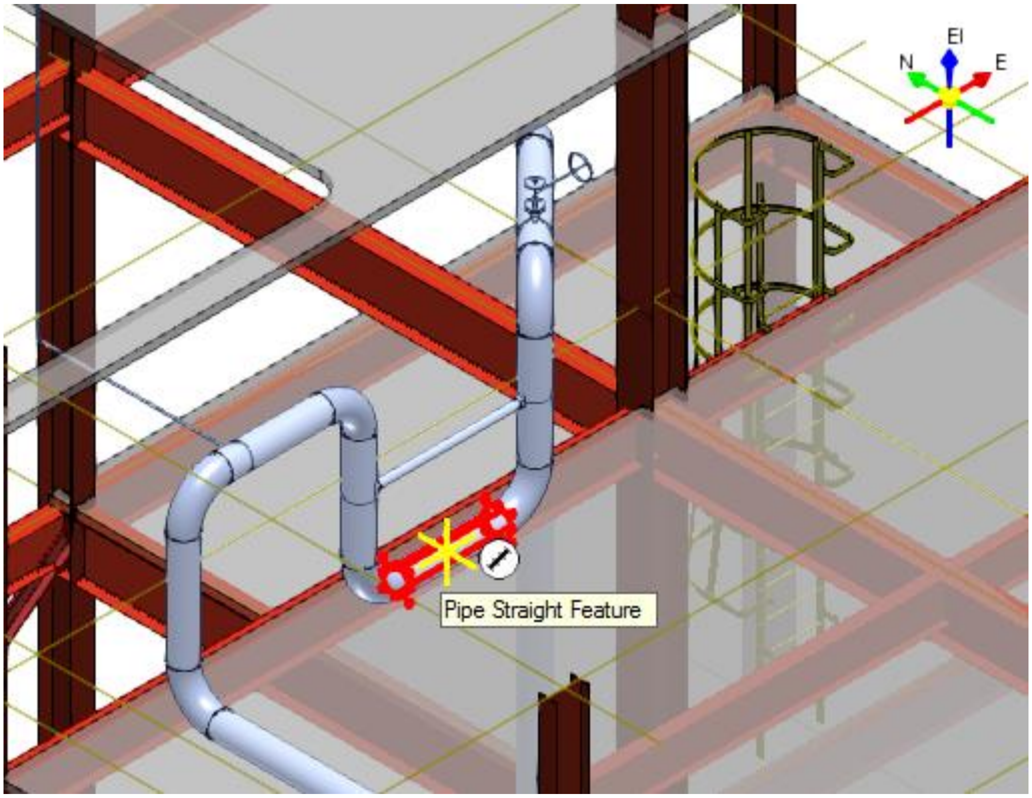
- Define your workspace to display **Unit U03** and coordinate system **U03 CS**. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Click **Zoom Area** on the Common toolbar and zoom in to the area where the Instrument needs to be placed.



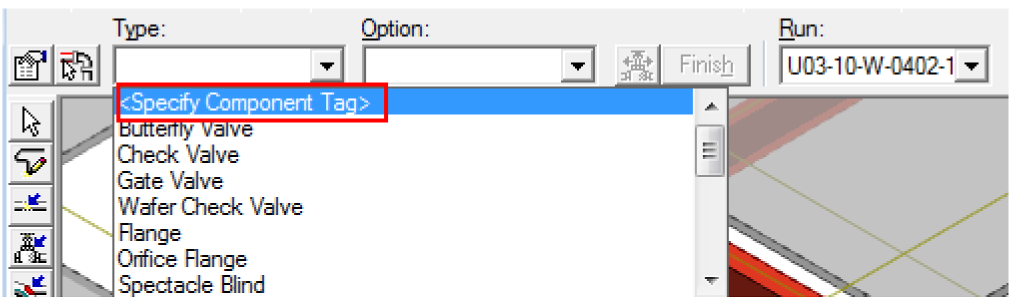
2. Click **Insert Component** on the vertical toolbar to start placing the instrument.

3. Using SmartSketch, locate the midpoint of the pipe.



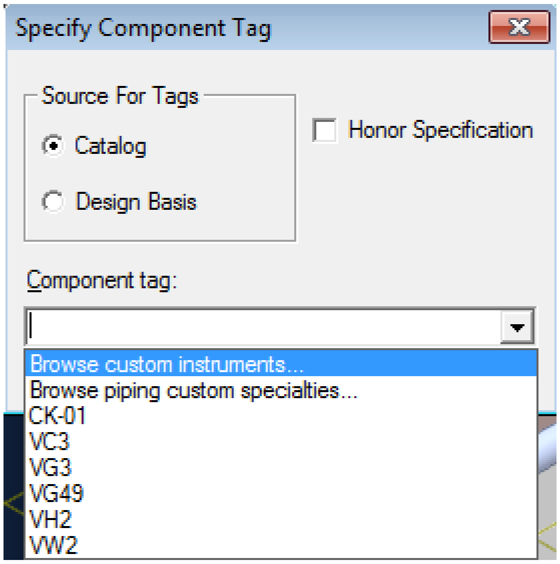
*The **Type** drop-down list on the **Insert Component** ribbon opens and displays the list of components that you can place in the pipe.*

4. Select the **<Specify Component Tag>** option in the list.



The Specify Component Tag dialog box appears.

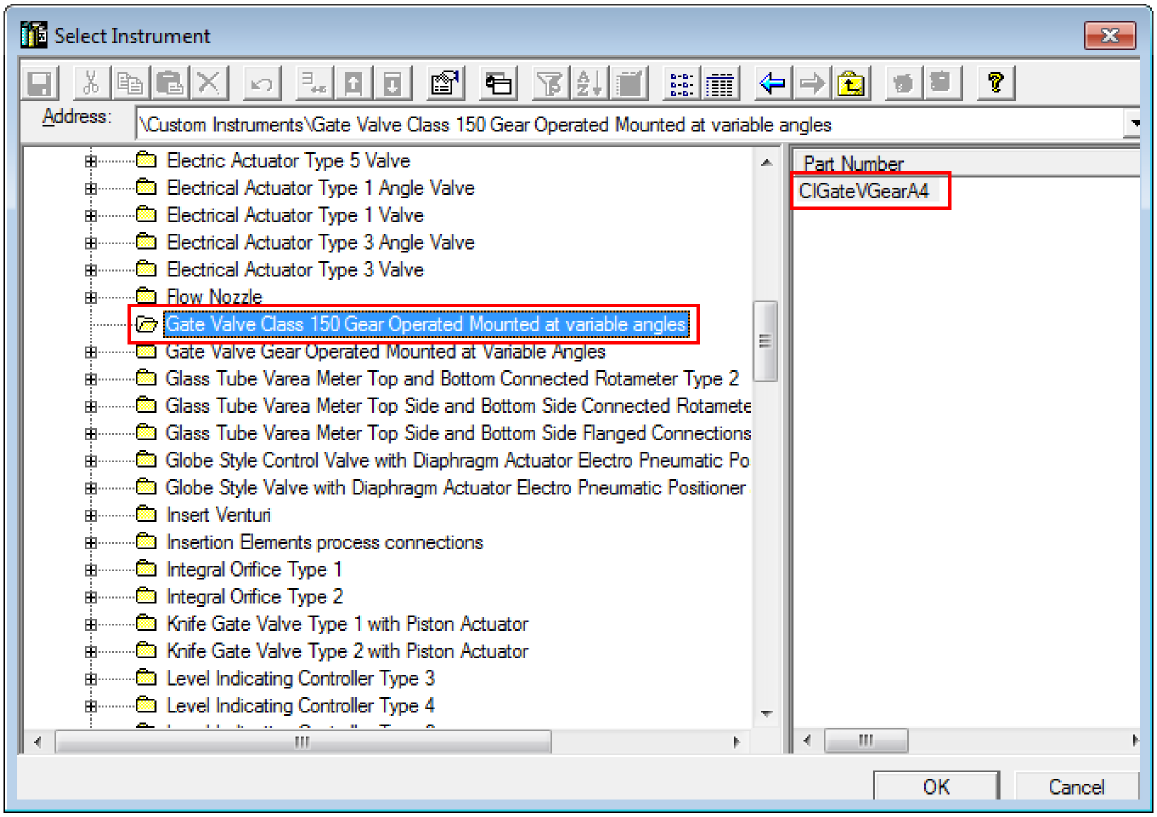
5. Select the **Browse instruments…** option in the **Component** tag drop-down list, and click **OK**.



The Select Instrument dialog box appears.

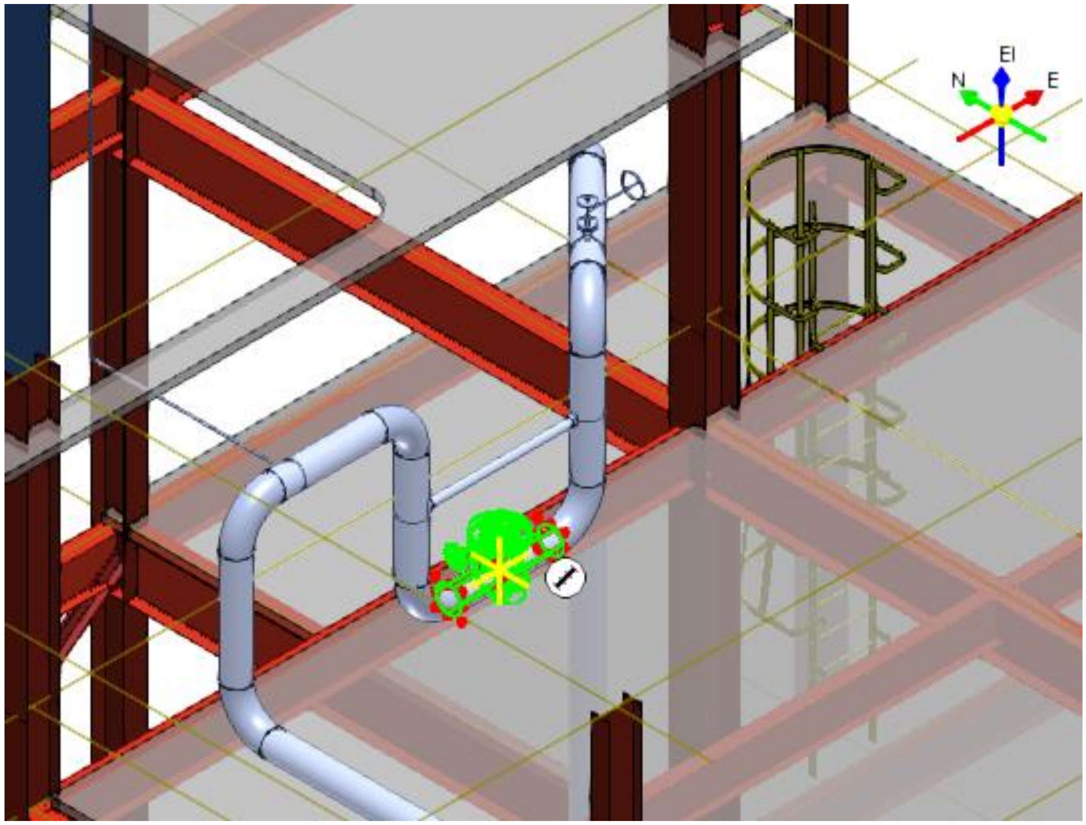
The Select Instrument dialog box displays a list of the available custom instruments that you can place in the pipe run. These custom instruments are already defined in the Catalog.

6. Navigate to **Gate Valve Class 150 Gear Operated Mounted at variable angles** and click the part number **ClGateVGearA4** in the right pane. Then, click **OK** to close the **Select Instrument** dialog box.



TIP You can preview the custom instrument that needs to be placed by clicking **Preview** on the toolbar.

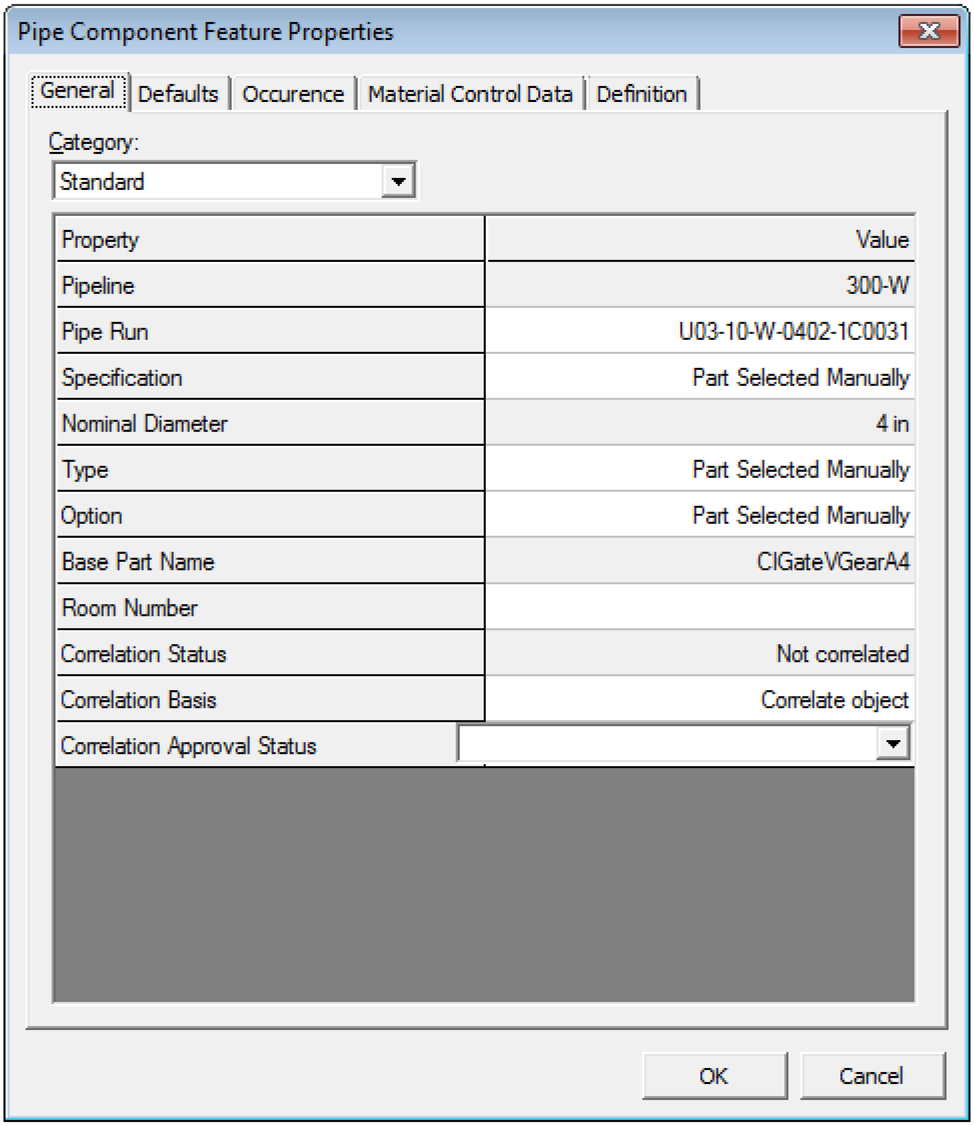
You will now see an outline of the valve in your model.



You can also change the properties of the valve before placing it. Perform the following steps to change the properties of the valve:

7. Click **Common Views** on the Common toolbar to change the view to **Looking North**. This will help you in getting a better view of the valve you are going to place.
8. Click **Properties** on the **Insert Component** ribbon.

*The **Pipe Component Feature Properties** dialog box appears. This dialog box has specifications automatically populated by the system.*

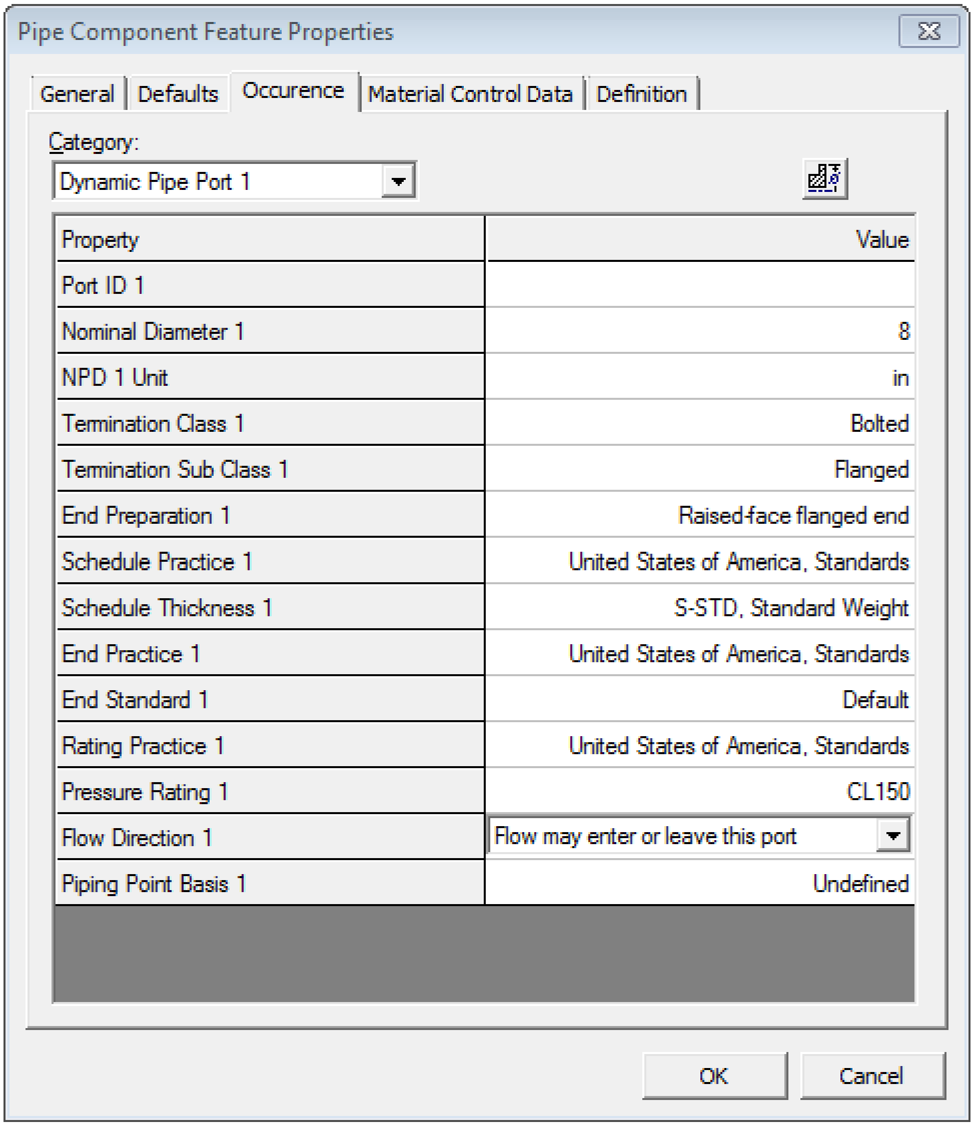


9. In the **Pipe Component Feature Properties** dialog box, click the **Occurrence** tab to change the properties of the valve.

10. The instrument valve you are going to place has two ports. Change the properties of the ports one by one. In the **Category** drop-down list, select the **Dynamic Pipe Port 1** option and set the following parameters:

    Nominal Diameter 1: 8
    NPD 1 Unit: in
    Termination Class 1: Bolted
    Termination Sub Class 1: Flanged
    End Preparation 1: Raised-face flanged end
    Schedule Practice 1: United States of America, Standards
    Schedule Thickness 1: S-STD, Standard Weight

End Practice 1: United States of America, Standards
End Standard 1: Default
Rating Practice 1: United States of America, Standards
Pressure Rating 1: CL150
Flow Direction 1: Flow may enter or leave this port
Piping Point Basis 1: <undefined value>

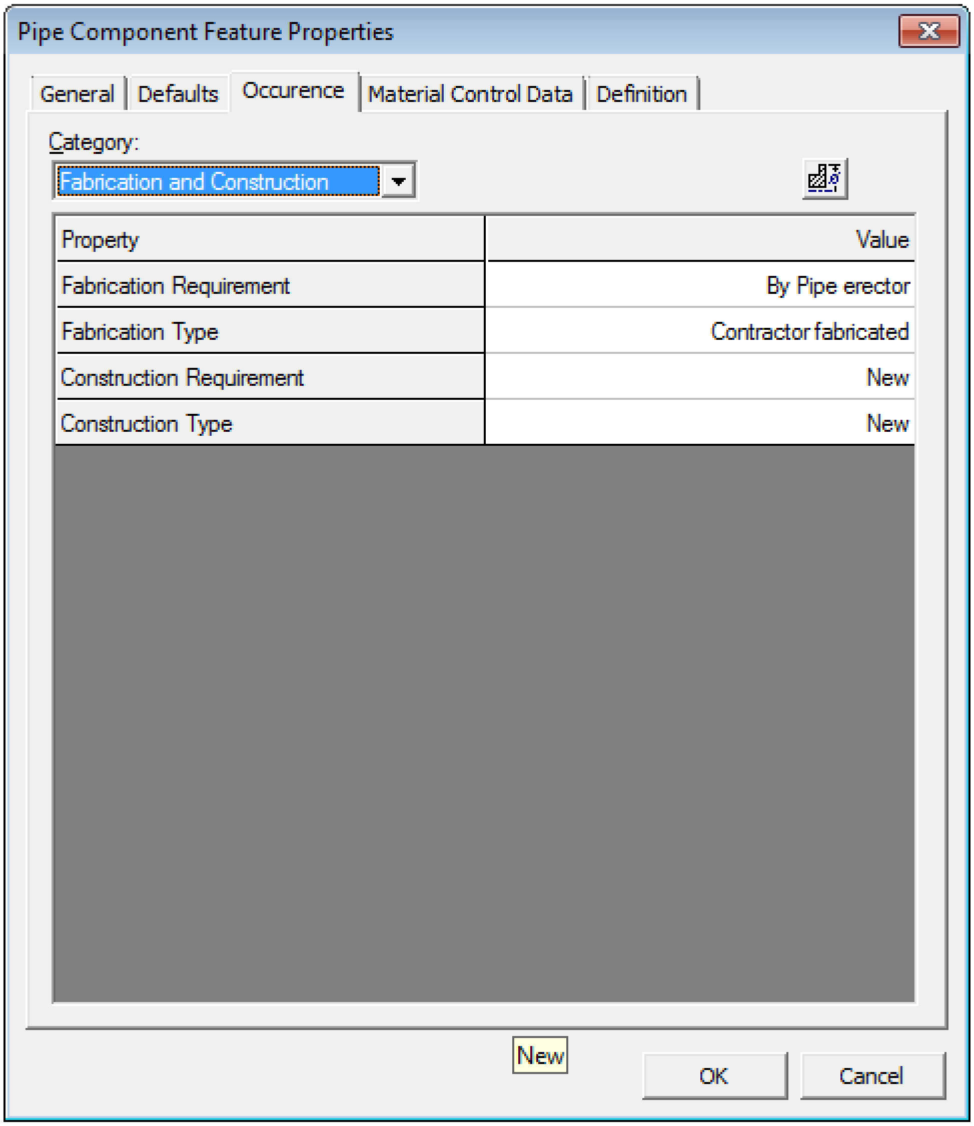


11. Now, in the **Category** drop-down list, select the **Port 2** option and set the same parameters that you set for Port 1.

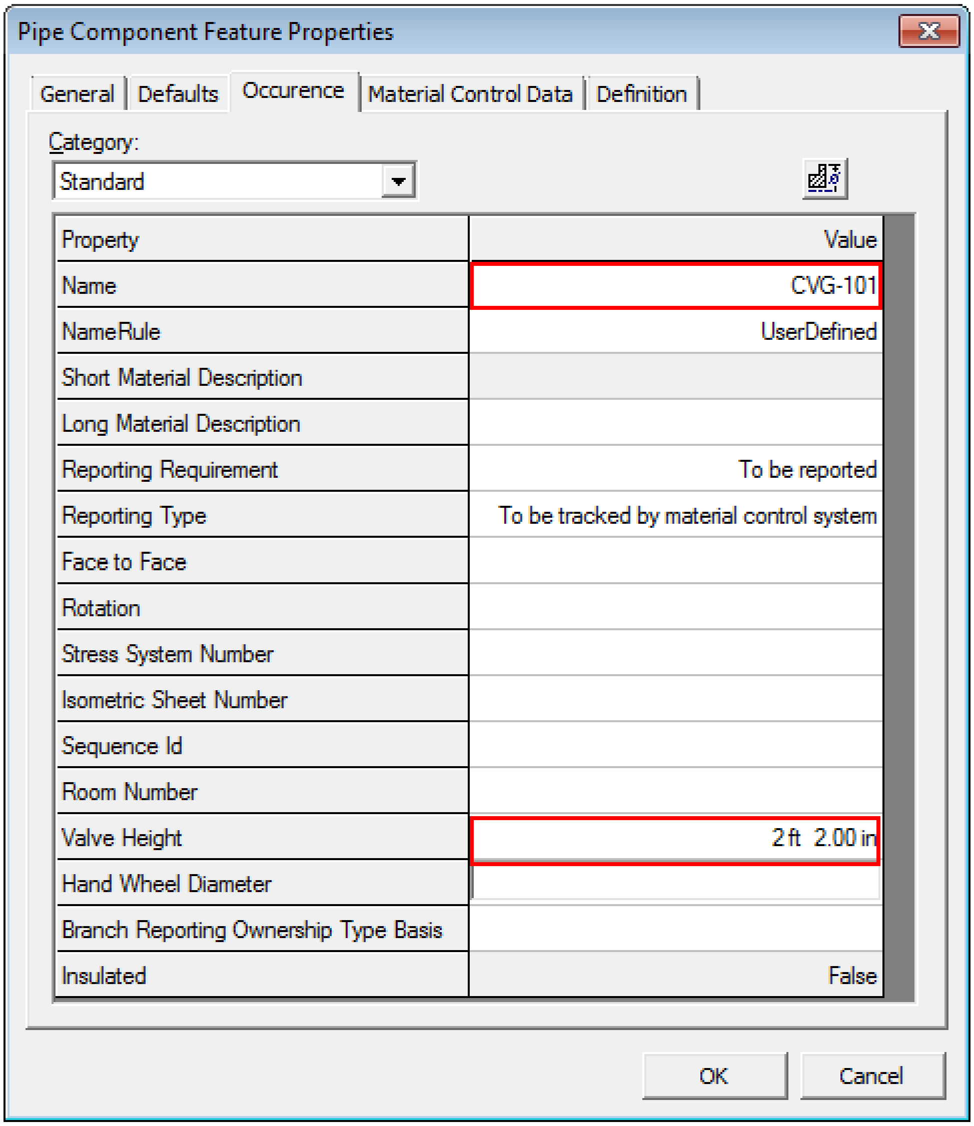
12. To define the fabrication and construction requirements for the instrument valve, in the **Category** drop-down list, select the **Fabrication and Construction** option and set the following parameters:

    Fabrication Requirement: By Pipe erector
    Fabrication Type: Contractor fabricated

Construction Requirement: New
Construction Type: New

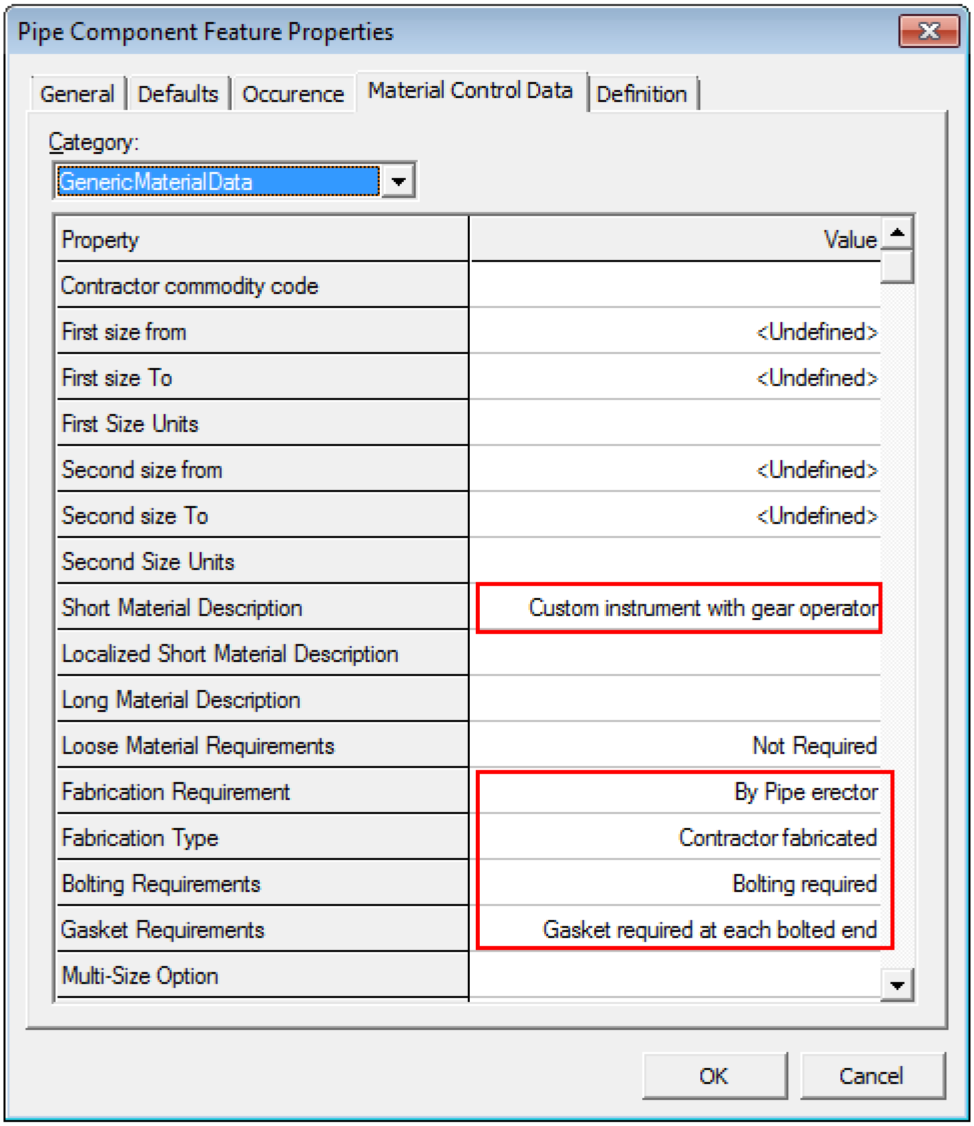


13. In the **Category** drop-down list, select the **Standard** option.
14. Key in **2 ft 2 in** the **Valve Height** field to define the height of the instrument valve and **CVG-101** as the tag number in the **Name** field for this instrument.



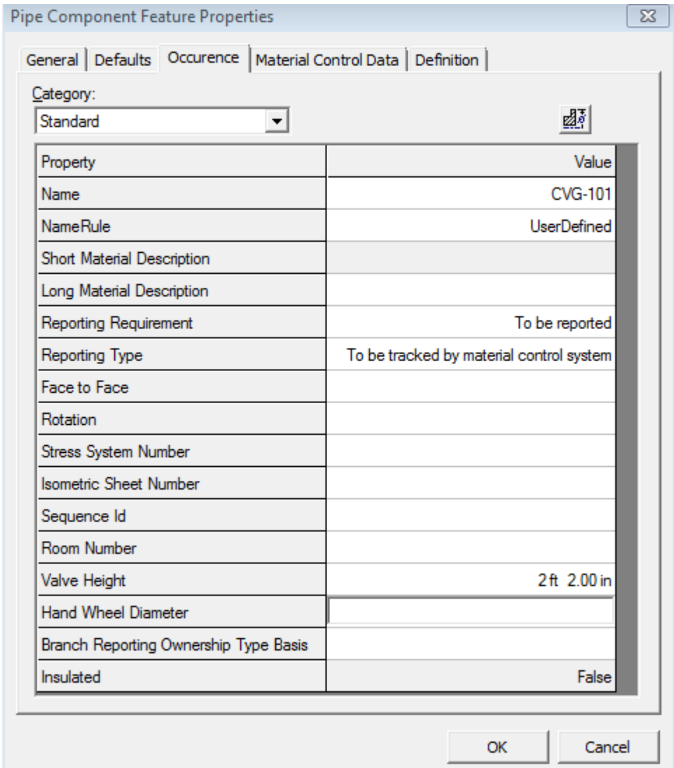
15. Click the **Material Control Data** and then select the **GenericMaterialData** option under the **Category** drop-down list. Change the material properties for the instrument valve you are placing. Make the following modifications:

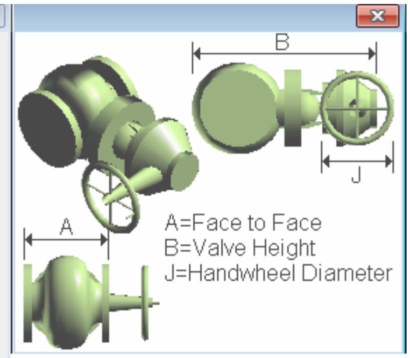
Short Material Description: Custom instrument with gear operator
Fabrication Requirement: By Pipe erector
Fabrication Type: Contractor fabricated
Bolting Requirements: Bolting required
Gasket Requirements: Gasket required at each bolted end



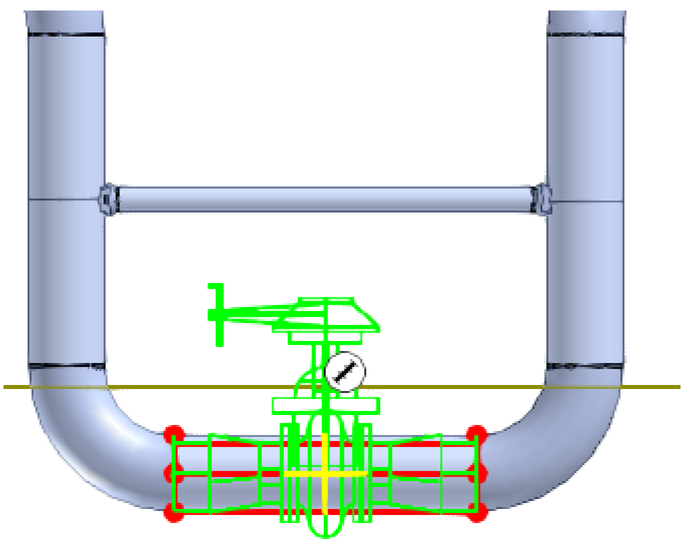
16. Click **OK** to close the **Pipe Component Feature Properties** dialog box.

TIP You can preview the instrument and the instrument specifications by clicking the **Show Dimensional Legend** button on the **Occurrence** tab in the **Pipe Component Feature Properties** dialog box.



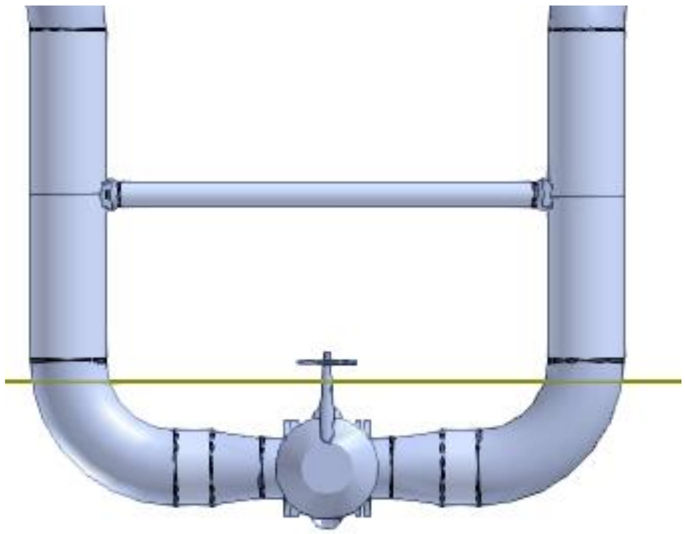


Notice that the system has placed appropriate mating components on the valve based on the data you entered in the Pipe Component Feature Properties dialog box.



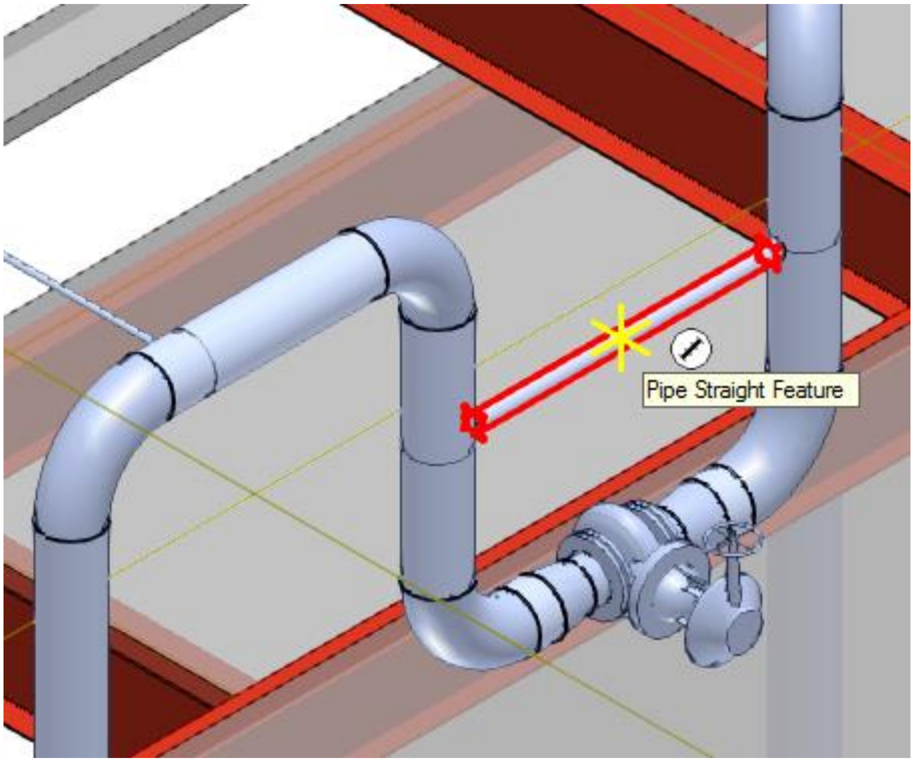
17. Now, you need to rotate the valve so that the valve operator is on the side so that it is easy to operate. On the **Insert Component** ribbon, key in **90 deg** in the **Angle** drop-down list and **0 deg** in the **Operator Angle** drop-down list.

18. Click **Finish** to place the valve.



19. Click **Common Views**  on the Common toolbar to change the view to **Isometric**.

20. Next, place a stock instrument on the pipe highlighted. Click **Insert Component** on the vertical toolbar and use SmartSketch to locate the midpoint of the pipe.

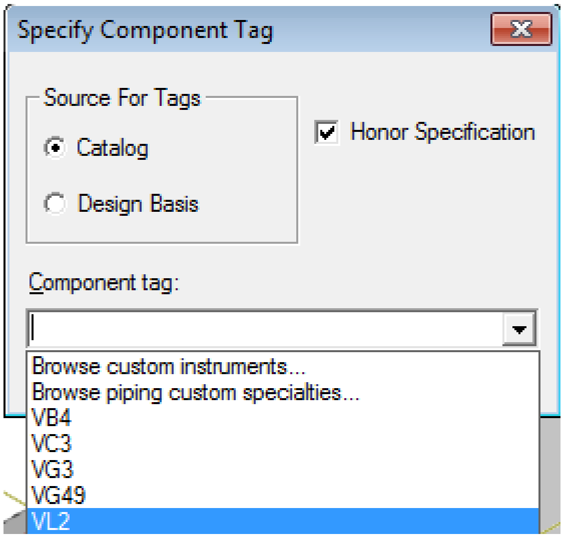


21. Click the midpoint of the pipe to start placing the stock instrument. The **Type** drop-down list on the **Insert Component** ribbon opens. Select the **<Specify Component Tag>** option in the list.

    The Specify Component Tag dialog box appears.

22. Select the **Catalog** option for source tags.
23. Click on the **Component** tag drop-down list to observe that all the Catalog items available are displayed irrespective of the Size and Spec used by Pipe Run.
24. Now check the **Honor Specification** option, and click on the **Component** tag drop down list to see only the Instruments/Specialties applicable to the selected Pipe.

25. Select **VL2** to place a Globe Valve as shown below.

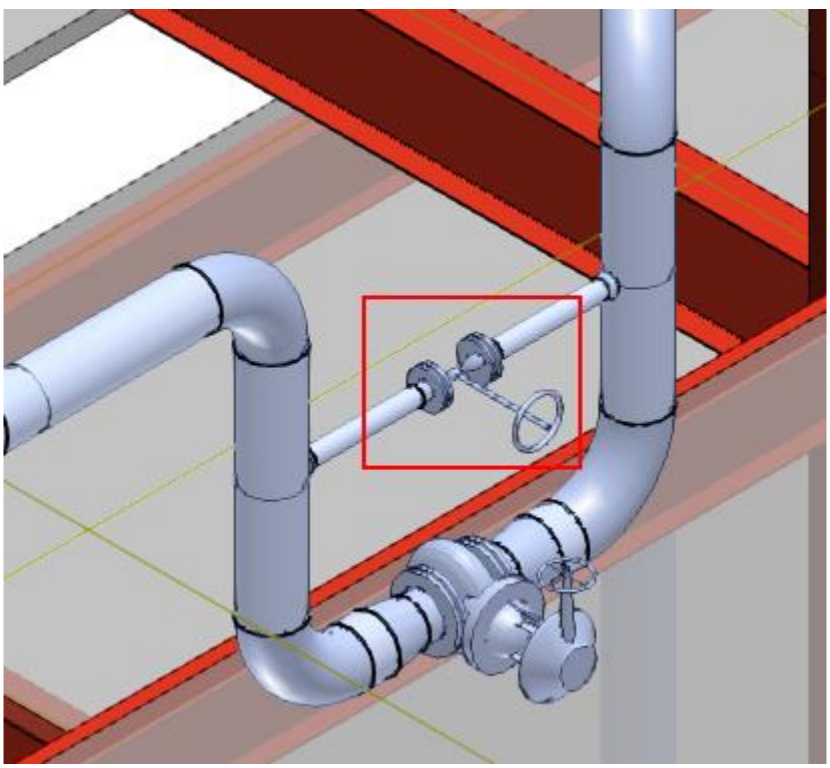


26. Click **OK** to close the box.

    An outline of the instrument will appear in the graphic view.

27. On the **Insert Component** ribbon, key in **270 deg** in the **Angle** drop-down list to rotate the instrument towards the south direction.

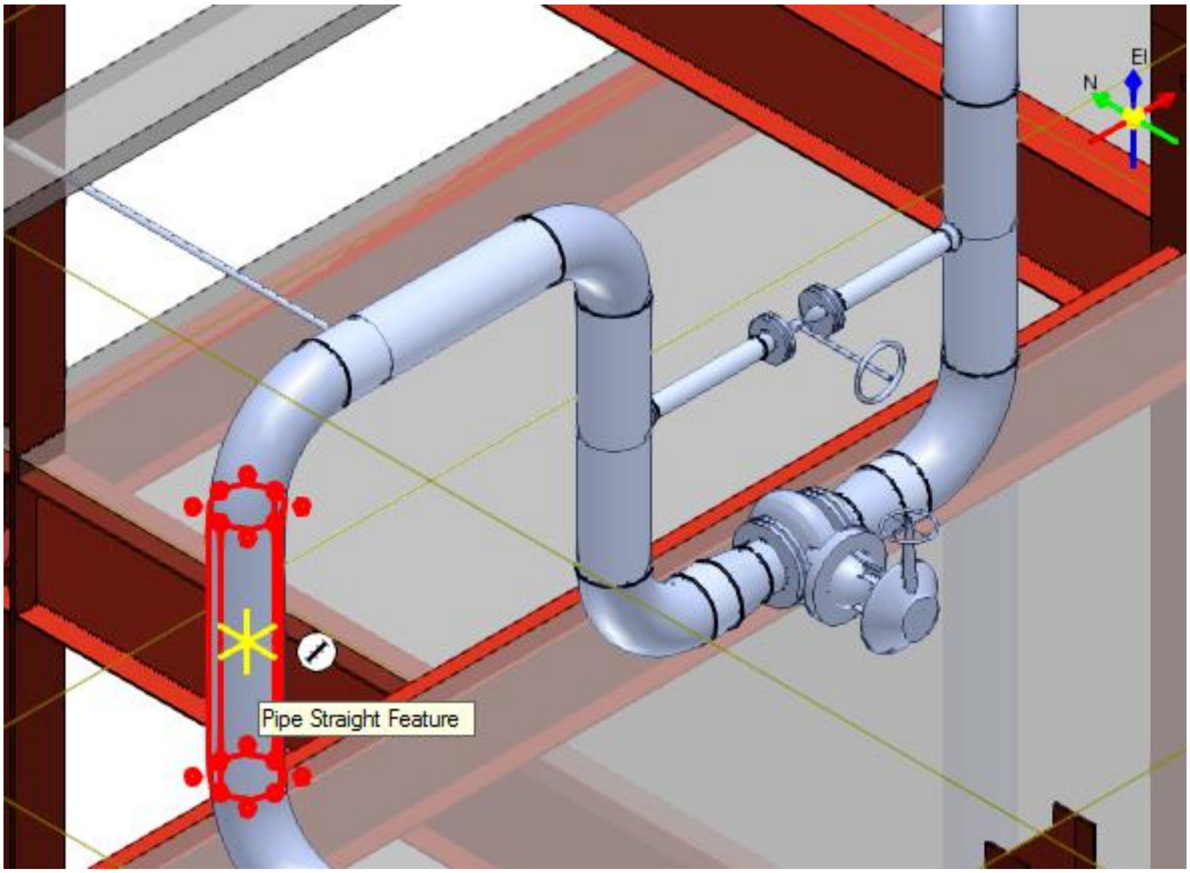
28. Click **Finish** to place the instrument.



Placing Instruments using Design Basis

Now, you need to place another custom instrument FV-311 by using the retrieved DDP on the pipeline 300-W highlighted in .

1. Click **Insert Component** and use SmartSketch to locate the midpoint of the pipe.

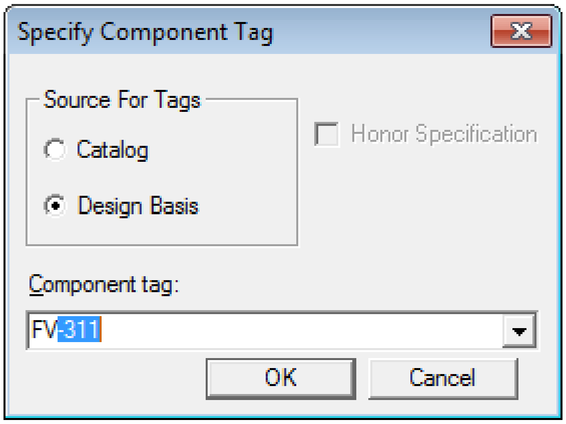


*The **Type** drop-down list on the **Insert Component** ribbon opens.*

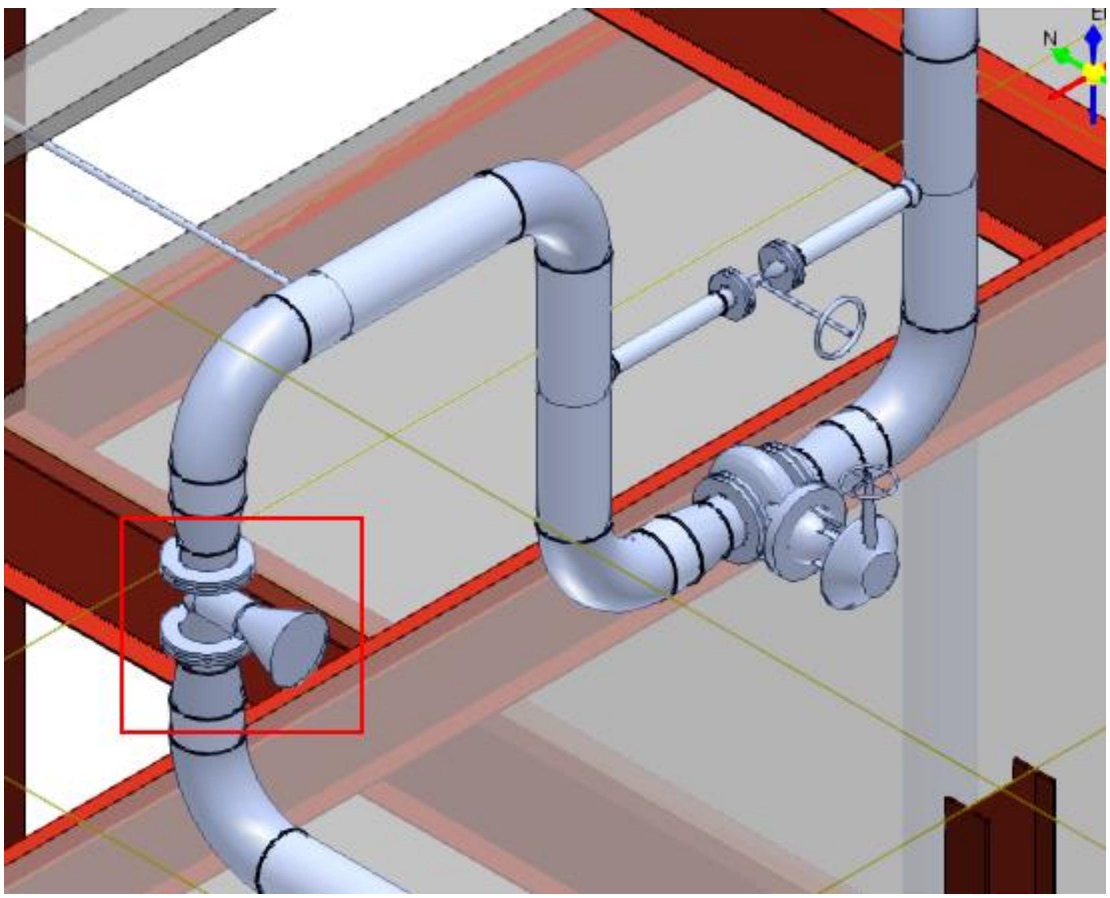
2. Select the **<Specify Component Tag>** option in the list.

*The **Specify Component Tag** dialog box appears.*

3. Select the **Design Basis o**ption to display the source tags and type "FV"in the Component tag drop-down list to choose FV-311 as shown below.



4. The placed FV-311 instrument should resemble the highlighted.
5. Key in **-90 deg** in the **Angle** drop-down list to rotate the instrument and click **Finish** to place it.



NOTE Smart 3D finds the dimensional data sheet published from SmartPlant Instrumentation using the tag number FV-311. It then reads the dimensional data sheet, applies those values to the custom instrument in the background, and then places the instrument on the pipe feature.

SESSION 8

Placing Piping Specialty Items

Objective

By the end of this session, you will be able to:

- Place a piping specialty item in a piping system.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

Overview

Specialty items are piping components that are not defined as a part of piping specification. Like instruments, you can place specialty items in a pipeline to perform a specific task. For example, you use specialty items such as separators, strainers, and screens to filter unwanted debris from the fluid before it reaches instrument valves or pumps. The specialty items are tagged by numbers.

In piping, you can place two types of specialty items, stock and custom. Stock specialty items represent those piping items that are purchased from a manufacturer’s catalog, where no real engineering is required other than selecting the correct size and material. These specialty items are usually placed by their tag numbers. Custom specialty items are custom-made according to the process and usually are placed by selecting from a pre-defined hierarchy in the catalog. The custom specialty items are typically driven by parameters. Therefore, you can change their size and shape after placing them in the model.

This session will cover the procedure for placing a specialty item silencer on a pipe run to reduce the noise when the fluid is released to the air.

Placing Piping Specialty Items

Objective

In this exercise you will be placing a specialty item silencer **CSInlineSilencer4** on equipment **T-101** in **Unit U03** of your workspace to reduce the noise. Before placing the silencer, you will need to position the following components and instruments on the nozzle **F** of equipment **T-101** on which the silencer will be positioned. These instruments and components will control the pressure of the fluid coming out of the vessel before it reaches the silencer.

Isolating Gate Valve:

Place an isolating gate valve to maintain the pressure of the following specifications:

Pipeline: 302-W
Specification: 2C0032
Angle: 270 deg

Pressure Relief Valve (PSV):

Place a pressure relief valve (PSV) IRVT 34 to protect the equipment from being subjected to pressures that exceed their design limits, of the following specifications:

Occurrence tab:

Dynamic Pipe Port 1 category

Nominal Diameter 1: 3
Pressure Rating 1: CL150

Fabrication and Construction category

Fabrication Requirement: By erector

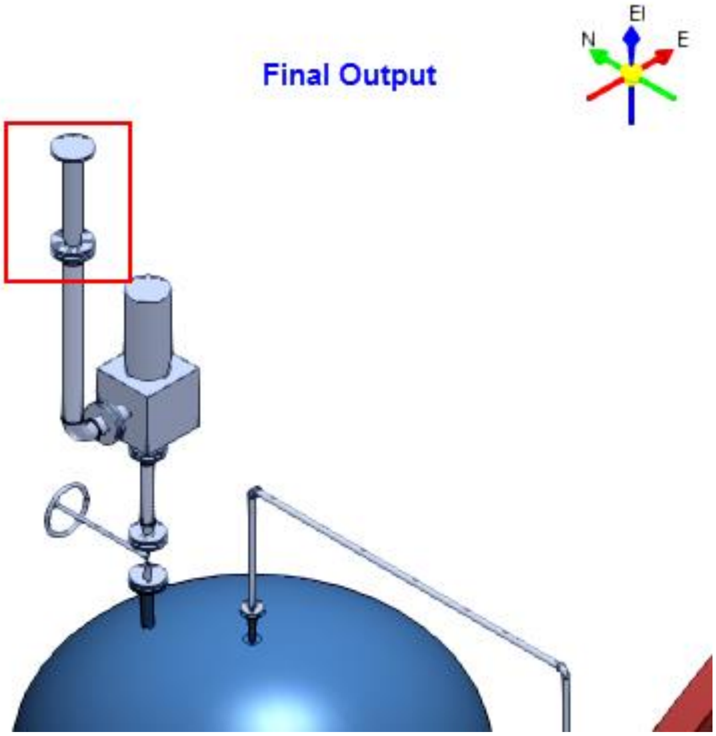
Material Control Data tab:

Short Material Description: Relief Valve
Fabrication Requirement: By erector
Bolting Requirements: Reportable bolts required
Gasket Requirements: Gasket required at each bolted end

Vent Pipe

Place a **3 ft** long vent pipe along with an elbow to control the fluid direction from the other side of the relief valve.

After the specialty item is placed, the view of the model should resemble this.

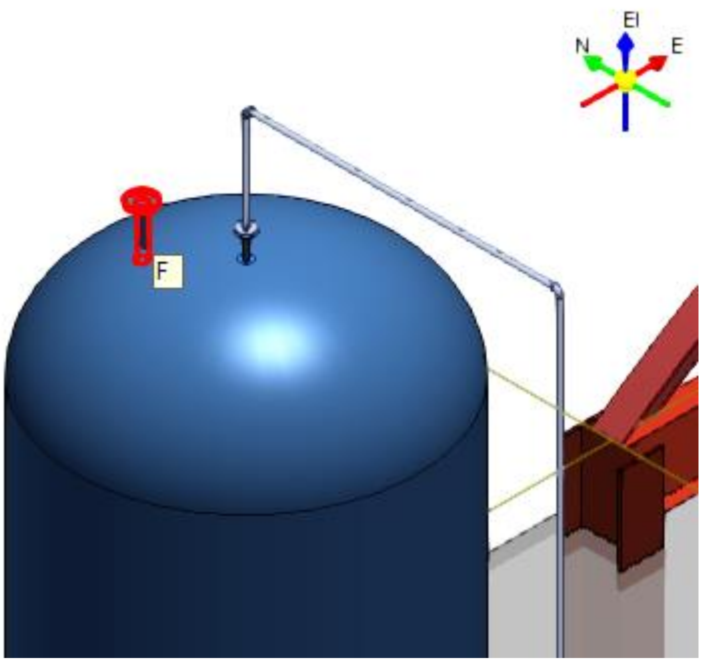


Before Starting this Procedure

- Define your workspace to display **Unit U03** and coordinate system **U03 CS**. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

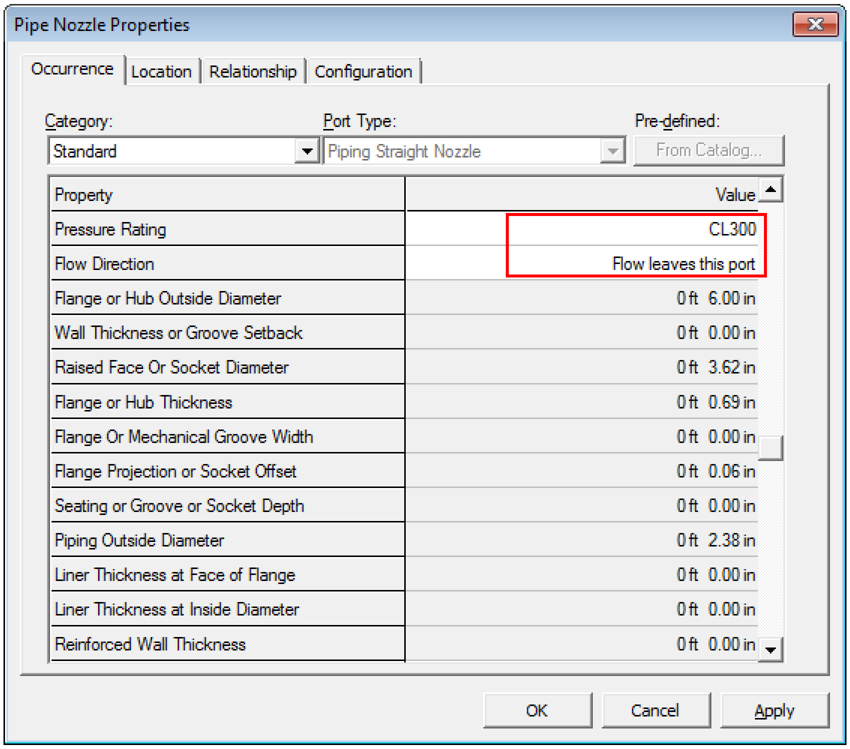
Placing Isolating Gate Valve

1. Locate nozzle **F** on equipment **T-101** in Unit **U03** on which you will insert components and specialty items.



2. Right-click on the nozzle, and open the **Pipe Nozzle Properties** dialog box to change the pressure rating and flow direction for placing components on the nozzle.
3. Change the following specifications in the **Pipe Nozzle Properties** dialog box, and click **OK**.

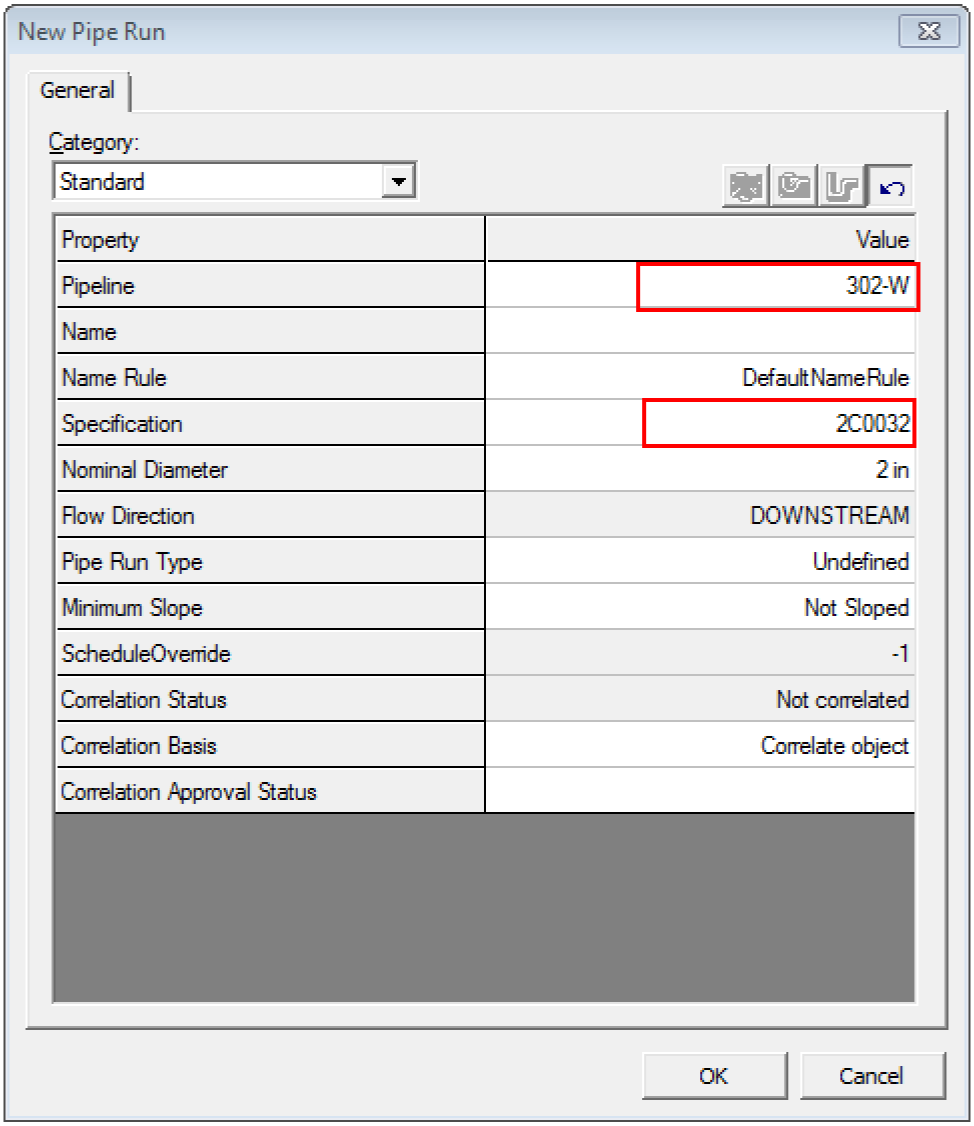
Pressure Rating: CL300
Flow Direction: Flow leaves this port



4. Click **Insert Component** .
5. Select the pipe nozzle **F** for placing the components.

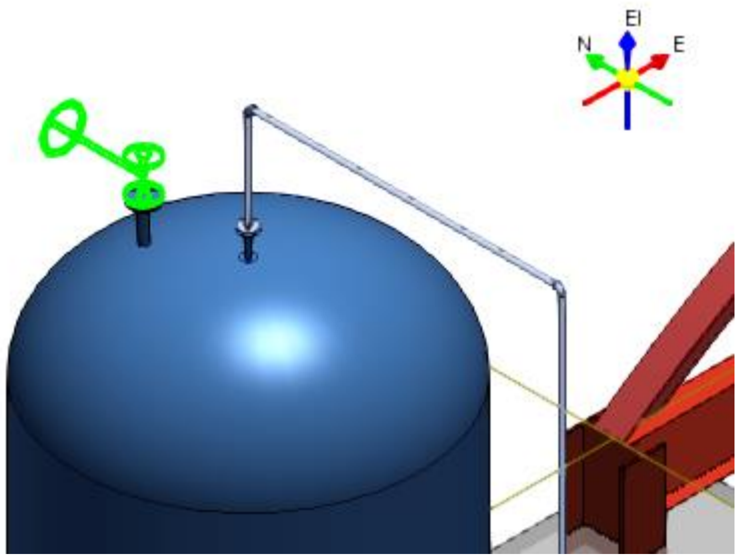
   The New Pipe Run dialog box appears
6. Change the following specifications in the dialog box, and click **OK**.

Pipeline: 302-W
Specification: 2C0032



7. Place an isolated valve Gate Valve on the nozzle F. Select the **Gate Valve** option from the **Type** drop-down list on the Insert Component ribbon.

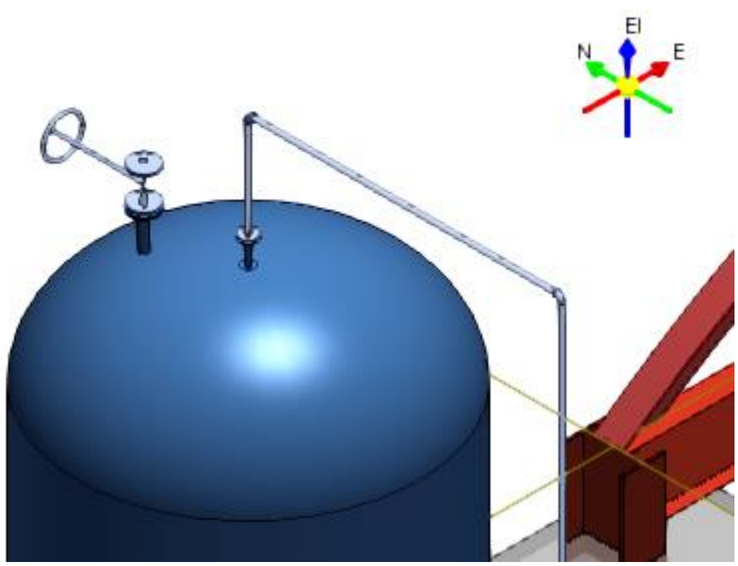
The gate valve appears in the graphic view.



8. Key in **270 deg** in the **Angle** box to rotate the gate valve so that the valve operator is oriented correctly.

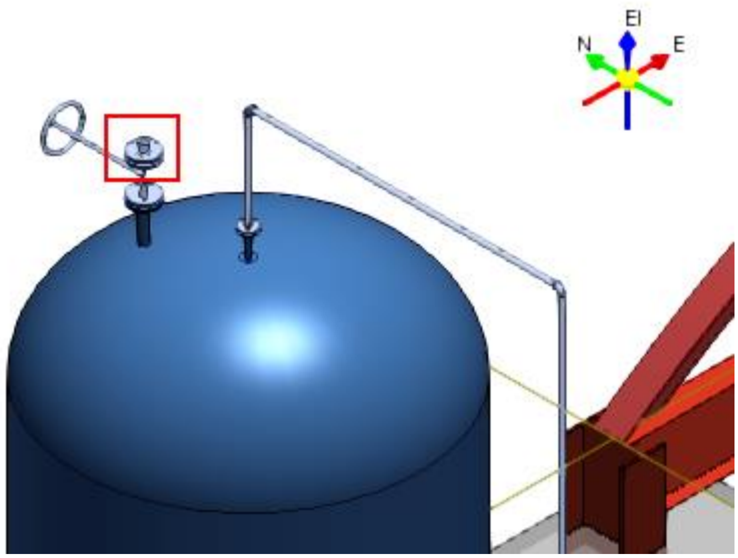
9. Click **Finish** to accept the placement of the gate valve.

   The placed gate valve should resemble this.

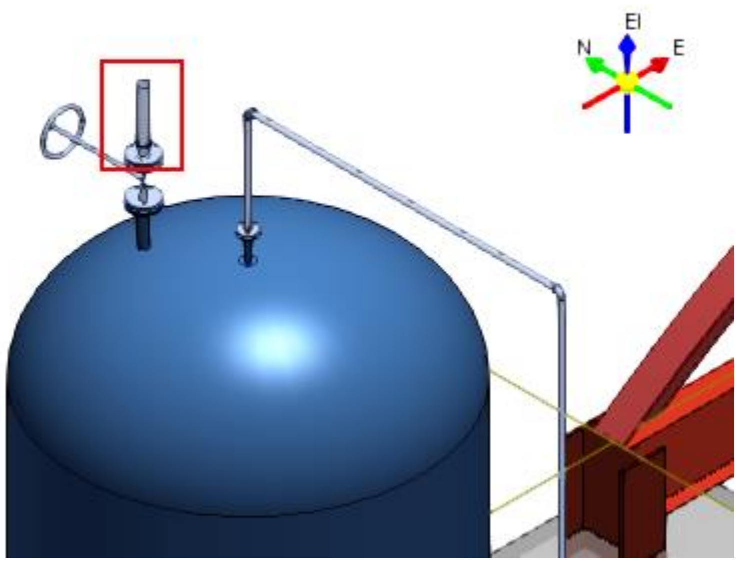


10. Select **Flange** from the **Type** drop-down list to place a flange on top of the gate valve.
11. Click **Finish** to accept the placement of the flange.

The flange placed on the gate valve should resemble this.



12. To let the fluid come out of the valve, place a pipe on the gate valve. Route a **1 ft** pipe from the flange placed above by using **Route Pipe** .

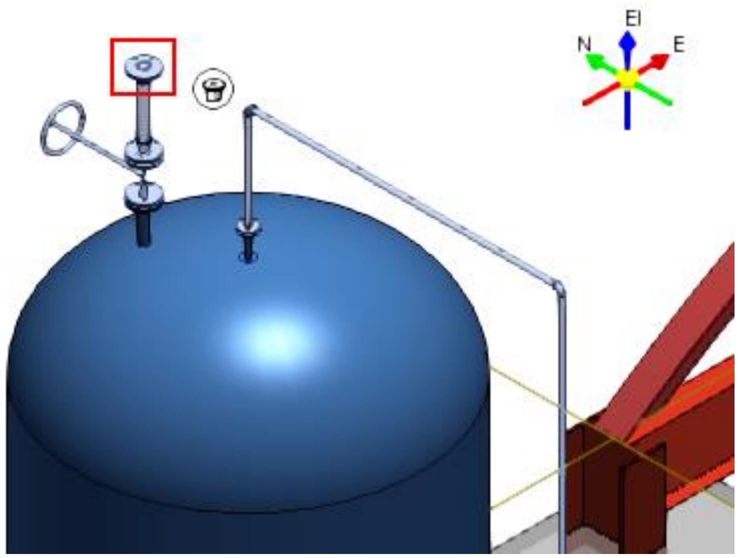


13. Right-click to terminate **Route Pipe**.
14. Click **Insert Component** and select the end feature of the pipe to cover the top of the pipe with a flange.

    Smart 3D prompts you to select type of the component to place.

15. Select **Flange** from the **Type** drop-down list and click **Finish** to accept the placement of the flange.

The flange is placed in the model.



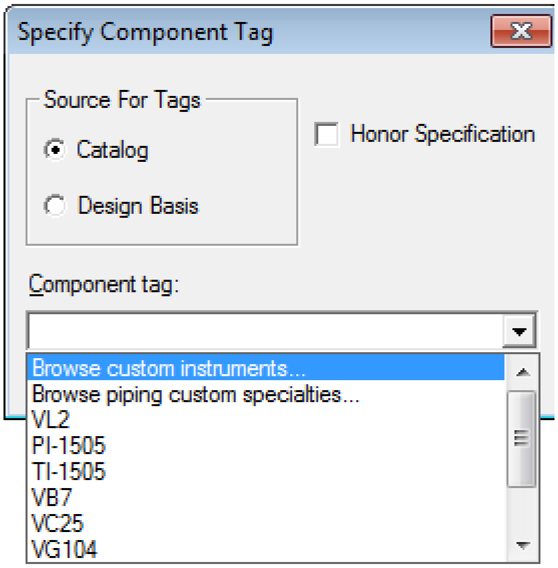
Placing Pressure Relief Valve

After placing the gate valve and a small spool, place a pressure relief valve IRVT 34 from the catalog. PSV is a type of instrument and refers to safety or relief devices. These devices are used to protect against emergency pressure conditions. This valve controls or limits the pressure in a piping system or vessel which can build up by an instrument or equipment failure, or fire. The pressure is relieved by allowing the pressurized fluid to flow from an auxiliary passage out of the system.

1. Select **Specify Component Tag** from the **Type** drop-down list to select the item from the catalog.

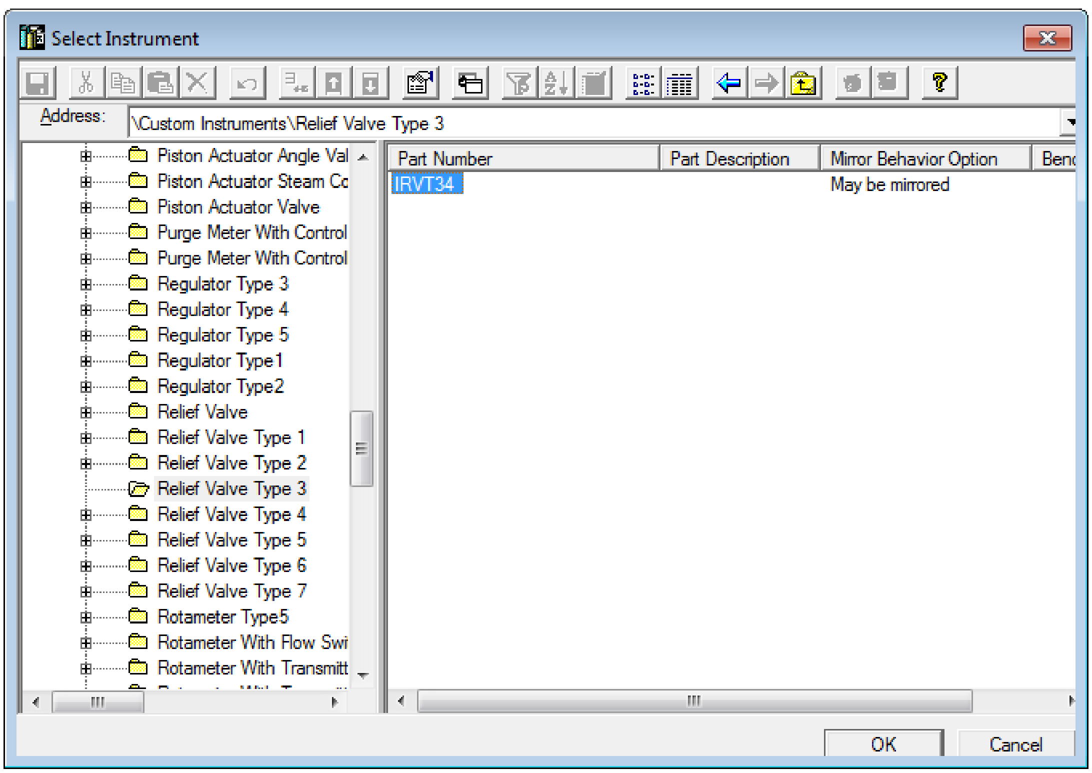
   The Specify Component Tag dialog box appears.

2. Select **Browse instruments…** from the **Component Tag** drop-down list.



The Select Instrument dialog box displays the instrument items from the catalog.

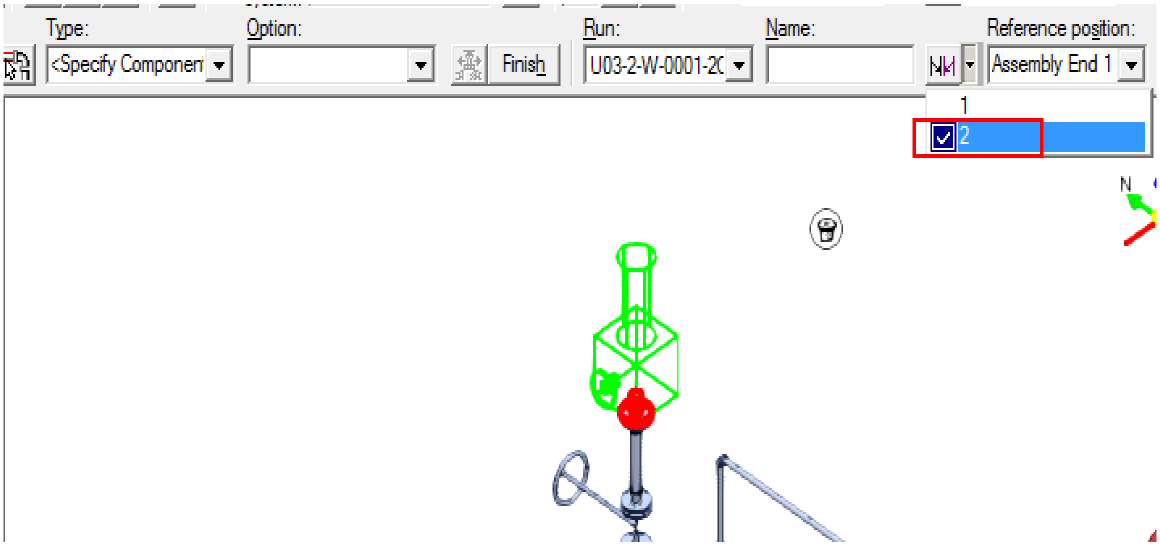
3. Expand **Custom Instrument > Relief Valve Type 3** and select the part **IRVT 34**. Click **OK**.



The valve appears in the graphic view.

4. Select Flip to change the port of the valve to 2.

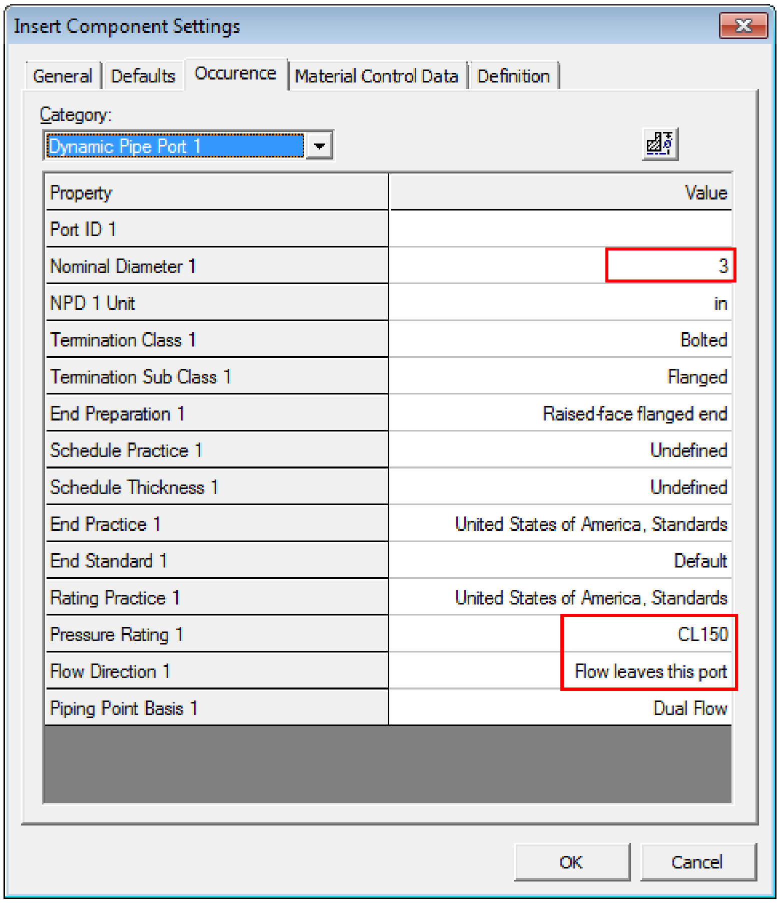
The valve in the graphic view appears.



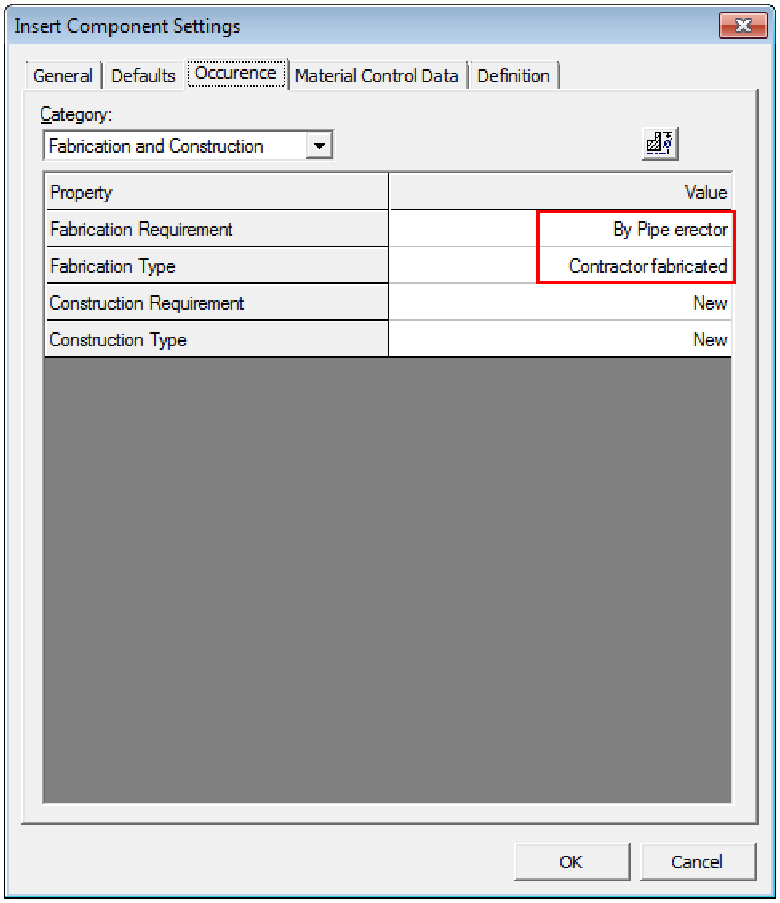
5. After you change the port of the valve to be placed, change the properties of the valve. Select **Properties** from the ribbon.

   The Insert Component Settings dialog box appears.

6. Switch to the **Dynamic Pipe Port 1** category on the **Occurrence** tab and change the **Nominal Diameter 1** to **3**, **Flow Direction 1** to **Flow leaves this port** and **Pressure Rating 1** to **CL150**.



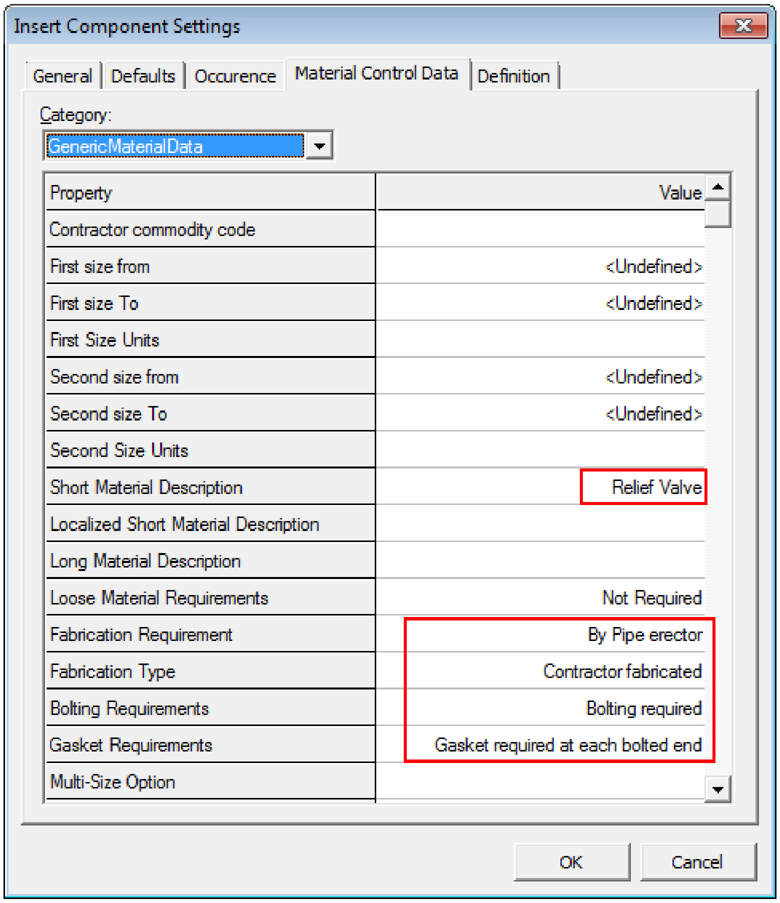
7. Switch to the **Fabrication and Construction** category on the **Occurrence** tab and set the **Fabrication Requirement** option to **By Pipe erector** and the **Fabrication Type** option to **Contractor fabricated** to specify the fabrication requirement for the valve.



8. Switch to the **Material Control Data** tab, and then select the **GenericMaterialData** category drop-down list.

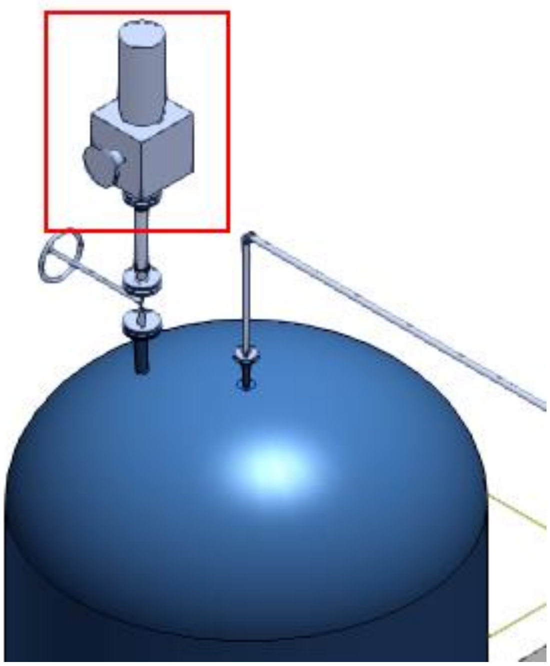
9. Change the following specifications, and then click **OK**.

Short Material Description: Relief Valve
Fabrication Requirement: By Pipe erector
Fabrication Type: Contractor fabricated
Bolting Requirements: Bolting required
Gasket Requirements: Gasket required at each bolted end



10. Click **Finish** to accept the placement of relief valve.

    The relief valve should resemble this.

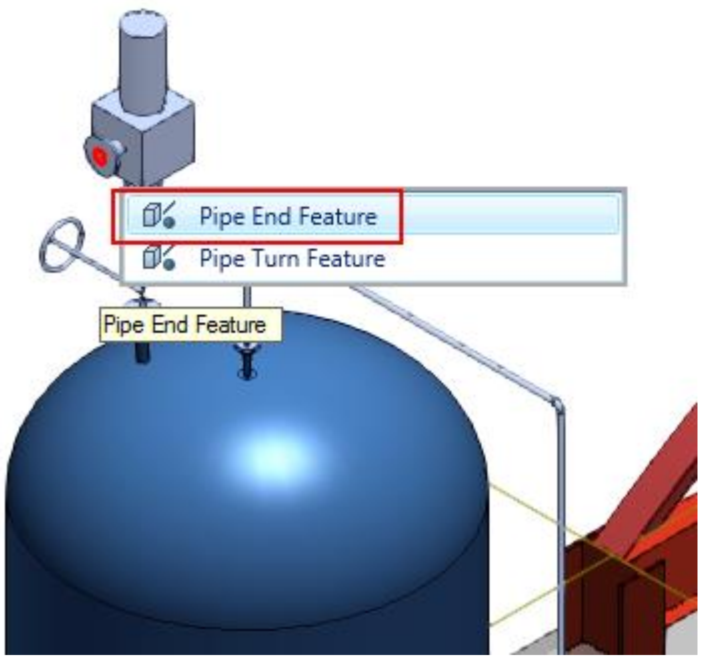


11. Right-click to terminate the **Insert Component** command.

Placing a Vent Pipe

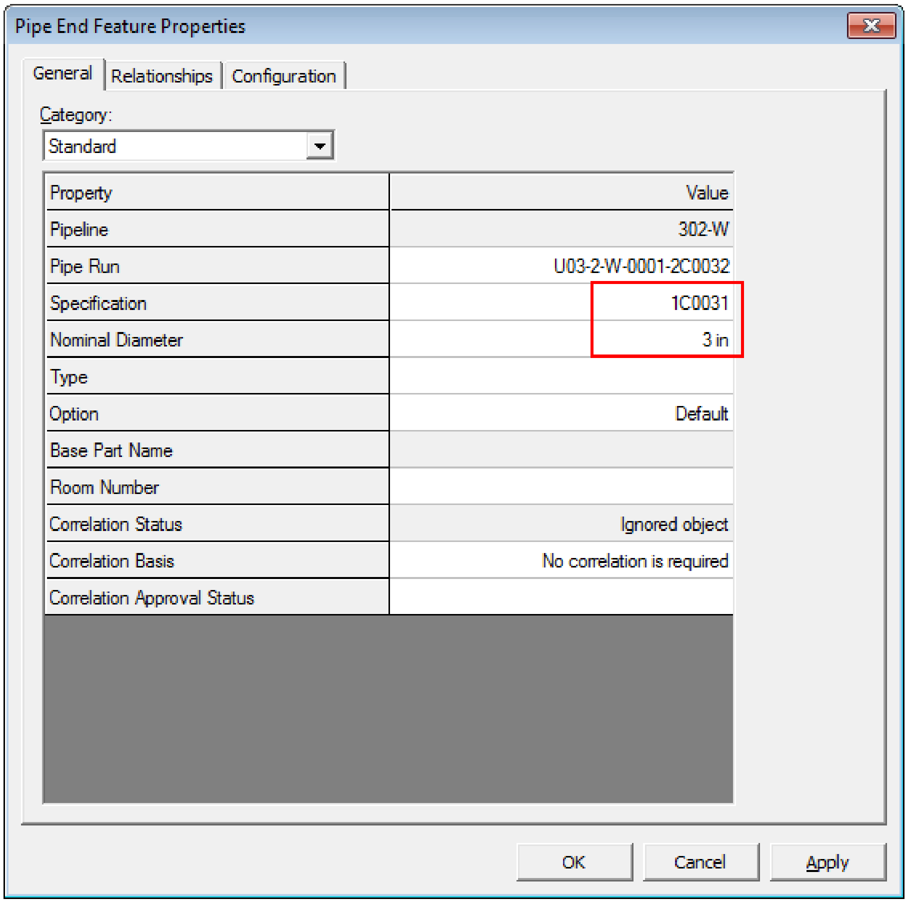
When the pressure setting in the vessel is exceeded, the relief valve is forced to open and a portion of the fluid is diverted through the other side of the line or vent pipe. In this exercise, you will place an elbow and then place a vent pipe on it.

1. Select the end feature of the valve and open the **Pipe End Feature Properties** dialog box to reduce the pressure and change the diameter.



2. Change the following specifications in the **Pipe End Feature Properties** dialog box, and then click **OK**.

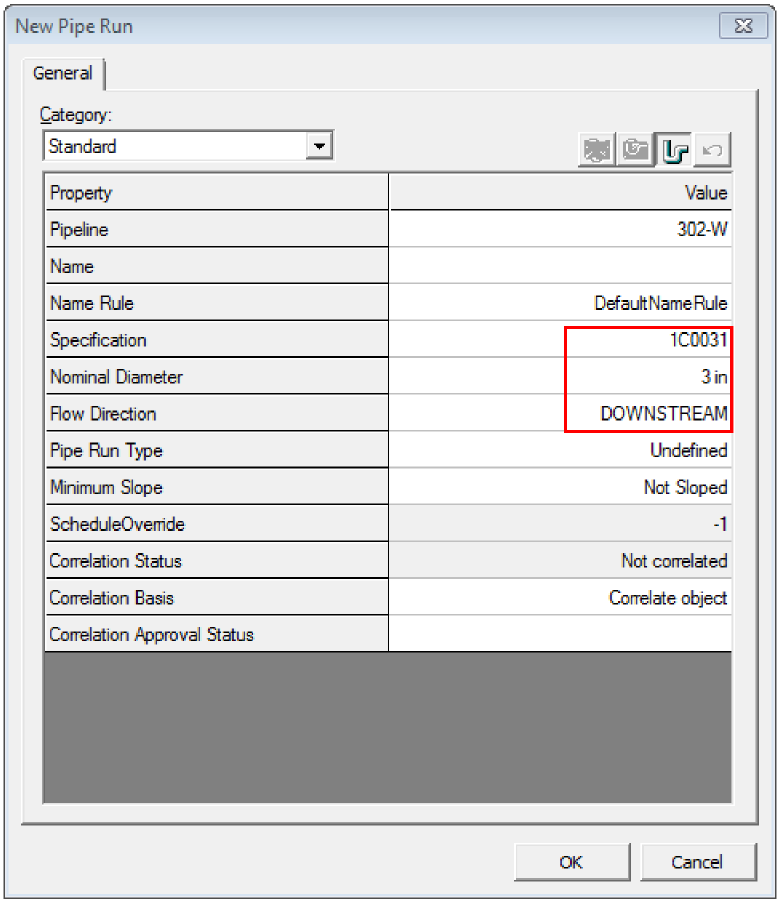
Specification: 1C0031
Nominal Diameter: 3 in



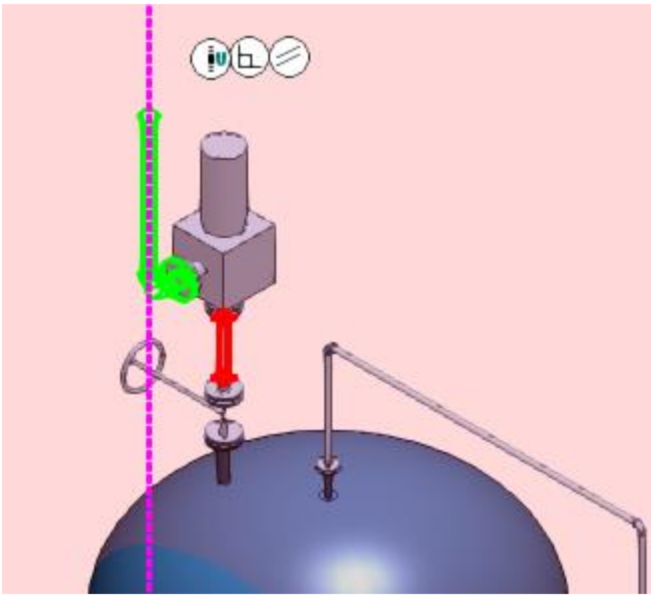
3. Select **Route Pipe** to begin routing from pipe end feature.
4. Select **New Pipe Run** option in the **Run** drop-down list to create a new pipe run.

5. Change the following specifications in the **New Pipe Run** dialog box, and click **OK**.

Pipeline: 302-W
Specification: 1C0031
Nominal Diameter: 3 in
Flow Direction: DOWNSTREAM

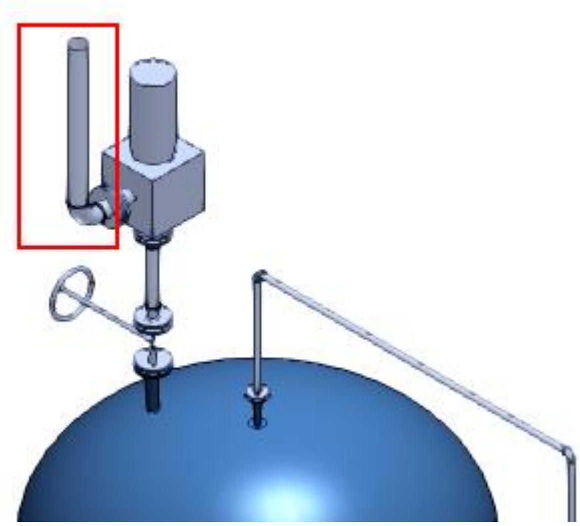


6. On the ribbon, lock the **Angle** at **90 deg** and lock **Length** at **3 ft**.



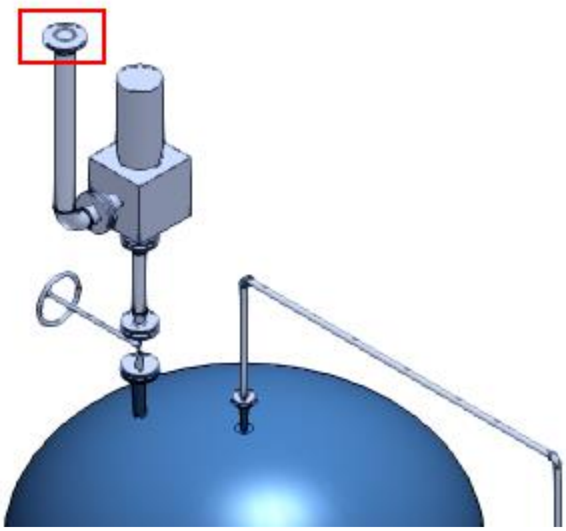
7. Click in the graphic view to place the pipe.

   Pipe run should resemble this.



8. Place a flange at the top of the pipe that you routed above using **Insert Component** .

The flange after placement will resemble this.



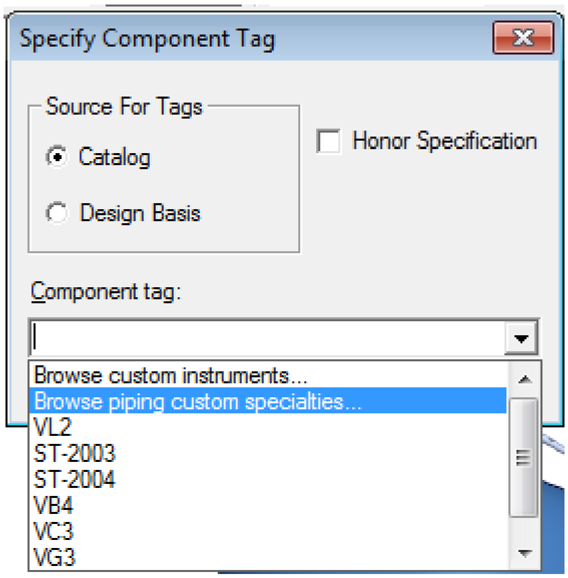
Placing a Silencer

After placing a vent pipe, now place a specialty item silencer on the vent pipe to reduce the noise when the fluid is released to the air.

1. To place the piping specialty item silencer on the flange, select **Specify Component Tag** from the **Type** drop-down list on the **Insert Component** ribbon.

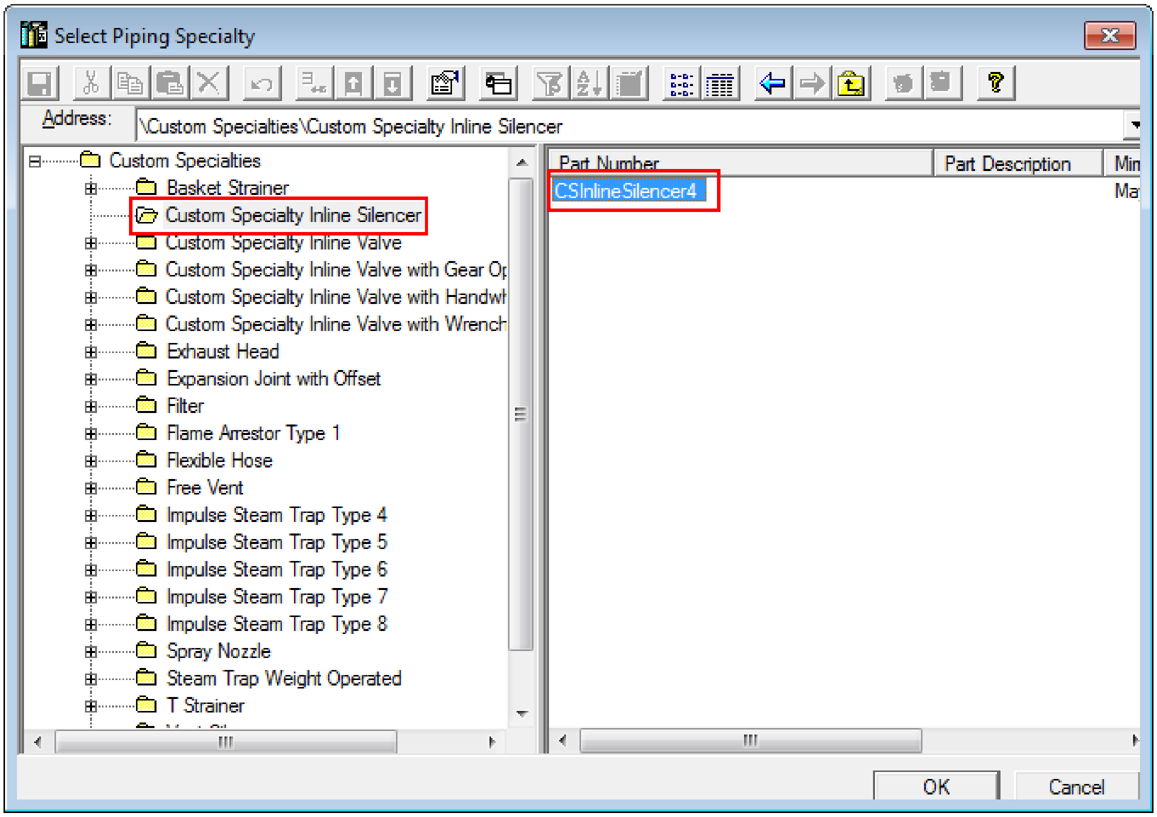
   The Specify Component Tag dialog box appears.

2. Select **Browse piping custom specialties…** from the **Component Tag** drop-down list.



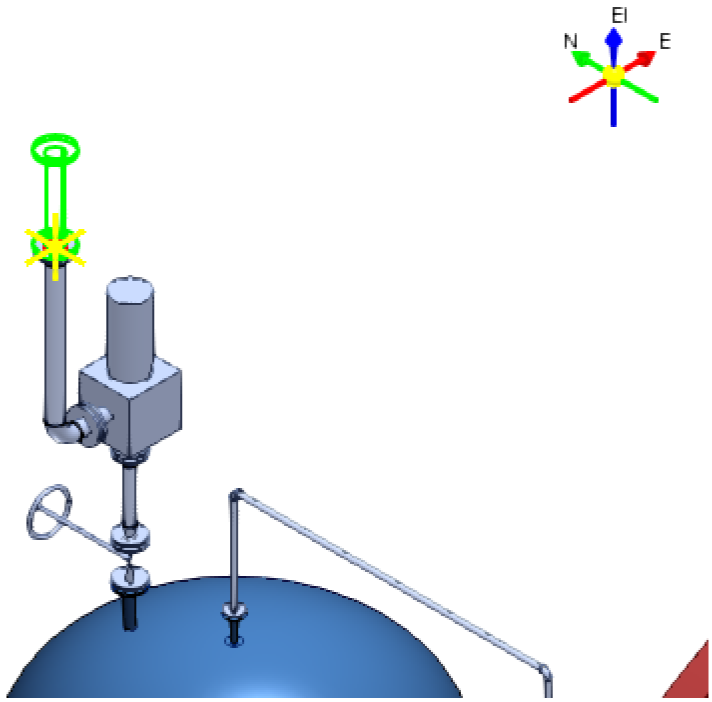
The Select Pipe Specialty dialog box displays the specialty items from the catalog.

3. Expand **Custom Specialties > Custom Specialty Inline Silencer**, and then select the part **CSInlineSilencer4**.



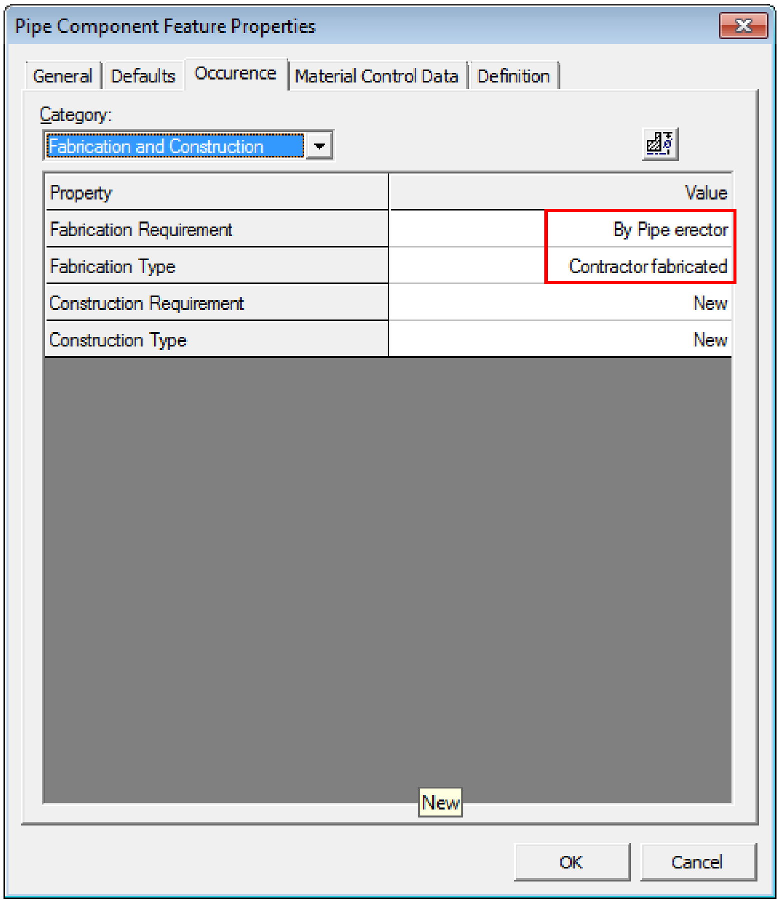
4. Click **OK**.

The specialty item silencer appears.



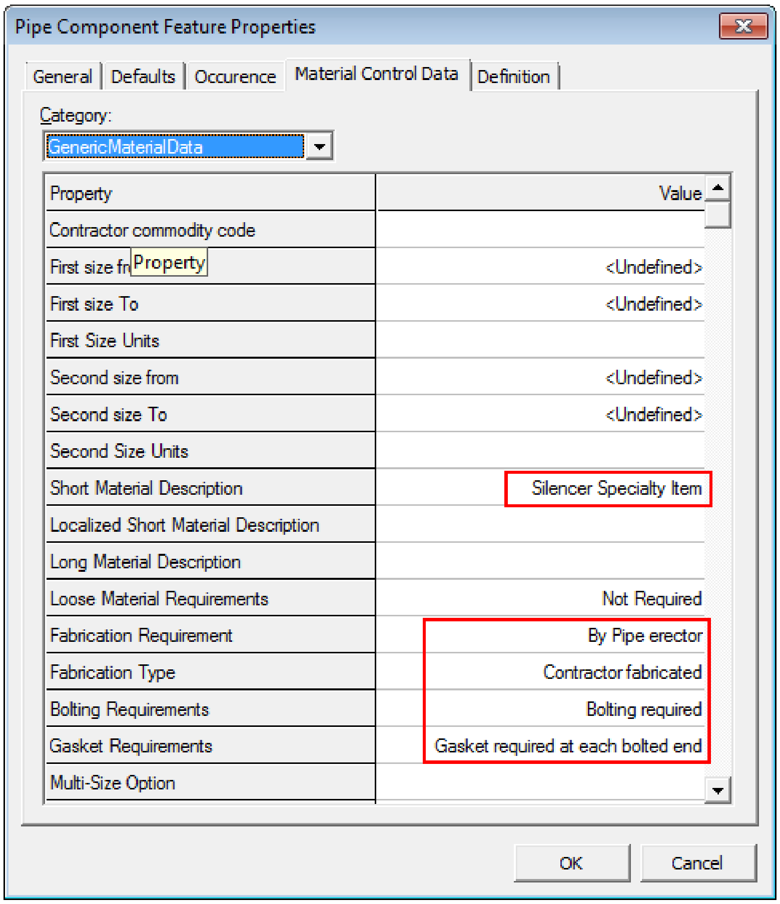
5. Now you will change the properties of the silencer so that the properties of the silencer are set with the flow direction of the fluid coming out of the tank. Click **Properties** to open the **Pipe Component Feature Properties** dialog box.
6. Select the **Fabrication and Construction** category on the **Occurrence** tab.
7. Select **By Pipe erector** from the **Fabrication Requirement** drop-down list.

8. Select **Contractor fabricated** from the **Fabrication Type** drop-down list.



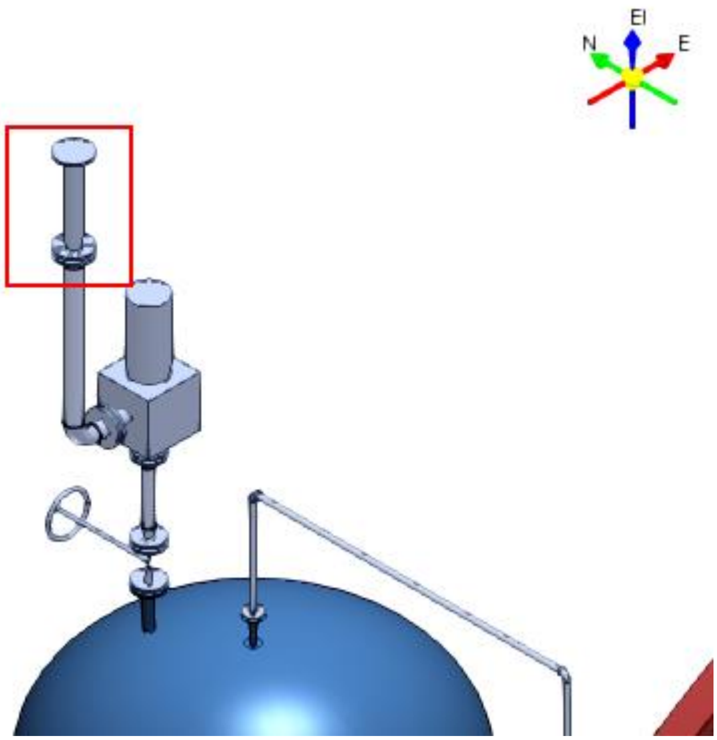
9. Select the **Material Control Data** tab, and then select the **GenericMaterialData** option under the **Category** drop-down list.
10. Change the following specifications, and then click **OK**.

Short Material Description: Silencer Specialty Item
Fabrication Requirement: By Pipe erector
Fabrication Type: Contractor fabricated
Bolting Requirements: Bolting required
Gasket Requirements: Gasket required at each bolted end



11. Click **Finish** to accept the placement of the silencer.
12. Right-click in the graphic view to terminate the command.

The silencer should resemble this.



For more information related to sequencing objects, refer to Insert a Piping Specialty Item topic in the user guide PipingUsersGuide.pdf.

Inserting Splits

Objective

By the end of this session, you will be able to:

- Insert splits in a pipe run.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

Overview

A split is a feature that represents a break or a joint in a pipeline. You can use any split feature defined in the reference data, such as, weld, clamp connection, flange set, union or sleeve to connect two pipes. **Insert Split** divides a pipeline into sections. You can select **Insert Split** while routing pipe or after routing pipe to insert a split in an existing pipe. There are two ways of inserting splits in a pipeline when using **Insert Split**:

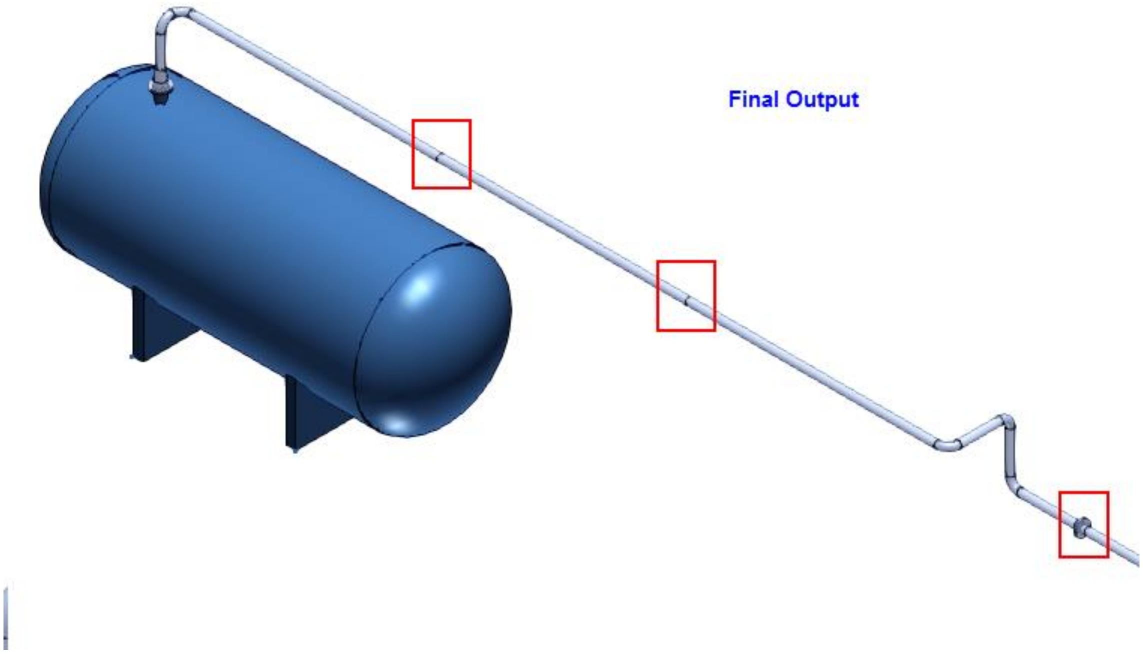
- You can cut the pipe feature into two features that represents two pipe parts joined by a weld joint or a takedown joint.
- You cut the pipe feature into two separate features in order to associate different property values to a single pipe part. By using a feature break, you can stop heat-tracing, insulation, or a surface coating at an arbitrary location along the pipe instead of at a weld or other joint in the line.

This session will cover the procedures for inserting splits by using weld joint, takedown joint, and feature break.

Inserting Weld and Takedown Joints

Objective

In this exercise you will be inserting a takedown joint and weld joints in a pipe run **400-P** in **Unit U04** of your workspace. After inserting the joints the view of the pipe run should resemble this.



Before Starting this Procedure

- Define your workspace to display **Unit U04** and coordinate system **U04 CS**. In your training plant, select **U04** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Select **Tools > PinPoint** to activate the **PinPoint** ribbon.
2. Set the active coordinate system to **U04 CS** and select the **Set Target to Origin** option on the **PinPoint** ribbon.
3. Click **Insert Split** to insert a takedown joint.

   The Insert Split ribbon appears to set the options for adding a break into a pipe run.

   The options on the **Insert Split** ribbon are:

   Pipe Split Feature Properties - Opens the **Pipe Split Feature Properties** dialog box in which you can define additional properties that are not available on the ribbon bar.

   Select Feature - Selects the pipe in which you want to insert a split.

   Enter Insertion Point - Defines the location to insert the split.

   Run - Displays the name of the pipe run to which the split belongs along with the names of all other runs associated with the same pipeline parent. You can select another pipe run if needed. You can select **More** to select a pipe run that is associated with a different pipeline parent.

   Weld Joint - Select this option when a welded joint is necessary.

Takedown Joint - Select this option when a takedown joint is necessary.

Feature Break - Select this option to place a feature break. Feature breaks are used to stop heat-tracing, insulation, and coatings at arbitrary locations along a pipe without having to break the pipe into two pipe parts.

Type - Select the type of split component to place. The software generates this list based on the components allowed by the pipe specification for the selected pipe. You can also specify the component that you want to use by selecting **Specify Component Tag** and then entering the component tag. See *Pipe Specifications* in *Piping Reference Data Guide* for more information on defining pipe specifications.

Option - Displays the option of the selected split component. In addition, the **Option** list contains any options defined in the pipe specification for the short code selected in the **Type** list. Select another value from the **Option** list to update the object.

Finish - Places the splits using the parameters that you have specified.

Split Mode - Specifies whether to place a single split or multiple splits. Select **Single Split** to place a single split or feature break at a specified location. Select **Multi Split** to place splits on straight features at a specified distance apart. You can select a single straight feature or an entire run in this mode. If you select an entire run, the splits are placed at the specified distance apart on each individual straight feature in the run, but not across the entire run itself. The **Multi Split** option will not place splits on bent pipe. Splits placed using the **Multi Split** option have no relation to each other after placement. Feature breaks cannot be placed in the **Multi Split** mode.

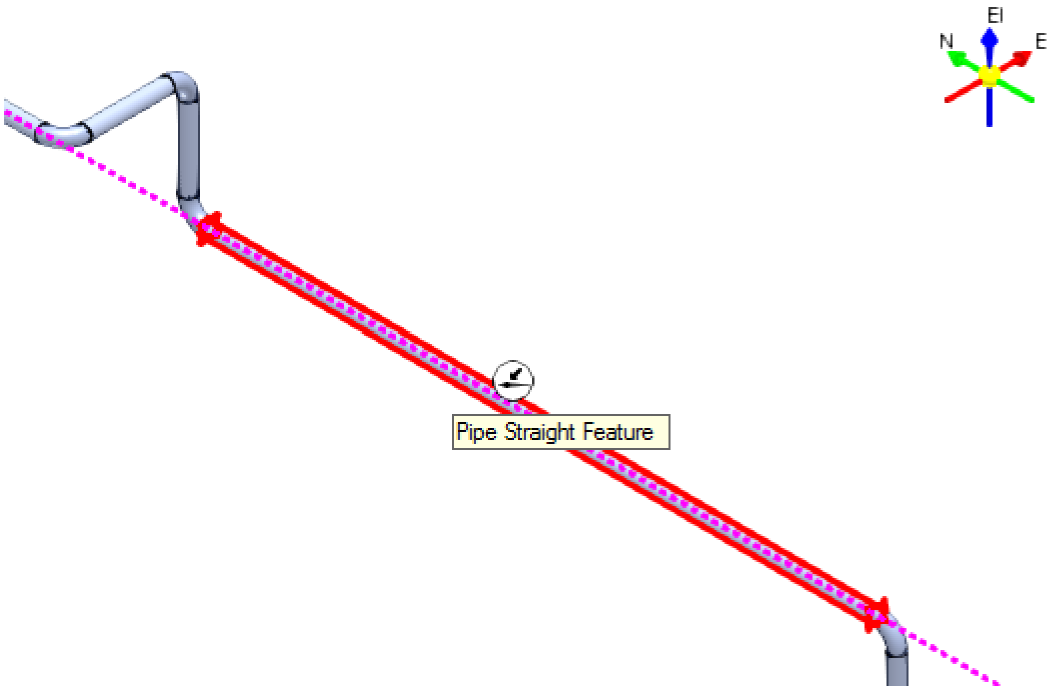
Pipe Length - Enter the distance between the splits. The default distance is the purchased pipe length defined in the piping specification. This option is only available when **Split Mode** is set to **Multi Split**.

Reference - Select **Start** to measure the distance from the start end (the first end that was placed) of the pipe. Select **End** to measure from the end of the pipe. This option is only available when **Split Mode** is set to **Multi Split**.

4. Select **Takedown Joint** to place takedown split in a pipe run.

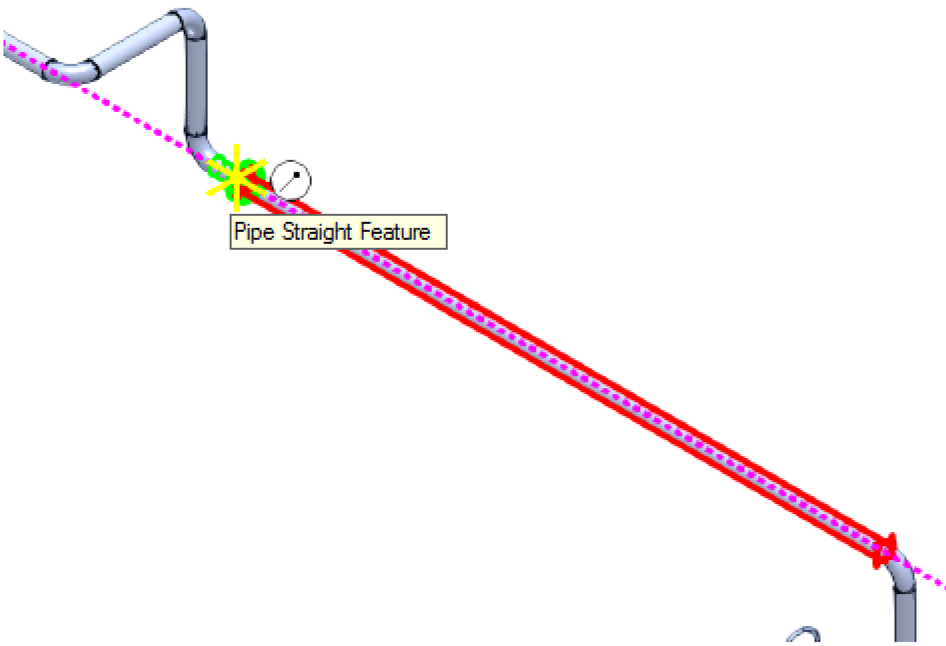
   The system prompts to select either a Pipe Straight feature.

5. Select the pipe run **U04-4-P-0203-1C0031** to specify the pipe where you want to insert the takedown joint.

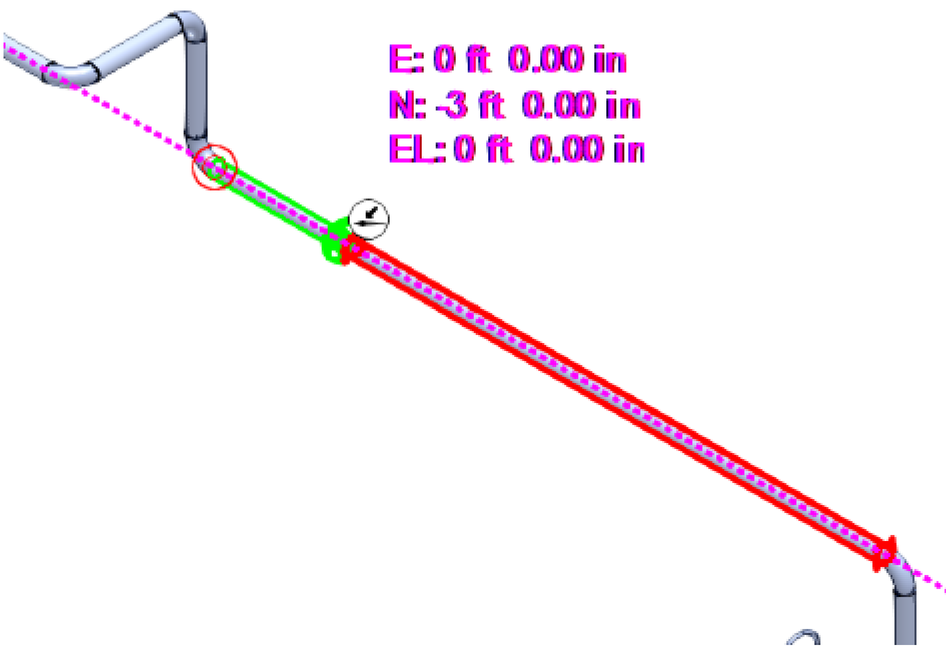


6. Select **Enter Insertion Point** on the ribbon.
7. Select **Reposition Target**  on the **PinPoint** ribbon to change the target origin.
8. Select the end point of the Pipe Straight Feature to specify the target origin.

The position of takedown joint will be specified from this target origin.

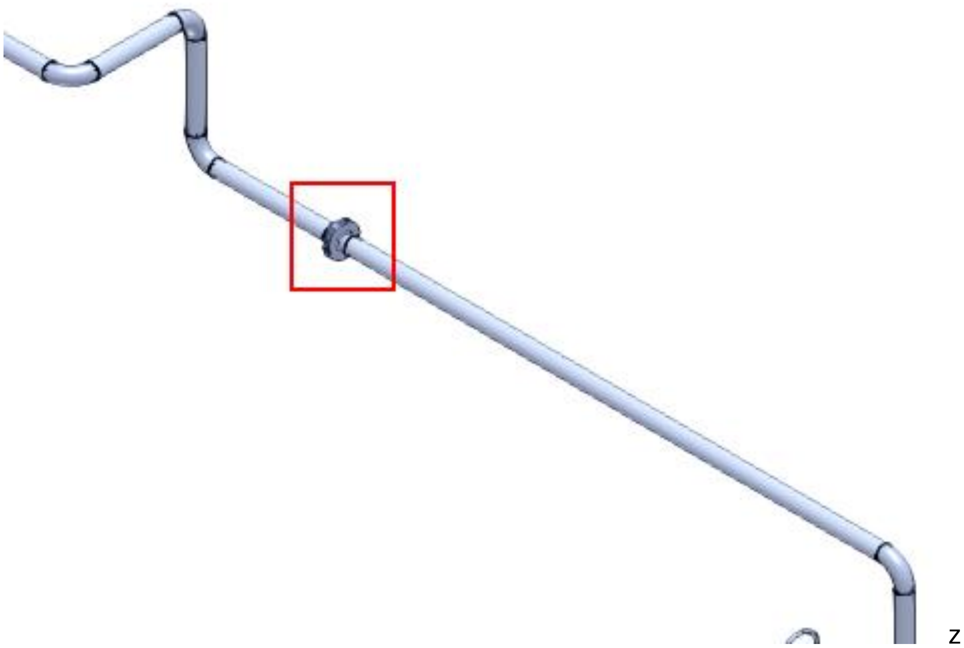


9. Key in **-3 ft** for **N** on the **PinPoint** ribbon to specify the position of takedown joint on the pipeline.

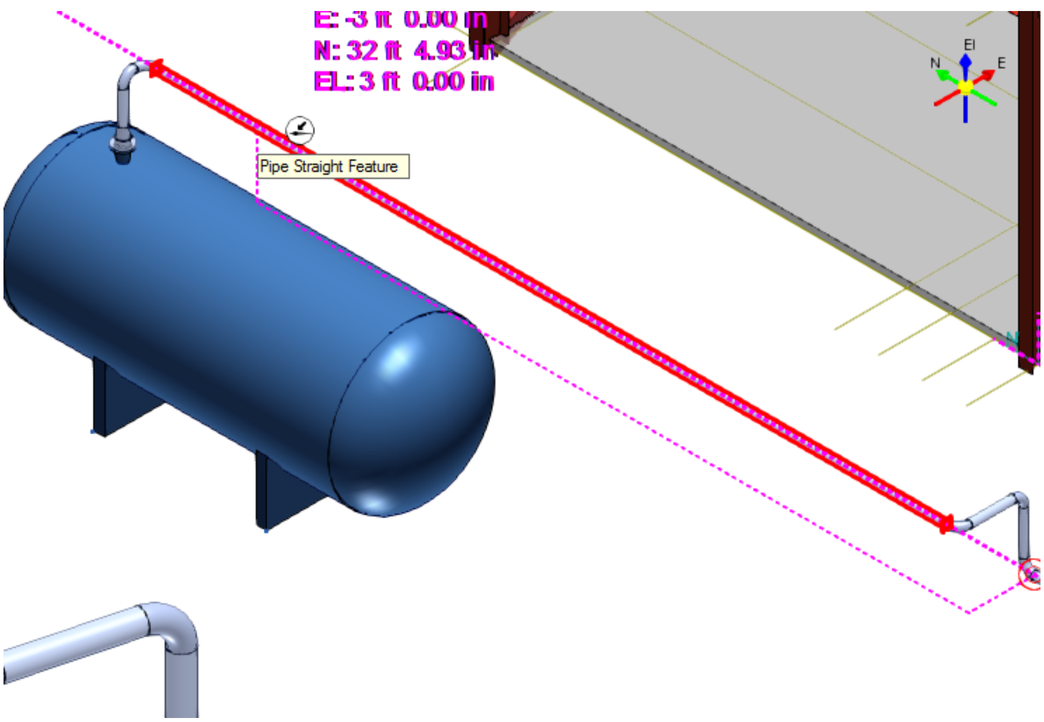


10. Click in the graphic view to accept the position of a split on the pipe run.
11. Click **Finish** to complete the placement

    The takedown joint (flange set) placed in a pipe run.

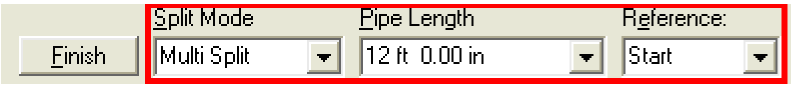


12. Click **Insert Split** to insert weld joints.
13. Select **Weld Joint** on the **Insert Split** ribbon to place welds in a pipe run.
14. Select **Pipe Straight Feature** of a pipe run to specify the pipe where you want to insert splits.



15. Specify the following values on the Insert Split ribbon to place multiple splits on the straight feature.

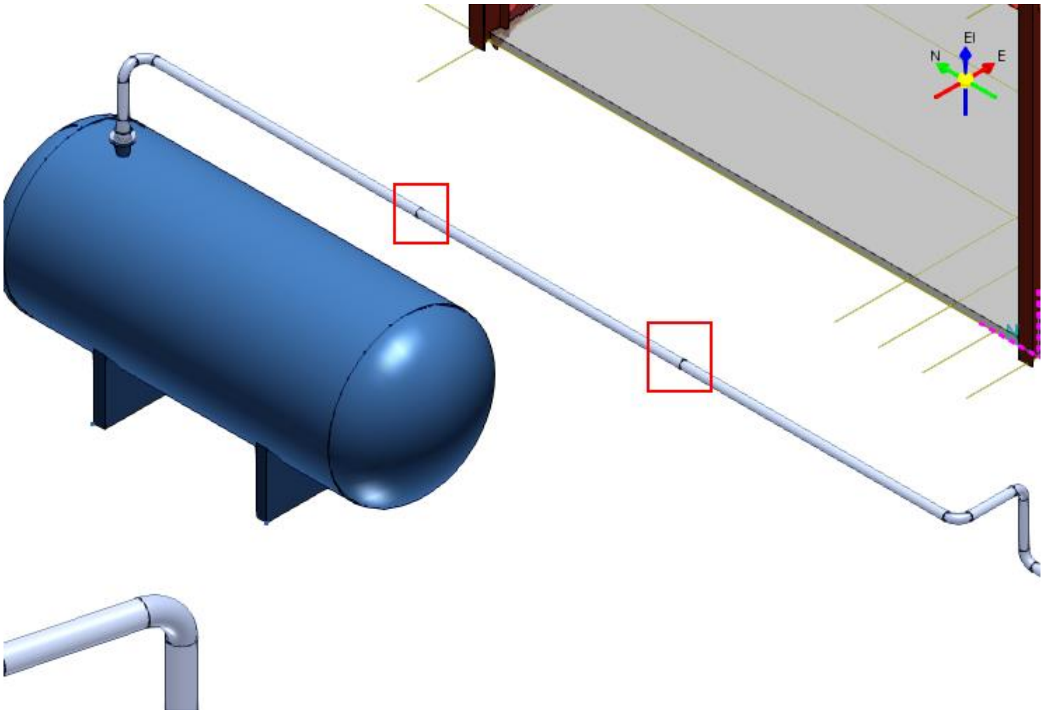
    Split Mode: Multi Split
    Pipe Length: 12 ft
    Reference: Start

    Smart 3D places splits at every 12 ft distance from the start end of the pipe.

16. Click **Finish** to accept the placement of weld joints.

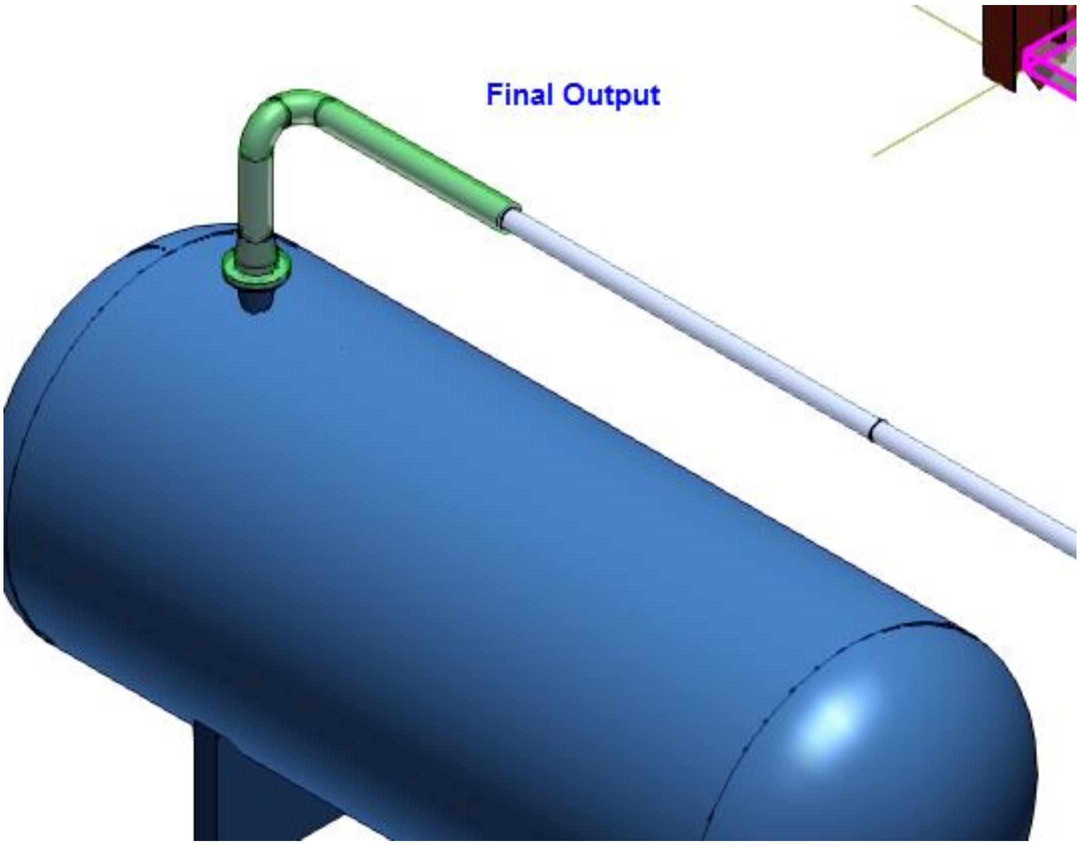
    Smart 3D places multiple weld joints in the pipe run.

Inserting a Feature Break Split

Objective

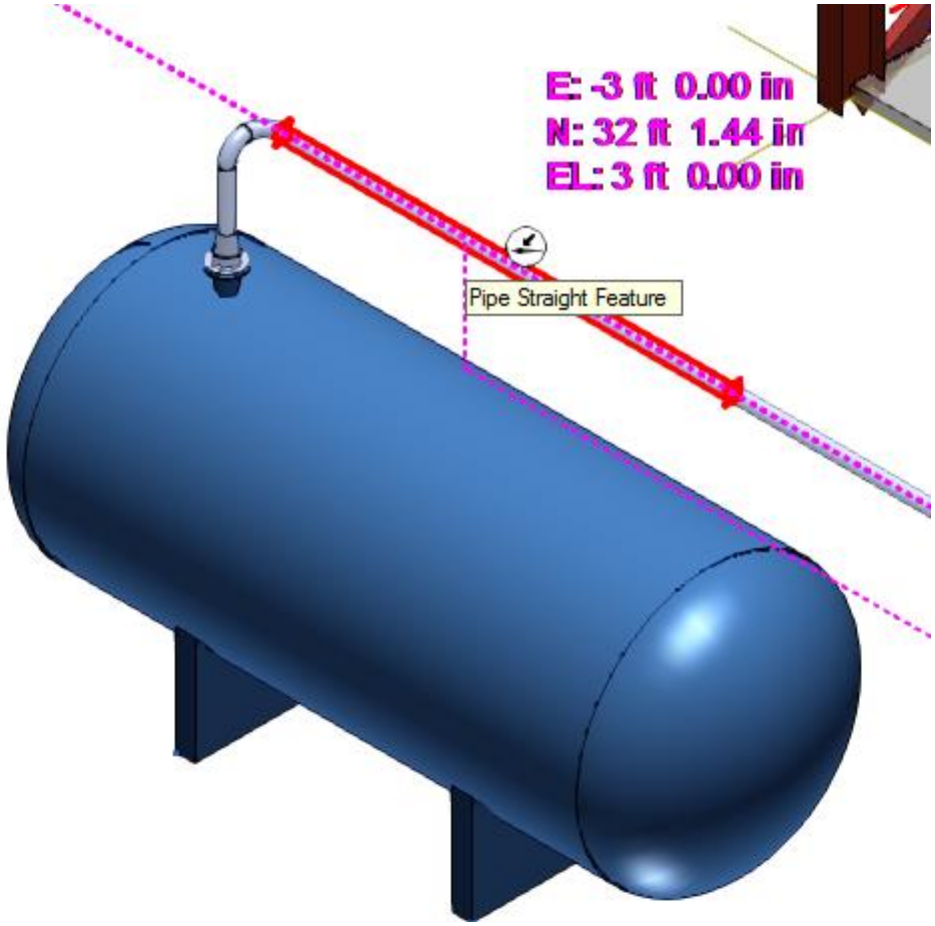
In this exercise you will be inserting a feature break split in a pipe run **400-P** in **Unit U04** of your workspace and insulate the section of the pipe run where feature break is inserted. After inserting the feature break and insulating it, the view of the pipe run should resemble this.



1. Click **Insert Split** .
2. Select **Feature Break** on the ribbon to place feature break split in a pipe run.

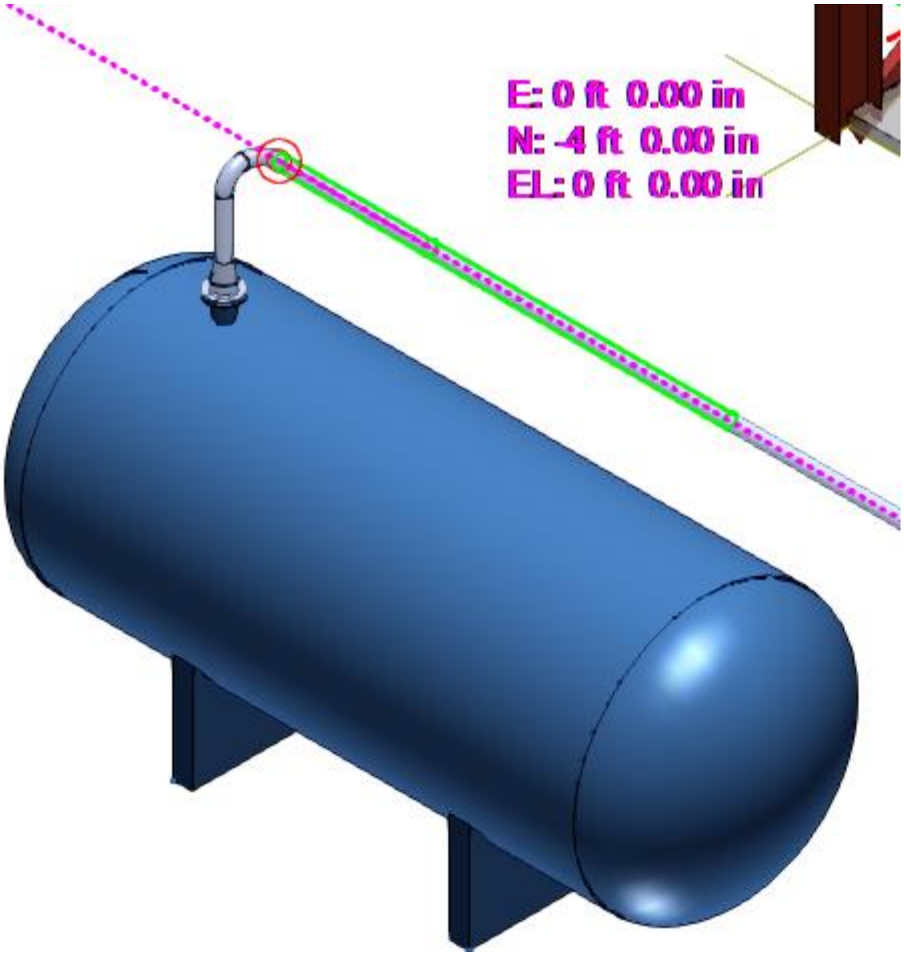
   TIP If the **Feature Break** option is not available, then it means that the last used value for the **Split Mode** was set to **Multi Split**. Select the pipe run so that Smart 3D opens the split ribbon bar. Now you can change the **Split Mode** back to **Single Split** to enable the feature break option.

3. Select the **Pipe Straight Feature** of pipeline **400-P**.

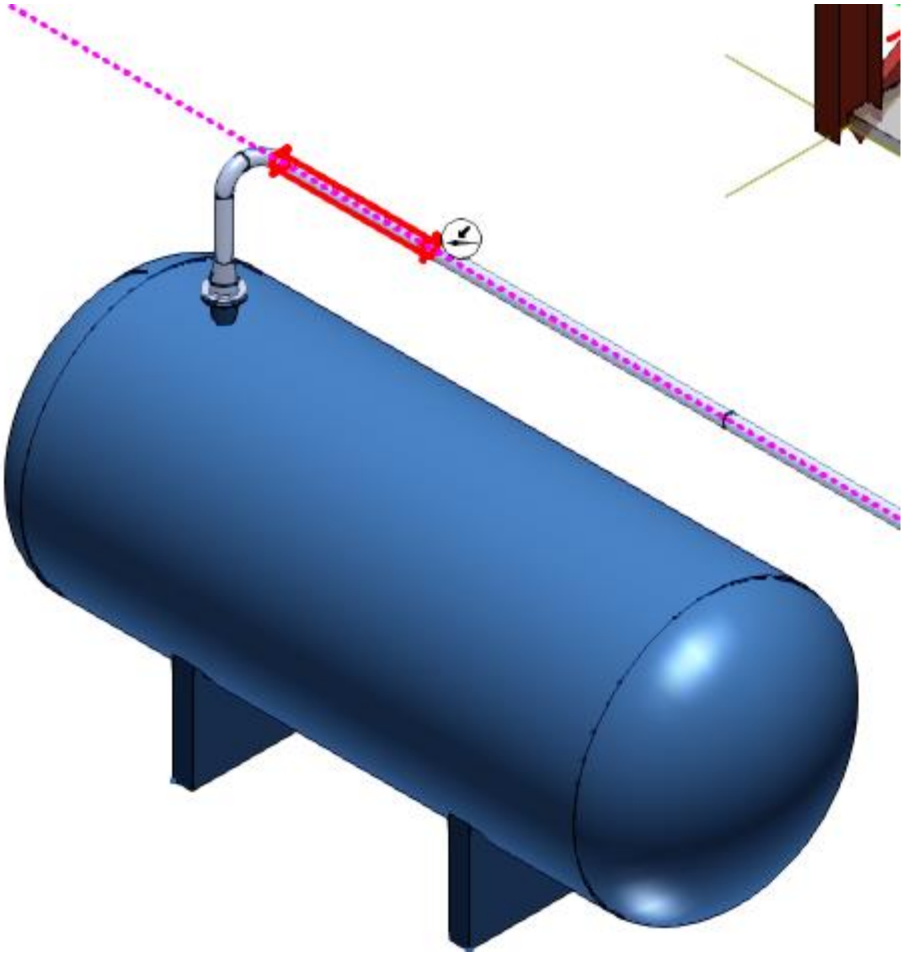


4. Click **Enter Insertion Point**
5. Select **Reposition Target** on the **PinPoint** ribbon.
6. Select the end point of the **Pipe Straight Feature** of the pipeline **400-P** to specify the target origin.

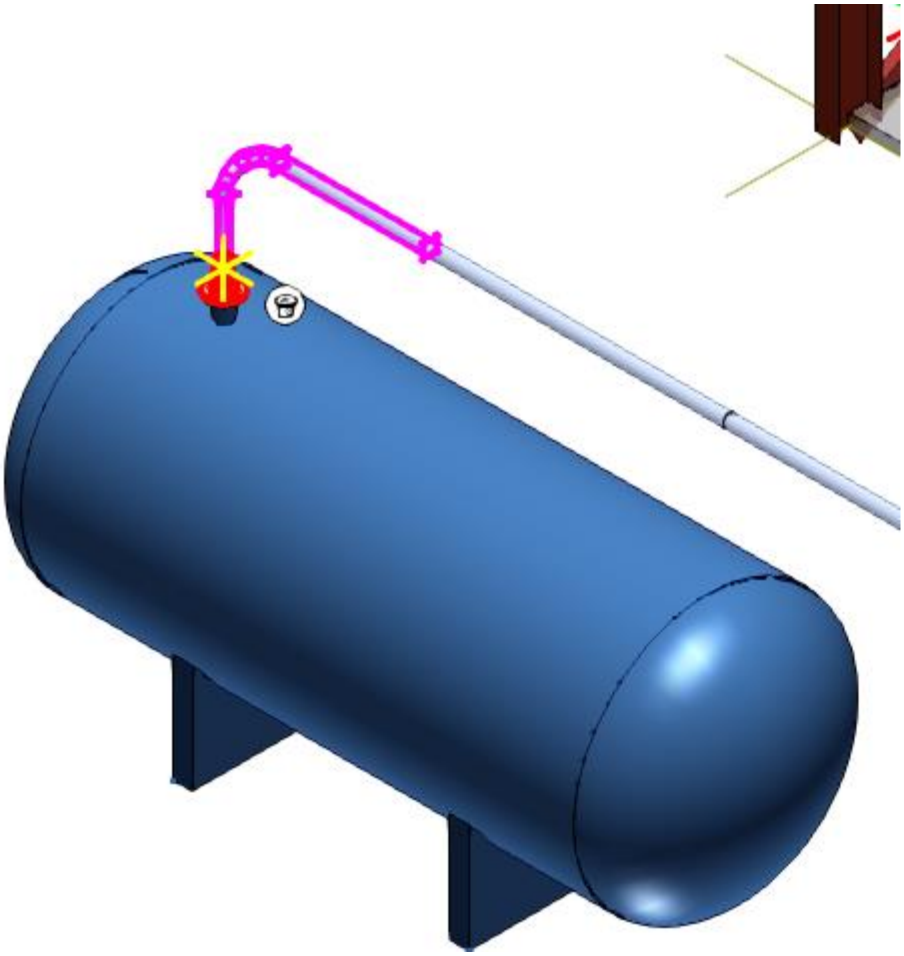
7. Key in **-4 ft** for **N** on **PinPoint** ribbon to specify the position of feature break.



8. Click on the graphic view to accept the placement of feature break. Click **Finish** to complete split placement. The placement of this split will not be visible in graphic view until you select the pipe run or the adjacent features.

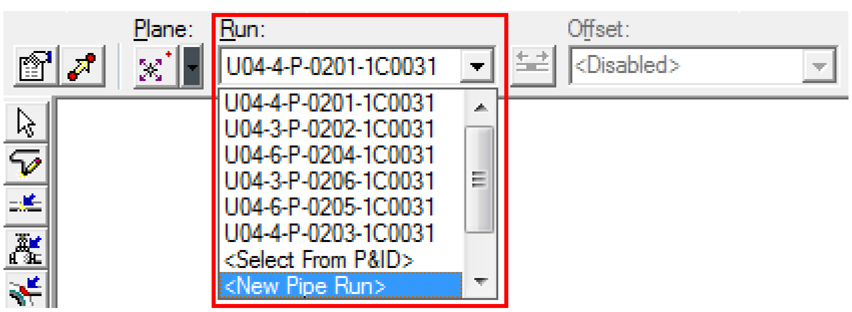


9. Select all straight and turn features between the equipment nozzle and the feature break by holding the SHIFT key as you select them.



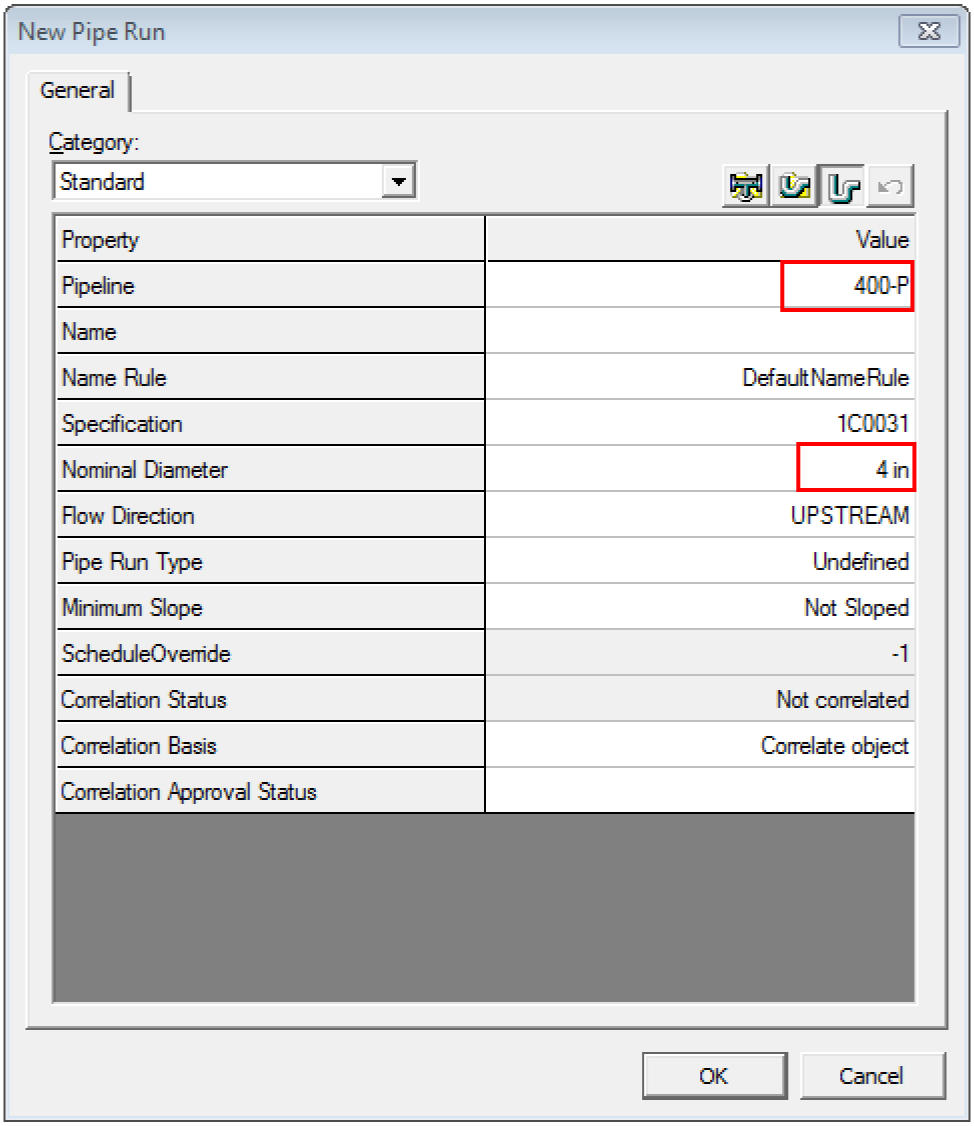
The Edit ribbon will appear.

10. Select the **New Pipe Run** option from the **Run** drop-down list on the **Edit** ribbon.



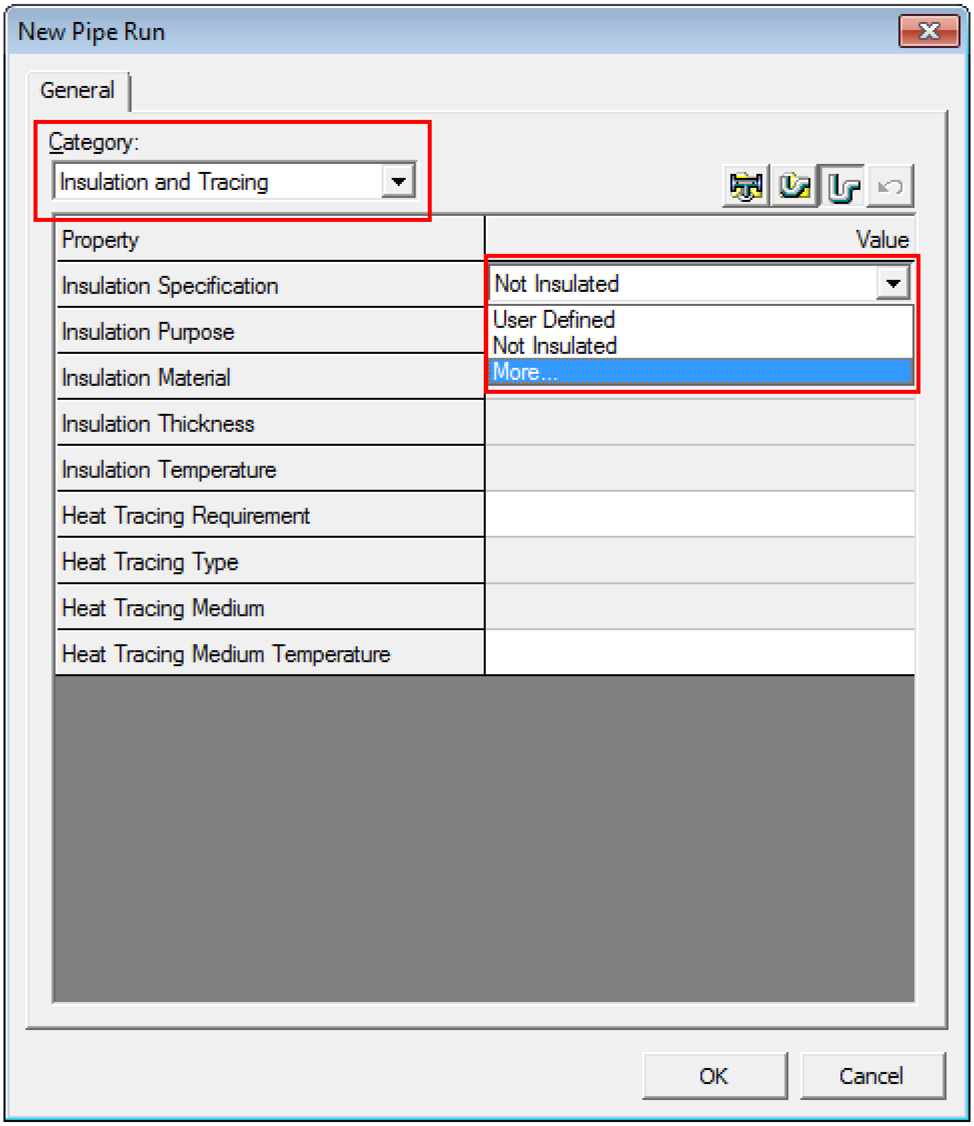
The New Pipe Run dialog box opens.

11. Select **400-P** for Pipeline and **4 in** for **Nominal Diameter** under the **Standard** category.



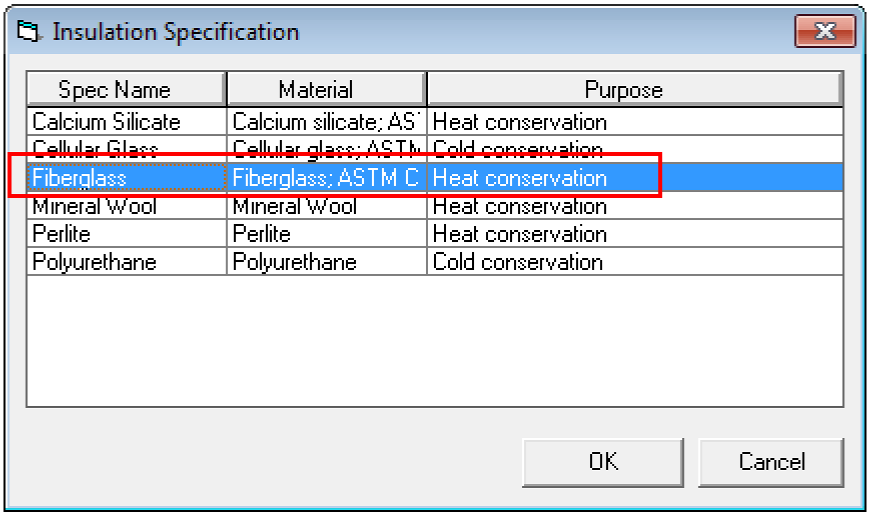
12. Select the **Insulation and Tracing** category.

13. Select **More…** in the **Insulation Specification** drop-down list to specify the insulation type for the pipe run.



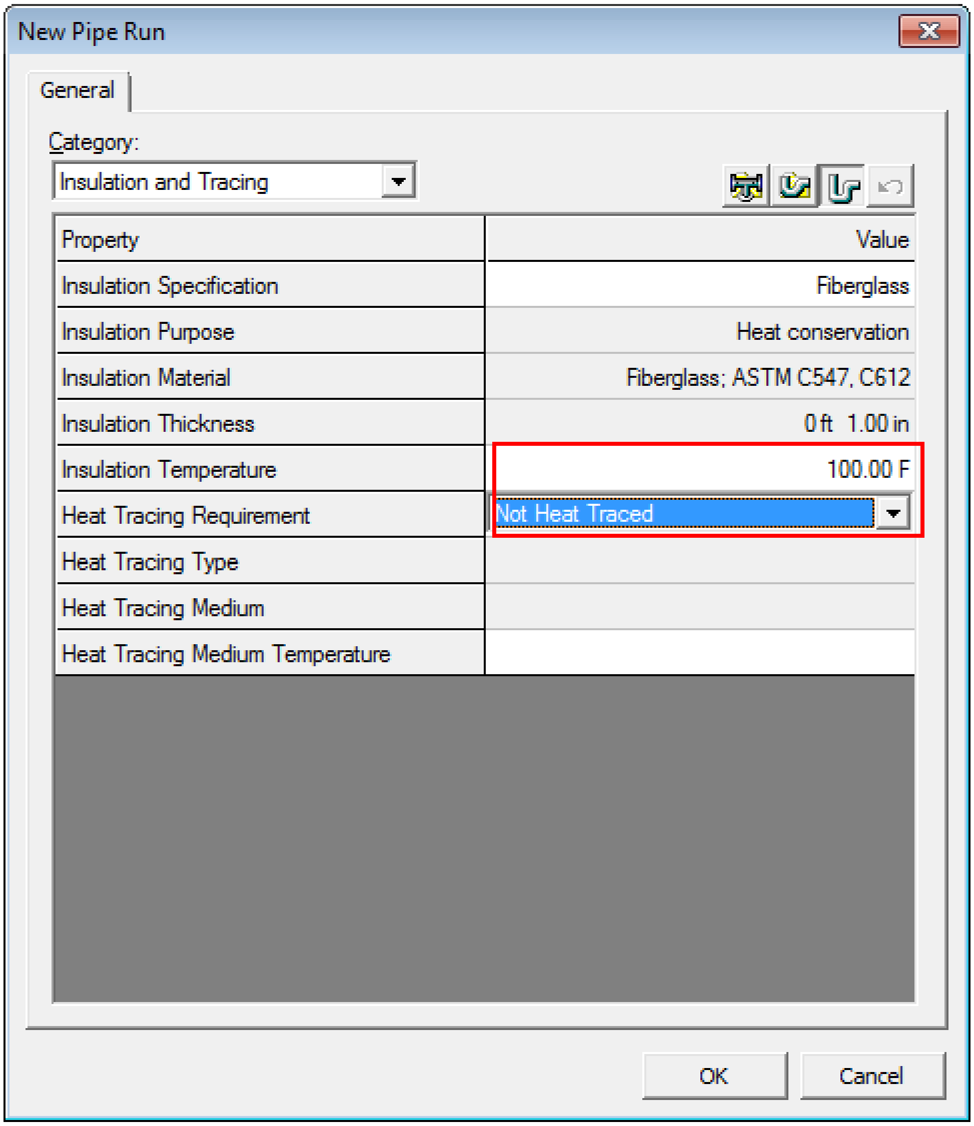
The Insulation Specification dialog box appears to select the type of insulation for the pipe run.

14. Select **Fiberglass**, and click **OK**.



15. Set the following specifications on the **New Pipe Run** dialog box under the **Insulation and Tracing** category:

Insulation Temperature: 100 F
Heat Tracing Requirement: Not Heat Traced



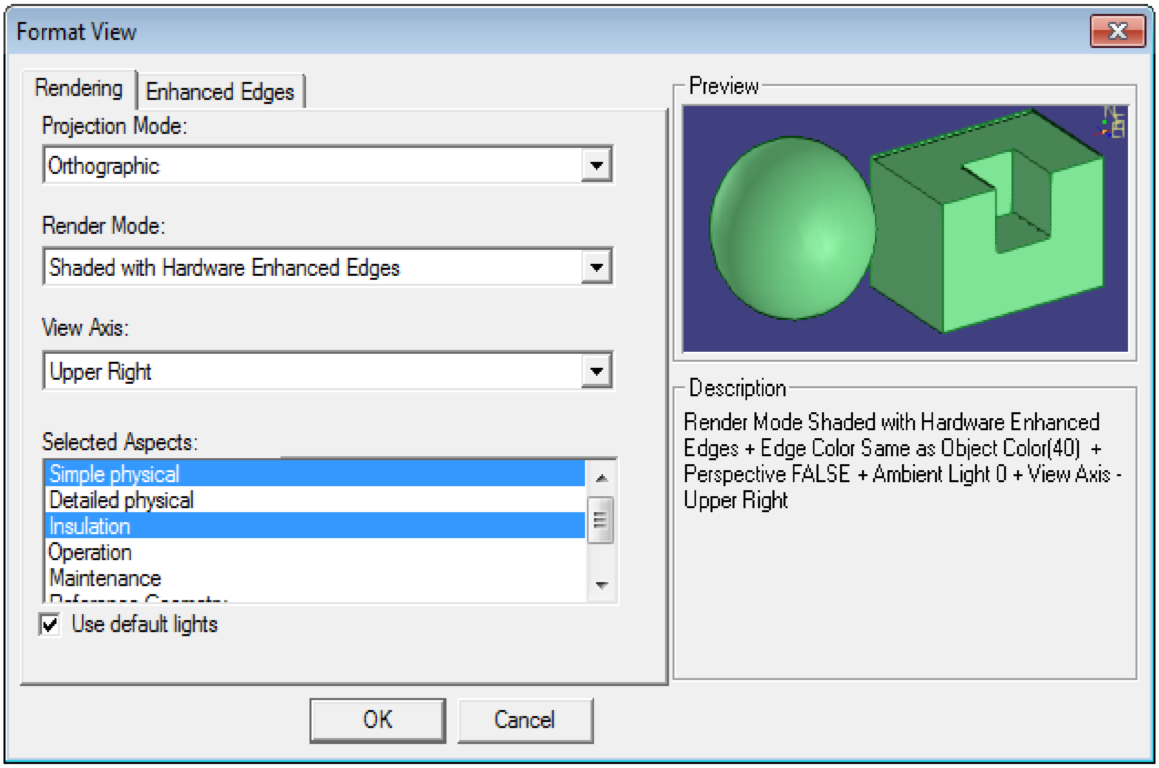
16. Click **OK**.

    Insulation is applied on the selected section of pipe run.

17. To see the insulated section change, select **Format > View**.

    The Format View dialog appears.

18. Select the **Insulation and Simple physical** in **Selected Aspects** in the **Format View** dialog box.



19. Click **OK**.
20. Now apply surface style rule on the insulated section using **Format > Surface Style Rules**. Select the **Piping Insulation – Delivered** rule, and click **OK** to apply the rule to the insulated section.

For more information related to sequencing objects, refer to **Insert Split** in the user guide PipingUsersGuide.pdf

SESSION 9

Placing Taps

Objective

By the end of this session, you will be able to:

- Place a tap on pipe parts.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

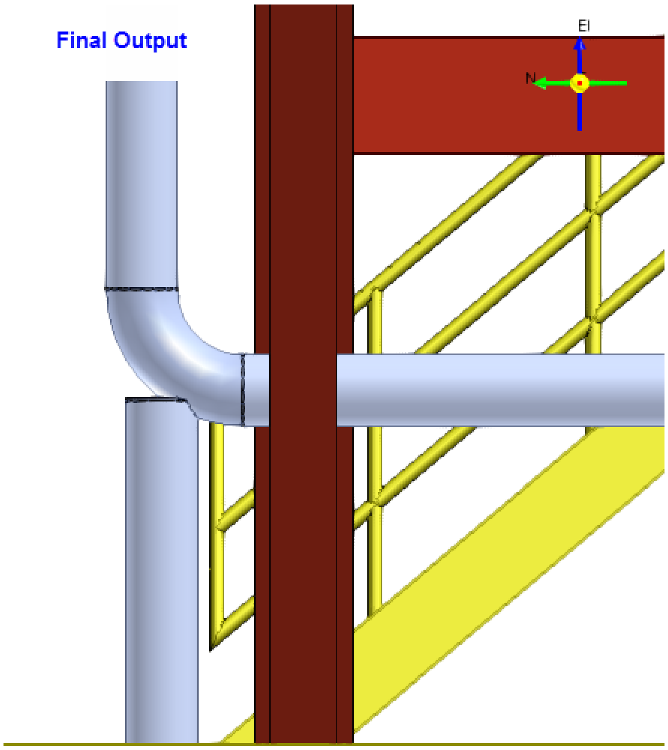
Overview

In Smart 3D taps are placed on pipe or piping components. You use **Insert Tap** to insert taps for venting, drainage, and for instrument component connections. The **Insert Tap** command inserts tap on pipe parts after the pipe part is placed in the model. You can place taps only on pipe parts referred as base part. A base part is a pipe part generated by a feature inserted by the user such as elbows, tees, caps, valves, pipes, and pipe bends. However, you cannot insert taps on components having mating parts. If you need to insert tap on a mating part, you need to change it to a base part using the **Edit Part** ribbon.

Inserting a Tap for a Pipe Trunnion

Objective

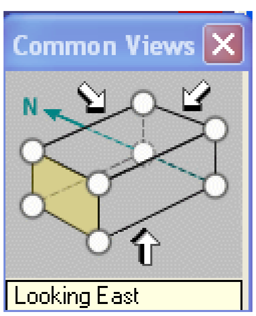
In this exercise you will be inserting a tap on the elbow 90 Degree Direction Change-0034 of the pipeline **300-W** using the **Insert Tap** command and then route a pipe from the end point of the placed tap to support the pipeline **300-W**. The pipe trunnion in the graphic view should resemble the highlighted section below.



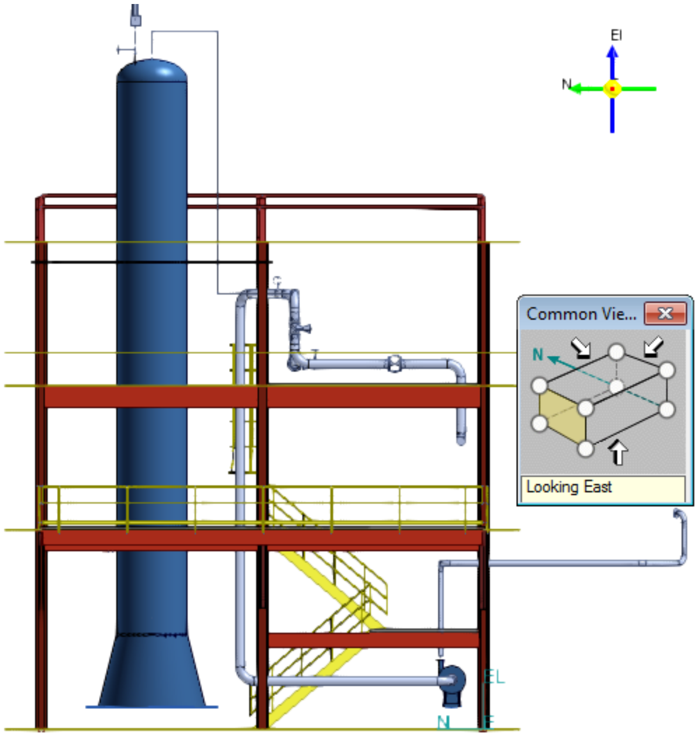
Before Starting this Procedure

- Define your workspace to display **Unit U03** and coordinate system **U03 CS**.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Change the model view to **Looking East** using the **Common Views** dialog box.

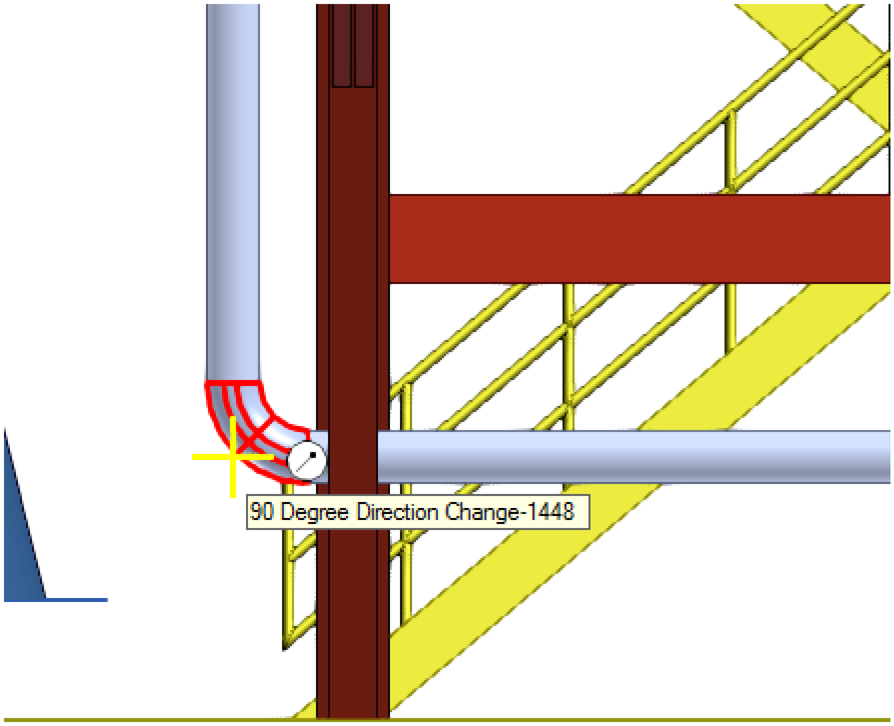


2. Locate the elbow 90 Degree Direction (Elbow) in the graphic view.

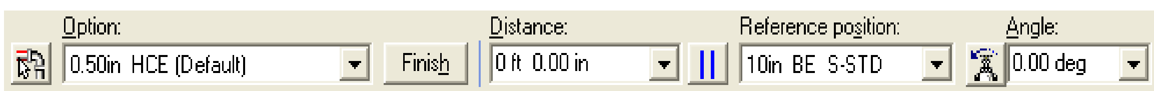


1. Click **Insert Tap**  on the vertical toolbar.

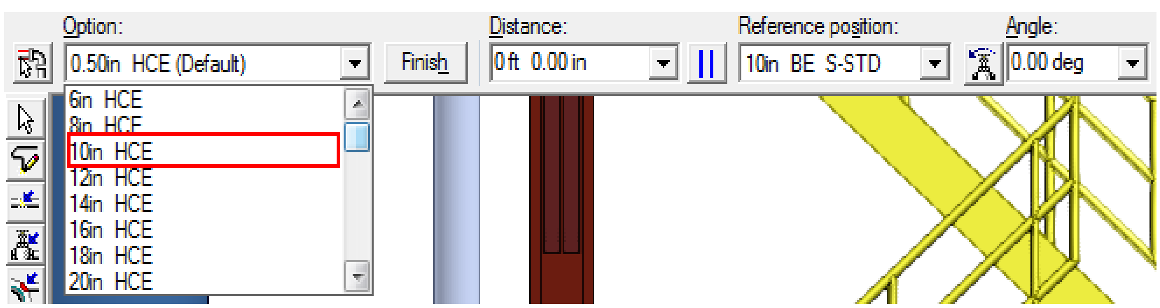
2. Select the elbow 90 Degree Direction Change (Elbow) (as shown below) of pipeline 300-W in the graphic view. This is the portion of the pipeline where you will place the tap.



When you select the elbow, the Insert Tap ribbon appears.



3. Select **10in HCE** in the **Option** drop-down list to specify a 10 inch Hole Circular End tap to be placed in the selected elbow. Smart 3D generates this list based on the pipe specification of the component into which you want to insert the tap.

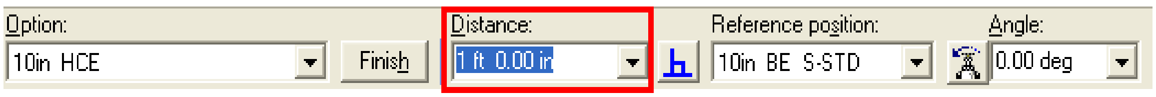


4. Set the **Orientation** option to perpendicular. This will measure the distance from the reference port to the tap location along the arc of the turn feature.

TIPS

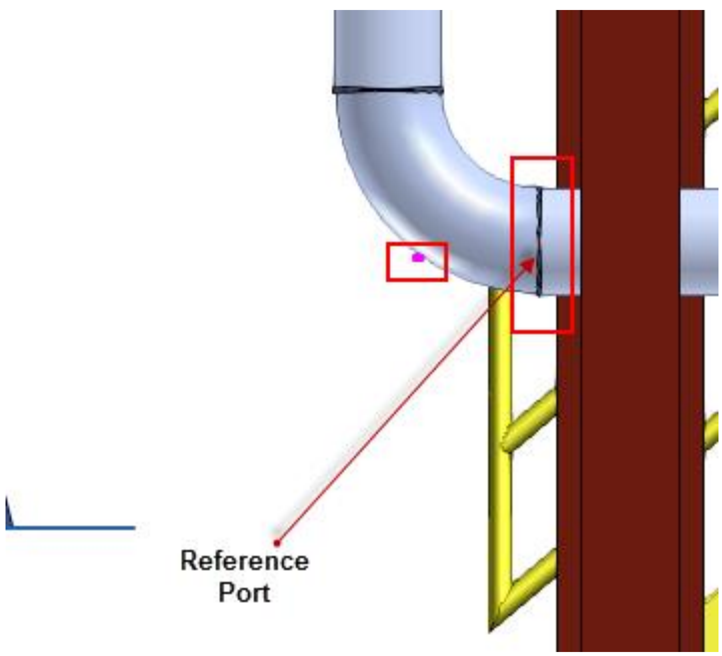
- If the orientation is set to parallel, the distance is measured from the specified reference point and at the specified angle.
- Orientation option is available only when you select an end feature such as cap or blind flange or a turn feature such as an elbow or pipe bend. For straight pipes and tubes, and the straight sections of a pipe bend, the orientation is always set to perpendicular. For pipe bends, the tap is always placed perpendicular to the flow direction throughout the turn.

5. Key in **1 ft** in the **Distance** drop-down list to reposition the tap point from port1 of the elbow. The distance is measured along the component between the tap and the specified reference position on the component.



6. Click **Finish** to accept the placement of the tap.

The tap placed on the elbow should resemble the highlighted section below.

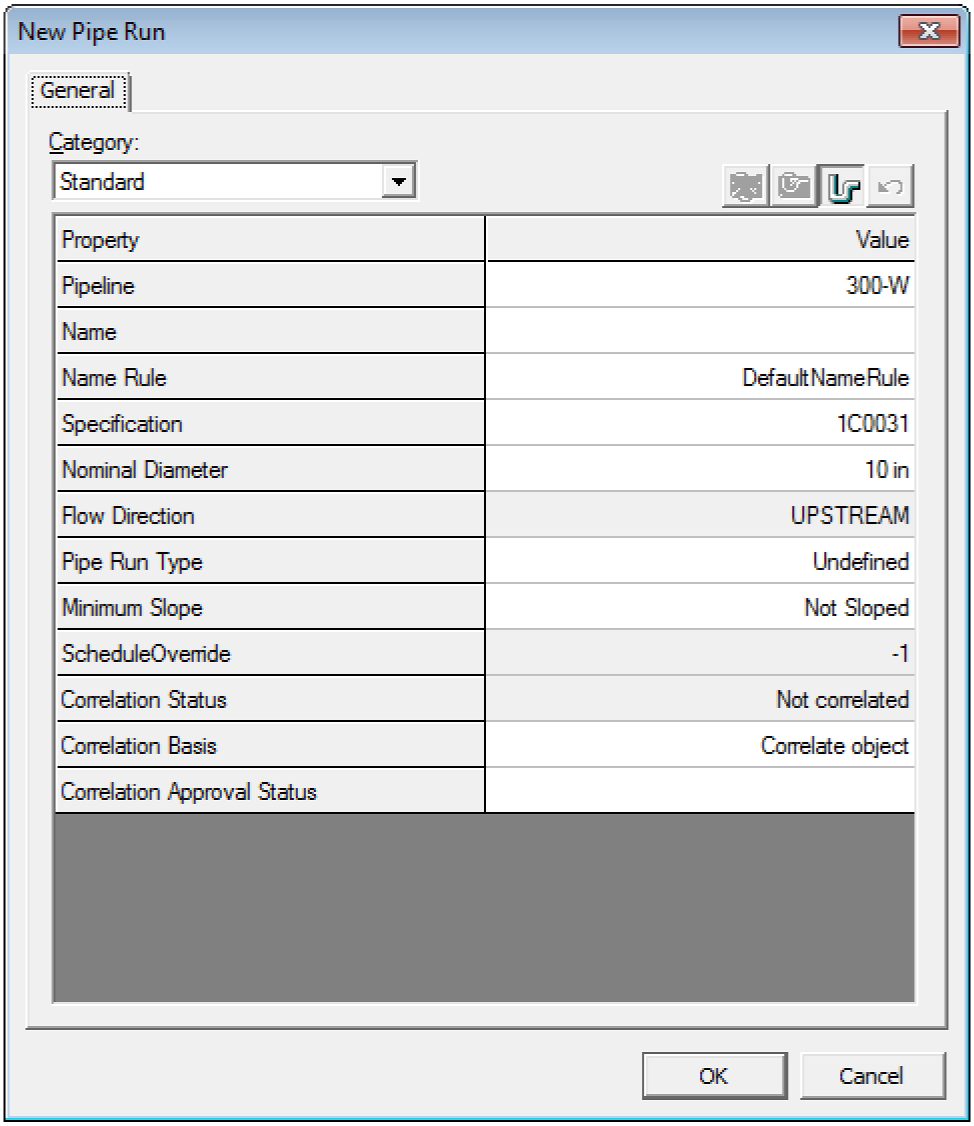


After placing the tap you next route a pipe starting from the end of the tap port to support the pipeline.

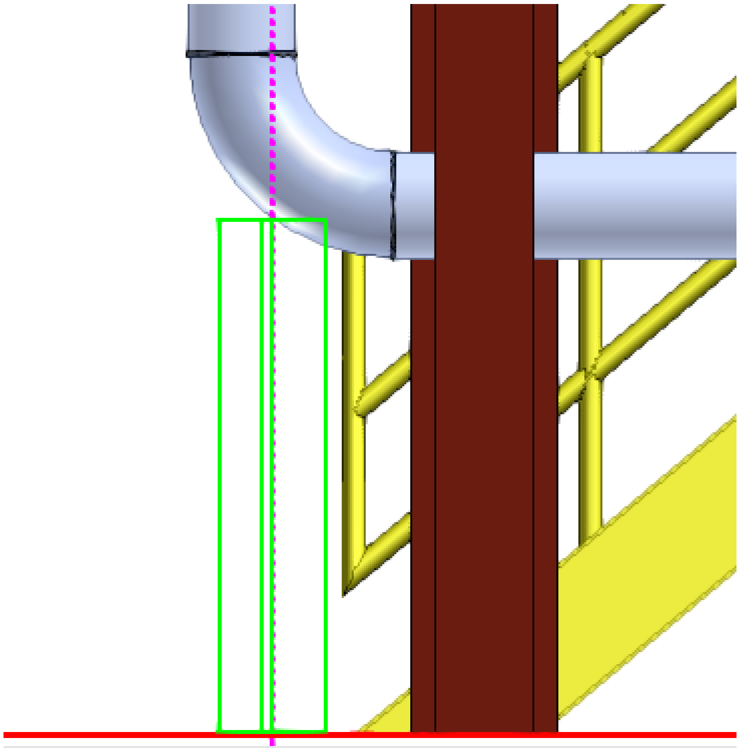
7. Click **Route Pipe**  and select the tap point.

The New Pipe Run dialog box appears.

8. Click **OK** to accept the default pipe run properties.



9. Define the length of the pipe trunnion by moving the cursor and locating the grid line

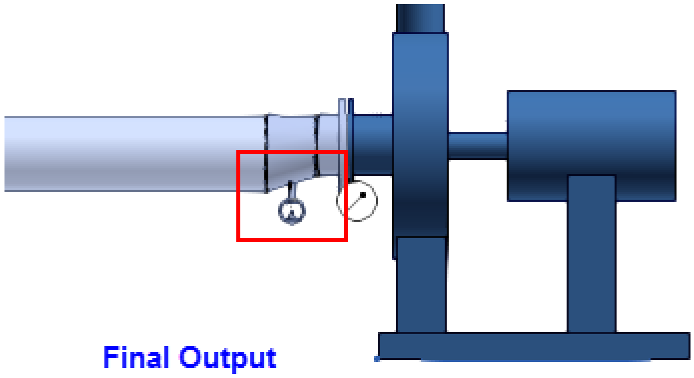


10. Click in the graphic view as soon as you locate the point to place the pipe trunnion to support the elbow.

Inserting a Tap for a Component

Objective

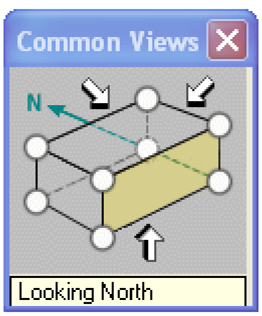
In this exercise you will be inserting a tap on the eccentric reducer Eccentric Size Change-0003 of the pipeline **300-W** by using the **Insert Tap** command and then place an instrument root valve rotated 270 deg to connect with the eccentric reducer Eccentric Size Change-0003. After inserting the tap and the instrument root valve the view of the model should resemble the highlighted section below.



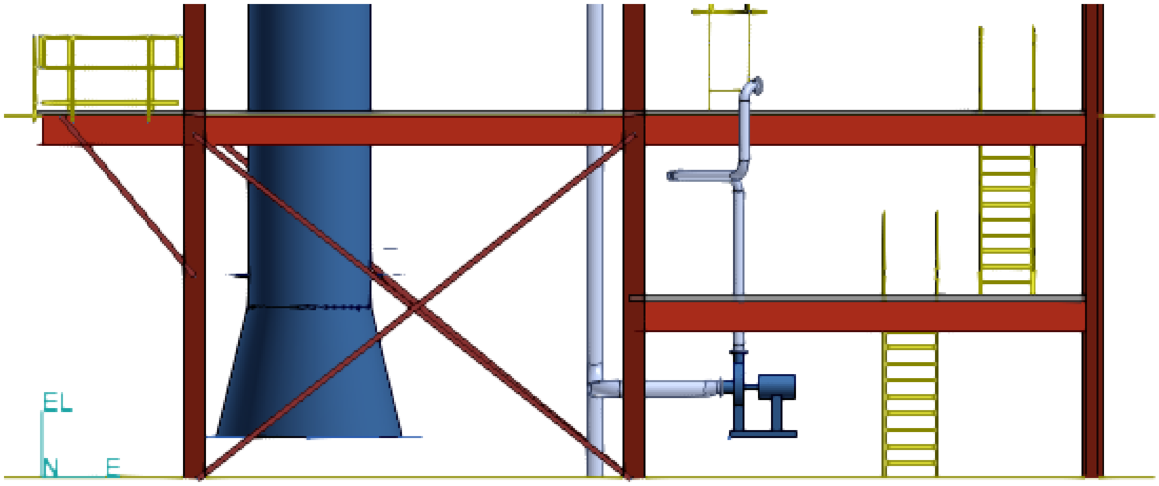
Before Starting this Procedure

- Define your workspace to display **Unit U03** and coordinate system **U03 CS**.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

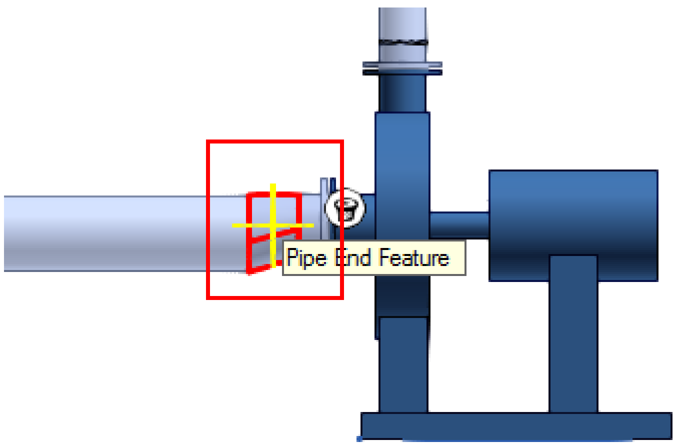
1. Change the model view to **Looking North** using the **Common Views** dialog box to focus on piping components.



2. Locate the Eccentric Size Change in the Graphic View.

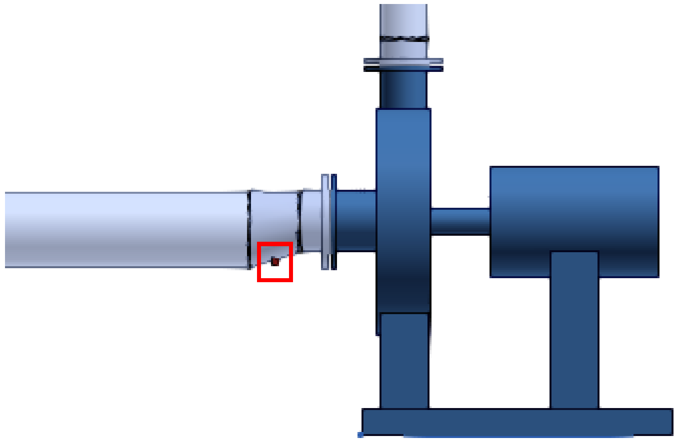


3. Click **Insert Tap** to activate the ribbon.
4. Select the eccentric reducer Eccentric Size Change, as shown below.



5. Select **0.75in SWE 3000** in the **Option** drop-down list to specify the tap ID to be placed.
6. Click Finish to accept the placement of the tap.

The tap placed on the eccentric reducer should resemble .

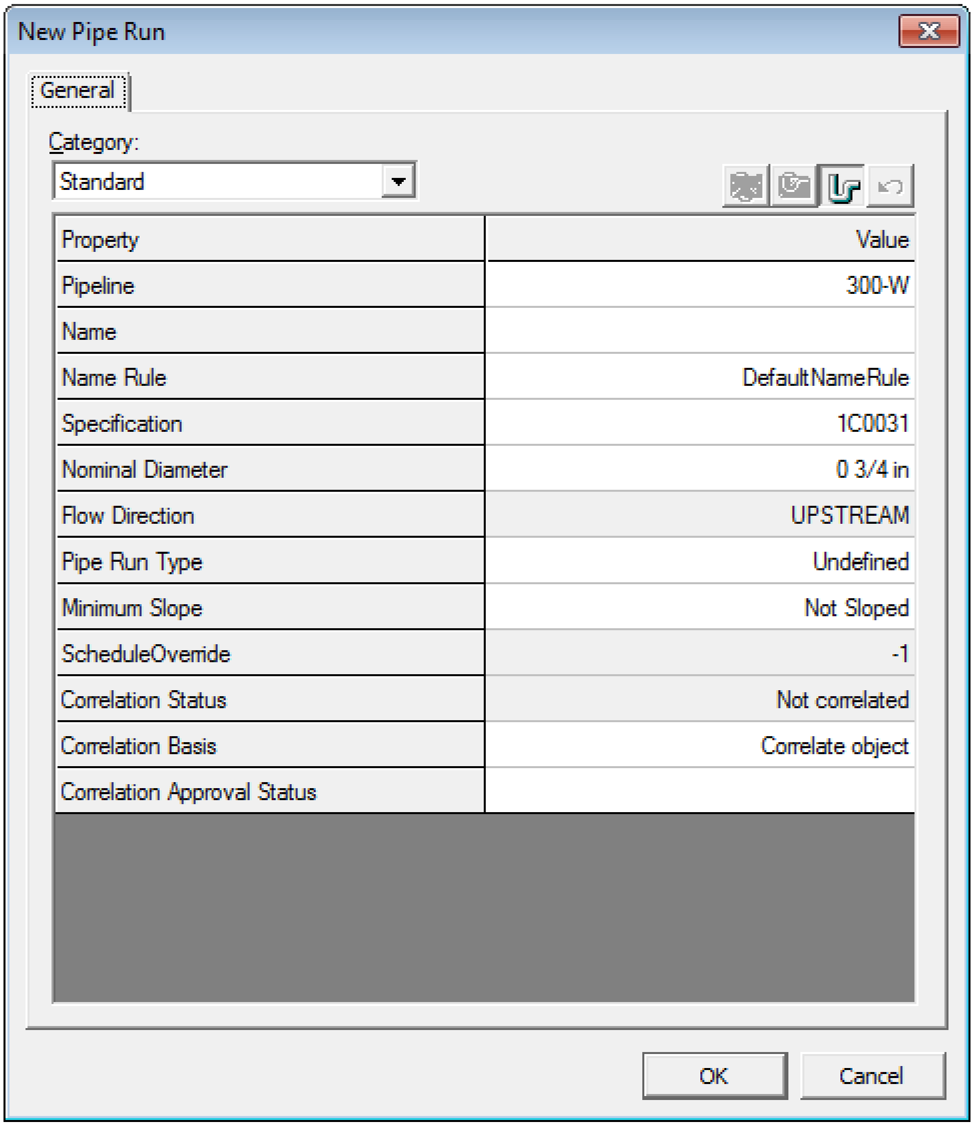


Now place a nipple at the end of the tap point to connect the instrument root valve with the tap.

7. Click **Insert Component**  and select the tap point in the graphic view.

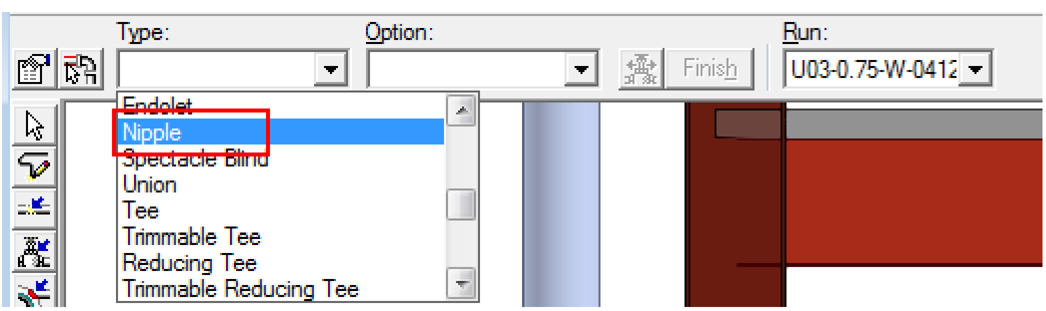
   The New Pipe Run dialog box appears.

8. Click **OK** in the dialog box to accept the default pipe run properties.



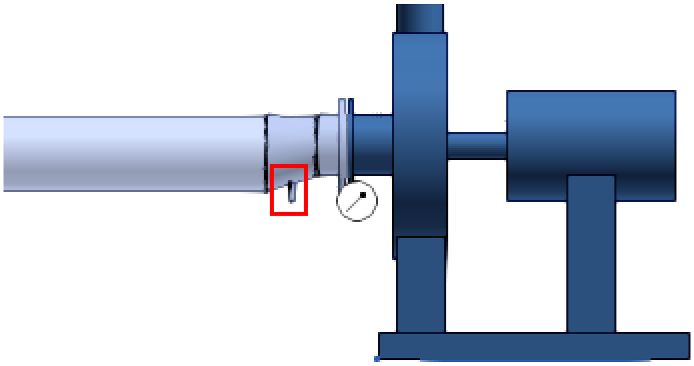
The Insert Component ribbon appears.

9. Select **Nipple** in the **Type** list to insert the nipple.



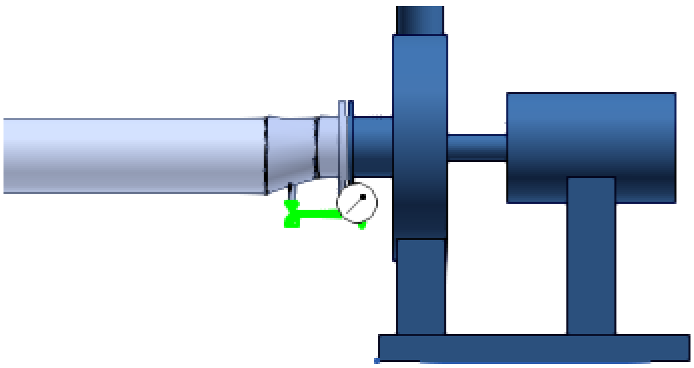
The nipple appears in the graphic view.

10. Click **Finish** to accept the placement of the nipple starting from the end point of the tap. After placing the nipple the view of the model should resemble the highlighted portion below.

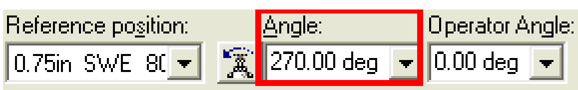


11. Now select **Instrument Root Valve** in the **Type** drop-down list to connect the root valve with the eccentric reducer.

    The Instrument Root Valve appears in the graphic view.



12. Key in **270 deg** in the **Angle** list to rotate the valve.



13. Click **Finish** to accept the placement of the Instrument Root Valve.

For more information related to sequencing objects, refer to **Insert Tap** topic in the user guide PipingUsersGuide.pdf.

SESSION 10

Routing Pipes from P&ID

Objective

By the end of this session, you will be able to:

- Route pipes using design bases from P&ID.
- View P&IDs.
- Correlate and compare model items with the design basis.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

Overview

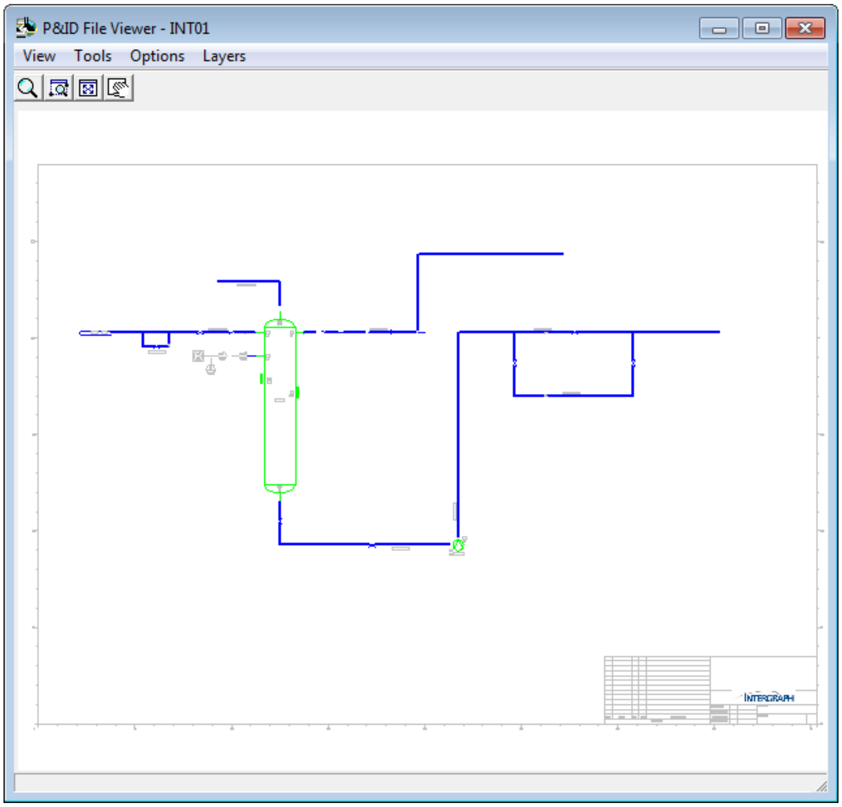
When designing or creating a plant in Smart 3D, you can reuse existing data from other design or authoring tools rather than creating a model from scratch. SmartPlant® Foundation (SPF) supports the integration of engineering tools, such as SmartPlant P&ID, Smart 3D, SmartPlant Instrumentation, and Aspen Zyqad. This integration addresses the flow of data as it moves from one engineering application to another through its lifecycle.

In an integrated environment, you publish and retrieve data from and to Smart 3D by using a central repository. During a publish operation, data such as drawings, reports, and 3D models transfers to a central repository. During a retrieve operation, the system retrieves P&IDs, Plant Break Down Structure, Project List, Work Breakdown Structure, Electrical Cable Schedules, and Instrumentation Dimensional Data Sheets from the central repository.

Retrieving Data from a P&ID

You can retrieve piping, instrumentation, and equipment data from P&ID in the integrated environment. You can use P&ID to access items, such as equipment, piping information, and SmartPlant Instrumentation dimension data, to help create the appropriate 3D design objects. For example, after you retrieve data from P&ID, you can use the P&ID File Viewer window in SP3D for guidance in routing pipes, inserting components and instruments, and placing equipment in the 3D model.

To perform these tasks, you use the P&ID File Viewer window in Smart 3D.



The SPF manages the retrieval of information and setting up of the model structure. Before any model is created, you need to create the model structure in SPF and then publish it. When you retrieve this information from the central repository, the same structure is automatically created in Smart 3D. When a model is created in SP3D, the publish functionality automatically groups items in SPF to that structure.

Design Basis

The information retrieved from P&ID is considered the design basis or design data in Smart 3D. The design basis is a collection of objects that represent pieces of data from different design and authoring tools. The administrator retrieves the design data and stores this data as design basis objects in the Smart 3D database. After the data is stored in the Smart 3D database, all users of Smart 3D can access this data.

For more information, refer to the *Design Basis Explanation* topic in *Smart 3D Integration Reference Guide*.

Comparing Model and Design Basis

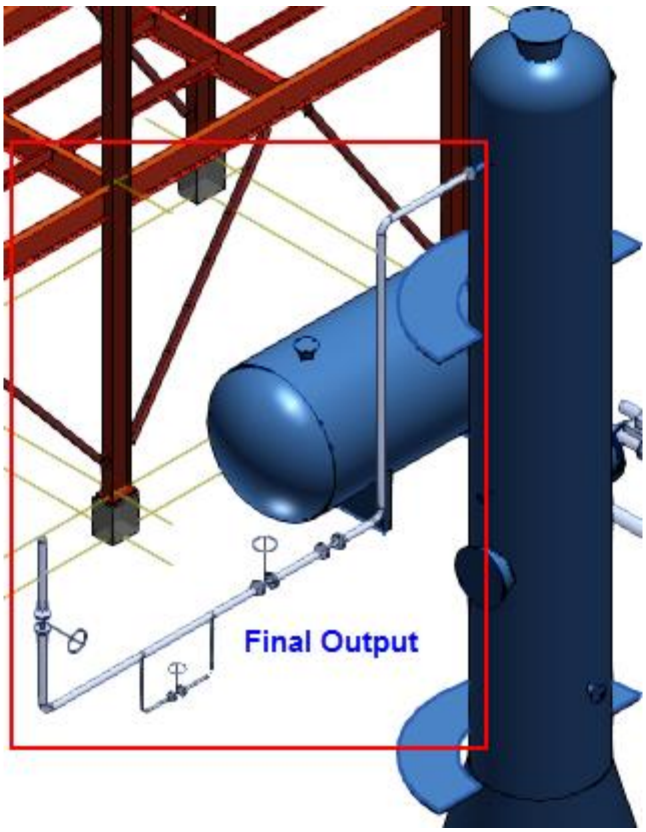
After you have routed pipelines in the model, you use P&ID to compare the pipelines in the model with the design basis. P&ID uses color codes to indicate correlation between the various components and whether the model matches the design basis and topology or not. You will learn more about comparing model and design basis by using P&ID later in the session. You will also learn to update your model to match the design basis.

Routing a Pipeline from P&ID

Objective

In this exercise you will be routing a pipeline from an equipment nozzle **A5** in **Unit U02** of your workspace by using P&ID. After routing the pipe run in the model, verify whether the pipe run you have routed matches the design basis or not. If it does not match, then you might need to update the properties of the pipe run to match the design basis. You might also need to insert pipe components to match the topology of the correlated pipe runs.

The routed pipe runs should resemble the highlighted section of .



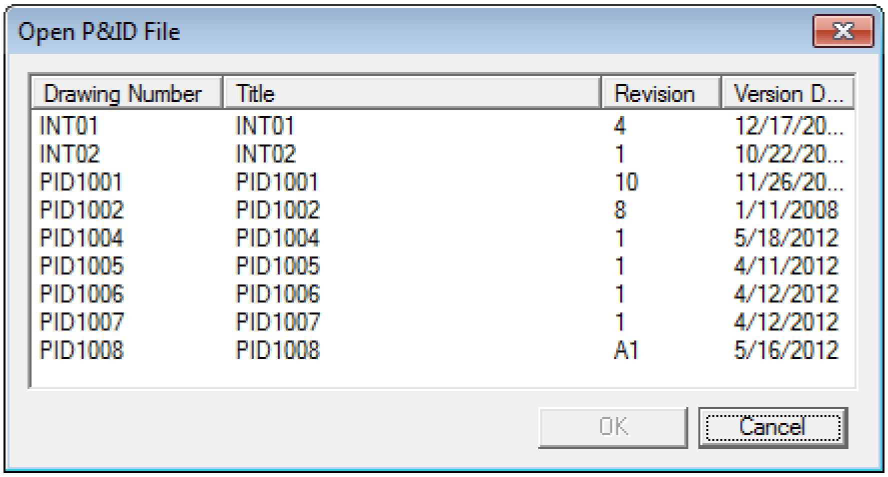
Before Starting this Procedure

- Define your workspace to display **Unit U02** and coordinate system **U02 CS**. In your training plant, select **U02** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

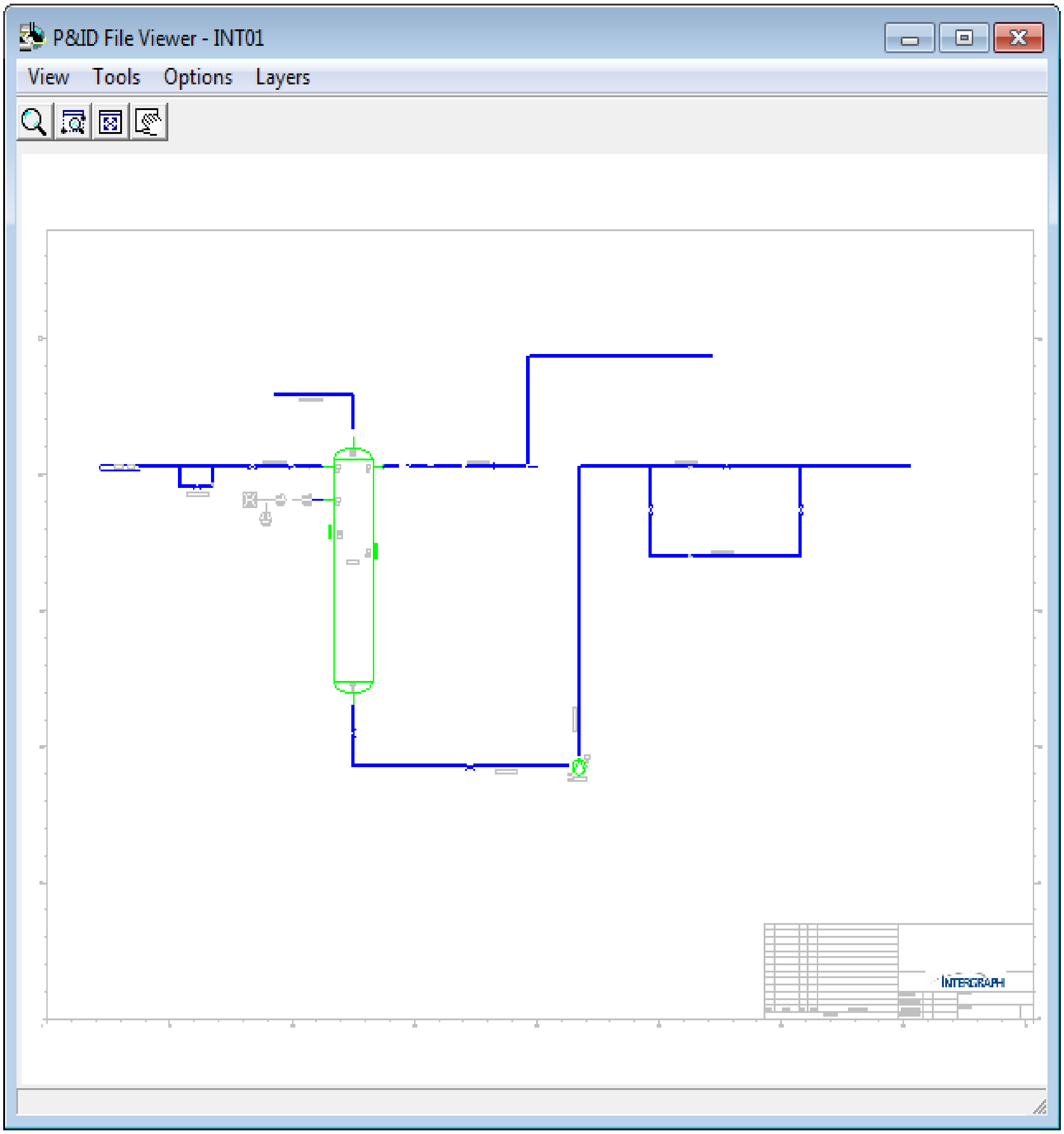
1. Select **SmartPlant > View P&ID** to open the P&ID file that contains the design basis of the pipeline you want to model.

   The View P&ID dialog box appears. This dialog box displays a list of the P&IDs available for the model.

2. Select the **INT01** drawing, and then click **Open**.

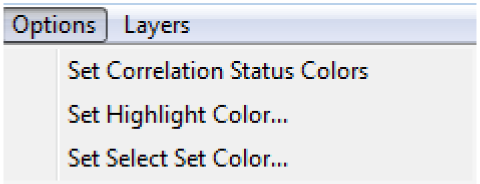

Open P&ID File
Drawing Number
Title
Revision
Version D...
INT01 INT01 4 12/17/20...
INT02 INT02 1 10/22/20...
PID1001 PID1001 10 11/26/20...
PID1002 PID1002 8 1/11/2008
PID1004 PID1004 1 5/18/2012
PID1005 PID1005 1 4/11/2012
PID1006 PID1006 1 4/12/2012
PID1007 PID1007 1 4/12/2012
PID1008 PID1008 A1 5/16/2012
OK
Cancel

The P&ID File Viewer window appears.



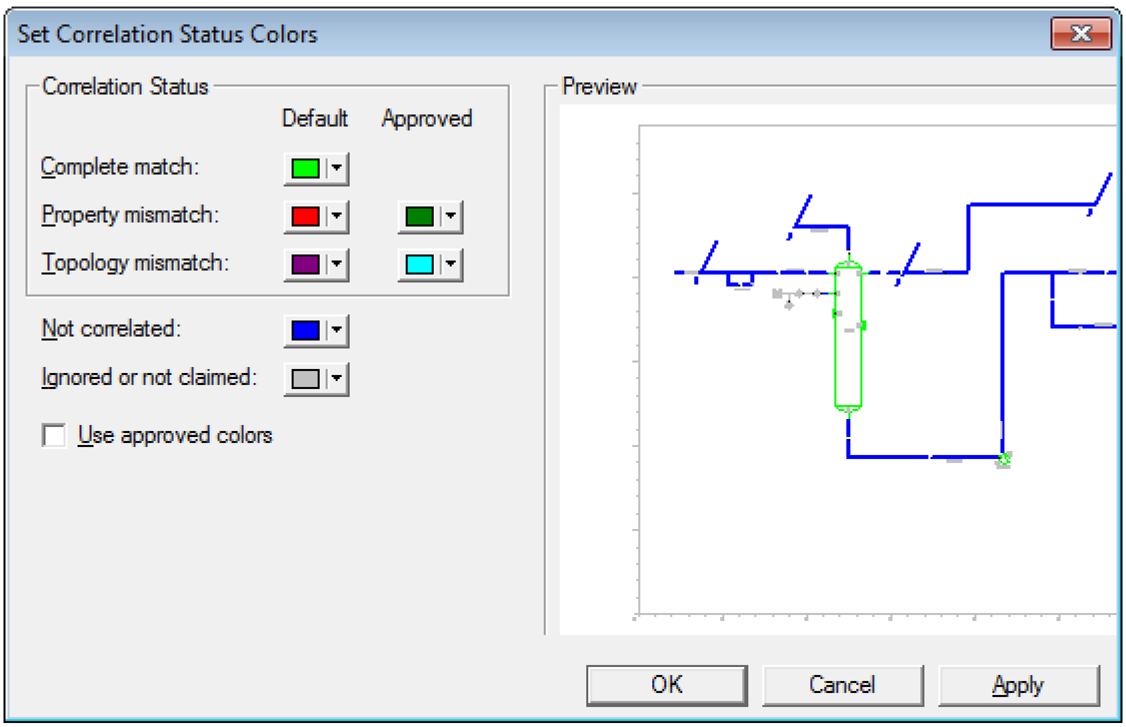
3. Click **Options > Set Correlation Status Colors** to view what the different colors in the P&ID File Viewer window indicate.

The colors of various objects in the P&ID File Viewer window reflect the correlation status between P&ID and the 3D model. You can use this as a graphical To-Do list to identify the additional tasks that you need to perform after using the **Retrieve** command.



The Set Correlation Status Colors dialog box appears. This window displays the different colors that are displayed in the P&ID File Viewer window and what they indicate.

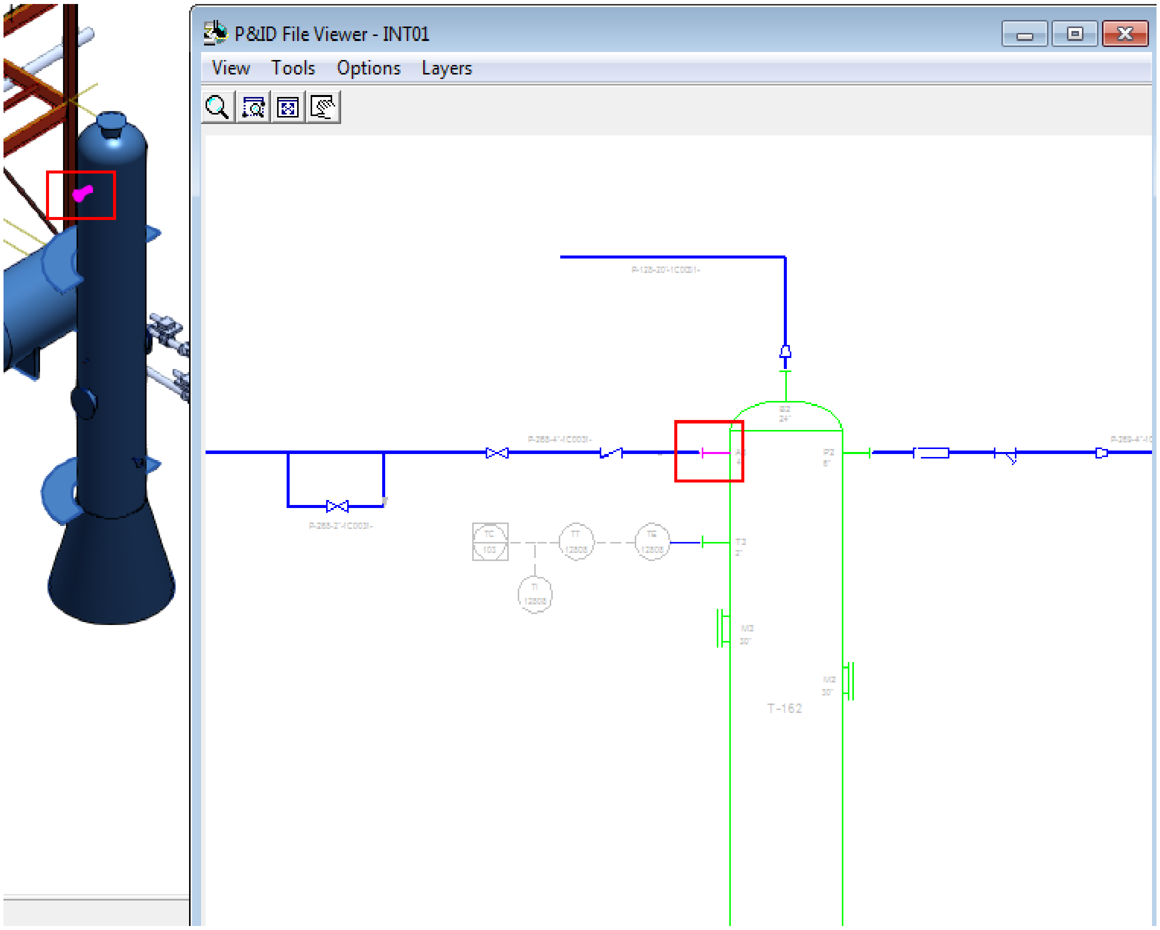
For example, in the **Set Correlation Status Colors** dialog box shows that components highlighted in red do not match the design basis and the components highlighted in green match the design basis. Knowing what the different colors in the **P&ID File Viewer** window indicate helps you in routing pipes in your model according to the design basis.



Notice that the equipment nozzles appear in green. This indicates that the equipment and nozzle are correlated and that the data for the nozzle in the model matches the P&ID design basis.

4. Click **Cancel** in the Set Correlation Status Colors dialog box to close it.
5. Select **All** in the Locate Filter to select all the objects in the model.
6. In the **P&ID File Viewer** window, click the equipment nozzle A5.

The nozzle A5 highlights in the graphic view.



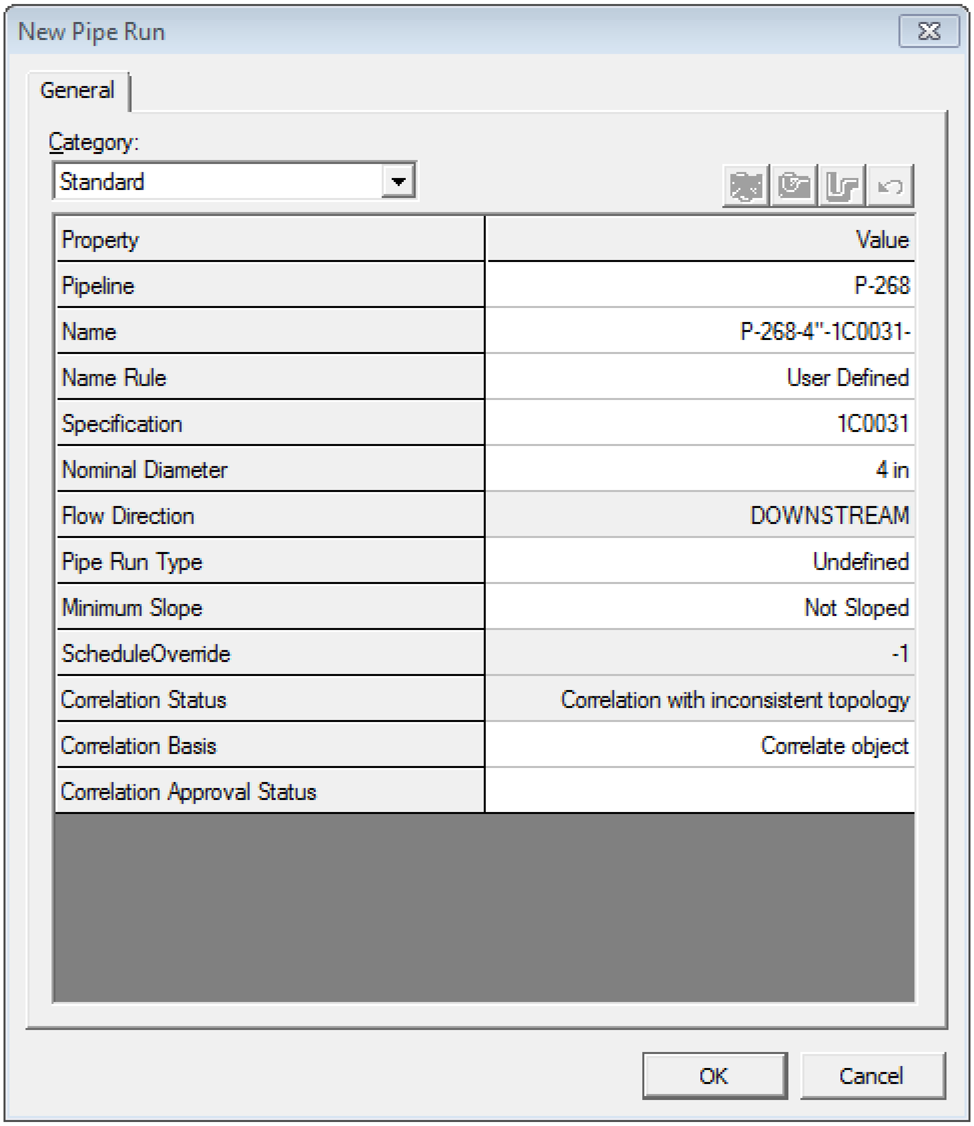
7. Click **Route Pipe** to start routing the pipe run.

   The New Pipe Run dialog box appears.

   The system automatically populates the **New Pipe Run** dialog box with design basis from P&ID because the equipment nozzle is already correlated.

8. Ensure that the New Pipe Run dialog box has the following parameters and then click **OK**:

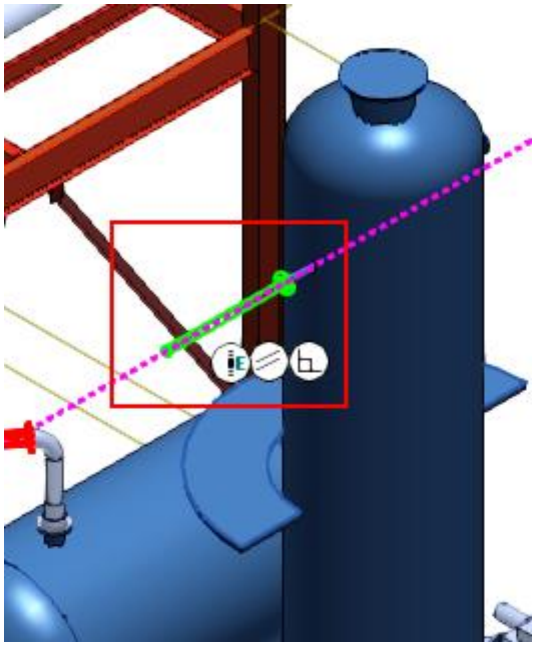
Pipeline: P-268
Name: P-268-4"-1C0031
Name Rule: User Defined
Specification: 1C0031
Nominal Diameter: 4 in
Flow Direction: DOWNSTREAM
Pipe Run Type: Undefined
Minimum Slope: Not Sloped
Schedule Override: <undefined value>
Correlation Status: Correlation with inconsistent topology
Correlation Basis: Correlate object



An outline of a pipe run will appear in the graphic view starting at pipe nozzle A5.

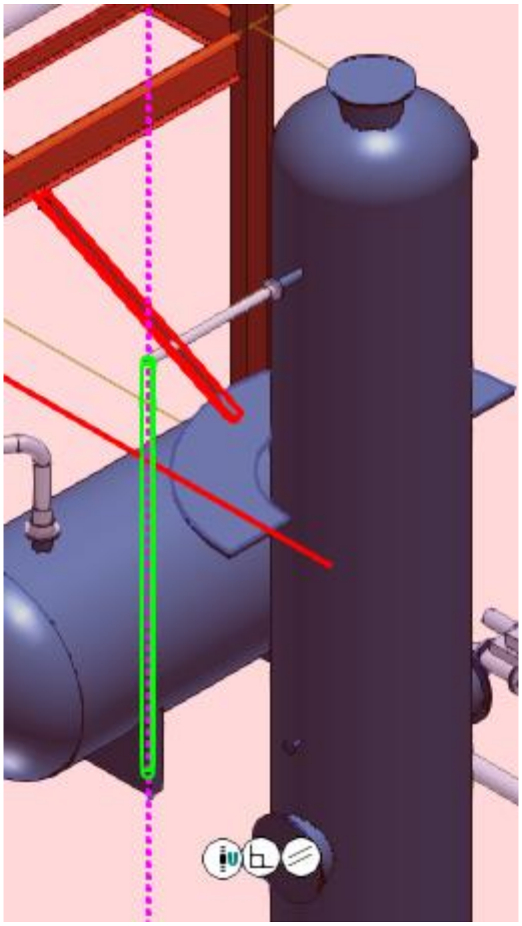
9. Key in **5 ft** in the Length box on the Route Pipe ribbon to lock the length of the pipe run.

10. Move the cursor towards the left until SmartSketch displays the E glyph, which indicates that you are routing the pipe in the East/West direction.
11. Click in the graphic view to place the pipe.



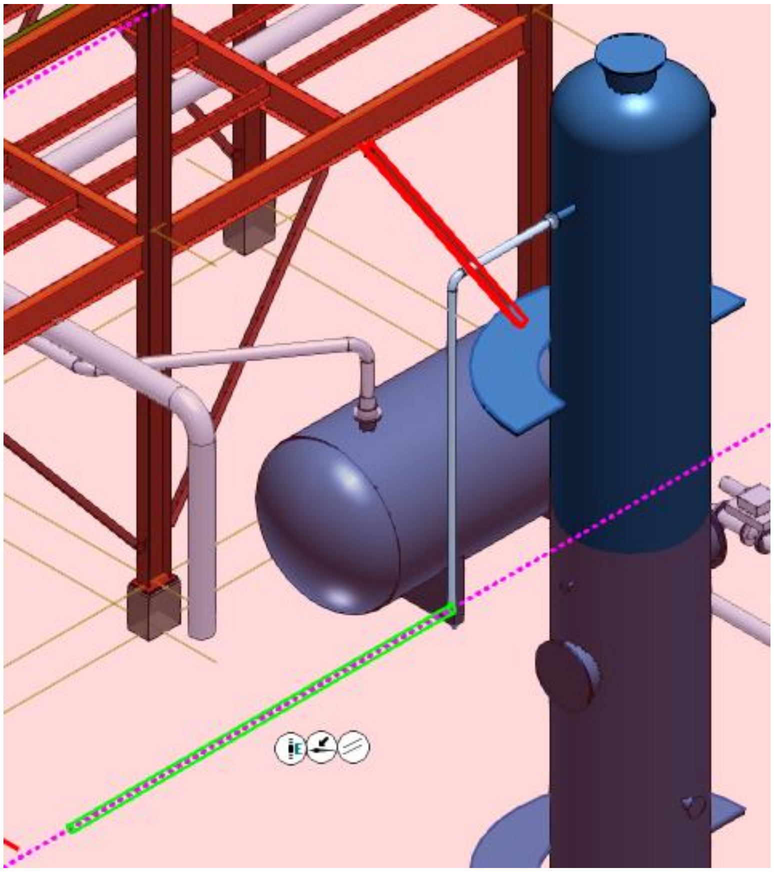
12. Key in **15 ft** in the **Length** box.
13. Move the cursor downwards until SmartSketch displays the U glyph. If you cannot position the cursor downwards, then select **No Plane** in the **Plane** drop-down list.

14. Click in the graphic view to place the pipe.



15. Key in **20 ft** in the **Length** box.

16. Move the cursor towards the left (west direction) until SmartSketch displays the E glyph, and click in the graphic view to place the pipe.

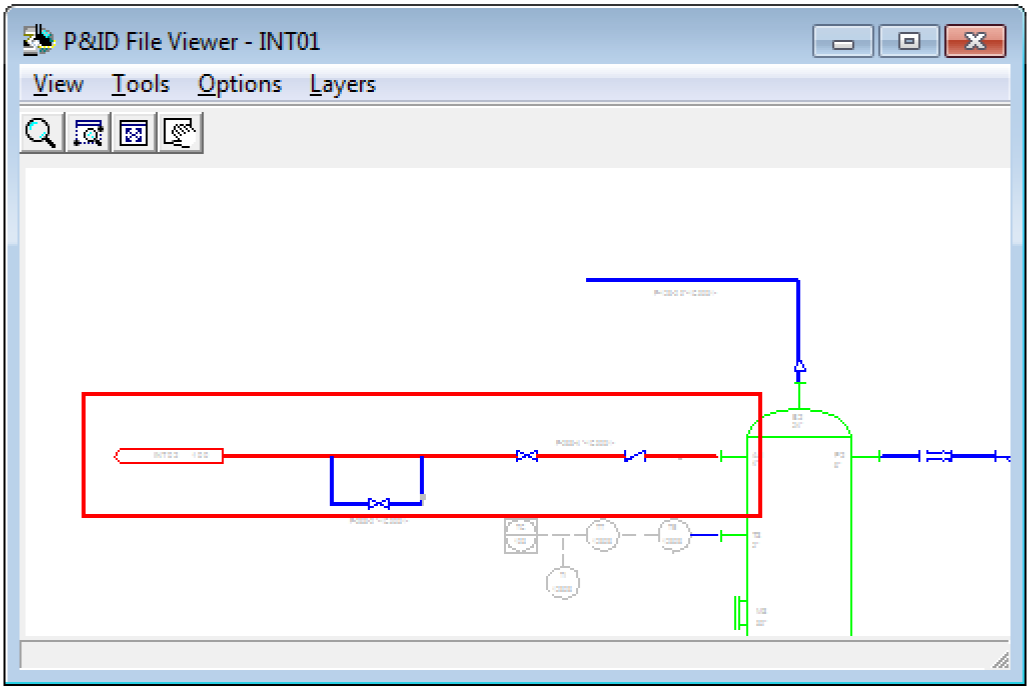


17. Right-click to end the **Route Pipe** command.

Match 3D Objects with P&ID Design Basis

1. Open the **P&ID File Viewer** window.

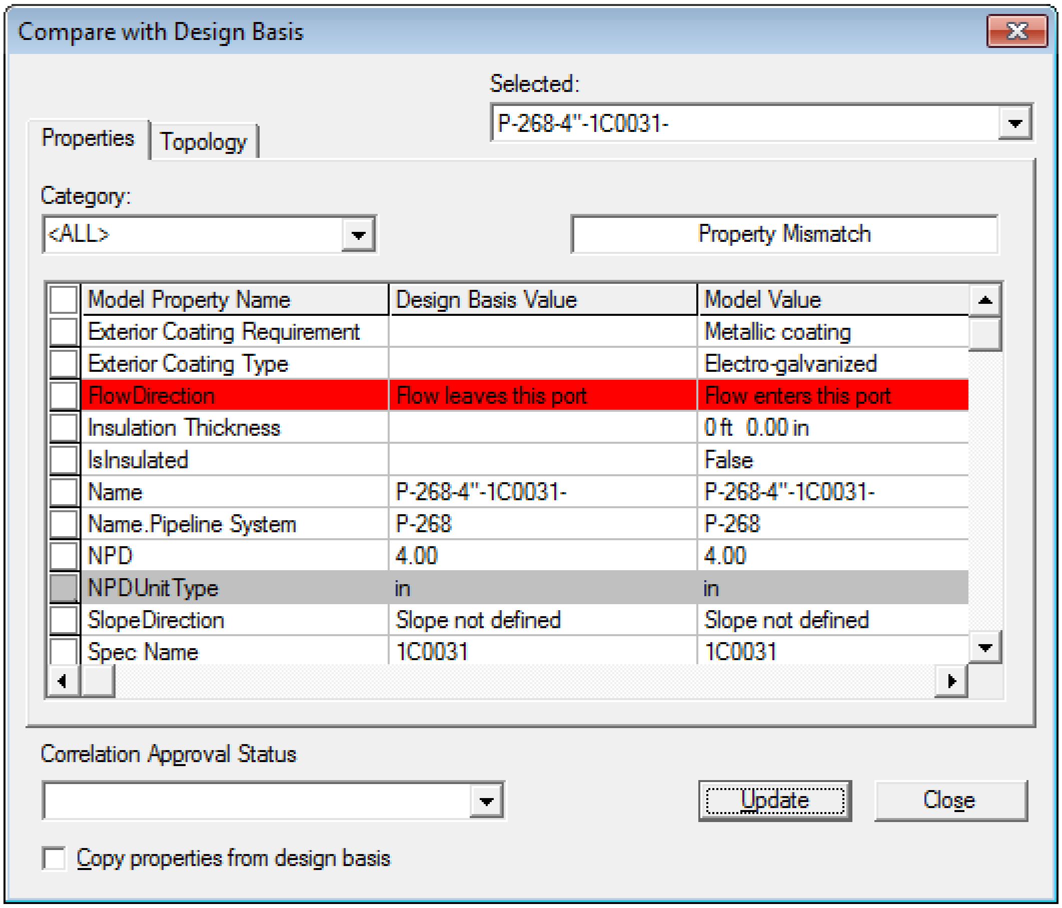
   The pipe run you have just routed will be highlighted in red. This indicates that the pipe run does not match the design basis.



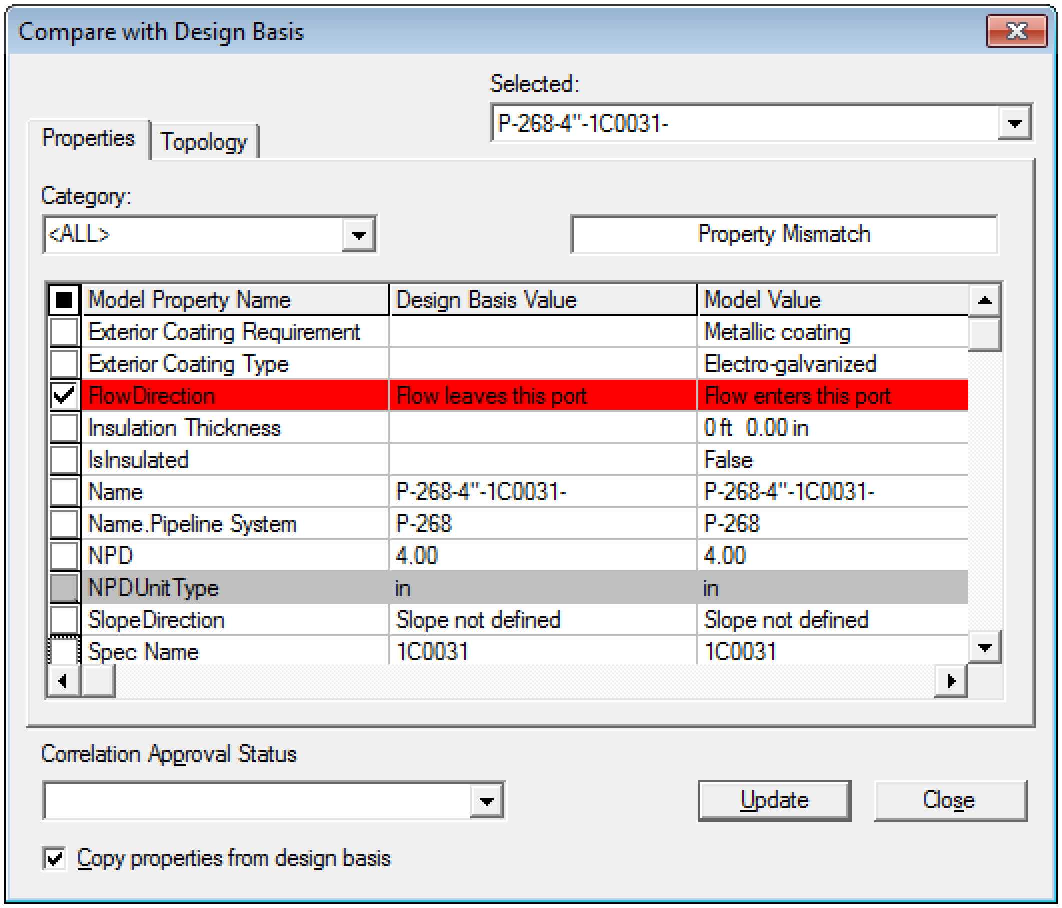
Perform the following steps to update the pipe run you have placed to match the design basis.

1. In the graphic view, select the pipe run that you have just routed.
2. Click **SmartPlant > Compare Design Basis** to compare the pipe run you have placed with the design basis.

The Compare with Design Basis dialog box appears. The Flow Direction row is highlighted in red. This indicates that the flow direction property of the pipe run does not match the design basis.



3. Select **Copy properties from Design Basis** (at the bottom of the dialog) and **Flow Direction**.



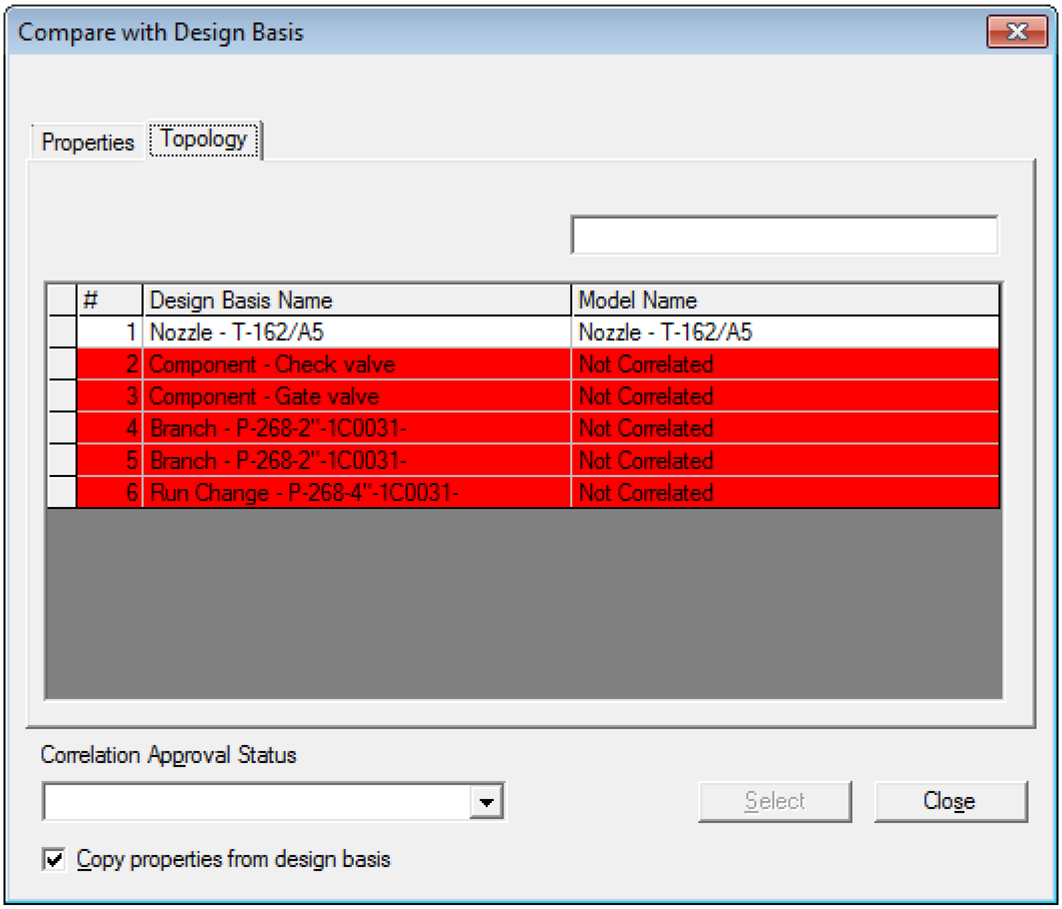
4. Click **Update** to update the flow direction property as per the design basis,

   The Flow Direction row now becomes white indicating that the flow direction on the modeled object now matches the design basis.

Correlate and Compare Model Objects with Design Basis

Objective

In this exercise you will use the **Topology** tab of the **Compare with Design Basis** dialog box. A couple of columns under the **Topology** tab are highlighted in red. This indicates that there are discrepancies between the design basis and the correlated pipe run. You did not place any components in the pipe run that you routed and hence, the pipe run topology does not match the design basis.

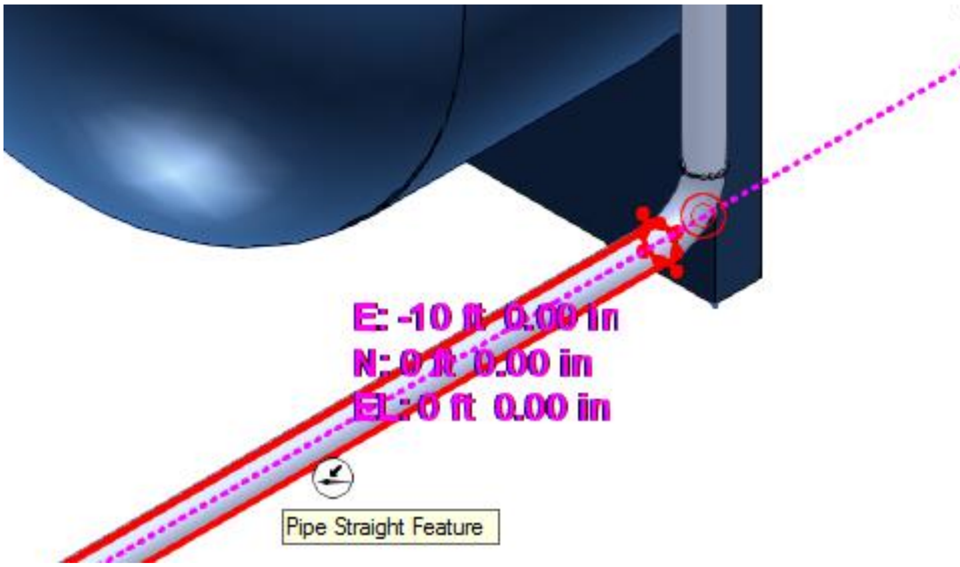


Now, continue modeling the pipeline by inserting a bypass line and pipe components until the pipe run topology matches the design basis. Perform the following steps:

1. Click **Close** to close the **Compare with Design Basis** dialog box.
2. Click **PinPoint** on the Common toolbar.
3. Select **Reposition Target**, and then click to select the origin of the elbow as the target to start routing the bypass line.

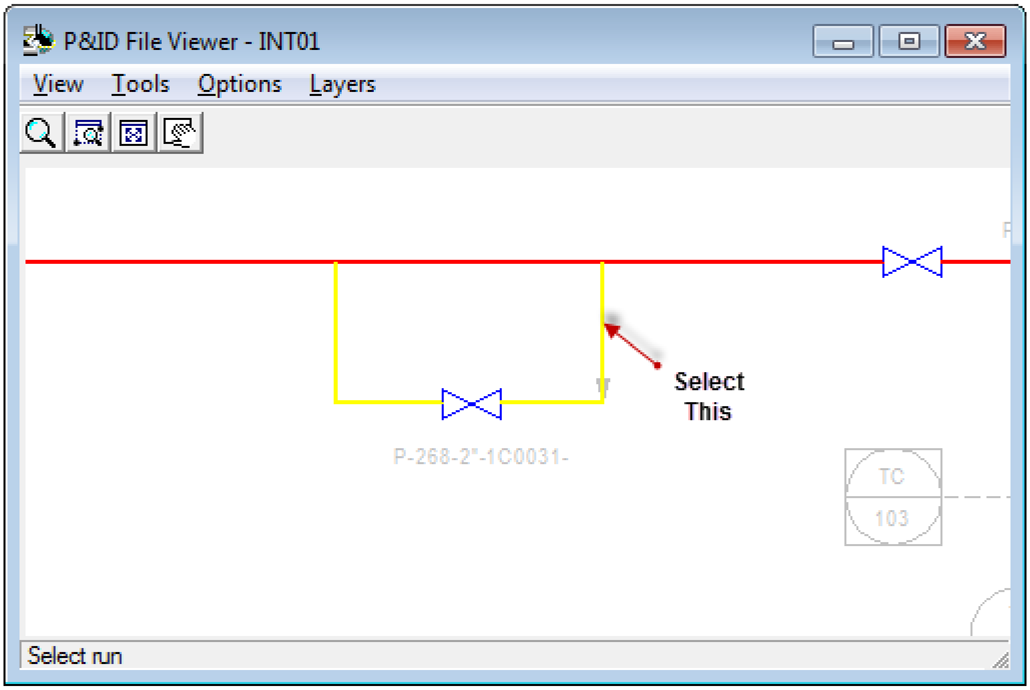
4. Key in **-10 ft** in the **E** drop-down field of the PinPoint ribbon to bypass the pipe at a distance of 10 ft in the West direction from the starting point.
5. Click **Route Pipe**  on the vertical toolbar.
6. Position the cursor along the **Pipe Straight Feature**, as shown and click to define the active placement point.

   The PinPoint constraints your cursor movement on an infinite East-West plane at 10 ft from the target. Therefore, you need to locate a projection line along the Pipe Straight Feature that indicates the intersection point between the plane and the line.



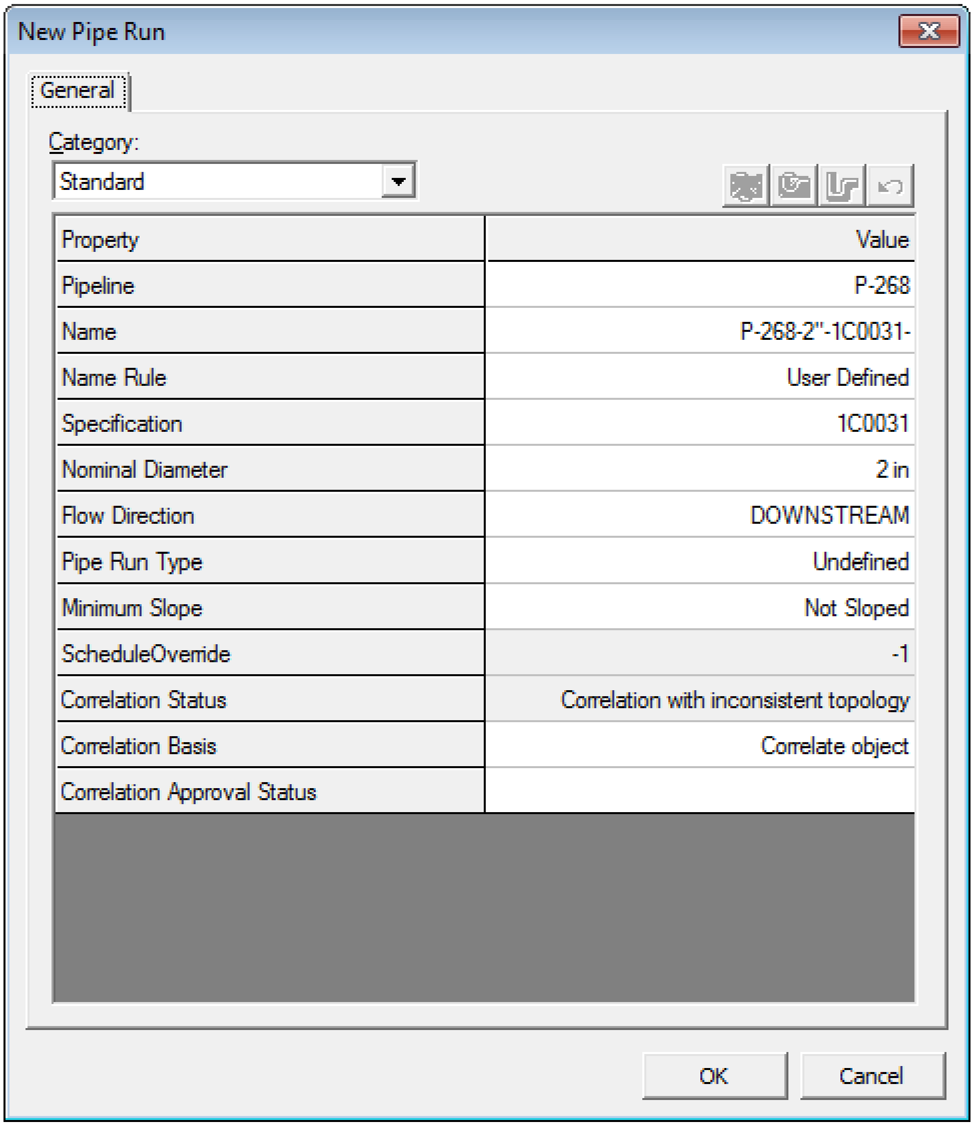
The P&ID File Viewer window appears.

7. In the P&ID File Viewer window, select the bypass line.



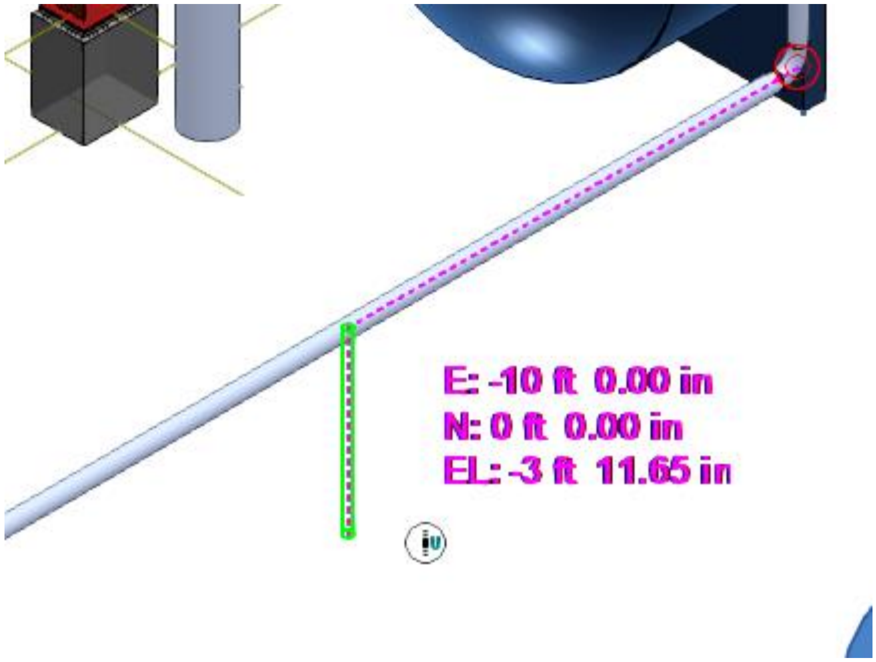
The New Pipe Run dialog box appears. The system populates the New Pipe Run dialog box automatically by using P&ID and design basis.

8. Click **OK** to accept the default values and close the dialog box.

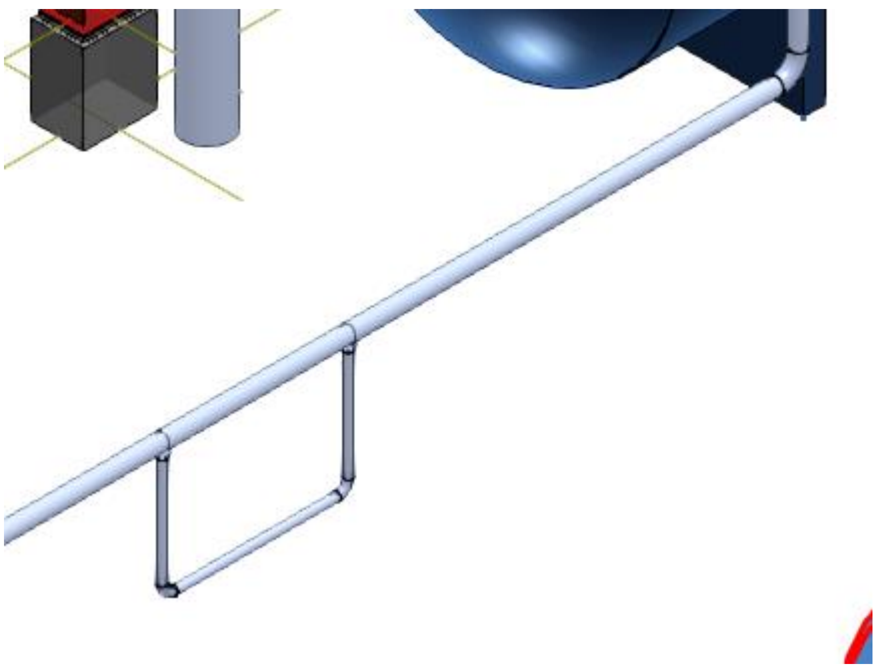


You now see an outline of the pipe run in the graphic view.

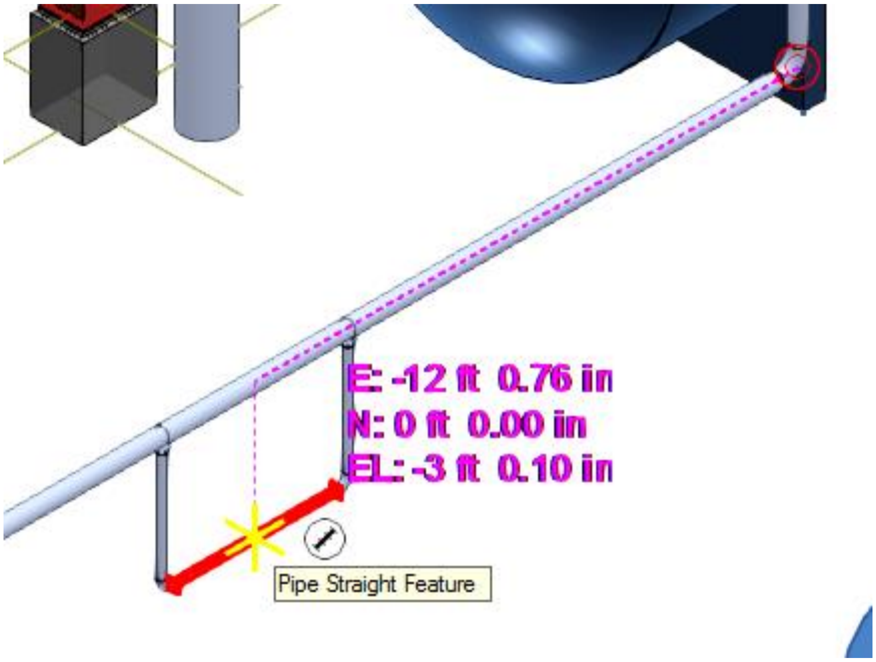
9. Click to place the pipe run.



You have now successfully placed the first pipe segment for this bypass line by using the SmartSketch glyphs to control the direction. Now, route two more pipe segments to complete the bypass line as shown.



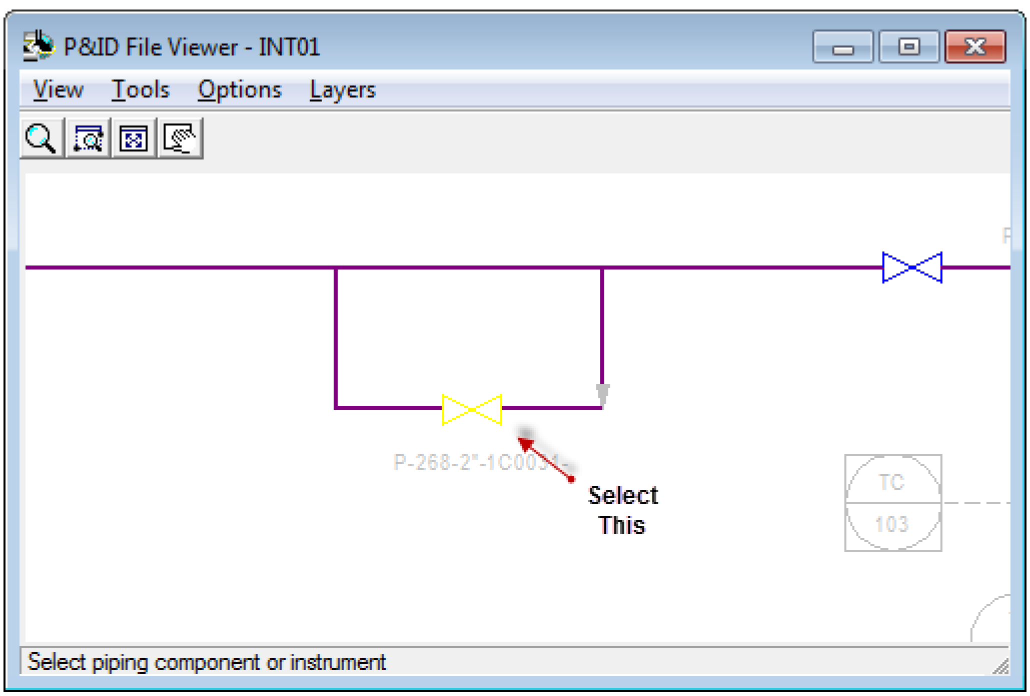
10. Now, place a gate valve on the bypass line you have just placed. Click **Insert Component** and use SmartSketch to locate the midpoint glyph of the pipe you have just placed. Use the midpoint as the active placement point for the component.



11. Click on the active placement point of the pipe.

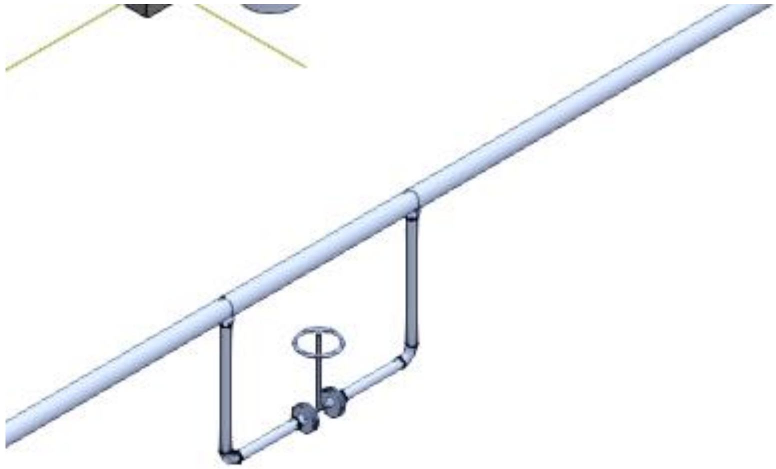
    The P&ID File Viewer window appears.

12. In the P&ID File Viewer window, click the gate valve located in the bypass line.



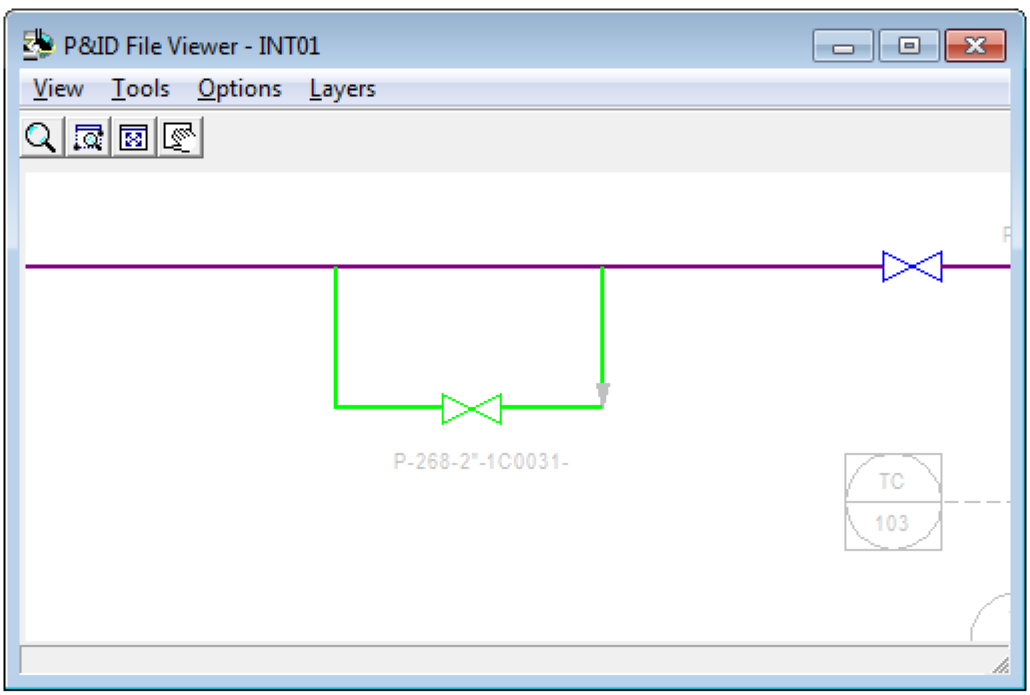
The software uses P&ID and design basis to select the appropriate pipe component.

13. Click **Finish** on the Insert Component ribbon to place the gate valve.



14. Open the P&ID File Viewer window.

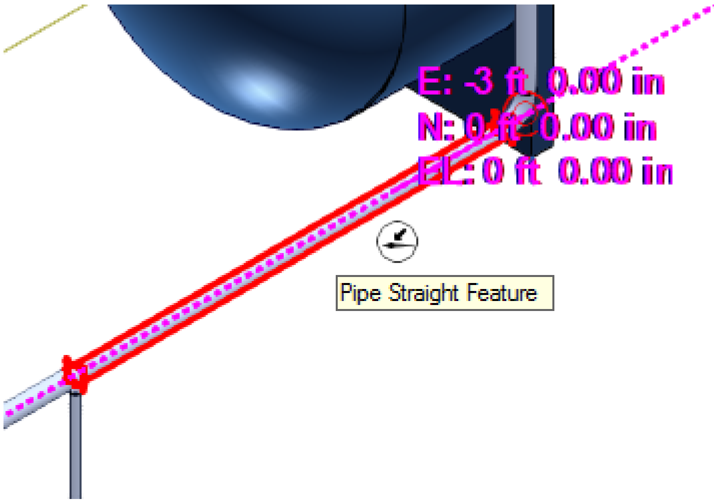
    The bypass line you just placed is highlighted in green. This indicates that the bypass line is correlated and its properties and topology matches the design basis.



Now, place a check valve at 3 ft distance from the center of the elbow you placed earlier.

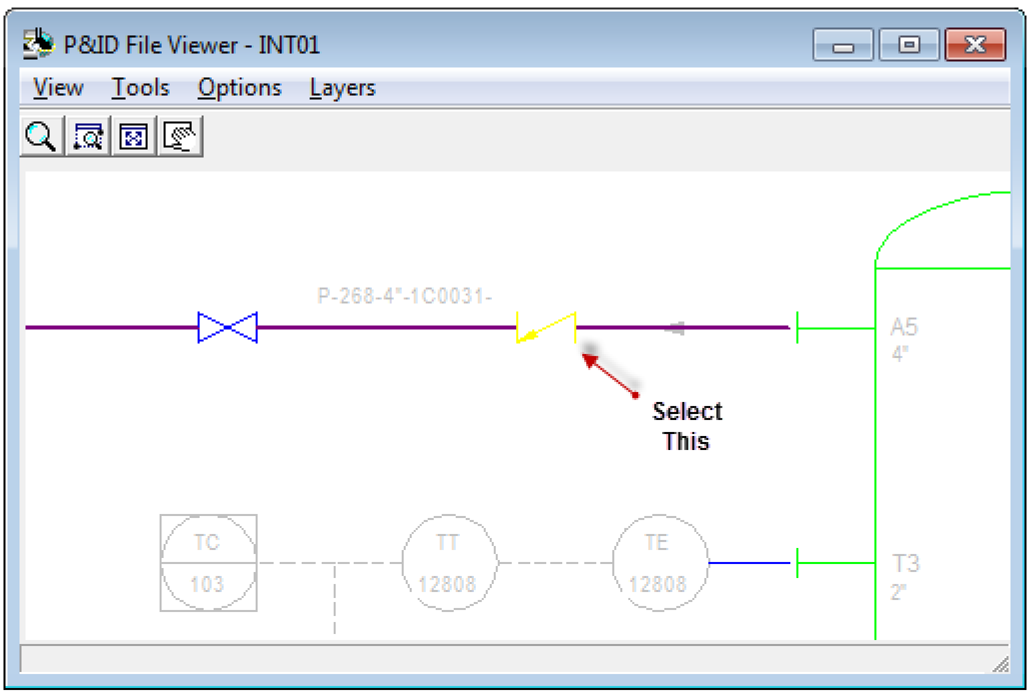
15. Key in **-3 ft** for **E** on the PinPoint ribbon to define the distance of 3 ft from the target.
16. Click **Insert Component** .

17. Now, position the cursor along the **Pipe Straight Feature**, and click to define the active placement point.



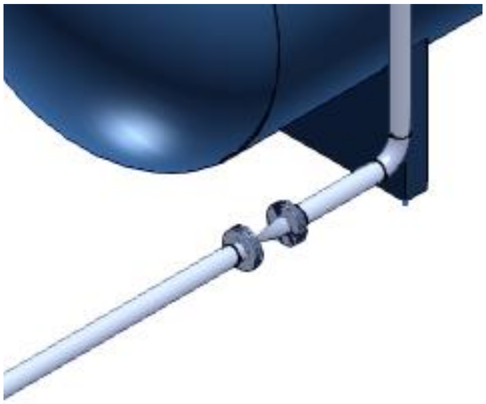
The P&ID File Viewer window appears.

18. In the **P&ID File Viewer** window, click the check valve.

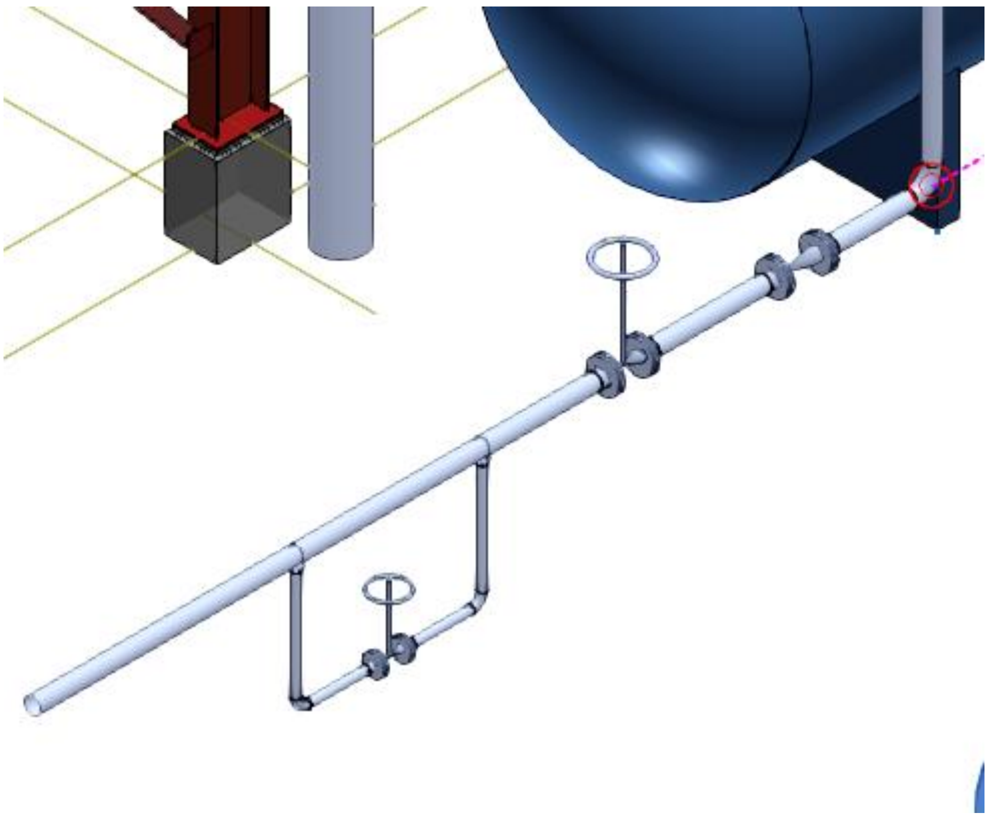


An outline of a check valve (base part) with mating flanges will appear in the graphic view at the active placement point. The software uses P&ID and the design basis to select the appropriate component.

19. Click **Finish** to place the pipe component.



20. Now, use PinPoint to locate the midpoint of the pipe run to the left of the check valve. Use it as the active placement point for the component you want to place. Then, place a gate valve.

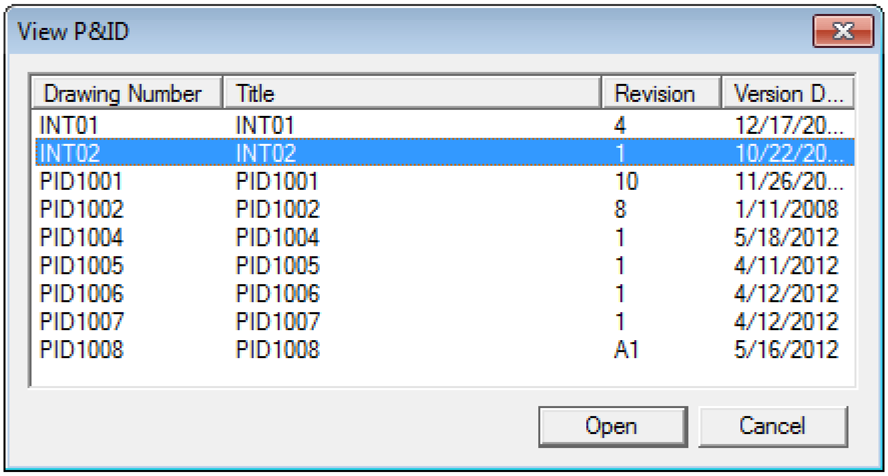


Now, continue routing the pipeline using the off-page connector (OPC). Open the continuation P&ID INT02.

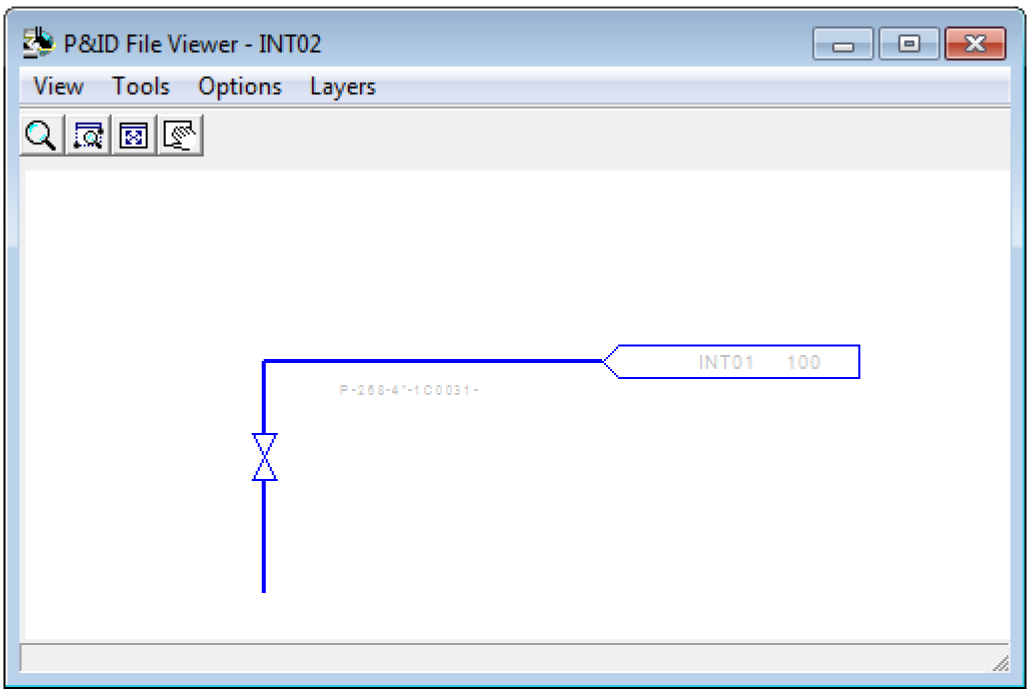
21. Click **SmartPlant > View P&ID**.

The View P&ID dialog box appears.

22. Select the drawing INT02 and click **Open**.

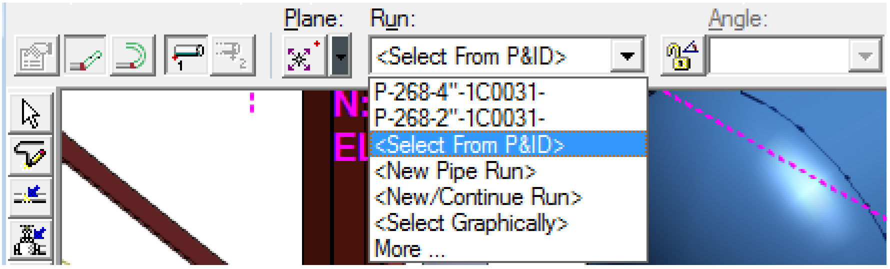


The P&ID File Viewer window appears.



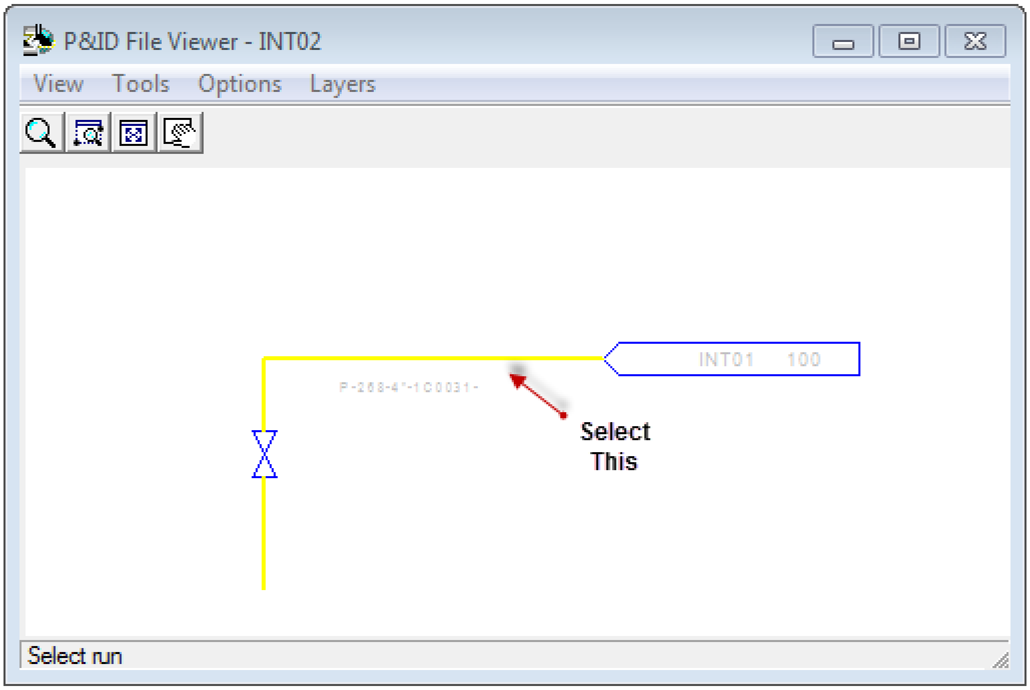
23. Now click **Route Pipe** .

24. Select the **Select From P&ID** option in the **Run** list on the ribbon.



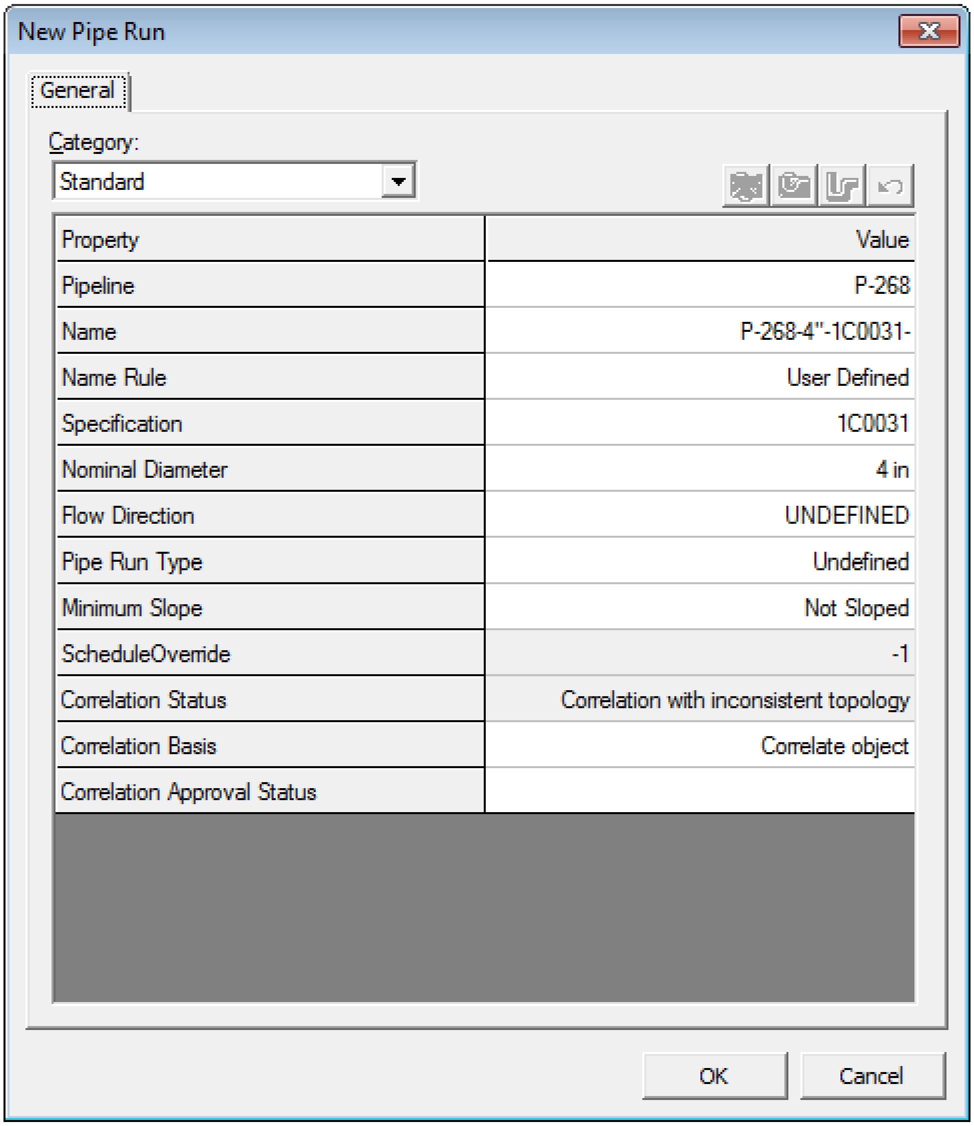
The P&ID File Viewer window appears.

25. In the **P&ID File Viewer**, click the **P-268** pipeline.



The New Pipe Run dialog box appears. The system populates the New Pipe Run dialog box automatically by using P&ID and design basis.

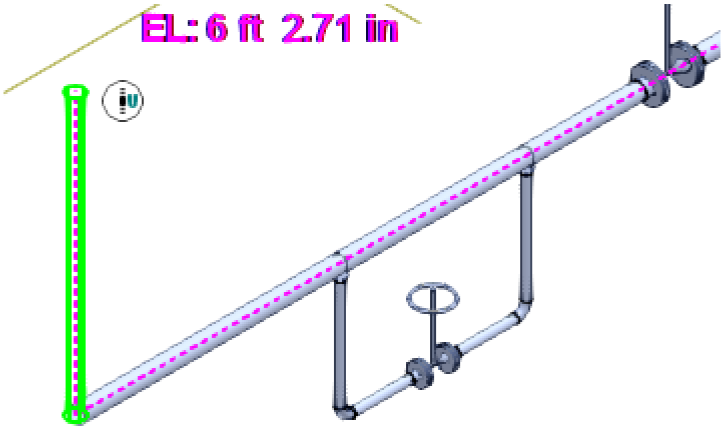
26. Click **OK** to accept the default values and close the dialog box.



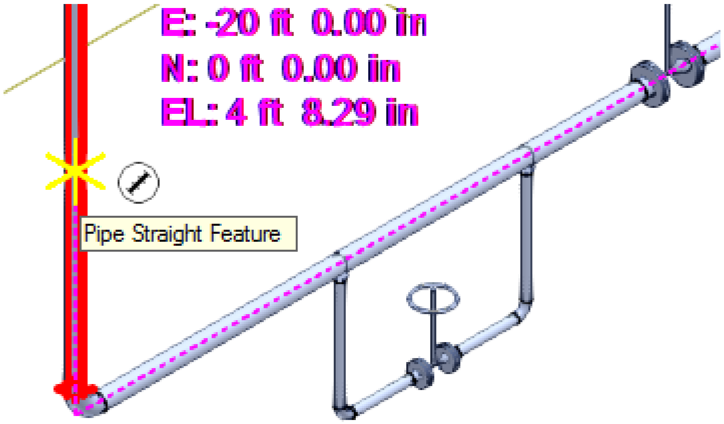
27. Select the end feature of the pipe to start the routing.

    An outline of the pipe run appears in your model.

28. Click in the graphic view to place the pipe run.

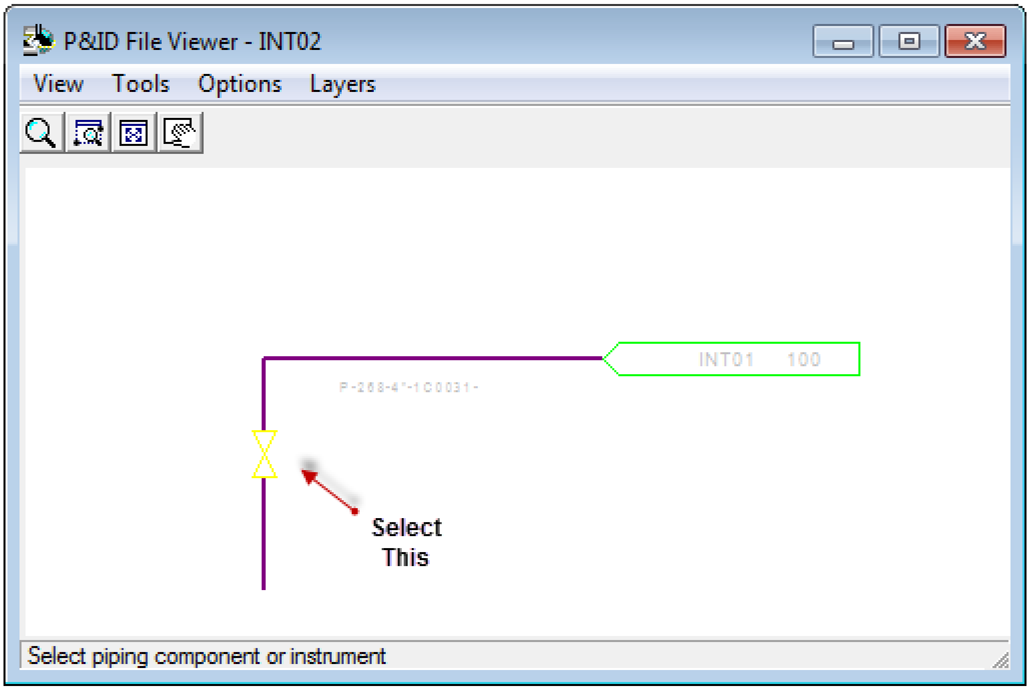


29. Click **Insert Component** .
30. Position the cursor along the **Pipe Straight Feature**, and click to define the active placement point.



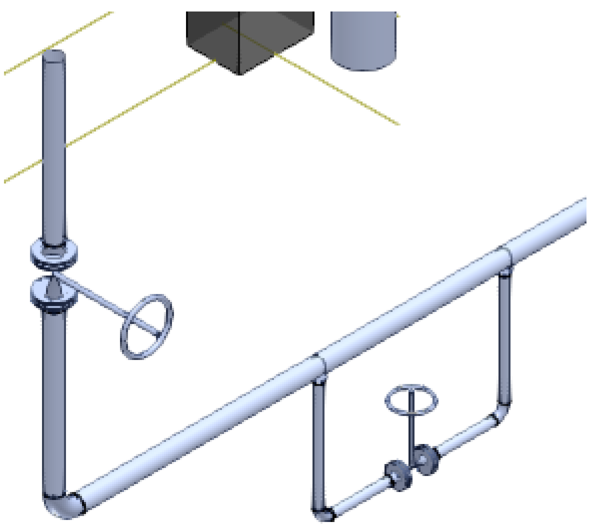
The P&ID File Viewer window appears.

31. In the **P&ID File Viewer** dialog, click the gate valve.



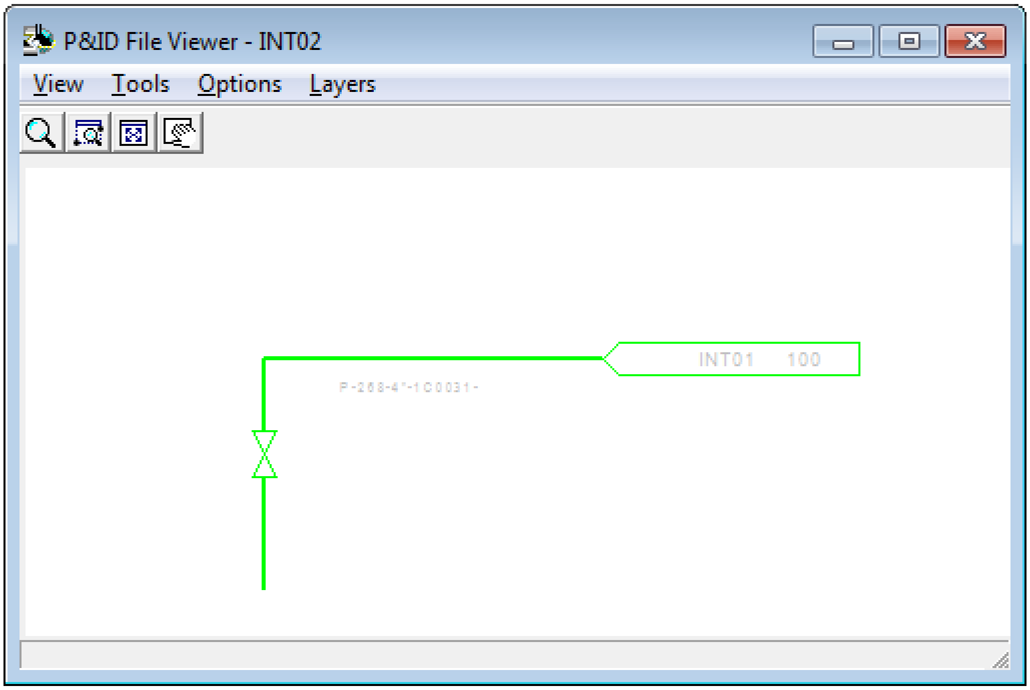
An outline of a gate valve (base part) with mating flanges will appear in the graphic view at the active placement point. The software uses P&ID and the design basis to select the appropriate component.

32. Key in **90 deg** in the **Angle** drop-down list to rotate the valve.
33. Click **Finish** to place the pipe component.



34. Open the **P&ID File Viewer** window.

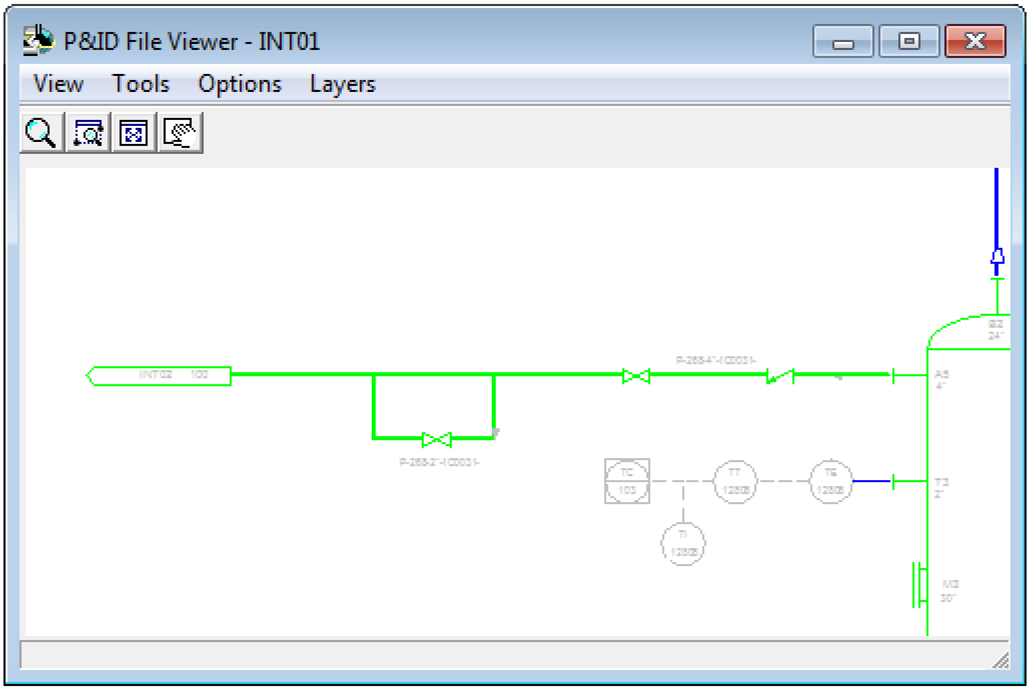
The pipe section you placed is highlighted in green, which indicates that the pipe section and the off-page connector now are correlated and its properties and topology match the *design basis.*



NOTE Unlike other P&ID elements, the OPC is correlated when the two pipelines are joined. The actual P&ID symbol is never selected or used for correlation. The main issue to know when correlating piping with an OPC is that a weld is placed where the two pipelines meet. Therefore, you should find a logical connection point for this weld in the model to avoid adding an additional unneeded weld. In this example, the weld at the 90 deg elbow was used for this logical connection point.

35. Open the P&ID INT01 drawing.

The pipe section you placed is highlighted in green, which indicates that the pipe section and the OPC now are correlated and its properties and topology match the design basis.

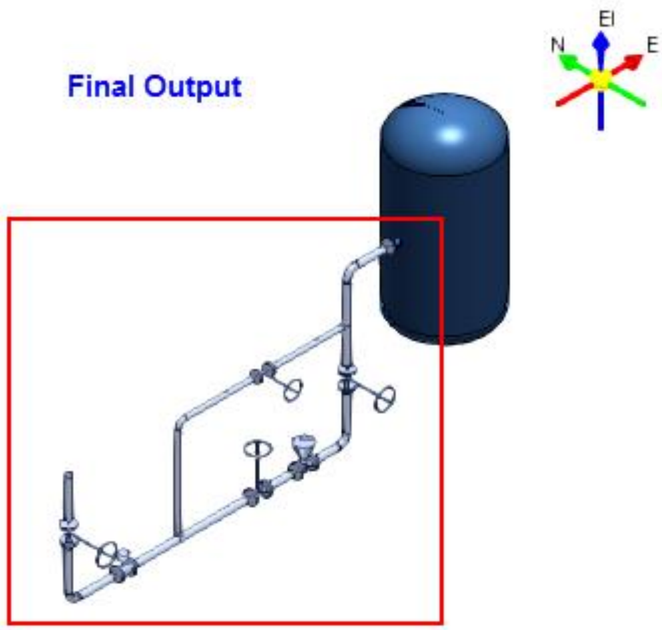


For more information related to correlate and compare the model items with the design basis refer to Piping in the Integrated Environment topic in the user guide PipingUsersGuide.pdf.

Placing Instruments from a P&ID

Objective

In this exercise you will be routing a pipeline from the equipment nozzle **V-129A/N1** and then insert instruments along this pipeline in **Area01**. The view of your model after placing these instruments should resemble the highlighted area in .



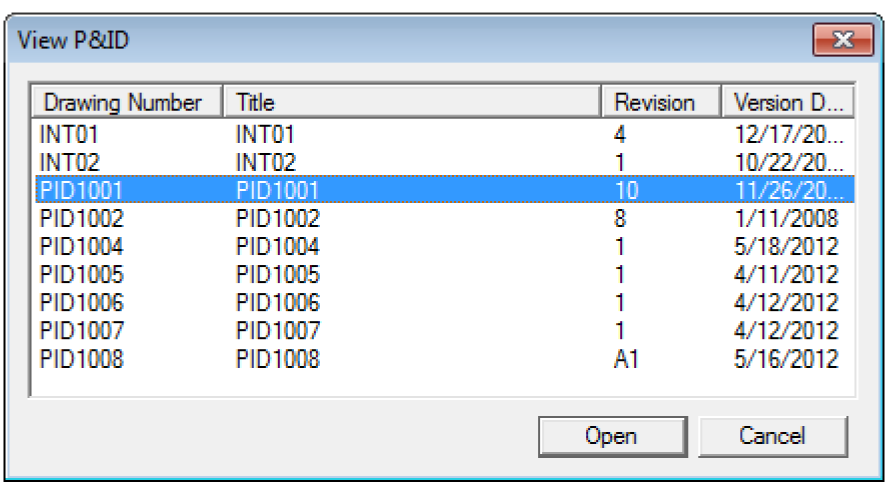
Before Starting this Procedure

- Define your workspace to display **Area01**. In your training plant, select **Area01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

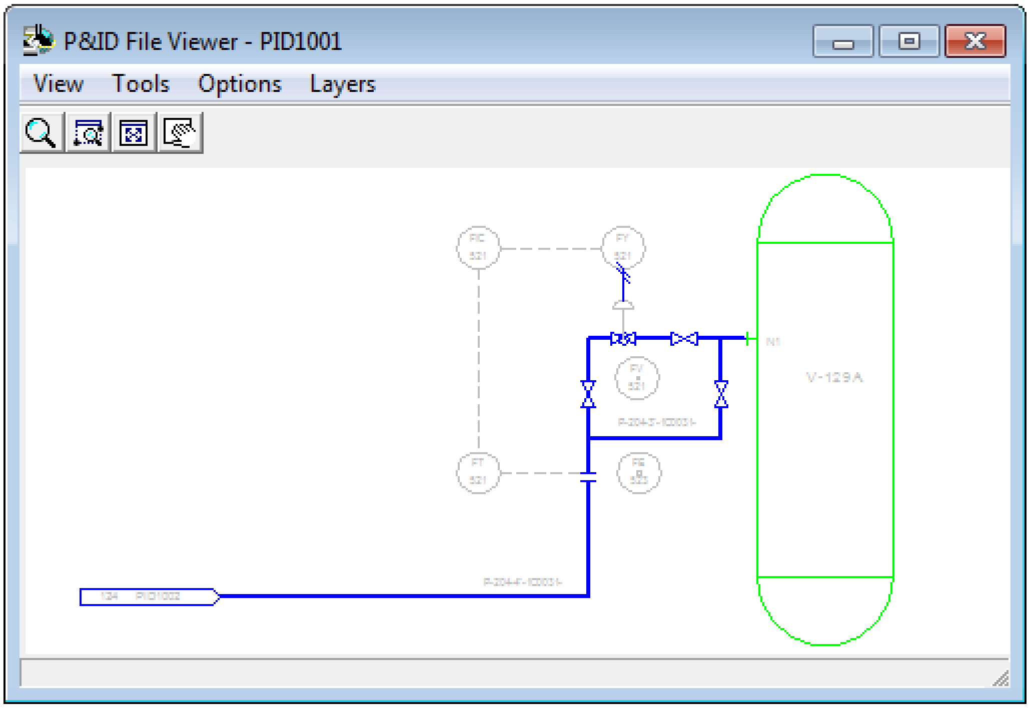
1. Select **SmartPlant > View P&ID** to open the P&ID that contains the design basis of the pipeline to model.

   The View P&ID dialog box appears. The View P&ID dialog box displays a list of the P&IDs available for the model.

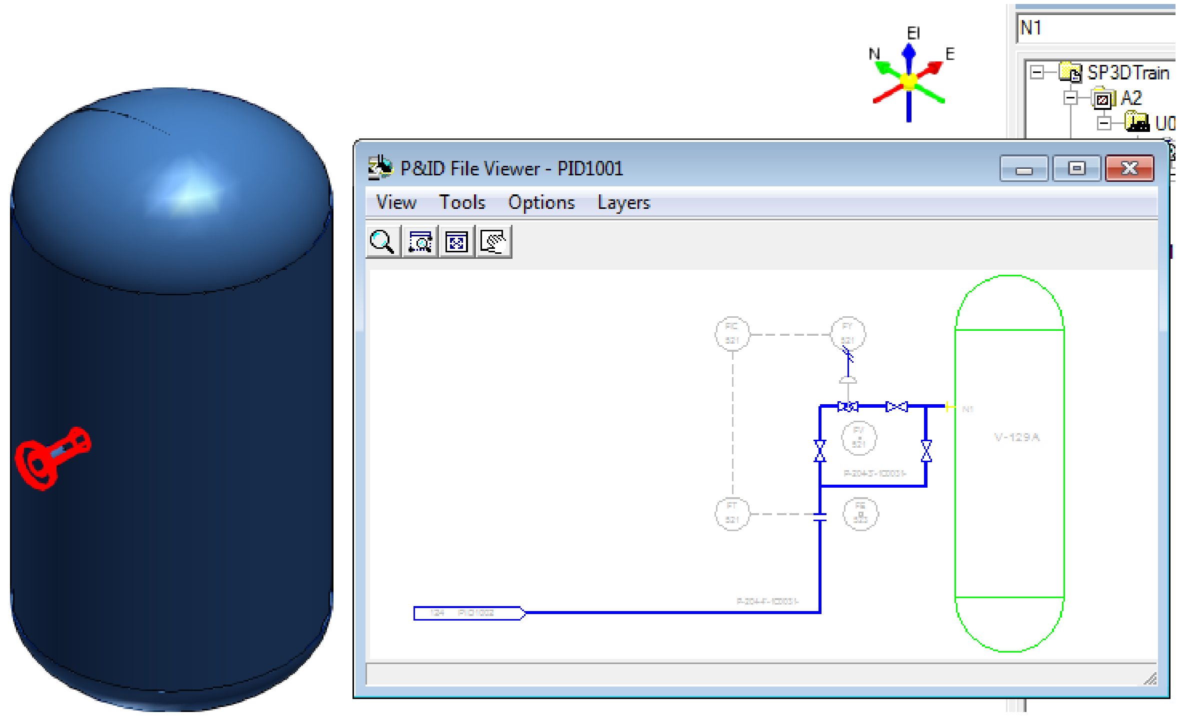
2. Select **PID1001** drawing, and click **Open**.



The P&ID File Viewer window appears.



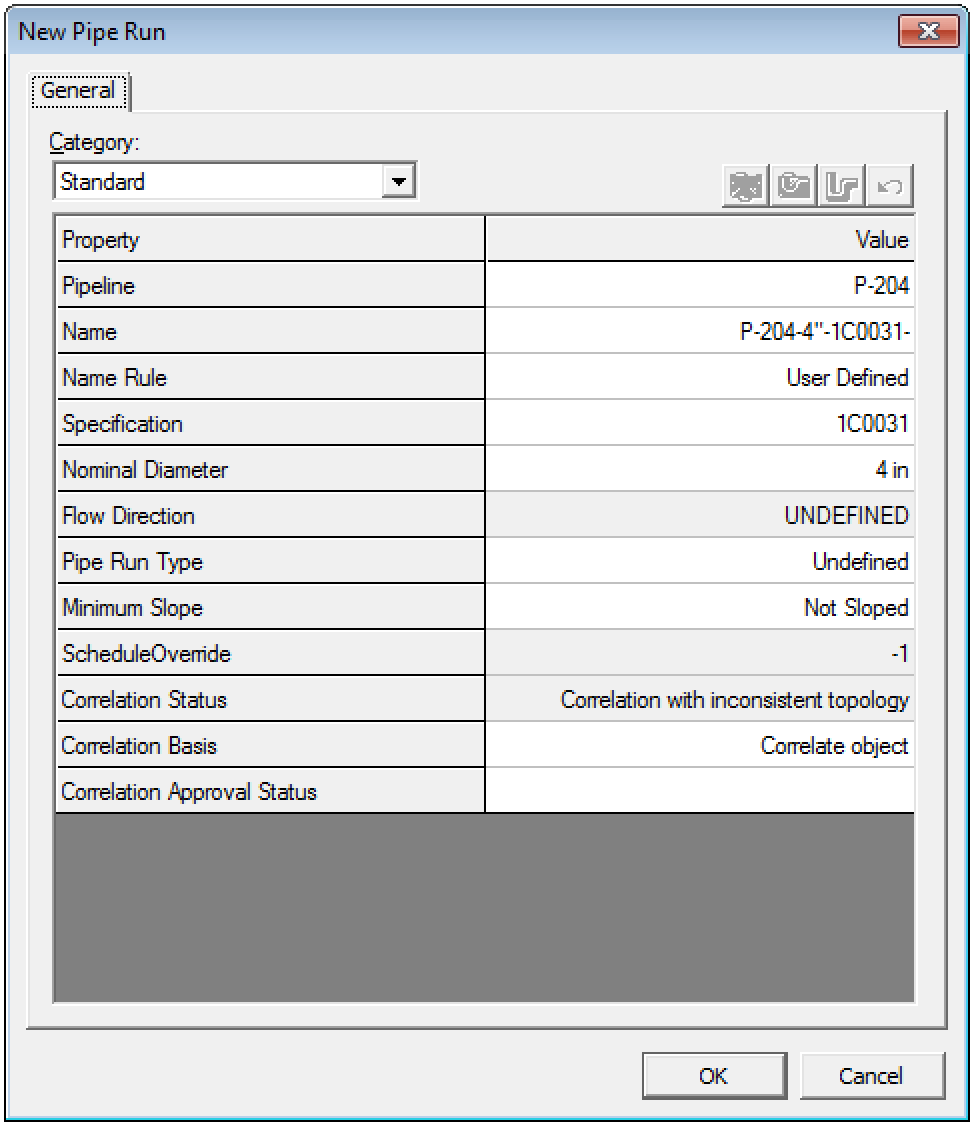
3. In the **Locate Filter** drop-down list, select **All**. This helps you select all the objects in the model and not a specific object.
4. Select the equipment nozzle **N1** from the P&ID File Viewer.
5. Click **Route Pipe**  to start routing the pipe run on the equipment nozzle N1.



The New Pipe Run dialog box appears. The system automatically populates the New Pipe Run dialog box with design basis from P&ID because the equipment nozzle is already correlated.

6. Verify that the **New Pipe Run** dialog box has the following parameters:

Pipeline: P-204
Name: P-204-4"-1C0031
Name Rule: User Defined
Specifications: 1C0031
Nominal Diameter: 4 in
Flow Direction: UNDEFINED
Pipe Run Type: Undefined
Minimum Slope: Not Sloped
Schedule Override: <undefined value>
Correlation Status: Correlation with inconsistent data
Correlation Basis: Correlate object

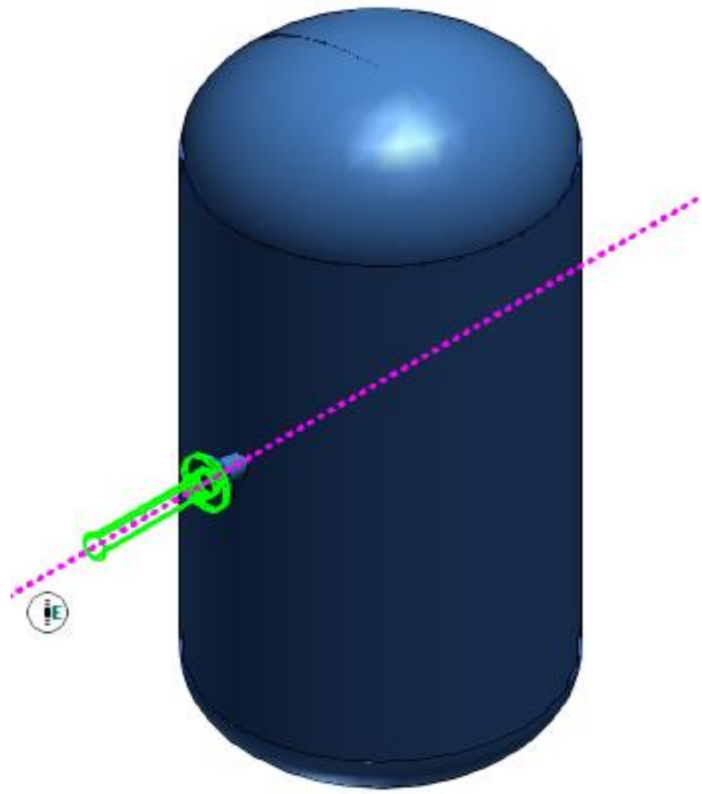


7. Click **OK** to close the dialog box.

   An outline of a pipe run appears in the graphic view starting at pipe nozzle N1.

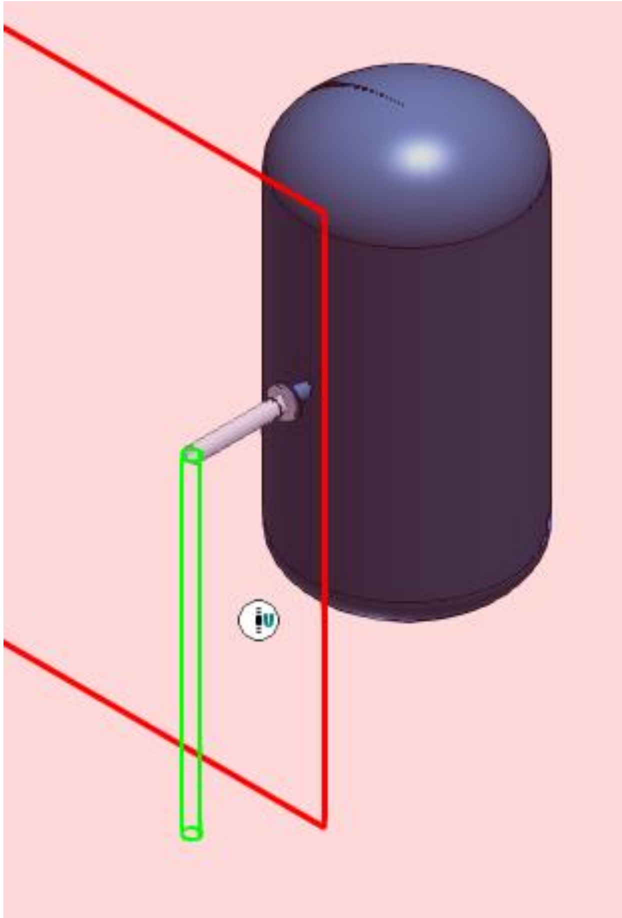
8. Key in **2 ft** in the Length box on the Route Pipe ribbon to lock the length of the pipe run.

9. Move the cursor towards the left until SmartSketch displays the E glyph, which indicates that you are routing the pipe in the west direction.



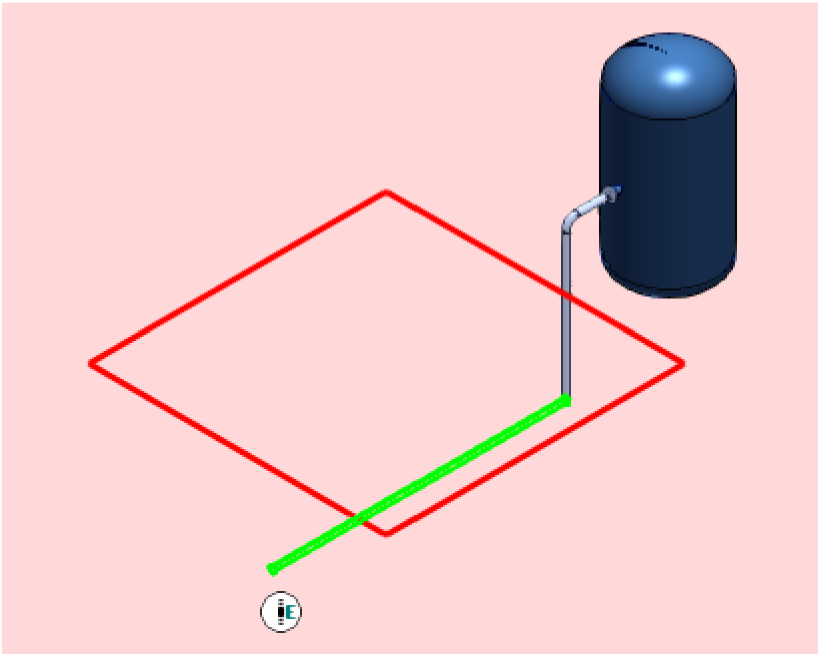
10. Click in the graphic view to place the pipe.
11. Key in **8 ft** in the Length box.
12. Move the cursor downwards until SmartSketch displays the U glyph.

13. Click in the graphic view to place the pipe.



14. Key in **15 ft** in the **Length** box.

15. Move the cursor towards the left until SmartSketch displays the E glyph, and click in the graphic view to place the pipe.

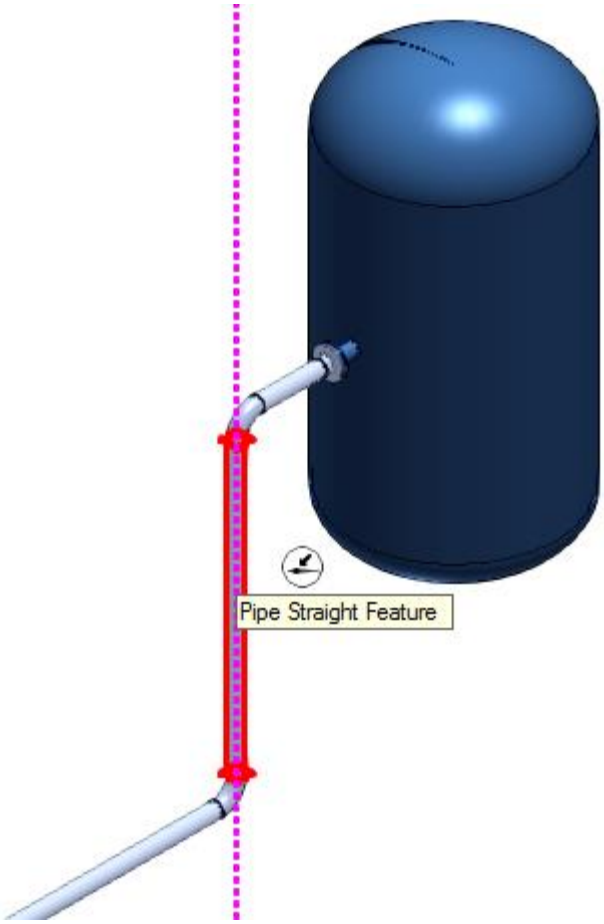


16. Right-click in the graphic view to terminate the Route Pipe command.

Now, continue modeling the pipeline by inserting a bypass line and instruments until the pipe run topology matches the design basis. Perform the following steps:

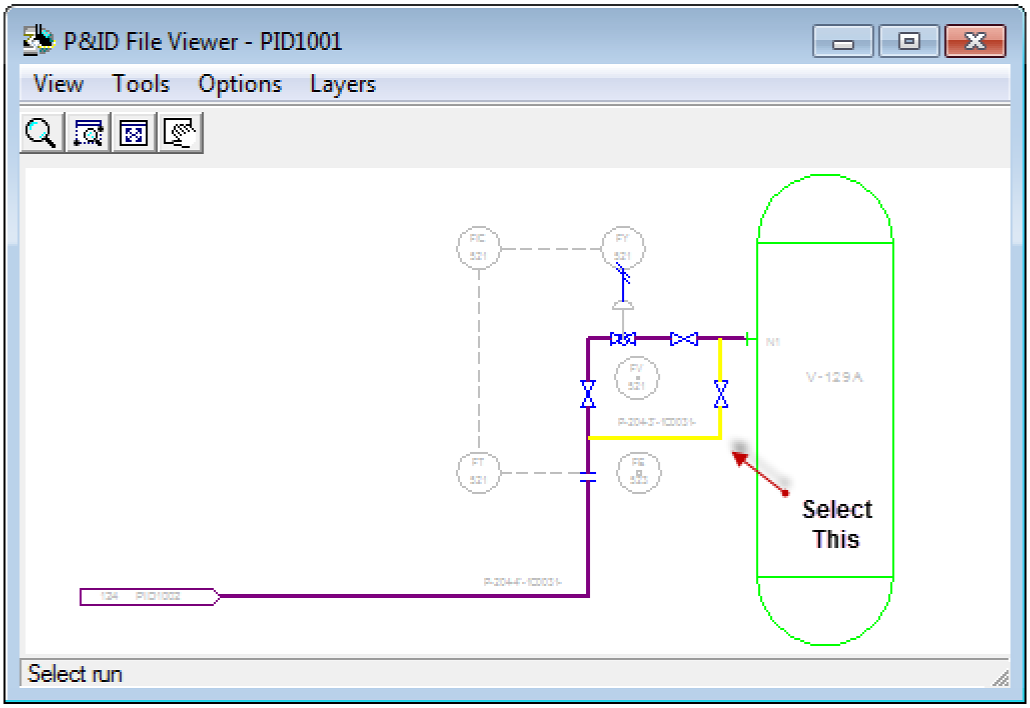
1. Click Route Pipe .

2. Position the cursor along the **Pipe Straight Feature**, and click to define the active placement point.

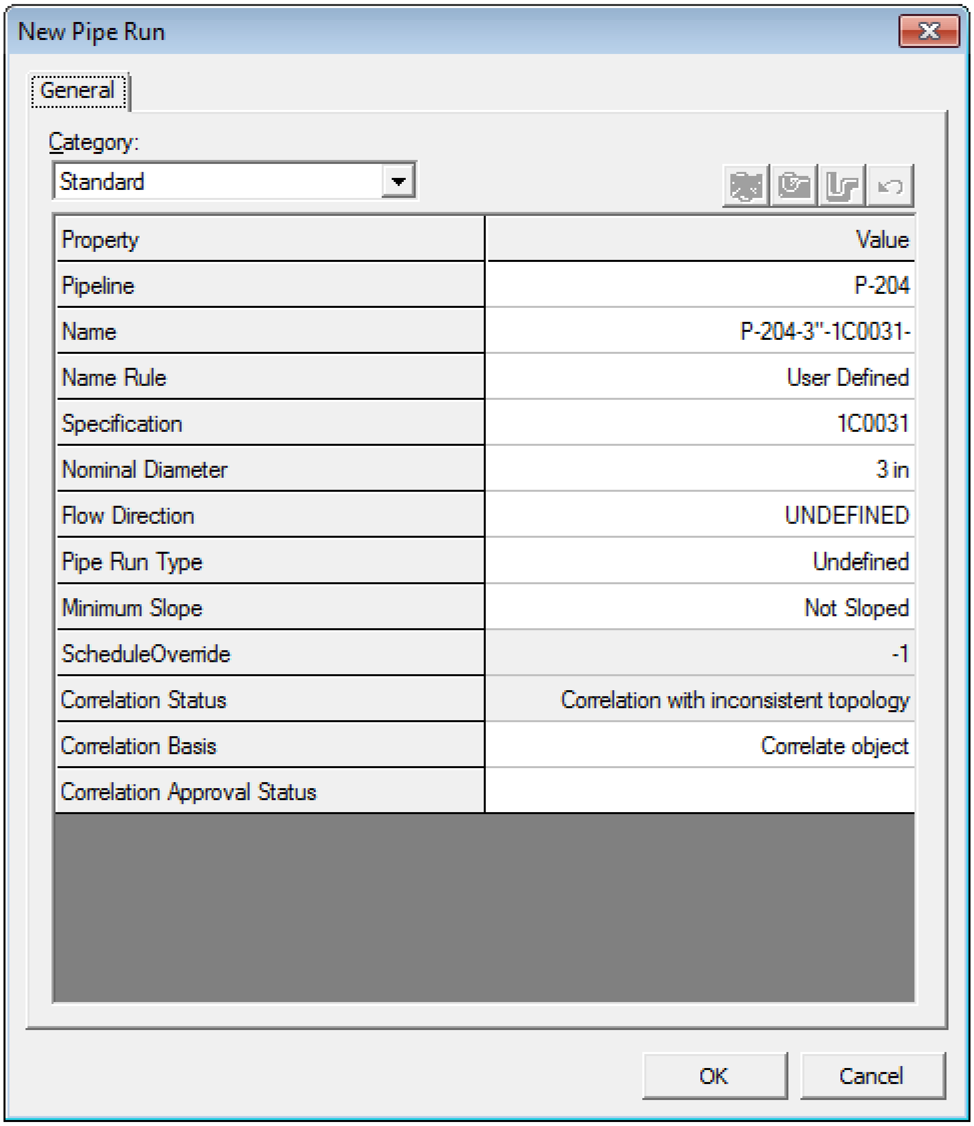


The P&ID File Viewer window appears.

3. In the P&ID File Viewer, select the bypass line.



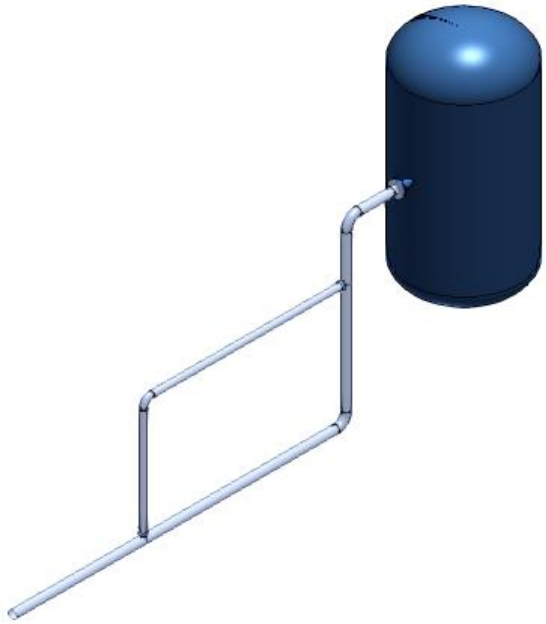
The New Pipe Run dialog box appears. The system populates the New Pipe Run dialog box automatically by using P&ID and design basis.



4. Click **OK** to accept the default values, and close the dialog box.

   You now see an outline of the pipe run in your model.

5. Route the pipe run to complete the bypass line as shown.

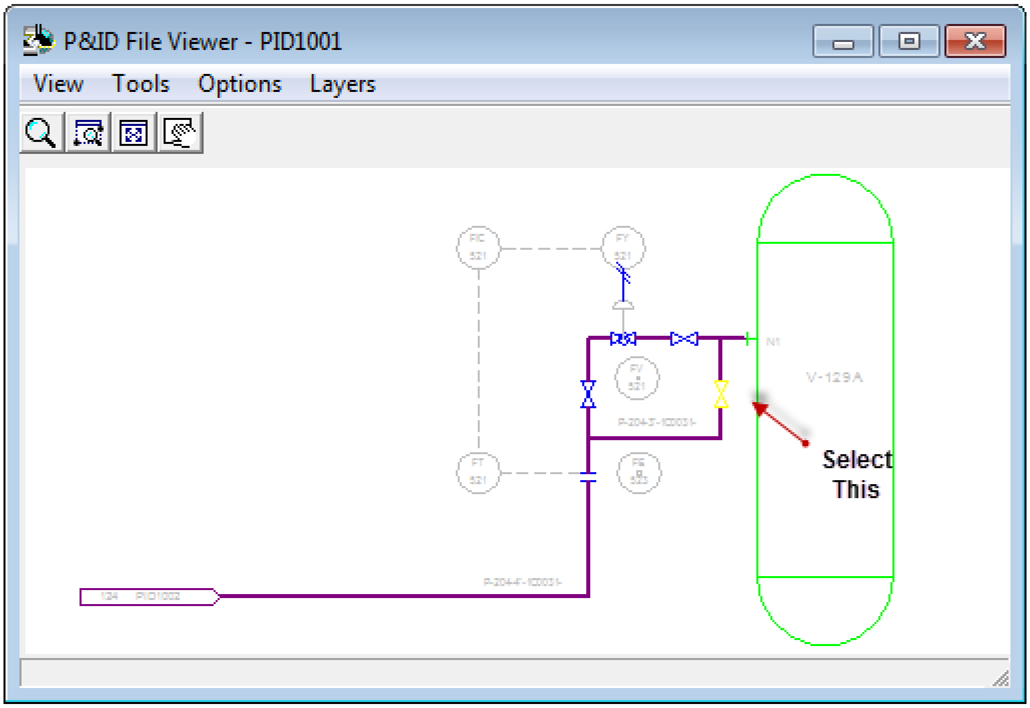


   Now, place a gate valve on the bypass line you have just placed.

6. Click **Insert Component** and use SmartSketch to locate the midpoint glyph of the pipe you have just placed. Use the midpoint as the active placement point for the component.
7. Click on the active placement point of the pipe.

   The P&ID File Viewer window appears.

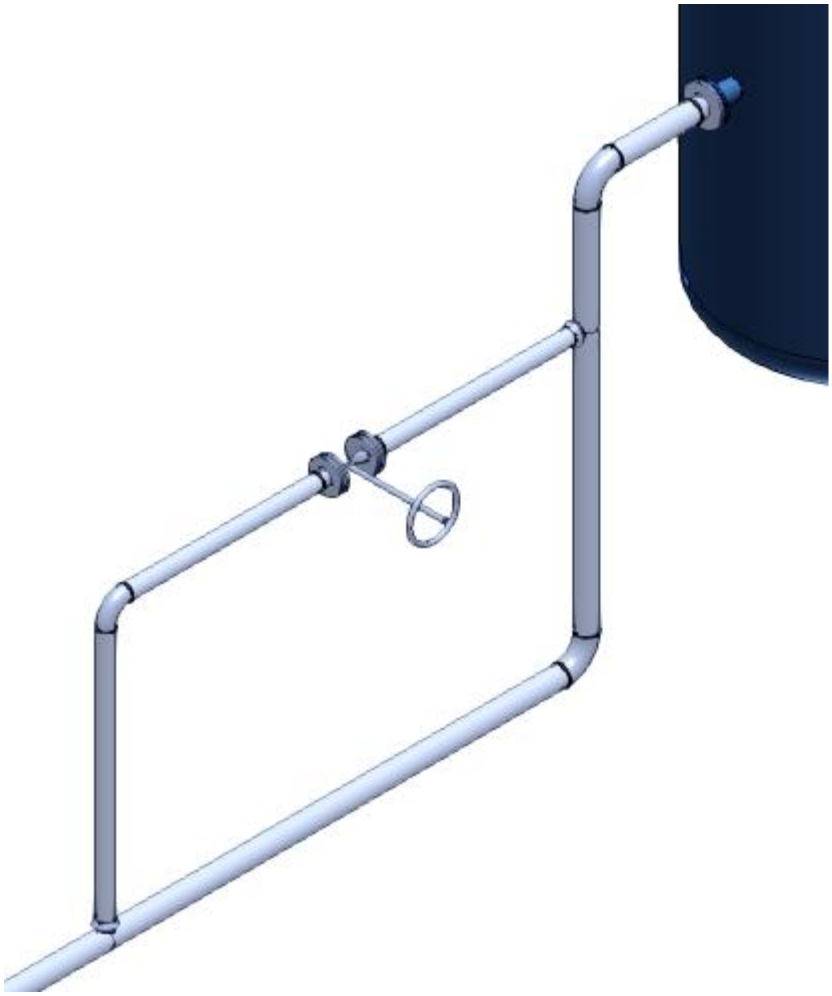
8. In the P&ID File Viewer window, select the gate valve located in the bypass line.



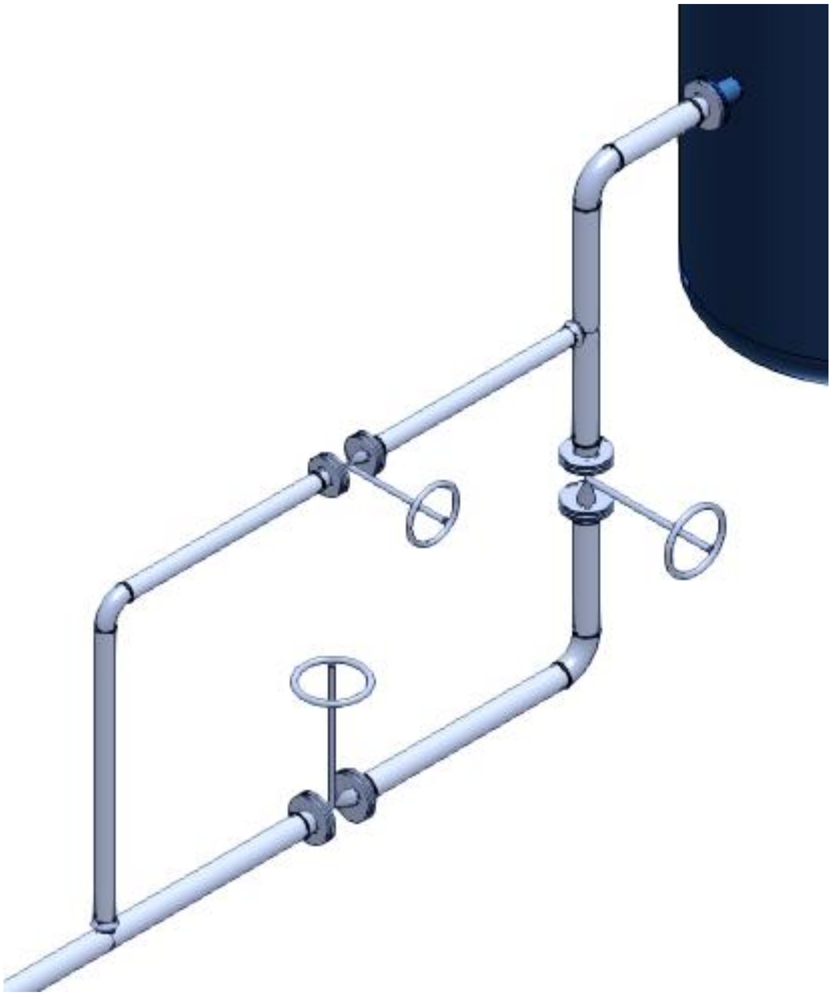
The software uses P&ID and design basis to select the appropriate pipe component.

9. Key in **90 deg** in the **Angle** list to rotate the valve.

10. Click **Finish** to place the gate valve.



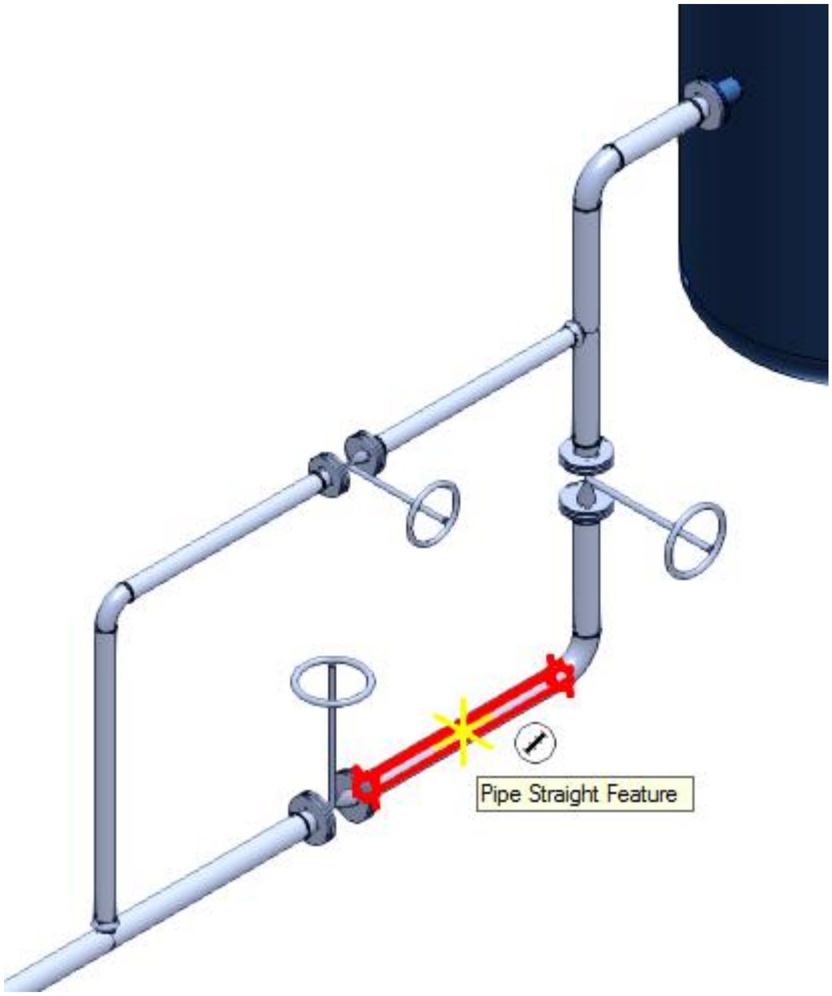
11. Repeat the above steps to place two gate valves.



Now, place a control valve on the pipeline P-204.

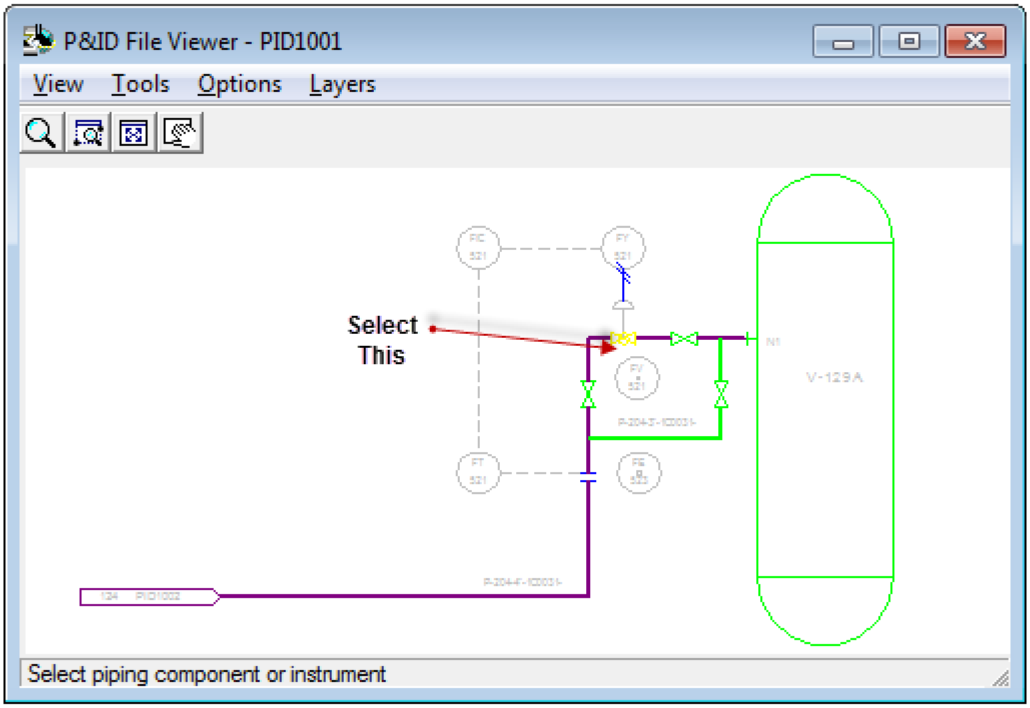
1. Click **Insert Component** . Use the midpoint of the pipeline P-204 as the active placement point for the instrument.

2. Click on the active placement point of the pipe.

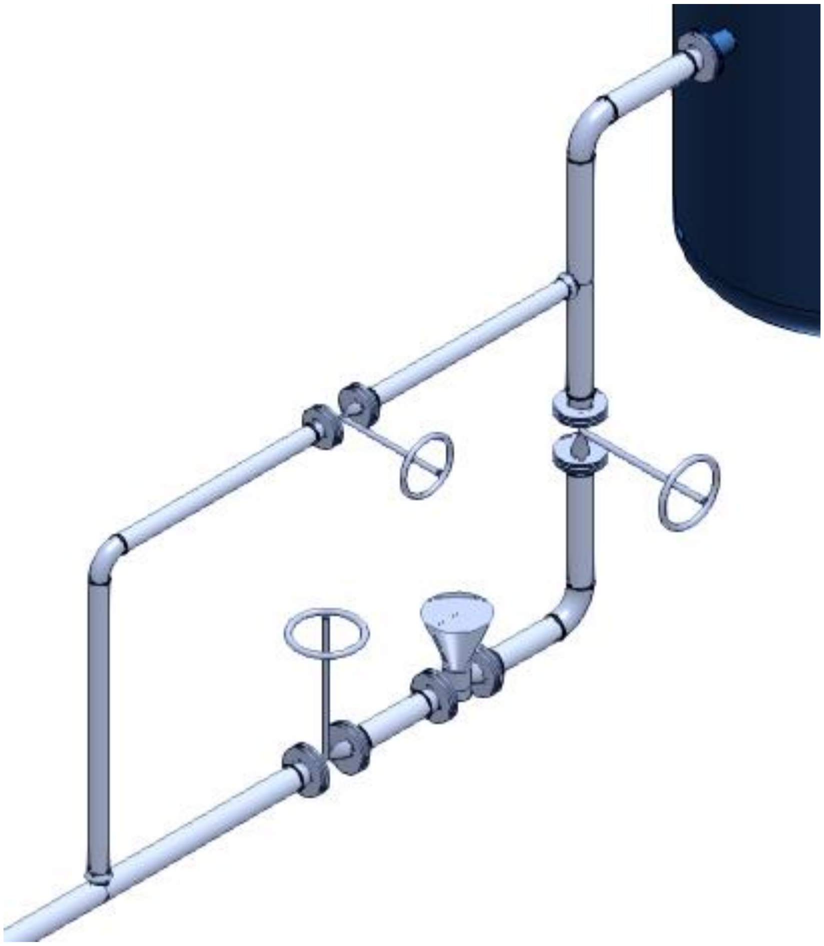


The P&ID File Viewer window appears.

3. In the **P&ID File Viewer** window, select the instrument located in the pipeline.



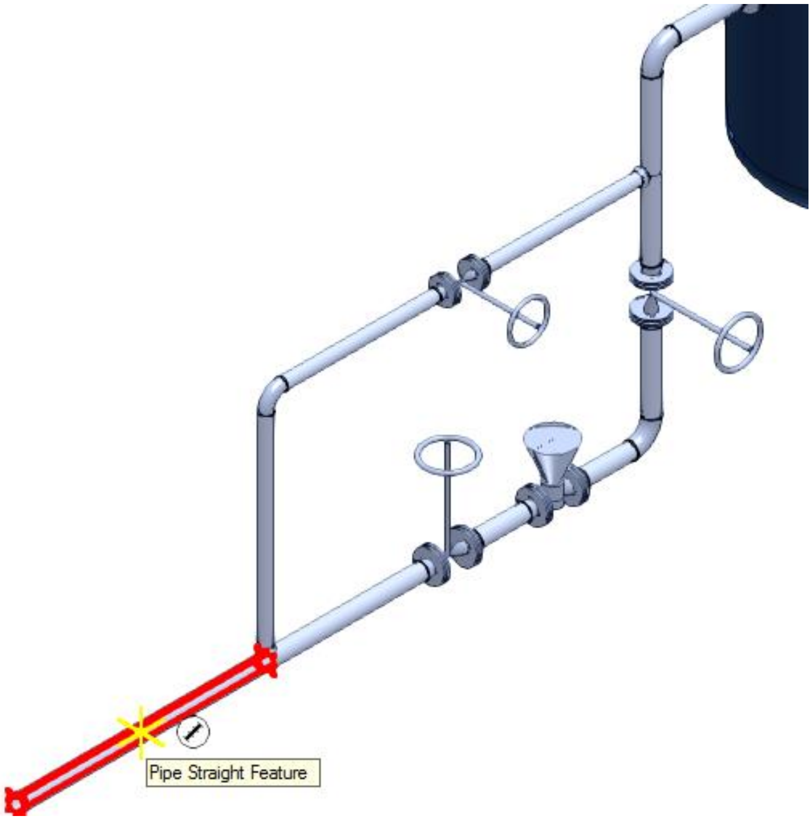
4. Click **Finish** to place the instrument.



Now, place a flow instrument on the pipeline P-204.

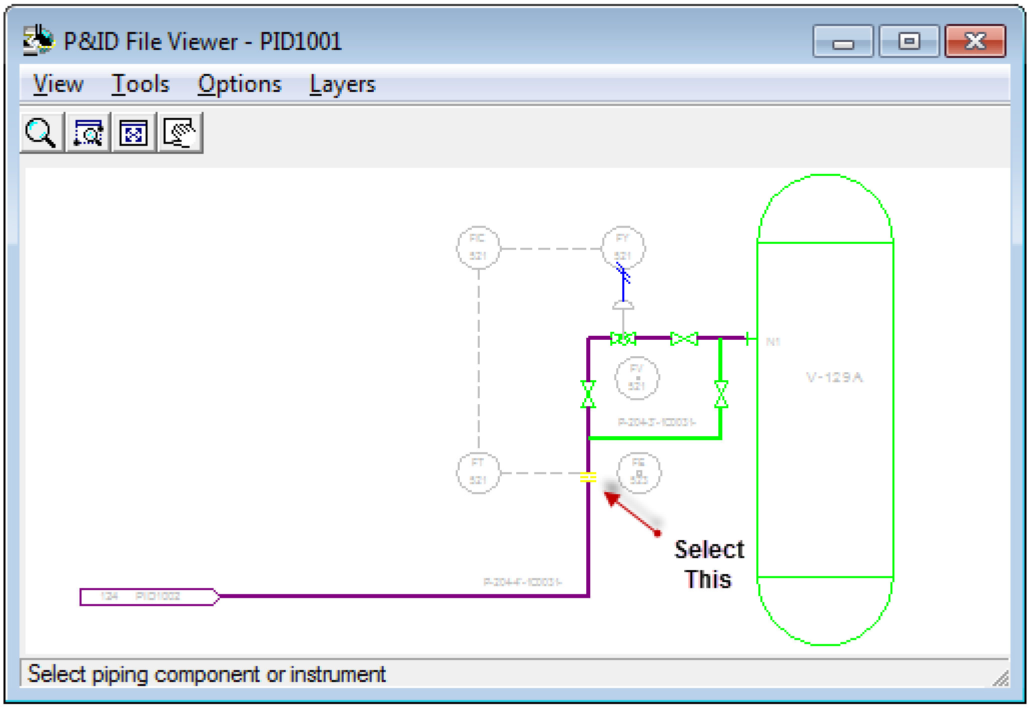
1. Click **Insert Component** . Use the midpoint as the active placement point for the flow instrument.

2. Click on the active placement point of the pipe.

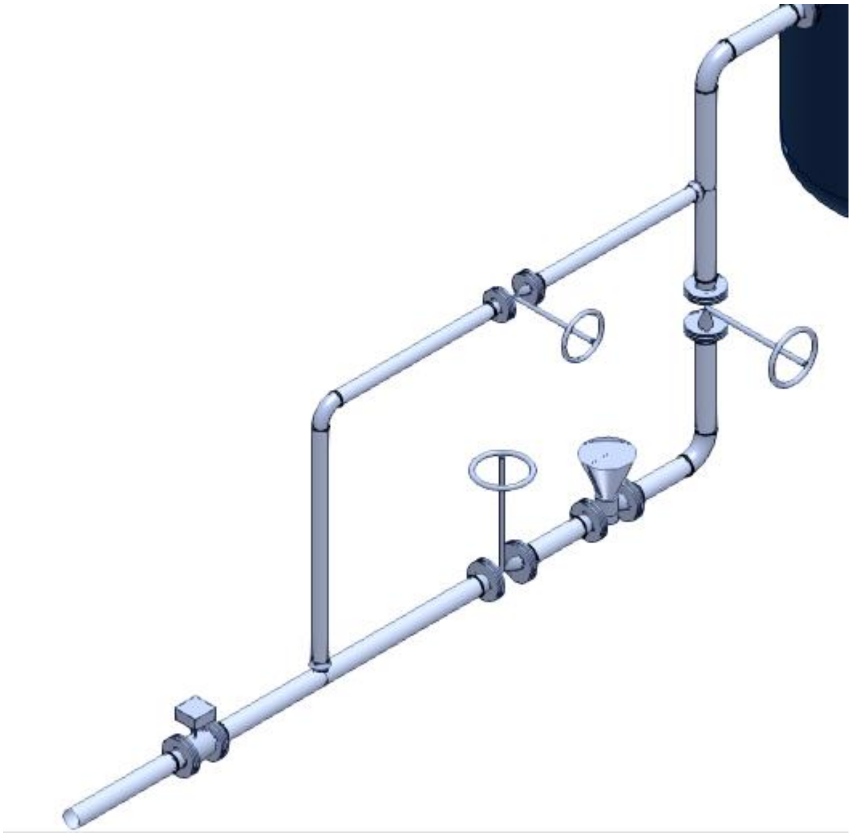


The P&ID File Viewer window appears.

3. In the **P&ID File Viewer** window, select the instrument located in the pipeline.



4. Click **Finish** to place the instrument.

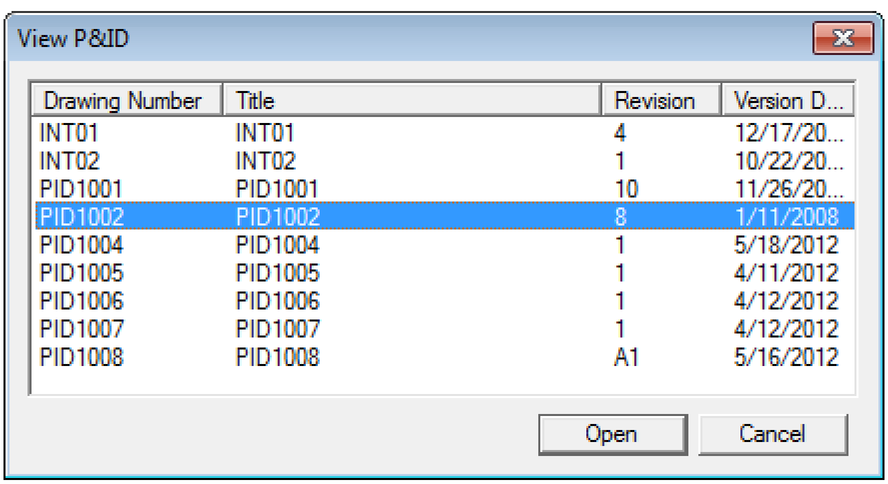


Now, continue routing the pipeline using the off-page connector (OPC).

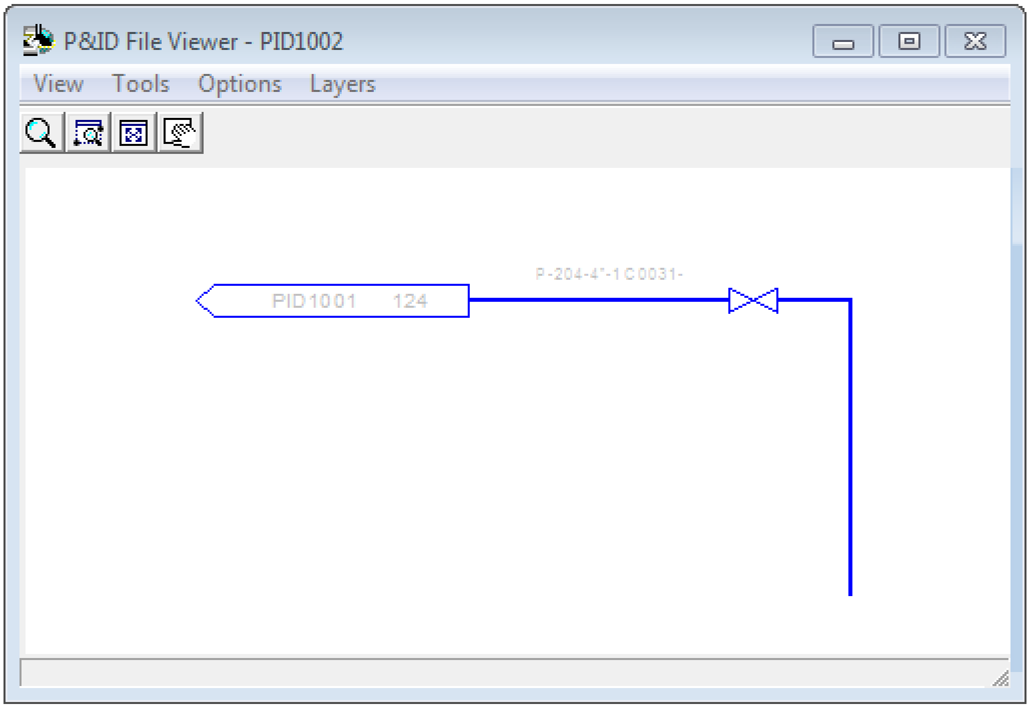
1. Click **SmartPlant > View P&ID** to open the continuation P&ID drawing.

   The View P&ID dialog box appears.

2. Select PID1002 drawing, and click **Open**.



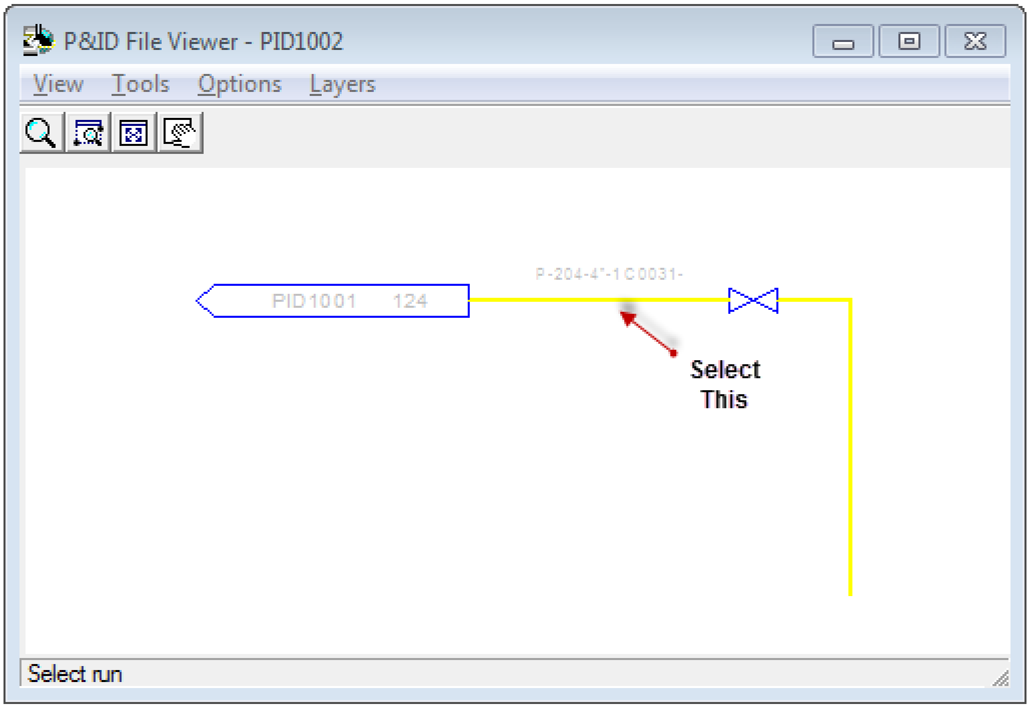
The P&ID File Viewer window appears.



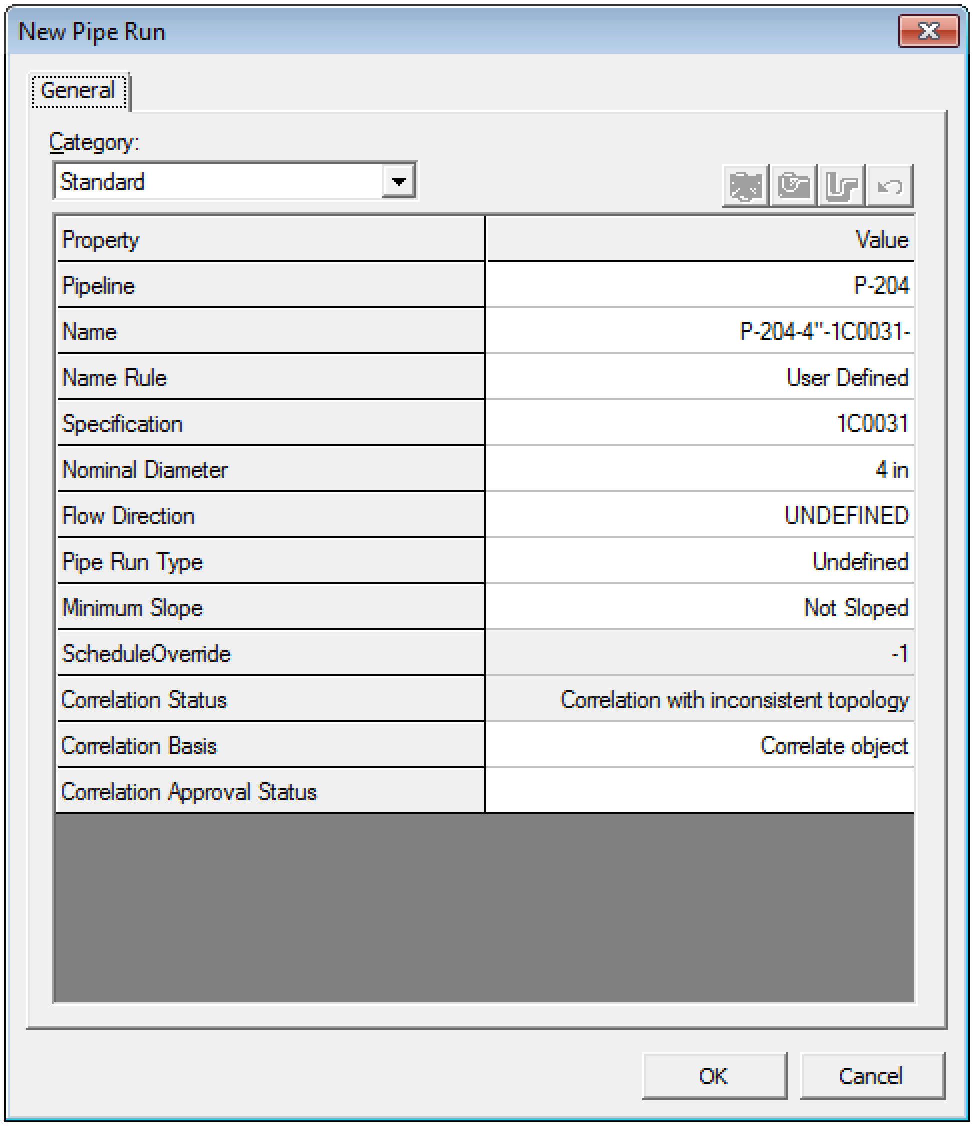
3. Click **Route Pipe** .
4. Select **<Select From P&ID>** on the **Run** list.

   The P&ID File Viewer window appears.

5. In the **P&ID File Viewer**, select the pipeline **P-204**.



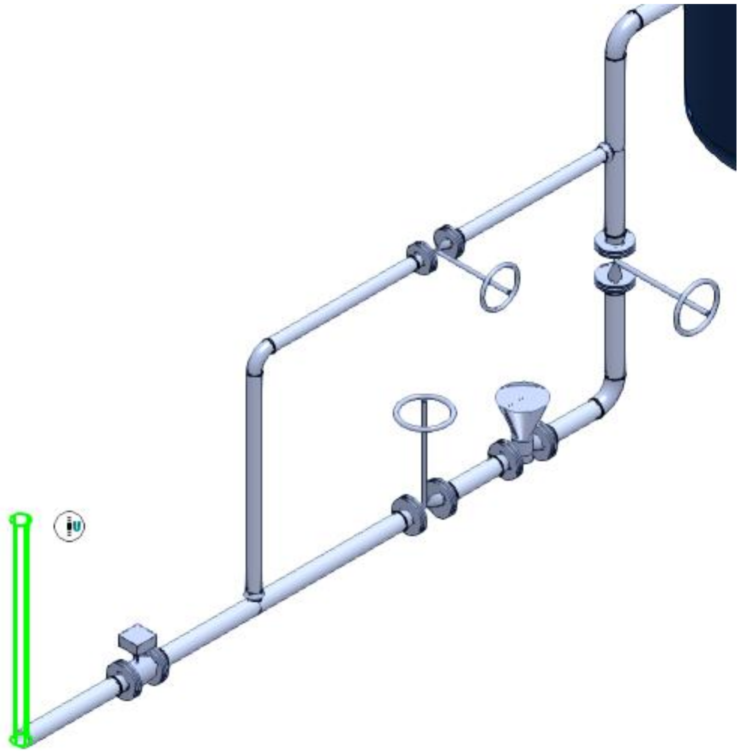
The New Pipe Run dialog box appears. The system populates the New Pipe Run dialog box automatically by using P&ID and design basis.



6. Click **OK** to accept the default values and close the dialog box.
7. Select the end of the pipe to start the routing.

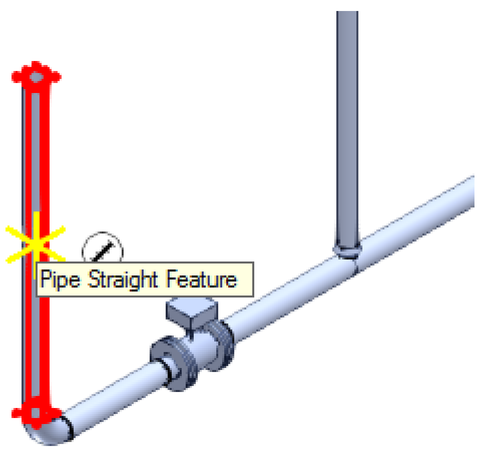
   You now see an outline of the pipe run in your model.

8. Click to place the pipe run.



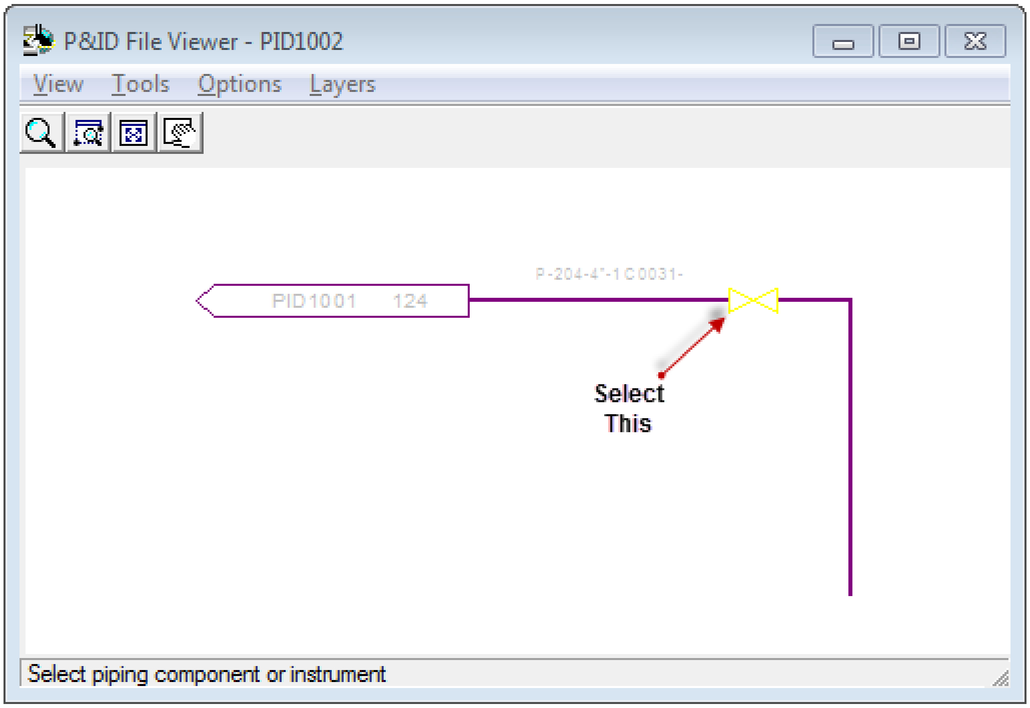
9. Click **Insert Component** .

10. Position the cursor along the **Pipe Straight Feature**, and click to define the active placement point.



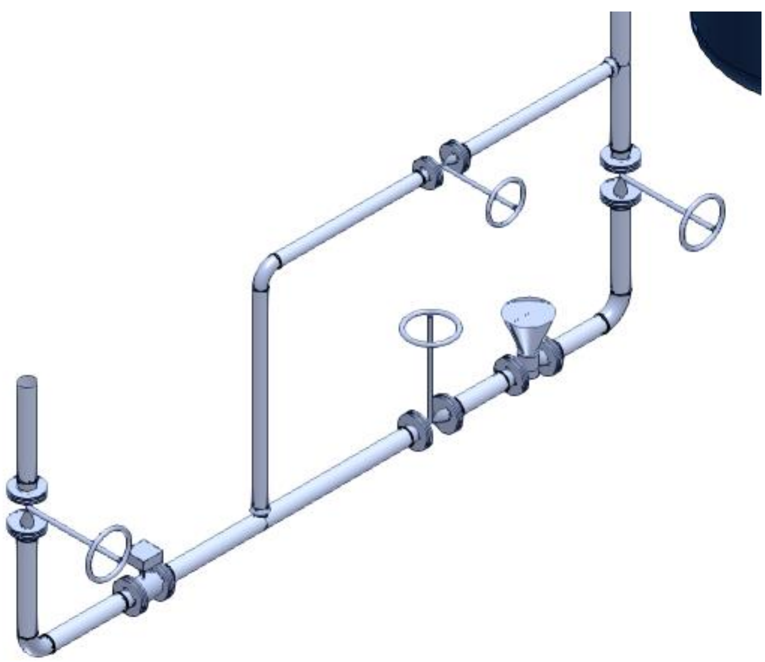
The P&ID File Viewer window appears.

11. In the **P&ID File Viewer**, select the gate valve.



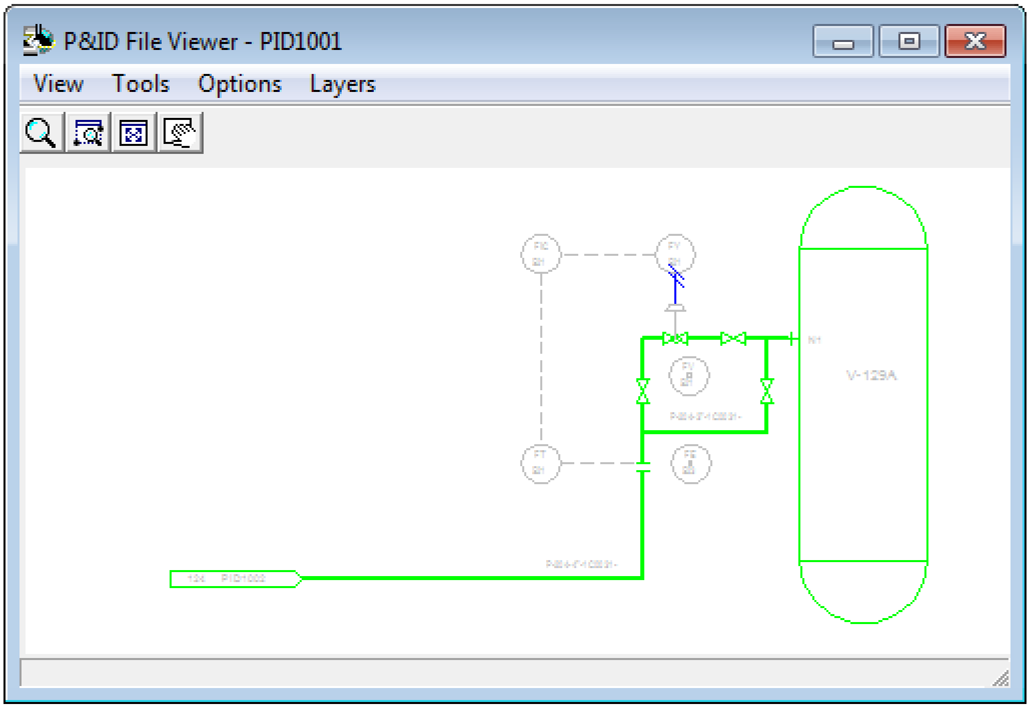
You will now see an outline of a gate valve (base part) with mating flanges at the active placement point. The software uses P&ID and the design basis to select the appropriate component.

12. Key in **90 deg** in the **Angle** list to rotate the valve.
13. Click **Finish** to place the pipe component.



14. Open the **P&ID PID1001** drawing.

    The pipe section you placed is highlighted in green, which indicates that the pipe section and the OPC are correlated now and its properties and topology matches the design basis.



For more information related to placing instrument, refer to the Insert an Instrument topic in the user guide PipingUsersGuide.pdf.

Routing Sloped Pipe

Objective

By the end of this session, you will be able to:

- Route a sloped pipe.

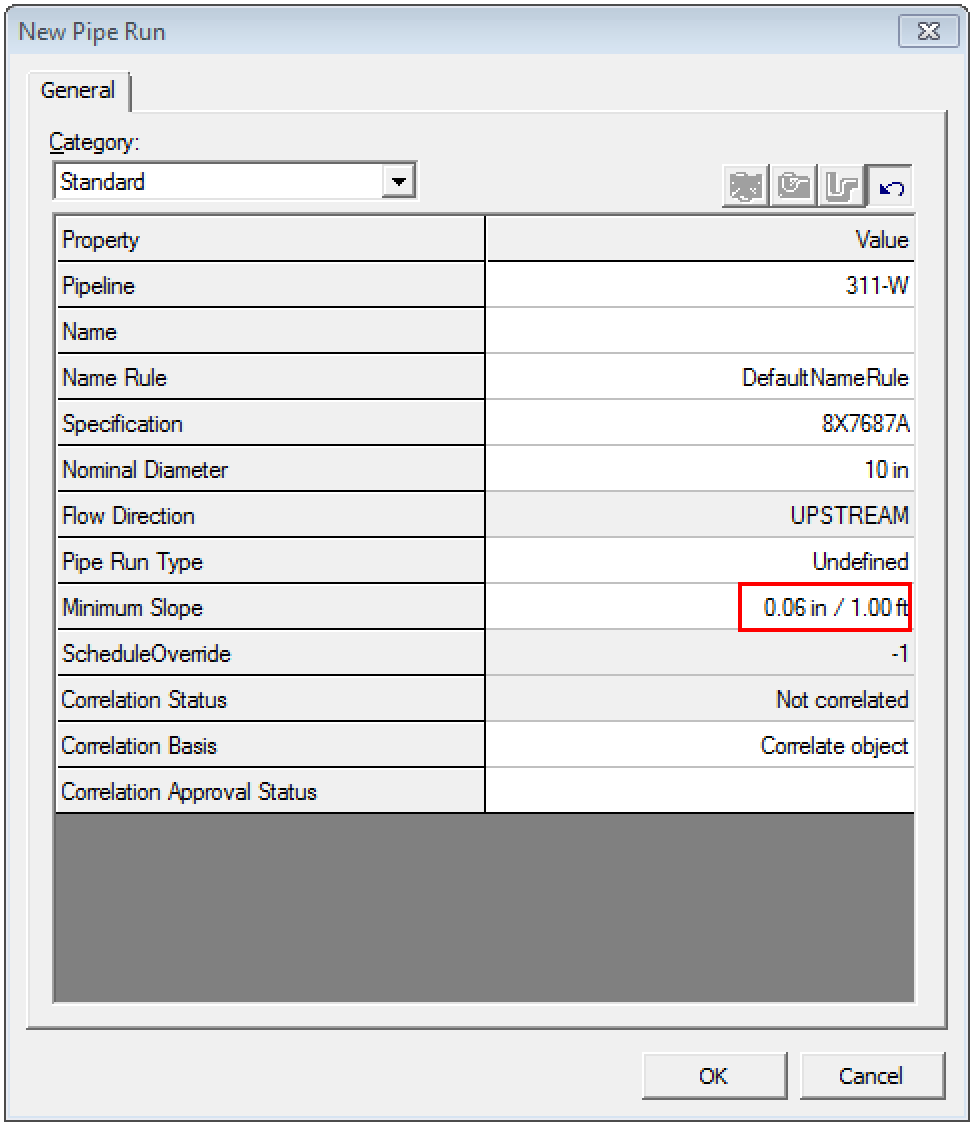
Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

Overview

Underground piping collects drains from funnels or catch basins and transports them to a disposal point. Since there is no pressure in this piping system, the pipe must slope for flow.

You use **Route Pipe**  to route a sloped pipe. The basic steps to route a sloped pipe or an underground pipe are similar to what you follow to route a straight pipe. The only difference is that you have to define a slope for the pipe. For this, you use the **Minimum Slope** field of the **New Pipe Run** dialog box. You define the slope by either defining the angle of the slope or the fraction to which you want the pipe to be sloped.

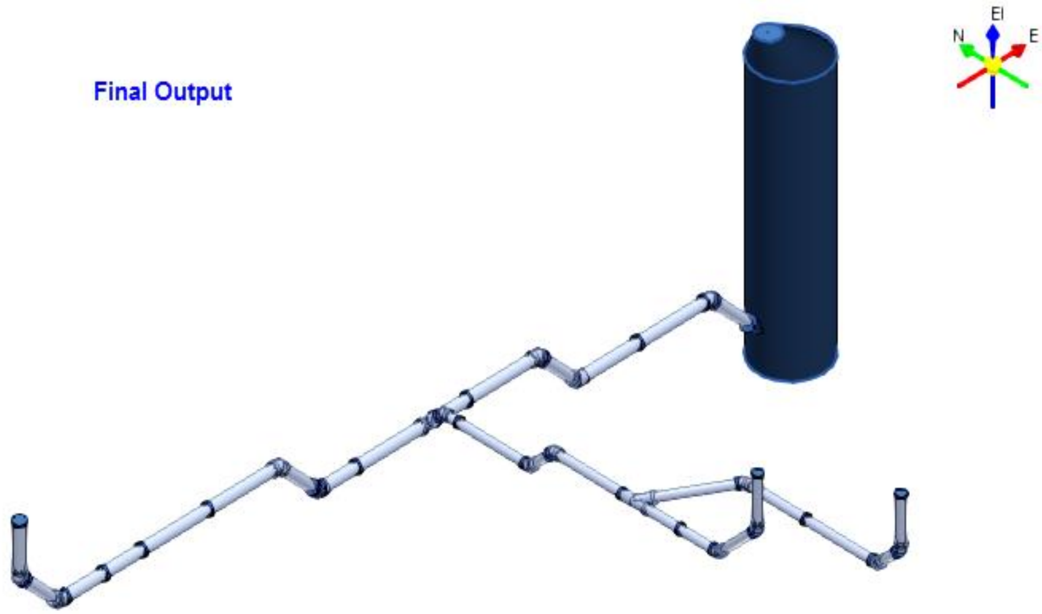


This session covers the procedure for routing a sewer line to a catch basin with laterals and cleanouts.

Routing a Sewer Line to a Catch Basin

Objective

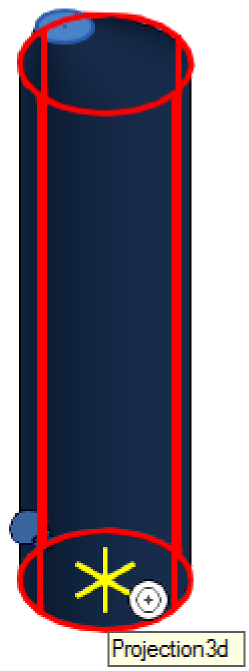
In this exercise you will be routing underground pipeline **311-W** from cleanouts to a catch basin **B-101** in **Unit U03** of your workspace. The routed underground piping system should resemble the highlighted area below.



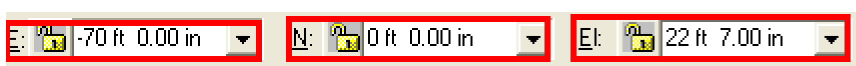
Before Starting this Procedure

- Define your workspace to display **Unit U03** and coordinate system **U03 CS**. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Locate the catch basin **B-101** from the **Workspace Explorer**.
2. Activate **PinPoint** using **Tools > PinPoint**.
3. Click Route Pipe to start routing the main sewer pipe.
4. Select **Reposition Target** on the **PinPoint** ribbon. The **Reposition Target** option helps define a reference point to route the pipeline.
5. Place the **PinPoint** target at the bottom center of the catch basin.



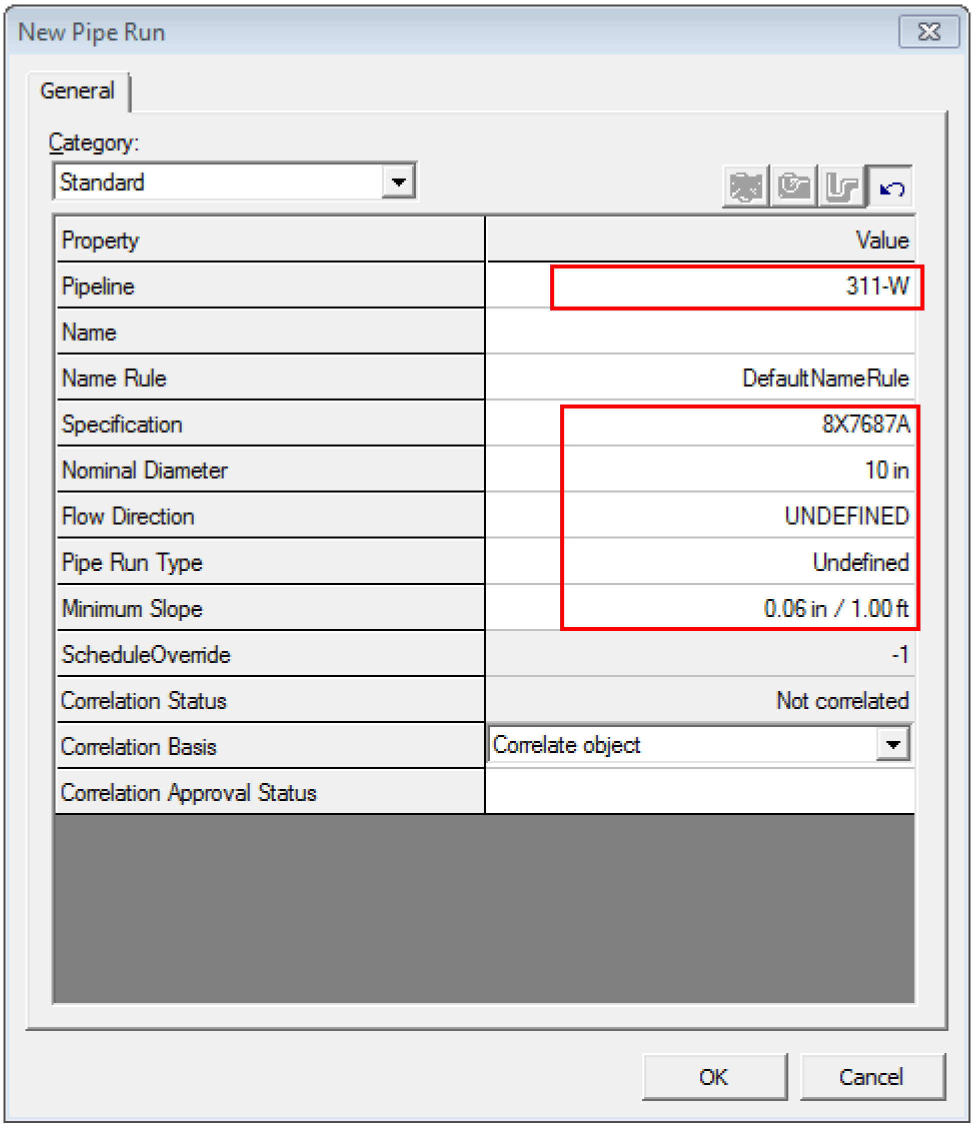
6. On the **PinPoint** ribbon, key in **-70 ft** for **E**, **0 ft** for **N** and **22 ft 7 inches** for **EL** to define the starting point of the route.



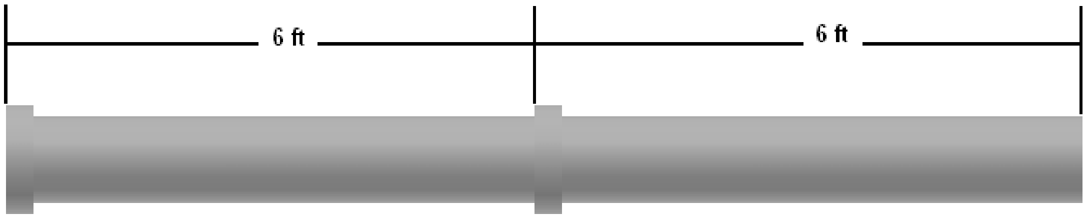
The New Pipe Run dialog box appears.

7. Set the following specifications on the **New Pipe Run** dialog box, and click **OK**.

Pipeline: 311-W
Name Rule: DefaultNameRule
Specification: 8X7687A
Nominal Diameter: 10 in
Flow Direction: DOWNSTREAM
Pipe Run Type: Undefined
Minimum Slope: 0.0625 in /1.0 ft
Schedule Override: <undefined value>
Correlation Status: Not correlated
Correlation Basis: Correlate object



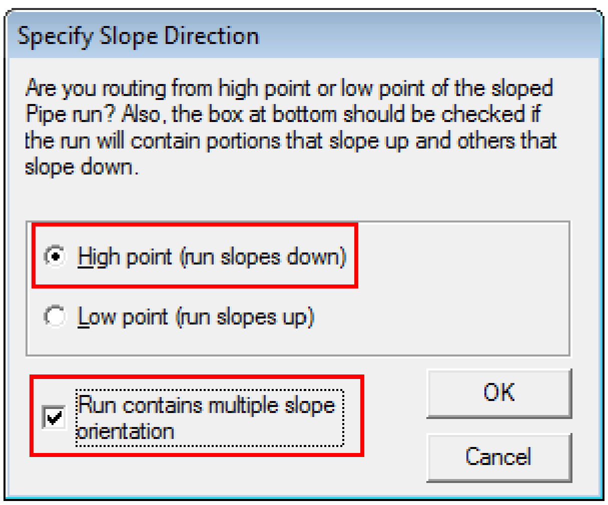
NOTE In this example, **Pipe Specification 8X7687A** contains **Vitrified Clay Pipe Stock** with differing end preparations at the two ends. An example of this is underground piping, where one end is a mechanical joint spigot end, while the other end is a mechanical joint bell end. Furthermore, a pipe of maximum **6 ft** length is defined for this bell and spigot pipe, which the system can use to place the Bell and Spigot joint automatically.



The Specify Slope Direction dialog box appears.

You need to route a pipe that slopes downwards.

8. Select the **High point (run slopes down)** option and the **Run contains multiple slope orientation** check box.

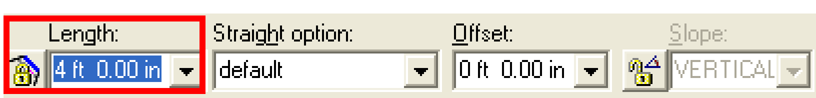


NOTES

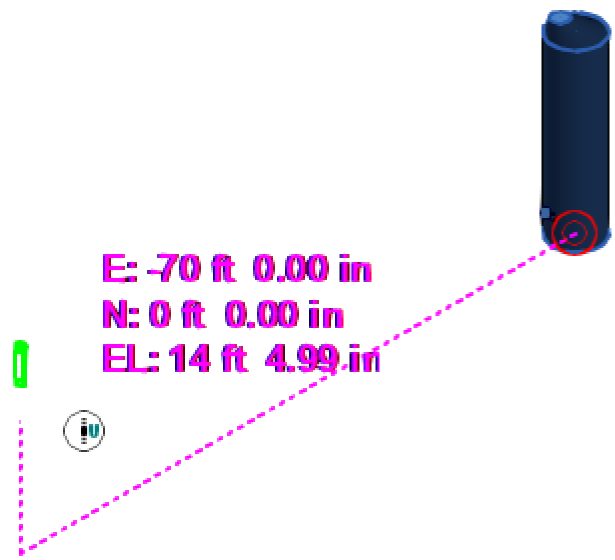
- The **Run contains multiple slope orientation** check box indicates that the pipe run will be sloped from both the up and down directions.
- After you specify that a pipe run has multiple slope orientations, you cannot clear the selection.

9. Click **OK**.

10. Key in **4 ft** in the **Length** box to define the length of the pipe. Make sure the **Plane** constraint is set to **No Plane** .



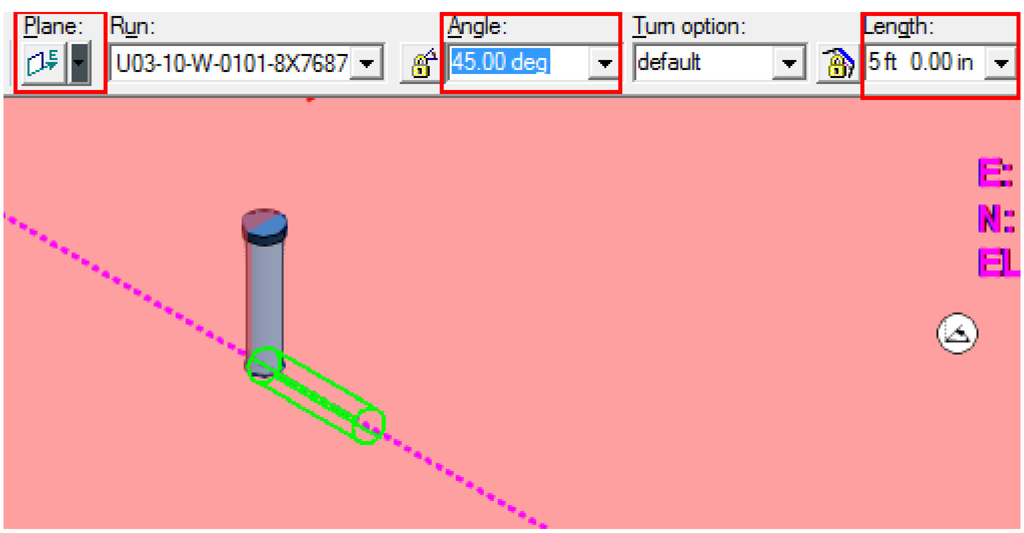
11. Select the **Lock Slope** option on the **Route Pipe** ribbon to unlock the slope constraint and position the cursor point down from the starting point. SmartSketch will display a U glyph. This glyph indicates that you are going down the Z-axis.



12. Click in the graphic view to accept the placement of this pipe.

13. Route a pipe by using the following specifications on the Route Pipe ribbon:

    Length: 5 ft, to constrain the length of the pipe to 5 ft
    Plane: Elevation Plane: East-West
    Angle: 45 deg

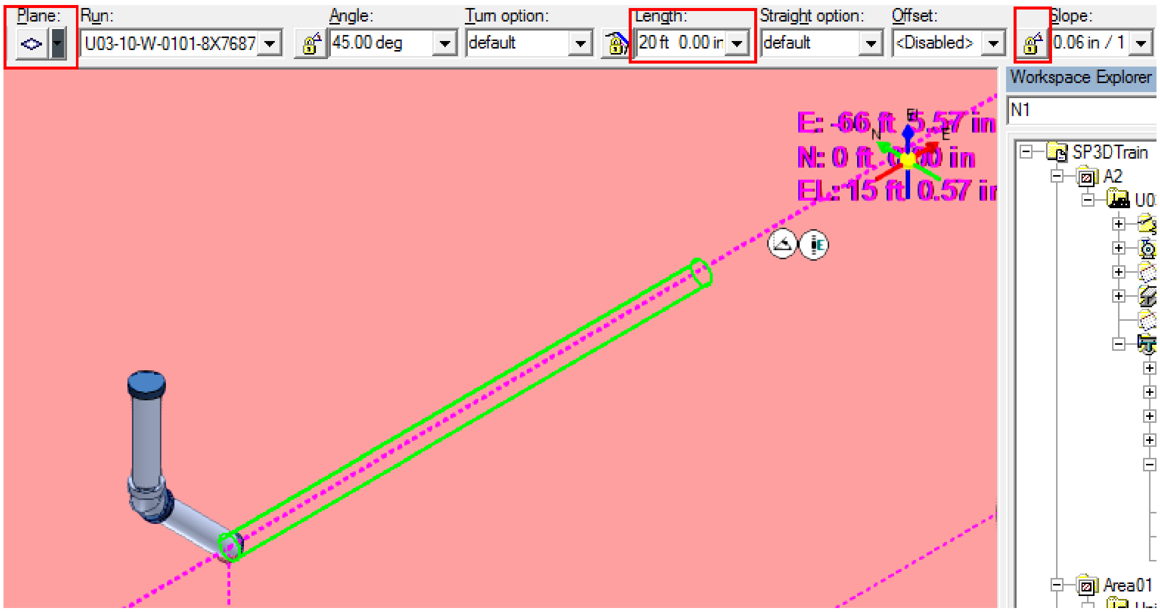


14. Click in the graphic view to accept the placement of this pipe.

15. Now, set the following specifications on the Route Pipe ribbon:

Length: 20 ft
Plane: Plan-Plane

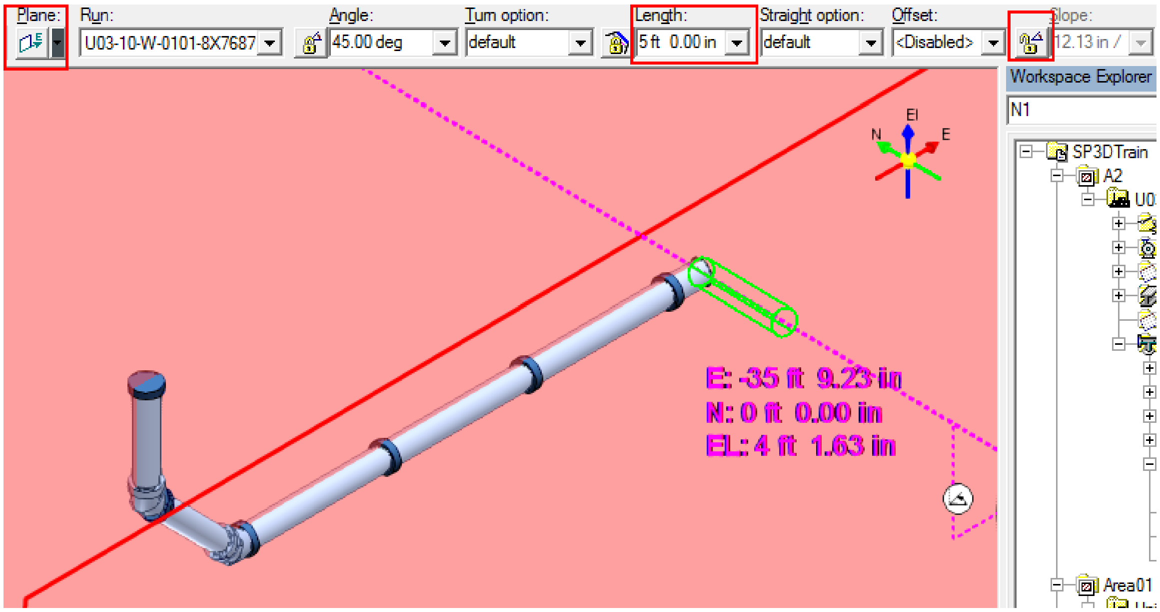
16. Select the **Lock Slope** option on the Route Pipe ribbon to constrain the routing to the minimum slope value and position the cursor point until the SmartSketch indicator displays an E glyph. This glyph indicates that you are going in the easting direction E.



17. Click in the graphic view to accept the placement of this pipe.
18. Set the following specifications on the **Route Pipe** ribbon:

Length: 5 ft
Plane: Elevation Plane: East-West
Angle: 45 deg

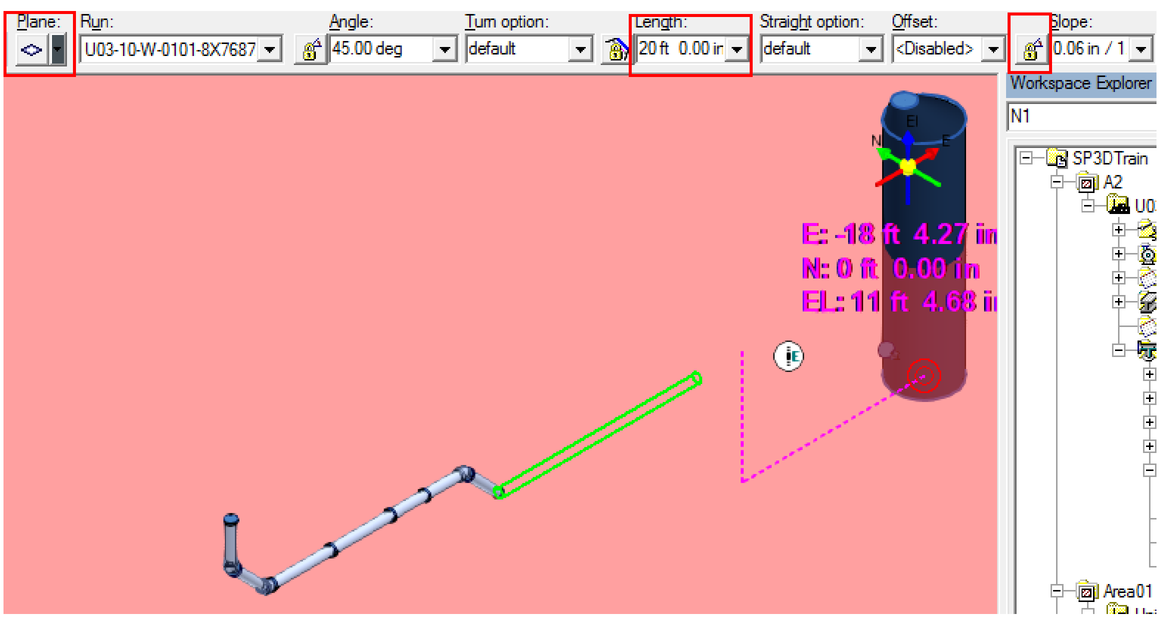
19. Select the **Lock Slope** option on the **Route Pipe** ribbon to unlock the slope constraint to route the sloped pipe.



20. Click in the graphic view to place the 45 deg elbow and a piece of the pipe.
21. Now, set the following specifications on the Route Pipe ribbon to route a pipe in the easting direction:

    Length: 20 ft
    Plane: Plan-Plane

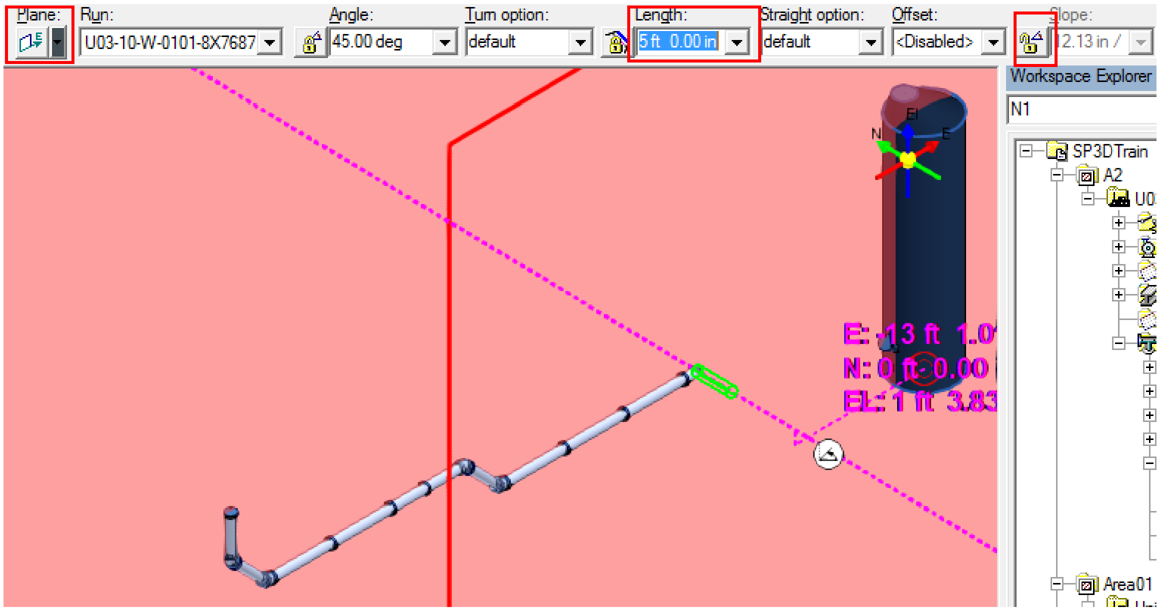
22. Select the **Lock Slope** option on the **Route Pipe** ribbon to constrain the routing to the minimum slope value and position the cursor until the SmartSketch indicator displays an E glyph. This glyph indicates that you are going in the easting direction.



23. Click in the graphic view to accept the placement of this pipe.
24. Now, route a **5 ft** pipe by using the following specifications on the **Route Pipe** ribbon:

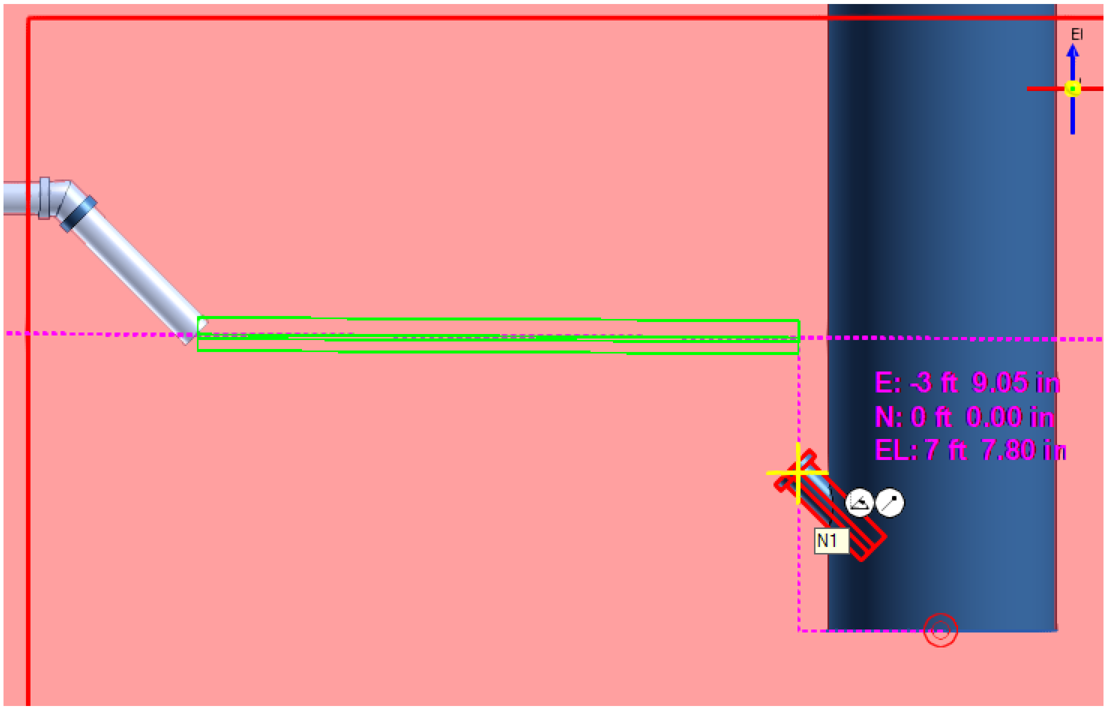
    Plane: Elevation Plane: East-West
    Angle: 45 deg

25. Select the **Lock Slope** option on the **Route Pipe** ribbon to unlock the slope constraint and route the sloped pipe.



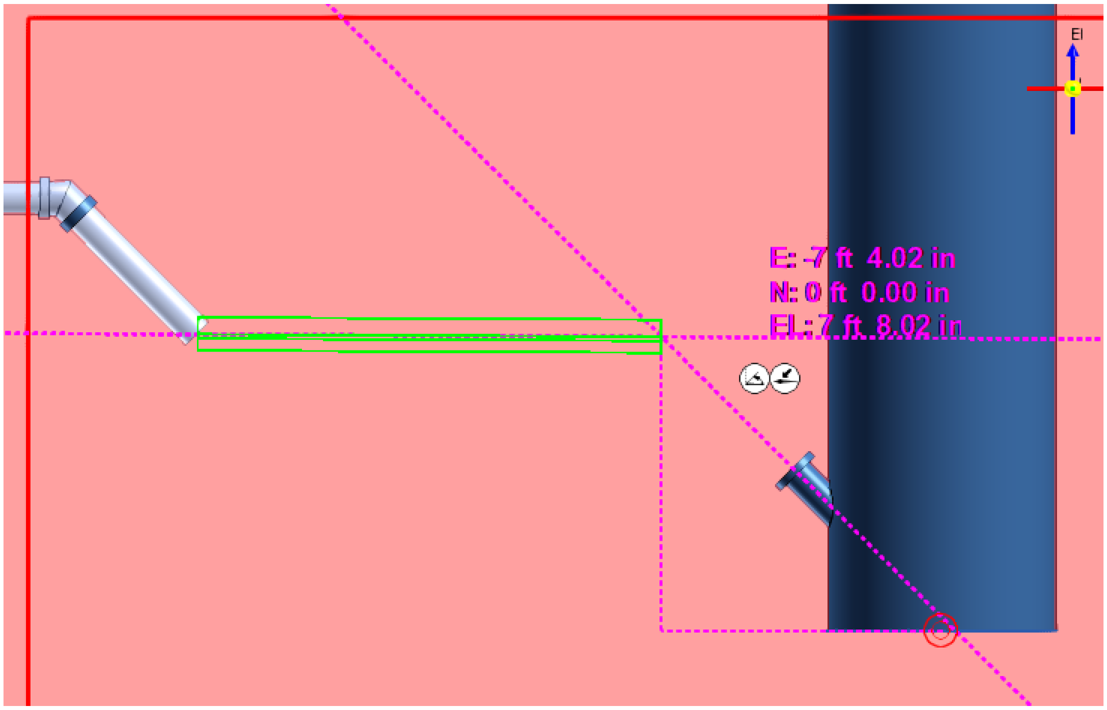
26. Click in the graphic view to place the 45 deg elbow and a piece of the pipe.
27. Change the view of the model to **Looking North** by using the **Common Views** button on the Common toolbar. This will enable you to get a better view of the routed sloped pipe.

28. Now, select **Elevation Plane: East-West** from the **Plane** drop-down list and select the **Lock Slope** option on the **Route Pipe** ribbon to constrain the routing to the minimum slope value. Move the cursor to the piping nozzle of the catch basin to add it to the SmartSketch stack.



The system will display a projection line that indicates the centerline of the piping nozzle.

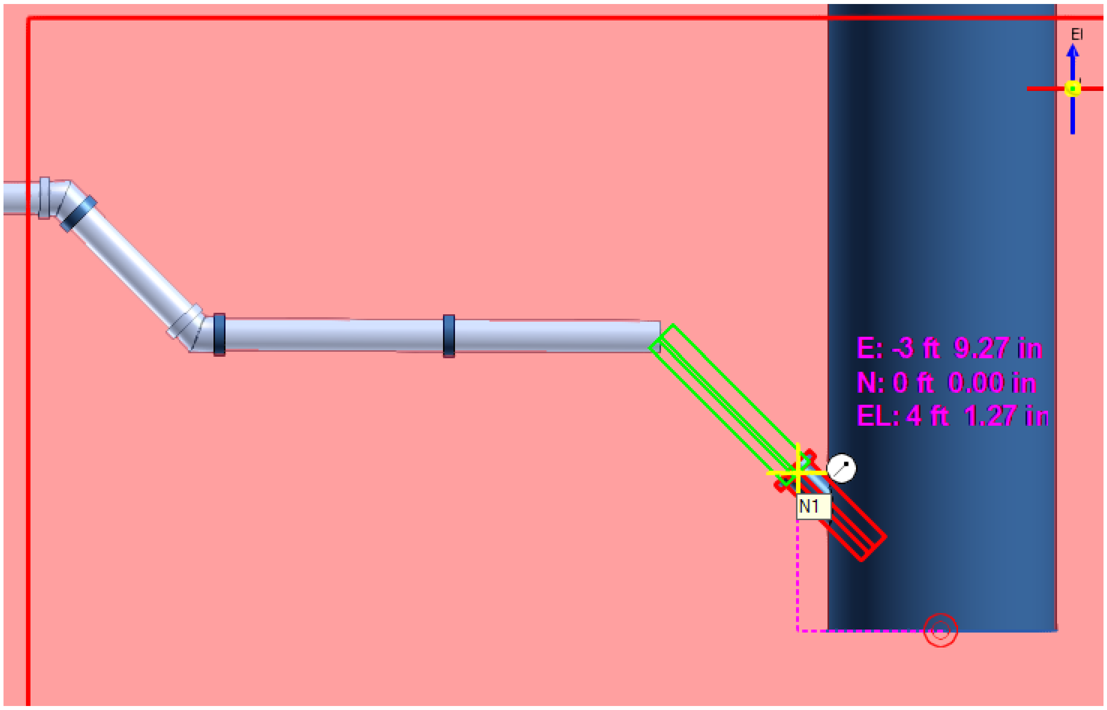
29. Locate the projection line displayed by the system until SmartSketch displays the point on curve indicator  and E glyphs. These glyphs indicate that you found the intersection point between the sloped pipe and the centerline of the pipe nozzle.



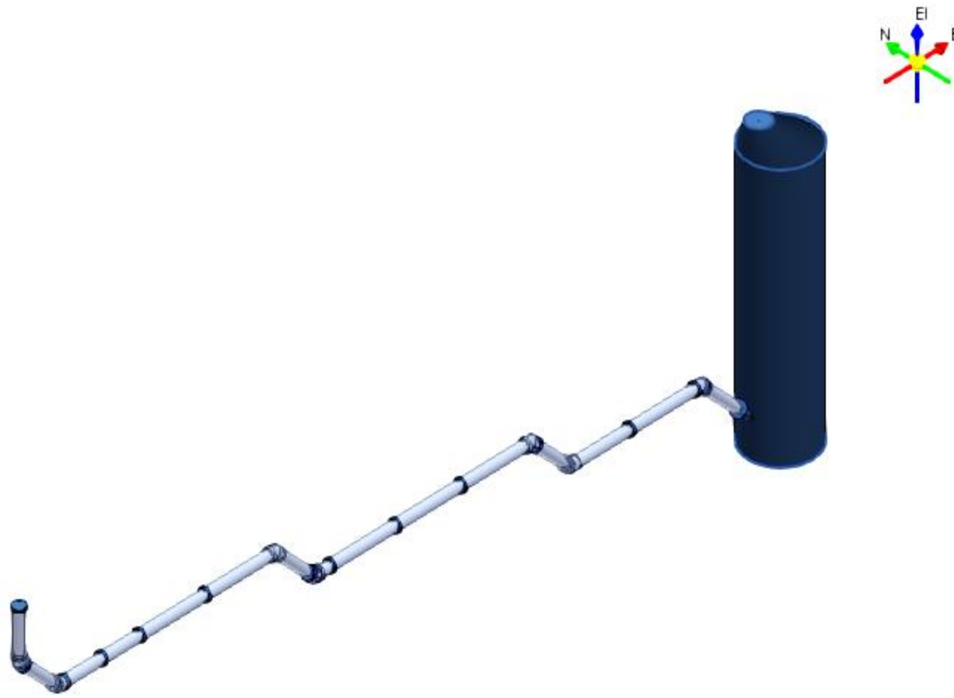
30. Click to accept the placement of this pipe.

31. Now, select **East\West in the Plane** drop-down list and the **Lock Angle** and **Lock Slope** options on the **Route Pipe** ribbon to unlock the slope and angle constraints, respectively.

32. Locate the piping nozzle **N1** until SmartSketch displays point port glyph.

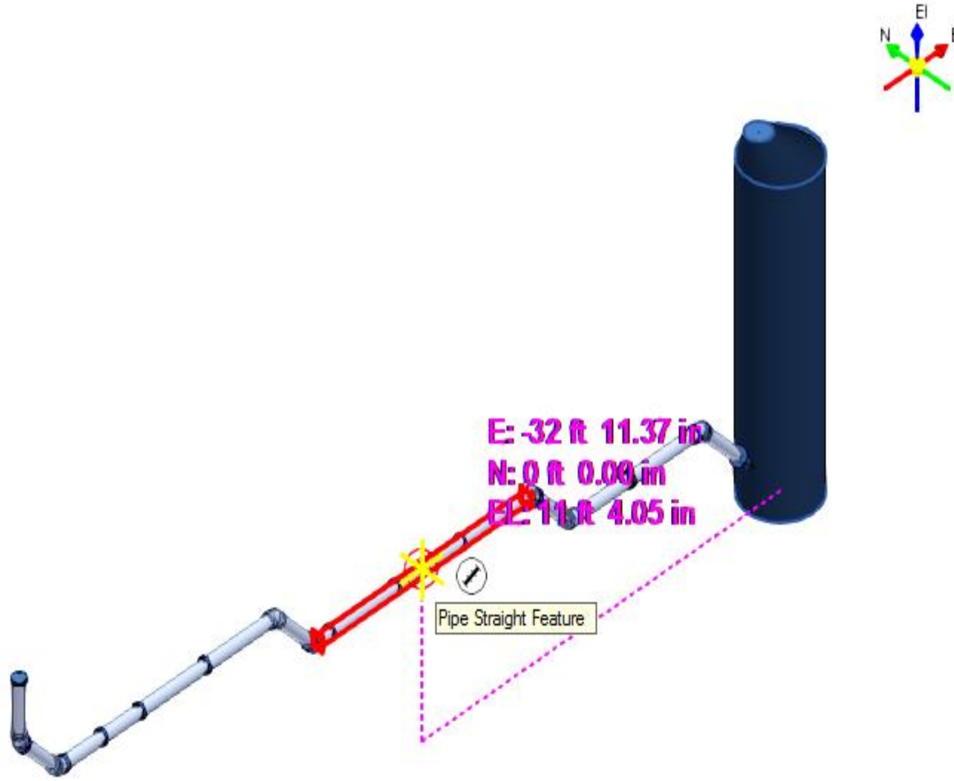


33. Click in the graphic view to finish routing the main sewer line.



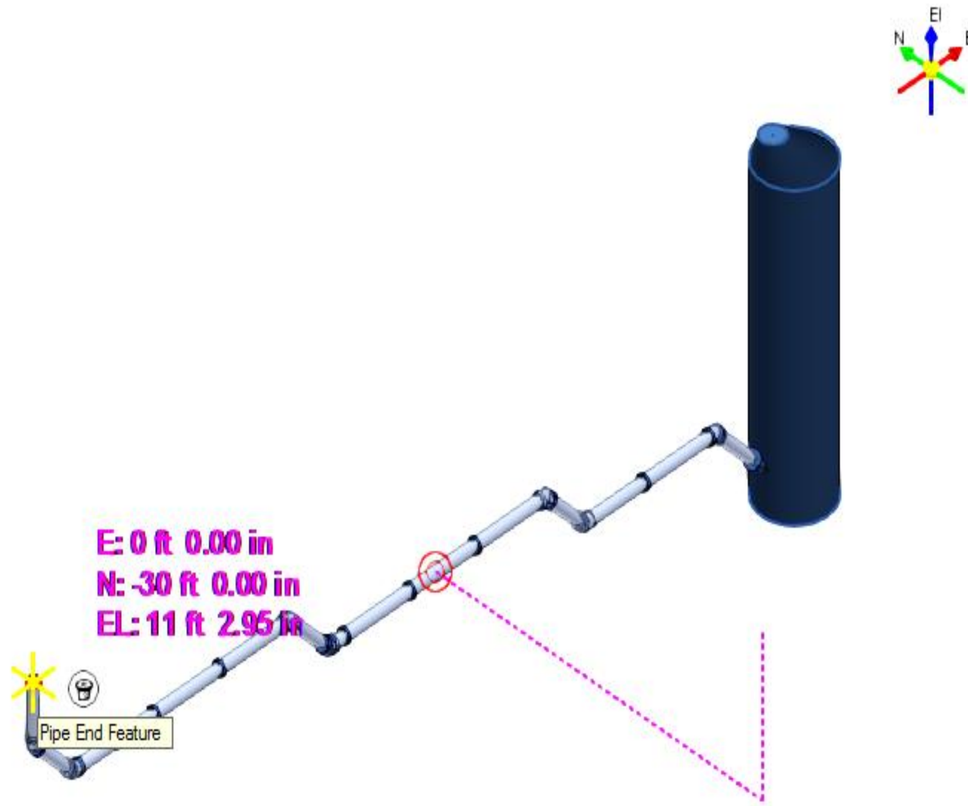
34. Now, click Route Pipe  to start routing the lateral sewer line.
35. Select Reposition Target  on the **PinPoint** ribbon to define a reference point to route the lateral sewer line.

36. Place the **PinPoint** target at the midpoint of the sloped pipe.



37. Key in **0 ft** for **E** and **-30 ft** for **N** on the PinPoint ribbon and locate the cleanout open port until the SmartSketch indicator key point glyph is displayed.

38. Press the function key **F8** to lock the **El** drop-down list by using the cleanout open port.

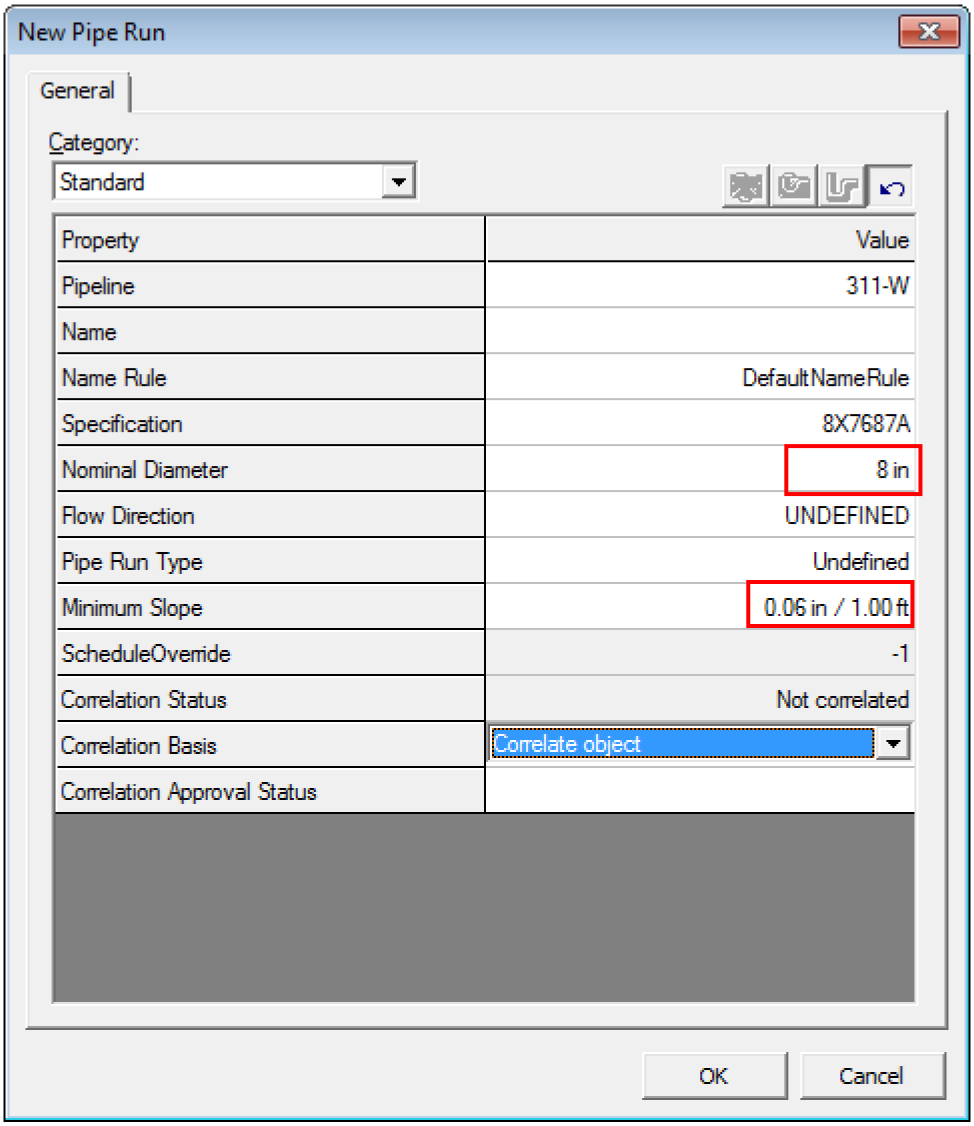


39. Click in the graphic view to accept pipe placement.

    The New Pipe Run dialog box appears.

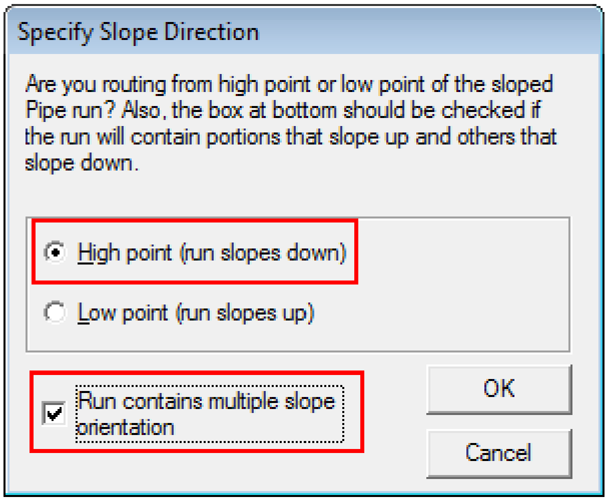
40. Select the following specifications in the **New Pipe Run** dialog box, and click OK.

Pipeline: 311-W
Name Rule: DefaultNameRule
Specification: 8X7687A
Nominal Diameter: 8 in
Flow Direction: DOWNSTREAM
Pipe Run Type: Undefined
Minimum Slope: 0.0625 in /1.0 ft
Correlation Status: Not correlated
Correlation Basis: Correlate object



The Specify Slope Direction dialog box appears.

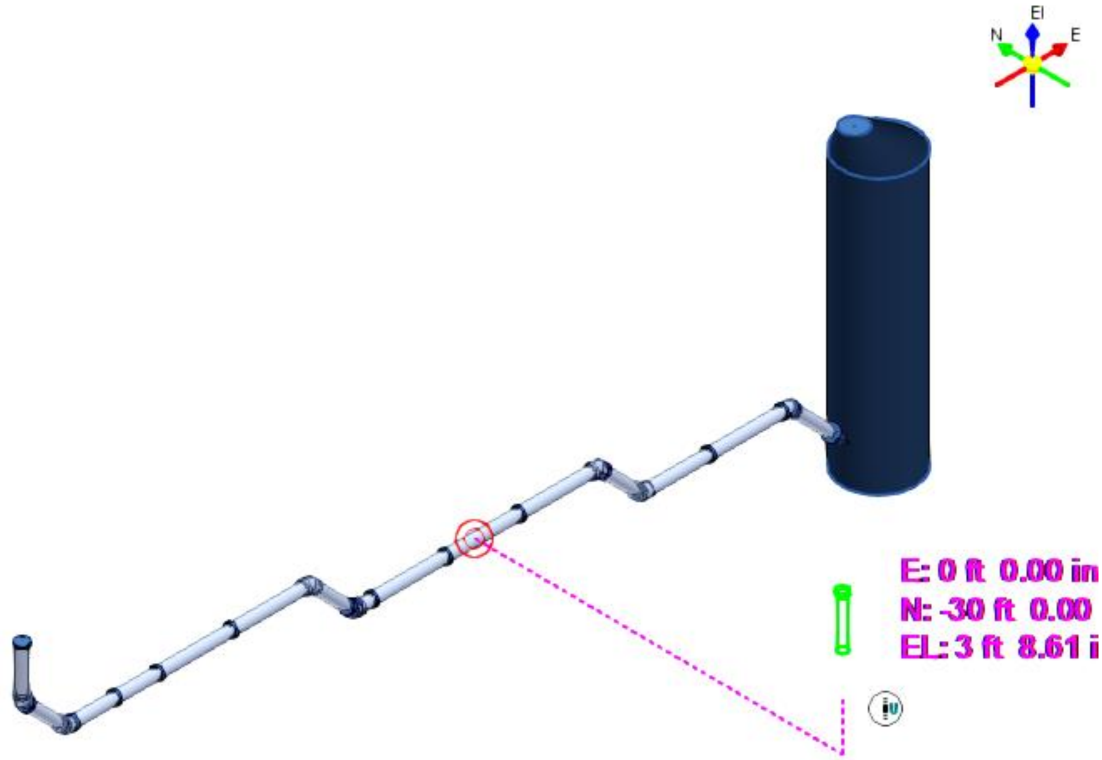
41. You need to route a pipe that slopes downwards. So, select the **High point (run slopes down)** option and the **Run contains multiple slope orientation** check box on the **Specify Slope Direction** dialog box.



42. Click **OK**.
43. Now, set the following specifications on the **Route Pipe** ribbon:

Length: 4 ft
Plane: No Plane

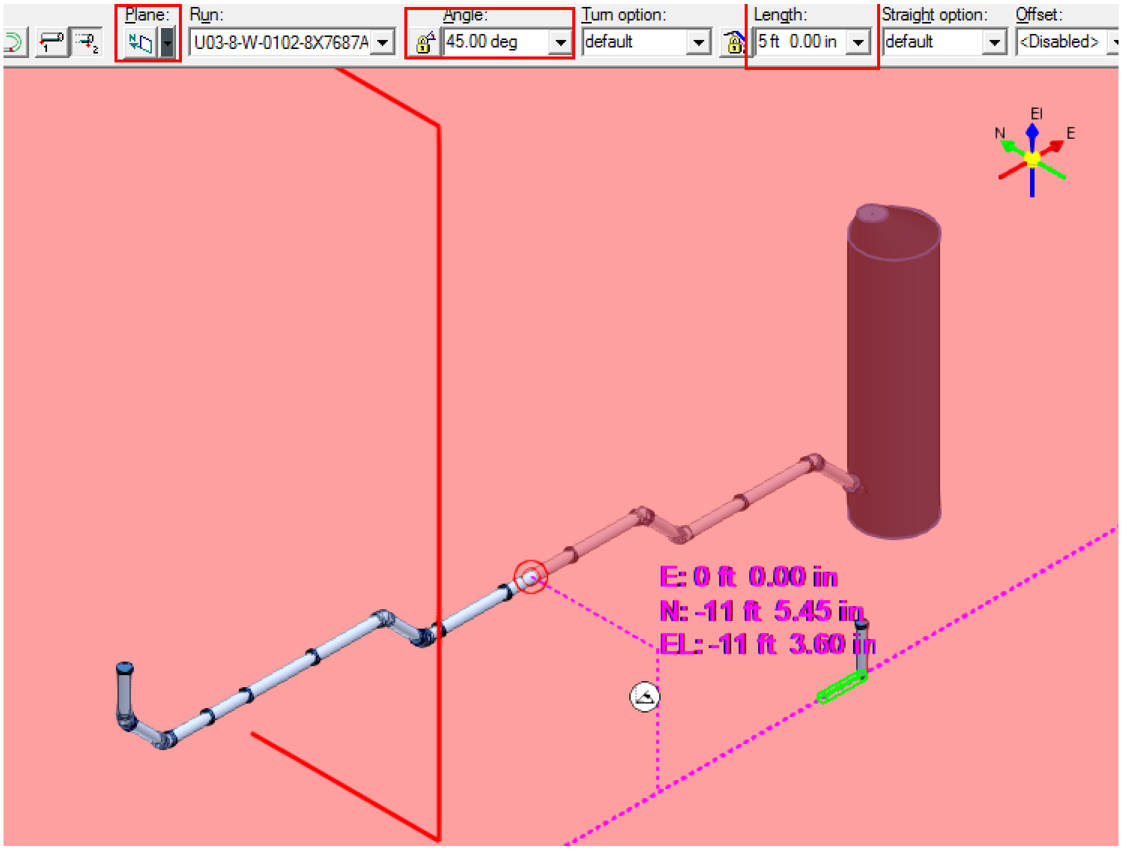
44. Also, click **Lock Slope** to unlock the slope constraint and position the cursor downwards from the starting point. The SmartSketch indicator will display a U glyph. This glyph indicates that you are going down the Z axis.



45. Click in the graphic view to accept the placement of the pipe.
46. Now, route a **5 ft** pipe by using the following specifications on the **Route Pipe** ribbon to route a pipe in the north-south direction:

    Plane: Elevation Plane: North-South
    Angle: 45 deg

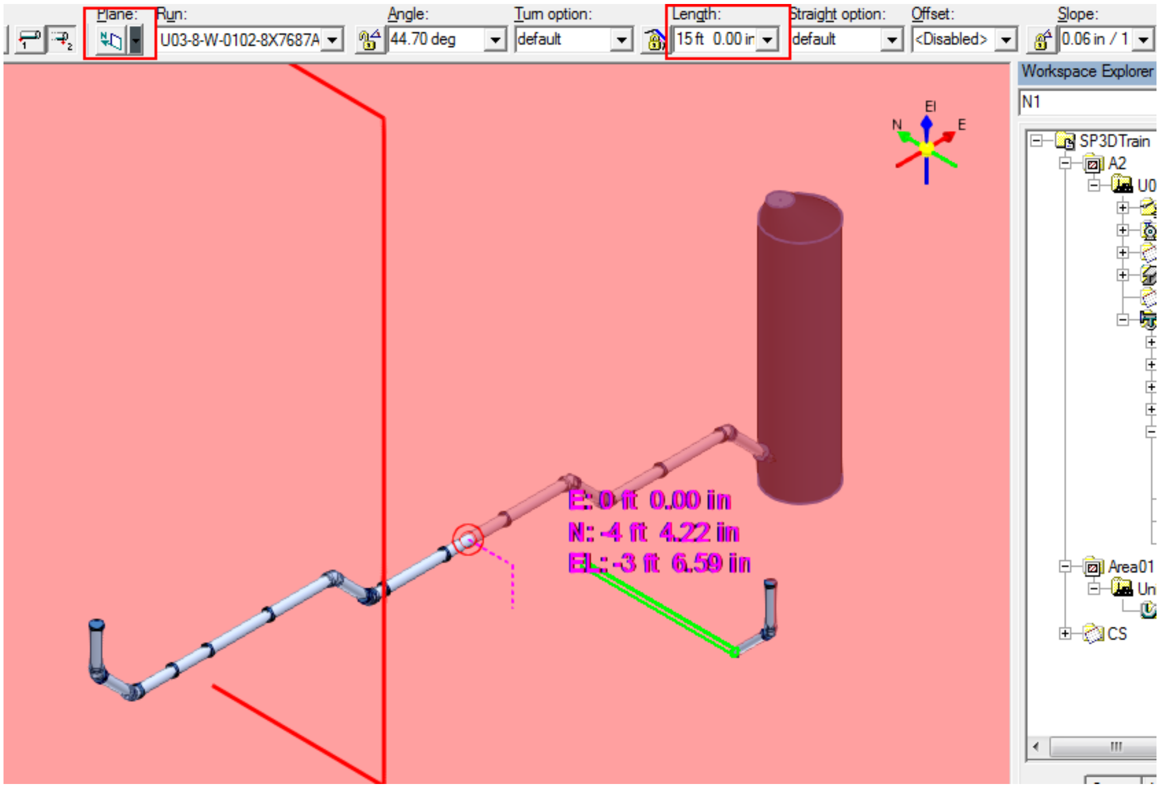
47. Then move the cursor in the north-south direction.



48. Click in the graphic view to place the 45 deg elbow and a piece of the pipe.
49. Now, set the following specifications on the **Route Pipe** ribbon to route a pipe in the north direction:

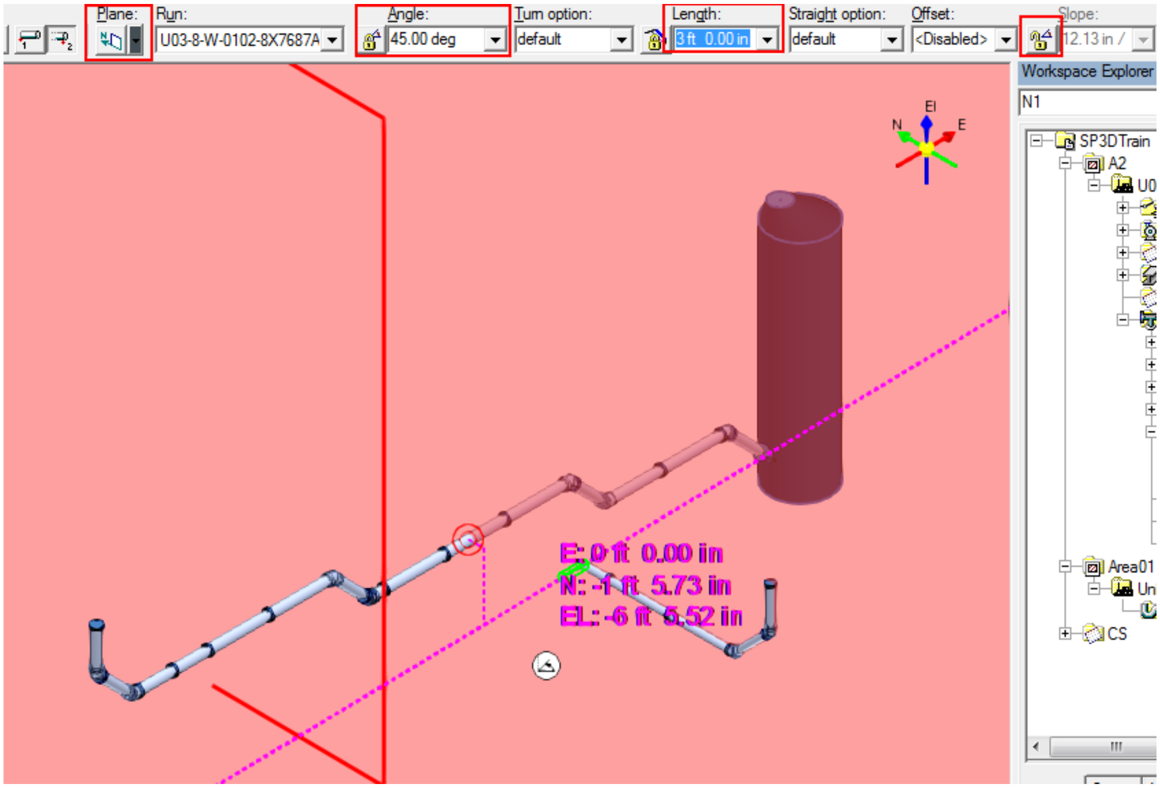
Length: 15 ft
Plane: Elevation Plane: North-South

50. Select **Lock Slope** on the **Route Pipe** ribbon to constrain the routing to the minimum slope value and position the cursor point until the SmartSketch indicator displays an N glyph. This glyph indicates that you are going in the northing direction.



51. Click in the graphic view to accept the placement of the pipe.
52. Now, route a **3 ft** pipe by using the following specifications on the **Route Pipe** ribbon:

Plane: Elevation Plane: North-South
Angle: 45 deg
Lock Slope option: Unlock

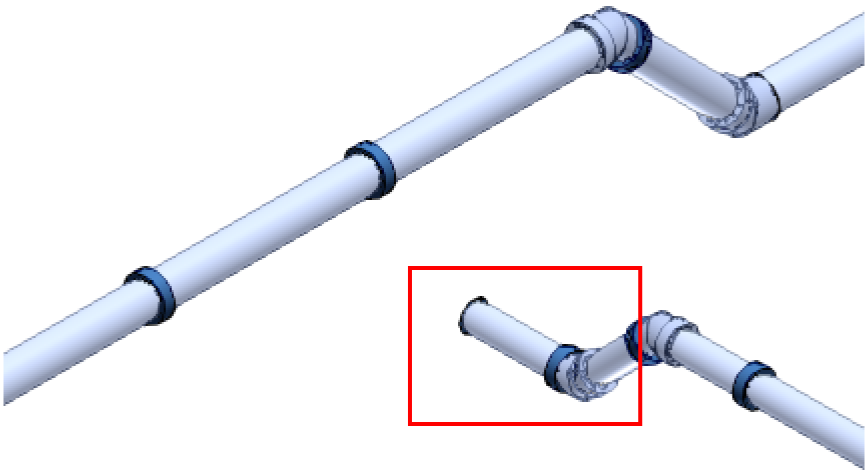


53. Click in the graphic view to accept the placement of this pipe.
54. Now, route another pipe North by using the following specifications on the **Route Pipe** ribbon:

    Length: 3 ft
    Plane: Elevation Plane: No Plane

55. Select **Lock Slope** to constrain the routing to the minimum slope value.
56. Click in the graphic view to accept the placement of this small sloped pipe.

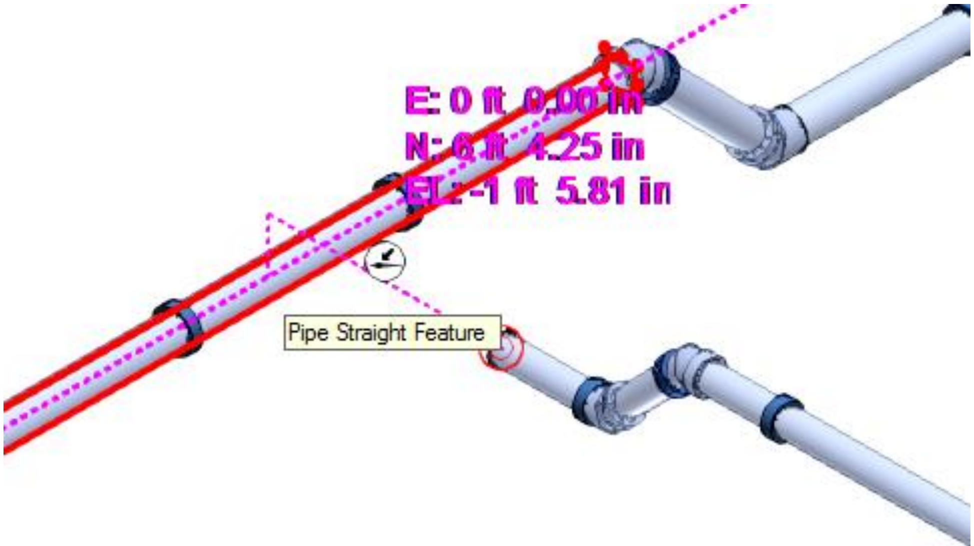
57. Right-click to terminate the **Route Pipe** command.



58. Select **Reposition Target** on the **PinPoint** ribbon and place the target at the end of the sloped pipe.

59. Select **Relative Tracking** and key in **0 ft** for **E** on the PinPoint ribbon.

60. Now, click **Route Pipe** and position the cursor on the **Pipe Straight Feature** of the main sewer line to start routing the lateral sewer line.



The system displays a projection line and locates the intersection point between the sloped pipe and the main sewer line.

61. Left-click to start routing the pipe.

    The New Pipe Run dialog box appears.

62. Set the following specifications in the **New Pipe Run** dialog box, and click **OK**.

    Pipeline: 311-W
    Name Rule: DefaultNameRule
    Specification: 8X7687A
    Nominal Diameter: 8 in
    Flow Direction: UPSTREAM
    Minimum Slope: 0.0625 in /1.0 ft
    Schedule Override: <undefined value>
    Correlation Basis: Correlate object

    The Specify Slope Direction dialog box appears.

63. You need to route a pipe that slopes downwards. **Select the High point (run slopes down)** option and the **Run contains multiple slope orientation** check box.

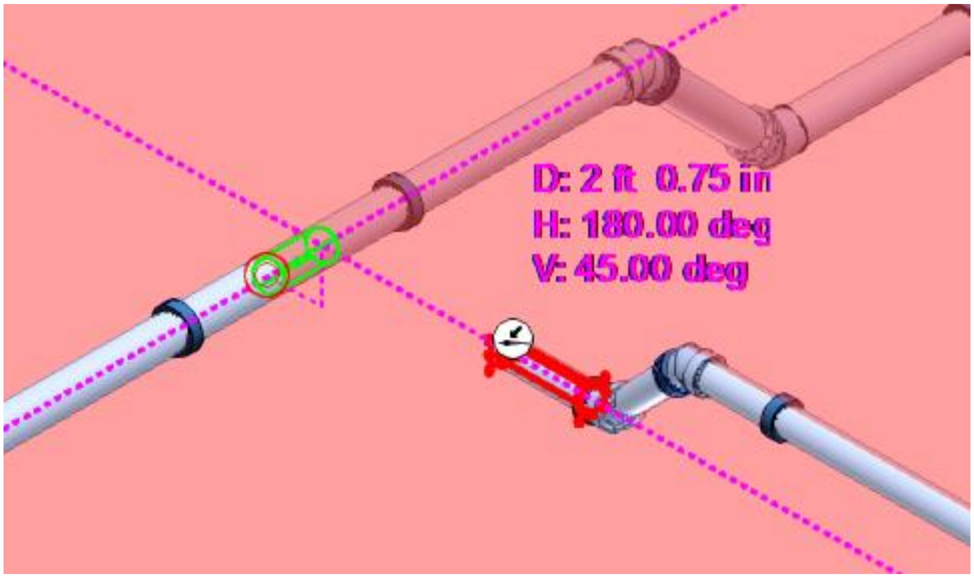
64. Click **OK**.

65. Select the **Elevation Plane: North-South** from the **Plane** drop-down list and the **Lock Slope** and **Lock Angle** options to unlock the slope and angle constraints, respectively, on the **Route Pipe** ribbon.

66. On the **PinPoint** ribbon, select **Spherical Coordinates** and select the **Down -45 deg** option in the **Vertical** drop-down list.

67. Move the cursor to the small sloped pipe to add it to the SmartSketch stack and locate the projection line until the SmartSketch indicator displays the point on curve glyph. This

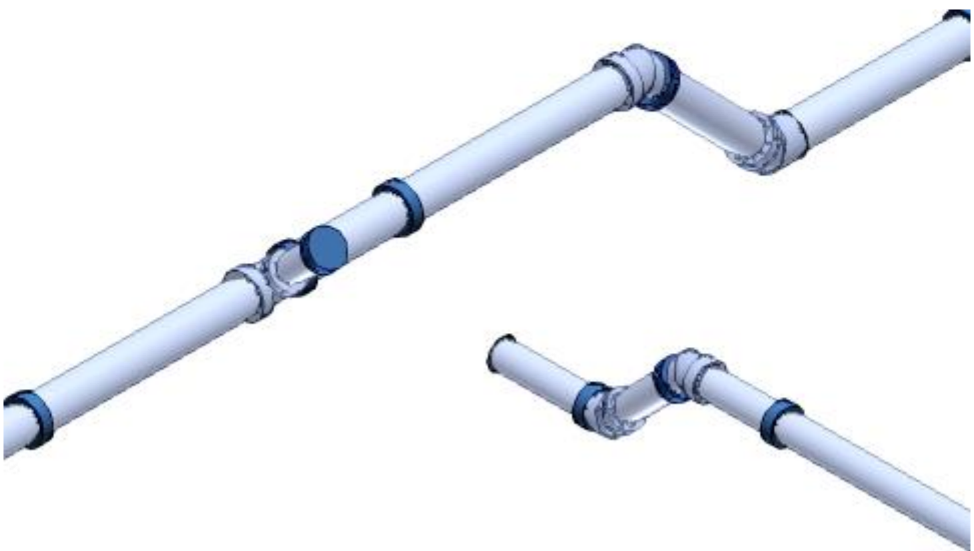
glyph indicates that you found the intersection point between the sloped pipe and the constraints (45 deg Up) created by the PinPoint.



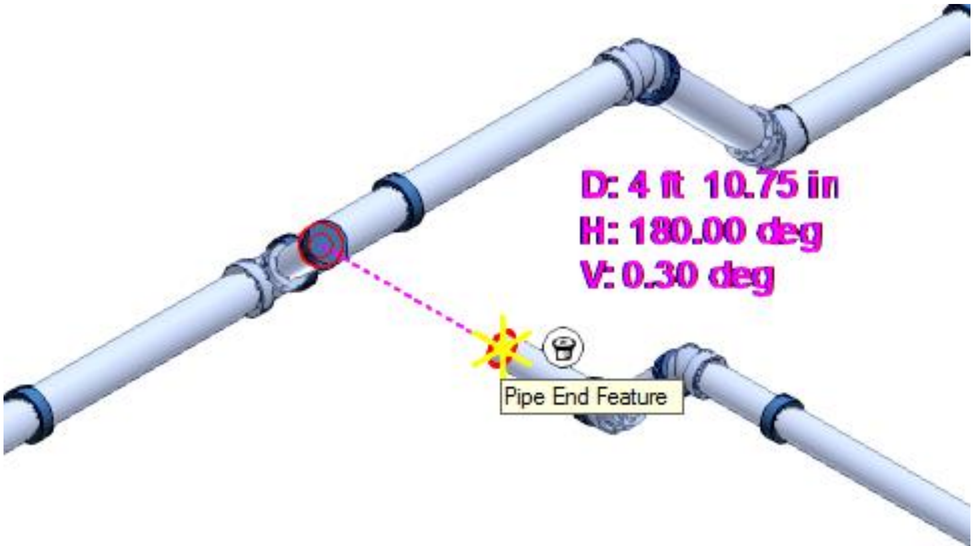
68. Click in the graphic view to place the data point.

    The system places a reducing tee and a small pipe.

69. Right-click to terminate the **Route Pipe** command.

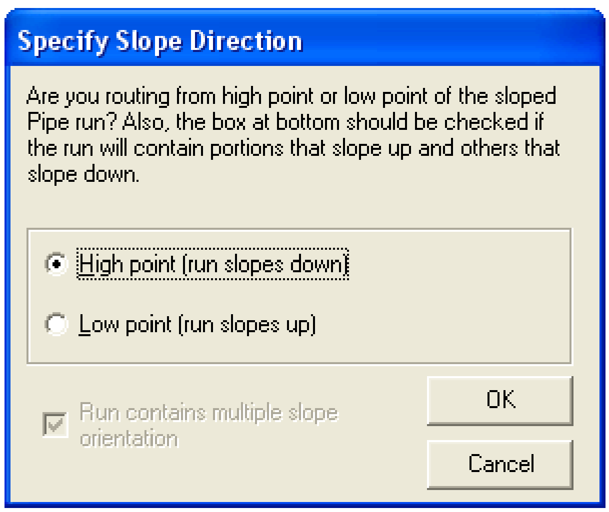


70. Now, click **Route Pipe** and position the cursor on the **End Feature** of the small sloped pipe to continue routing the lateral sewer line.



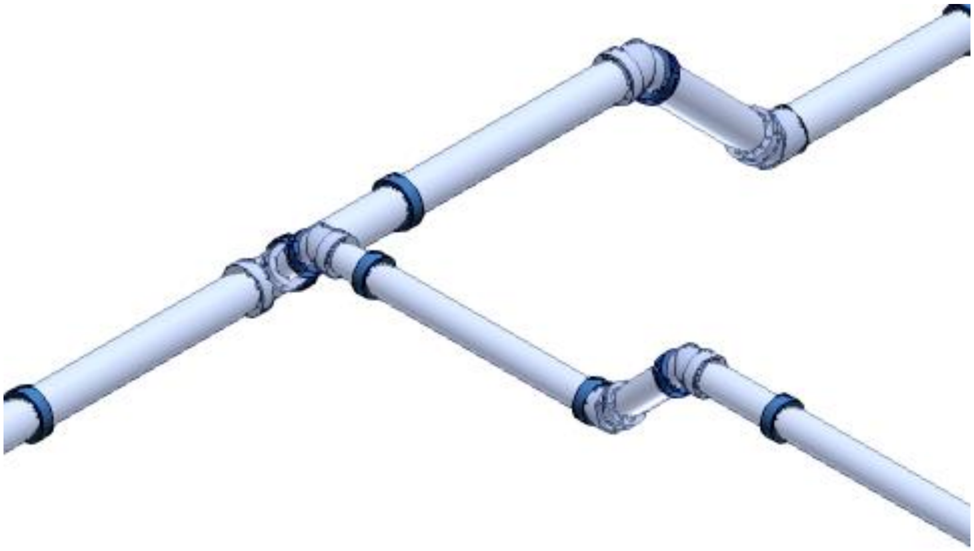
The Specify Slope Direction dialog box appears

71. You need to route a pipe that slopes downwards. Select the **High point (run slopes down)** option. Then, click **OK**.



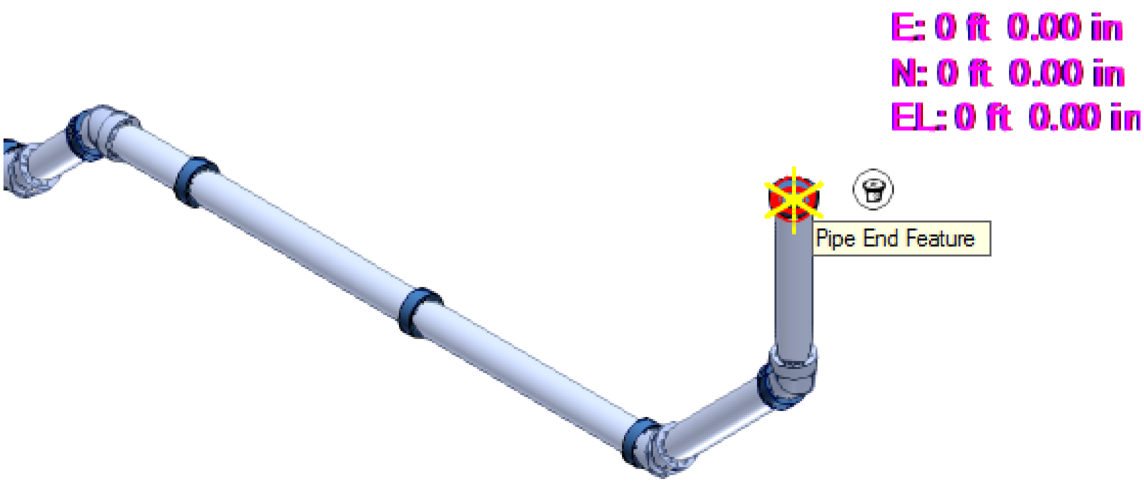
72. Locate the **End Feature** on the pipe coming out from the reducing tee.

73. Click in the graphic view to finish routing the lateral sewer line.

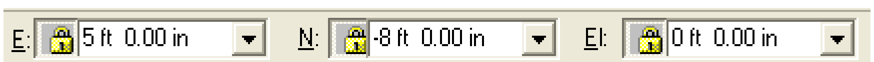


74. Now, click **Route Pipe** to start routing another cleanout line.
75. Set the **PinPoint** mode to **Rectangular Coordinates** .

76. Select **Reposition Target** on the PinPoint ribbon and place the target at the cleanout located in the lateral sewer line.



77. Key in **5 ft** for **E**, **-8 ft** for **N**, and **0 ft** for **El** on the PinPoint ribbon.



The New Pipe Run dialog box appears.

78. Set the following specifications in the **New Pipe Run** dialog box, and click **OK**.

Pipeline: 311-W
Name Rule: DefaultNameRule
Specification: 8X7687A
Nominal Diameter: 8 in
Flow Direction: DOWNSTREAM
Minimum Slope: 0.0625 in /1.0 ft
Schedule Override: <undefined value>
Correlation Basis: Correlate object

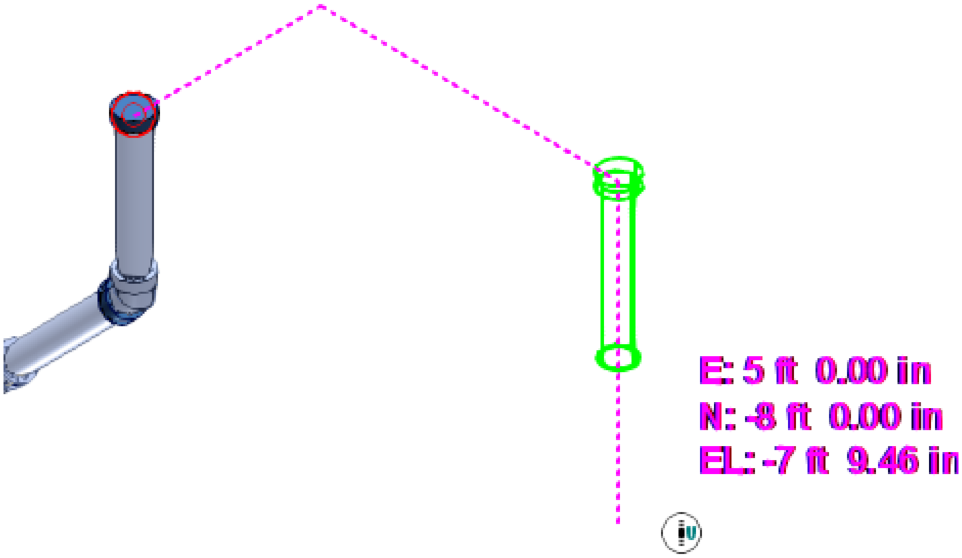
The Specify Slope Direction dialog box appears.

79. You need to route a pipe that slopes downwards. So, select the **High point (run slopes down)** option. Click **OK**.

80. Now, specify the following specifications on the Route Pipe ribbon:

Length: 4 ft
Plane: No Plane

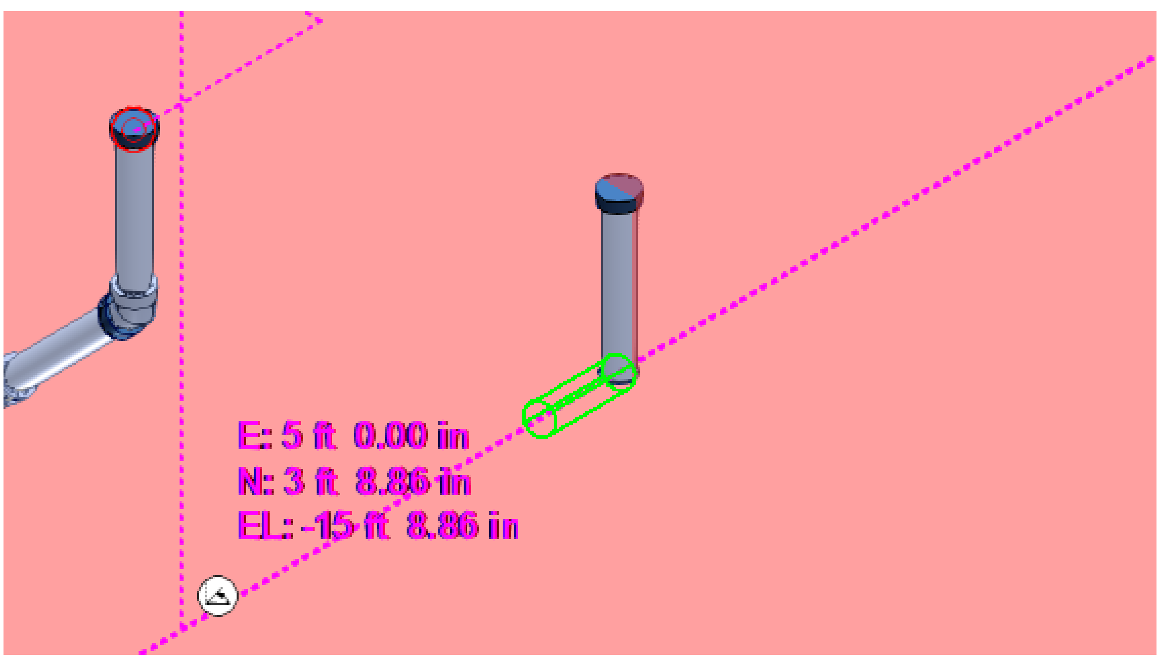
81. Select **Lock Slope** on the **Route Pipe** ribbon to unlock the slope constraint and position the cursor downwards from the starting point. A SmartSketch indicator will display a U glyph. This glyph indicates that you are going down the Z-axis.



82. Click in the graphic view to accept the placement of the pipe.

83. Route a pipe by using the following specifications on the Route Pipe ribbon:

Length: 3 ft
Plane: Elevation Plane: North-South
Angle: 45 deg

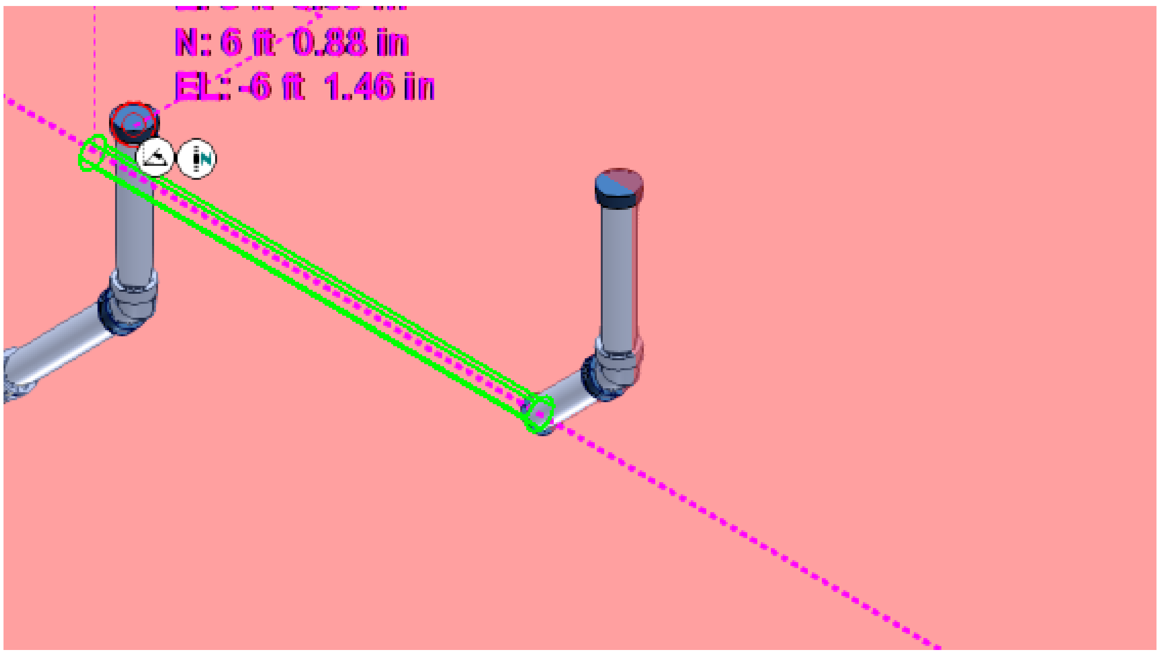


84. Set the following specifications on the **Route Pipe** ribbon to route a pipe in the north direction:

    Length: 12 ft
    Plane: Elevation Plane: North-South

85. Select the **Lock Slope** option on the **Route Pipe** ribbon to constrain the routing to the minimum slope value and position the cursor point until SmartSketch displays an N glyph. This glyph indicates that you are going in the north direction.

86. Click in the graphic view to accept the placement of the pipe.

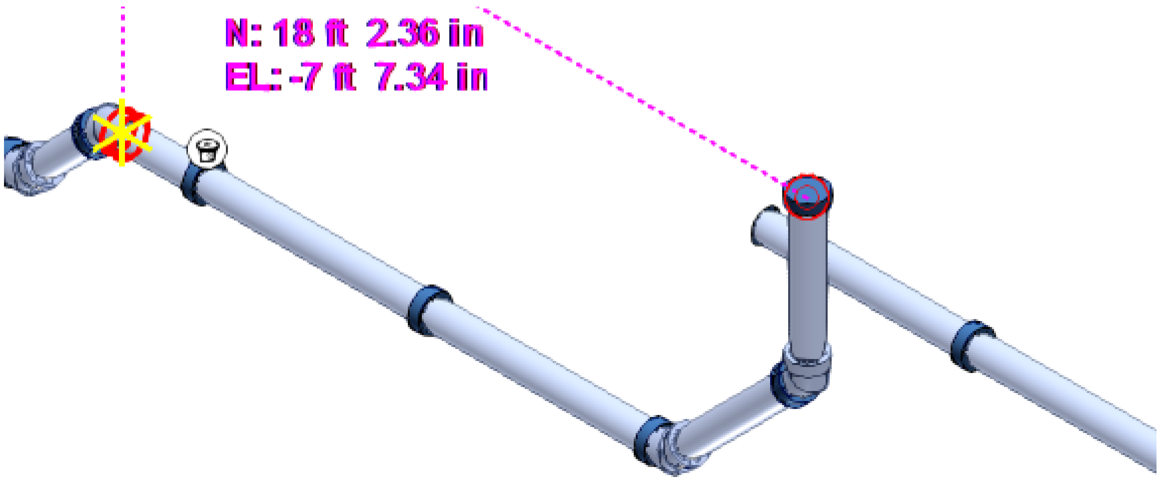


87. Now, place a 45 deg elbow in the direction of the lateral sewer line. This step can be done by creating a plane constraint, which forces the routing at 45 deg.

88. Select **Plane by Three Points** from the **Plane** drop-down list on the Route Pipe ribbon.

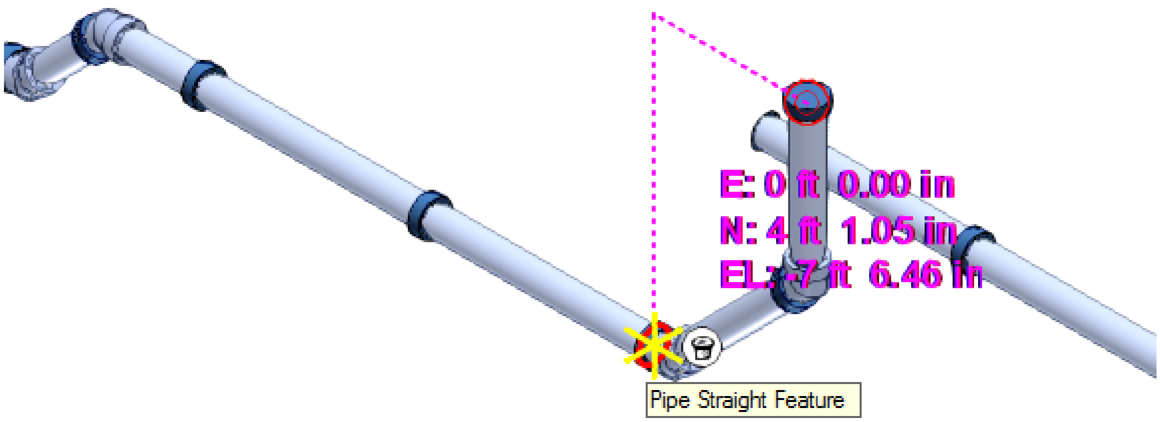
    The system prompts you to specify the first point of the plane.

89. Select a port on the lateral pipeline to define the first point of the plane.



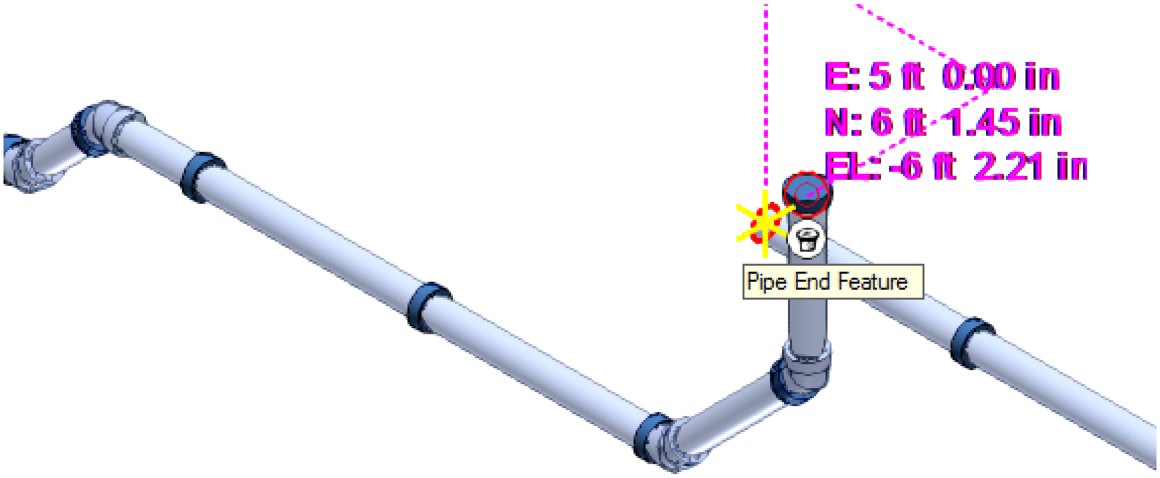
The system prompts you to specify the second point of the plane.

90. Select another port on the lateral pipeline to define the second point of the plane.

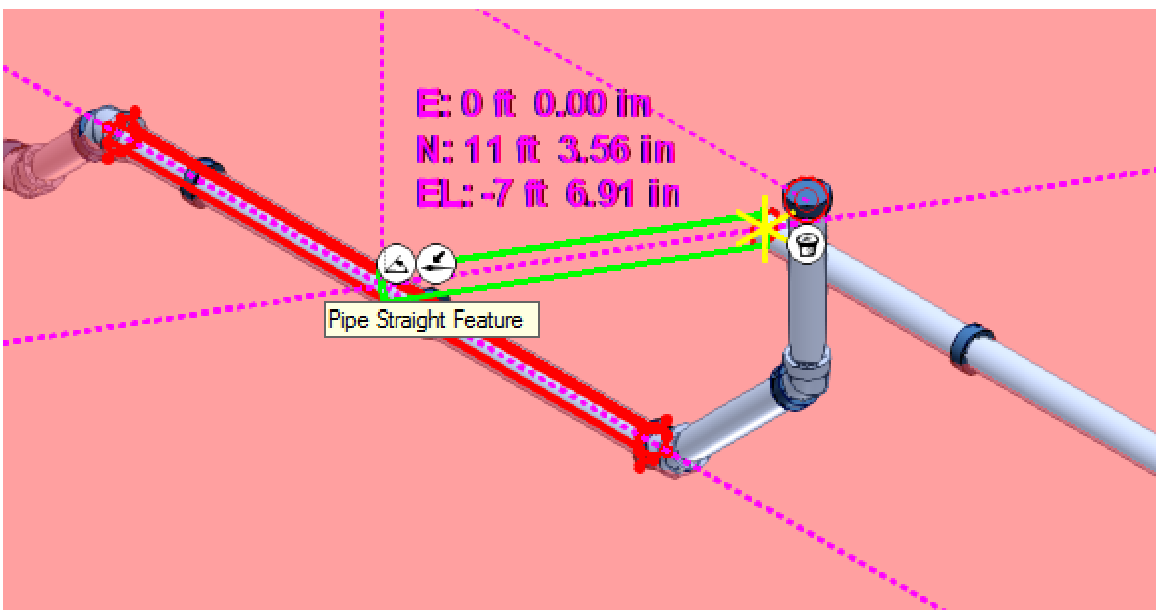


The system prompts you to specify the third point of the plane.

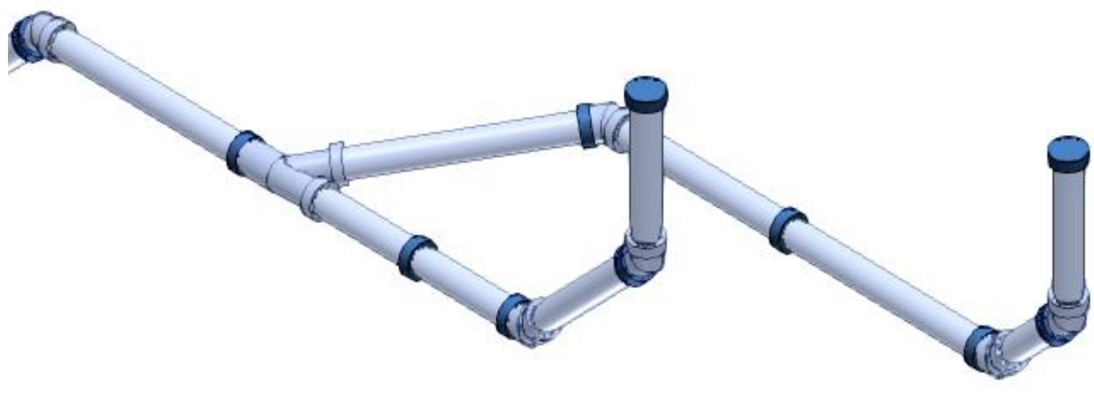
91. Select the turning point of the routing line to define the third point of the plane.



92. Key in **45 deg** in the **Angle** drop-down list and select the **Lock Slope** option on the **Route Pipe** ribbon to unlock the slope constraint.
93. Position the cursor point along the lateral pipeline. SmartSketch will display a point on curve glyph.



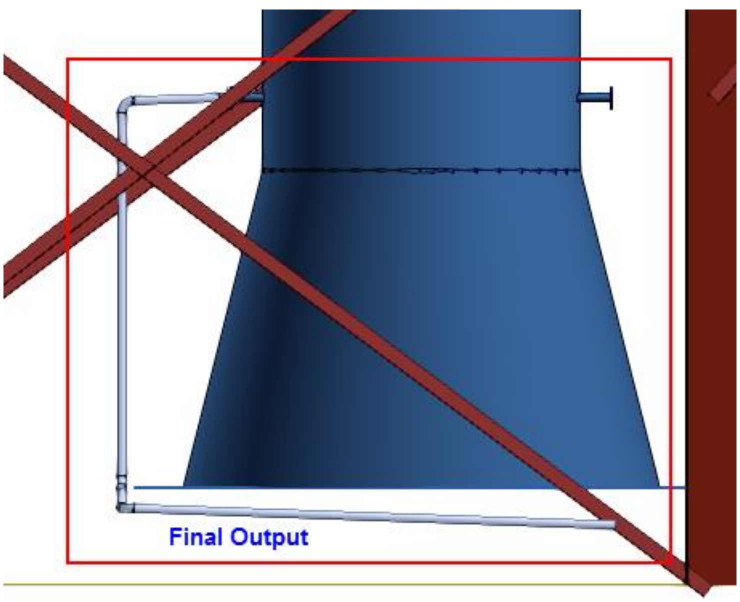
94. Click in the graphic view to place the 45 degree elbow. Notice the system places a 45 deg wye fitting based on the sizes of the lines, the rolling offsets, and the branch table defined in the pipe specification.



Routing a Sloped Pipe Run

Objective

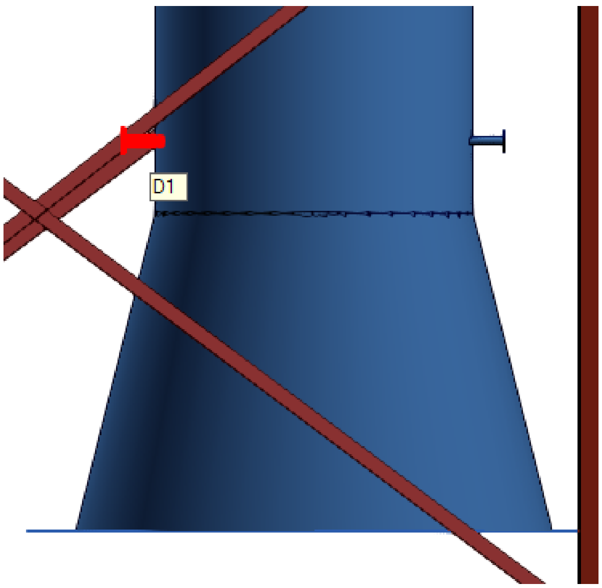
In this exercise you will be routing **Pipeline 302-W** from the tower **T-101/Nozzle D1** in **Unit U03** of your workspace. The routed pipe should look like the highlighted area below.



Before Starting this Procedure

- Define your workspace to display **Unit U03** and coordinate system **U03 CS**. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Locate the equipment nozzle **D1** in the Workspace Explorer.
2. Click **Common Views** on the Common toolbar to switch to the **Looking South** view. You might have to switch the view a couple of times during the procedure to get a better view of the area where you are performing a task.
3. Click **Route Pipe** .
4. In the graphic view, locate and click the equipment nozzle **D1**.



The New Pipe Run dialog box appears.

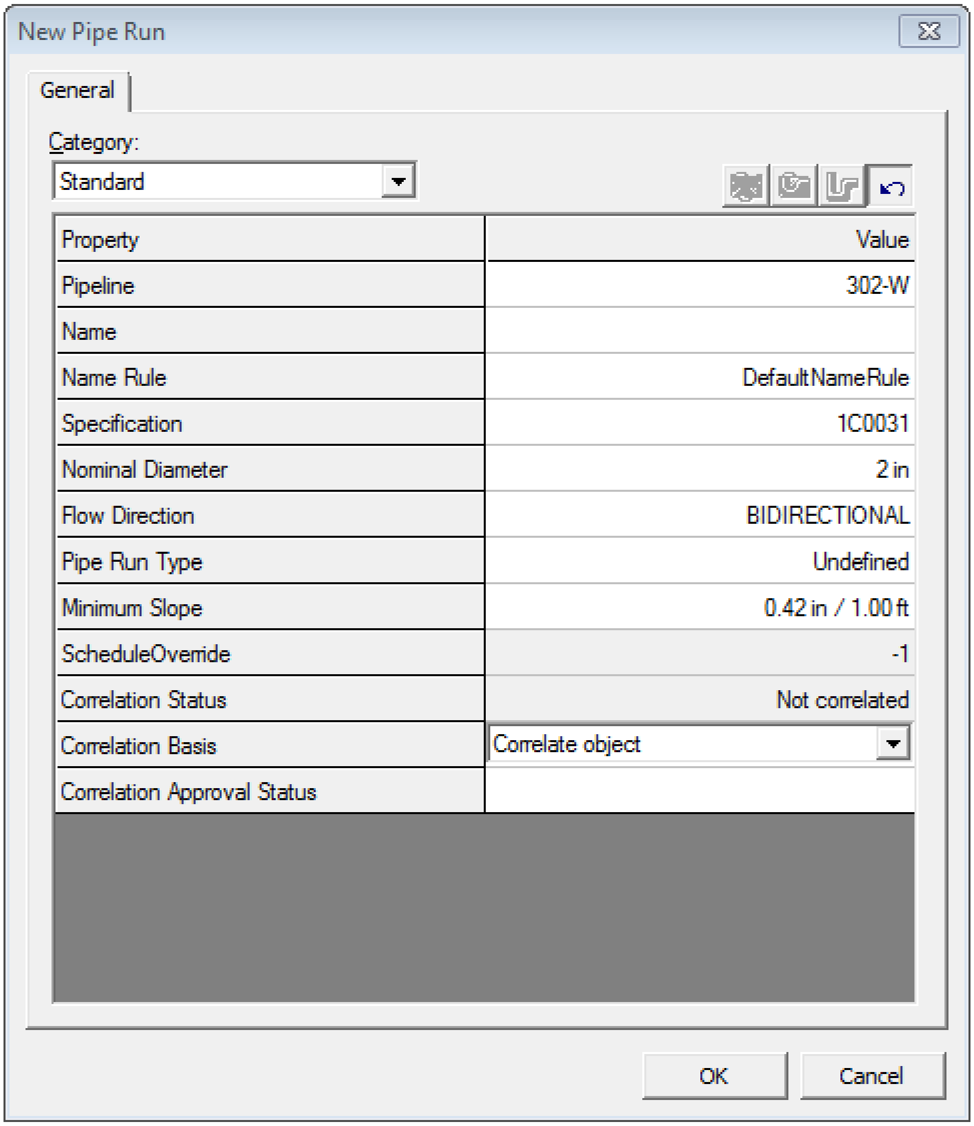
5. Select **302-W** in the **Pipeline** field.

   The system automatically selects the other parameters by using the piping specification.

   The specifications in the **New Pipe Run** dialog box are:

   Pipeline: 302-W
   Name Rule: DefaultNameRule
   Specification: 1C0031
   Nominal Diameter: 2 in
   Flow Direction: BIDIRECTIONAL
   Pipe Run Type: Undefined
   Minimum Slope: 2 deg
   Schedule Override: <undefined value>
   Correlation Status: Not Correlated
   Correlation Basis: Correlate object

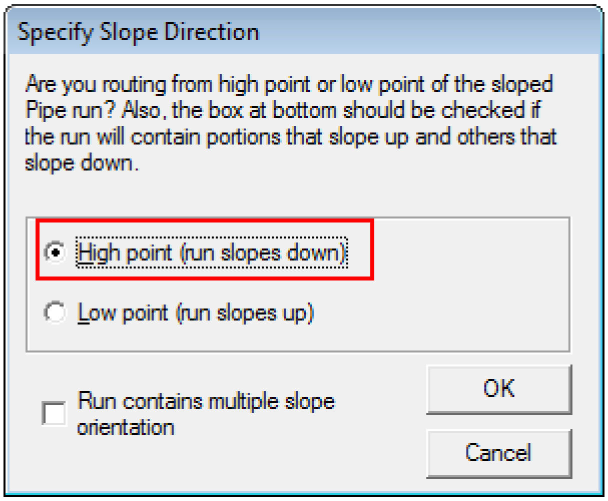
The software changes the Minimum Slope values to the current display unit of measurement 0.42 in/1.00 ft.



6. Click **OK**.

   The Specify Slope Direction dialog box appears.

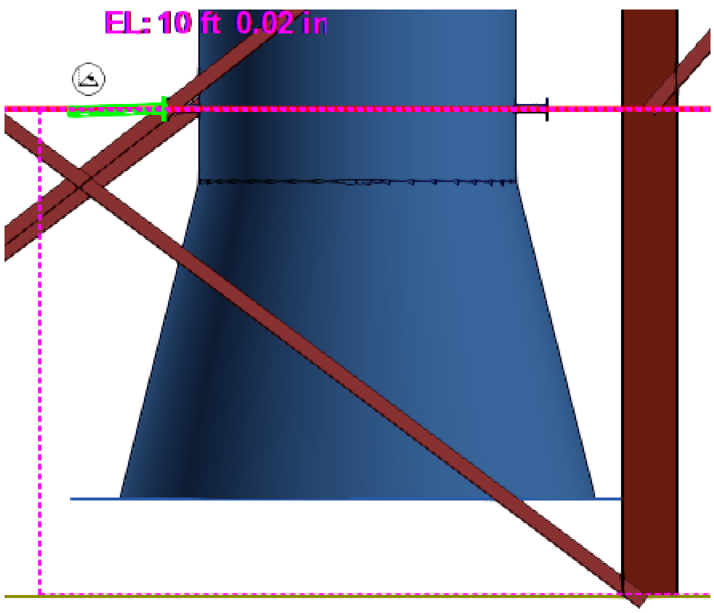
7. You need to route a pipe that slopes downwards. Select the **High point (run slopes down)** option, and click OK.



You now see an outline of a pipe in your model.

8. On the Route Pipe ribbon, select the **Plan-Plane** option from the **Plane** drop-down list and key in **2 ft** in the **Length** field.

9. Click in the graphic view to place the pipe. Ignore any warnings.

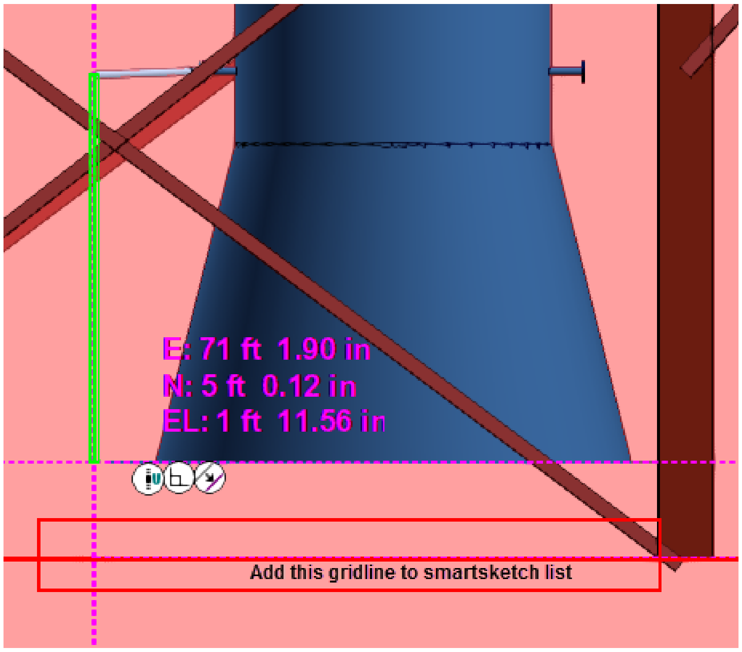


10. On the **Route Pipe** ribbon, unlock the **Angle**, **Slope**, and **Length** fields. Key in **2 ft** in the **Offset** field.

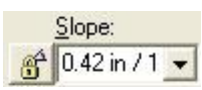
11. Click the **Plane** drop-down arrow on the **Route Pipe** ribbon and select **Elevation Plane: East-West** to route the pipe.

12. Move your cursor downwards to locate the grid line and add it to the SmartSketch list. Adding an object to the SmartSketch list helps define the exact point at a particular distance from the object where you want to place the pipe.
13. Move your cursor straight upwards.

    The system displays a projection line that indicates 2 ft offset from the selected grid line.
14. Click in the graphic view to place the pipe.



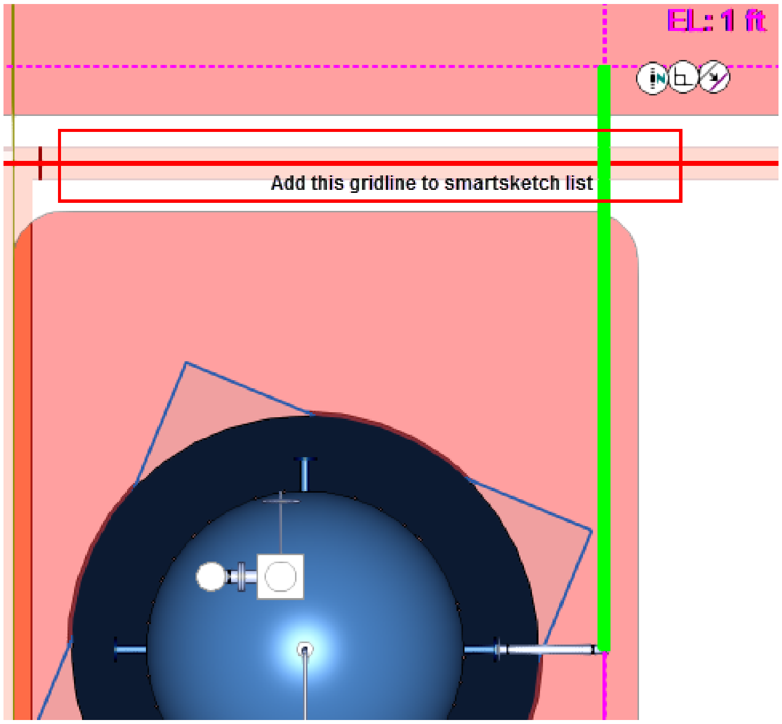
15. Click **Common Views** to switch to the **Looking Plan** view. On the **Route Pipe** ribbon, lock the **Slope** field again.



16. Click the **Plane** drop-down arrow on the **Route Pipe** ribbon and select **Plan Plane** to route the pipe.
17. Move the cursor upwards to locate the grid line and add it to the SmartSketch list.
18. Move your cursor straight upwards.

    The system displays a projection line that indicates a 2 ft offset from the selected grid line.

19. Click in the graphic view to place the pipe.

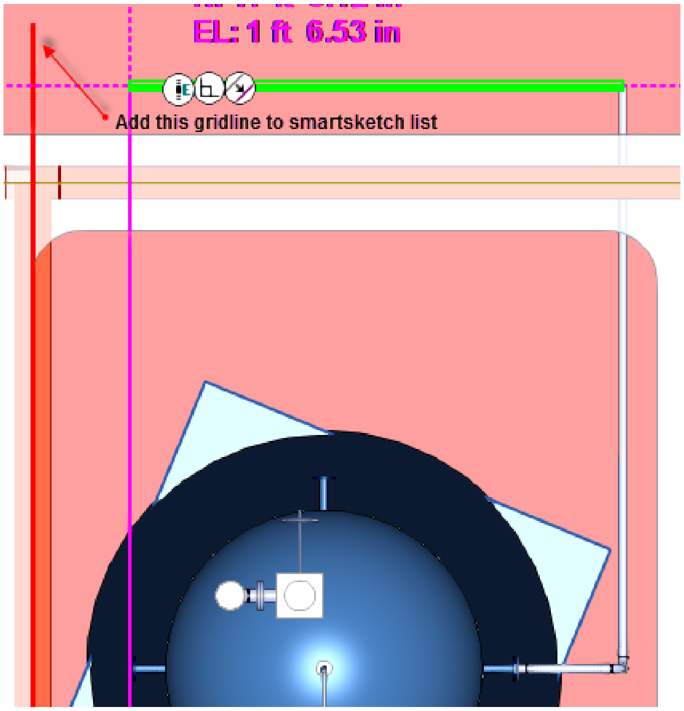


20. Move the cursor in the west direction to locate the grid line and add it to the SmartSketch list.

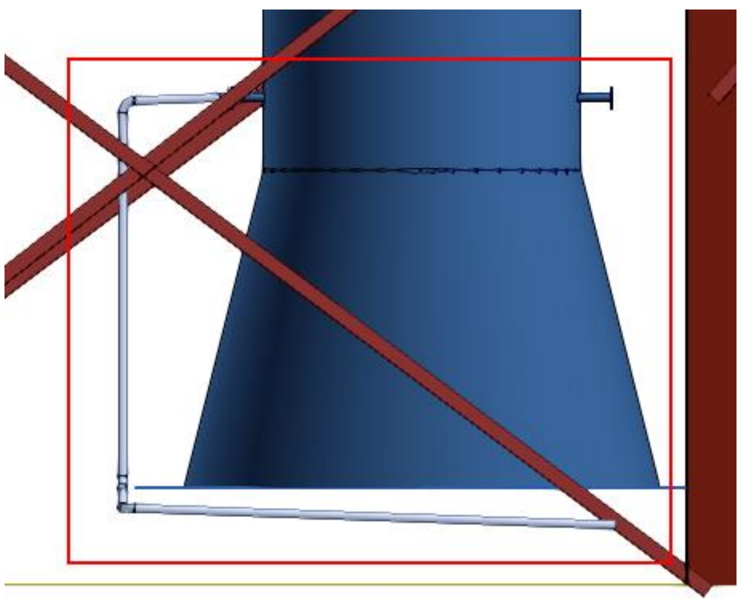
21. Move your cursor straight towards the right.

    The system displays a projection line that indicates a 2 ft offset from the selected grid line.

22. Click in the graphic view to place the pipe.



23. Right-click in the graphic view to terminate the **Route Pipe** command.
24. Click **Common Views** to switch to the **Looking South** view. This gives you a better view of the pipe you have placed. The view of your model should now resemble below.



After routing the sloped pipe, you use Insert Component  to insert components, where required, in the pipe. For steps to insert components in a pipe run, see *Inserting Components in a Pipe Run* (on page 69).

For more information related to routing a sloped pipe, refer to the following topics in the user guide PipingUsersGuide.pdf:

- Route a Sloped Pipe Run
- Route a Multi-Sloped Pipe Run

SESSION 11

Routing Arc and Flex Pipe

Objective

By the end of this session, you will be able to:

- Use the Arc Pipe command to place Arc and Circular Piping.
- Use the Flex Pipe command to route radial and offset Flex Pipe.
- Insert various components on Arc Piping.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)

Overview

Special circumstances may occur where arced or circular piping needs to be routed. Applications of for this could include routing ring headers in heater or fuel systems. The arc routing command is useful for modeling these types of systems, which would otherwise prove difficult to represent using the straight piping command.

Additionally, this section covers the flex pipe command. Again, applications for this could be on headers and fueling stations where piping requires flexibility. The flex pipe command uses an auto-routing capability to efficiently model flex pipe between two given points.

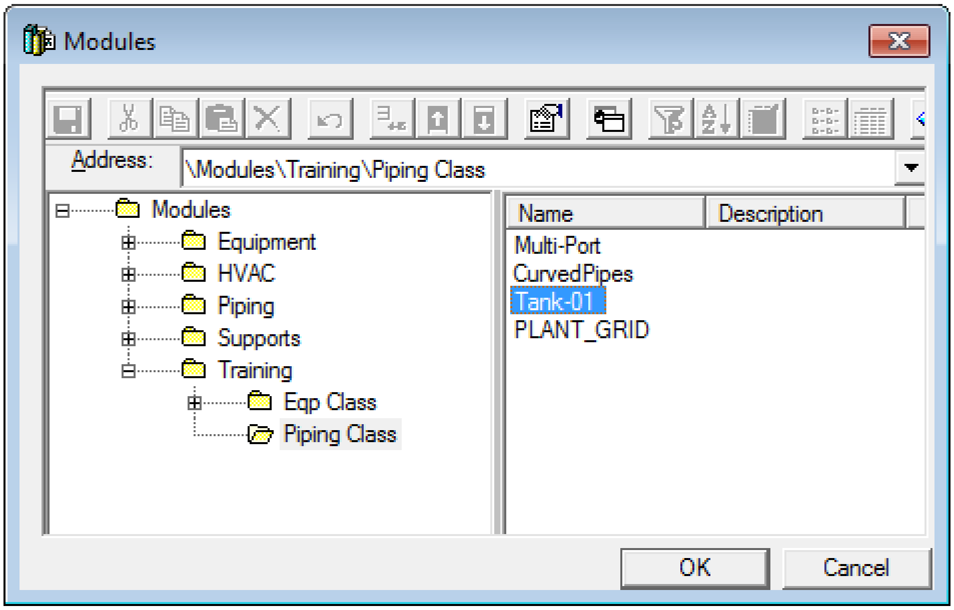
Routing Arc Pipe

Before Starting this Procedure

- Define your workspace to display Unit U03 and coordinate system U03 CS.
- Make sure you are in the Piping task and the Active Permission Group is set to Piping.

1. Activate PinPoint. Set the active coordinate system to **U03 CS** and select **Set Target to Origin**.

2. Select **Edit > Paste from Catalog**. Browse for **Tank-01** in **Modules > Training > Piping Class**, and click OK.



3. In the **Place Macro Dialog Box**, click **OK**.
4. In the pinpoint ribbon, key in the coordinates shown below.

5. Left click in the graphic view to accept the position of Tank-01.

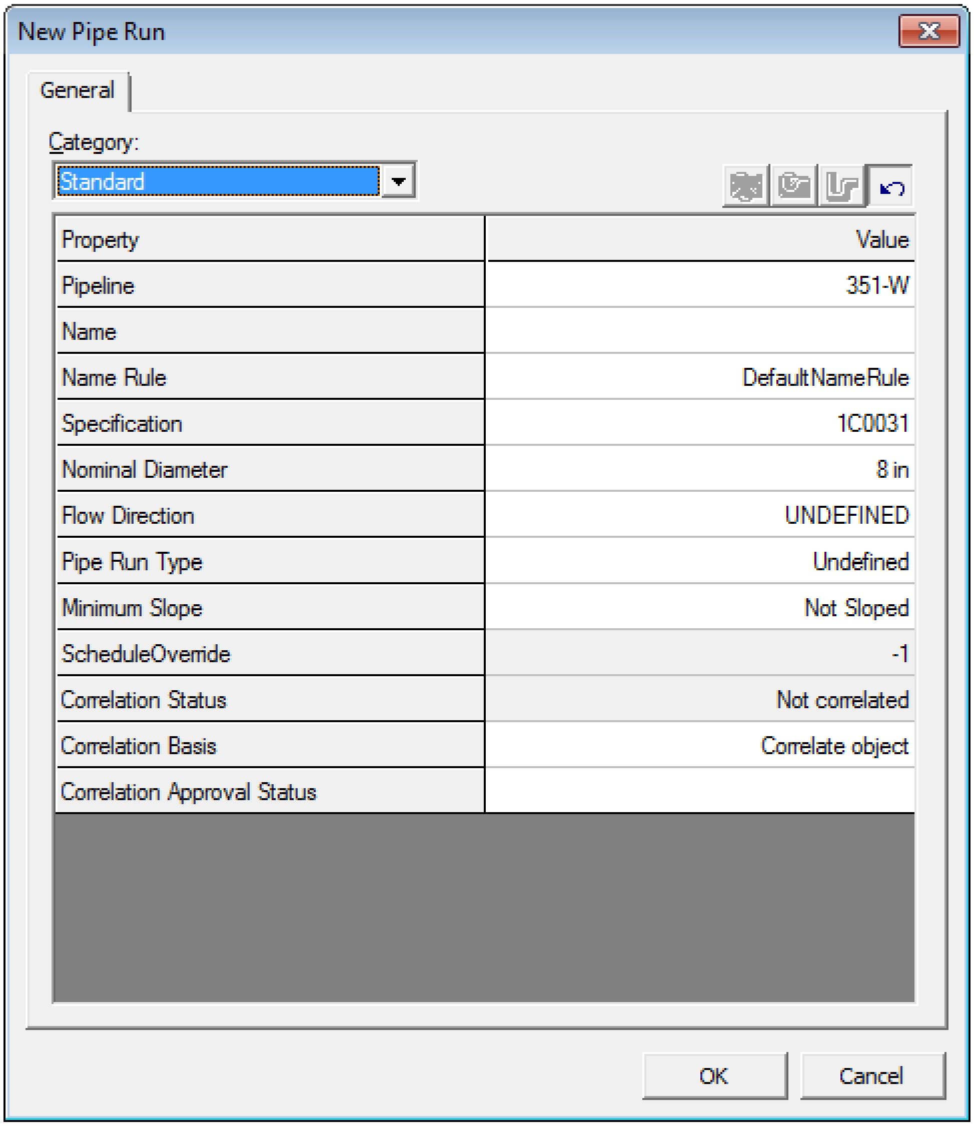
6. Select **Reposition Target** . Find and select the center point of the base of the tank.

7. Click **Route Pipe** .
8. Select **Arc Routing**
9. Input the coordinates shown below in the **PinPoint** ribbon bar to establish the starting location of the arced piping.

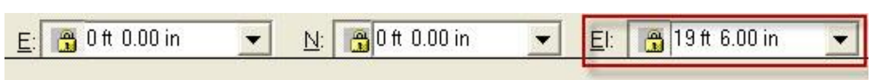
10. Left-click to accept the location.

    The New Pipe Run dialog will appear.

11. Enter the data shown below, and then click **OK**.

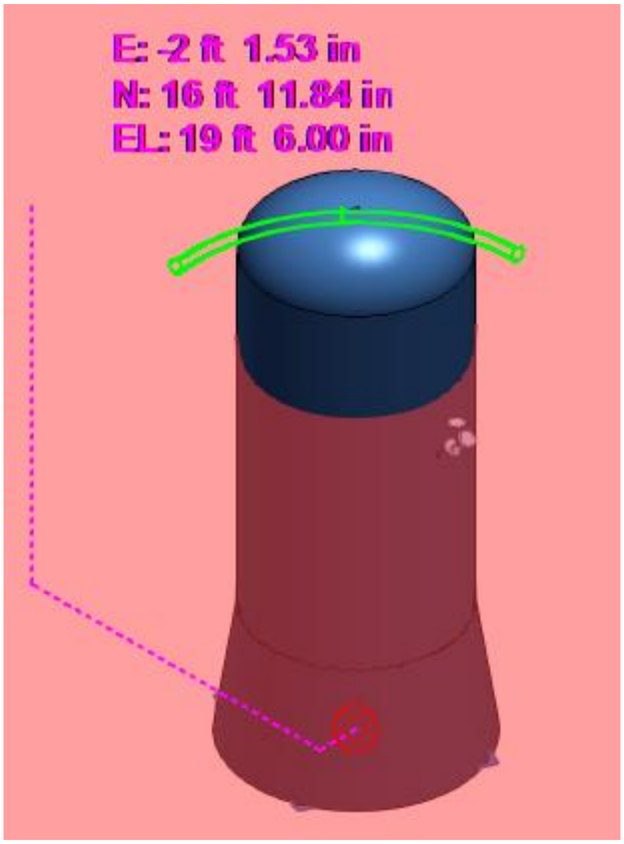


12. To select the center location of the arc, key in the following in the Pin Point ribbon bar and then left-click to anywhere in the graphical display to accept the center location.



13. Select **Plan Plane** to lock the path the arc piping will take.

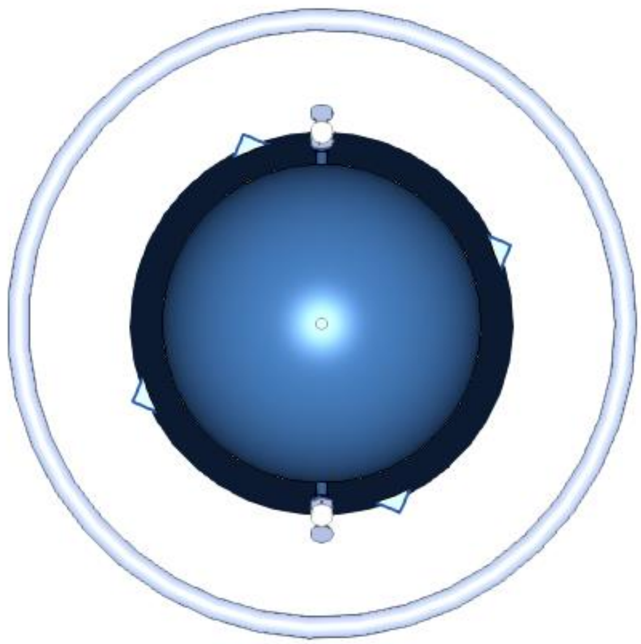
14. Left click in the graphic view at any point to accept the arc path. The length of the route will be established next.



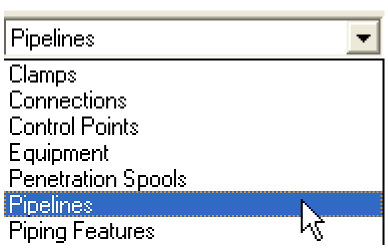
15. Change **Angle** to **360 deg**

16. Click **Finish**.

The piping should look like this.



17. Click Select, and select **Pipeline** from the **Locate Filter**.

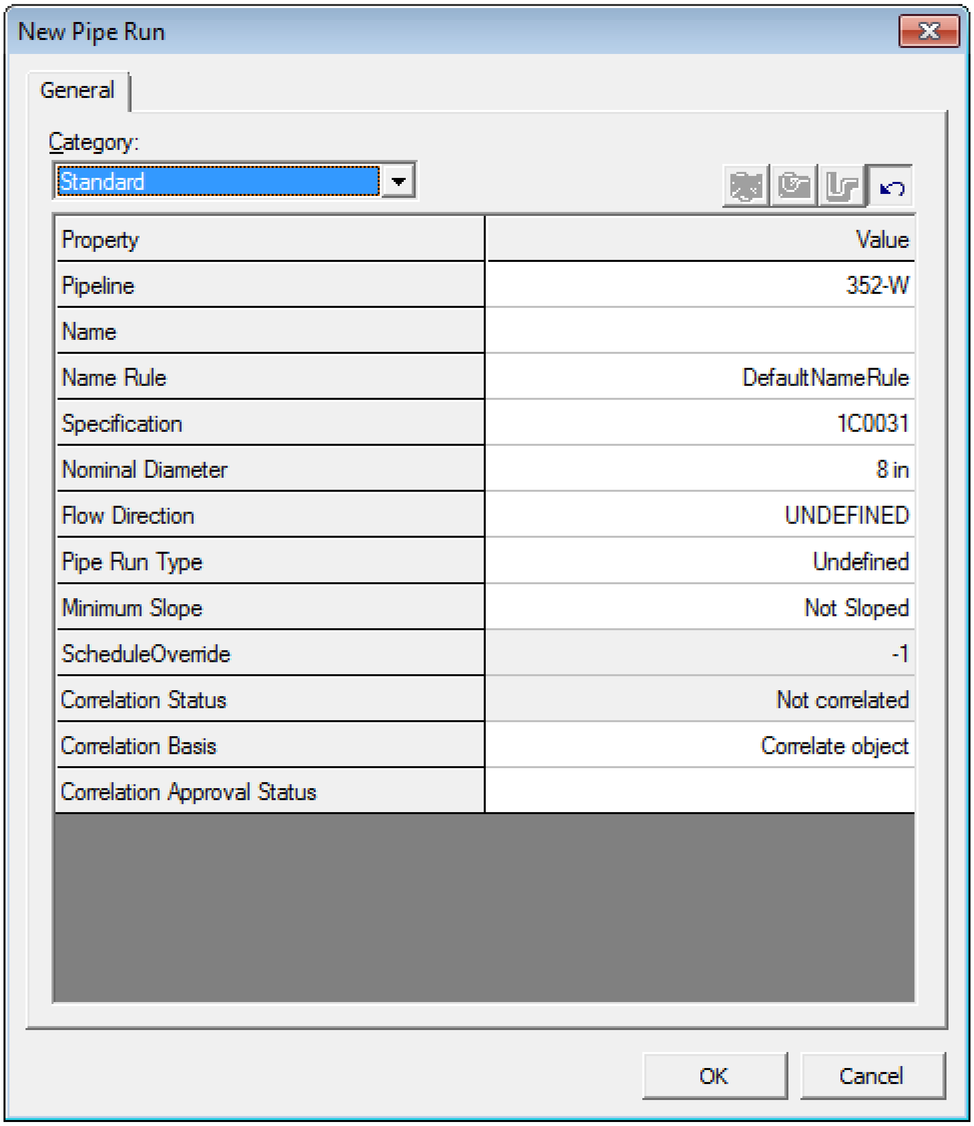


18. Select the **351-W Pipeline** that you just routed.
19. Select **Tools > Hide**.

    The pipeline should not be visible after executing this command.

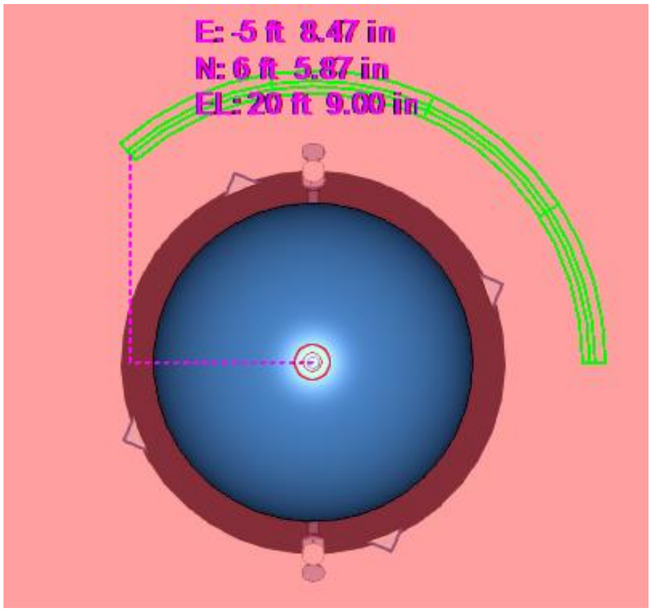
20. Select **Route Pipe**.
21. Select **Arc Routing**.
22. Select the starting location at East 8’ 9", North 0, and Elevation 20’ 9" by entering these values in the pinpoint ribbon.
23. Left click in the graphical display.

24. In the **New Pipe Run** dialog, enter data to match below, and then click OK.

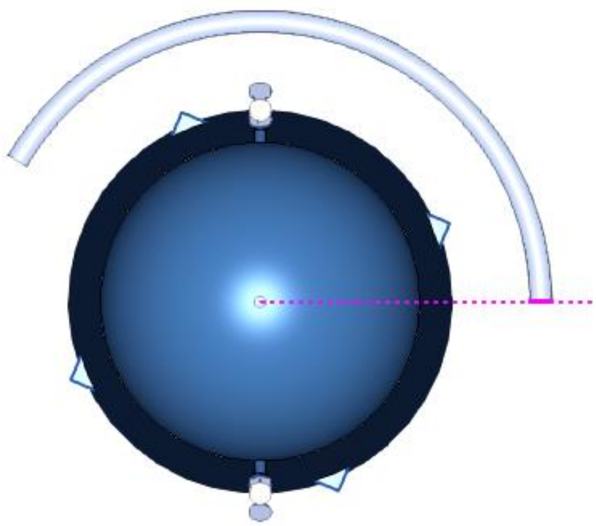


25. For the center point use East 0, North 0, Elevation 20’ 9". Key these values into the pinpoint ribbon. Left click to accept the center location of the arc piping.

26. Select **Plan Plane** to lock the path the arc piping will take. Left click in the graphic view at any point to accept the arc path. The length of the route will be established next.

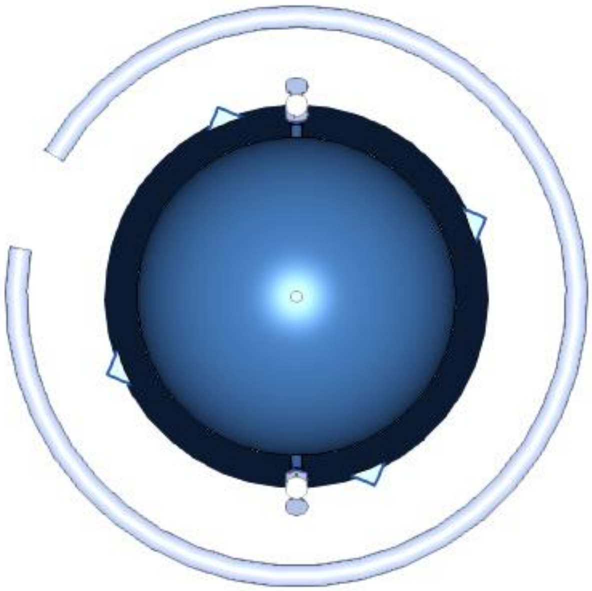


27. Change the **Angle** to **150.00 deg** and click **Finish**.
28. Click Select, and select **Piping Features** from the **Locate Filter**.
29. Select the **End Feature** of the first placement point of the pipe as shown.



30. Change the **Angle** of this location to **340 deg**.

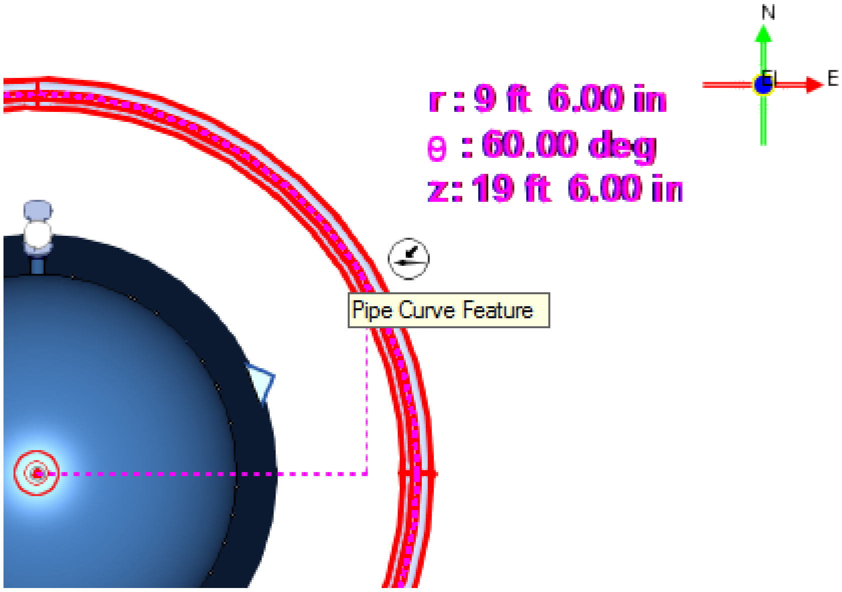
The finished piping should look like this.



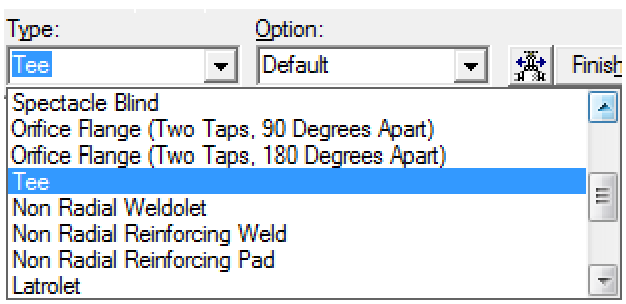
Inserting Components on Arc Pipe

1. Select **Tools > Show All**.
2. Change the **Locate Filter** to **Pipelines.**
3. Select the **352-W Pipeline**, and right-click and select **Hide**.
4. Click **Cylindrical Coordinate System**  on the PinPoint ribbon.
5. Click **Insert Component** , and enter **60 deg** for **Theta** in the ribbon bar.

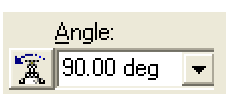
6. Select the pipe by left-clicking it in the graphic view. Be sure that **Theta** is **60 deg** on the pinpoint coordinate rather than 240 deg.



7. Select **Tee** from the **Type** list.



8. Set **Angle** to **90 deg**.



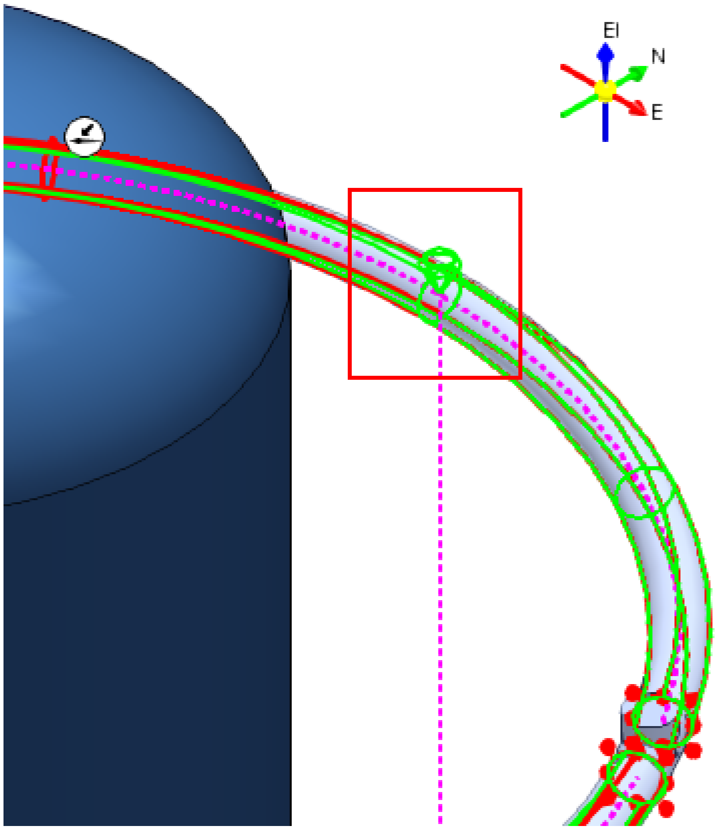
9. Click **Finish**.

NOTE The branch port of the **Tee** should be in the UP direction. Adjust the **Angle** if needed.

10. Click **Insert Component**, and enter N 0 deg for **Theta**.

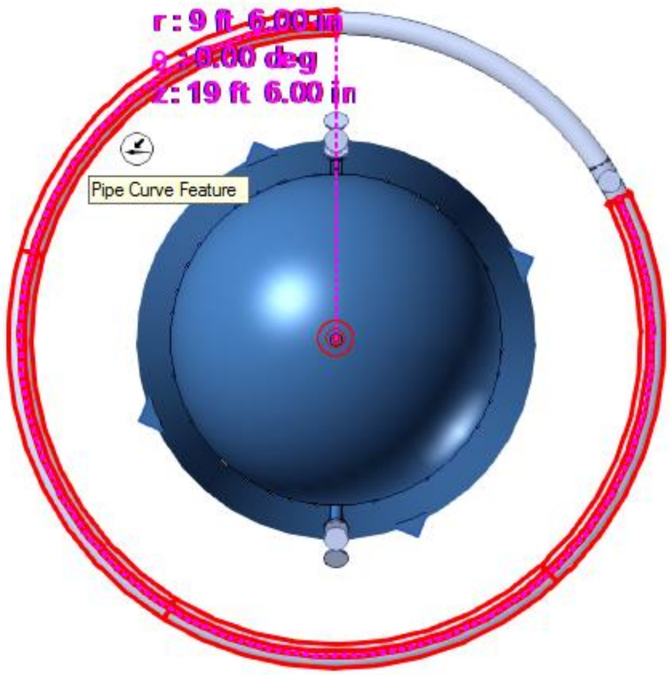
11. Select the pipe by left-clicking it in the graphic view.
12. Select **Weldolet** from the **Type** list.
13. Then select **<New Pipe Run>** from **Run** list.
14. Keep all other default settings but choose **4"** for the **Nominal Diameter**, and then click **OK**.

15. Leave **Angle** as **0 deg**, which should point the outlet of the weldolet up as shown below, and then click **Finish**.



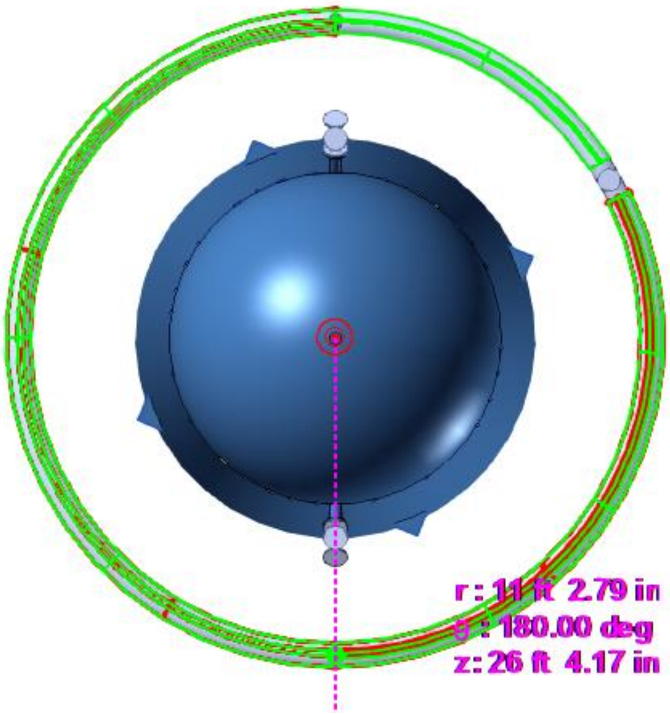
16. Click **Insert Component**, and enter N 0.00 deg for **Theta**.

17. Hover over the pipe in the graphic view. The graphical display should resemble below. Left-click on the pipe.

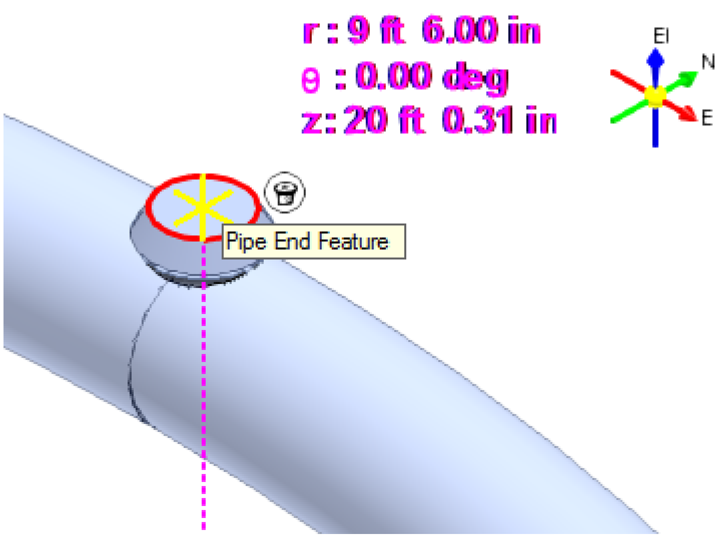


18. Select **Weldolet** from the **Type** dropdown.
19. Then select **<New Pipe Run>** from the ribbon bar.
20. Keep all other default settings but choose **4"** for the **Nominal Diameter**, and then click **OK**.
21. Click **Enter Insertion Point** on the ribbon. Key in **S 180 deg** for **Theta**.

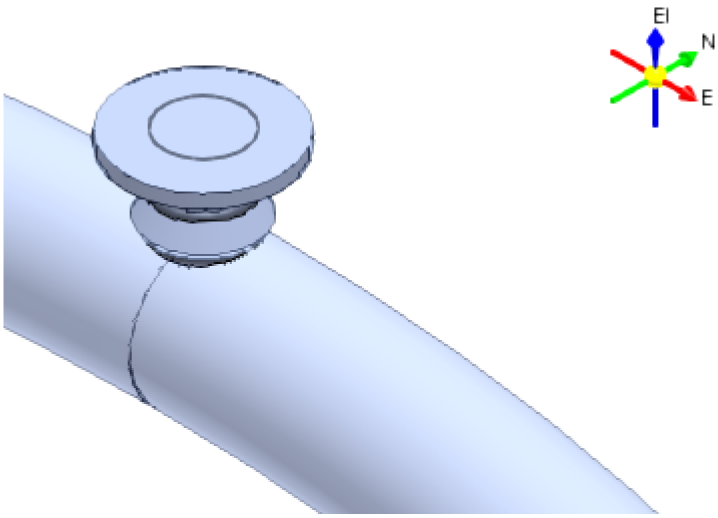
22. Move the mouse to the southern most point of the display as shown below. Left-click in the graphic view. Click **Finish**.



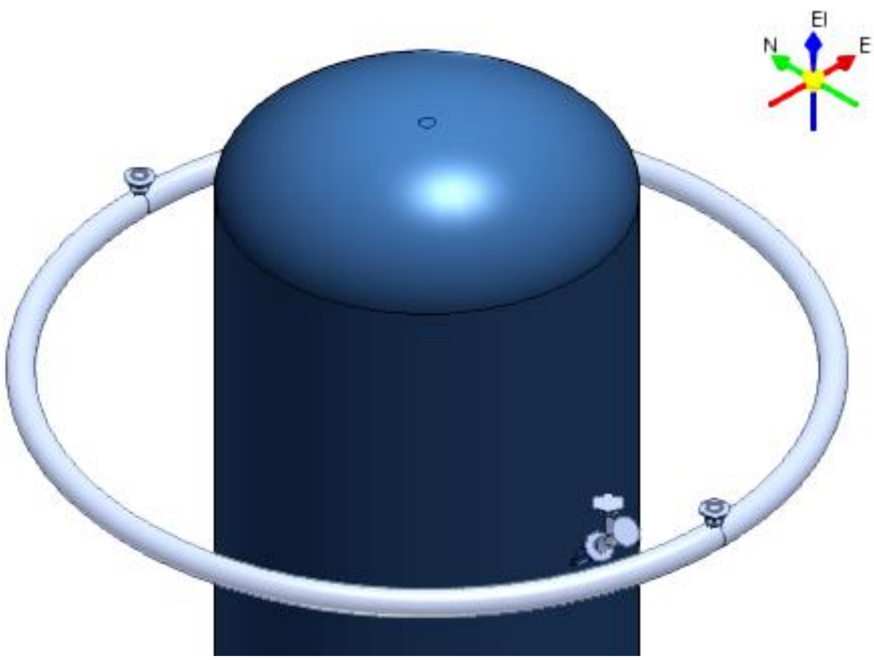
23. Click **Insert Component** , and select the outlet of either weldolet.



24. From **Type** list, select **Flange**, and then click **Finish**.

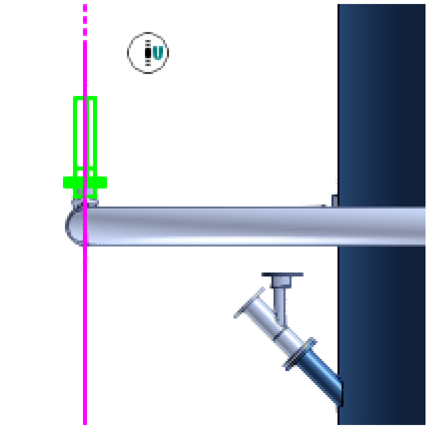


25. Insert a Flange on the other weldolet. After inserting the Flanges, the configuration should look like this.

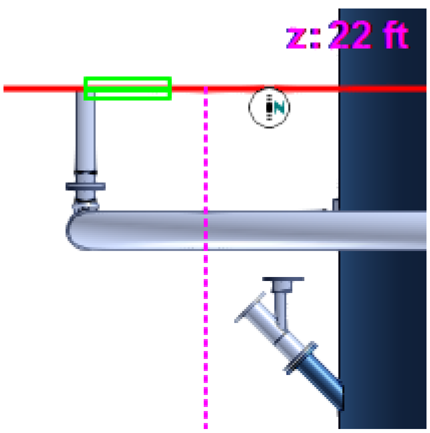


26. Change the view to **Looking East**, and click Route Pipe. Keep the straight routing option then select the flange on the north side of the tank to begin routing. Keep all defaults in the **New Pipe Run** dialog.
27. Confirm that the plane is set to **No Plane**.
28. Enter **1' 6"** for length of the pipe.

29. Route the pipe in the Up direction as shown below.

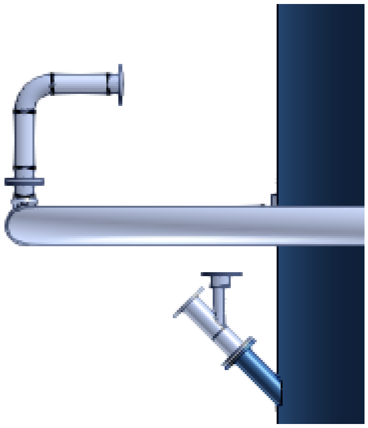


30. Left-click in the graphic view to place the pipe.
31. While still in the **Route Pipe** command, route **1’ 6"** south towards the centerline of **Tank-01**. A Northing Glyph should appear.

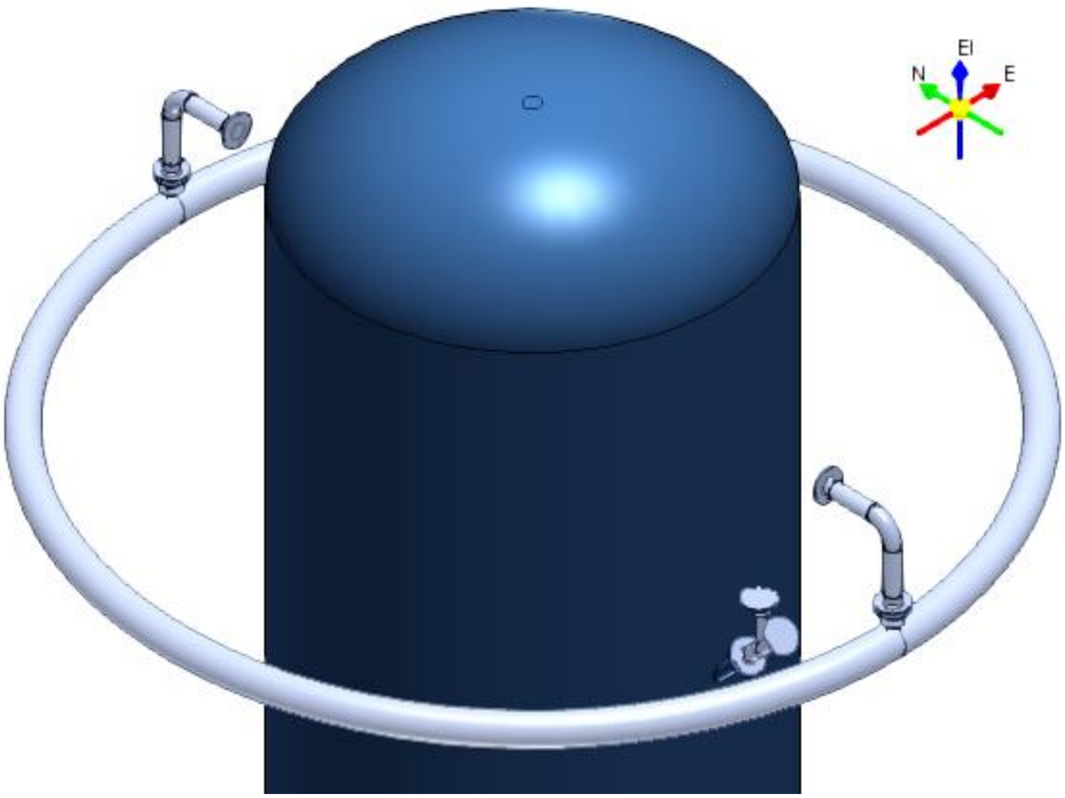


32. Left-click in the graphic view to place the pipe.
33. While still in the route command, click **Insert Component** .

34. Select **Flange** from the **Type** list, and then click **Finish**.



35. Right-click in the graphic view to terminate the route. Repeat the routing procedures for the other weldolet. Be sure to route toward the centerline of **Tank-01**. The configuration should look like this when complete.



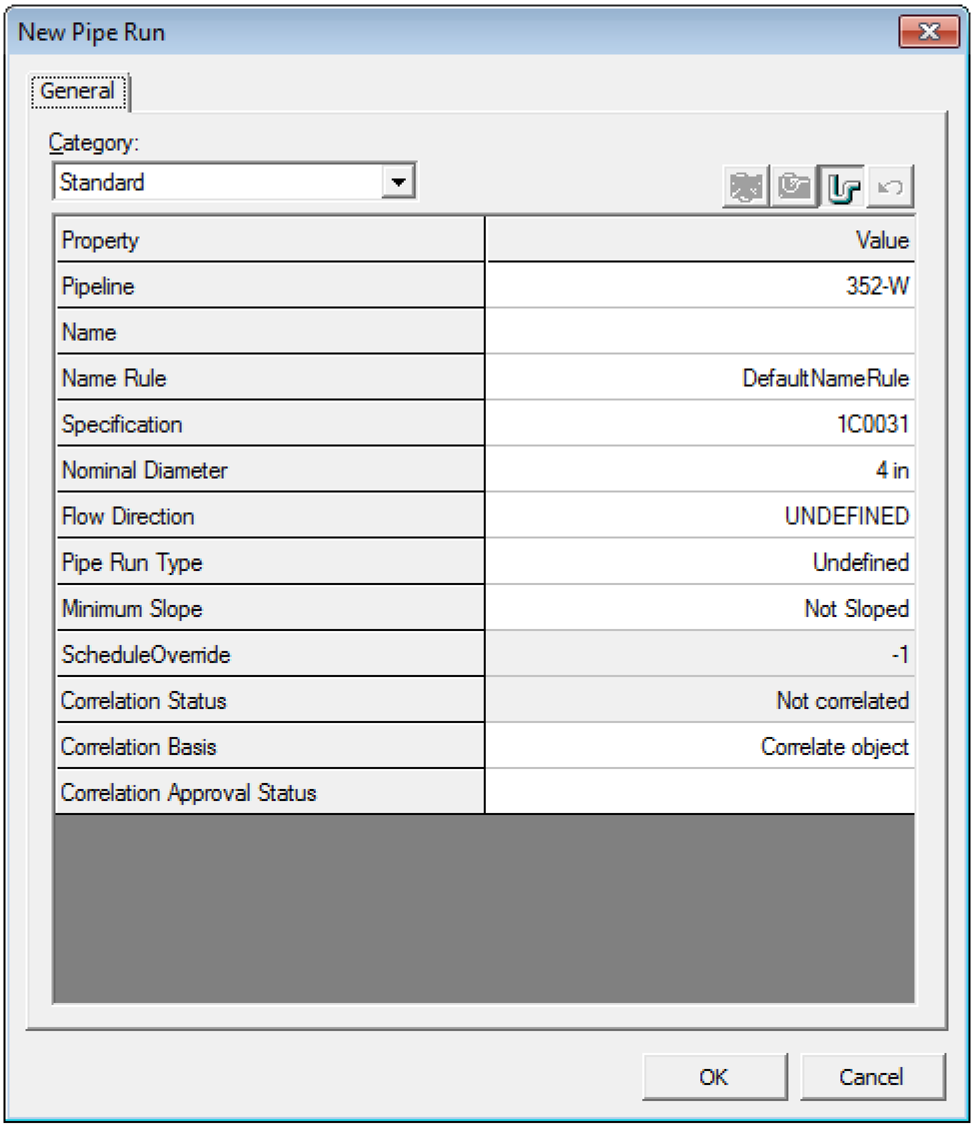
36. Select **Tools > Show All**.

37. Select Pipeline 351-W by left clicking it in the graphic view. Be sure that the **Selection Filter** is set to **Pipelines**.

38. Select **Tools > Hide**.

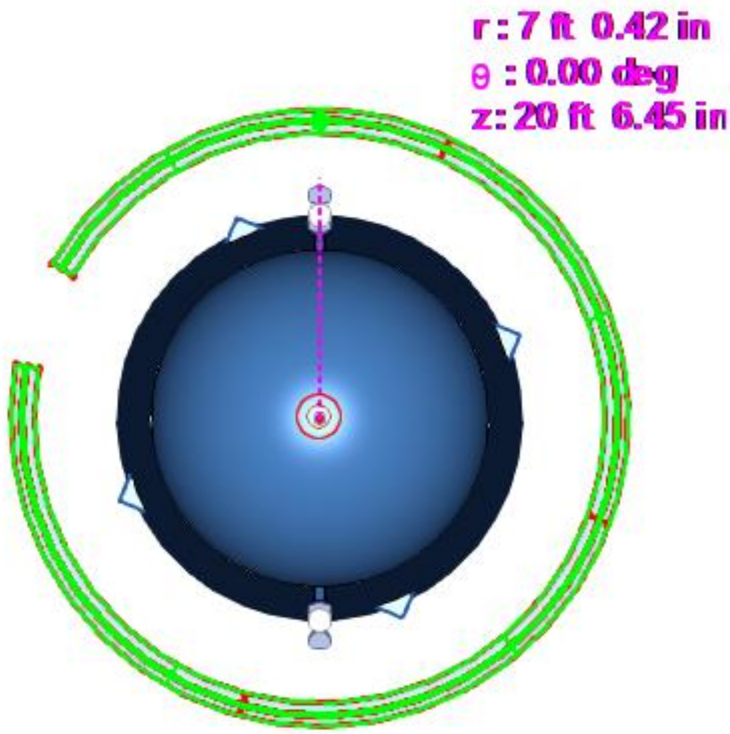
39. Click **Insert Component** .

40. Select the pipe by left-clicking it in the graphic view.
41. Select **Weldolet** from the **Type** dropdown.
42. Select **New Pipe Run** from **Run** list.
43. Choose **4"** for the **Nominal Diameter** in the **New Pipe Run** dialog, and then click **OK**.



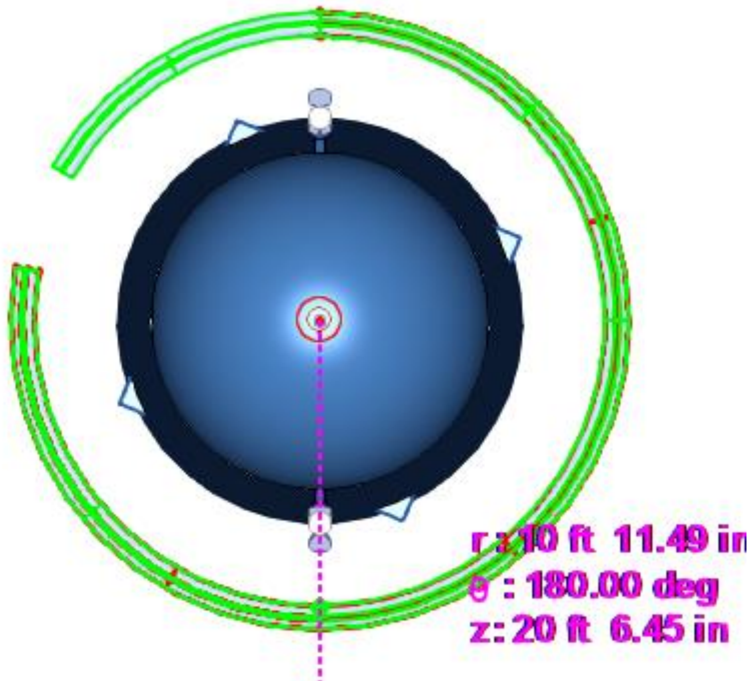
44. Click **Enter Insertion Point** . Enter **0 deg** for **Theta** in the ribbon bar.

45. Keep the mouse at the north end of the vessel as shown. Left-click in the graphic view.

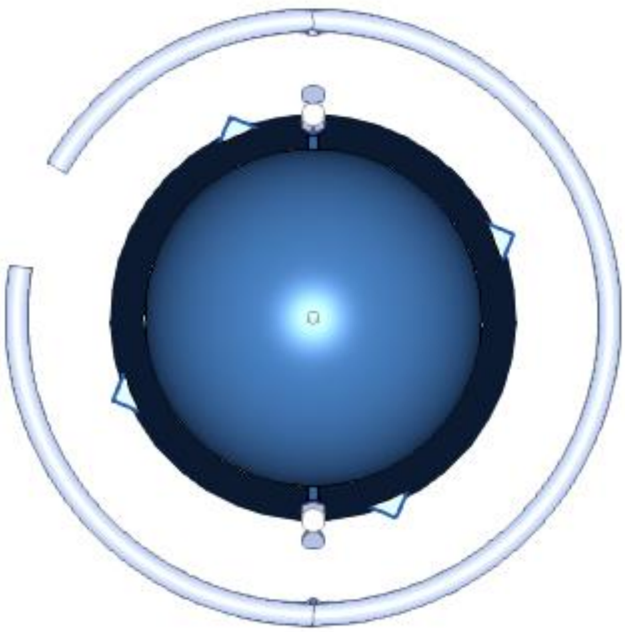


46. Enter an **Angle** of **135 deg**, and click **Finish** to place the weldolet in the graphic view.

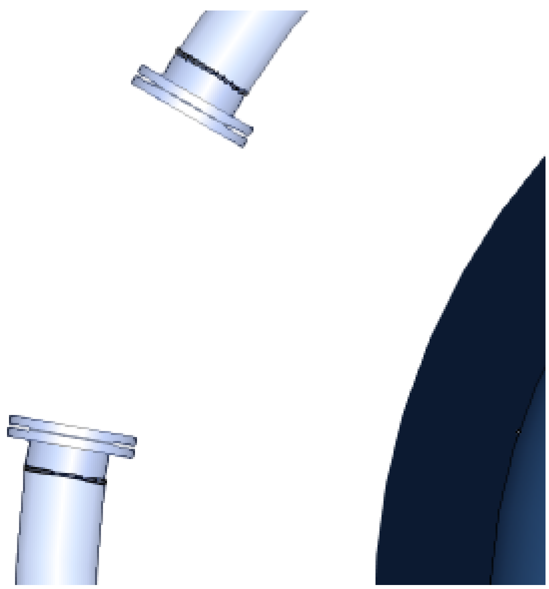
47. Insert another weldolet where Theta equals 180 degrees. This means keeping the mouse at the south end of the vessel when positioning the weldolet. Before the weldolet is placed, the graphic display should resemble this.



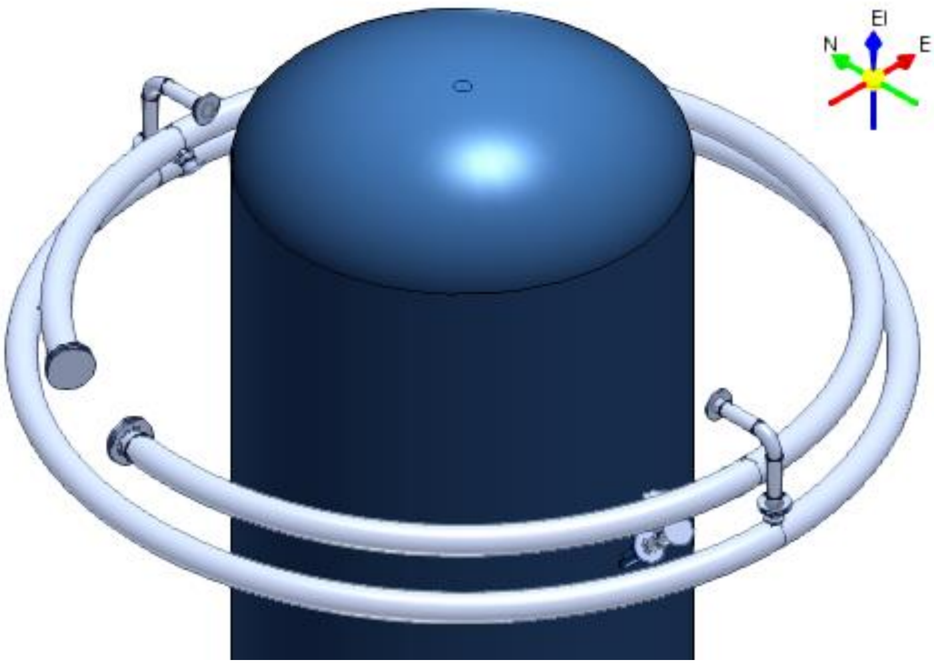
The configuration should look like this when completed.



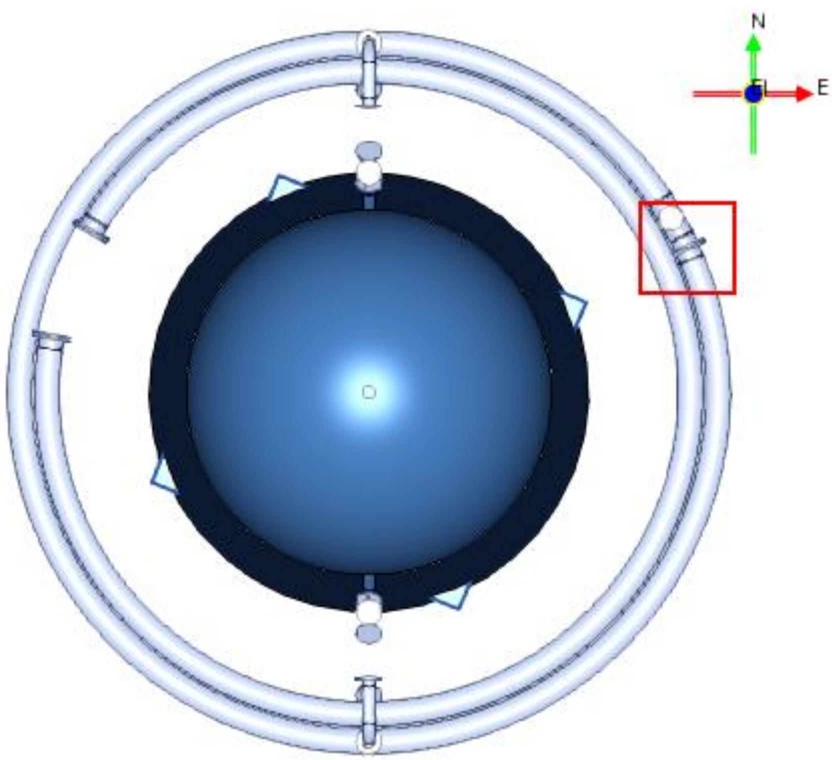
48. Insert flanges on the weldolets just placed by using **Insert Component**.
49. Insert a Blind flange on each open end of Pipeline 352-W using **Insert Component**. The configuration should look like this.



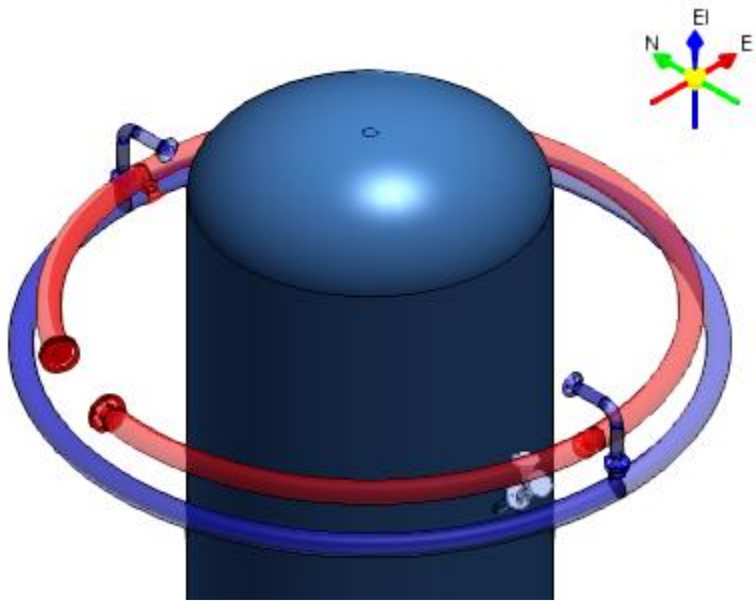
50. Select **Tools > Show All**. The result should resemble this.



51. On Pipeline 351-W, insert a set of Flanges adjacent to the Tee and with Theta angle of NE 65 degrees similar to this:



52. Select **Format > Surface Style Rules**.
53. Select Flex, 351-W, and 352-W from the left pane and click **Add**.
54. Click **OK** on the Surface Style Rule dialog. The result will look like this.

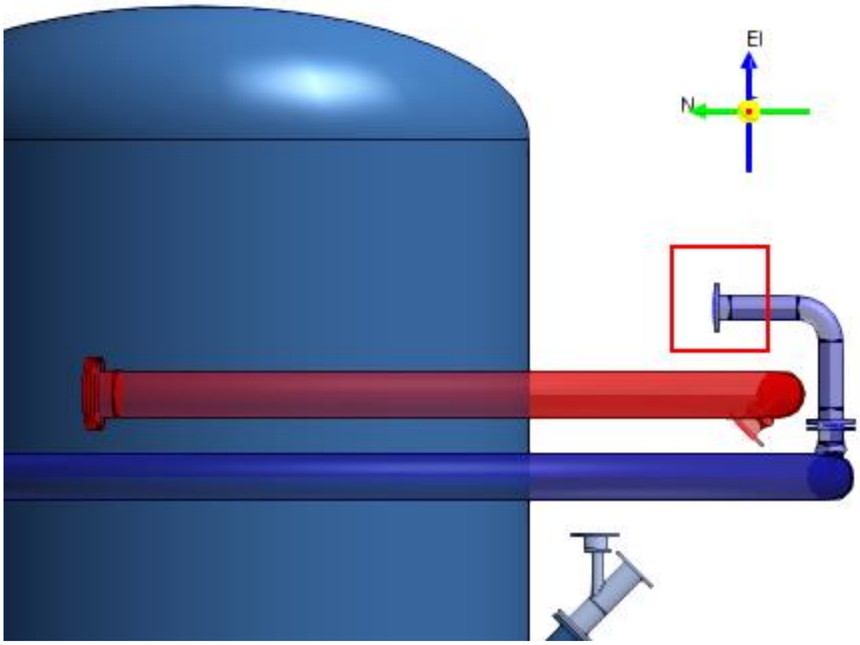


Routing Flex Pipe

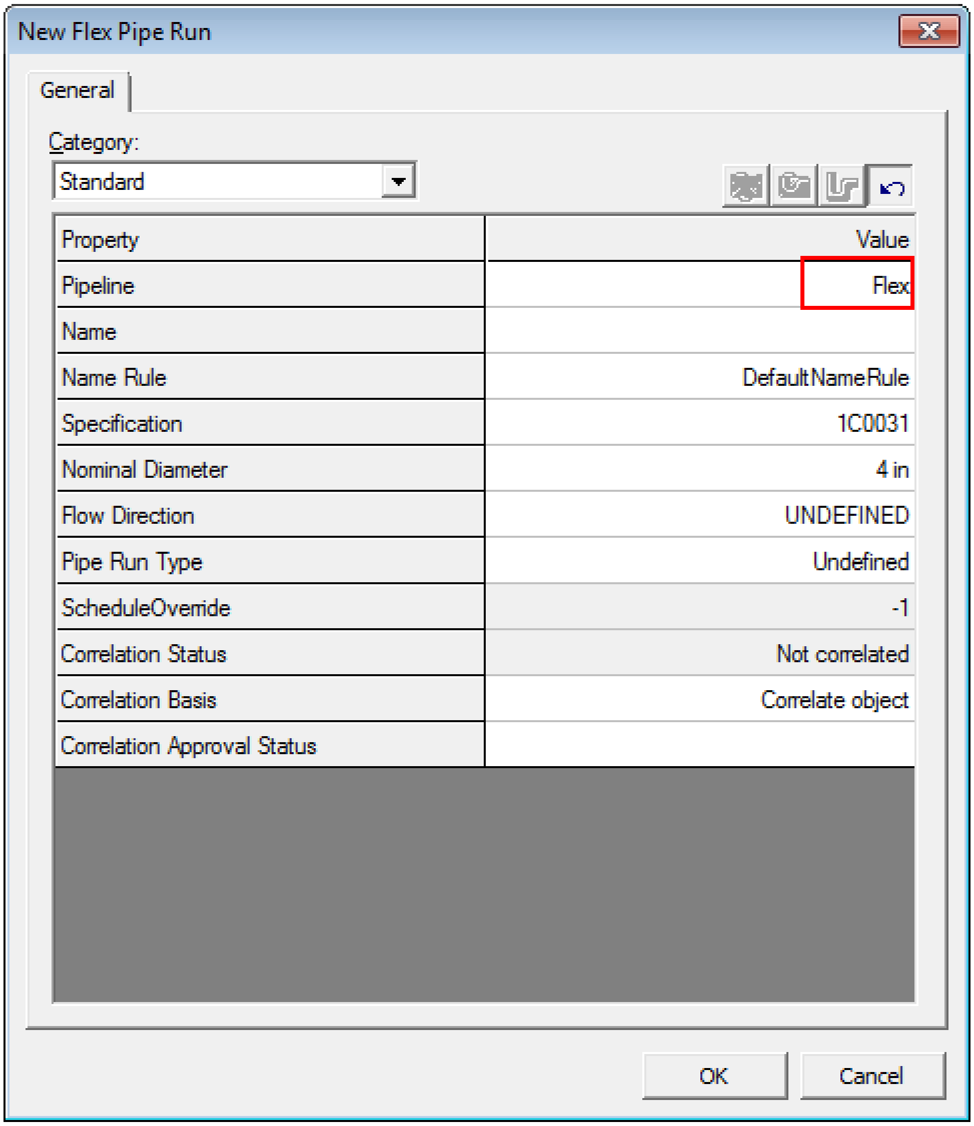
1. Click **Route Flex Pipe**

You are prompted to select an End\Nozzle\Component port to Begin Routing.

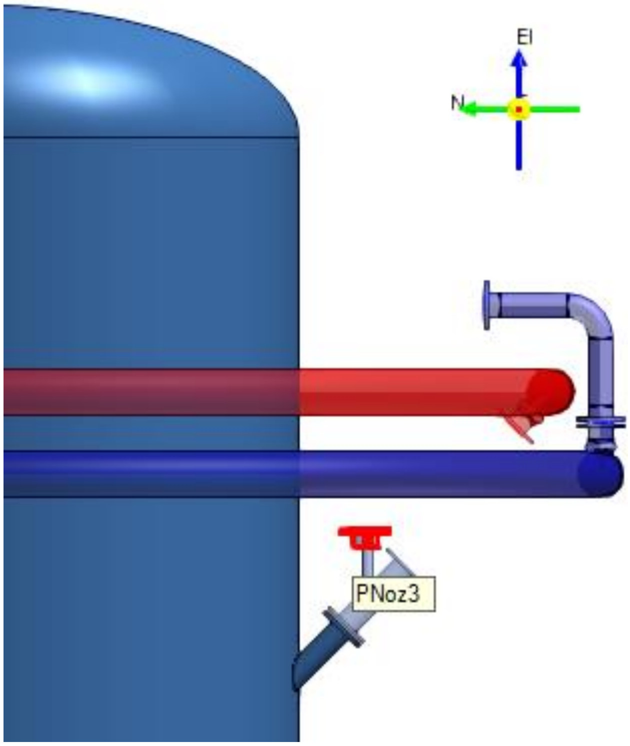
2. Choose flange pointing N on south of tank from pipeline **351-W** as shown.



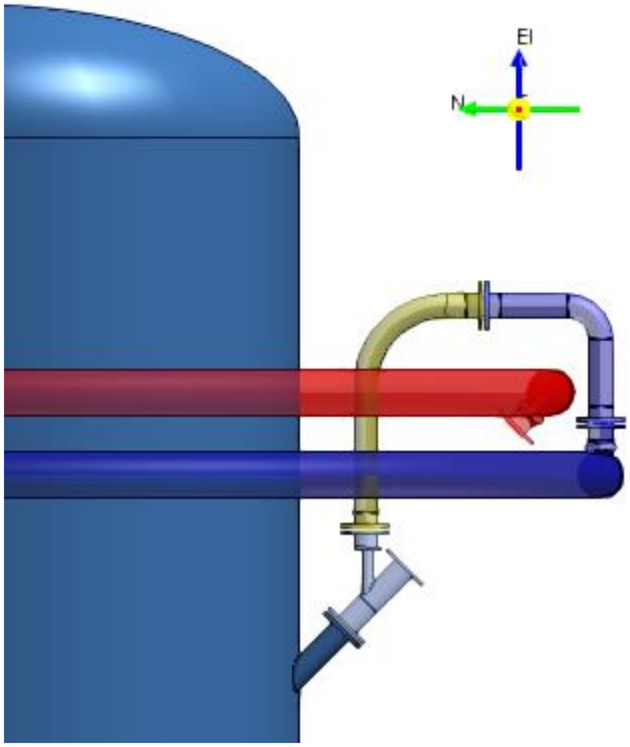
3. For the Pipeline Name, browse for the Flex Pipeline when the New Flex Pipe Run dialog appears, and then click **OK**.



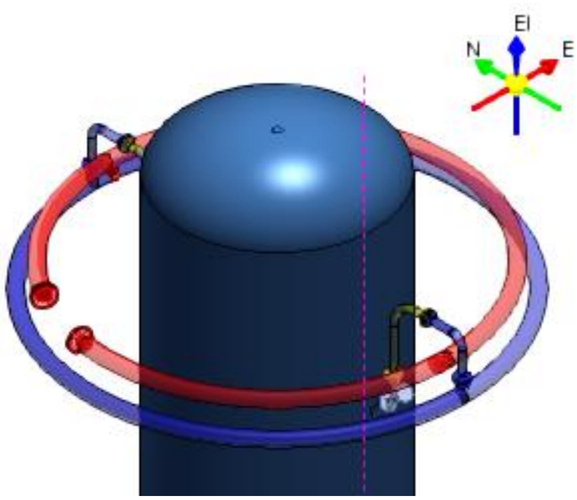
4. For the second point, choose the flange on the component on the Y-Strainer as indicated below.



5. Click **Finish** to place the pipe in the model.



6. Repeat these steps for the other flange on N side of tank in pipeline 351-W.
7. The finished configuration should look like the following.

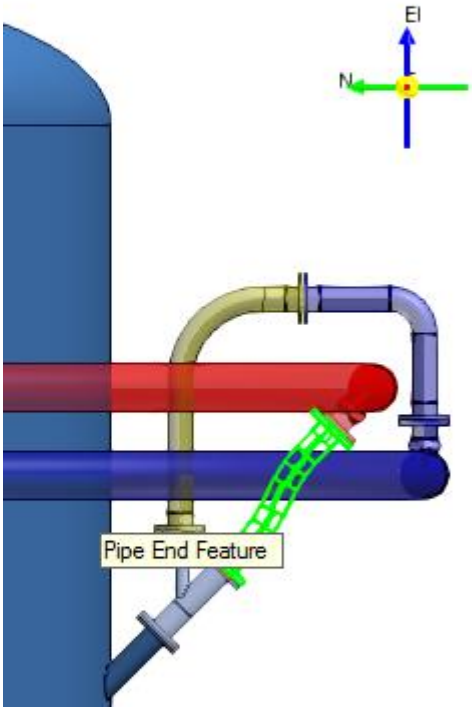


8. Click **Route Flex Pipe** , if it is not already selected.

   You will be prompted to select an End\Nozzle\Component port to Begin Routing.

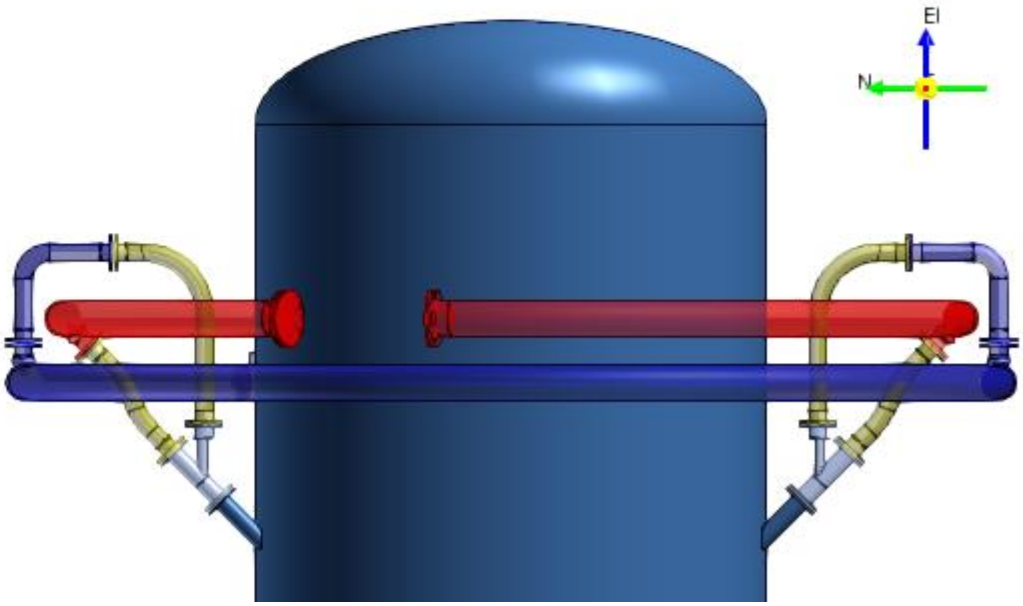
9. Choose a flange on Pipeline 352-W.

10. For the Pipeline Name, browse for the Flex Pipeline when the **New Flex Pipe Run** dialog appears, and then click **OK**.
11. For the second point, choose the flange on the component on the equipment nozzle closest to the initial route point.



12. Click **Finish** to place the pipe in the model.
13. Repeat these steps for the other flange in 352-W.

14. The finished configuration should look like the following.



SECTION 1

Manipulating Piping Objects

Objective

By the end of this session, you will be able to:

- Modify pipes, their features, and piping objects.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)

Overview

Smart 3D provides commands to modify a pipe run and its features. With these commands you can edit, copy, move, rotate and delete pipe runs and their features such as end, along leg, straight, branch and turn. To use the manipulation commands you first need to route a pipe run or complete the placement of components. These commands require the selection of a pipe run, or its feature as first step by using **Select** on the vertical toolbar.

The manipulations that you can perform to pipe runs and their features are as follows:

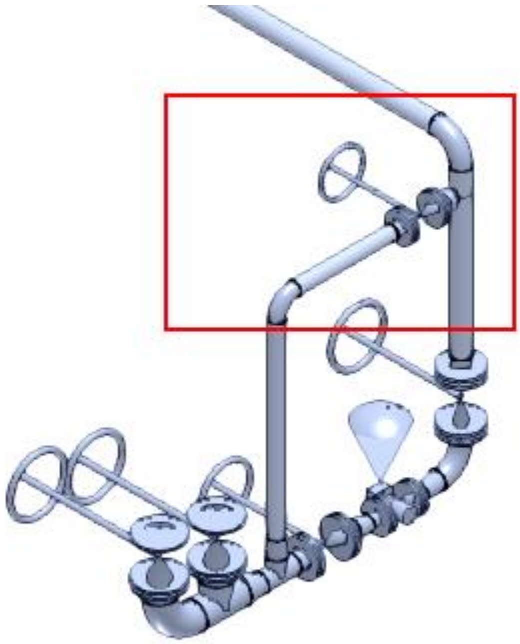
- **Move:** After routing a pipe run, you can move the features of that run. You can use the **PinPoint** or **SmartSketch** tools to move the desired feature to any location by typing delta distances, or absolute distances. As you move the features, the components are regenerated driven by the topology and the specifications that are accessed during the move operation.
- **Copy:** You can copy a pipe run or a feature to place it in a different pipeline.
- **Modifying Properties:** All the pipe runs and their features have properties that you can edit.
- **Delete:** Just as you move features to re-generate the parts in the model, you can also delete features to remove unwanted parts.
- **Rotate:** You can rotate a pipe, a piping object, or a branch port interactively in Smart 3D. You can rotate the branch port only when nothing has been connected to the branch port.

While working in an integrated environment you can also update the custom instruments in Smart 3D after Instrumentation Dimensional Datasheet (DDP) is changed in SmartPlant Instrumentation.

Moving Pipe Straight Features

Objective

In this exercise you will be moving the pipe runs from pipeline 400-P 1 ft from the runs current position. After moving these pipe runs the graphic view should resemble this.

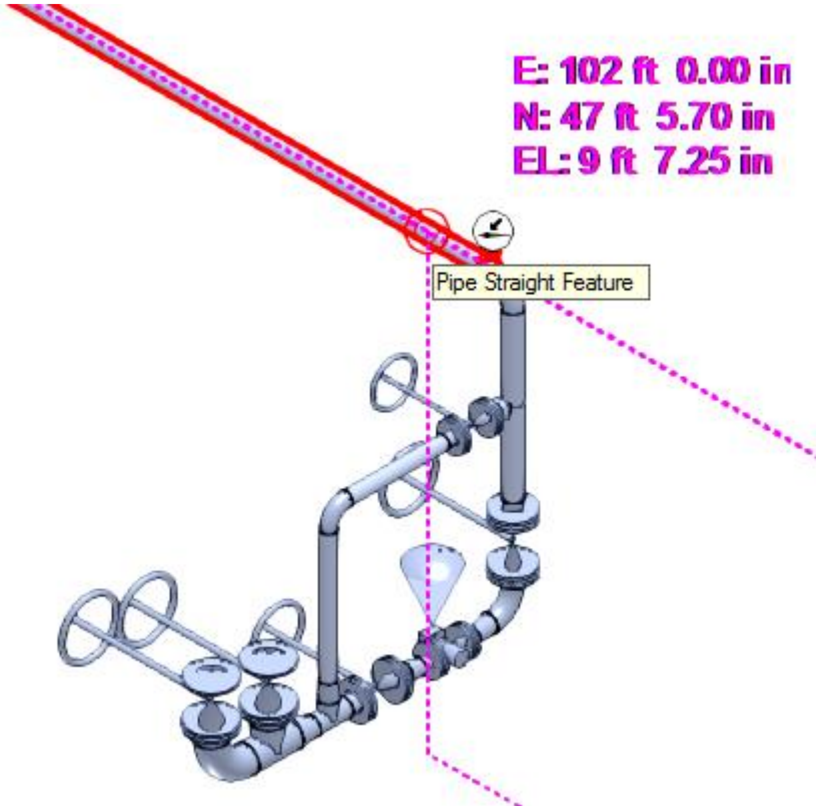


Before Starting this Procedure

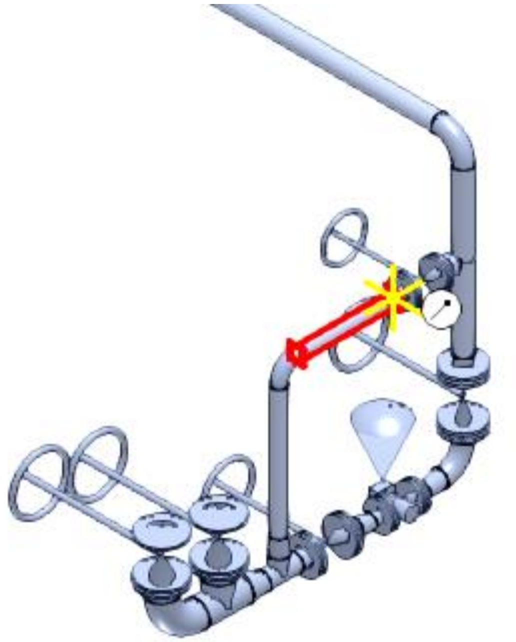
- Define your workspace to display Unit **U04** and the coordinate system **U04 CS**.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Select **Tools > Pinpoint** to activate the **PinPoint** ribbon.
2. Select **Piping Features** in the **Locate Filter** drop-down list to select only piping features in the graphic view.
3. Select **Reposition Target**  on the **PinPoint** ribbon to change the target origin.

4. Select the **Pipe Straight Feature** of the pipeline 400-P, as shown, to specify the centerline of the pipe as origin.



5. Select the **Pipe Straight Feature**, as shown, to specify the piping feature that you will move with respect to 400-P.



TIPS

- While moving a Pipe Straight Feature, the entire pipe run to which the feature is connected moves.
- The move direction is always perpendicular to the axis of the **Pipe Straight Feature**.
- A branch feature connected to the moved leg maintains its original angle.

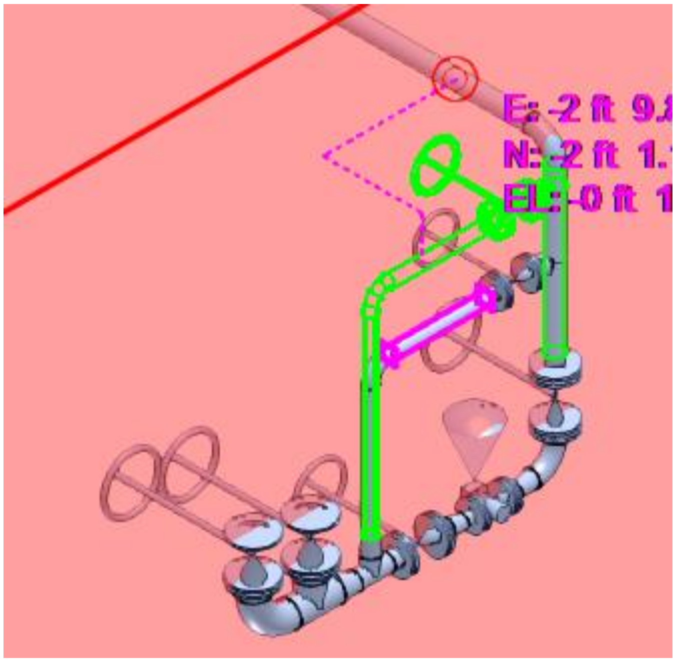
When you select the Pipe Straight Feature, an Edit ribbon appears.

6. Select **Move From** option on the **Edit** ribbon to specify the new position of the pipe feature to be moved.

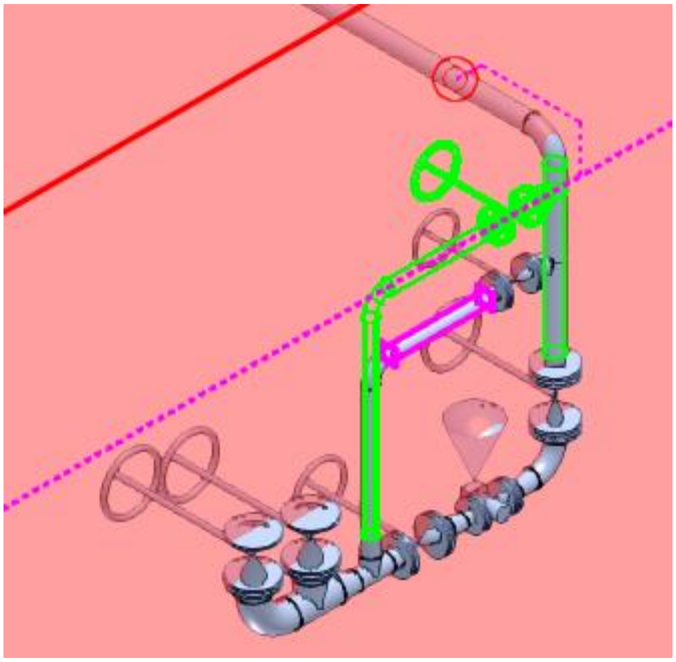
The selected pipe appears with bordered outline in the graphic view.

7. Select the **Pipe Straight Feature** to be moved.

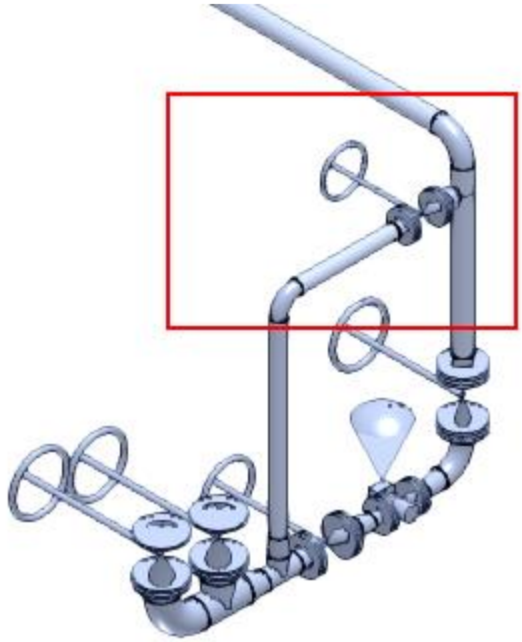
The selected pipe appears with bordered outline in the graphic view, as shown.



8. Key in **-1 ft** for **El** on the PinPoint ribbon to move and define the position of the pipe straight feature 1 ft from the centerline of the pipe 400-P. Before you accept the placement position, the pipe feature after moving resembles this.



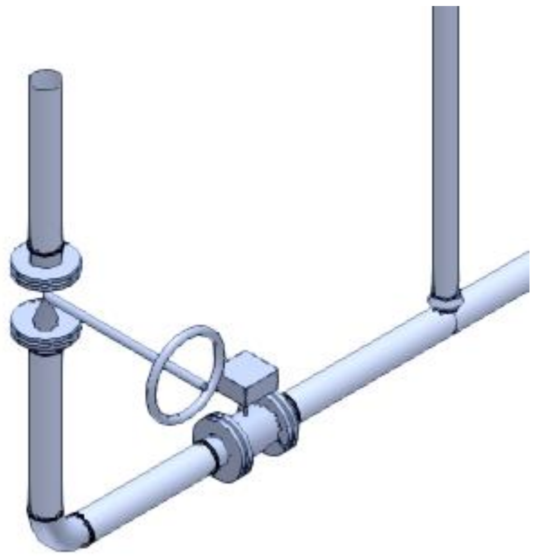
9. Click in the graphic view to accept the position of the moved pipe feature, as shown.



Moving Pipe End Feature

Objective

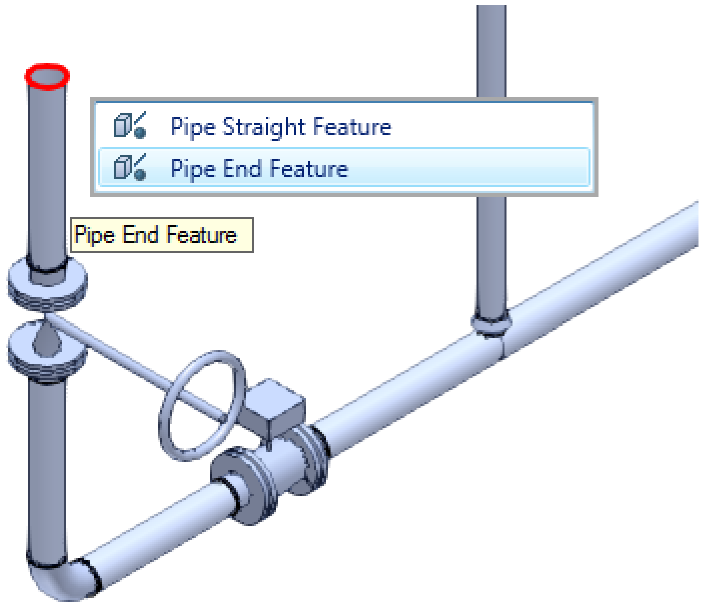
In this exercise, you will be moving the end point of a pipeline **P-204** to a new location by moving the end feature of the pipe run by **6 ft** in the upward direction. After moving the end feature the view of the model in the graphic view should resemble this.



Before Starting this Procedure

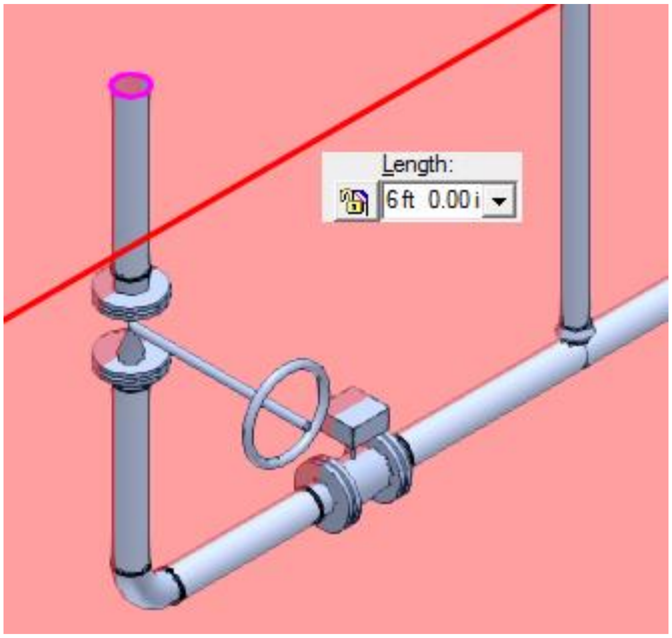
- Define your workspace to display **Area01 > Unit01**.

1. Set the active coordinate system to **Global CS** on the **PinPoint** ribbon and activate the **Set Target to Origin** option.
2. Select the **Pipe End Feature** of the pipeline **P-204**, as shown. You can use the quick pick tool to help you locate the end feature of the pipe.

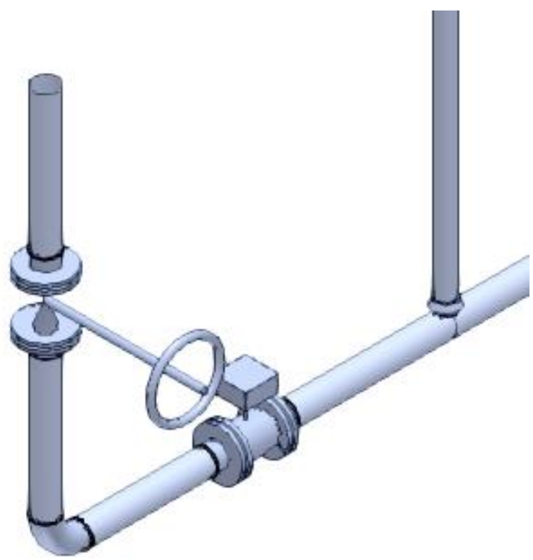


The Edit End Feature ribbon appears. The end feature can be moved by using the Smart Step options available on the Edit End Feature ribbon. You can use the Length drop-down list to type a new length for the pipe. Smart 3D moves the end feature to the specified length.

3. Key in **6 ft** in the **Length** drop-down list on the **Edit End Feature** ribbon to extend the pipe.



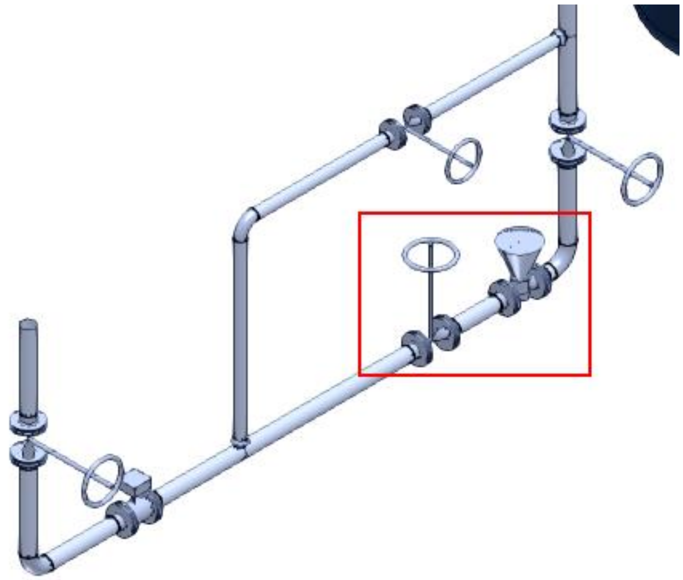
The pipe end is moved 6 feet.



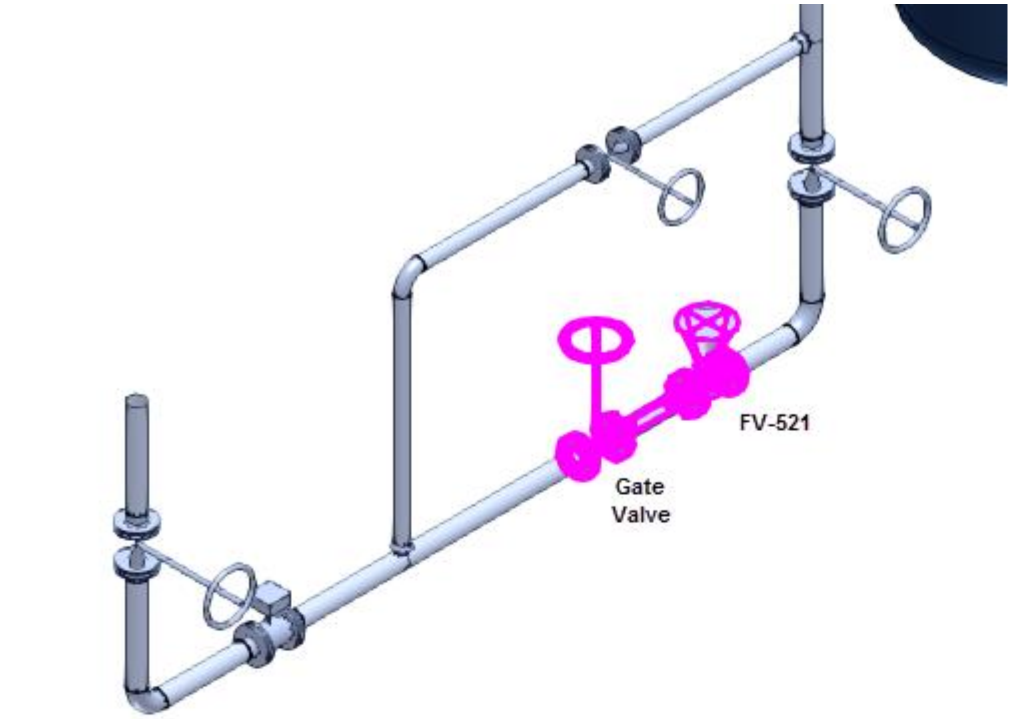
Moving Multiple Pipe Features

Objective

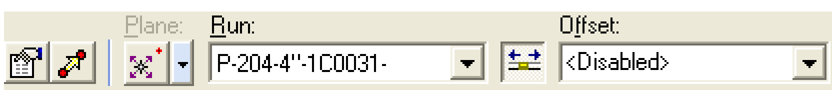
In this exercise, you will be moving the in-line contiguous pipe parts such as a gate valve and the **FV-521** instrument of a pipeline **P-204** along a straight pipe by using the Multi-Select ribbon. After moving the pipe parts the view of the model in the graphic view should resemble this.



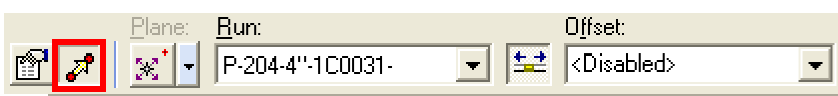
1. Select the features that generate the **FV-521** instrument, pipe and the gate valve located in line **P-204**, as shown. You can use the SHIFT+Select method to help you select all the features.



The Multi-Select ribbon appears when the select set contains more than one feature.

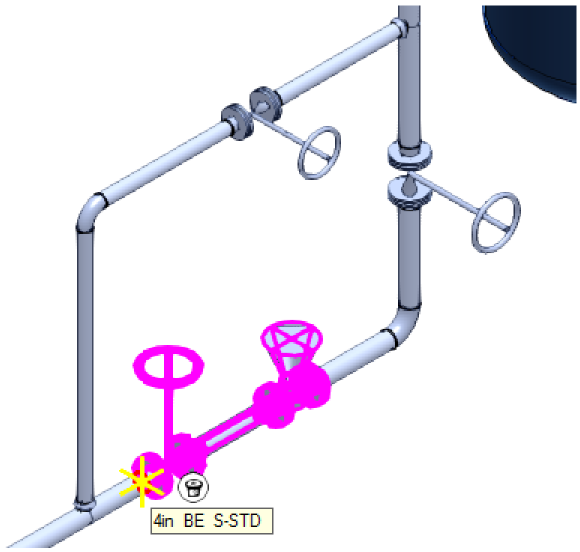


2. Select **Move** on the Multi-Select ribbon to start moving the pipe parts.

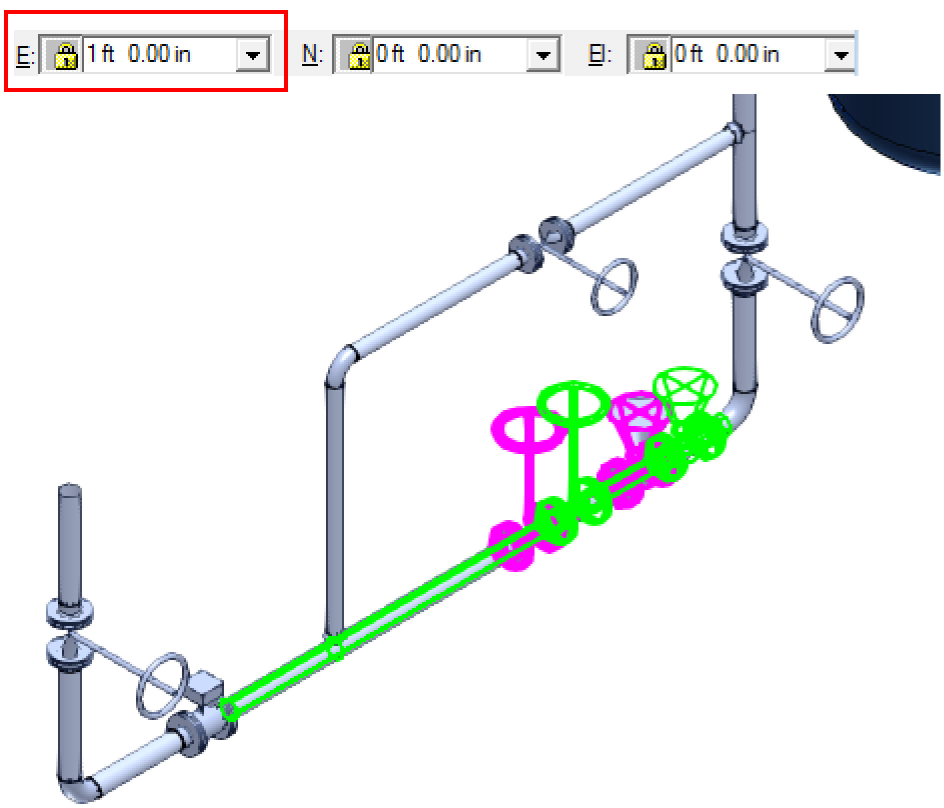


3. Activate the **PinPoint** ribbon and then select **Relative Tracking**

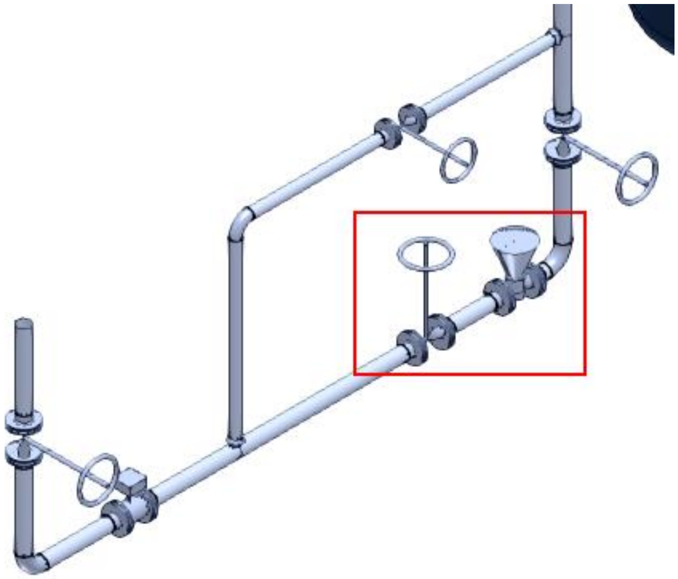
4. Select the buttweld end port of the flange, as shown, to specify the origin for moving the pipe parts.



5. On the PinPoint ribbon, key in **1ft** for easting **E** drop-down list.
6. Move the cursor along the path and Smart 3D constraints your movement along the path, as shown.



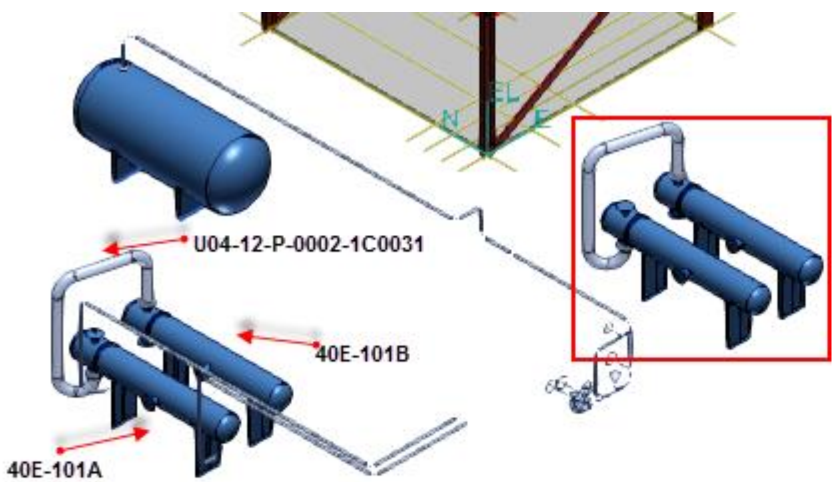
7. Click in the graphic view to finish the move operation.



Copying and Pasting a Pipe Run

Objective

In this exercise you will be copying a pipe run in Pipeline **403-P** along with the equipment **40E-101A** and **40E-101B** to which the pipe run is connected. After moving these pipe runs the graphic view should resemble this.

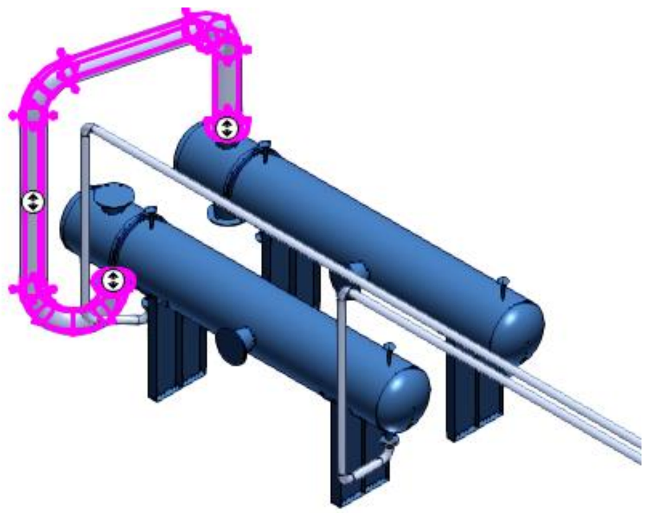


Before Starting this Procedure

- Define your workspace to display **U04** and coordinate system **U04 CS**. In your training plant, select **U04** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

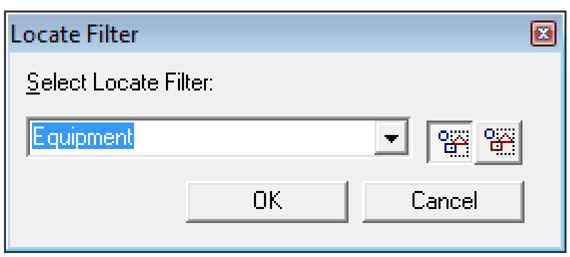
1. Select the **Piping Runs** option in the **Locate Filter** drop-down list to select only pipe runs in the graphic view.
2. Select the pipe run in the pipeline **403-P**, as shown.



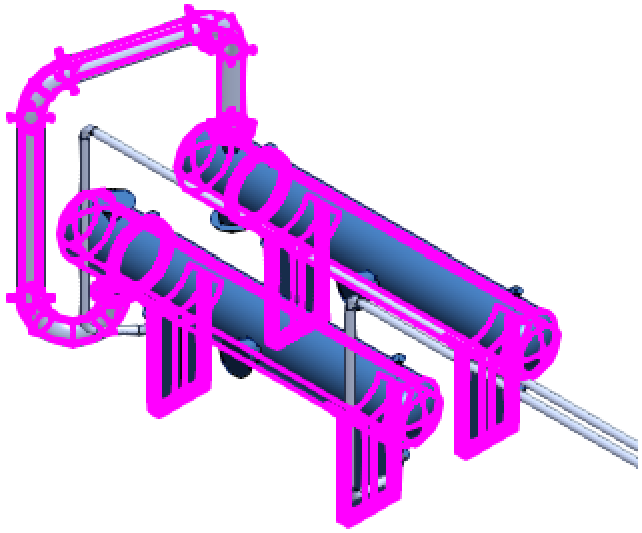
3. Select both equipment to which the pipe run is connected.
4. Select **Edit > Locate Filter** (you may also use shortcut key **Ctrl + E)** to set the locate filter to equipment along with the pipe run.

   The Locate Filter dialog box appears.

5. Select **Equipment** in the **Select Locate Filter** drop-down list, and click **OK**.



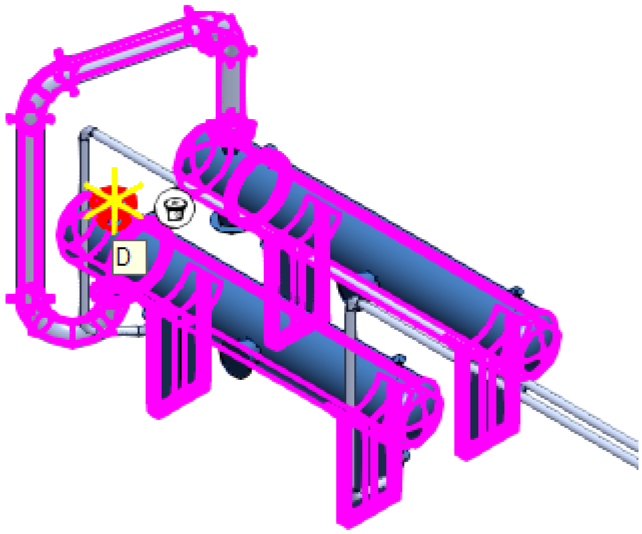
6. While holding the CTRL key, select the equipment **40E-101A** and **40E-101B** along with the pipe run, as shown.



7. Select **Edit > Copy** to copy the objects from the graphic view.

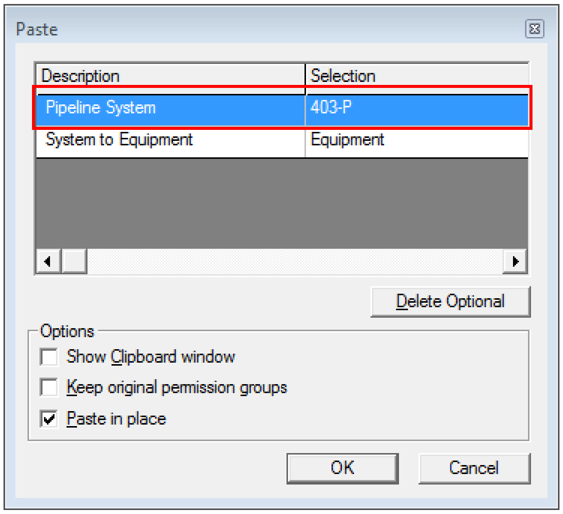
   Smart 3D prompts you to select the reference point within the selected set of objects.

8. Select the pipe nozzle **D** on equipment **40E-101A**, as shown.

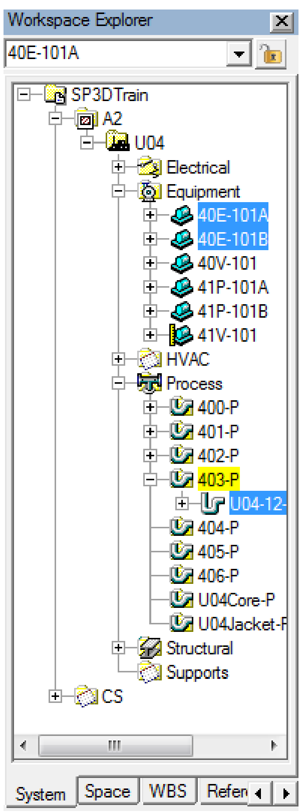


9. Select **Edit > Paste** to paste the objects.

The Paste dialog box appears. In this dialog box you define the parent system Pipeline System for piping objects and System to Equipment for equipment objects separately. The parent system is the system where you will paste the copied objects. You can paste the objects in the same parent system that they belong or can define different parent system.

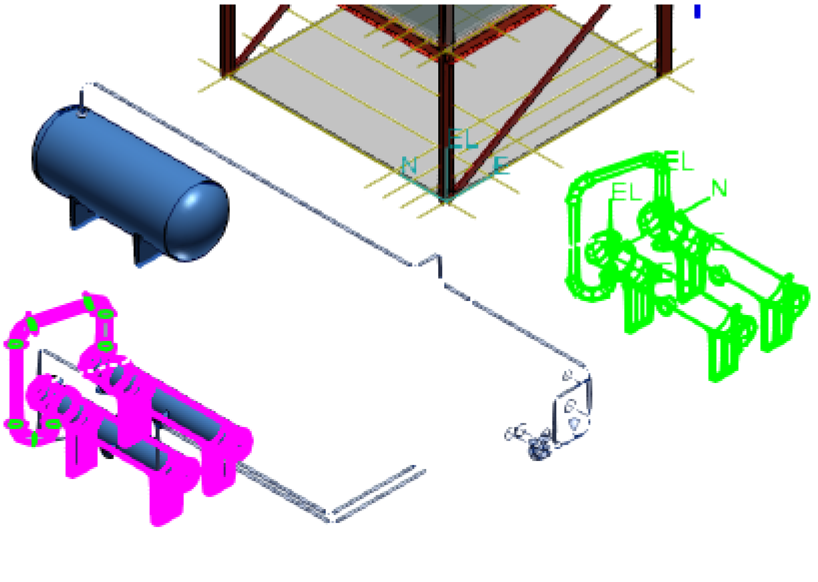


10. You can change the **Pipeline System** by highlighting it in the **Paste** dialog and selecting the different piping system from the Workspace Explorer. The parent system will be highlighted in yellow in the Workspace Explorer corresponding to the selection in the Paste dialog box, as shown. Similarly the parent system for copied equipment can also be changed from the Workspace Explorer.

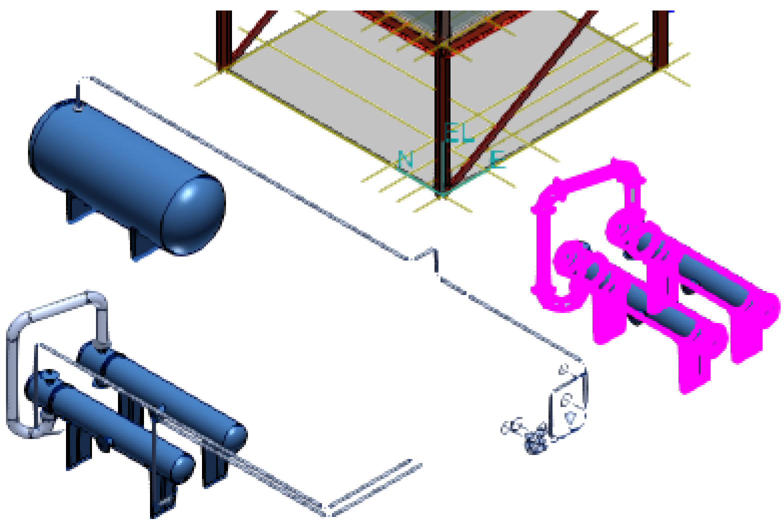


11. Keep the parent system for piping as **403-P** and equipment system for equipment **40E-101A** and **40E-101B** from where they have been copied.
12. Clear the **Paste in place** option in the **Paste** dialog box to paste the copied objects in different location, and click **OK**.

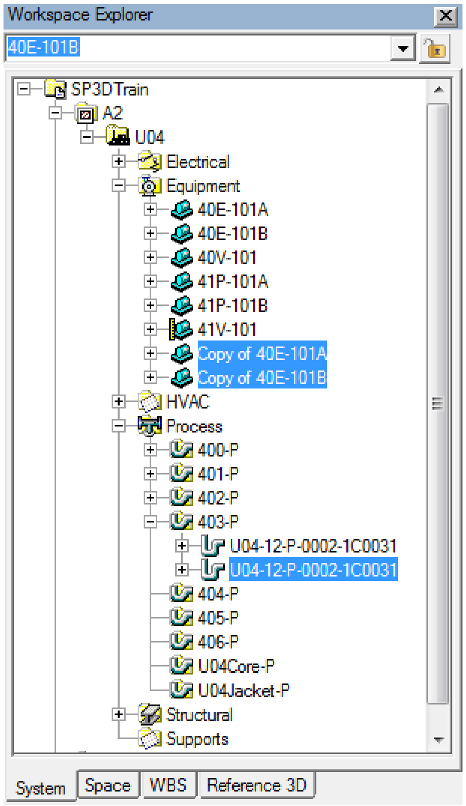
    The copied objects appear with green bordered outline in the graphic view, as shown.



13. Click in the graphic view to paste the objects as shown.



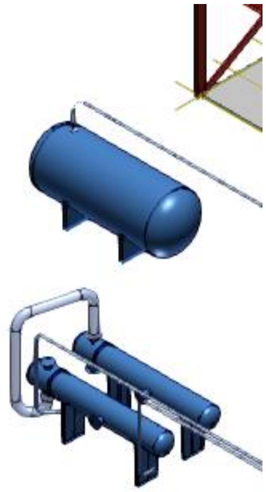
The pasted objects also appear in the Workspace Explorer, as shown in the highlighted sections.



Mirror Copying Pipe Runs/Equipment

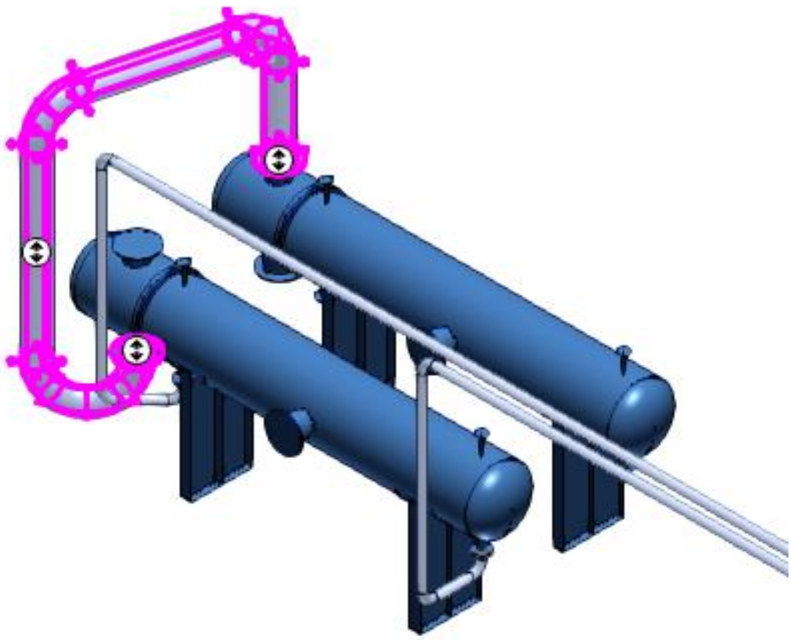
Objective

In this exercise you will be mirror copying the pipe run from the pipeline **403-P** along with the equipment **40E-101A** and **40E-101B** to which the pipe run is connected. The mirrored pipe run and equipment should resemble this.



1. Select **Piping Runs** in the **Locate Filter** drop-down list to select only pipe runs in the graphic view.

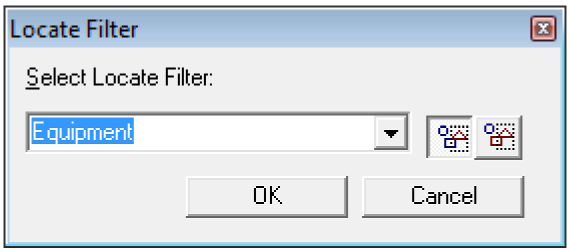
2. Select the pipe run in the pipeline **403-P**, as shown.



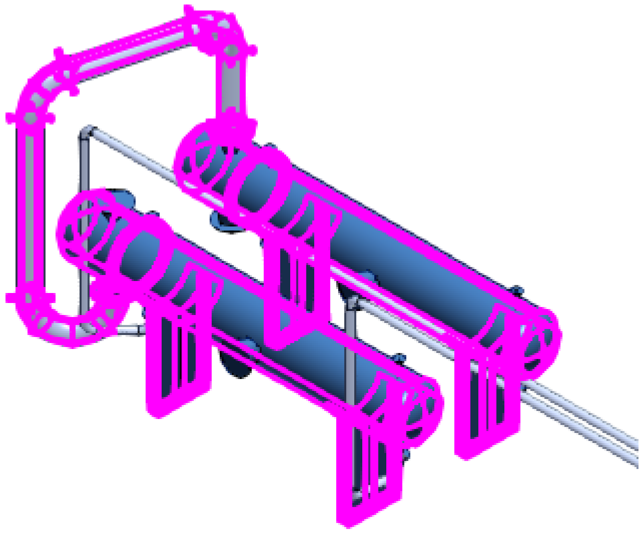
3. Select both equipment **40E-101A** and **40E-101B** to which the pipe run is connected.
4. Select **Edit > Locate Filter** to set the locate filter to equipment along with the pipe run.

   The Locate Filter dialog box appears.

5. Select **Equipment** from the **Select Locate Filter** drop-down list, and click **OK**.



6. While holding the CTRL key, select equipment **40E-101A** and **40E-101B** along with the pipe run.

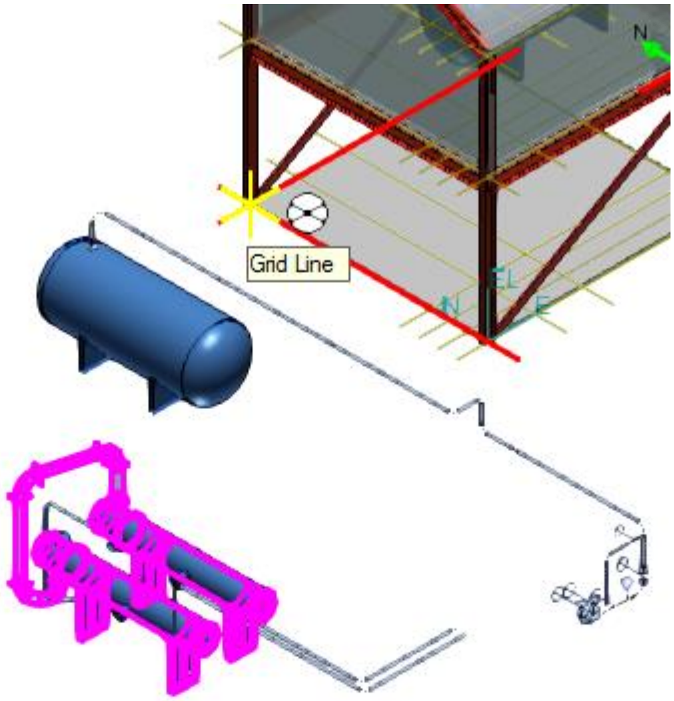


7. Select **Edit > Mirror Copy** to mirror copy the selected objects from the graphic view.

   The Mirror Copy ribbon appears.

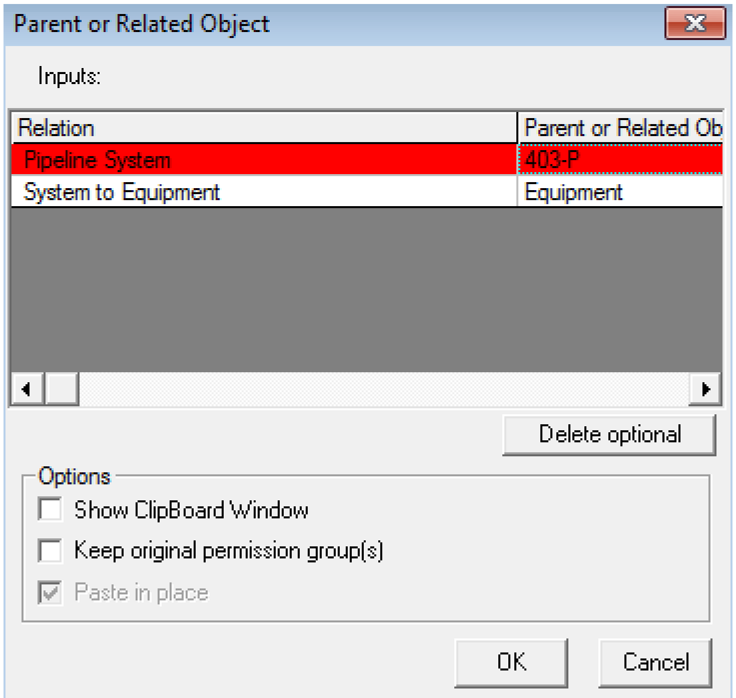
8. In this ribbon define the mirror plane and the **Point to Mirror About** in which the selected objects are mirrored. Select the **East-West** option in the **Direction** drop-down list and **Point to Mirror About** as the **Destination** mode.

9. Select the end of the column as the point to mirror about, as shown.

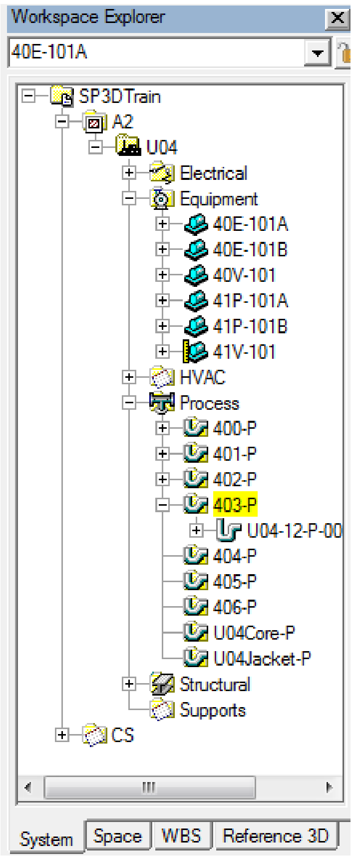


The Parent or Related Object dialog box appears. In this dialog box you define the parent system Pipeline System for piping objects and System to Equipment for equipment objects separately.

The parent system is the system where you will paste the copied objects. You can paste the objects in the same parent system that they belong or can define different parent system.

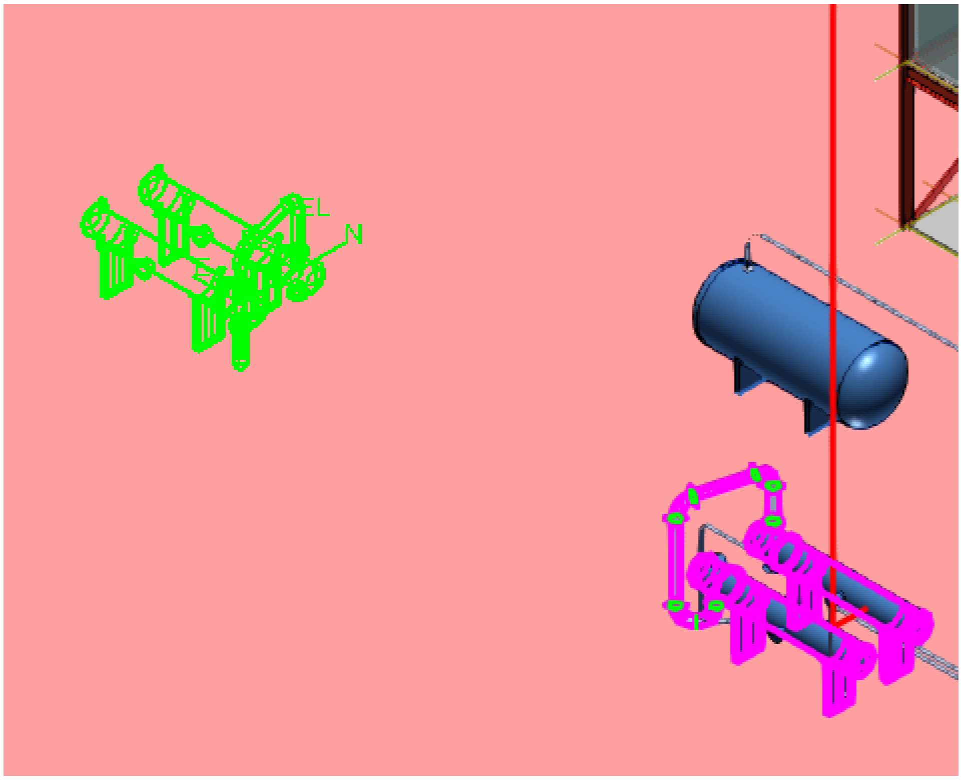


You can change the Pipeline System by highlighting it in the Paste dialog box and selecting the different piping system from the Workspace Explorer. The parent system will be highlighted in yellow in the Workspace Explorer corresponding to the selection in the Paste dialog box, as shown. Similarly the parent system for copied equipment can be changed from the Workspace Explorer.



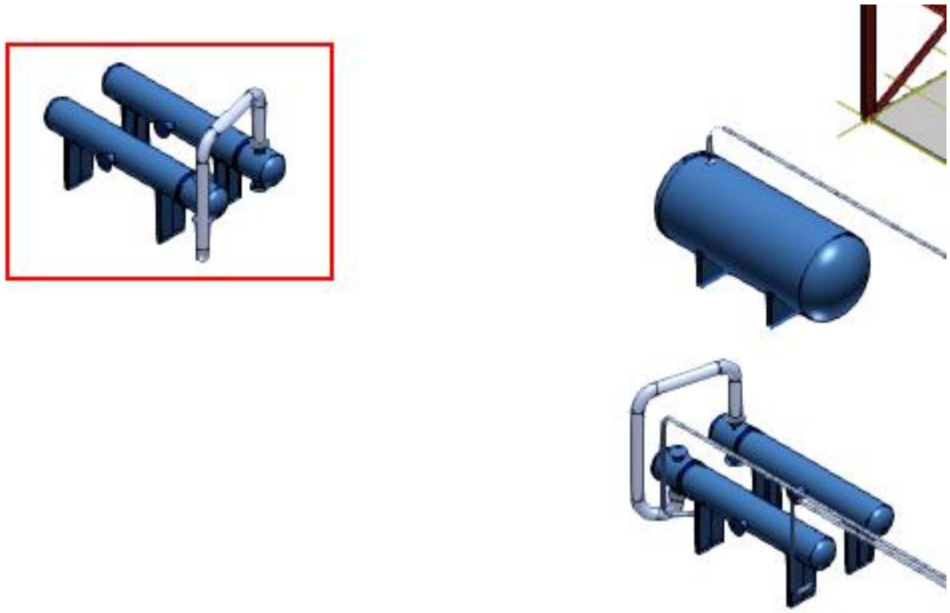
10. Keep the parent system for piping as **403-P** and equipment system for the equipment **40E-101A** and **40E-101B** from where they have been copied, and click **OK**.

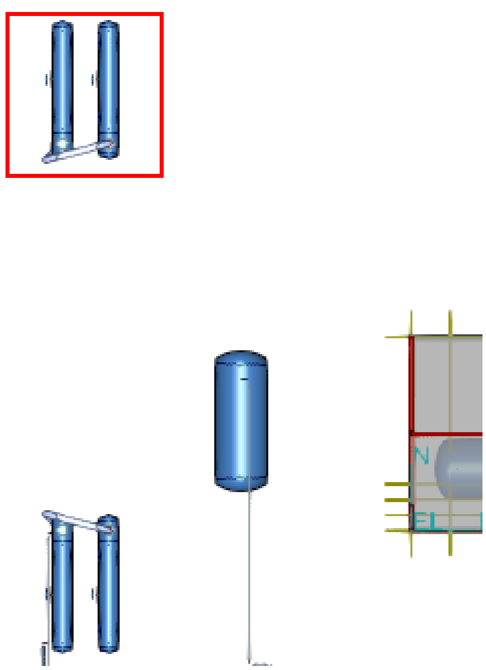
The mirrored objects appear with green bordered outline in the graphic view, as shown.



11. Click **Finish** on the Mirror Copy ribbon.

The mirrored objects will appear in the graphic view, as shown.

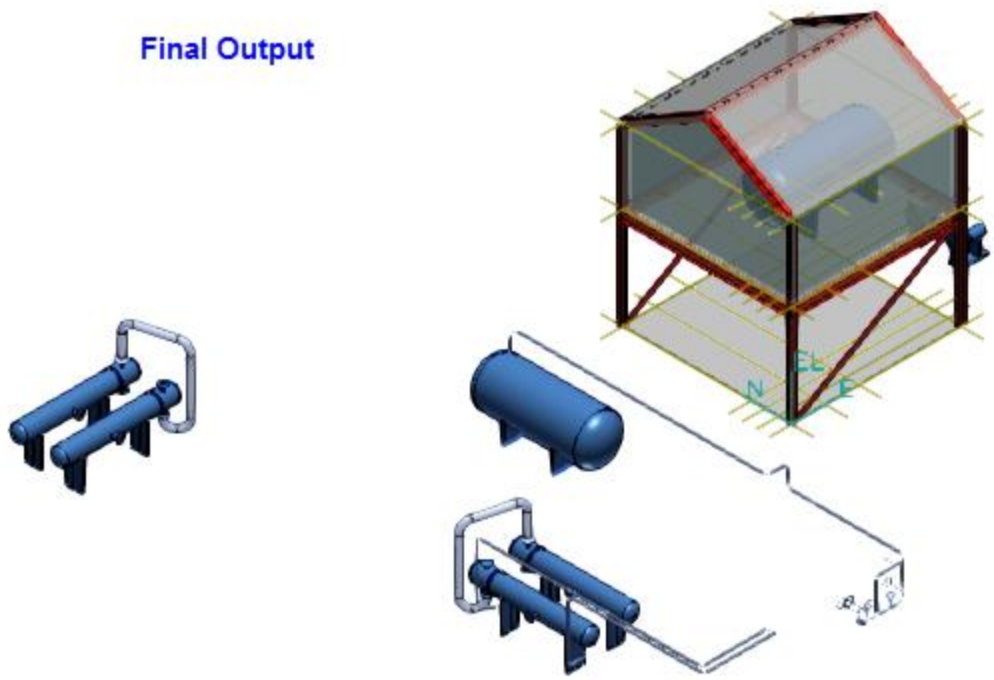




Rotating Pipe Runs and Equipment

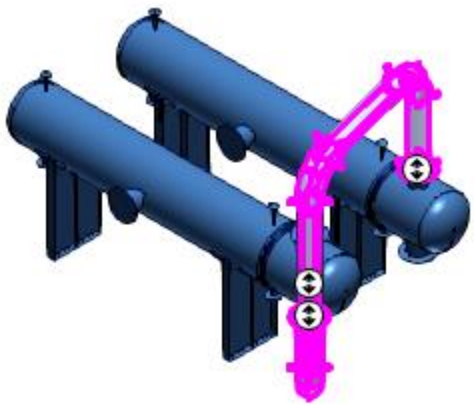
Objective

In this exercise you will be rotating mirrored objects about the midpoint of the pipe located in the mirrored pipe run. The mirrored pipe run and equipment should resemble this.



1. Select **Piping Runs** in the **Locate Filter** drop-down list to select only pipe runs in the graphic view.

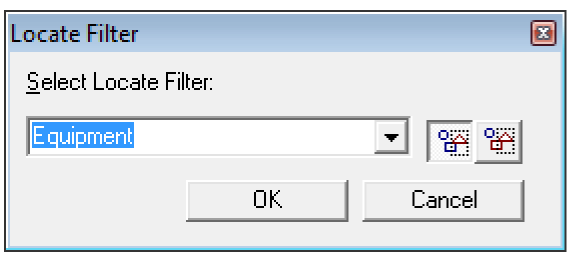
2. Select the mirrored pipe run in the pipeline 403-P, as shown.



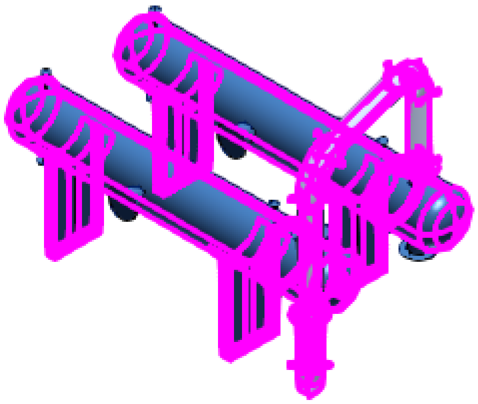
3. Select both the equipment to which the pipe run is connected.
4. Select **Edit > Locate Filter** to set the locate filter to equipment along with pipe run.

   The Locate Filter dialog box appears.

5. Select Equipment in the **Select Locate Filter** drop-down list, and click **OK**.



6. While holding the CTRL key, select the equipment **Copy of 40E-101A** and **Copy of 40E-101B** along with the mirrored pipe run, as shown.

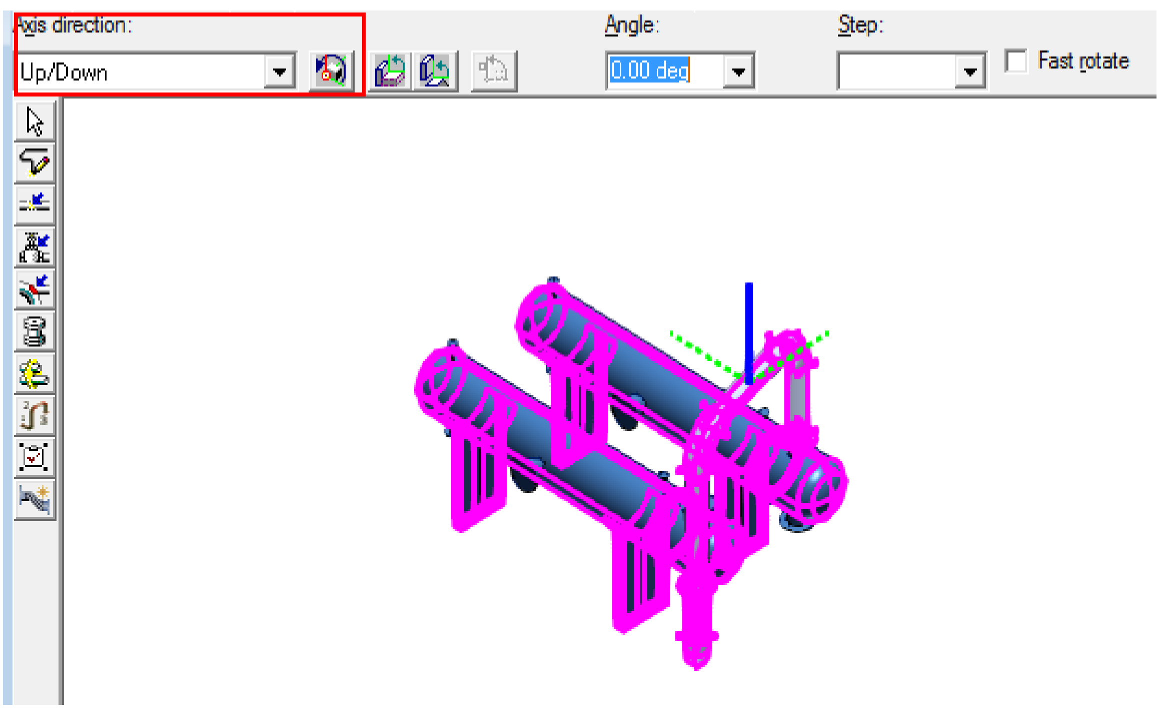


7. Select **Edit > Rotate Object** to rotate the selected objects in the graphic view.

   The Rotate Object ribbon appears. In this ribbon define the axis direction and the origin point for the rotation.

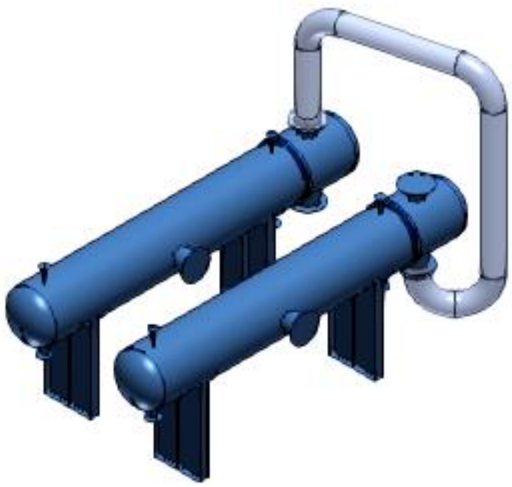
8. Select **Up/Down** in the **Axis Direction** drop-down list.

9. Select **Axis Position Point** and select the midpoint of the pipe run as the origin point for the axis of rotation.



10. Key in **90 deg** in the **Angle** drop-down list.

The selected objects will rotate in the graphic view, as shown.



11. Undo the rotation for a later exercise.

Deleting Pipelines

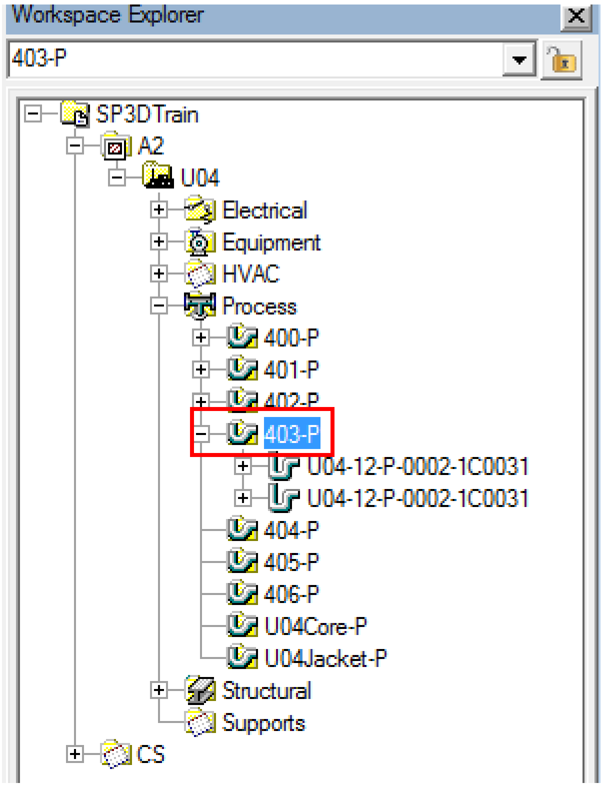
Objective

In this exercise you will be deleting a Pipeline from your model. You have the ability to select a pipeline system in the Workspace Explorer or in the graphic view and then select the Delete command to delete a pipeline.

Delete the pipeline 403-P in Unit U04 of your workspace.

1. Define your workspace to include unit **U04**.
2. Click **Select** on the vertical toolbar.
3. Select the **Pipelines** option in the **Locate Filter** drop-down list.

4. Select the pipeline **403-P** from the **Workspace Explorer**.



5. Click **Delete**.

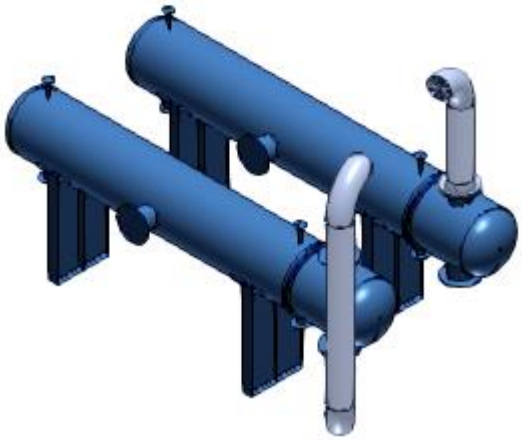
6. Click **Undo**. Pipeline 403-P is needed in later labs.

CAUTION While deleting a pipeline and pipe run, remember that deleting a pipeline deletes all the pipe runs, features, and parts associated with that pipeline. Likewise, deleting pipe run deletes features, and parts associated with that pipe run.

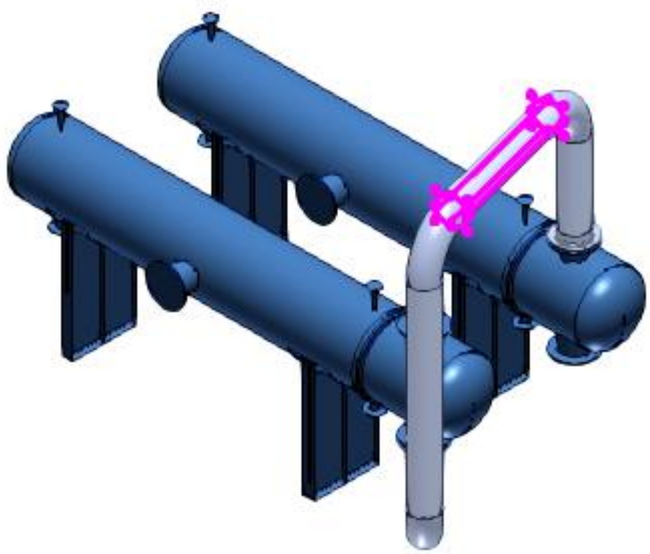
Deleting Pipe Straight Features

Objective

In this exercise you will be deleting a pipe straight feature of the mirrored pipeline **403-P** in **Unit U04** of your workspace. The view of the model after deleting the pipe straight feature will resemble this.



1. Select **Piping Features** in the **Locate Filter** drop-down list.
2. Select the **Pipe Straight Feature** of pipeline **403-P**.



3. Click **Delete** to delete the Pipe Straight Feature.

Deleting a pipe straight feature does not remove connected turn or branch features.

Changing Mating Part to a Base Part

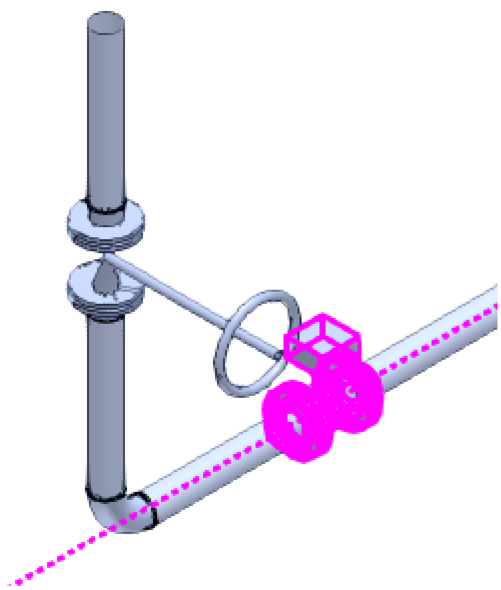
Objective

In this exercise you will be changing a Mating part to a Base part.SP3D automatically deletes all mating parts that were placed with the base part that you deleted. You can change a mating part to a base part so that SP3D will not automatically delete it.

Before Starting this Procedure

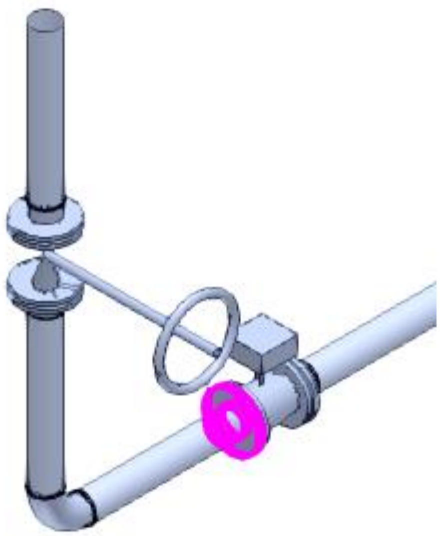
- Define your workspace to display **Area01**. In your training plant, select **Area01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

1. In the **Locate Filter** list, select the **Piping Features** option.
2. In the graphic view, select **FE-523** from pipeline **P-204**, as shown.



3. Click **Delete**. Notice how both the flanges and the instrument are deleted.
4. Click **Undo**.
5. In the **Locate Filter** list, select the **Piping Parts** option.

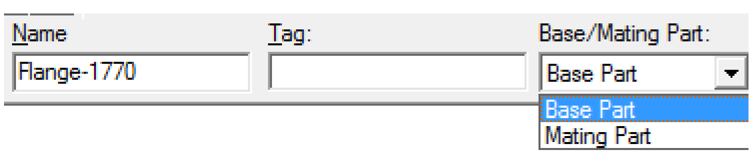
6. In the graphic view, select the mating flange of **FE-523** from pipeline **P-204**.



The Edit Pipe Part ribbon bar appears.

The **Base/Mating Part** drop-down list displays the selected object, which has the mating part.

7. Select the **Base Part** option in the drop-down list.



CAUTION Use caution however, because after a mating part has been changed to a base part, it cannot be changed back to a mating part again.

8. Repeat the above steps for the other mating flange.
9. In the **Locate Filter** list, select the **Piping Features** option.
10. In the graphic view, select **FE-523** from pipeline **P-204**.
11. Select **Delete**. Notice how the instrument is deleted, leaving the flanges. New mating flanges and a piece of pipe are automatically inserted to bridge the gap.

S E S S I O N 1 2

Creating Spools

Objective

By the end of this session, you will be able to:

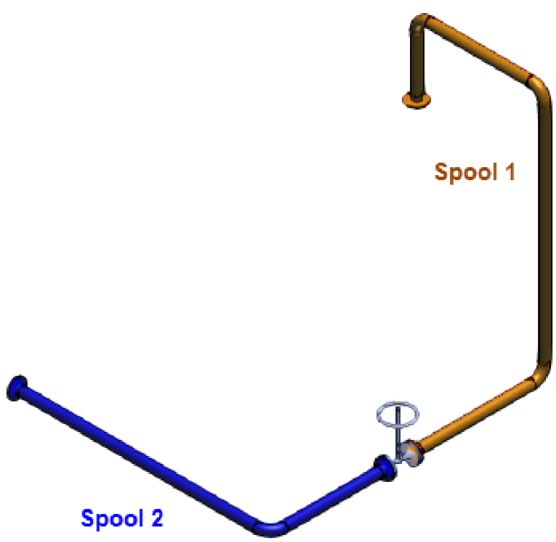
- Create spool assemblies in a pipe run using the **Generate Spools** command.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)

Overview

The **Generate Spools**  command breaks pipelines into spools. Spools are collections of piping parts and welds that hold them together. Smart 3D creates a spool assembly by applying a set of rules that breaks the pipeline system into pieces. Each piece ends with a spool-breaking component or connection type.



There are two basic approaches for creating spool assemblies in Smart 3D.

- **By WBS items method:** Smart 3D collects all the pipe parts grouped by a WBS item for generating spool assemblies.

- **By Pipeline objects method:** Smart 3D collects all the pipe parts that make up a pipeline to generate spool assemblies.

After a WBS item or a pipeline is spooled, it becomes the parent object for the spool. After a spool is generated, it can be used to create an isometric drawing for fabrication. Spools include any piping component or a piece of pipe that is assigned a fabrication type by a fabricator. Implied objects such as bolts, nuts, and gaskets that are associated with these parts are not included in spools.

NOTES

- The **Spooling Basis** option is used to set up the spool generation based on WBS items or pipelines.

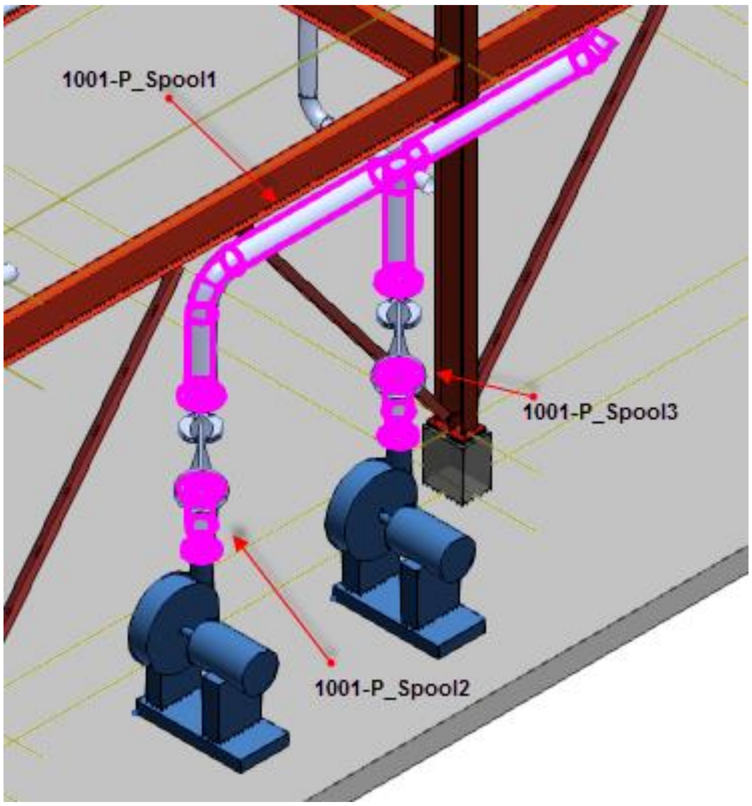
To change the basis of spooling, use Microsoft Excel to open the [Product Directory]\SmartPlant\3D\CatalogData\Bulkload\Datafiles\BulkLoadIsoKeys.xls workbook. In the SpoolingBasis column in the PipeMfgSpoolRule sheet, enter 0 to spool by pipelines or enter 2 to spool by WBS items. In column A of that row, be sure to type M to indicate that you have modified the line. Save and exit the workbook. Bulk load the workbook to the Catalog by using the Add, Modify, or Delete records in the Existing Catalog option.

This session covers the procedure for generating spool assemblies by WBS items and by Pipeline objects. Both the procedures are supported in Smart 3D. The Smart 3D administrator can setup to use one or the other.

Creating Standard Spools from Pipeline Objects

Objective

In this exercise you will be creating standard spools on the pipeline **1001-P** in **Unit U01** of your workspace by using the Generate Spools command. After creating the spools, the workspace should resemble this.



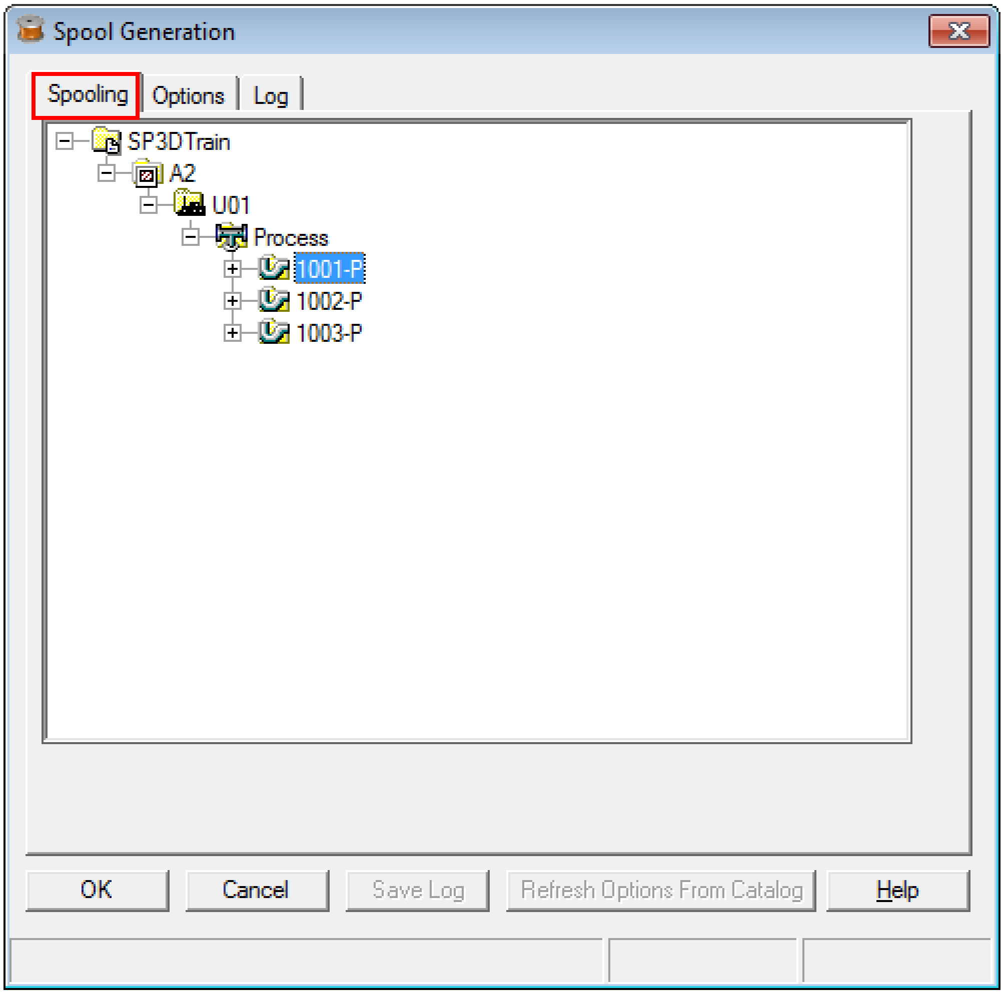
Before Starting this Procedure

- Define your workspace to display **Unit U01** and coordinate system **U01 CS**. In your training plant, select **U01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

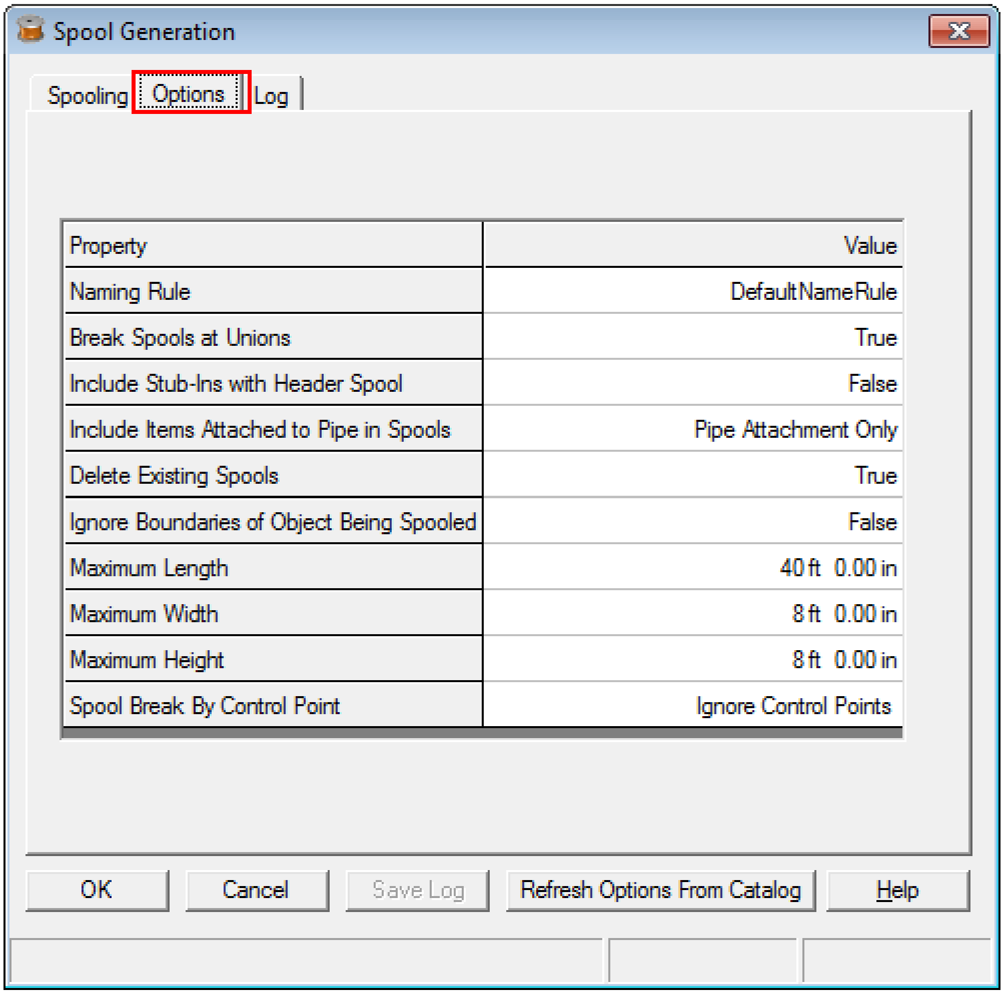
1. Click **Generate Spools** .

   The Spool Generation dialog box appears.

2. On the **Spooling** tab, expand **A2 > U01 > Process** and select **1001-P**, to specify the pipeline where you want to generate spools. Each spool must consist of connected objects; otherwise, errors might occur in spooling.

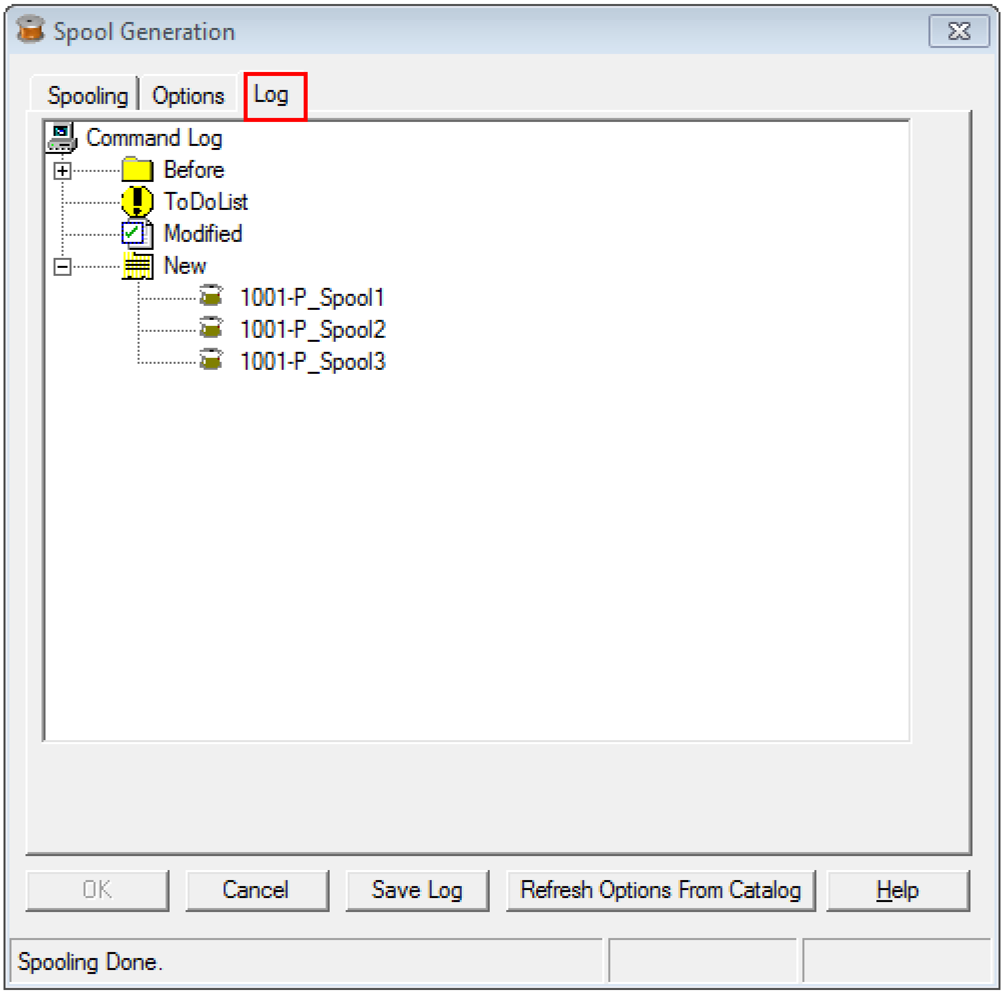


3. Select the **Options** tab in the **Spool Generation** dialog box to review the spooling options.



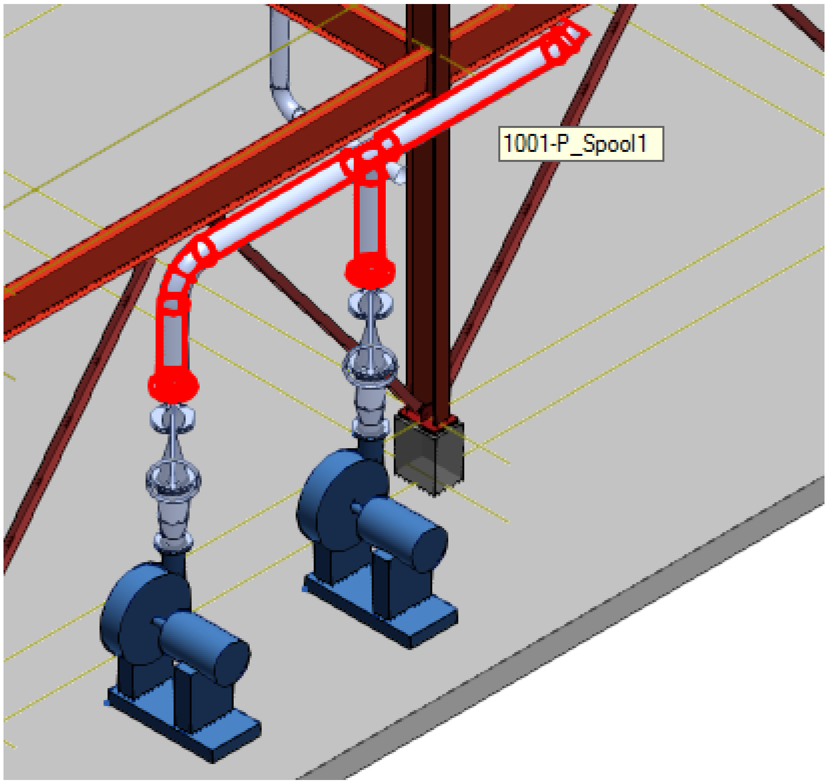
4. Click **OK** to apply the default settings to the spool that is generated.

5. Select **Log** tab in the **Spool Generation** dialog box and expand the New node to review the spooling log then click **Cancel** close the dialog box.



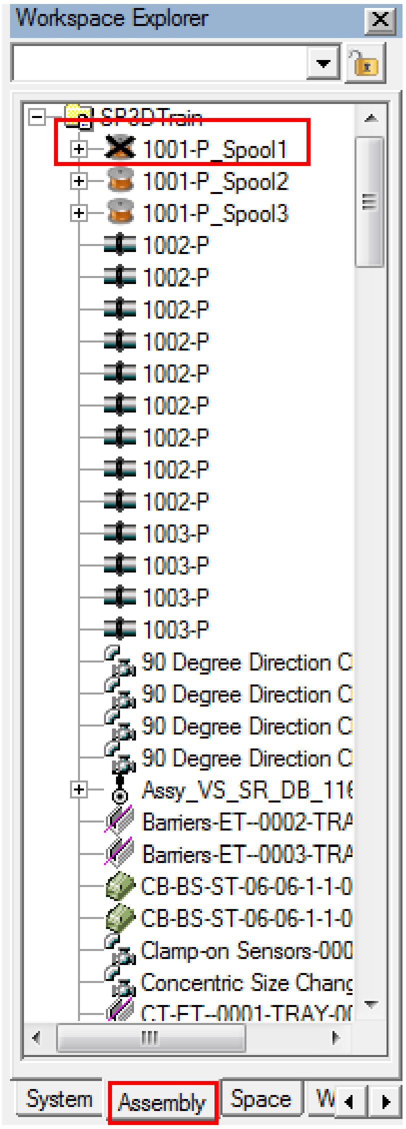
6. To view the spool you created, select the **Spools** option in the **Locate Filter** drop-down list and point to the pipeline 1001-P.

A tool tip will appear for the spool in the graphic view.



While generating spools, Smart 3D checks the maximum length, height, and width of a spool, which you specify in the **Spool Generation** dialog box. If the spool exceeds the maximum size, Smart 3D marks that spool as oversized so that the user can insert additional breaks as required. This check helps ensure that the spools that need to be dipped in a tank for galvanization, de-scaling, and pickling will actually fit in the immersion tank.

Spools can also be marked oversized if they are too large for either shipping or fitting in the fabrication equipment such as bending machines or annealing ovens. You can check for oversized spools on the **Assembly** tab in the **Workspace Explorer**. The icon for an oversized spool contains a black X symbol, as shown.

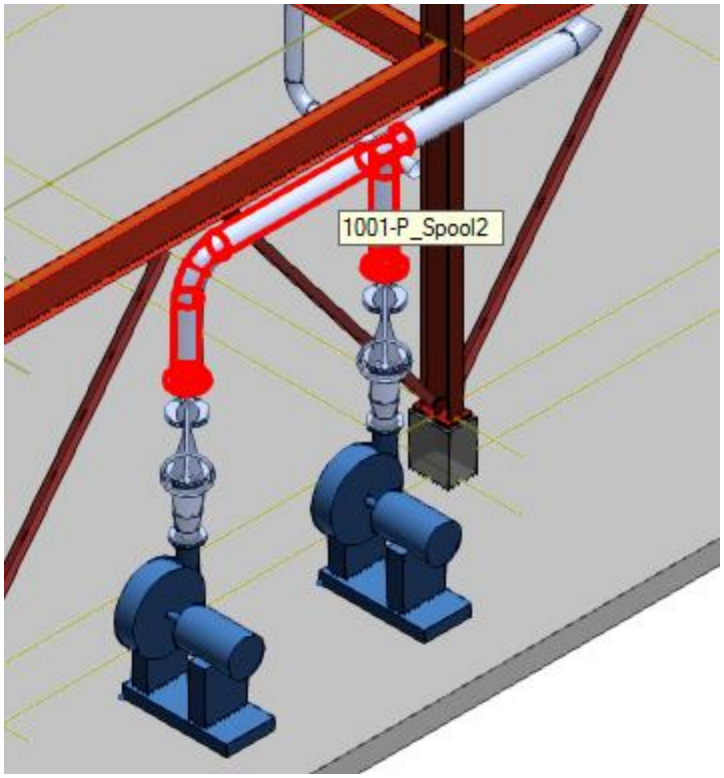


To reduce the oversized spools to the maximum size defined in the **Spool Generation** dialog box, you can insert manual break points or control points on the oversized spools by using the **Insert > Control Point** command. A control point breaks oversized spools along the pipeline. The control point must be placed on the connection object of a pipeline.

Creating Spools using Manual Break Points

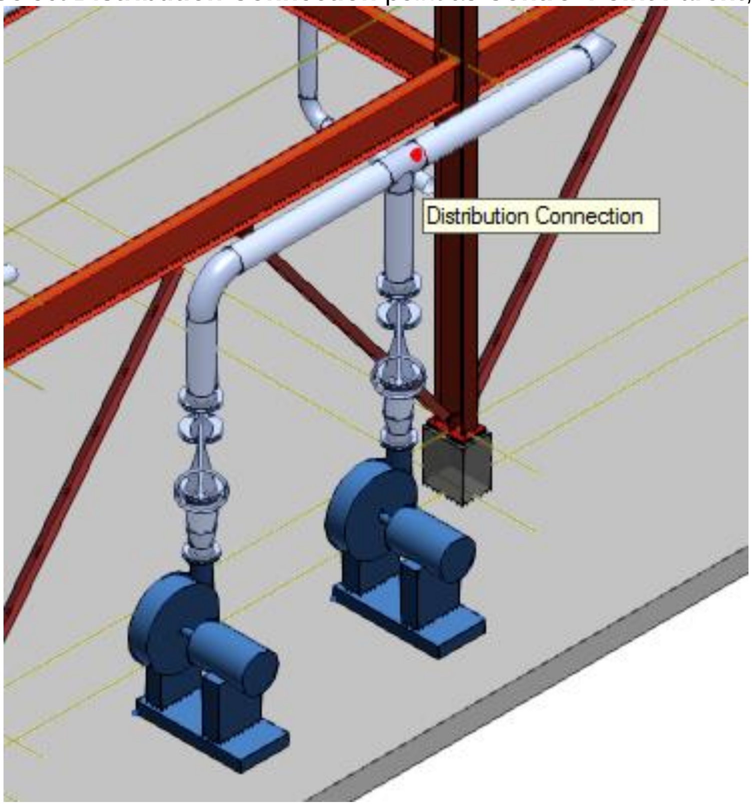
Objective

In this exercise you will be inserting control points in the oversized spool **1001-P_Spool1** shown below to manage the size of the spool assembly. The view of the pipe at the end of the procedure should resemble this.

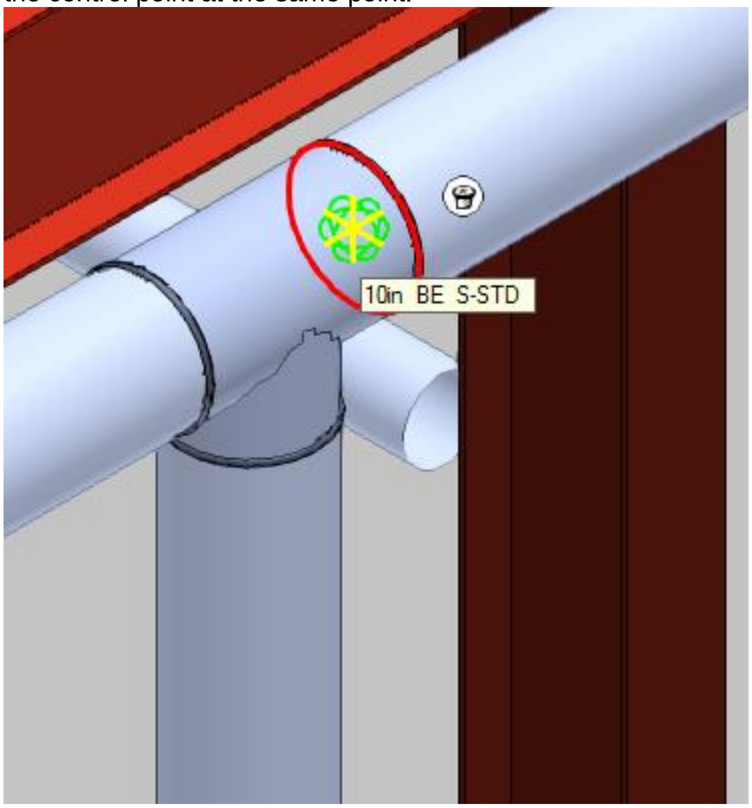


1. Make sure that the **Locate Filter** is set to **Connections**.

2. Select **Distribution Connection** point as **Control Point Parent**, on **1001-P_Spool1**.

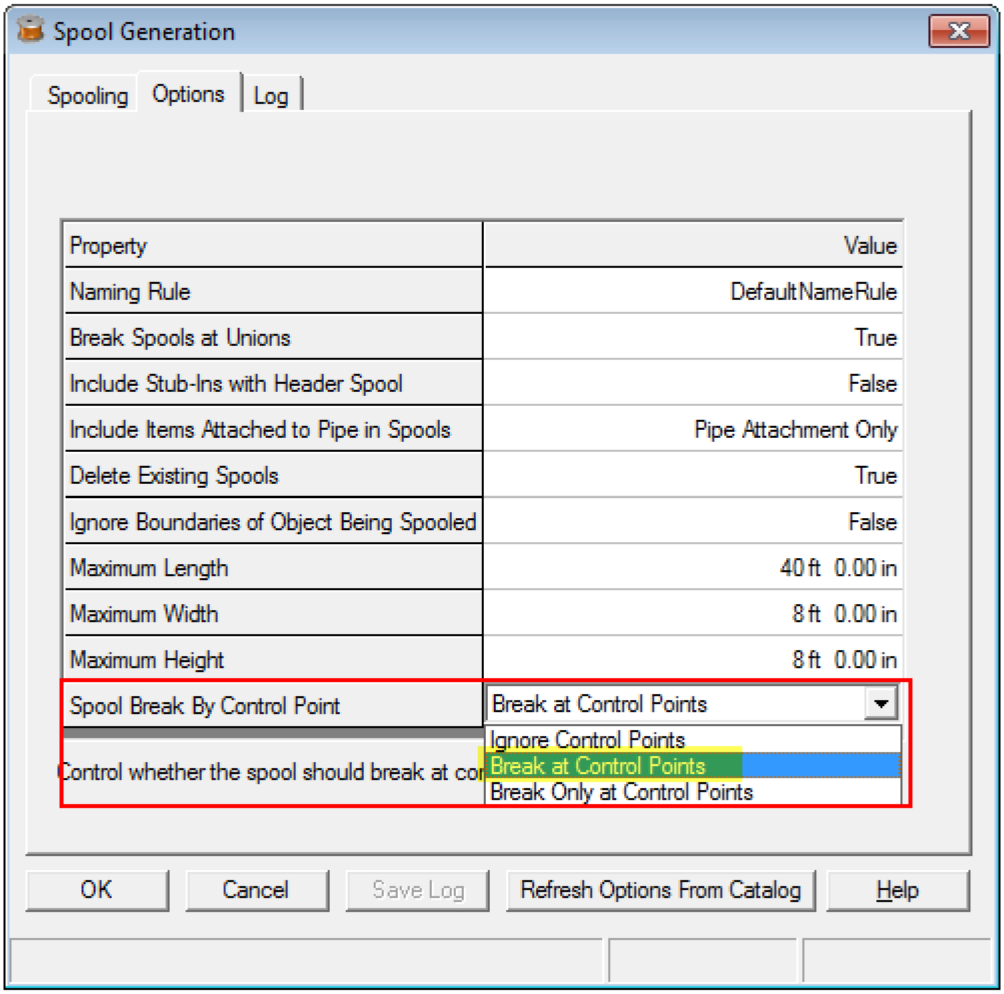


3. Select **Insert > Control Point**.
4. Select **Spool Break** in the **Subtype** drop-down list on the **Control Point** ribbon and place the control point at the same point.



5. Click **Generate Spools** .
6. On the **Spooling** tab in the **Spool Generation** dialog box, expand **A2 > U01 > Process** and select **1001-P**.

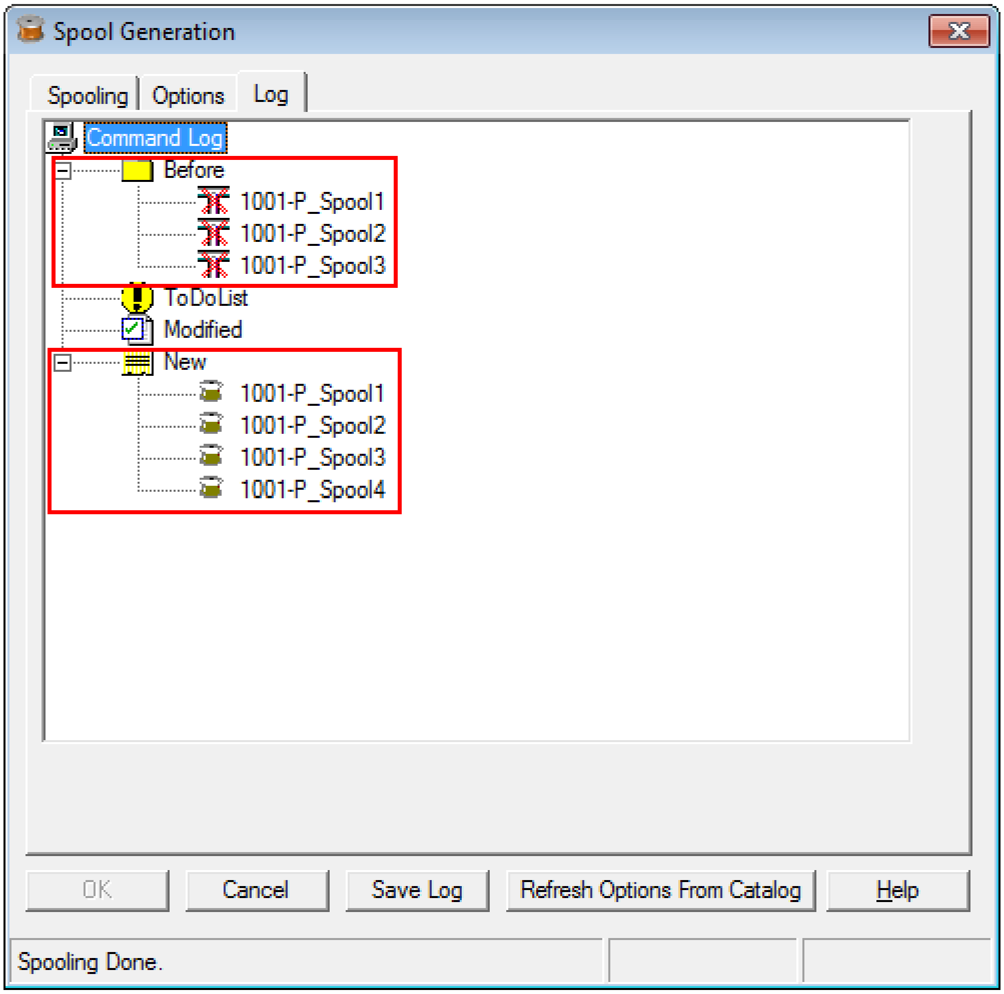
7. On the **Options** tab in the **Spool Generation** dialog box, select the **Break at Control Points** option in the **Spool Break By Control Point** drop-down list.



8. Click **OK** to process the pipelines into spools.

   This will regenerate spools, and a new spool break will be inserted at the control point.

9. Now select the **Log** tab in the **Spool Generation** dialog box. This will show you the number of spools before and after inserting the control point.



10. Click **Cancel** in the Spool Generation dialog box.

TIP Generating spools at break points or control points will work only when the control points are linked to connection objects. If welds or features are used, the spooling process will ignore the control points.

For more information related to spooling, refer to **Spool Generation** in the user guide PipingUsersGuide.pdf.

SESSION 13

Sequencing Objects

Objective

By the end of this session, you will be able to:

- Sequence piping objects in a model.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Creating Spools* (on page 379)

Overview

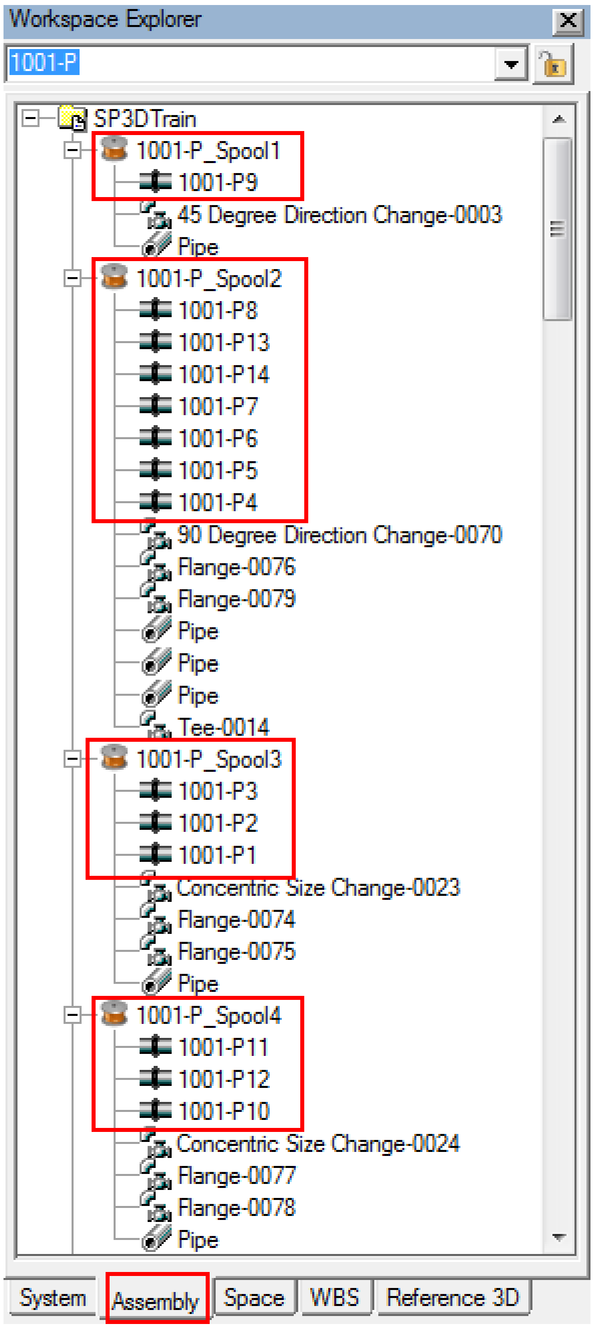
In piping, renaming objects in logical order using unique names is called sequencing. Sequencing makes it easy for you to locate piping objects from the Workspace Explorer, without checking them in the graphic view of the software. You can sequence piping objects by using the **Sequence Objects** command, which renames the target objects in the selected grouping object. This command renames target objects such as pipe welds or spools located in the grouping object such as a pipeline, a pipe run, a spool, or WBS items. You can select to rename the target objects based on their flow direction or topology.

This session will cover the procedure for sequencing piping objects in a WBS item and pipeline.

Sequencing Piping Objects in a Pipeline

Objective

In this exercise you will be sequencing the pipe welds of a pipeline 1001-P in Unit U01 of your workspace. The sequenced pipe welds in the Assembly tab of the Workspace Explorer should resemble the highlighted sections.



Before Starting this Procedure

- Define your workspace to display **Unit U01** and the coordinate system **U01 CS**. In your training plant, select **U01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

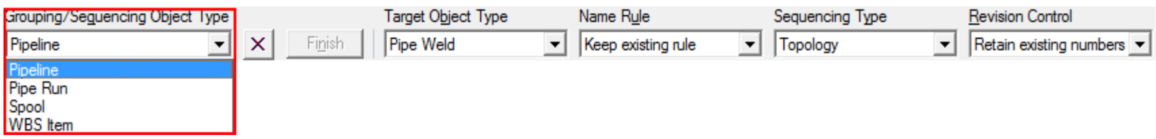
1. Click **Sequence Objects** on the vertical toolbar.

   The Sequence Objects ribbon appears.

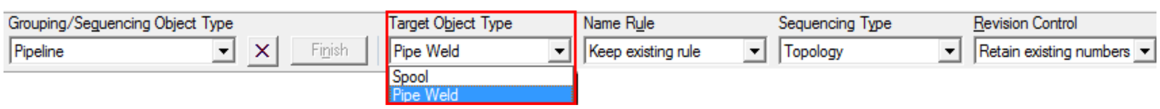
   The first option on the ribbon is **Grouping/Sequencing Object Type**, which specifies the grouping object type by what you can sequence the object. You can select a pipeline, a pipe run, a spool, or a WBS item as a grouping object. This option defines the collection of target objects to be sequenced and defines the boundaries of the sequencing.

2. Select **Pipeline** in the **Grouping/Sequencing Object Type** drop-down list.

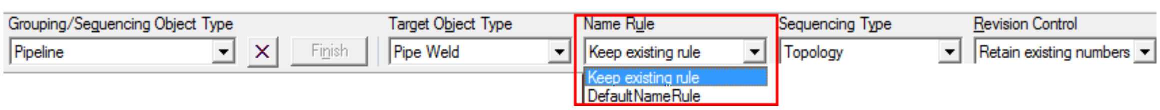
   The second option is Target Object Type. You use this option to specify the target object that you want to rename on the selected grouping object.

3. Select **Pipe Weld** option in the **Target Object Type** drop-down list to specify the target object to be renamed on the pipeline.

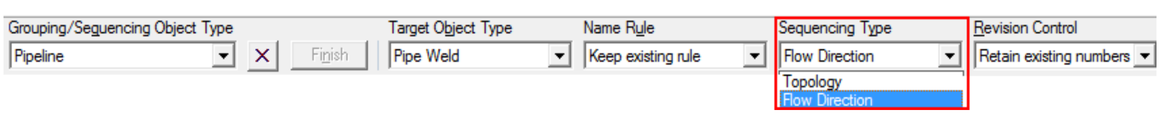
   TIP You can sequence the spools of a pipeline by selecting the Spool option in the Target Object Type drop-down list.

4. Select **DefaultNameRule** in the **Name Rule** drop-down list to specify the naming rule for renaming the target object.

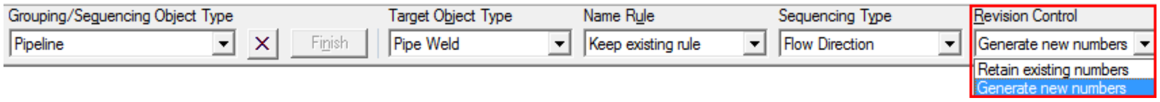
   After selecting the target object, you define the sequencing type to specify the logical order of the target object.

5. Select **Flow Direction** option in the **Sequencing Type** drop-down list to define the sequence type for the welds.

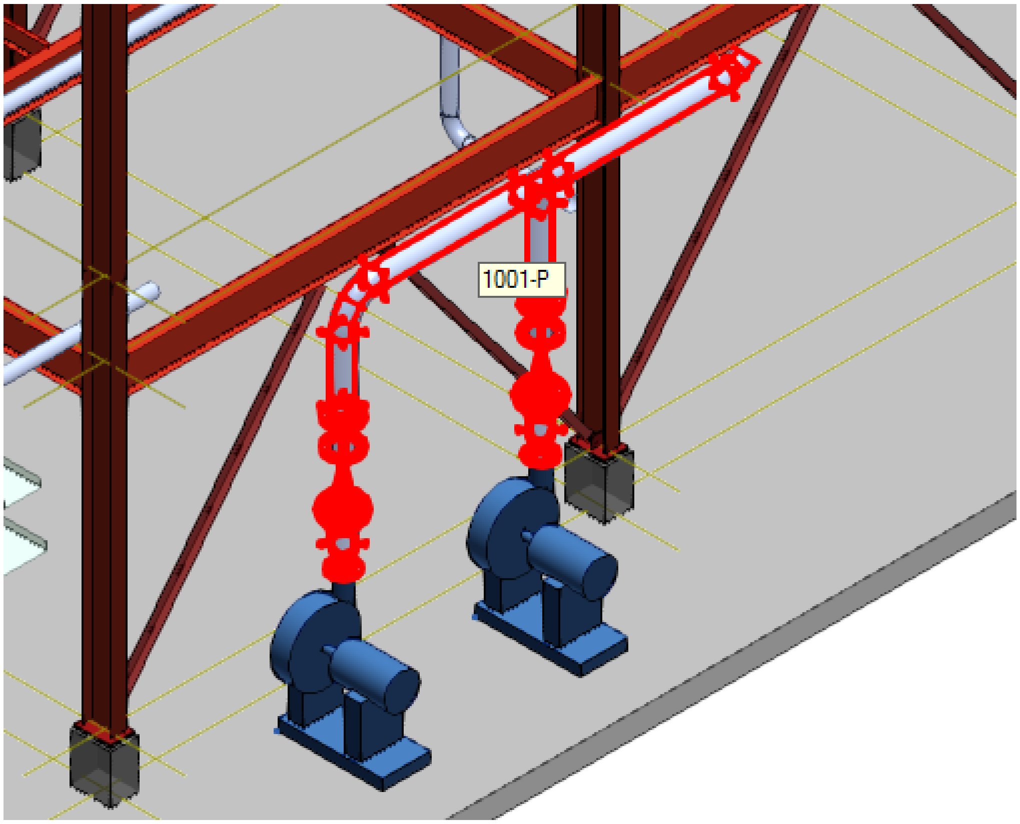
   

   TIP In Smart 3D, spools are sequenced mainly by using the flow direction sequencing type.

6. Select **Generate new numbers** in the **Revision Control** drop-down list to generate new numbers for the welds.



7. Now select the pipeline **1001-P** in the graphic view.



8. Click **Finish** to execute the command and apply the options set for renaming the pipe welds.

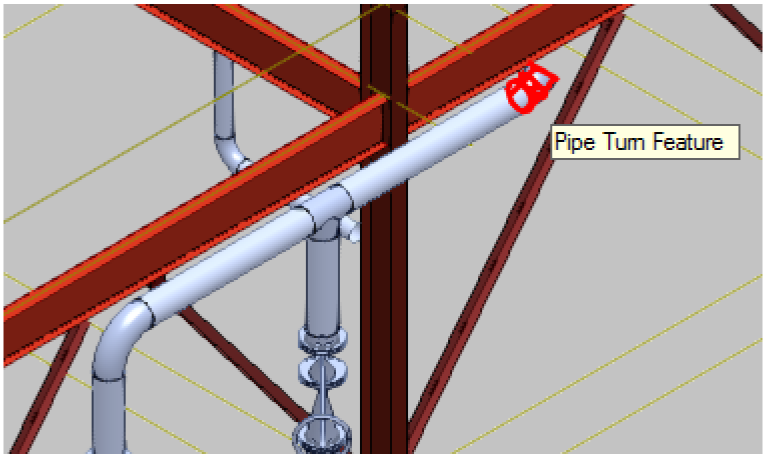
For more information related to sequencing objects, refer to Sequence Objects Command, topic in the user guide PipingUsersGuide.pdf.

Sequencing Piping Objects Avoiding Reuse of Sequence ID in a Pipeline

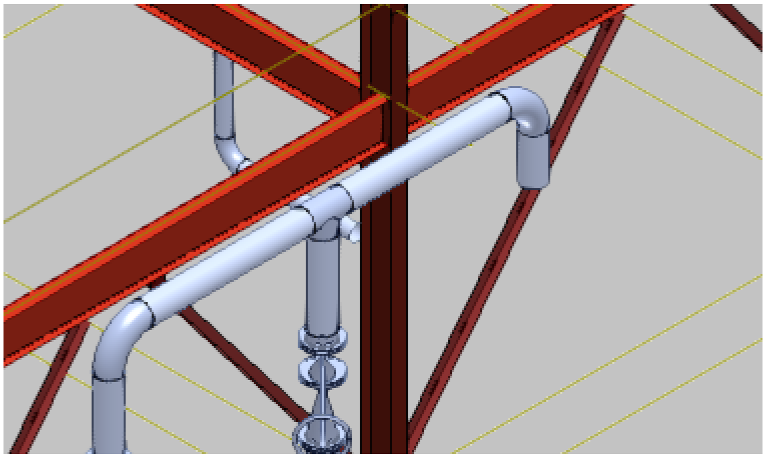
Objective

In this Lab you will sequence the weld ID's after changing the routing of 1001-P in U01 of your Workspace and Observe that there is no reuse of Weld and spool sequence ID that are deleted.

1. Delete the **Pipe** turn feature of **1001-P**.

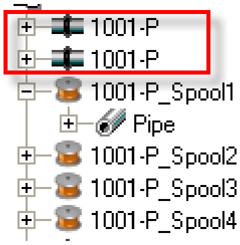


2. Now select the **End Feature** of the pipe and route a **3 ft** Pipe in the downward direction.



3. Right-click in the graphic view to terminate **Route Pipe** to resemble as below

4. Observe that two new welds are created in the **Assembly** tab and the **1001-P9** weld under the **1001-P_Spool1** is deleted.



5. Repeat the steps and **Select the Retain** existing numbers option in the **Revision Control** drop-down list.

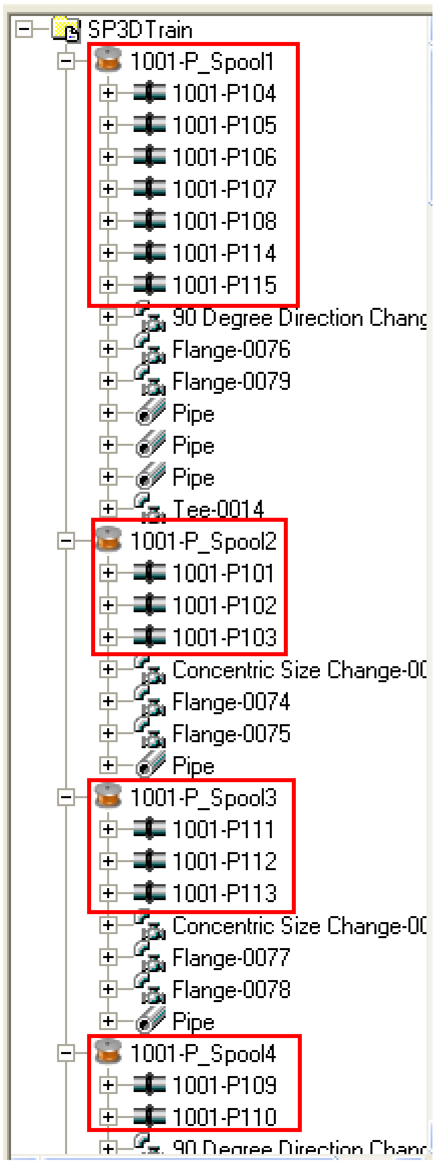
6. Now select the pipeline **1001-P** in the graphic view and click **Finish**.

7. Observe that the weld Sequence ID **1001-P9** is not reused by the new welds.

8. Now the Weld ID's can be sequenced again to completely **Regenerate New Numbers** by selecting **Generate New Numbers** option and key in **101** in the **Start Sequence ID** ribbon.

TIP The **Generate New Numbers** option regenerates all the Weld Sequence ID's as per the modified routing taking into consideration the **Start Sequence ID**.

The 1001-P pipeline with newly Generated Sequence ID's will resemble this.



SESSION 14

Creating Isometric Drawings

Objective

By the end of this session, you will be able to:

- Create isometric drawings in a model and update them.

Prerequisite Sessions

- Smart 3D Overview
- Smart 3D Common Sessions
- *Piping: An Overview* (on page 9)
- *Routing Pipes* (on page 15)
- *Inserting Components in a Pipe Run* (on page 69)
- *Creating Spools* (on page 379)
- Smart 3D Drawings Sessions

Overview

In piping, isometric drawings communicate several important types of information to a pipe fabrication workshop. This information includes pipe cut lengths, bend angles, and welds. Isometric drawings are created by specifying a filter-based query to collect the objects to associate a **Piping Isometric Drawing by Query** component to a **Query Manager**. Each component has an associated isometric style such as **Iso_Pipeline**, **Iso_Piperun**, **Iso_PenSpool**, **Iso_Spool**, **Iso_WBS**, and **Iso_Stress**. So, you or the Smart 3D administrator need to set the isometric options for each isometric style needed in your project.

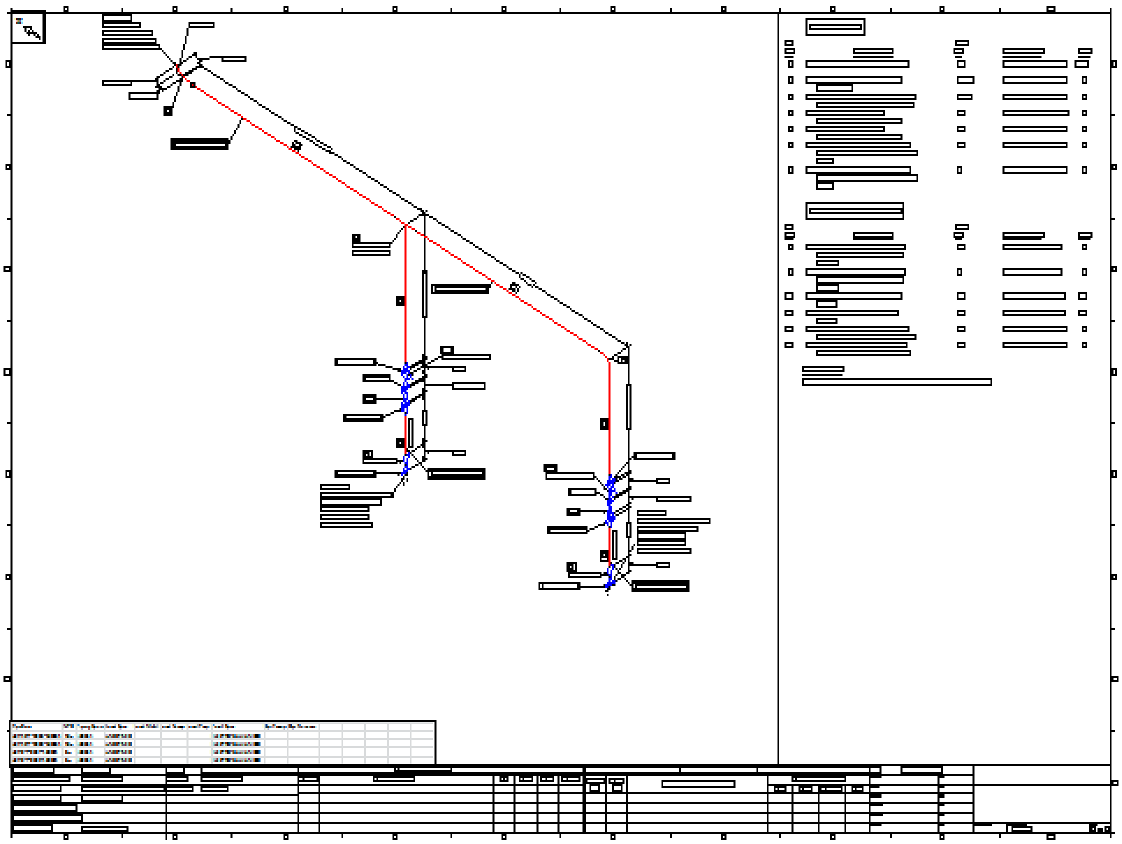
There are two basic approaches by which piping parts can be automatically grouped for creating isometric drawings: From Pipeline Objects or From WBS Items. When using the From Pipeline Objects approach, you collect all pipe parts that make up the pipeline. When using the From WBS Items approach, you collect all the pipe parts according to a query defined in a label and assign them to these WBS group items.

This session covers the procedure for creating isometric drawings from WBS items and pipeline objects. Both of these will be supported in SP3D. That is, the SP3D administrator can setup to use one or the other or can allow both.

Creating Isometric Drawings from Pipeline Objects

Objective

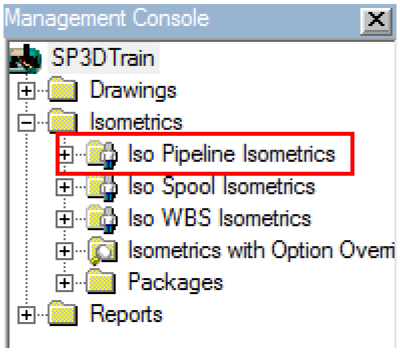
In this exercise you will be creating an isometric drawing for the pipeline 1001-P in Unit U01 of your workspace. After creating the isometric drawing for the pipeline 1001-P the view will resemble .



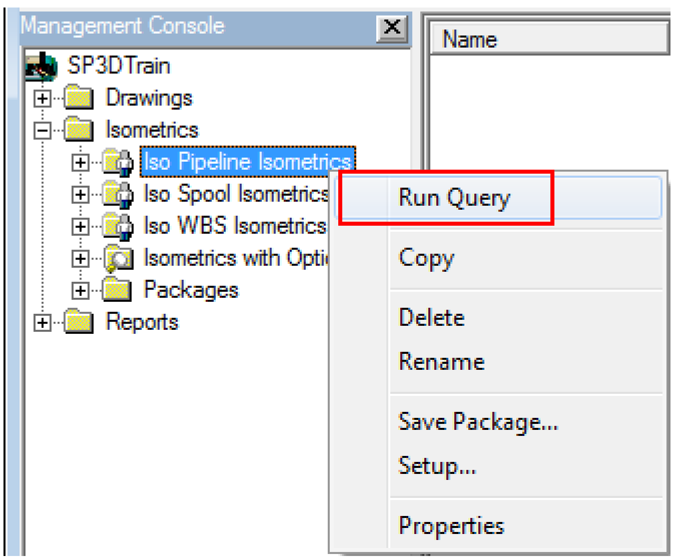
Before Starting this Procedure

- Define your workspace to display Unit U01 and coordinate system U01 CS.
- Make sure you are in the Drawings and Reports task to create an isometric drawing for pipeline 1001-P.

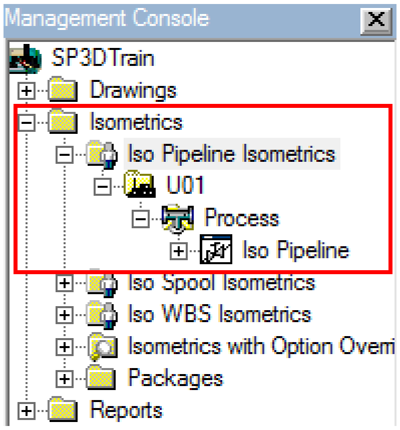
In this task, the Management Console appears in the left panel of the SP3D window, as shown in . The Management Console has components to create the isometric drawings. The administrator should have already created Drawings by Query Manager components needed in the project.   In this tutorial session, a Drawings by Query Manager component called Iso Pipeline Isometrics is created which specifies the portion of the model from where you select the piping objects.



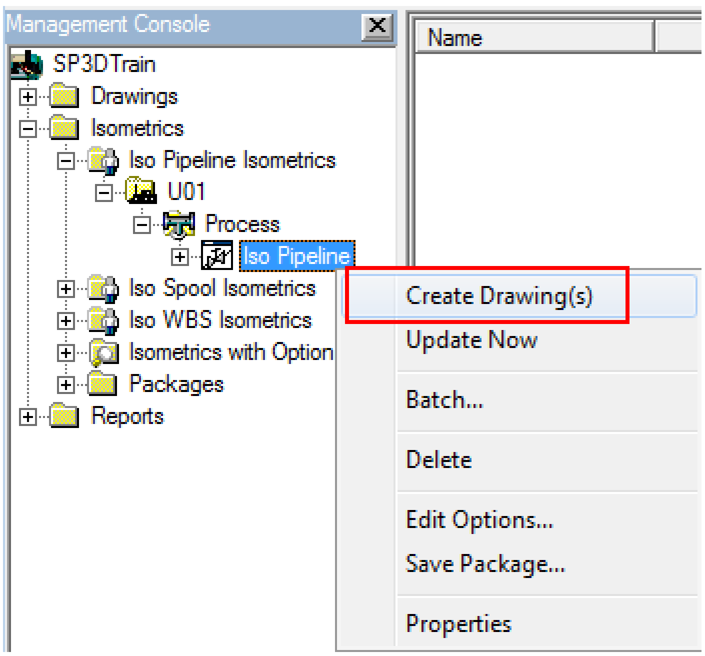
1. In the **Management Console**, expand the drawing hierarchy to **Isometrics > Iso Pipeline Isometrics**.
2. Right-click the **Iso Pipeline Isometrics** component in the **Management Console** and click **Run Query**. Smart 3D uses the query defined in the Iso Pipeline Isometrics component to find the pipelines in the model and display the results of the search in the drawing hierarchy.



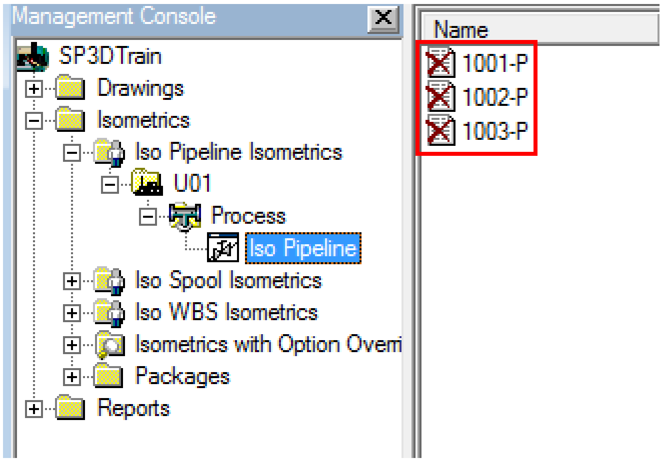
The Unit U01 and the piping system available in U01 appear in the hierarchy of Management Console. The piping isometric drawing type Iso Pipeline also appears in the hierarchy.



3. Right-click **Iso Pipeline** component in the hierarchy, and click **Create Drawings** to create the isometric drawing documents.

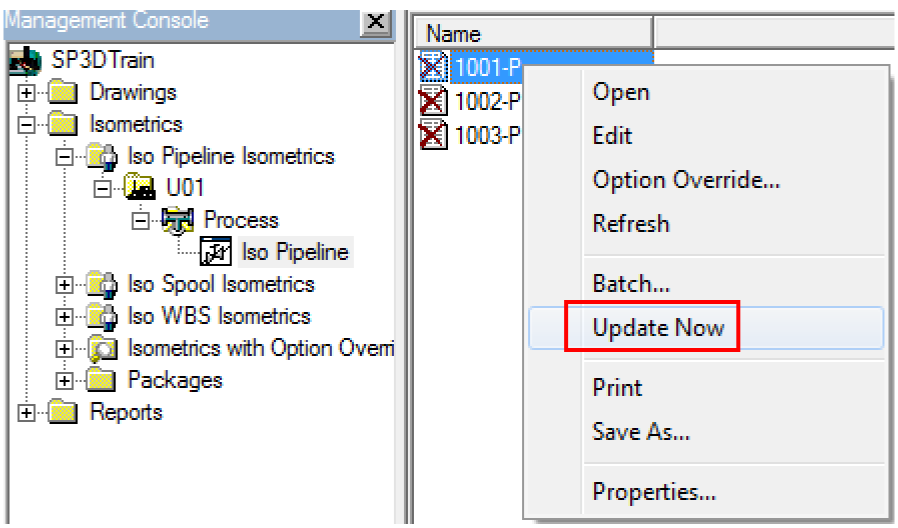


Smart 3D generates isometric drawing documents for all the pipelines available in Unit U01.

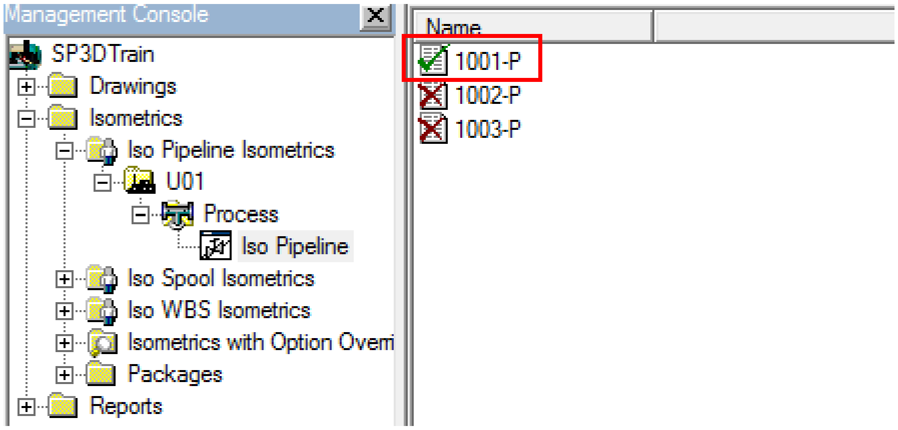


The cross mark on the drawings for each documents of Unit U01 shows that either the drawings are not created yet or they are not updated.

4. Right-click the isometric drawing document of 1001-P, and click **Update Now** to update the drawing.

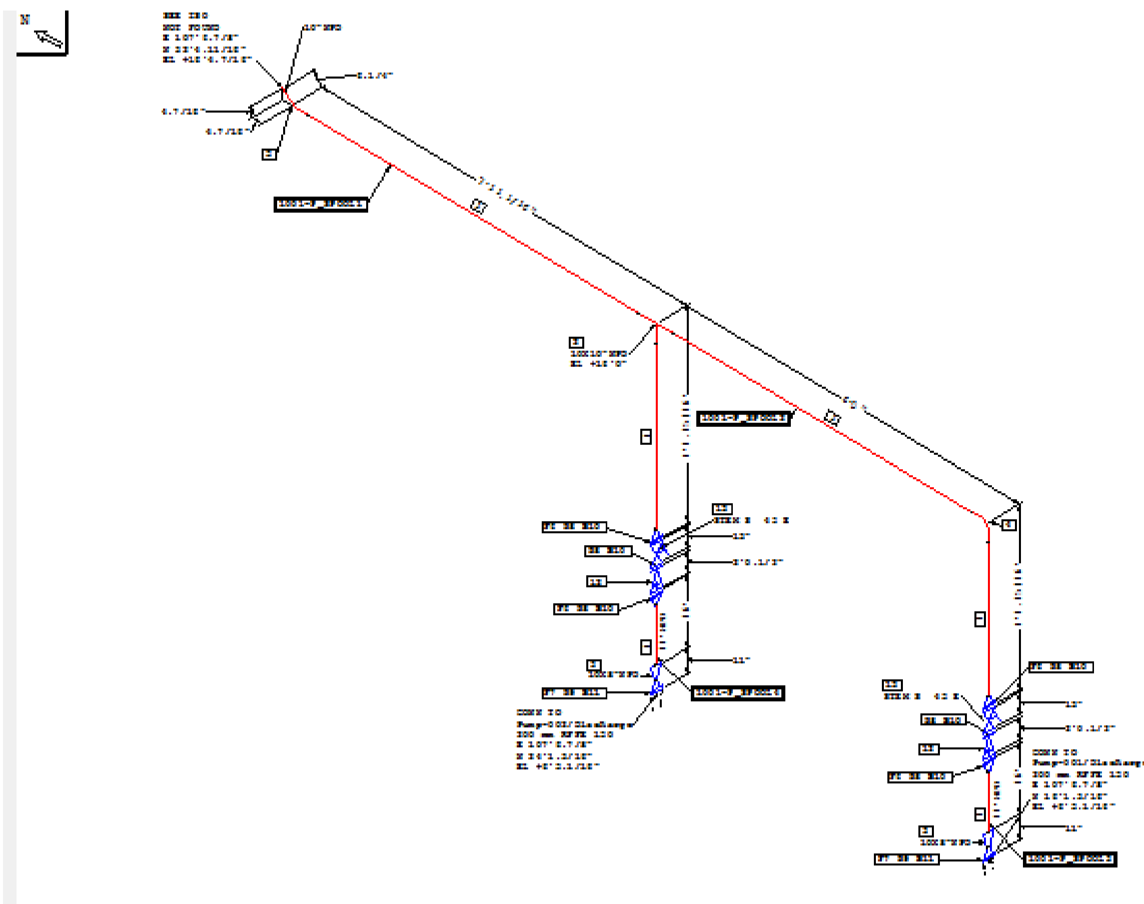


Smart 3D gets the piping parts and generates the isometric drawing of the pipeline 1001-P. A green check mark appears on 1001-P that indicates that the isometric drawing for 1001-P has been generated.



5. Double-click **1001-P** document to open the drawing viewer.

   The drawing viewer displays the pictorial representation of isometric drawing on the left panel of the screen and the material take-off for the pipeline 1001-P on the right panel of the screen.



The drawing viewer also displays the material take-off sheet for 1001-P. This sheet displays the details of the material required for 1001-P.

SHOP MATERIALS

PT NO	DESCRIPTION	NPD (IN)	CMDTY CODE	QTY
1	Pipe, S-STD, BE, ASTM-A53-B Type S	10	[illegible]	20.5'
2	Tee, S-STD, BE, ASTM-A234-WPB, ASME-B16.9	10x10	[illegible]	1
3	Concentric reducer, S-STD x S-STD bore, BE, ASTM-A234-WPB, ASME-B16.9	10x8	[illegible]	2
4	90 deg LR elbow, S-STD, BE, ASTM-A234-WPB, ASME-B16.9	10	[illegible]	1
5	45 deg LR elbow, S-STD, BE, ASTM-A234-WPB, ASME-B16.9	10	[illegible]	1
6	Flange, CL150, RFFE BE, ASTM-A105, ASME-B16.5, WN, S-STD bore bore to match	10	[illegible]	4
7	Flange, CL150, RFFE BE, ASTM-A105, ASME-B16.5, WN, S-STD bore bore to match	8	[illegible]	2

OTHER THAN SHOP MATERIALS

PT NO	DESCRIPTION	NPD (IN)	CMDTY CODE	QTY
8	Gasket, CL150, 0.125" thk, 304 spiral wnd, graph filled, CS center ring, API-601	10	[illegible]	6
9	Gasket, CL150, 0.125" thk, 304 spiral wnd, graph filled, CS center ring, API-601	8	[illegible]	2
10	Studbolts, ASTM-A193-B7 - 4.75 in. Length	7/8	[illegible]	72
11	Studbolts, ASTM-A193-B7 - 4.5 in. Length	3/4	[illegible]	16
12	Check valve, CL150, RFFE, BC, swing, ASTM-A216-WCB, trim 8, Pacific 180	10	[illegible]	2
13	Gate valve, CL150, RFFE, BB, OS&Y, ASTM-A216-WCB, trim 8, Crane 47	10	[illegible]	2

PIECE MARKS

1001-P_SPOOL1 1001-P_SPOOL2 1001-P_SPOOL3 1001-P_SPOOL4

Resolving Isometric Extraction Problems

Because there are many steps involved in the generation of a completed isometric drawing, so there are many ways for the process to fail. When a problem occurs and a drawing generation fails, you should follow a check list mentioned below to determine the cause of the problem:

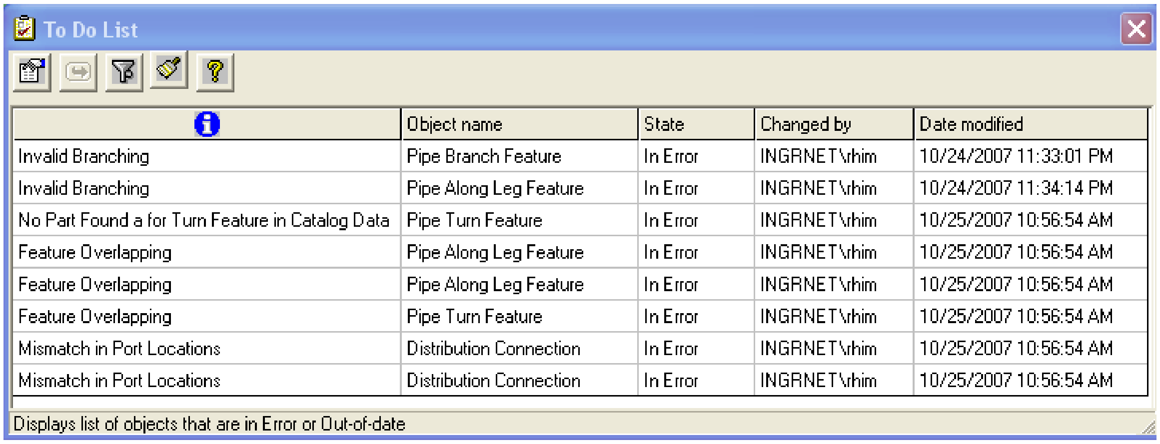
- Review Model Integrity
- View Extraction Data Dialog Box
- Review the Error Log and Iso Gen Error log
- Look up the Error Message

Review Model Integrity

The isometric extraction is dependent on the data in the pipelines, so it is essential to verify the integrity of the pipeline before trying to extract isometrics. The following lists the commands that can be used to identify these problems:

- **To Do List Command**

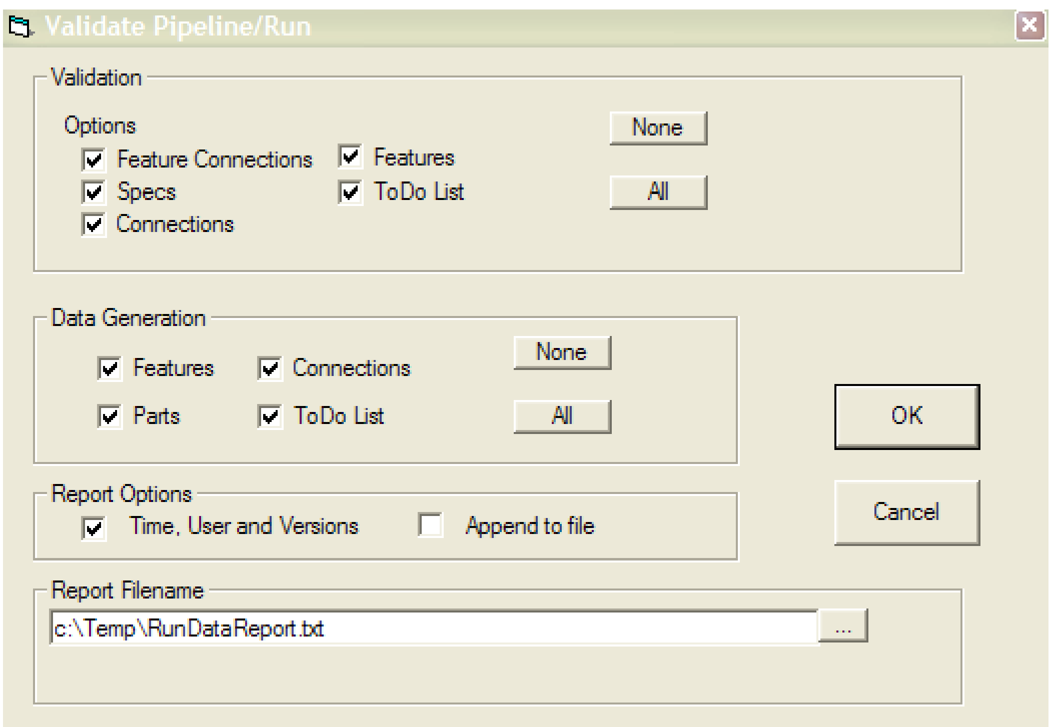
The To Do List command is available on the View menu in any modeling task. It provides users a dialog box to see objects in the workspace that have inconsistent data. The note column indicates the description of the problem.



Refer to **To Do List** of Smart 3D Common Section for more information on To Do List Command.

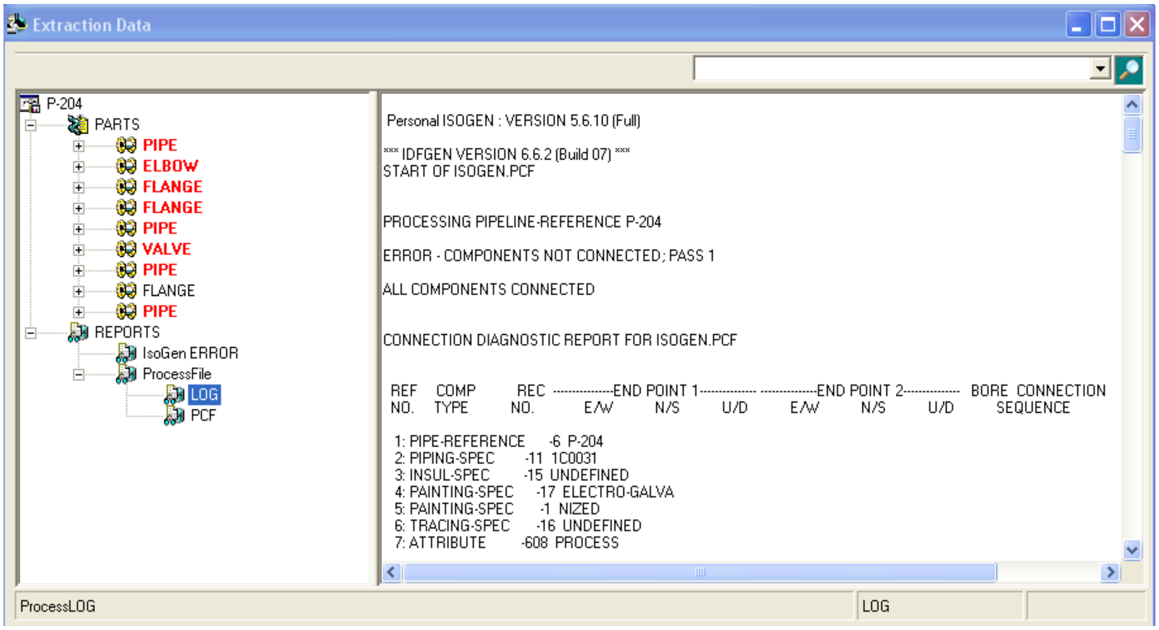
- **Validate Pipeline/PipeRun Command**

  In the Piping Task, run the Validate Pipeline/Run command as the pipelines are routed. This command checks the selected pipeline or pipe run for inconsistent and illegal pipe run topology, port overlaps, and provides user a validation report to check for errors/warnings on the selected objects. This command is available when selecting a pipe run or pipeline and press Ctrl + Shift + D keys.



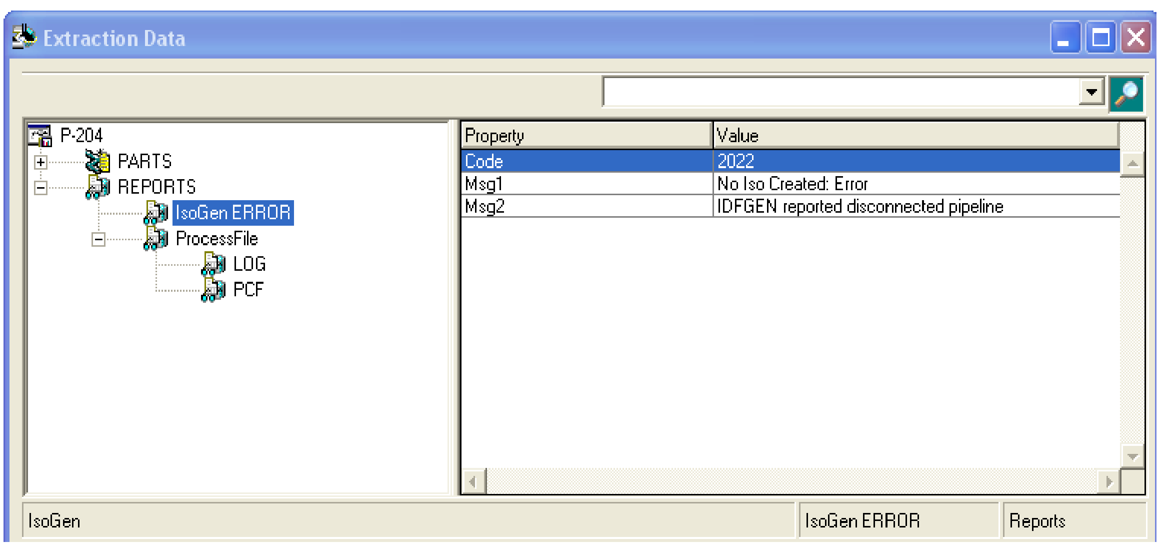
View Extraction Data Dialog Box

This dialog box provides troubleshooting information for the isometric drawing extraction process by displaying parts and reports. You can access this dialog box by right-clicking a piping isometric drawing and selecting the View Extraction Data option on the menu.



Review the IsoGen Error Log

The system creates a log file and is displayed in the View Extraction Data dialog box. The log file provides basic information about the job submission and the processing of the piping objects.



Lookup the Error Message:

Refer to the help file (Appendix: Personal ISOGEN Return Values) for an explanation of any error or warning messages. In many cases, the message text might be enough to enable you to resolve the problem easily.

Error Value	Description
1999	Isogen has returned a FAIL for one or more drawings in the set. This means it has been unable to layout the drawing successfully.
2001	Inconsistent units combination (imperial coordinates and metric bores).
2022	IDFGEN has detected disconnected pipeline. Drawings have still been created that will indicate the cause of the disconnection.

For more information related to Automated WBS Creation refer to the Group Pipe Parts Command topic in the user guide PipingUsersGuide.pdf:

For more information related to creating isometric drawings, refer to the Piping Isometric Drawings by Query: An Overview topic in the user guide PipingIsometricDrawingsUsersGuide.pdf.

Routing Piping from a Reference 3D Model

Objective

By the end of this session, you will be able to:

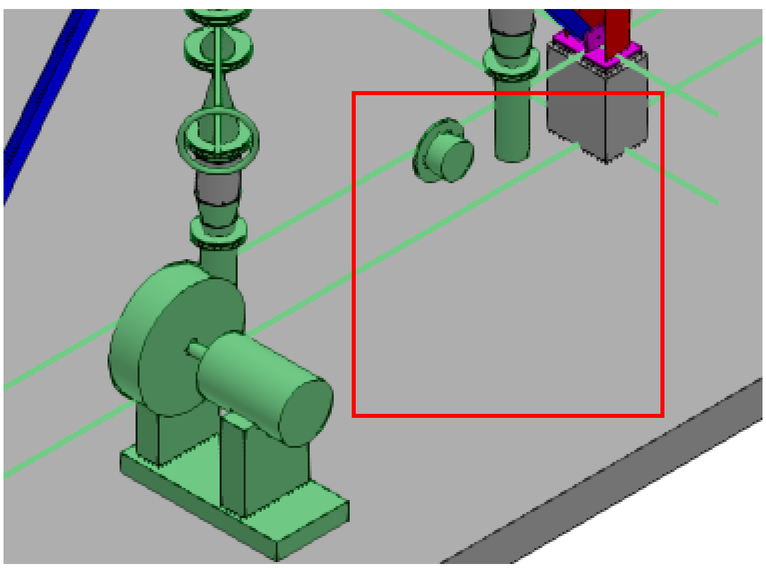
- Route to and from nozzles in a Reference 3D Model

Overview

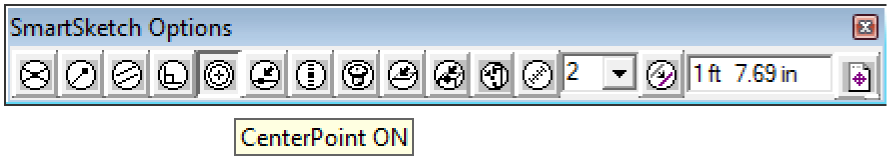
In Smart 3D, you can to route and insert components on Reference 3D Model nozzles. Smart 3D allows you to begin modeling from the nozzles that are in imported Reference 3D Models. This includes the generation of appropriate mating parts and connection items such as gaskets or welds. Because not all required information is available from the R3D nozzles, the user will still need to include any specific properties in the new modeled piping.

1. Define the workspace using the **Reference 3D** filter found in **Training Filters**.
2. Activate the **Piping** task.
3. Change the **Locate Filter** to **All**, and make sure that the active permissions group is set to **Admin**.
4. Left-Click on the kettle heat exchanger in the Reference 3D model and then hover over it.

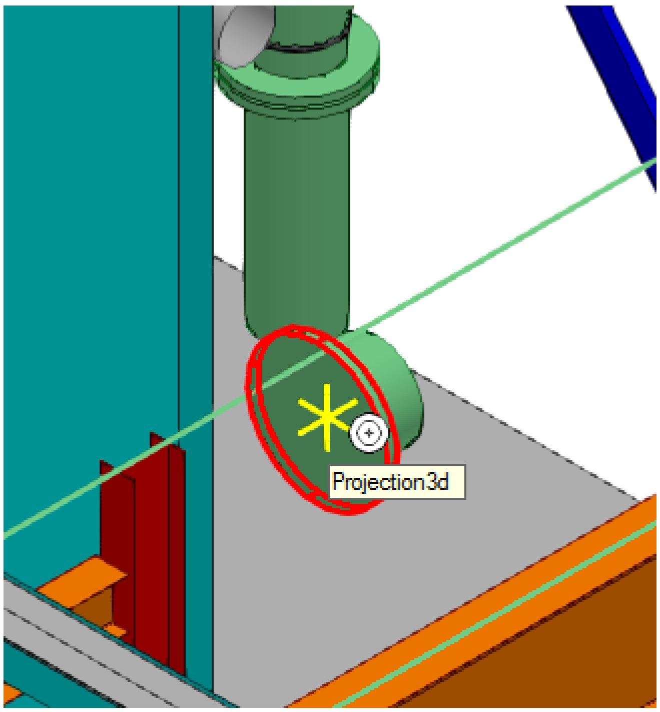
   Notice that E-102 shows up in the graphic display. This displays the intelligence that comes with importing a Reference 3D model.
5. Find **Pump-002** and select it. Go to **Tools > Hide**. Only the nozzles that were originally on the pump should remain in the graphic view.



6. Select **Tools > SmartSketch Options** and turn every option off except CenterPoint.

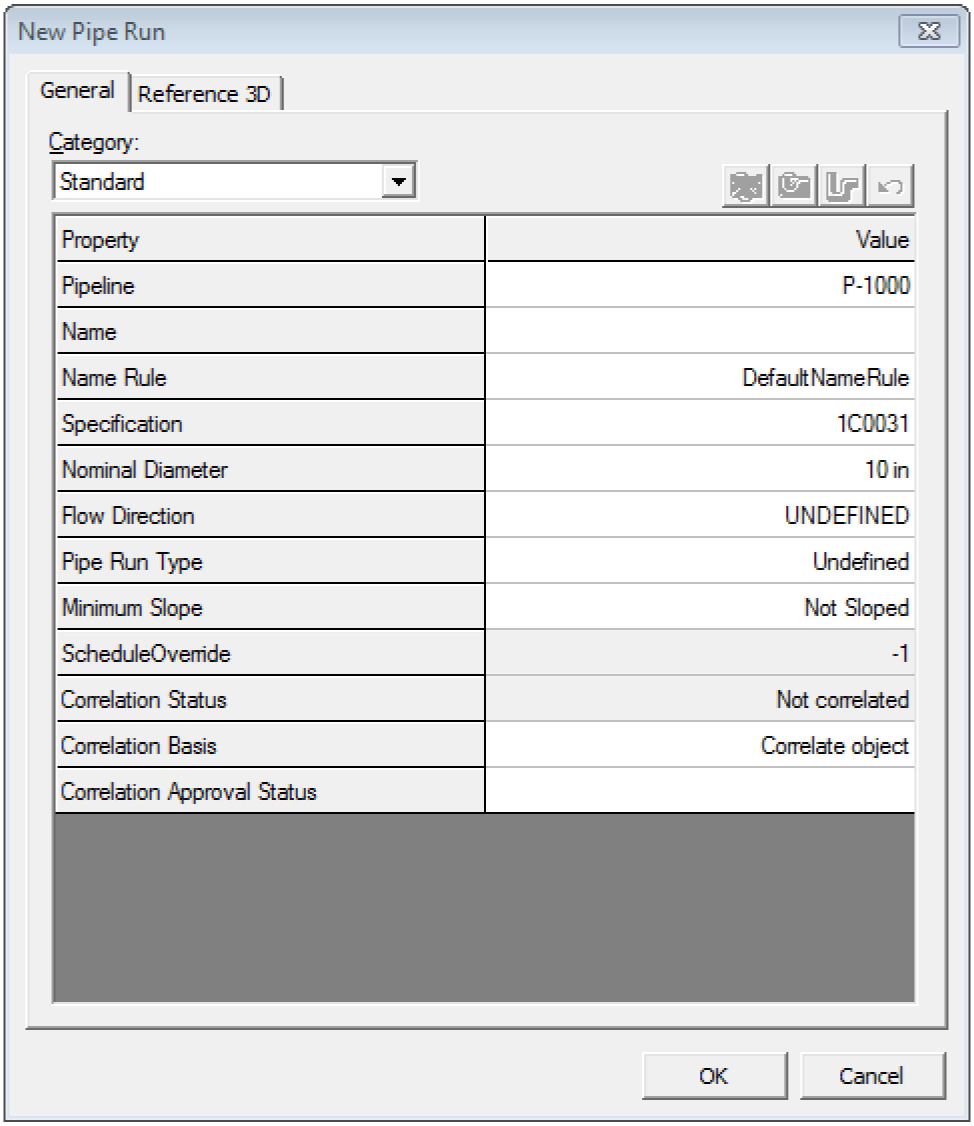


7. Exit the **SmartSketch Options** dialog.
8. Click **Route Pipe** and hover over the suction nozzle to add it to SmartSketch.
9. After the nozzle is added to the Smart Sketch list, left-click on the Projection 3d surface to route from.

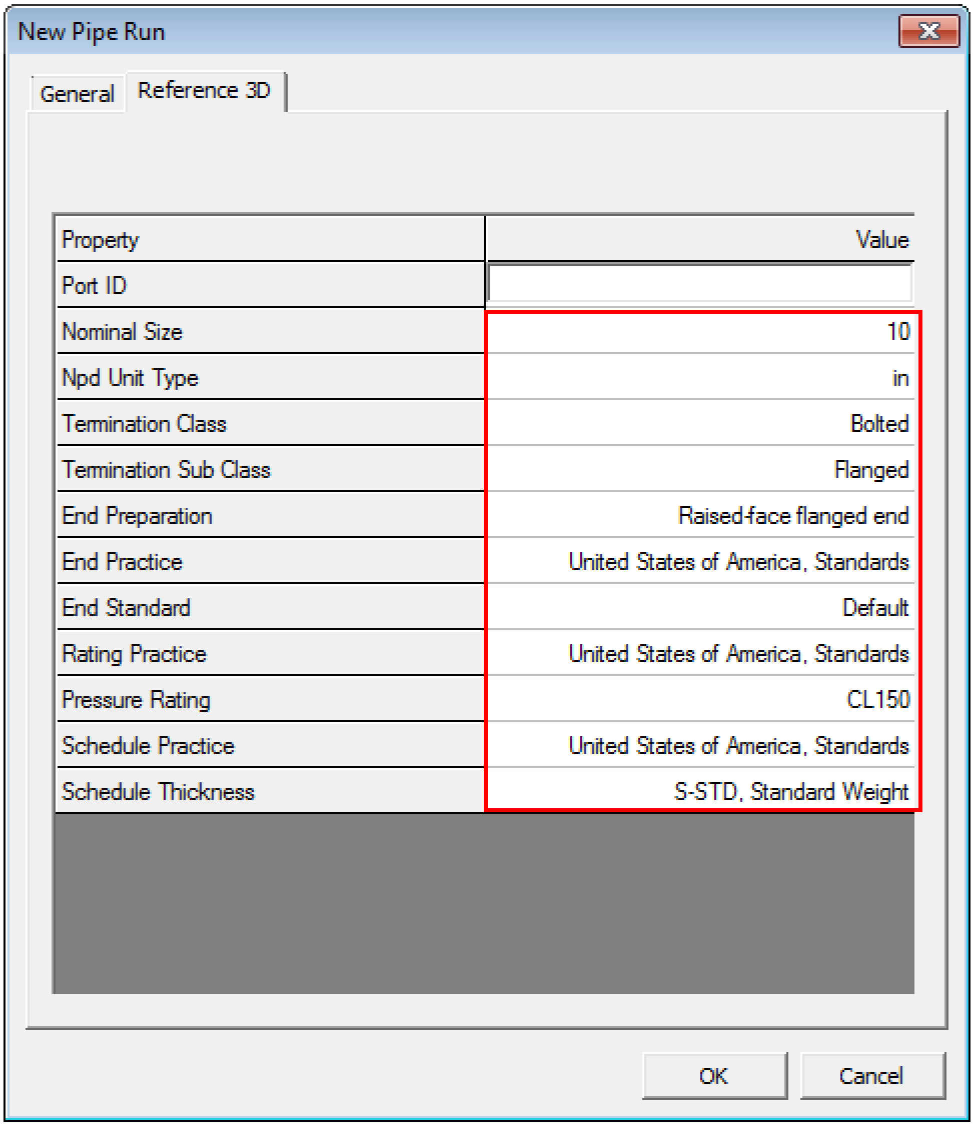


The New Pipe Run Dialog appears.

10. On the **General Tab**, browse for pipeline **P-1000** in **A1 > U11 > Process**. Change the rest of the properties to match below.

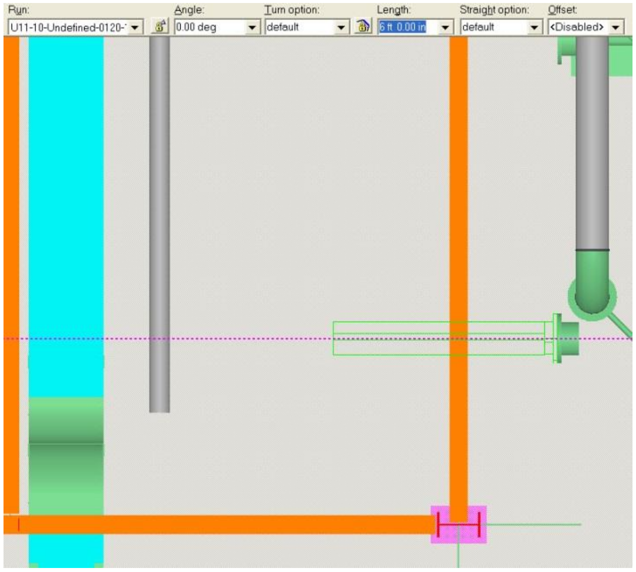


11. On the Reference 3D tab, change the properties to match below.



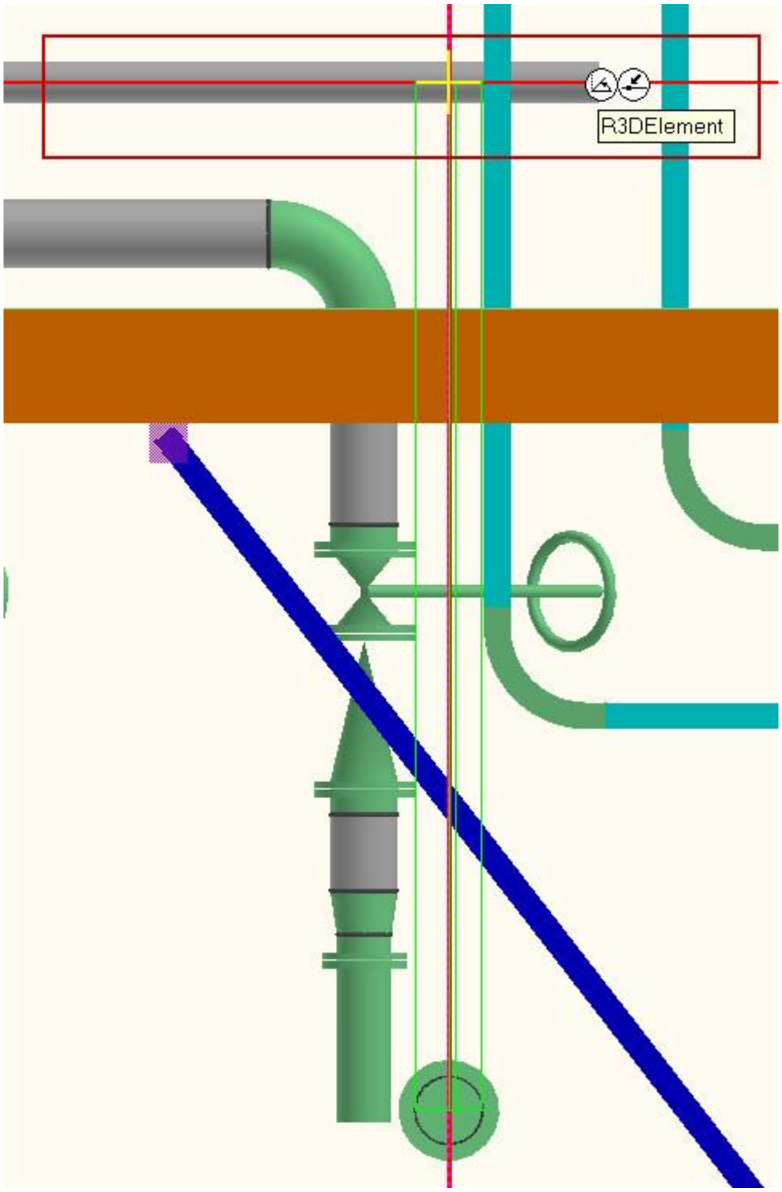
12. Click **OK**.

13. Switch the view to **Looking Plan** and route the pipe with the angle locked at 0 degrees. Input **6 ft** for the length. Left Click in the graphic view to place the pipe.



14. Go to **Tools >SmartSketch Options** and turn every option on. Exit the dialog.
15. Switch the view to **Looking East** and change the **Plane** to **East-West**.

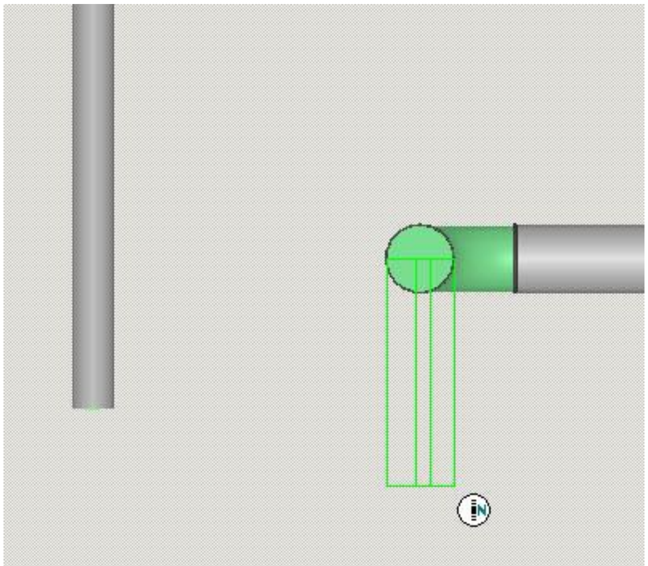
16. Route the pipe upwards with the length unlocked. Find the centerline of the pipe highlighted below and left-click to place the pipe at this elevation.



17. Close any dialog box that might appear.

    The referenced pipe causes this dialog to appear.

18. Switch the view to **Looking Plan** and continue routing the pipe **4 ft** North with **No Plane** selected. Left-click to place the pipe.



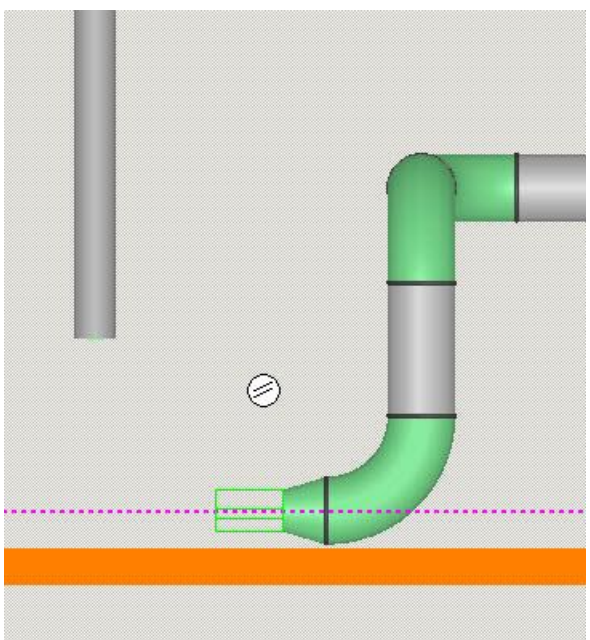
19. Select **Insert Component** and select **90 Degree Direction Change** from the **Type** dropdown. Enter **270 degrees** in the **Angle** box. Click **Finish**.

20. Select **Insert Component** and select **Concentric Size Change**.

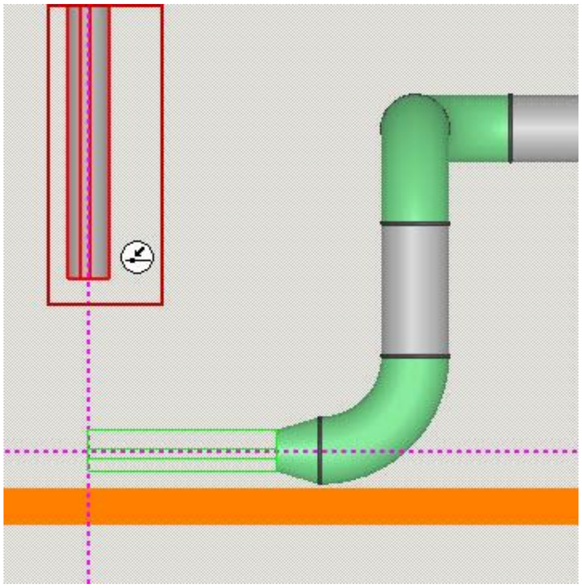
    The Run dropdown will prompt for a Run selection.

21. Choose **<New Pipe Run>** and change the **Nominal Diameter** to **6 in**. Click **OK**.

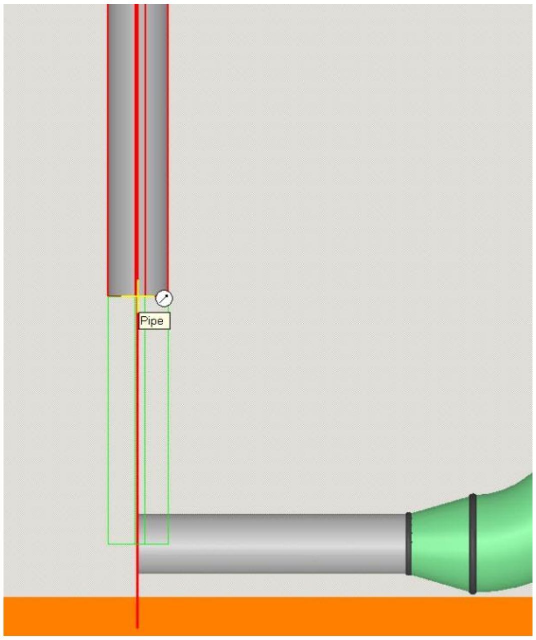
    The graphic view should be similar to this.



22. With the route plane turned to **Plan Plane**, find the centerline of the pipe highlighted below and left-click to place the pipe.

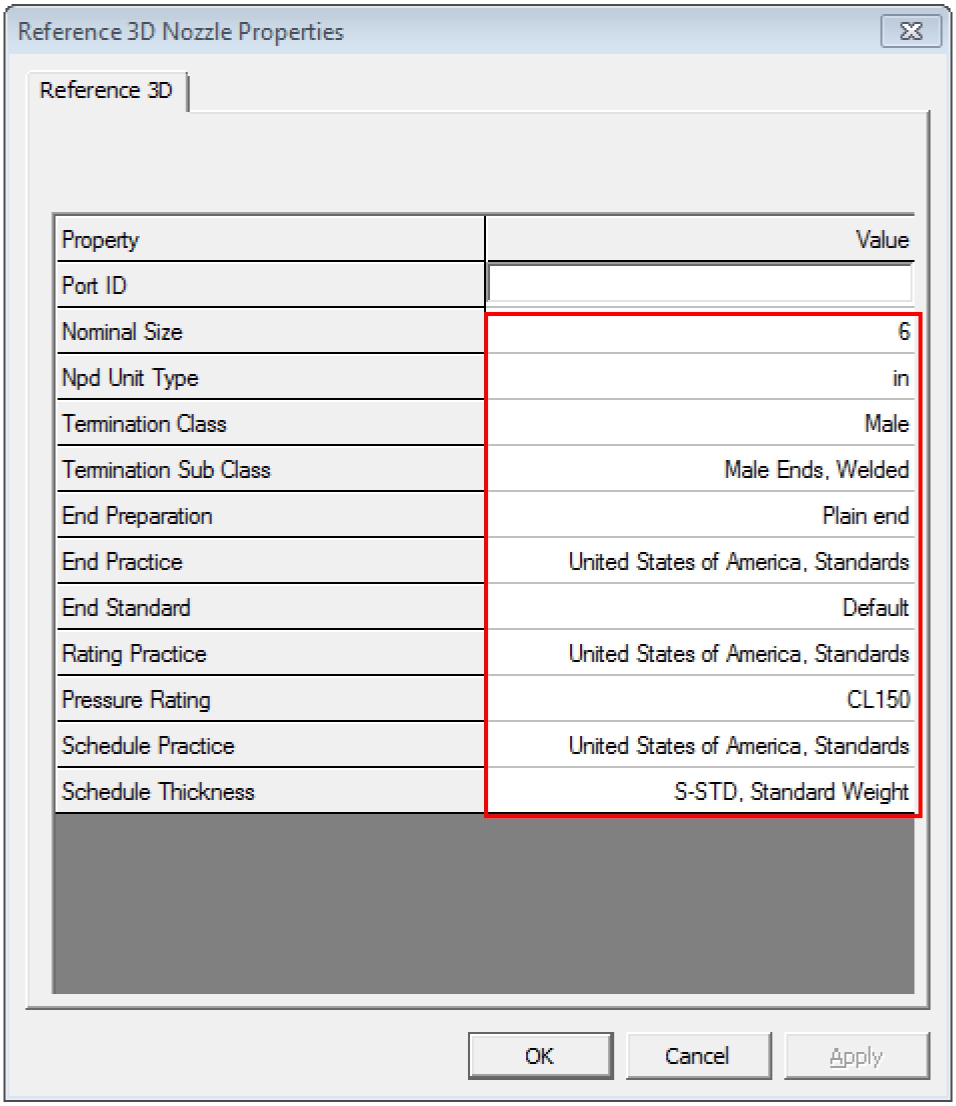


23. Route toward and connect to the end of the Reference 3D pipe. Left-click to place the pipe.

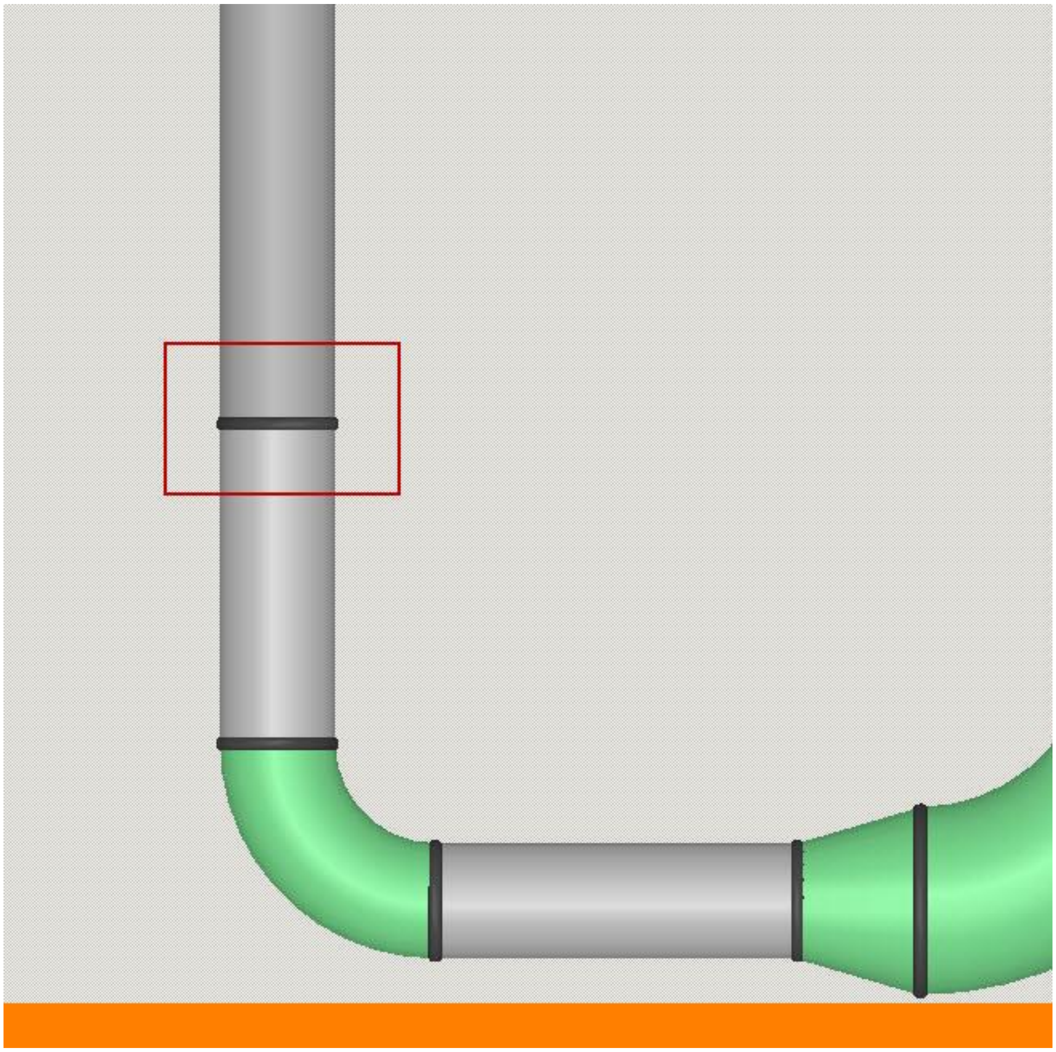


The Reference 3D Nozzle Properties dialog will appear.

24. Change the properties of this dialog to match below.



25. Notice that a weld specified in the **Reference 3D Nozzle Properties** appears at the connection point as shown.



SECTION 2

Working with WBS Items

Objective

- By the end of this session, you will be able to:
- Sequence piping objects in a model.

Prerequisite Sessions

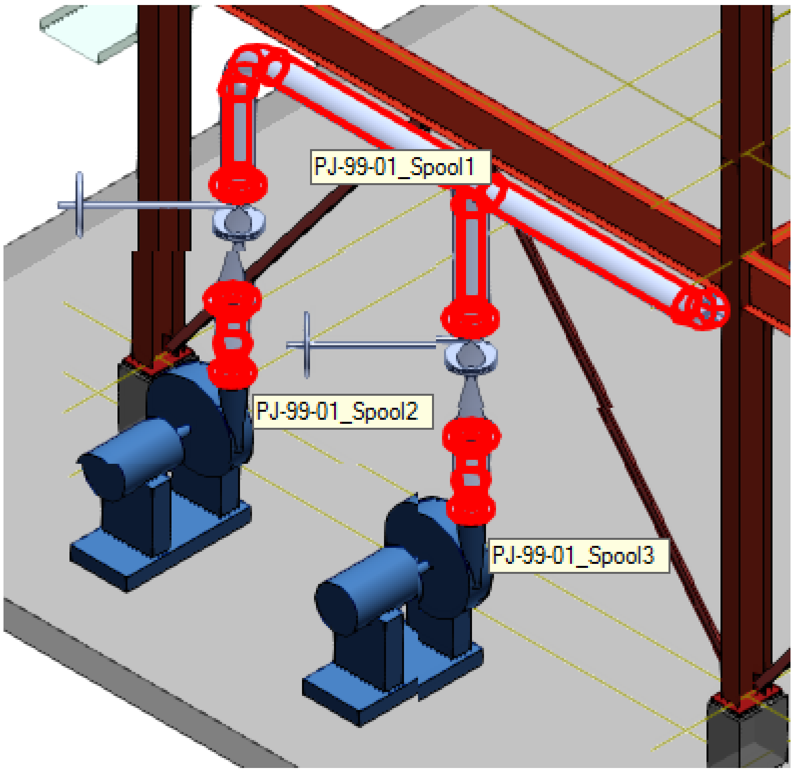
- Smart 3D Overview
- Smart 3D Common Sessions
- Piping: An Overview
- Routing Pipes

Overview

The WBS is a means to define additional groupings of design objects for whatever work purposes your company might need. It is well suited for dividing the design for contracting or fabrication purposes.There are two basic approaches for creating spool assemblies, sequencing piping objects, and for creating isometric drawings. These are by Pipeline Objects, and By WBS Items. In a previous session, you learned there are several methods for working with WBS Items. This session will cover the procedure for creating standard spools, sequencing pipe welds, and create isometric drawings from WBS items.

Creating Standard Spools from WBS Items

In the following exercise, you will be creating standard spools on the pipeline **1001-P** in **Unit U01** of your workspace by using the **Generate Spools** command. Before creating the spools, assign the piping objects to an active project and a WBS item. After the spools are created, the workspace should look like this.



Before Starting this Procedure

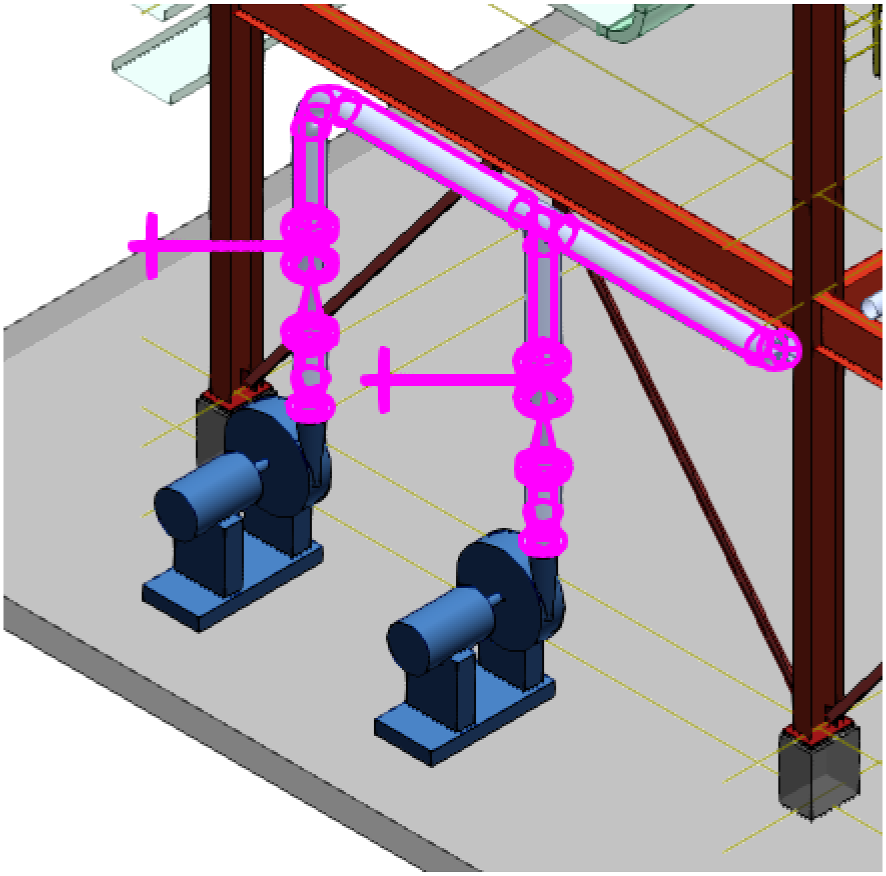
- Define your workspace to display **Unit U01**, coordinate system **U01 CS**, and **Projects in the WBS** hierarchy. In your training plant, select **U01 and WBS Items** compound filter from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

Manually Assigning Piping Objects to an Active Project and a WBS Item

Assign all the piping objects of the pipeline 1001-P to an active project PJ-99 in Unit U01 of your workspace.

1. Select **Piping Parts** in the **Locate Filter** to select only the piping parts in the graphic view.

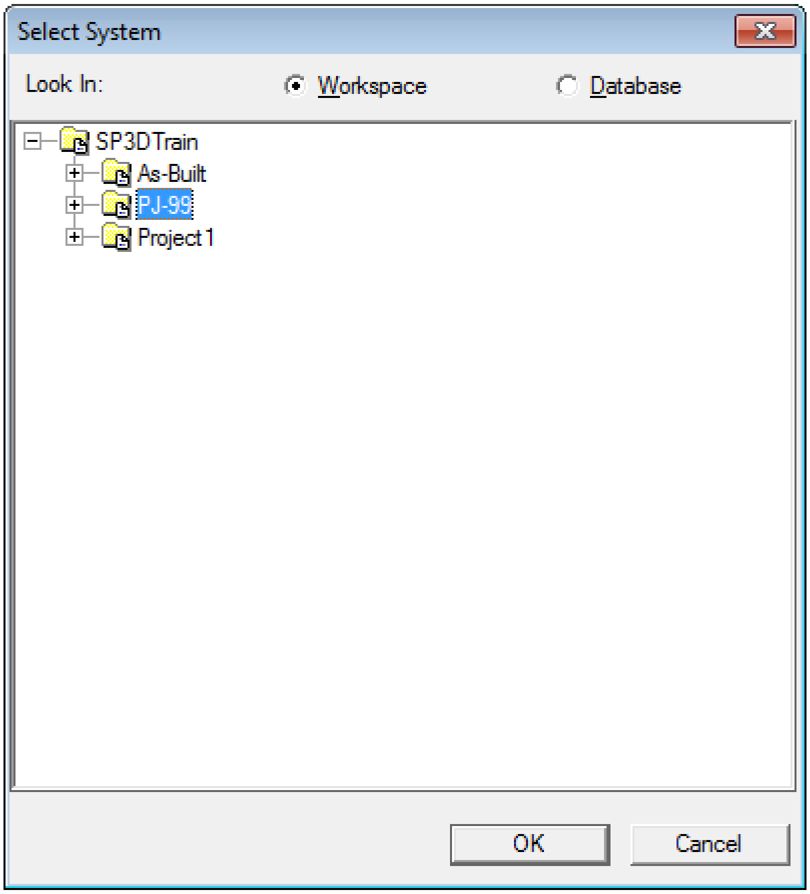
2. Select **Inside Fence**  on the Common toolbar to select all the piping parts of the pipeline **1001-P**.



3. To specify an active project, select **More…** in the **Active Project** drop-down list on the main toolbar.

The Active Project dialog box appears.

1. Select the **Database** option to see all the WBS projects in the dialog box. Next, select **PJ-99** to specify PJ-99 as an active project.

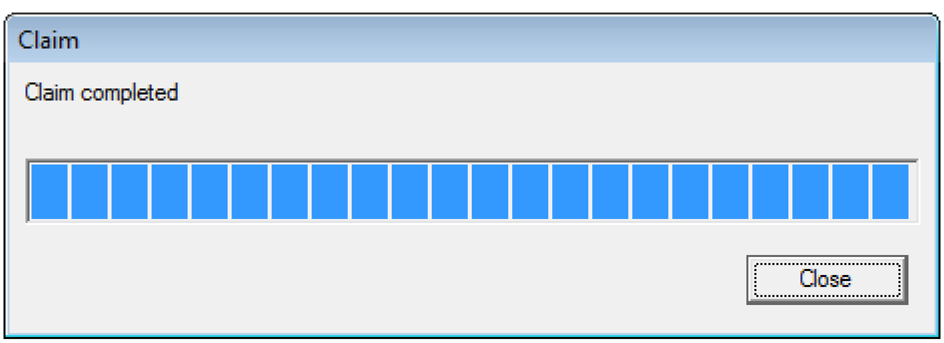


2. Click **OK** in the Active Project dialog box.

The selected active project should appear on the main toolbar.

1. Select **Project > Claim** to associate the selected piping objects with the active project PJ-99.

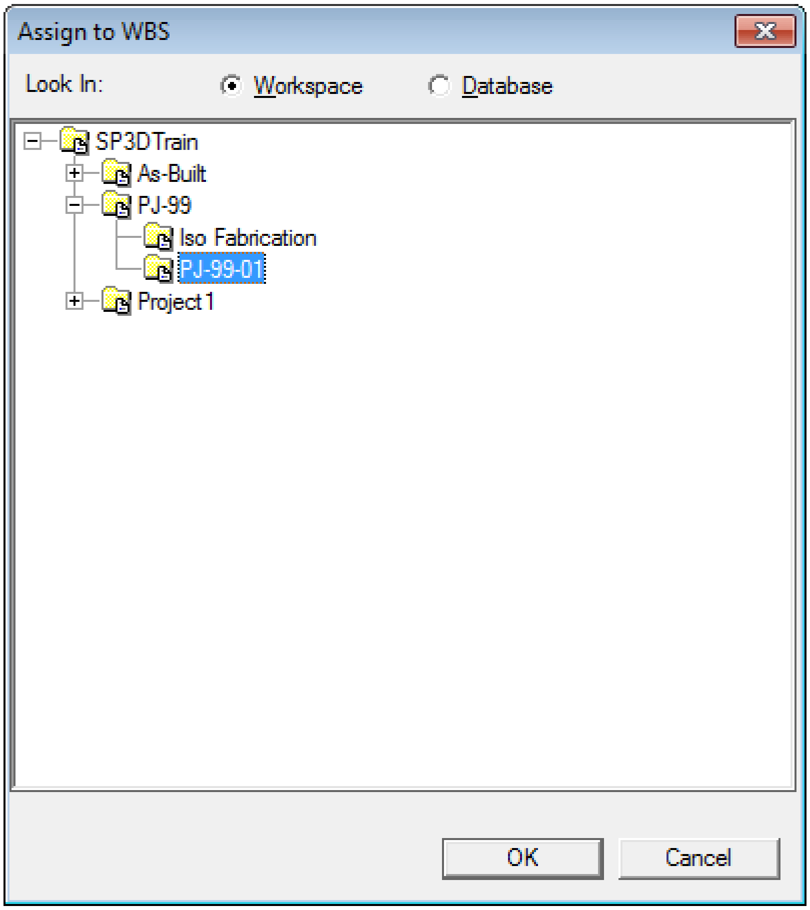
After the claim process is complete, Smart 3D displays the Claim dialog box.



1. Click **Close** to close the dialog box.
2. Select **Project > Assign to WBS…** to associate the selected piping objects as a WBS item.

The Assign to WBS dialog box appears.

1. Select the **Workspace** option and expand the project PJ-99.
2. Select the WBS item PJ-99-01 and click **OK** in the **Assign to WBS** dialog box.

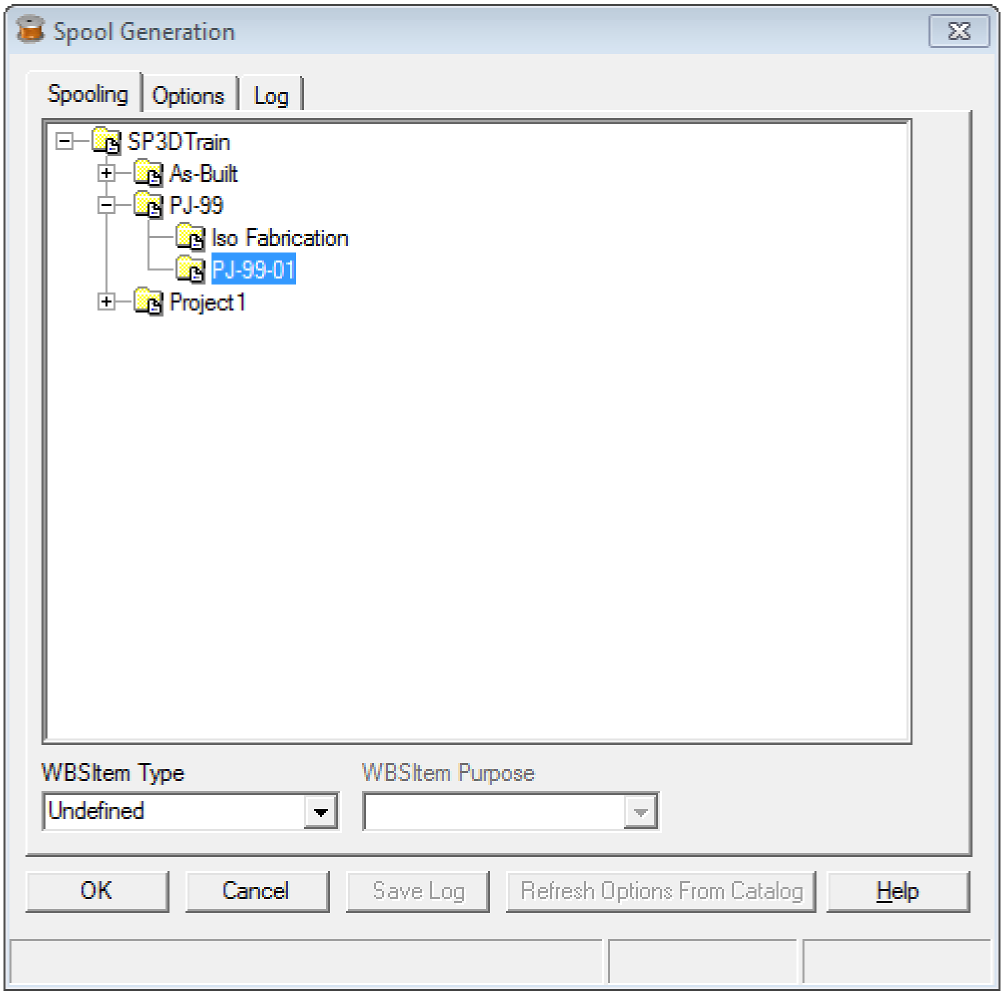


Creating Spool Assemblies from a WBS Item

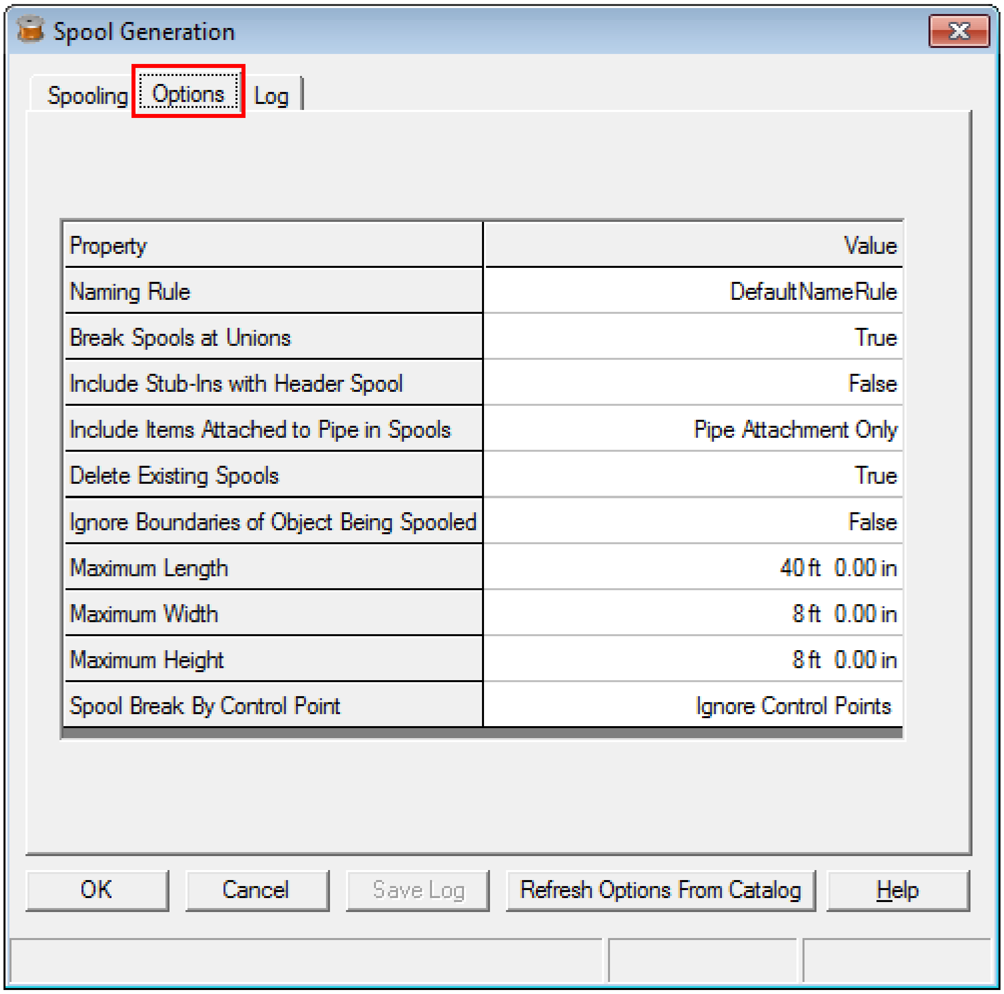
1. Click **Generate Spools** .

The Spool Generation dialog box appears.

1. On the **Spooling** tab, expand the project PJ-99 and select the WBS item PJ-99-01 to specify the WBS item where you want to generate spools. Each spool must consist of connected objects; otherwise, errors may occur in spooling.



2. Select the **Options** tab in the Spool Generation dialog box to review the spooling options.

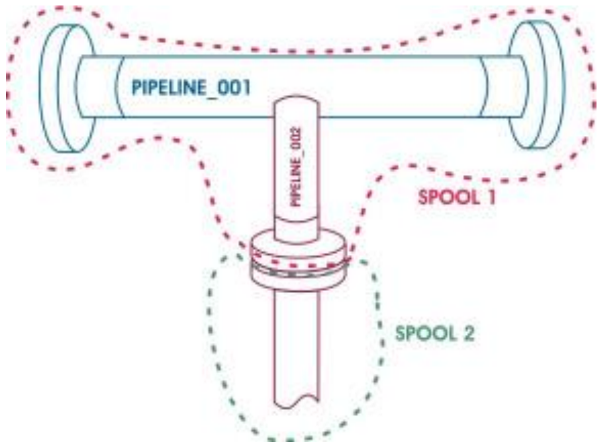


The settings on this dialog box are:

Name Rule - Specifies the naming rule for spools.

Break Spools at Union - Specifies that the software breaks the spools at unions. The union part must have its **Commodity Type** property set to "Union" for the spooling software to recognize the union as a break point.

Include Stub Ins with Header Spool - Specifies that a spool can include the stub-in pipe and all the parts of this stub-in branch until the first spool-breaking component is encountered. For example, by setting this option to **True**, the software spools the stub-in and flange for the isometric drawing with Pipeline_001 even though the stub-in and flange do not belong to Pipeline_001. When creating isometric drawings for Pipeline_001 in the Drawings and Reports task, you must use the **Isometric Style Options Browser** to set the **Drawing.Content.HonorSpoolAtBranch** option to **True**.



Include Items Welded to Pipe in Spools - Includes welded objects, such as pipe hanger or support parts, in the same spool as the components to which they are welded. Surface mounted components are included in the spool if the fabrication type for the surface mounted component is set to "By Fabricator".

Delete Existing Spools - Changes the spool numbering only at modified or added piping. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers.

This setting guarantees that the software applies the same piece marks (spool names) to spools, even if spools are regenerated.

Ignore Boundaries of Object Being Spooled - When set to **False**, the software will not cross the boundary of the pipeline or block. When set to **True**, the software will cross the boundary of the pipeline or block for spool generation until an intrinsic spool break is found. This feature is intended for use when spooling by block.

To use this option correctly, you need to understand the concept of intrinsic breaks. The spooling software starts a new spool any time it hits an intrinsic break:

- extents of the collection being spooled (pipeline or assembly)
- bolted connection
- (field) welded connection
- change in fabrication category from shop to field
- union (optional controlled by the **Break Spools at Union** option described previously)
- clamped connections

The extent of the collection being spooled is the most important intrinsic break condition. If a pipeline is spooled, the software breaks the spool at the end of the pipeline regardless of what is there. Normally, there is no conflict with the other intrinsic break conditions; however, a conflict can occasionally happen. For example, it does not matter if the pipeline is connected to another pipeline by a shop weld (which would normally not be a spool break), the software will still start a new spool at this point.

In listing the intrinsic breaks in order of importance, the extent of the collection being spooled should always be at the top because it normally trumps any other intrinsic break condition if a conflict arises. Whatever is being spooled, pipeline or block, is the "basket of parts" with which the spool software works. The software does not go outside of that basket for any reason unless the **Ignore Boundaries of Object Being Spooled** is set to **True**. In that case, the spool software steals connected parts from outside of its basket until it hits an intrinsic break, an open end, or it runs out of parts. The spool software will not steal a part from outside the provided basket if that part already belongs to another spool.

TIP In other words, to turn off the first intrinsic break, set **Ignore Boundaries of Object Being Spooled** to **True**.

Maximum Spool Length - Provides the maximum length of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Width - Provides the maximum width of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Height - Provides the maximum height of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

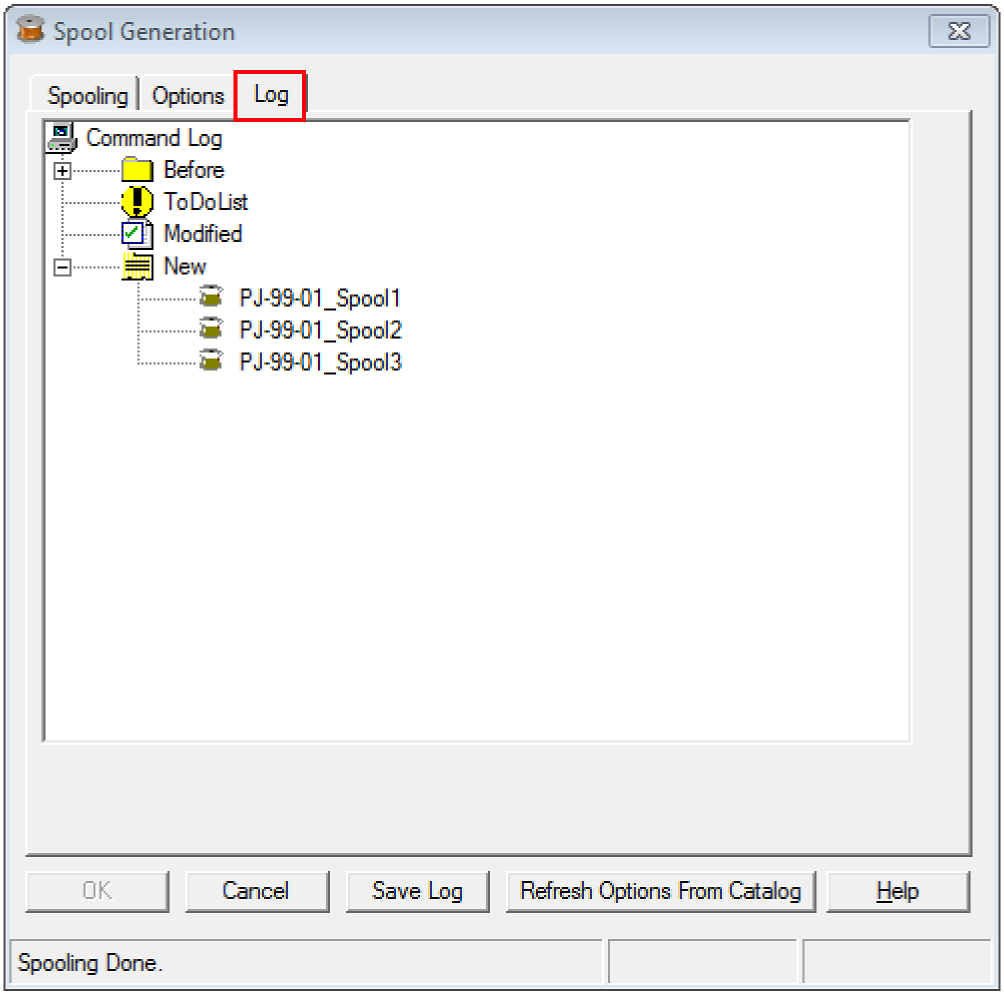
Spool Break By Control Point - Specifies if spools should break at control points.

You can place control points using the **Insert > Control Point** command. You must set the control point **Subtype** to **Spool Break** in order to use the control point with this option. In addition, the control point must be placed on the connection object.

Select **Ignore Control Points** to ignore the control points during spooling. Select **Break at Control Points** to break spools at the normal intrinsic line breaks and at control points. Select **Break Only at Control Points** to break spools only at control points.

3. Click **OK** to apply the default settings to the spool that is generated.

4. Select the **Log** tab in the **Spool Generation** dialog box to review the spooling log and close the dialog box.



Before - Lists all the spools that existed in the model before you ran the last spooling process. This is the same list of spools that displays if you select this tab before processing spools.

To Do List - Lists spools that have objects in the **To Do List**. The spools cannot be created until you fix the objects on the **To Do List**. Click **View > To Do List** (or press CTRL+T) to view the **To Do List**.

Modified - Lists all the spools that were modified during the last spooling process.

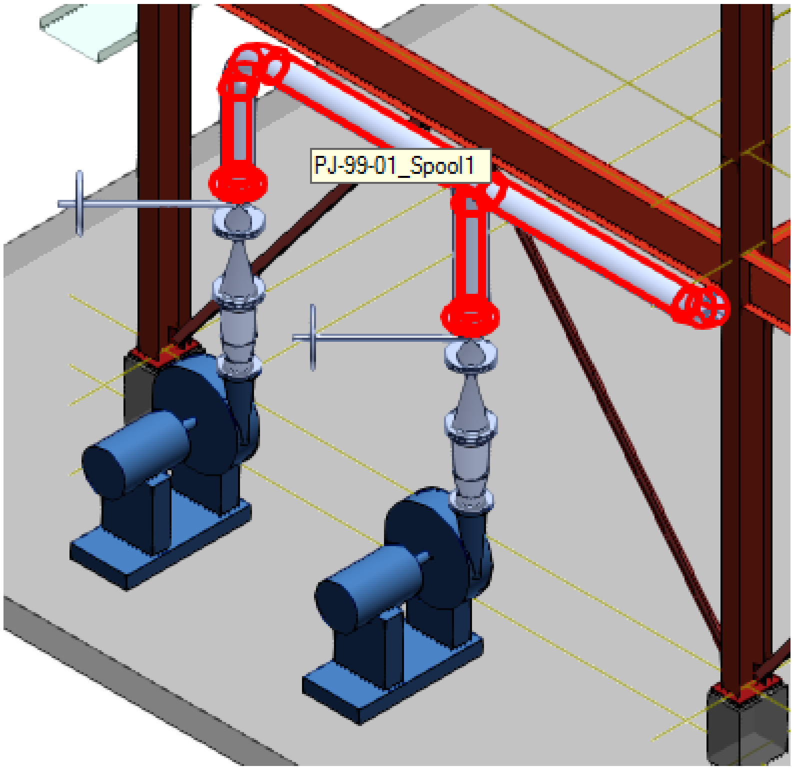
New - Lists all the spools that were created during the last spooling process.

TIPS

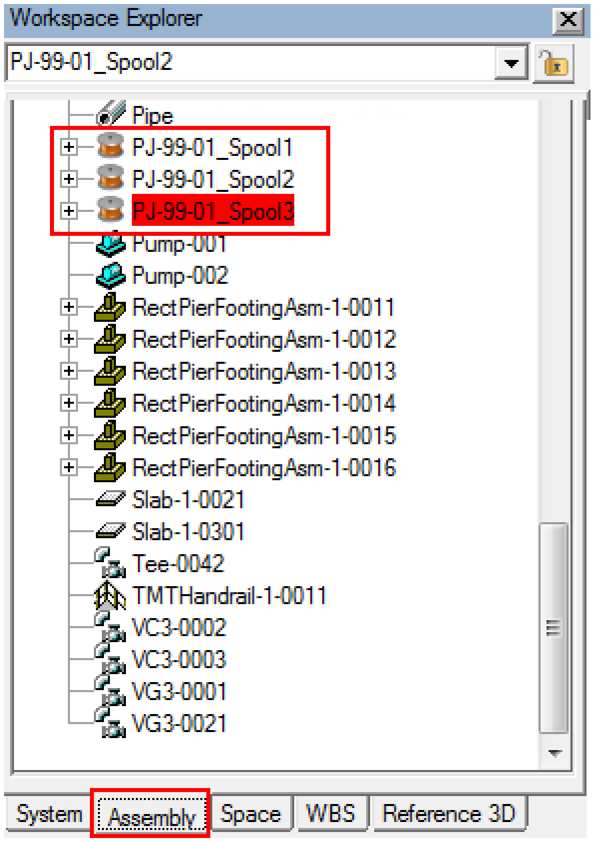
- You can select a spool, pipe, or pipe component in the list to highlight it in the model.
- You can click Save Log in the Spool Generation dialog box to save the log as a text file.
- After a spool is generated, it is stored in the model database and displayed on the **Assembly** tab in the **Workspace Explorer**. This tab is not visible by default. To display this tab, select the **Tools > Options** command and then, select the **Assembly** check

box. Save and exit the session file. After you reopen the session file, you will see the **Assembly** tab in the Workspace Explorer. The **Assembly** tab can also appear if you switch to a non-modeling task such as systems and specifications task.

5. To view the spool you created, select the **Spools** option in the **Locate Filter** drop-down list and point to the pipeline 1001-P. A tool tip will appear for the spool in the graphic view.



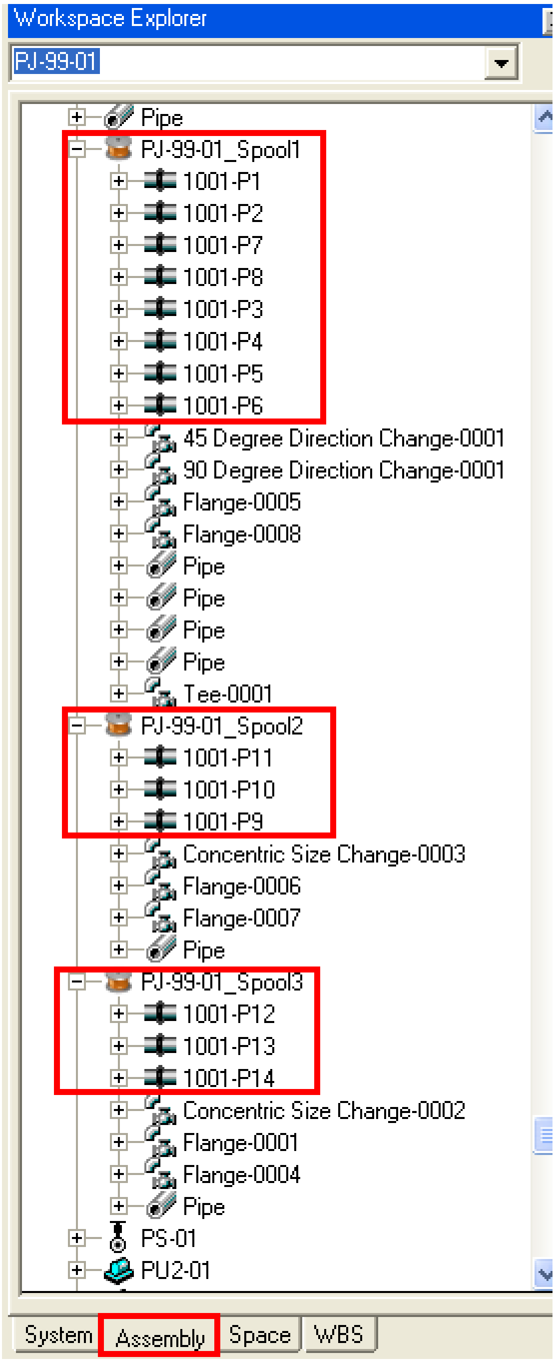
6. You can check the spools on the **Assembly** tab in the Workspace Explorer.



Sequencing Piping Objects in a WBS Item

Objective

In this exercise you will be sequencing the pipe welds of a pipeline **1001-P** in **Unit U01** of your workspace by assigning the piping objects to a WBS item **PJ-99-01**. The sequenced pipe welds on the Assembly tab of the Workspace Explorer should resemble the highlighted sections.



Before beginning the procedure:

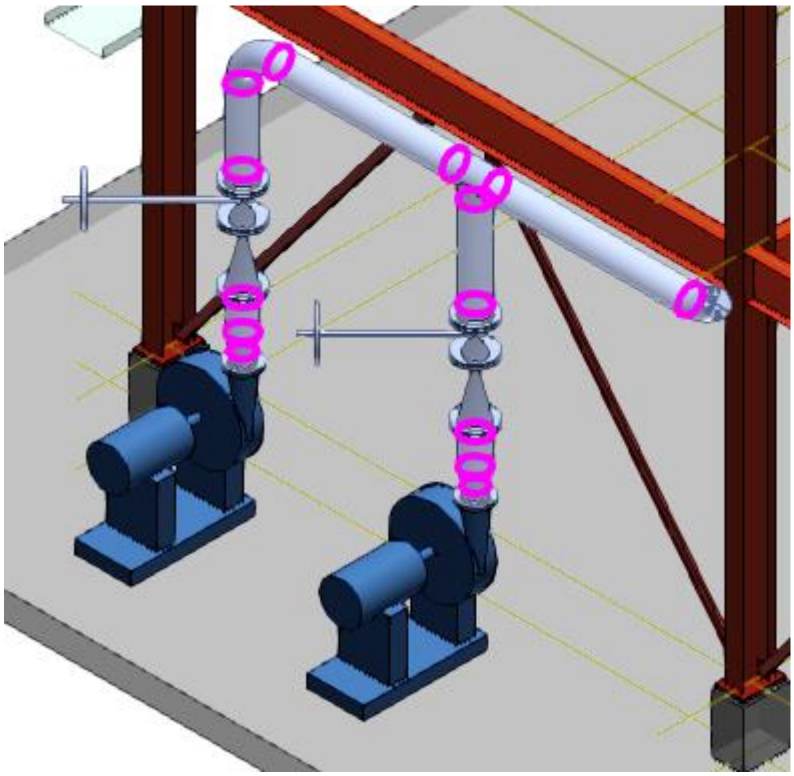
- Define your workspace to display **Unit U01**, coordinate system **U01 CS**, and **Projects** in the WBS hierarchy. In your training plant, select **U01** and **WBS Items** compound filter from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

Manually Assigning Piping Objects to a WBS Item:

Assign all the welds of the pipeline 1001-P to a WBS item PJ-99-01.

1. Select **Welds** in the **Locate Filter** drop-down list on the Common toolbar to select only the welds in the graphic view.
2. Select Inside Fence on the Common toolbar to select all the welds that are part of the pipeline 1001-P.

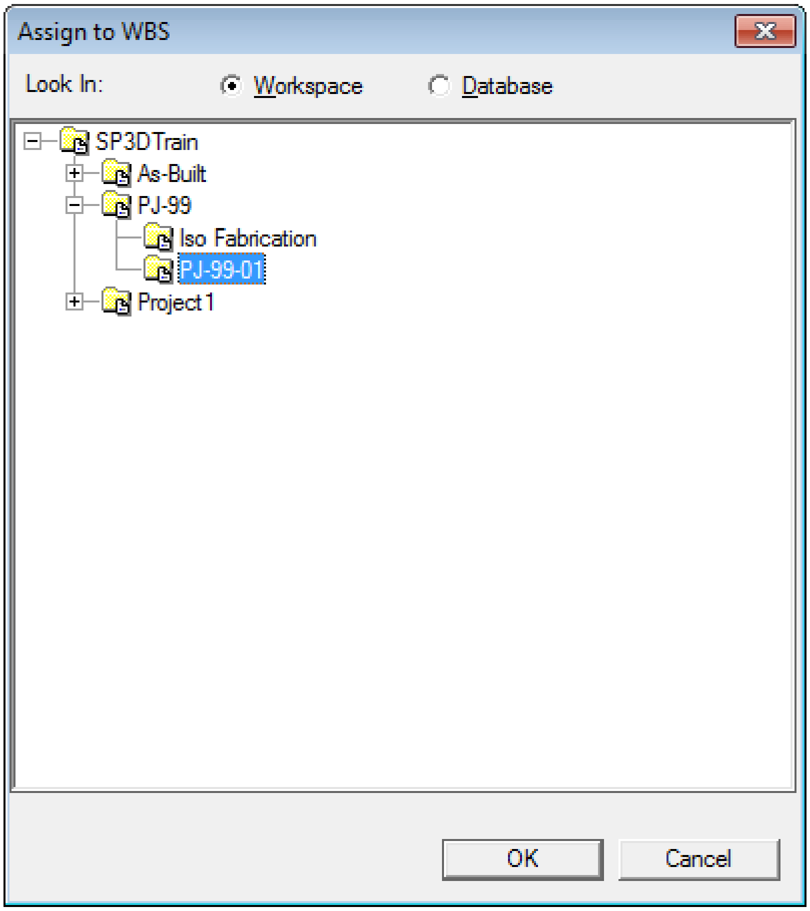


3. Select **Project > Assign to WBS…** to associate the selected piping objects to a WBS item.

The Assign to WBS dialog box appears.

1. Select **Database** in this dialog box.

2. Expand the project **PJ-99** and select the WBS item **PJ-99-01**.



3. Click **OK** in the Assign to WBS dialog box.

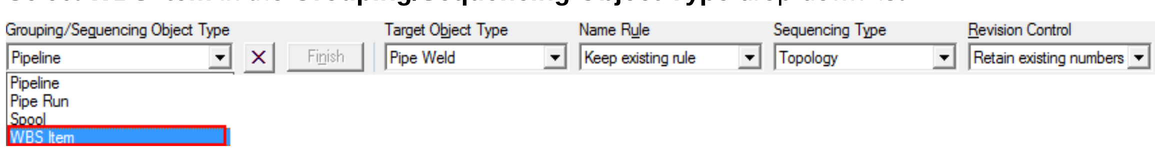
Sequencing Welds from WBS Items:

1. Click **Sequence Objects**  on the vertical toolbar.

The Sequence Objects ribbon appears.

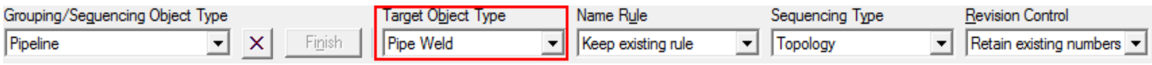
The first option on the **Sequence Objects** ribbon is **Grouping/Sequencing Object Type**, which specifies the grouping object type by what you can sequence the object. You can select a pipeline, a pipe run, a spool, or a WBS item as the grouping object. This option defines the collection of target objects to be sequenced and defines the boundaries of the sequencing.

1. Select **WBS Item** in the **Grouping/Sequencing Object Type** drop-down list.

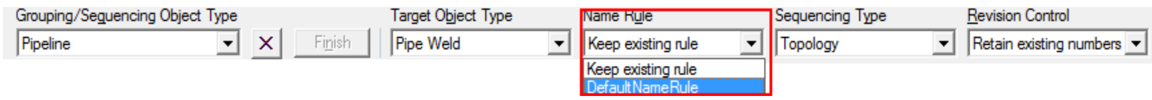


The second option on the **Sequence Objects** ribbon is **Target Object Type**, which you can select. You use this option to specify the target object that you want to rename on the selected grouping object.

2. Select the **Pipe Weld** option in the **Target Object Type** drop-down list to specify the target object to be renamed on the pipeline.

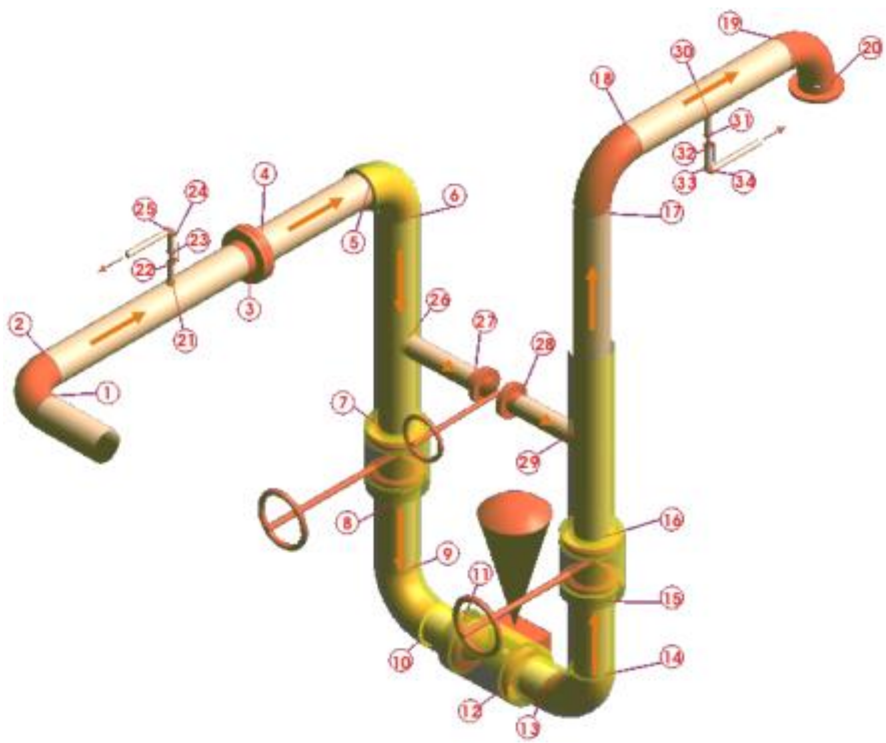


3. Select the **DefaultNameRule** option in the **Name Rule** drop-down list to specify the naming rule for renaming the target object.

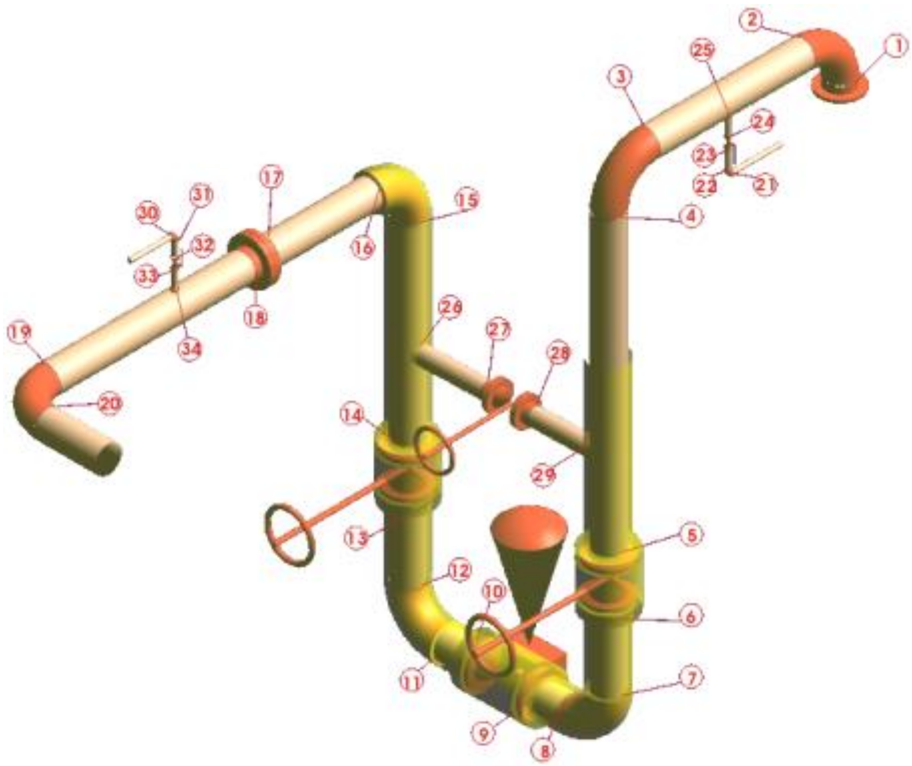


After selecting the target object, you define the sequencing type to specify the logical order of the target object. There are two options used to define the sequencing type:

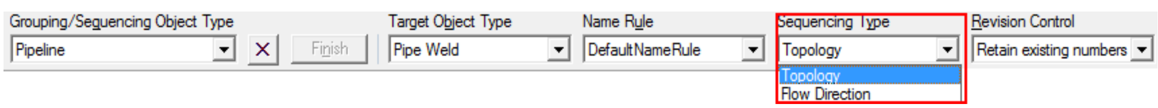
- **Flow Direction** – This option begins sequencing from the furthermost upstream point on the line and proceeds along the line with the flow direction. The subsequent branches are processed according to their flow direction.



- **Topology** - This option begins sequencing starting from the point on the primary header of the spool, pipeline, or pipe run with the highest coordinate value and works back along the target objects until all have been processed. The subsequent branches are processed from the furthermost point into the branch connection.

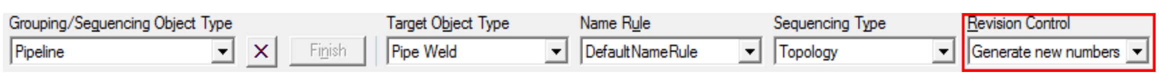


4. Select **Topology** in the **Sequencing Type** drop-down list to define the sequence type for the welds.

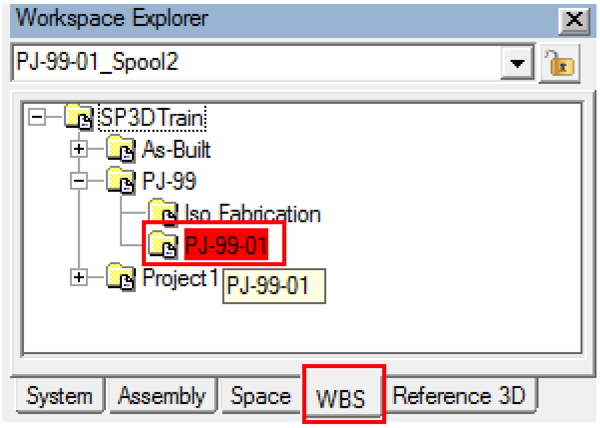


After selecting the target object, naming rule, and sequence type, you define the revision control, which specifies how you want to use the existing sequence numbers for the target object. The Retain existing numbers option keeps the sequence number of the target object that already has it, but creates a new sequence number for objects that do not have a sequence number. The Generate new numbers option discards all the sequence numbers for the selected target object and generates new number for all the target objects.

5. Select **Generate new numbers** option in the **Revision Control** drop-down list to generate new numbers for the welds



6. Select WBS item **PJ-99-01** in the Workspace Explorer.

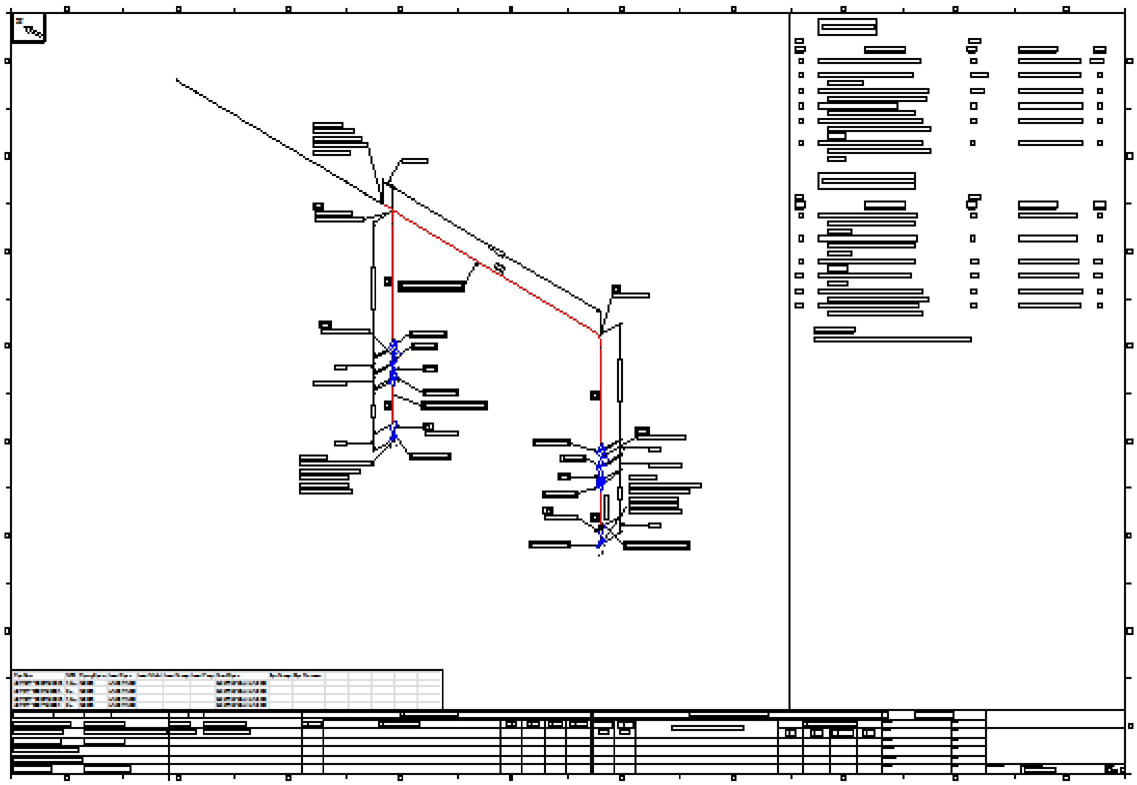


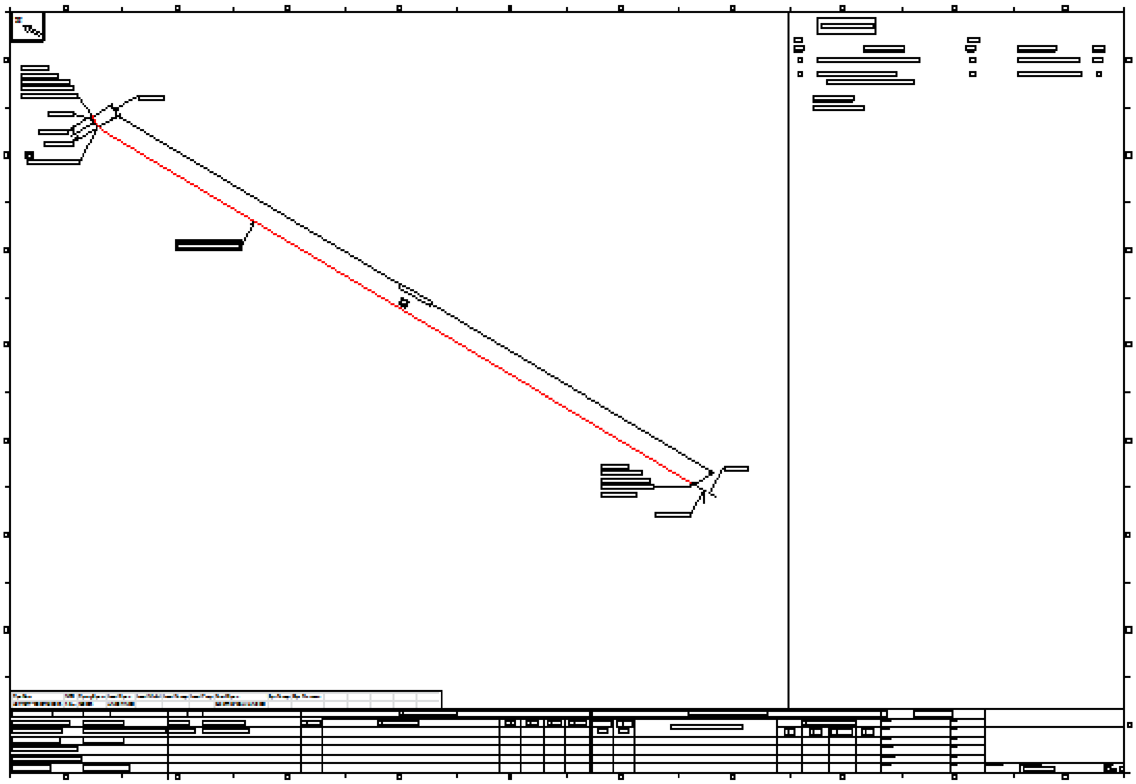
7. Click **Finish** to execute the command and apply the options set for renaming the pipe welds.

Creating Isometric Drawings by Automatically Assigning Objects to WBS Items

Objective

In this exercise you will be creating isometric drawings for the pipe parts belonging to the pipeline **1001-P** in **Unit U01** of your workspace by automatically assigning piping parts to the created WBS items. After creating the isometric drawing for the created WBS items the view will resemble these.





Before beginning the procedure:

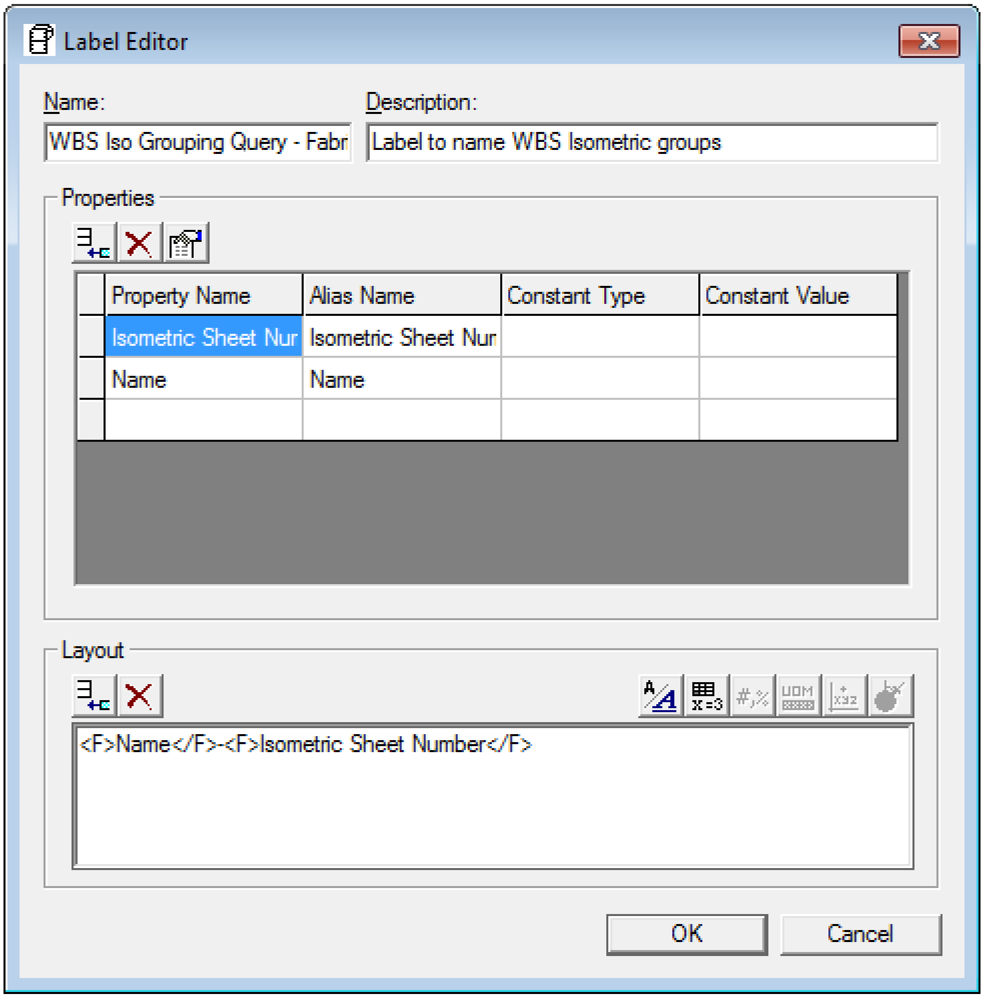
- Define your workspace to display **Unit U01**, coordinate system **U01 CS**, and **Projects** in the WBS hierarchy. In your training plant, select **U01** and **WBS Items** compound filter from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and **Active Permission Group** is set to **Piping**.

Before beginning the procedure for creating isometric drawing, you will first claim and manually assign piping objects to an active project. You claim all the objects to a WBS project before you run the **Group Pipe Parts** command on the objects to automatically assign them to create WBS item.

Automatically Assigning Piping Objects to the Created WBS Items:

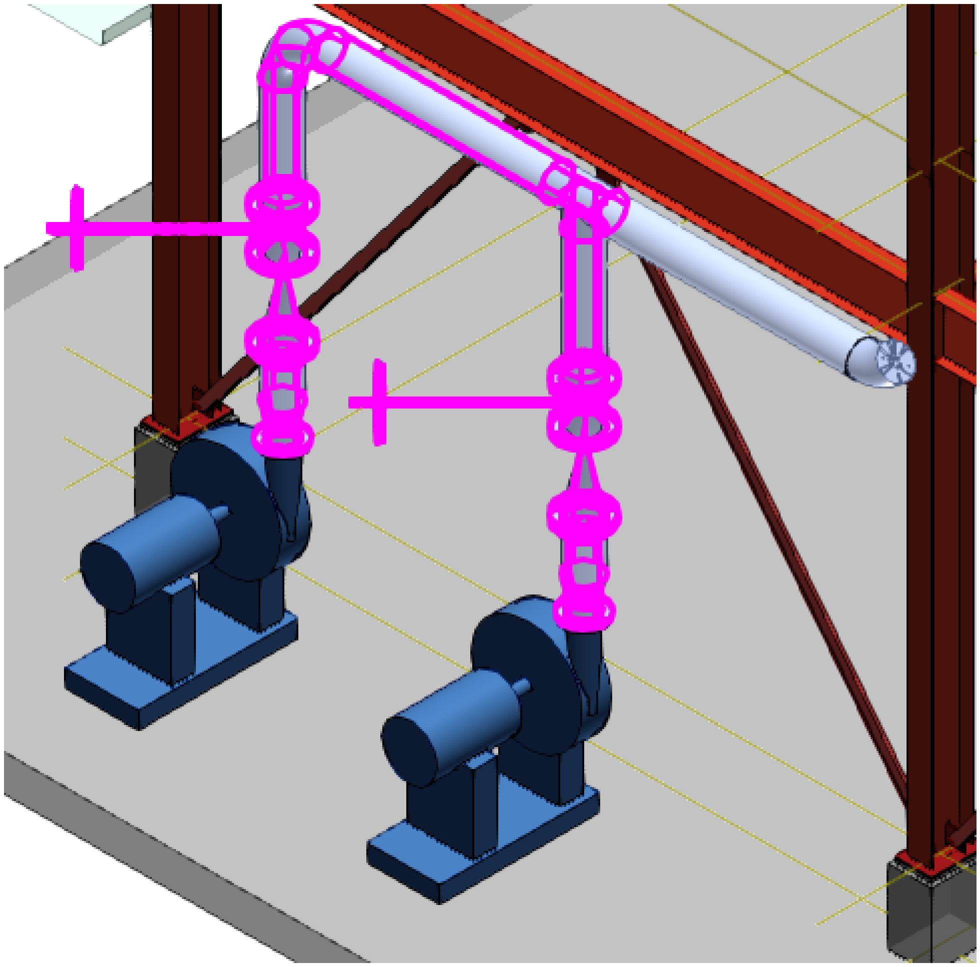
The **Group Pipe Parts** command on the vertical toolbar is used to create WBS items, name them and assign a group of piping parts to the created WBS items. The created WBS items are then used to create isometric drawings. Grouping the piping parts makes it easy to define the objects that you want to include in an isometric drawing. You need to group the piping parts into discrete sets by using their properties in a label definition query. In this case, the discrete sets are the WBS items.

The figure below shows the label used in this session for the query upon which the grouping of objects will be based.



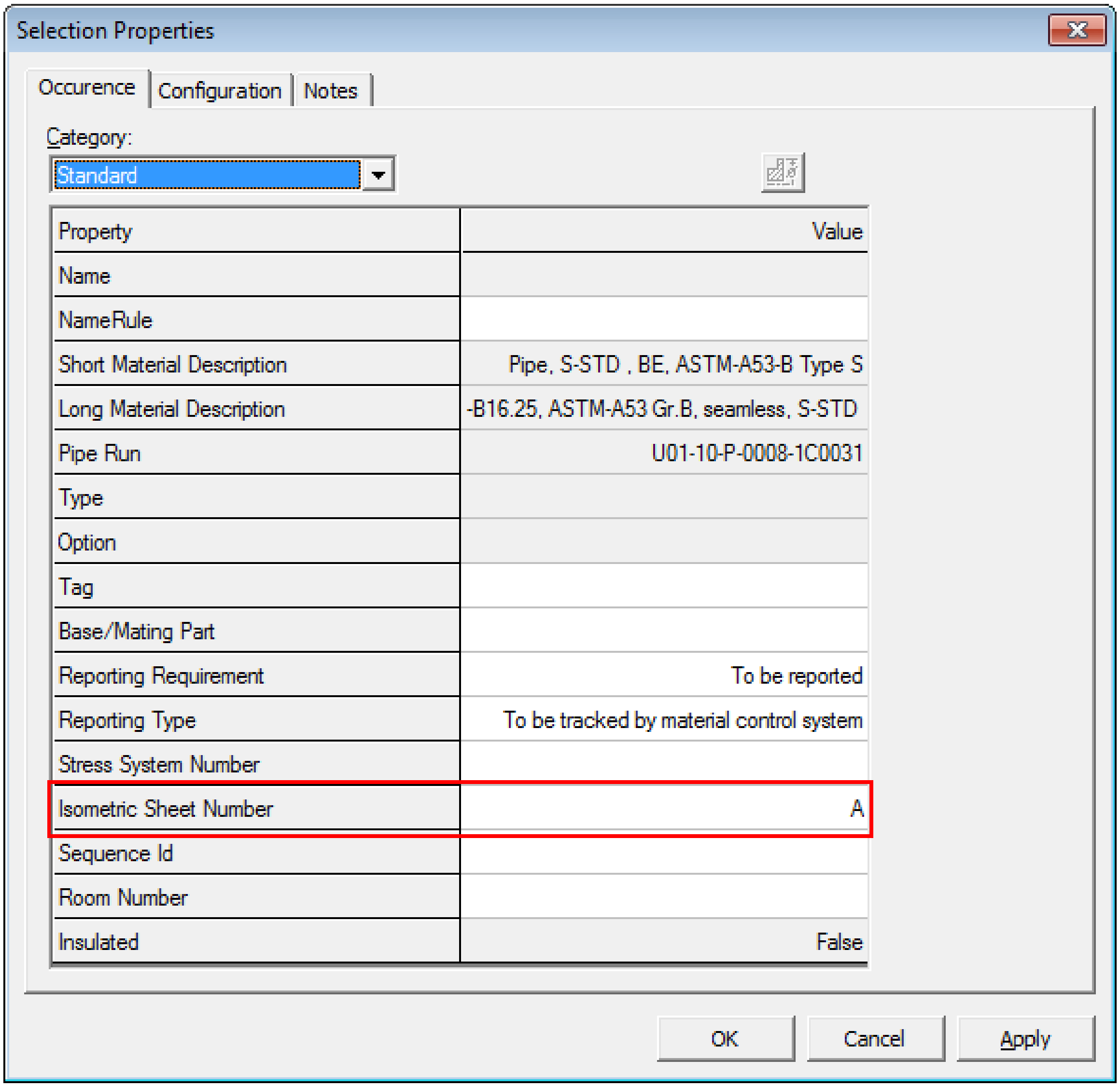
Name: Pipeline attribute
Isometric Sheet Number: Pipe Part attribute

1. Select the piping parts of the pipeline 1001-P to assign them to isometric sheet number A.

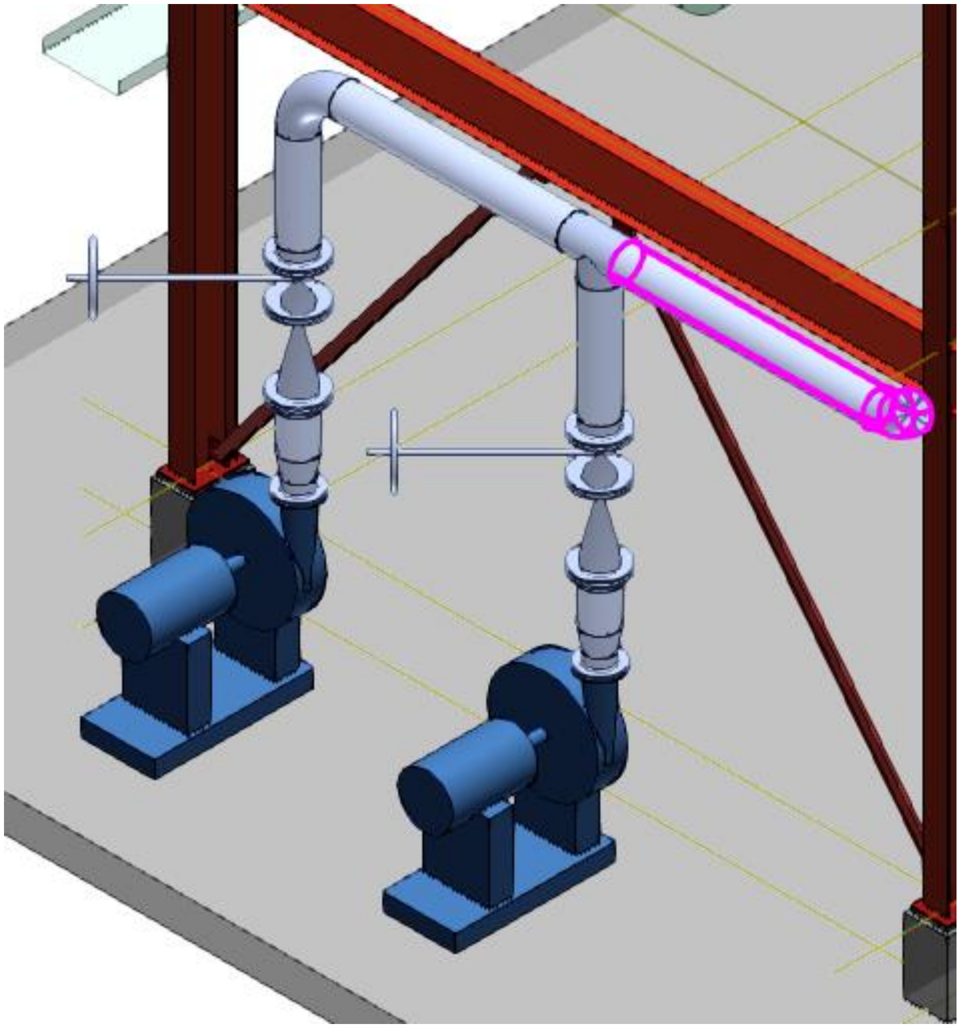


2. Right-click the selected parts and open the properties dialog box to assign the selected piping parts to an isometric sheet number A for creating an isometric drawing.

3. Key in **A** in the **Isometric Sheet Number** box in the **Selection Properties** dialog box, and click **OK**.

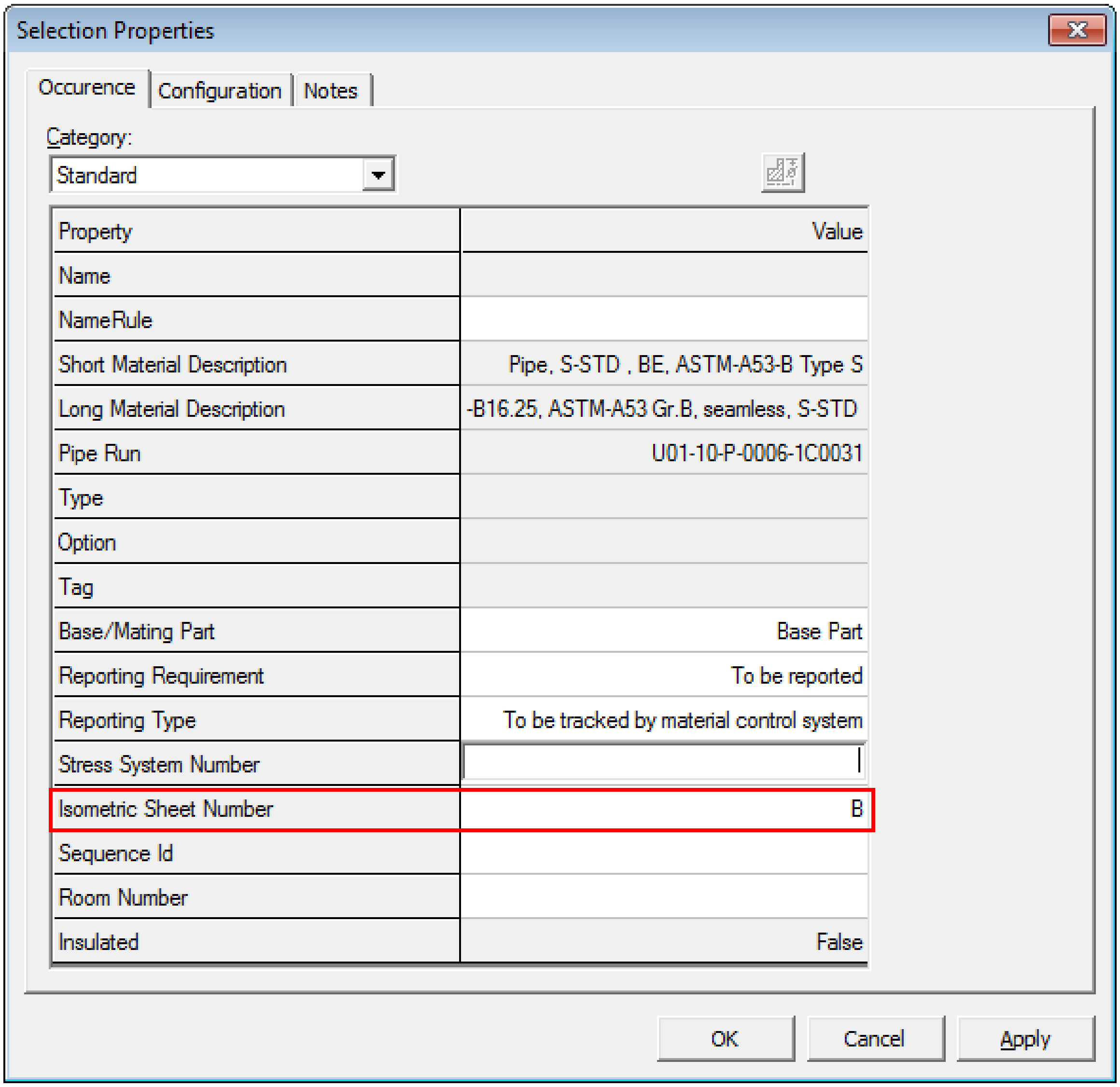


4. Now select the remaining pipe parts belonging to 1001-P.



5. Right-click the selected pipe parts and open the **Selection Properties** dialog box to assign them to another isometric sheet number B for creating an isometric drawing.

6. Key in **B** in the **Isometric Sheet Number** box in the **Selection Properties** dialog box, and click **OK**.



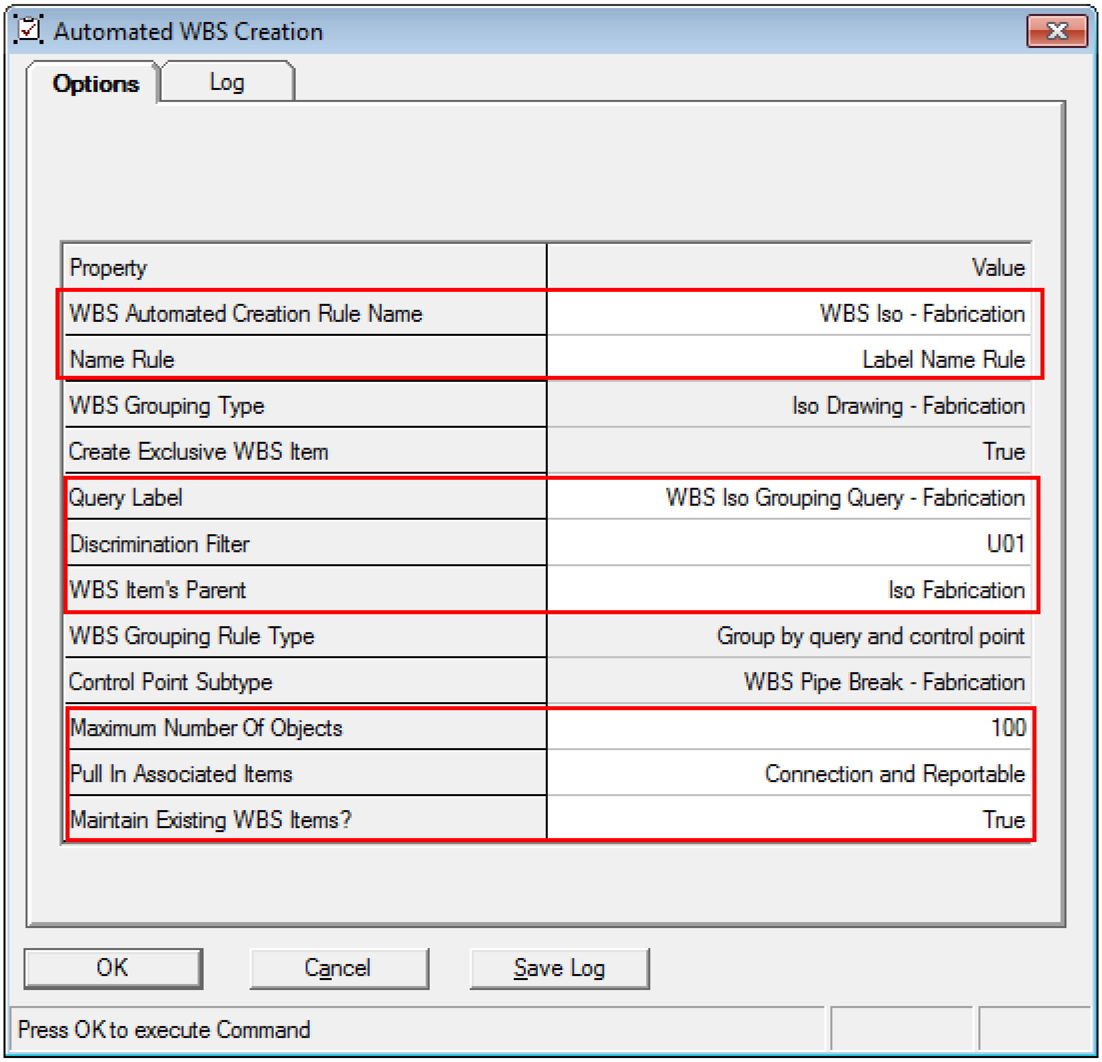
7. Now to assign these pipe parts to the appropriate WBS items, select **Group Pipe Parts**  on the vertical toolbar.

The Automated WBS Creation dialog box appears.

1. Set the following specifications in this dialog box, and click **OK**.

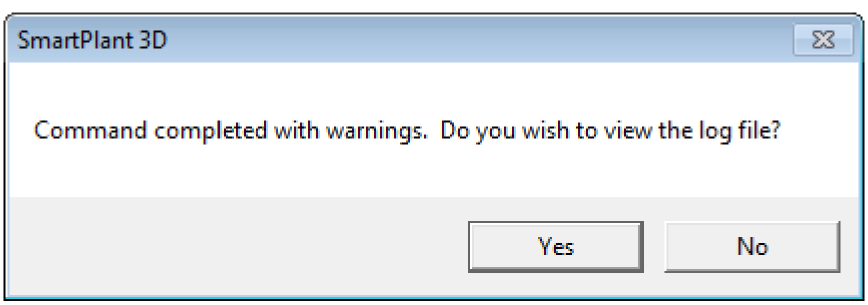
WBS Automated Creation Rule Name: WBS Iso- Fabrication
Name Rule: Label Name Rule
Query Label: WBS Iso Grouping Query- Fabrication
Discrimination Filter: Plant Filters > Training Filters > U01
WBS Item's Parent: PJ-99> Iso Fabrication
Maximum Number Of Objects: 100

Pull In Associated Items: Connection and Reportable
Maintain Existing WBS Items?: True



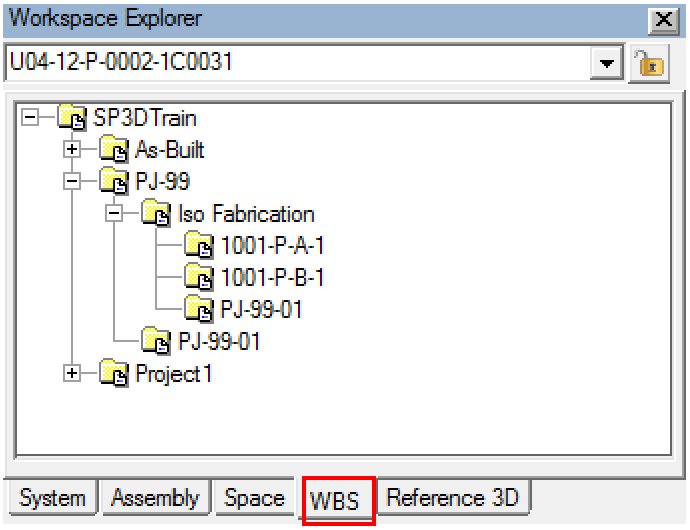
Smart 3D prompts you to view the log file after the process is complete.

1. Click **No** to close the log dialog box, and click **Cancel** on the **Automated WBS Creation** dialog box.



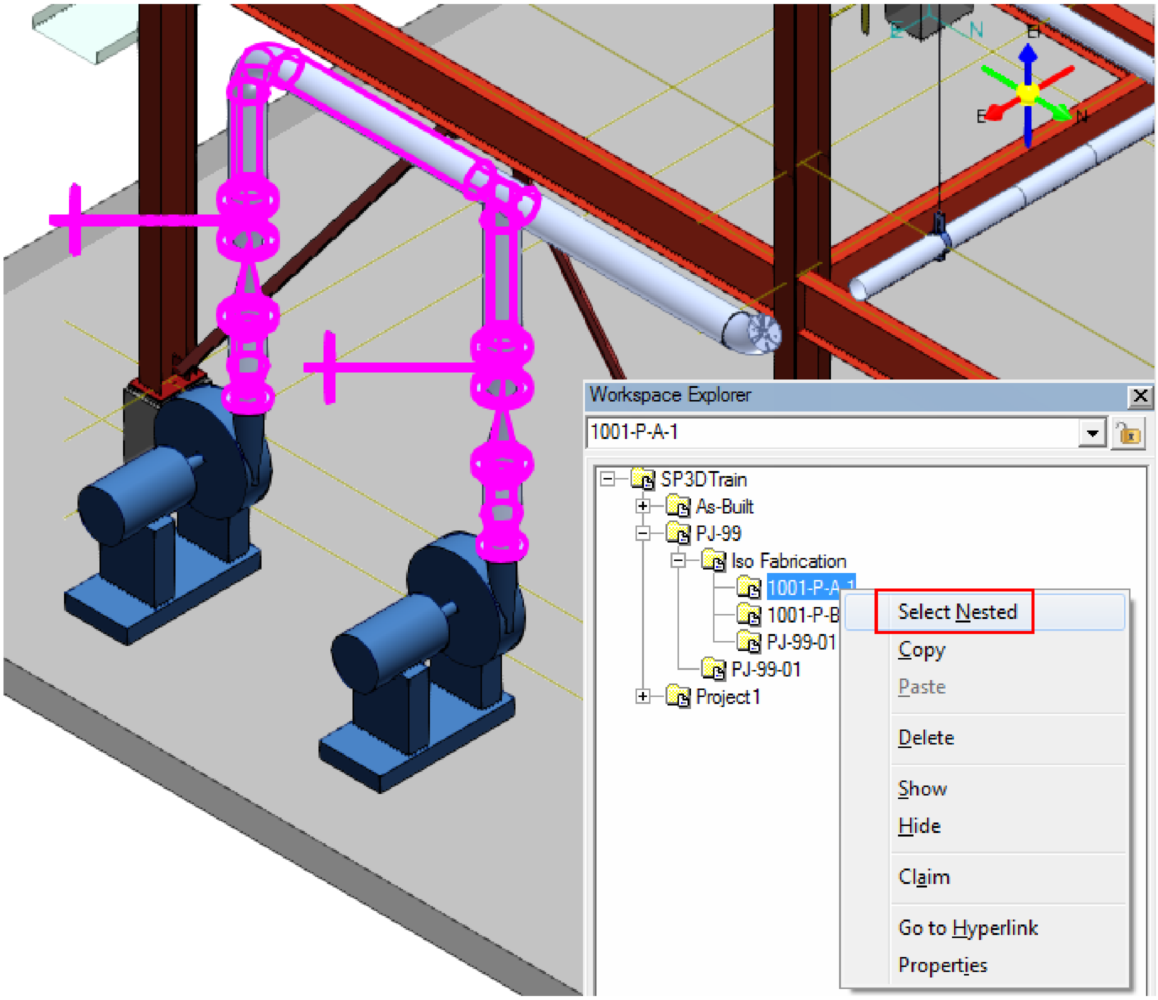
Two WBS items are created in the WBS hierarchy.

1. Select the **WBS** tab in the **Workspace Explorer** to view the **WBS item Iso Fabrication**.



The naming of the WBS item is based on the pipeline name (fluid type and sequence number) and isometric sheet number that we defined in the label. For example, in P-1001-A-1, P-101 is the pipeline name, A is the isometric sheet number, and 1 is the sequence number.

2. Right-click the WBS item **P-1001-A-1** in the **Workspace Explorer** and select **Select Nested** option to highlight all the pipe parts that belong to P-1001-A-1 WBS item.



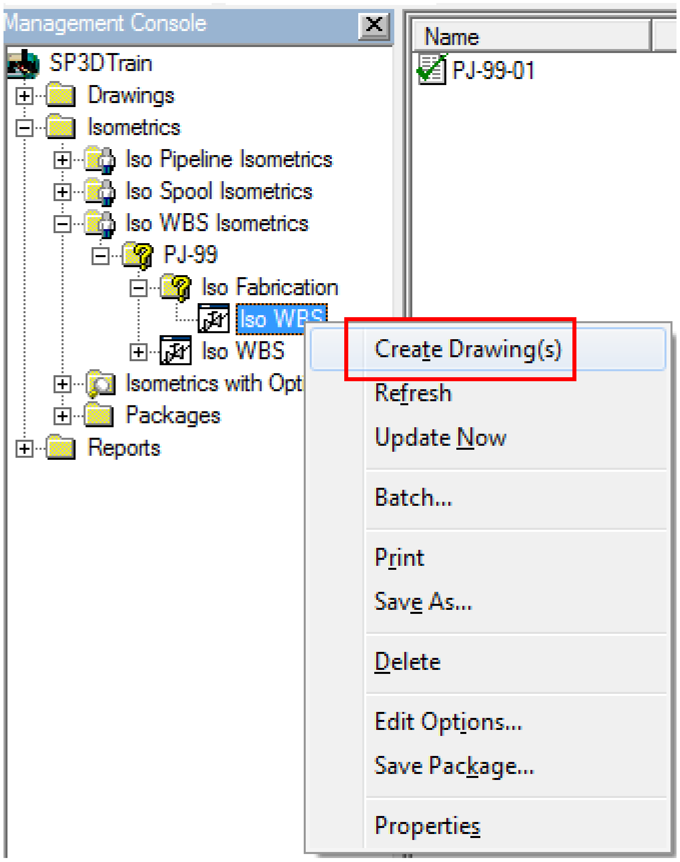
Creating Isometric Drawing from WBS items:

3. Switch to the **Drawings and Reports** task to create isometric drawings for the created WBS items.

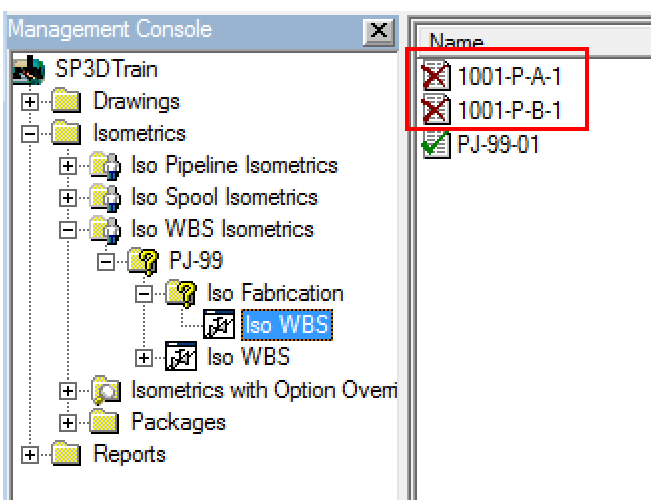
   The administrator should have already created **Drawings by Query Manager** components needed in the project. In this tutorial session, a **Drawings by Query Manager** component called **Iso WBS Isometrics** is created.

4. In the **Management Console** and expand the drawing hierarchy to **Isometrics > Iso WBS Isometrics**.

5. Now right-click **Iso WBS** in the hierarchy and click the **Create Drawings** option to create the isometric drawing documents.

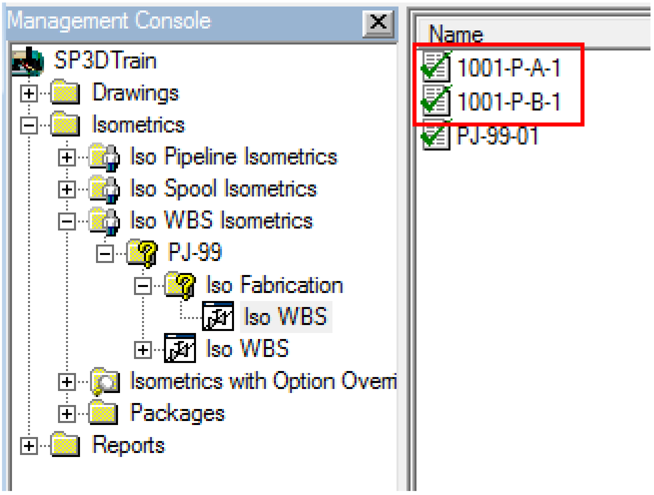


Smart 3D generates isometric drawing documents for all the WBS items available in the Iso Fabrication WBS parent.

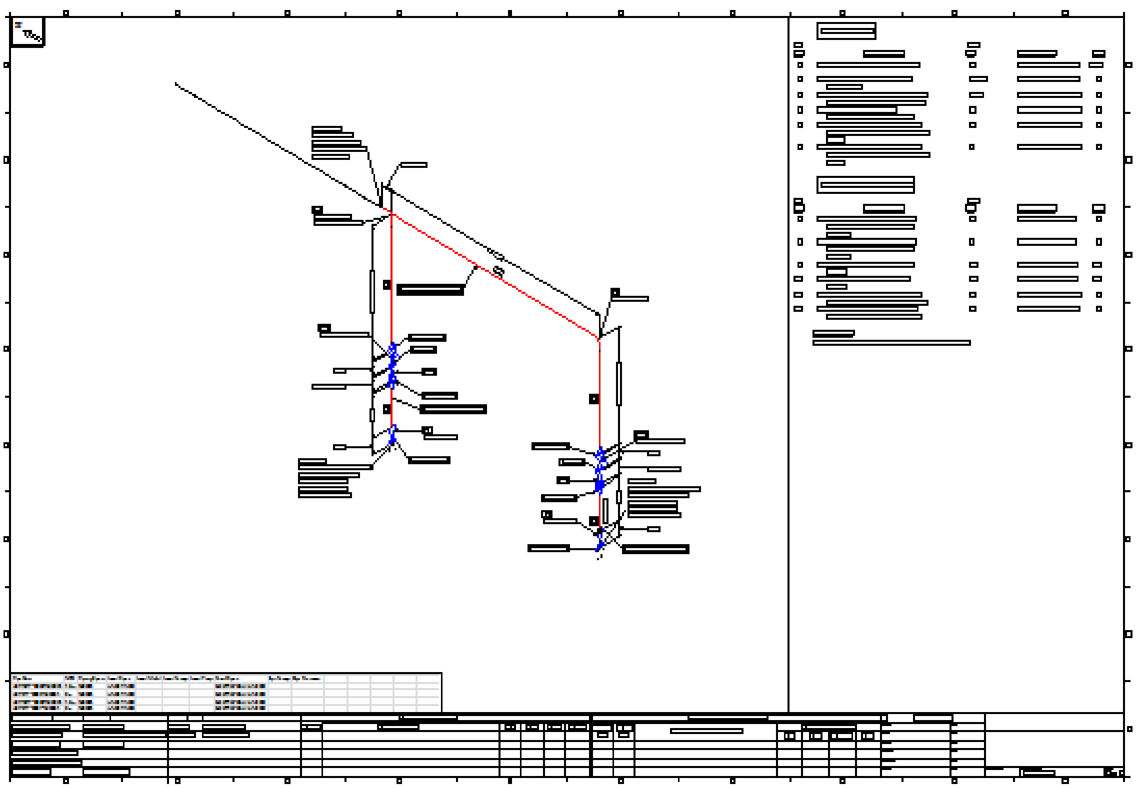


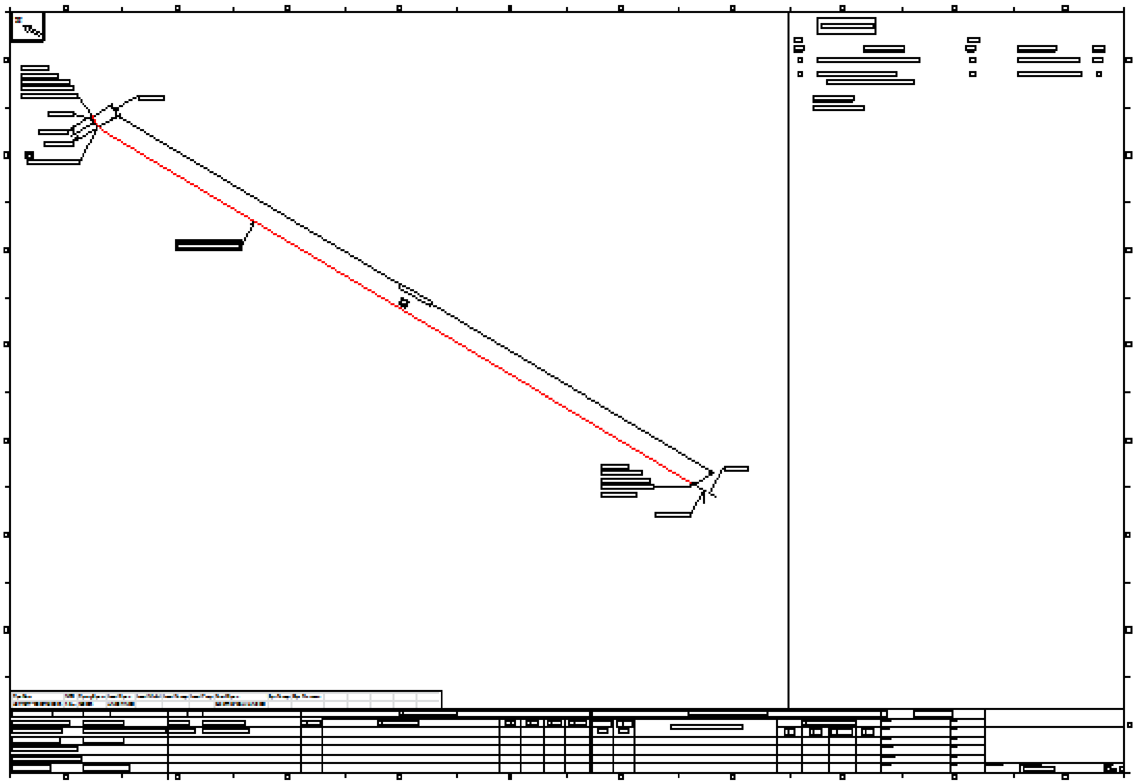
1. Right-click the **Iso WBS Package** isometric drawing type and click **Update Now** to update the drawings for both the WBS groups.

Smart 3D finds all the piping parts belonging to the created WBS items and generates the isometrics drawings for both the WBS groups 1001-P-A-1 and 1001-P-B-1. The green check mark then appears on both the groups in the Management Console that indicates that the isometric drawing is generated.



1. After the updation process is complete, double-click the isometric drawings created for the WBS items one by one. A pictorial representation of both isometrics is displayed.



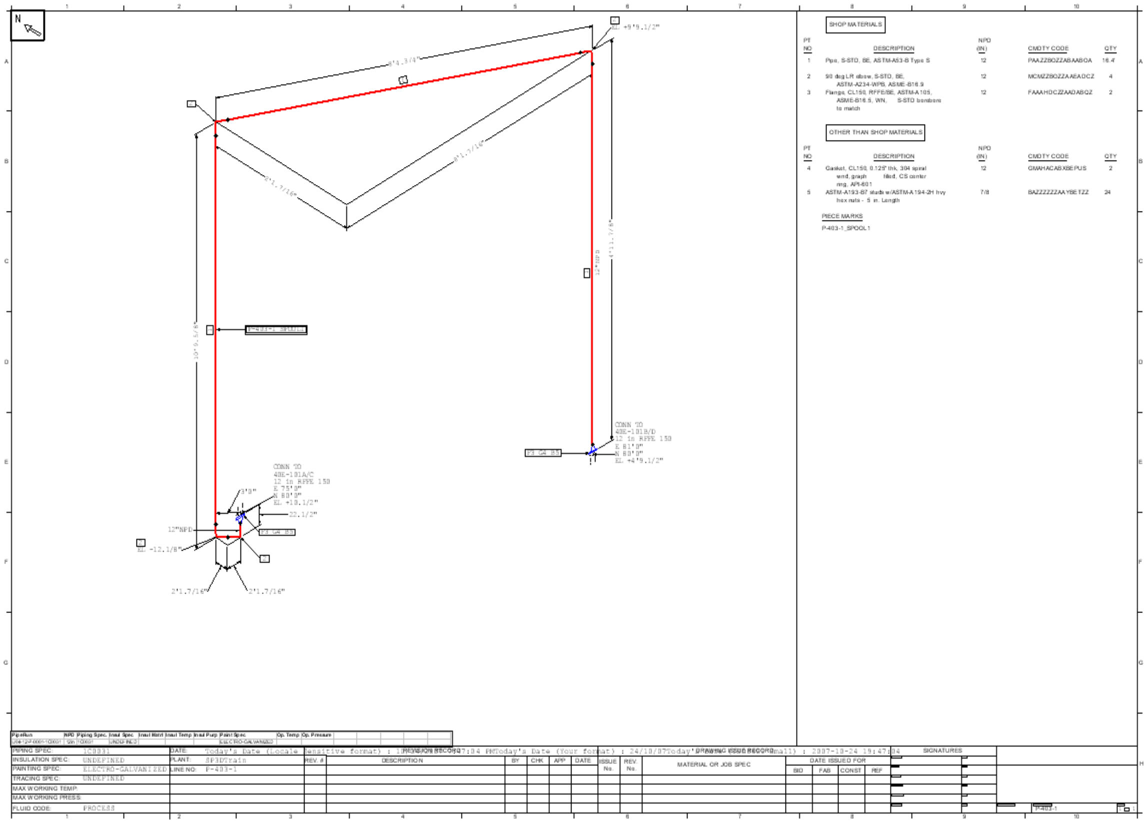


Creating Isometric Drawings from WBS Items

In Smart 3D you can create isometric drawings from WBS items by manually assigning piping objects to WBS items and by automatically assigning piping objects to WBS items. This will create one drawing per WBS items.

Objective

In the following exercise you will be creating isometric drawings for the pipeline 403-P in Unit U04 of your workspace by manually assigning piping parts to the created WBS items. After creating the isometric drawing for the created WBS items the view will resemble this.



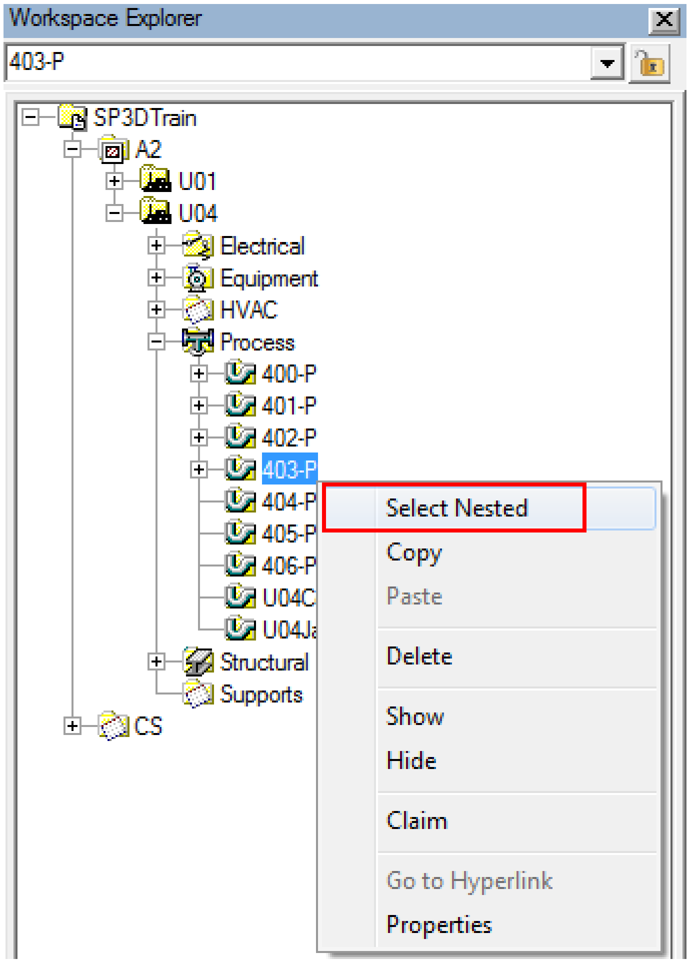
Before Starting this Procedure

- Define your workspace to display Unit U04, coordinate system U04 CS, and Projects in the WBS hierarchy. In your training plant, select U04 and WBS Items compound filter from Plant Filters > Training Filters in the Select Filter dialog box.
- Make sure you are in the Piping task and Active Permission Group is set to Piping.

Before beginning the procedure for creating isometric drawing, claim all the piping objects of a pipeline **403-P** of **Unit U04** to an active project **PJ-99** for assigning piping objects to active project. Then manually create a WBS item under the active project and assign all the piping objects to the created WBS item.

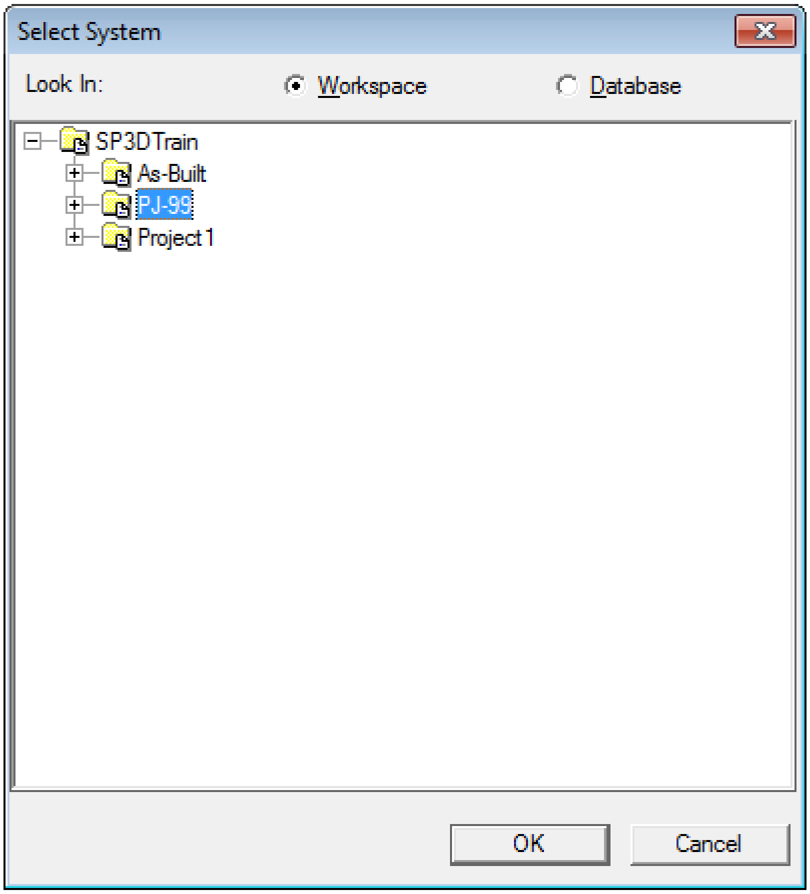
1. Select **All** in the **Locate Filter**.
2. In the **Workspace Explorer** window, expand the hierarchy **A2 > U04 > Process**.

3. Right-click the pipeline system **403-P** and click **Select Nested** in the menu to select all the piping objects in the graphic view.



4. Select **More…** in the **Active Project** drop-down list on the toolbar to specify the active project.

5. Select **Database** option to see all the WBS projects in the displayed dialog box and then select **PJ-99**, to specify PJ-99 as an active project.



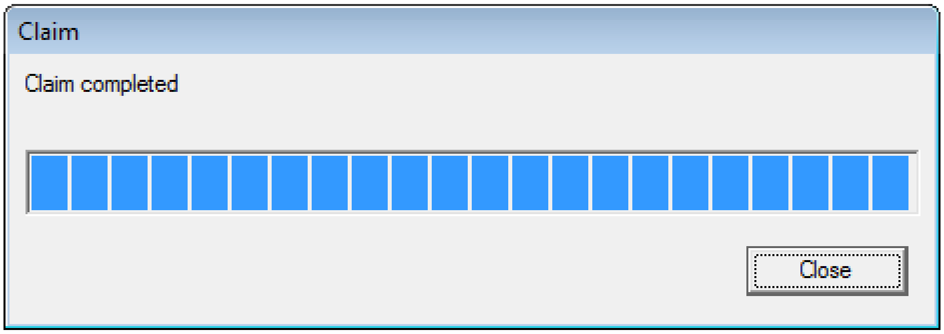
6. Click **OK**.

The selected active project should display.

1. Select **Project > Claim** to associate the selected piping objects with the active project PJ-99.

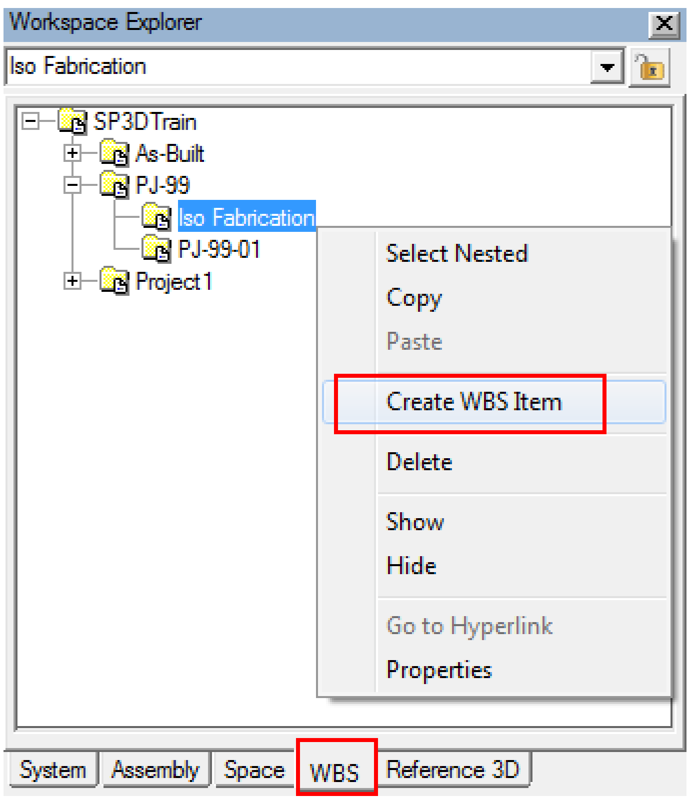
After the claim process is complete, Smart 3D displays the **Claim** dialog box.

1. Click **Close**.

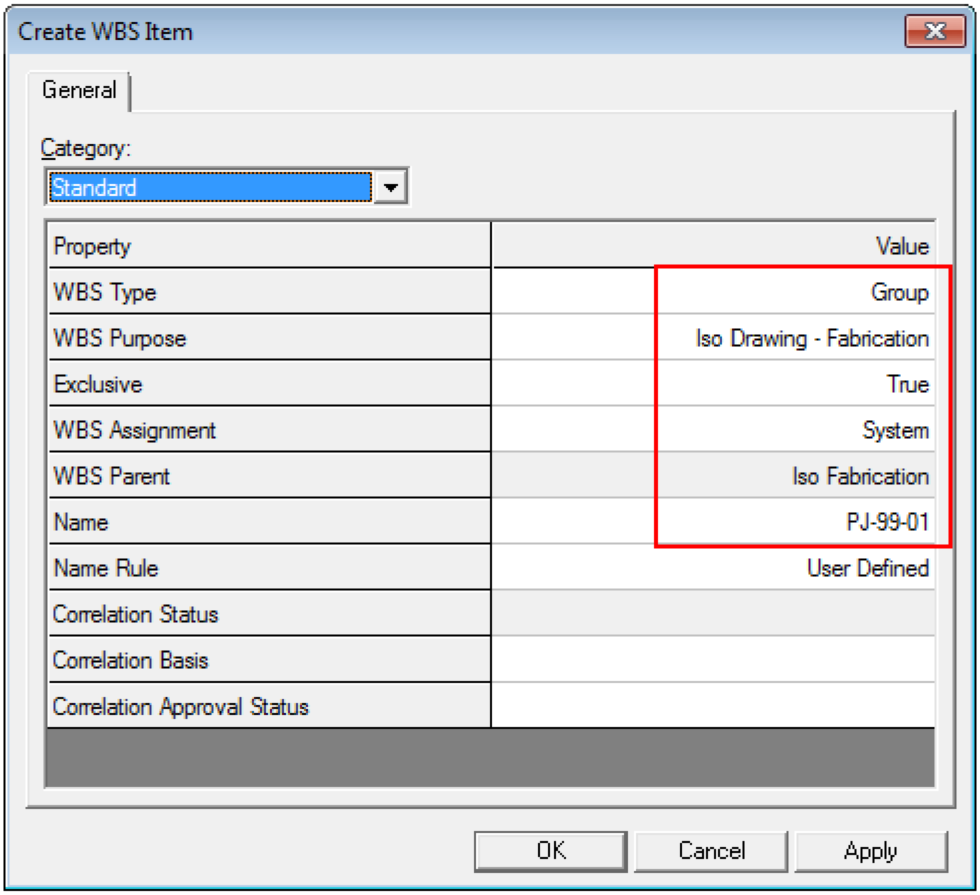


Manually Assigning Piping Objects to WBS item

2. In the **Workspace Explorer**, select the **WBS** tab and expand **PJ-99 > Iso Fabrication**.
3. Right-click the **Iso Fabrication** system in the **Workspace Explorer** and select the **Create WBS Item** command from the menu to create a new WBS item in the system Iso Fabrication.

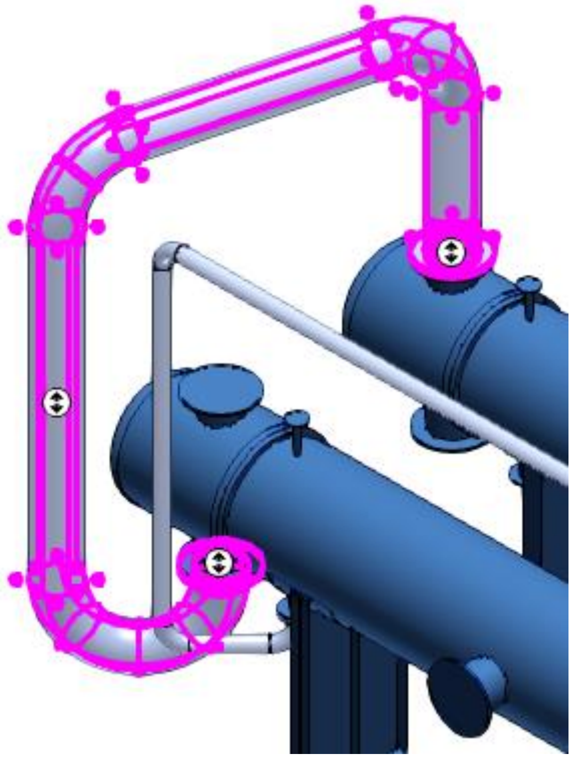


4. Select the property specifications, and click **OK**.



NOTE When the WBS Assignment property is set to System, then you can ensure that all the components of a given system will be assigned to the WBS item automatically.

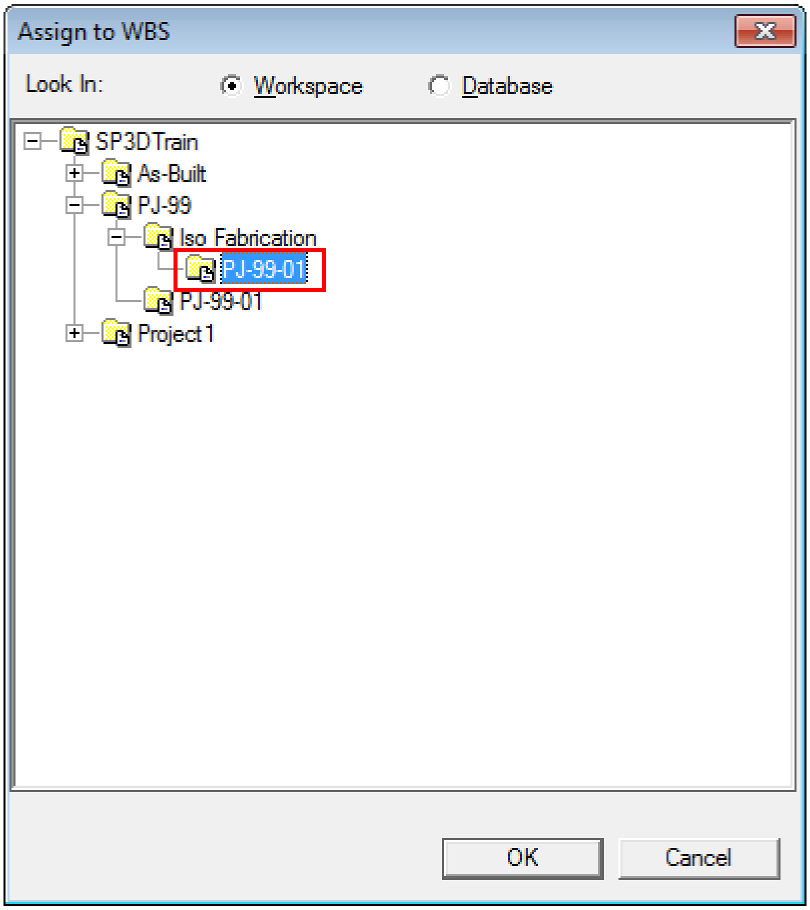
5. In the **Workspace Explorer** window, select the **System** tab to expand **A2 > U04 > Process** and select the pipeline system **403-P**. This will select the pipeline **403-P** in the graphic view.



6. Select **Project > Assign to WBS…** This command creates a relationship between all the components of the selected pipeline system and a selected WBS item.

The Assign to WBS dialog box appears.

1. In the **Assign to WBS** dialog box, expand **PJ-99 > Iso Fabrication** and select the WBS item **PJ-99-01** to which the piping objects need to be assigned. Then, click **OK**.



2. Select the **WBS** tab in the **Workspace Explorer**.
3. Expand **PJ-99**. Right-click **PJ-99-01** and click **Select Nested** in the menu.

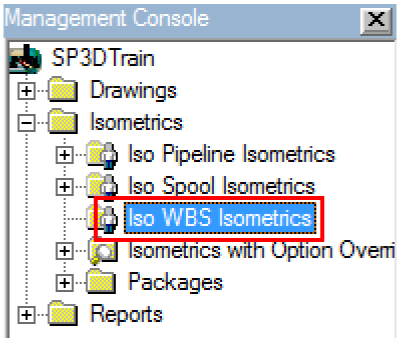
The highlighted objects in the graphic view are assigned to the PJ-99-01 WBS item.

Creating Isometric Drawing from WBS item:

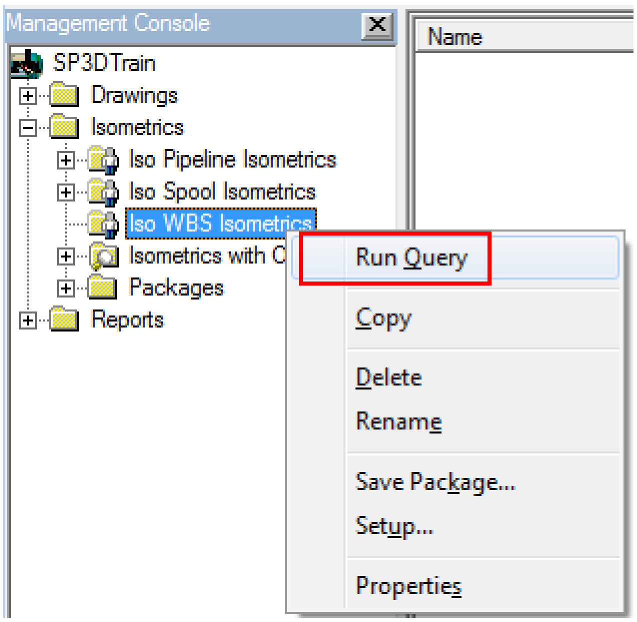
1. Switch to the **Drawings and Reports** task from the **Tasks > Drawings and Reports** command to create isometric drawings for the created WBS items.

   In this task, the **Management Console** appears in the left panel of the Smart 3D window. The Management Console has components to create isometric drawings. The administrator should have already created the Drawings by Query Manager components needed in the project. In this session, the Drawings by Query Manager component is called Iso WBS Isometrics.

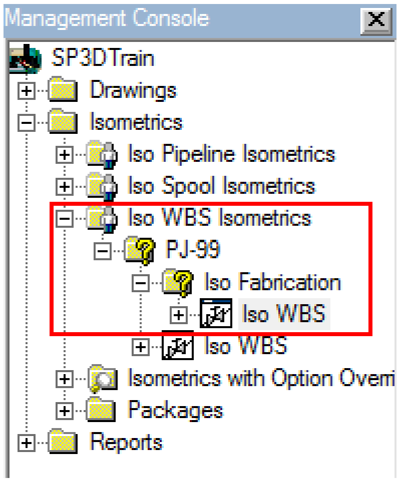
2. In the Management Console, expand the drawing hierarchy to **Unit 01 > Isometrics > Iso WBS Isometrics**.



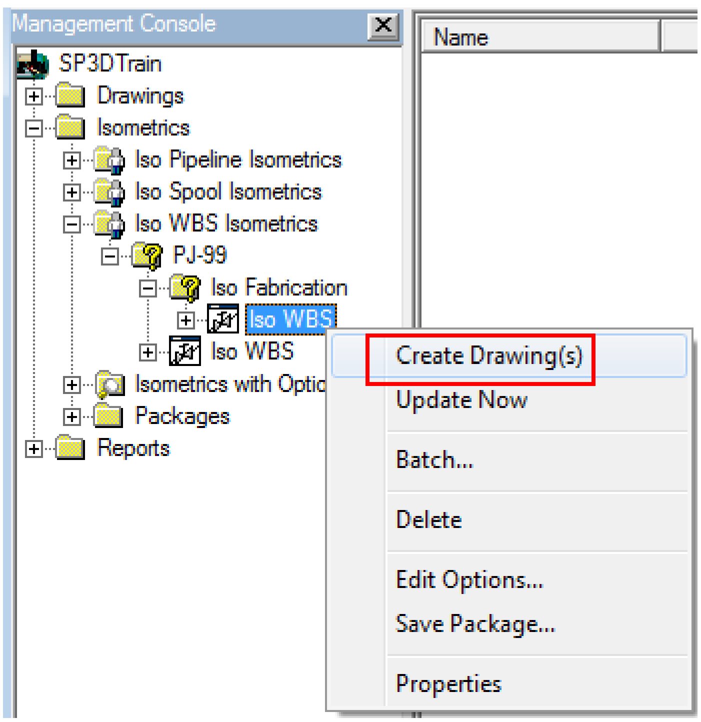
3. Right-click the **Iso WBS Isometrics** component in the **Management Console** and select **Run Query** option. Smart 3D uses the query defined in the Iso WBS Isometrics component to find the WBS items with Iso Fabrication in the model and displays the result of the search in the drawing hierarchy.



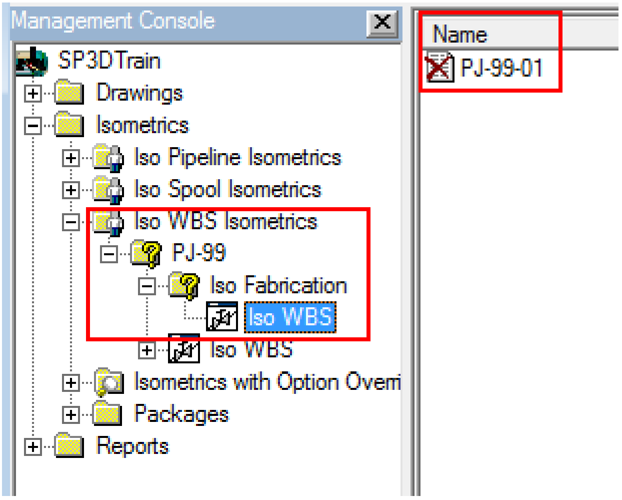
4. The active project **PJ-99** and WBS item Iso Fabrication appears in the hierarchy of the **Management Console**.



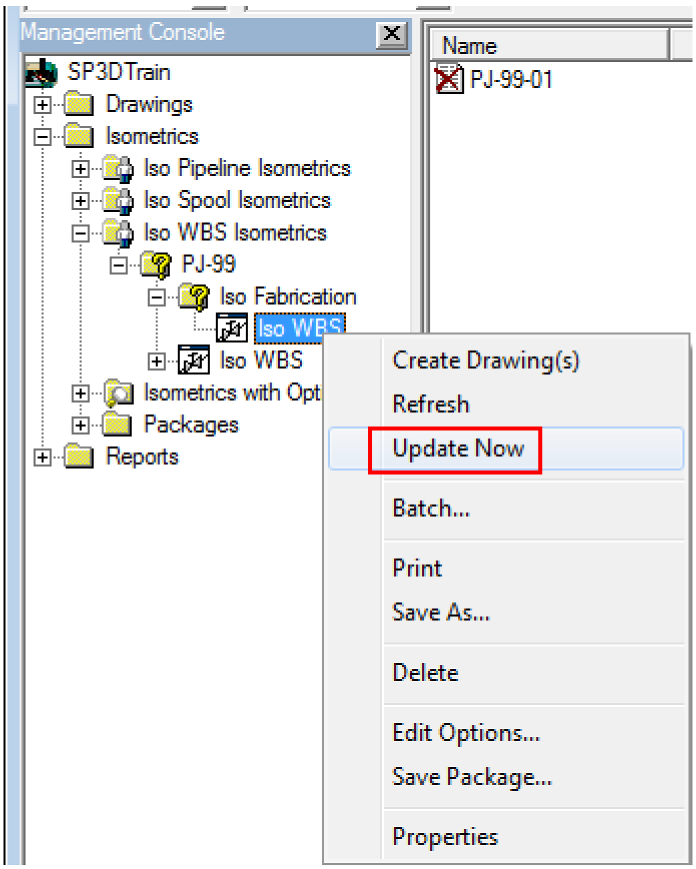
5. Now right-click the **Iso WBS** Package under **Iso WBS Isometrics > PJ-99 > Iso Fabrication** and click **Create Drawings** option to create the isometric drawing documents.



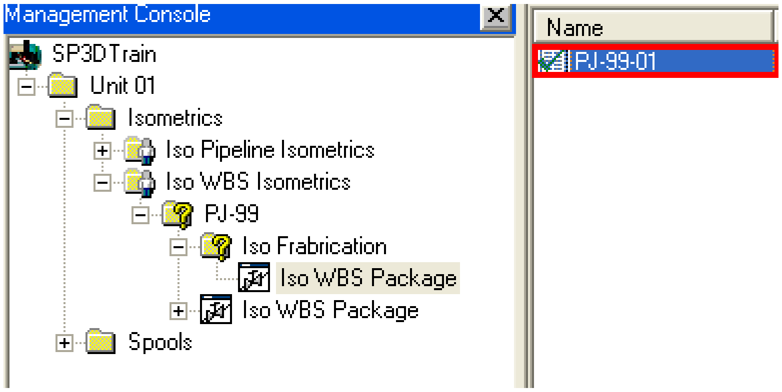
Smart 3D generates isometric drawing documents for all the WBS items available in the Iso Fabrication WBS parent.



1. Right-click the **Iso WBS Package** isometric drawing type and click **Update Now** to update the drawings for the WBS group.

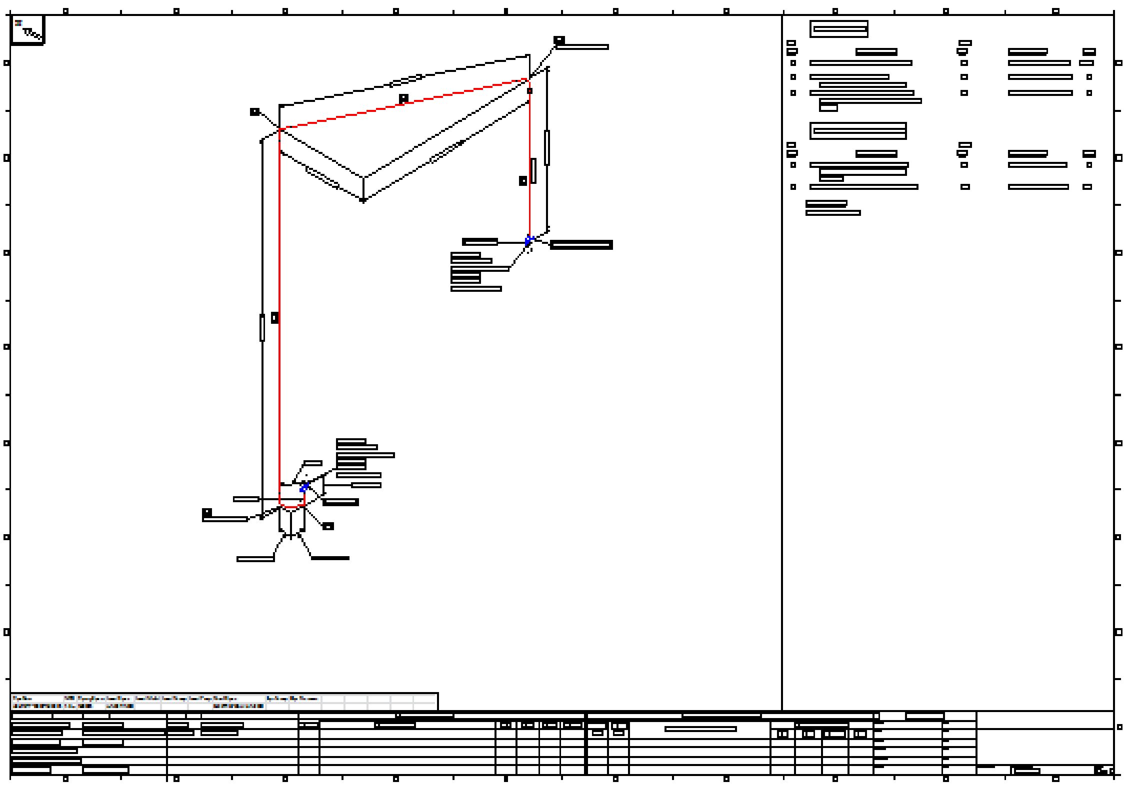


Smart 3D finds all the piping parts belonging to the created WBS items and generates the isometric drawings for the WBS group PJ-99-01. A green check mark appears on the PJ-99-01 group in the Management Console that indicates that the isometric drawing is generated.



1. After the update process is complete, double-click the isometric drawings created for the WBS item.

A pictorial representation of the isometric drawings will be displayed.

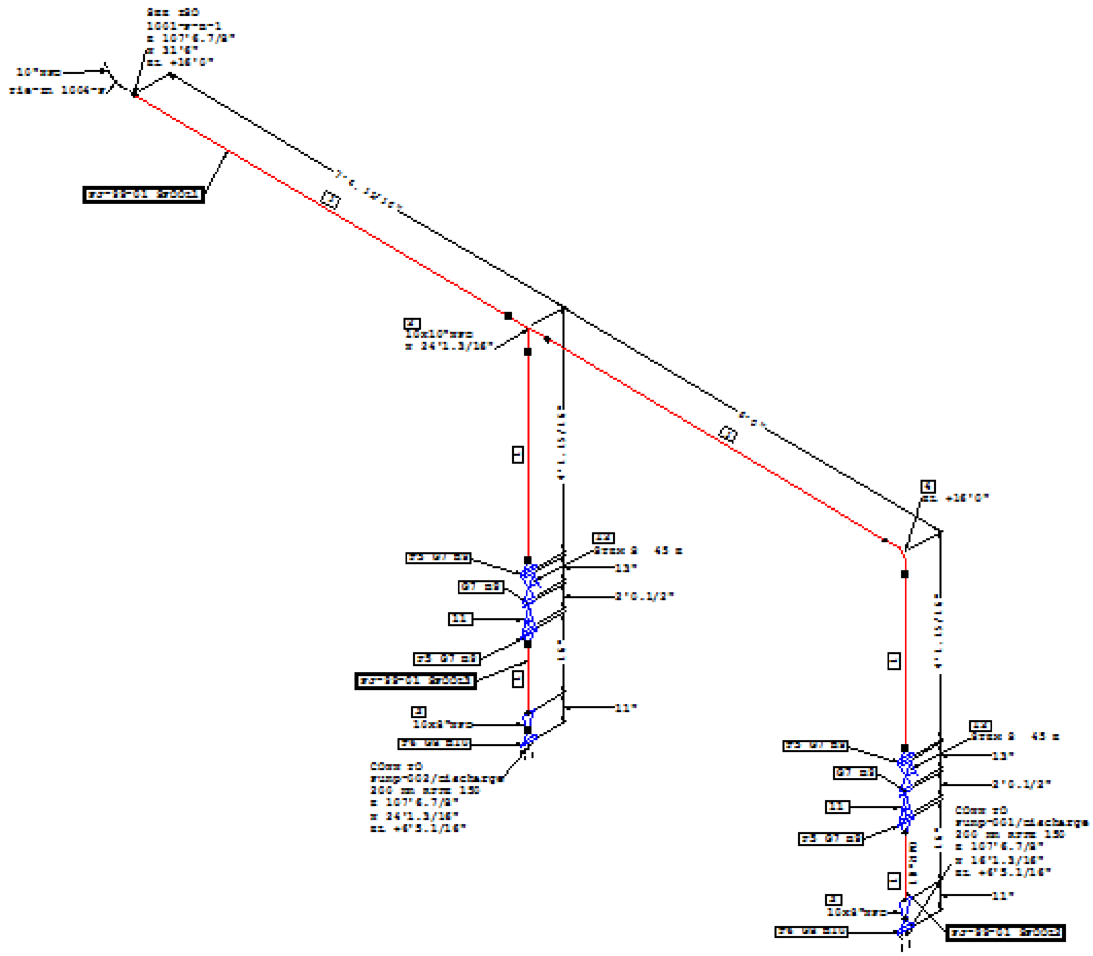


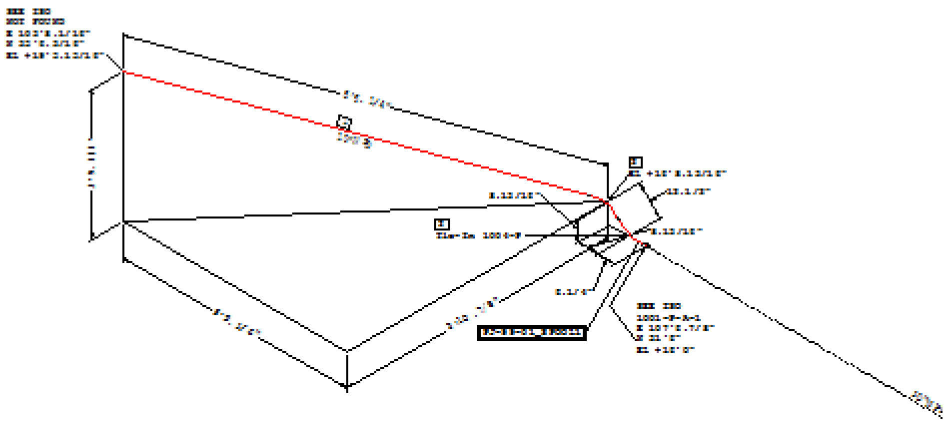
Updating Isometric Drawings

After the isometric drawings are created for a pipeline, you can add components and route any additional pipe in the pipeline 1001-P. When the structure of the pipeline changes, you need to update the WBS item and re-assign the objects once again to create the WBS item. After updating the WBS item you need to update the drawing for the pipeline as well.

Objective

In this exercise you will be placing a 45 deg elbow and route a 5 ft pipe from the free end of the pipeline 1001-P. After updating the pipeline, update the WBS item and then update the isometric drawing of the pipeline 1001-P.After updating the drawing, the view of the drawings should resemble this.

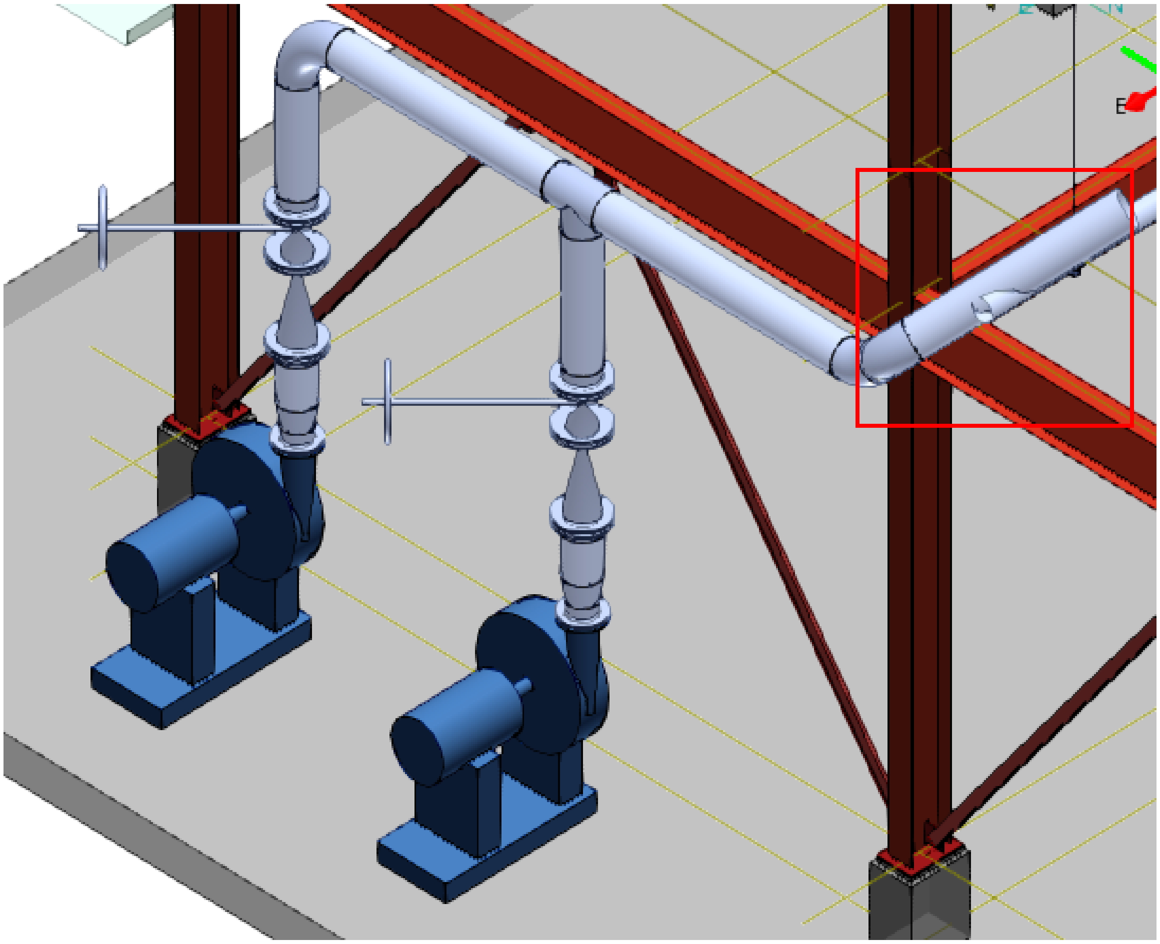




1. Select **Tasks > Piping** to switch to the **Piping** task to work in the piping environment.

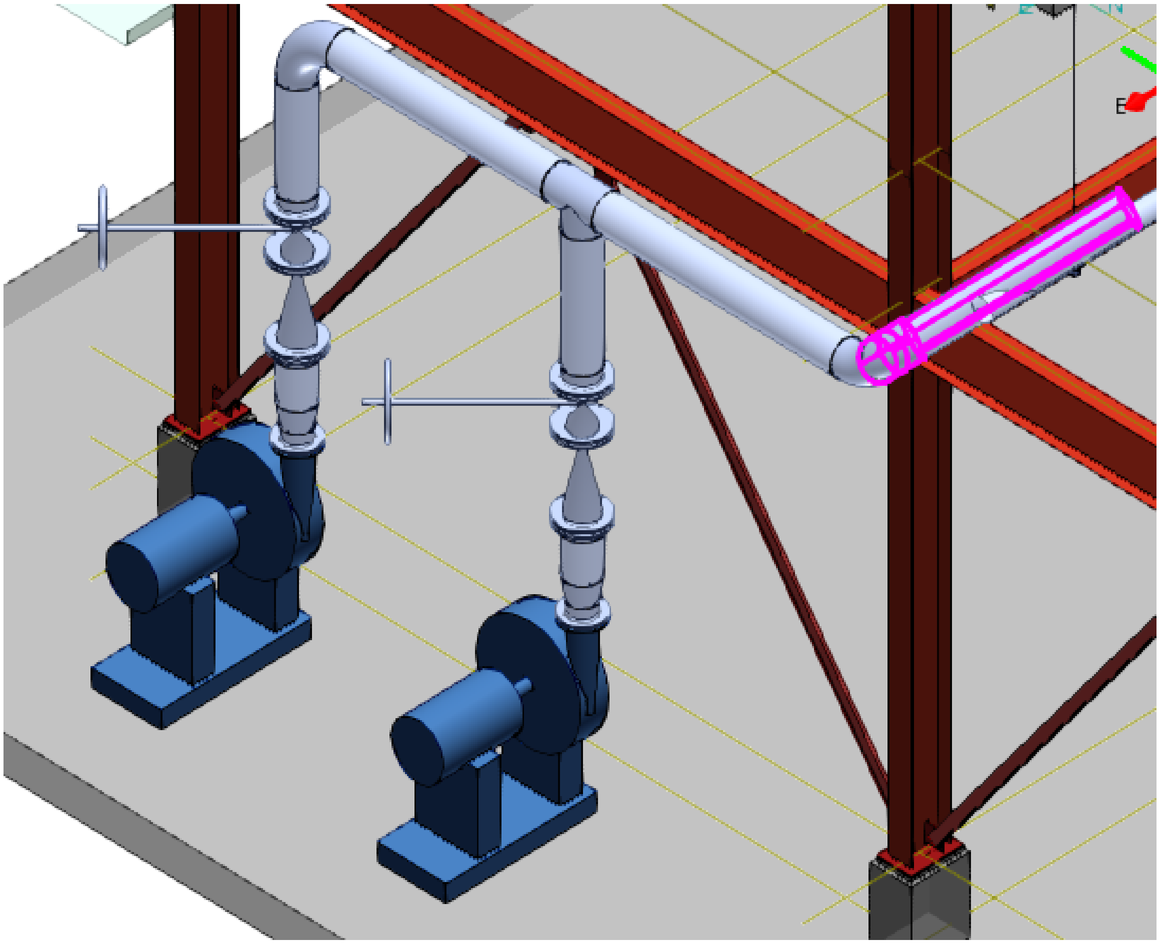
   Notice that the active WBS project is PJ-99. Therefore, when routing or inserting components, the new objects will be claimed to WBS project PJ-99 automatically.

2. Insert a 45 deg elbow and route a 5 ft pipe from the free end of the pipeline 1001-P in Unit U01 of your workspace by using the **Insert Component** and **Route Pipe** commands. After inserting the elbow and routing the pipe the view of the model should resemble this.

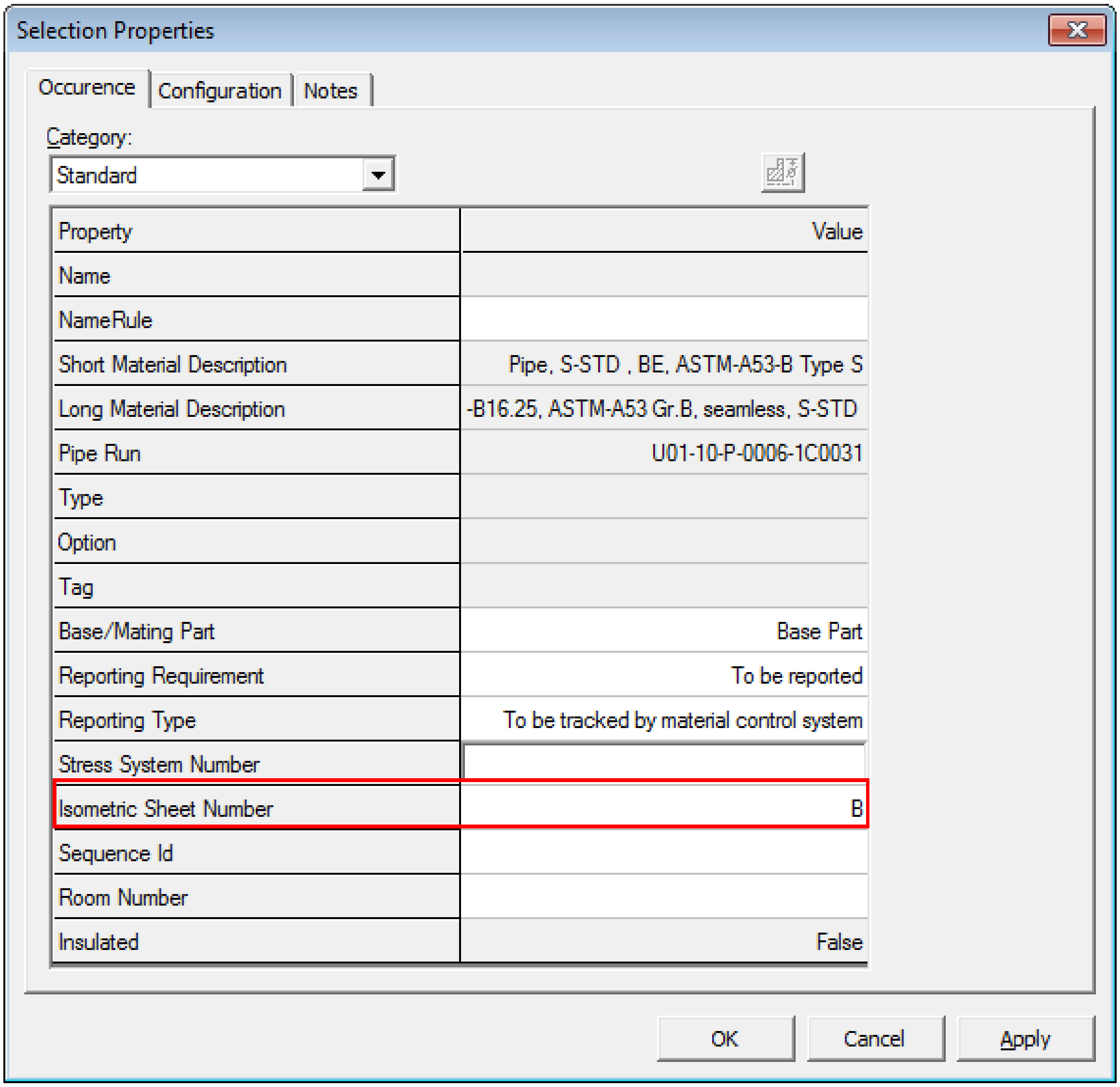


3. Make sure the **Locate Filter** is set to **Piping Parts**.

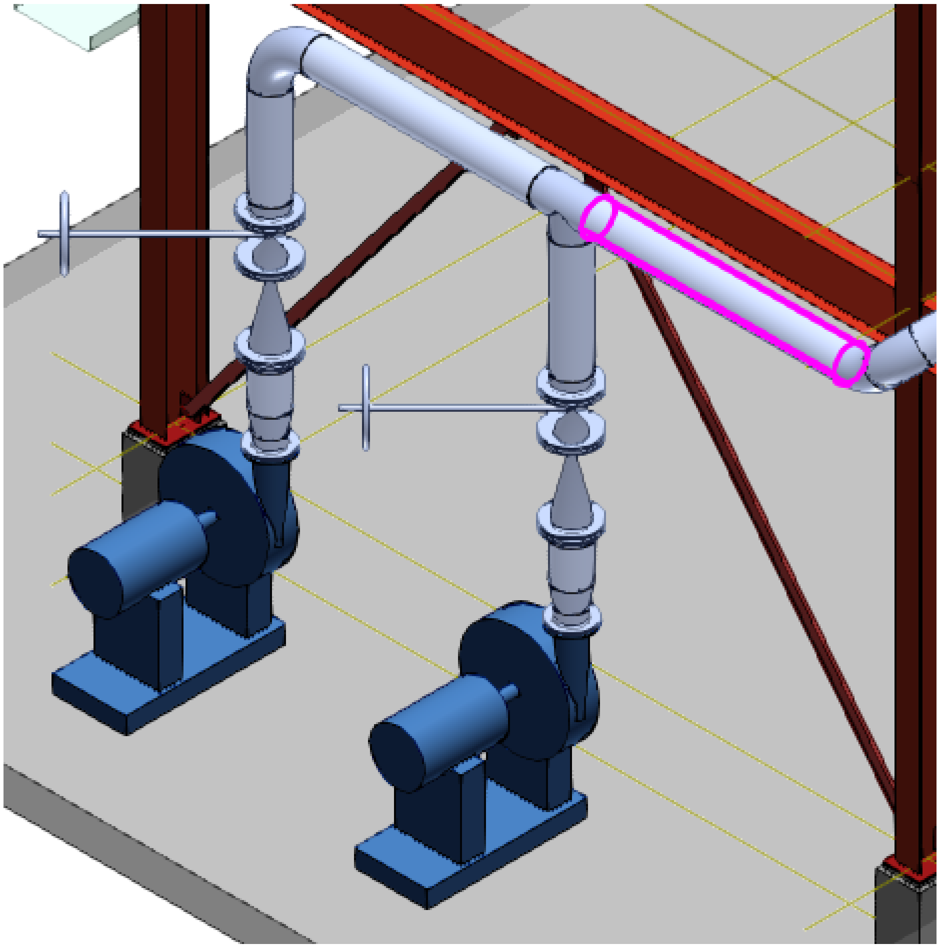
4. Select the 45 deg elbow and pipe you have placed.



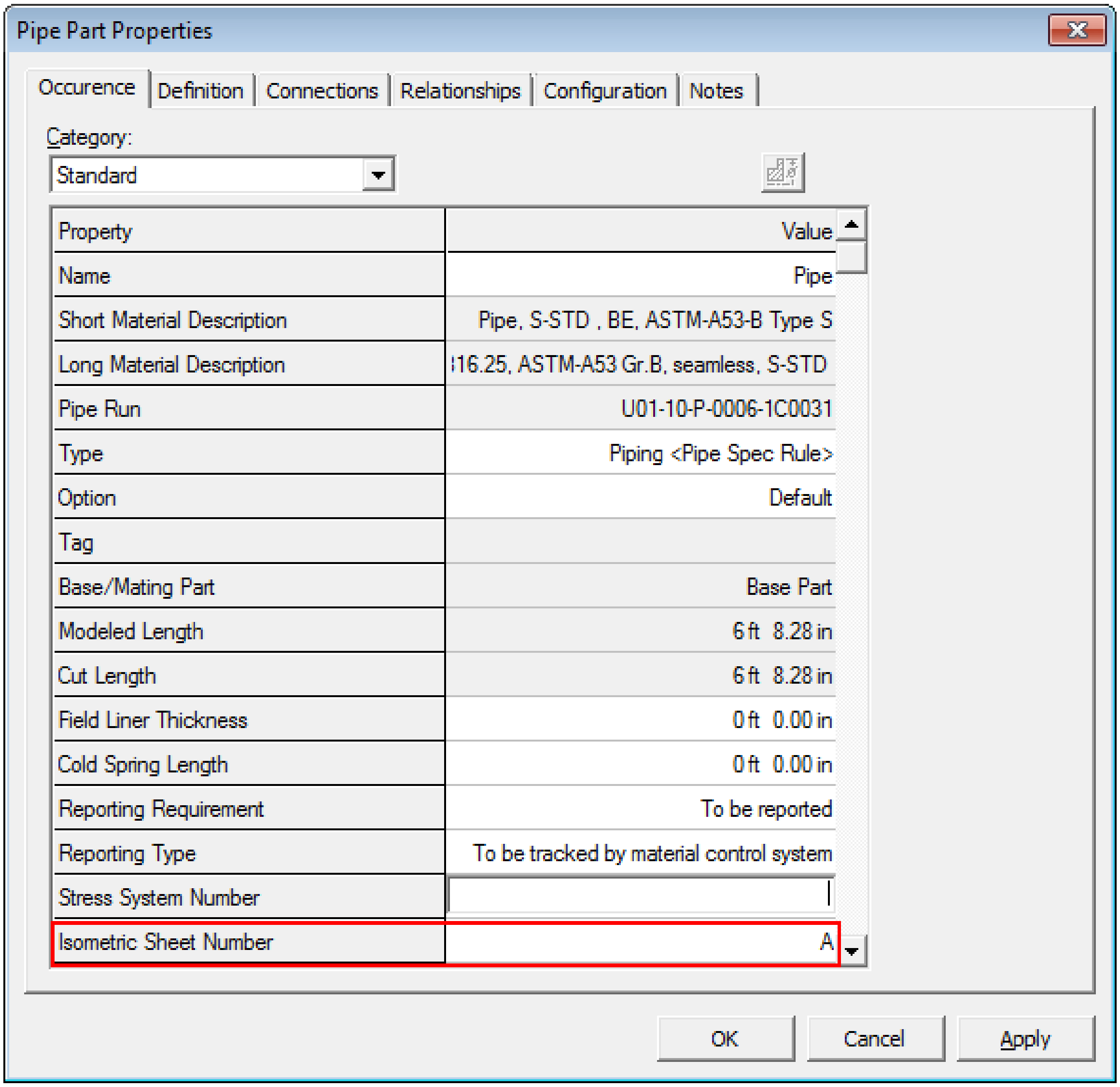
5. Right-click the selected pipe parts and open the **Selection Properties** dialog box to assign the pipe parts to the isometric sheet number B for creating an isometric drawing. Type **B** in the **Isometric Sheet Number** field in the **Selection Properties** dialog box, and click **OK**.



6. Select the pipe.



7. Right-click the selected pipe and open the properties dialog box re-assign it to isometric sheet number A for creating an isometric drawing. Type **A** in the **Isometric Sheet Number** box in the **Selection Properties** dialog box, and click **OK**.



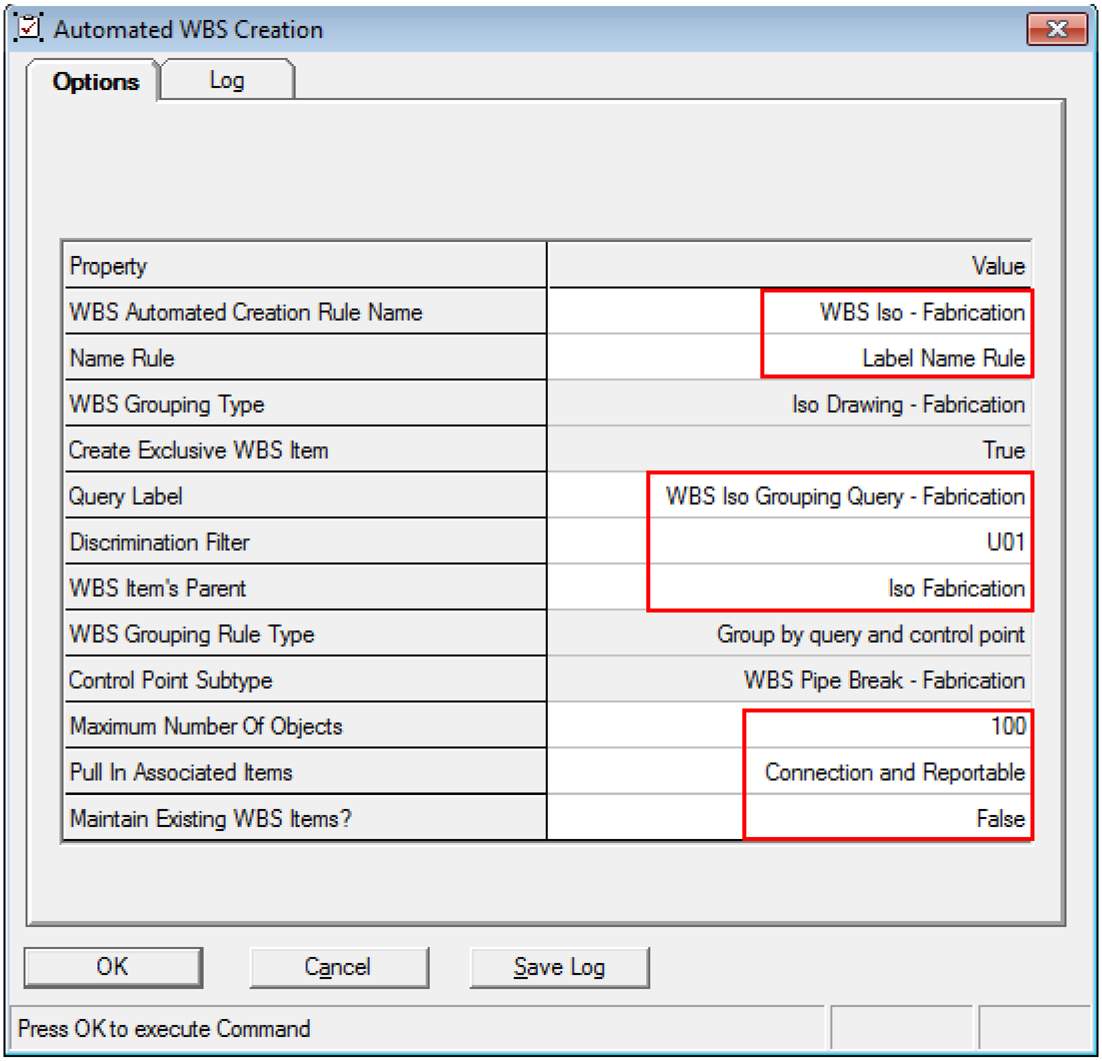
8. Now to assign these pipe parts to the appropriate WBS items, click **Group Pipe Parts** on the vertical toolbar.

The Automated WBS Creation dialog box appears.

1. Set the following specifications in this dialog box, and click **OK**.

WBS Automated Creation Rule Name: WBS Iso- Fabrication
Name Rule: Label Name Rule
Query Label: WBS Iso Grouping Query- Fabrication
Discrimination Filter: Plant Filters > Training Filters > U01
WBS Item's Parent: PJ-99 > Iso Fabrication
Maximum Number Of Objects: 100

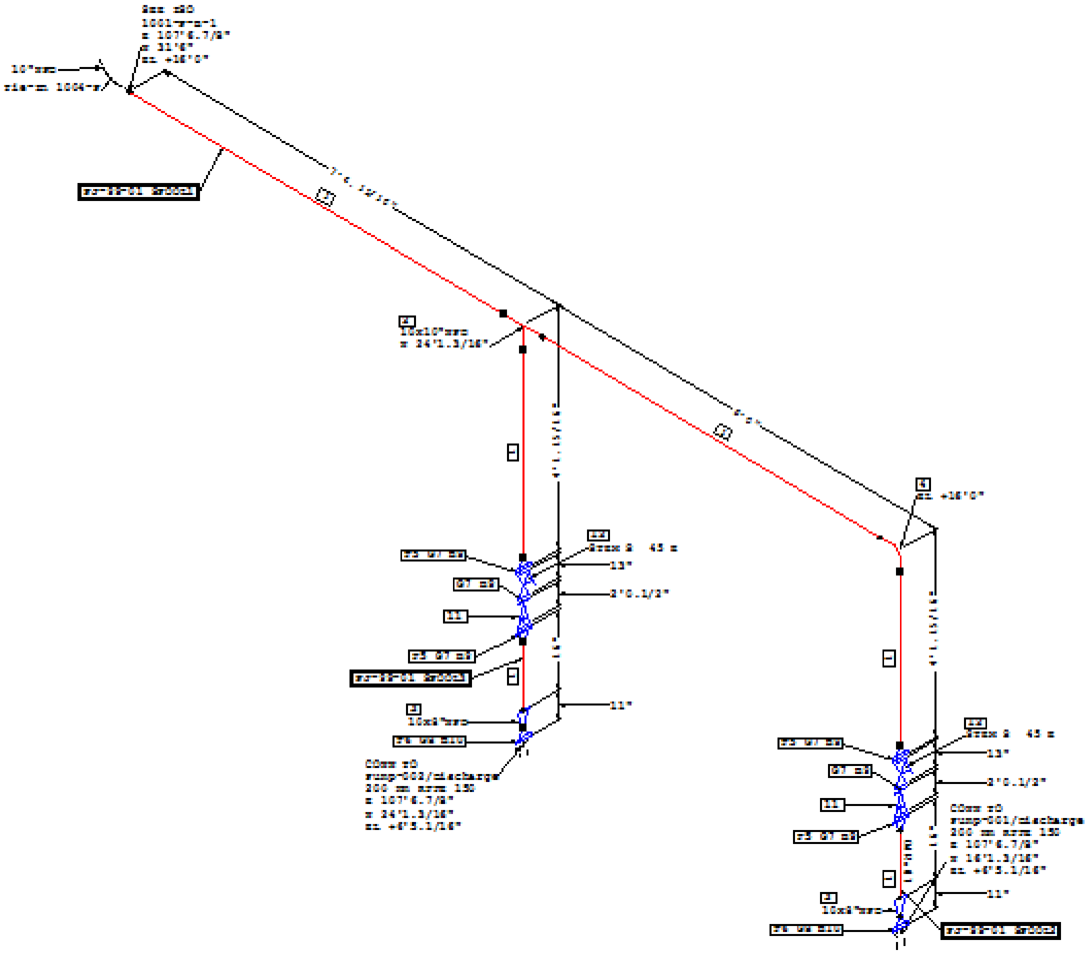
Pull In Associated Items: Connection and Reportable
Maintain Existing WBS Items?: False

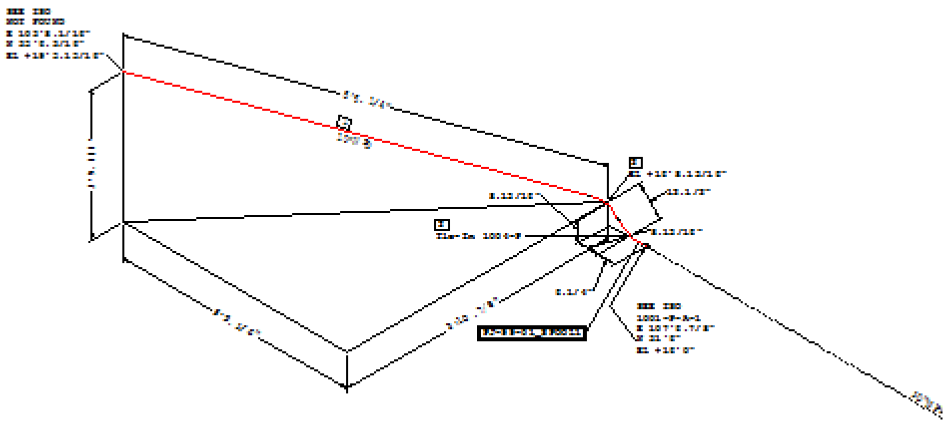


Smart 3D prompts you to view the log file after the process is complete.

1. Click **No** to close the log dialog box, and click **Cancel** on the **Automated WBS Creation** dialog box.
2. Switch to the **Drawings and Reports** task to update the isometric drawings for the created WBS items.
3. In the **Management Console**, expand the drawing hierarchy to **Isometrics > Iso WBS Isometrics > PJ-99 > Iso Fabrication > Iso WBS**
4. Right-click the Iso WBS isometric drawing type, and click **Update Now** to update the isometric drawings.

5. After the update is complete, double-click the isometric drawings created for the WBS items one-by-one. A pictorial representation of both isometrics will be displayed.





Changing the Status of WBS Items

The WBS item must be in the Working status so that users can assign objects to it. This prevents user from assigning objects within groupings that are already in the Approved status.

Objective

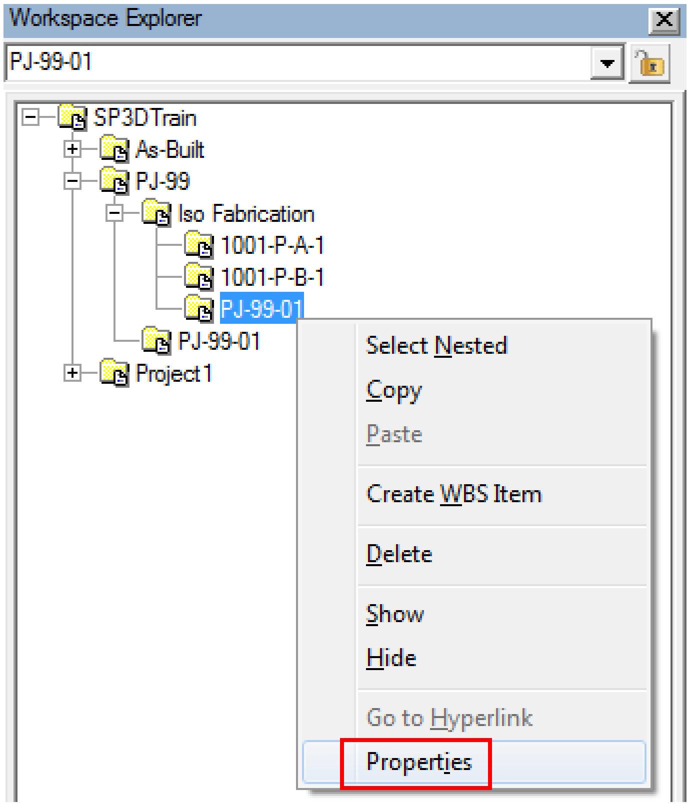
In this exercise you will be changing the status of the WBS item PJ-99-01to Approved. Then, you will place a temperature indicator **TI-504** on the pipeline **403-P** and assign all the piping objects of the pipeline **403-P** to the WBS item **PJ-99-01**. This will show that the temperature indicator, placed after changing the status of the WBS item to Approved, cannot be assigned to the WBS item.

Before Starting this Procedure

- Define your workspace to display **Unit U04**, coordinate system **U04 CS**, and **Projects** in the WBS hierarchy. In your training plant, select **U04** and **WBS Items** compound filter from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

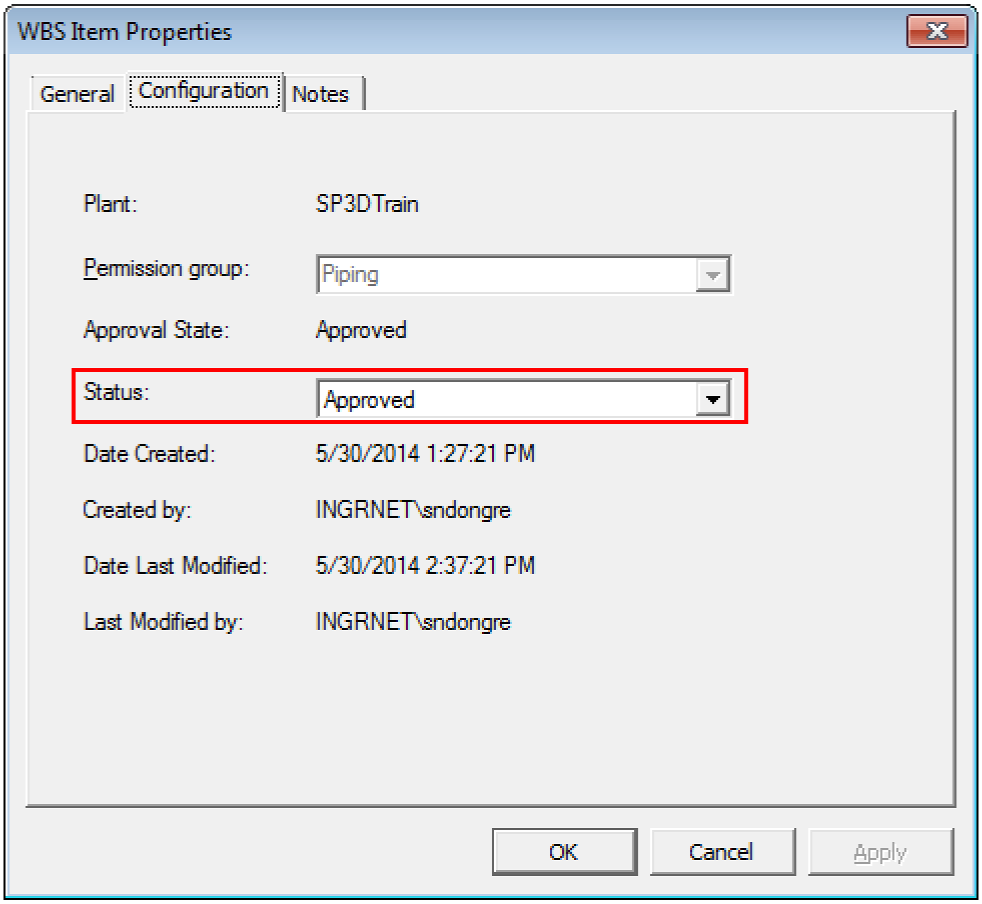
1. Switch to the **Piping** task.
2. In the Workspace Explorer, select the WBS tab.

3. Right-click the WBS item **PJ-99-01** and select the **Properties** option from the menu to change the properties of the WBS item PJ-99-01.

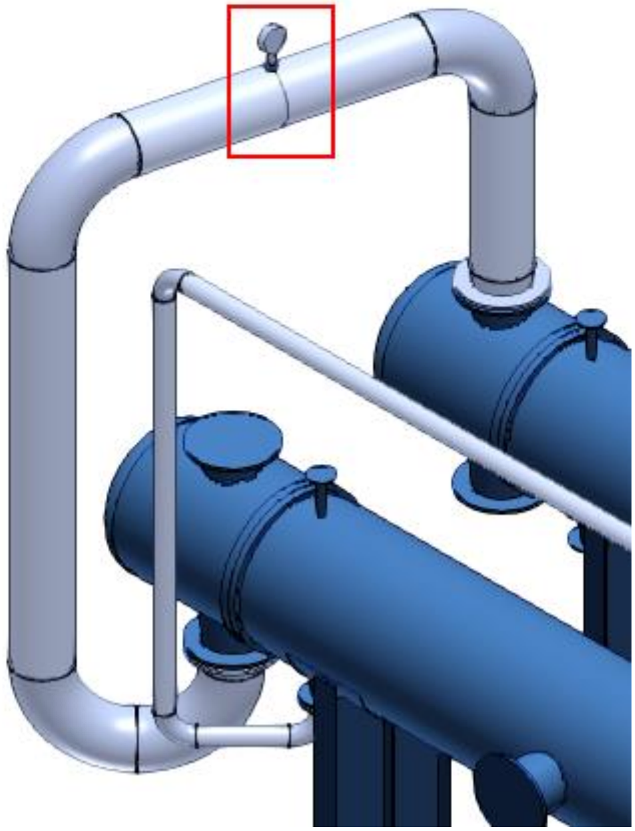


The WBS Item Properties dialog box appears.

1. On the **Configuration** tab, select **Approved** option in the **Status** drop-down list, and click **OK**.



2. Now place a temperature indicator **TI-1504** on the pipeline 403-P.



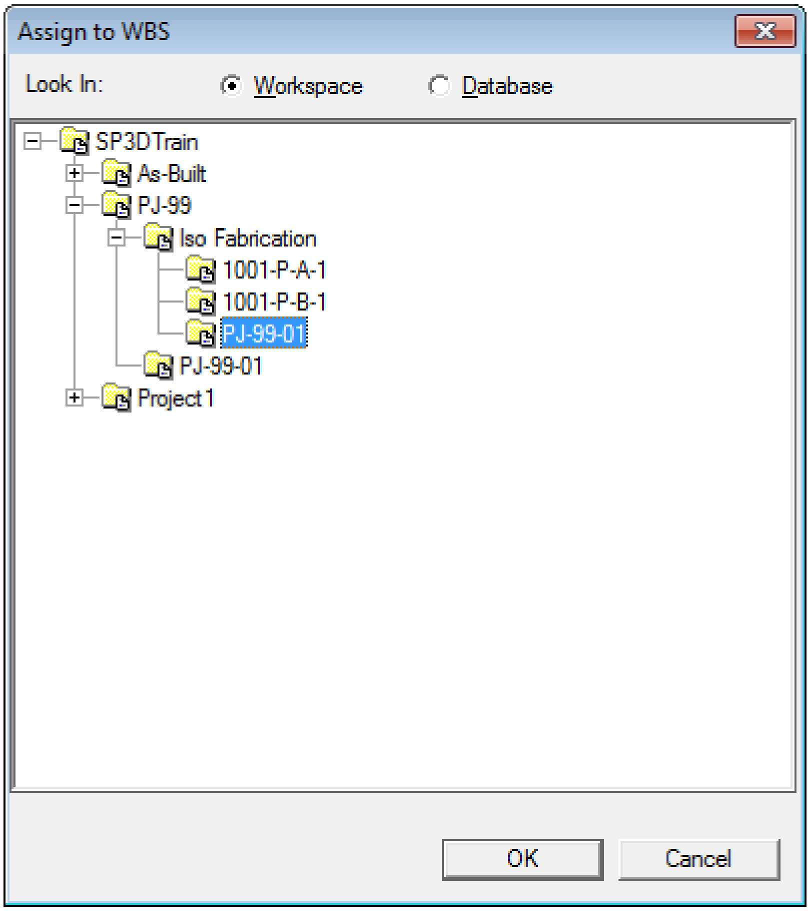
3. In the Workspace Explorer window, select the **System** tab.
4. Expand **A2 > U04 > Process** and select the pipeline system **403-P**.

The pipeline 403-P will be highlighted in the graphic view.

1. Now select **Project > Assign to WBS…**

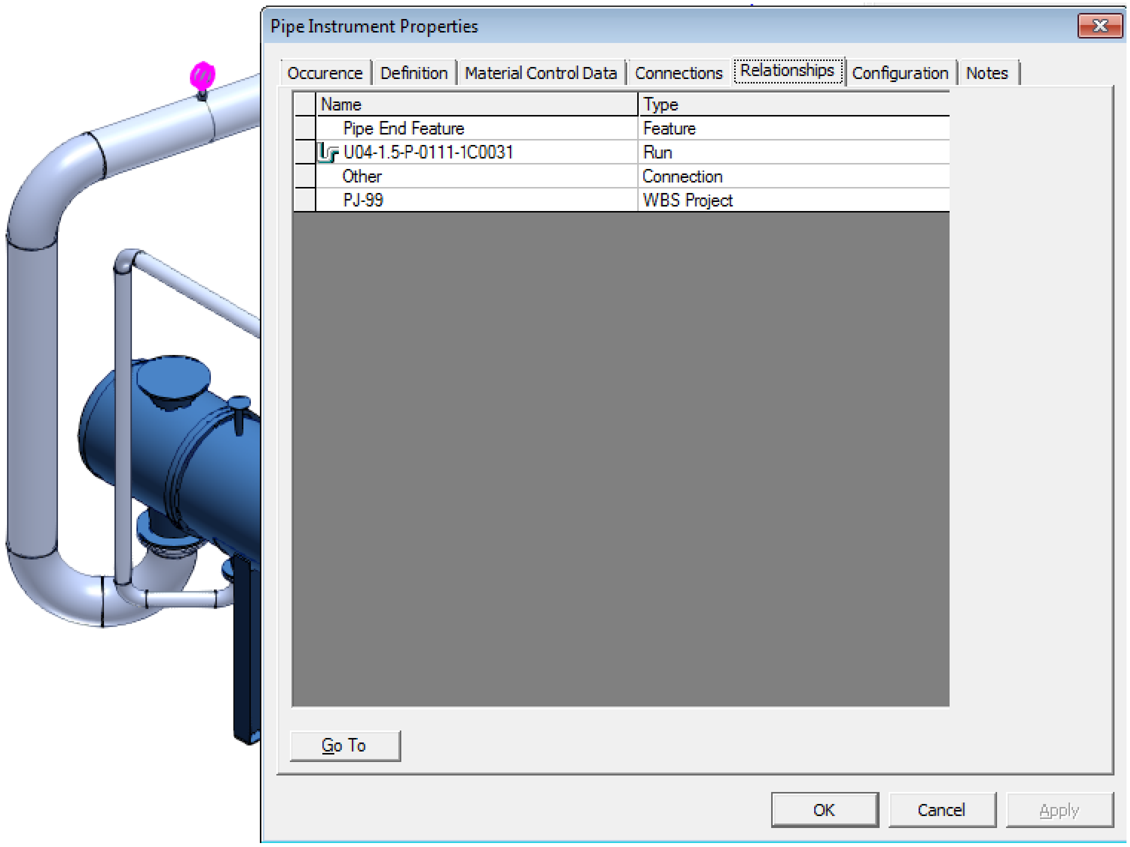
The Assign to WBS dialog box appears.

1. In the **Assign to WBS** dialog box expand **PJ-99** folder and select **PJ-99-01**. Click **OK**.



2. Select **Piping Parts** in the **Locate Filter** and select the temperature indicator **TI-1504** in the graphic view.
3. Right-click the temperature indicator TI-1504 and open the **Pipe Instrument Properties** dialog box.

The WBS Item record is not shown on the Relationships tab on the Pipe Instrument Properties dialog box.



Auto Correlate with P&ID

Objective

By the end of this session, you will be able to:

- Use the Auto-Correlate feature to correlate Smart 3D piping objects with the corresponding P&ID objects.

Overview

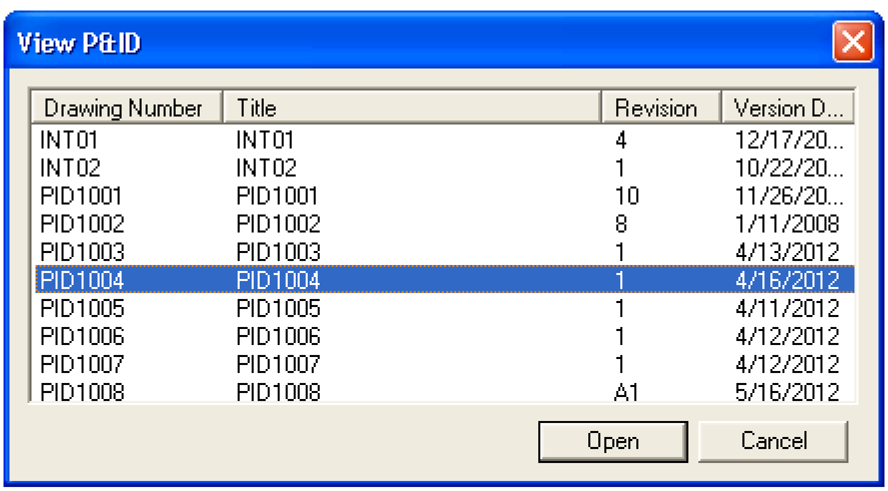
In an integrated environment, you publish and retrieve data from and to Smart 3D by using a central repository. During a publish operation, data such as drawings, reports, and 3D models transfers to a central repository. During a retrieve operation, the system retrieves P&IDs, Plant Break Down Structure, Project List, Work Breakdown Structure, Electrical Cable Schedules, and Instrumentation Dimensional Data Sheets from the central repository.

The P&ID's that were retrived from the central repository will be used to automatically correlate with the SP3D model.

Before Starting this Procedure

- Define your workspace to display **Unit U14** from **A3**. In your training plant, select **U14** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

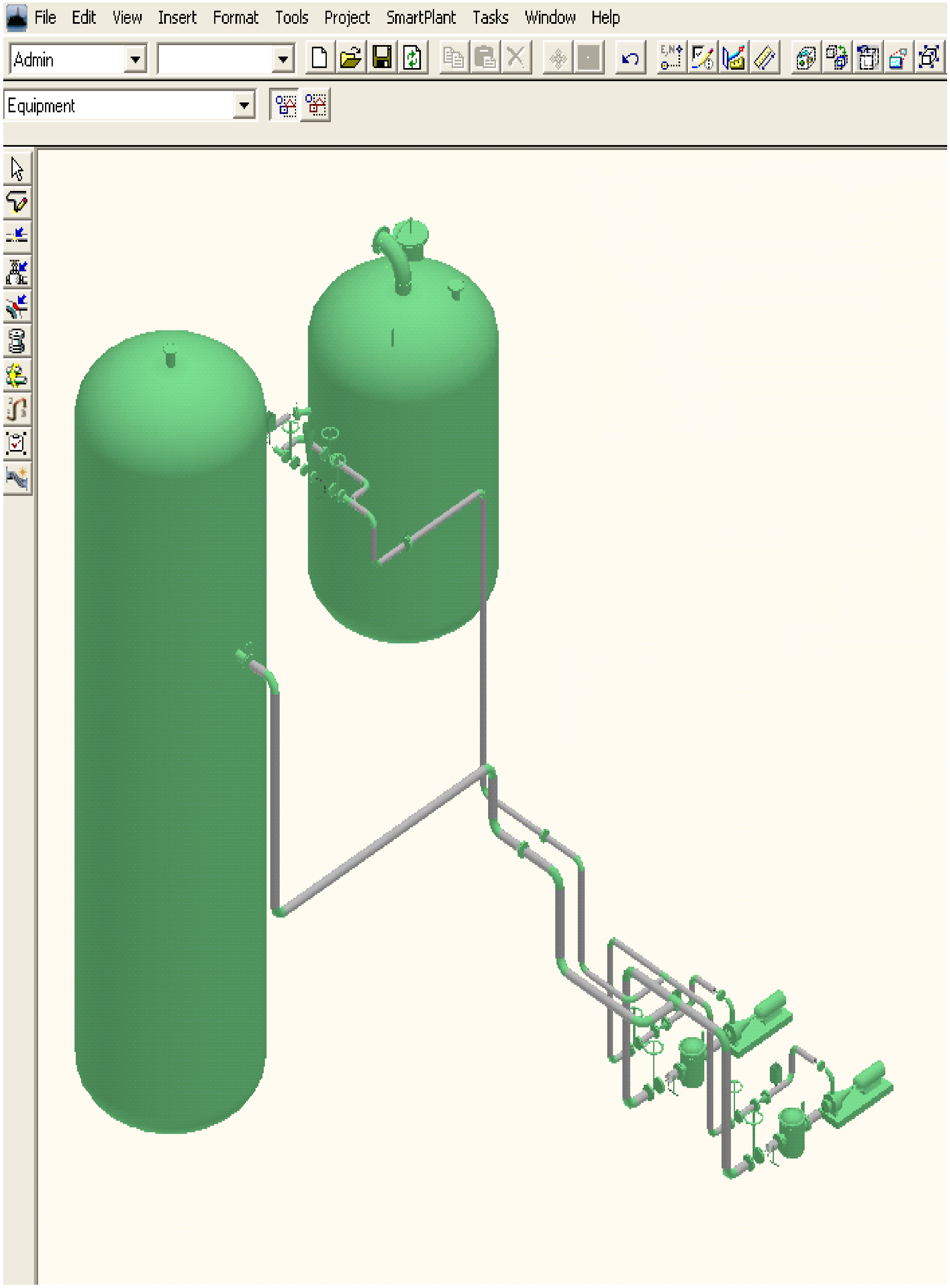
1. Select **SmartPlant > View P&ID**.
2. Select **PID1004**, and click **Open**.



NOTE This is the P&ID we will be working with during this exercise. We will start off by selecting a few objects in the 3D model that belong to the P&ID that we opened. These are the objects we will be working on correlating.

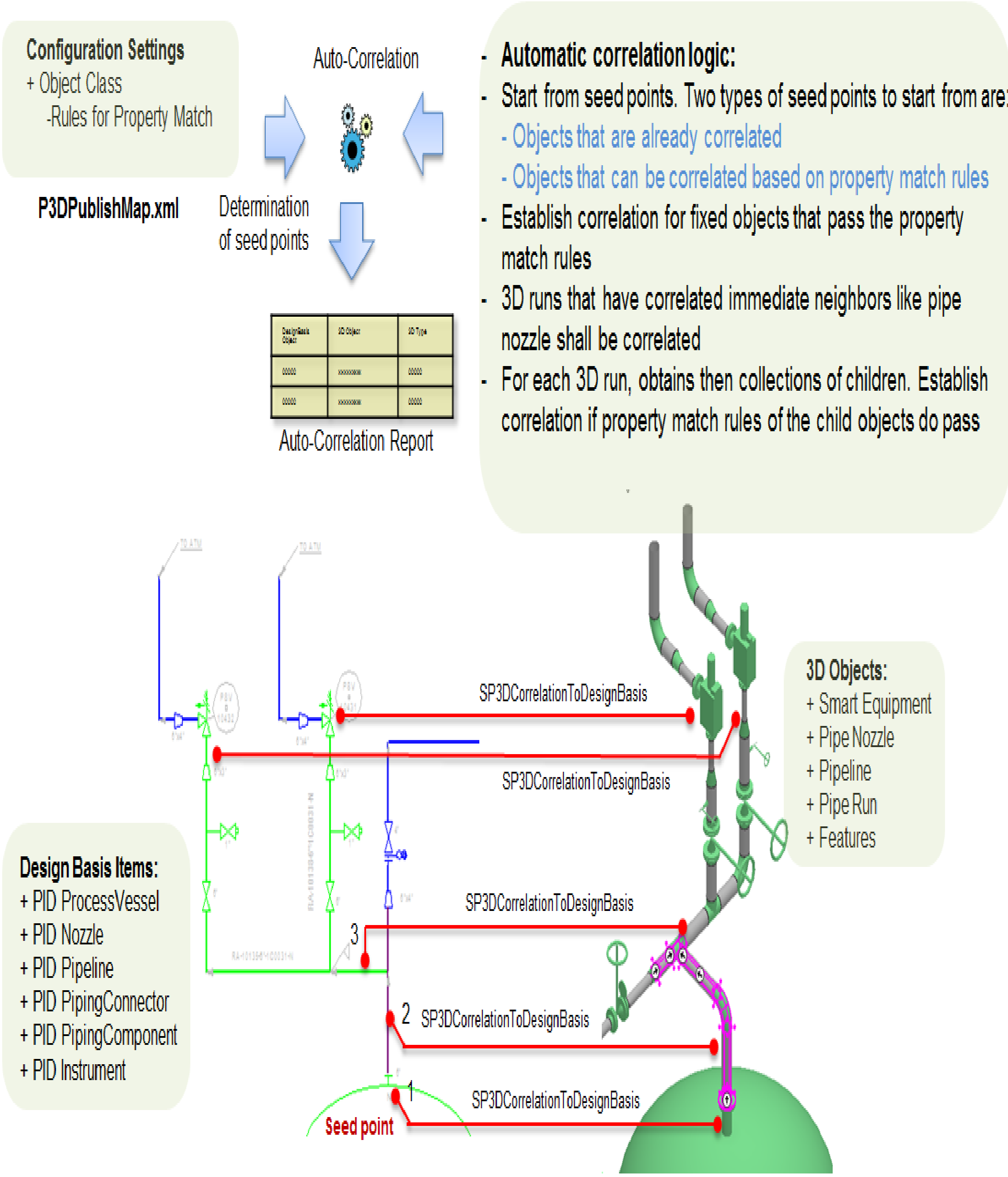
3. Select **Tools > Select By Filter…**

4. Select the **Filter** which contains a particular part of the P&ID you would like to correlate. In the example below, U14 is selected.


File Edit View Insert Format Tools Project SmartPlant Tasks Window Help
Admin
Equipment

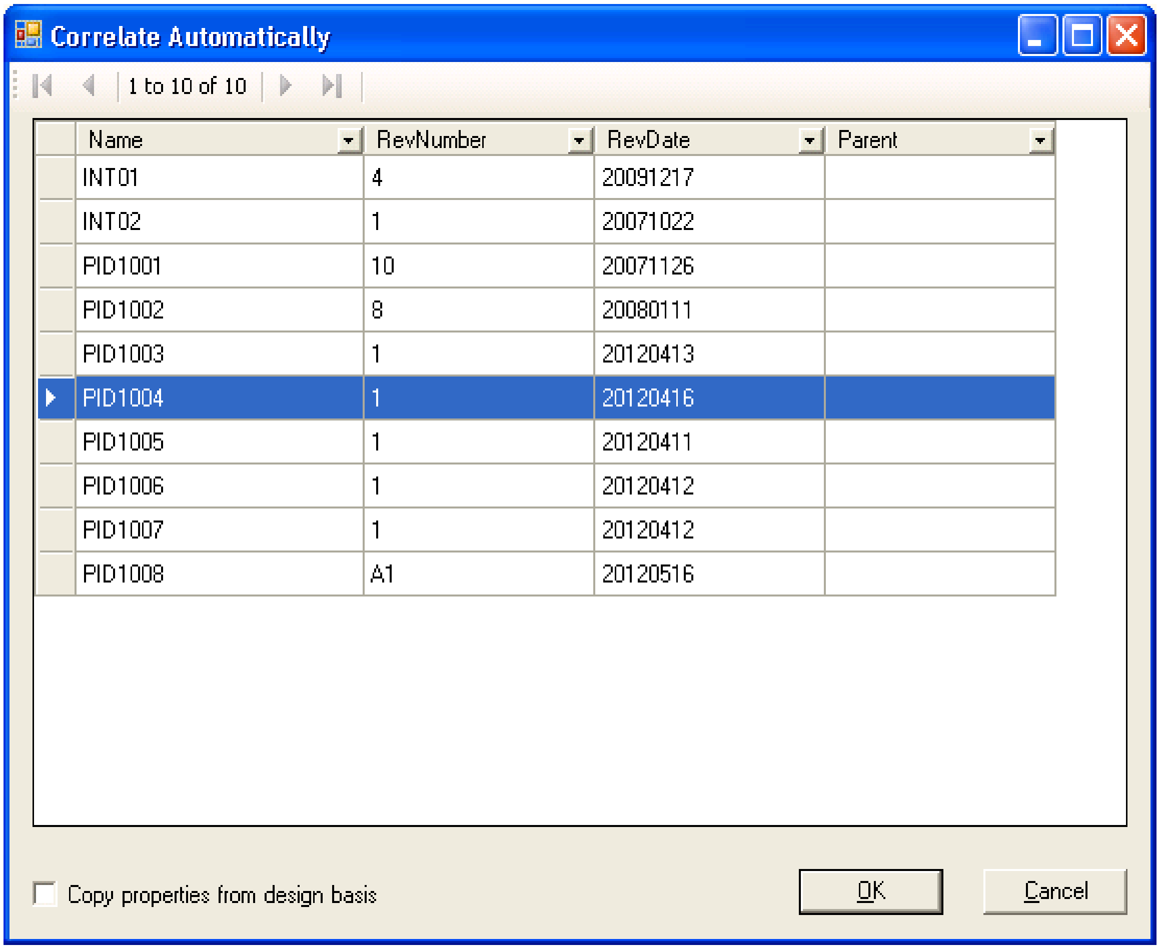
Smart 3D has a command to automatically correlate 3D objects to match design basis objects on the P&ID's. A report comprised of three sections, the ambiguous objects section, not correlated objects section and correlated objects section, is displayed at the end of the auto correlation process. This report allows users to review the results and resolve any correlation ambiguities.

One of the significant enhancements of the auto-correlate command is to correlate based on topology. The command starts the automatic correlation logic. Equipment, pipe nozzle, and pipeline are first correlated based on the property match rules. After pipelines have been correlated, the process shall correlate runs that have correlated immediate neighbors like equipment pipe nozzles. For each 3D run, the process obtains the collections of children and starts navigating through the collections one step at a time starting from the seed points. If property match rules of the child objects do pass, then a correlation is established.



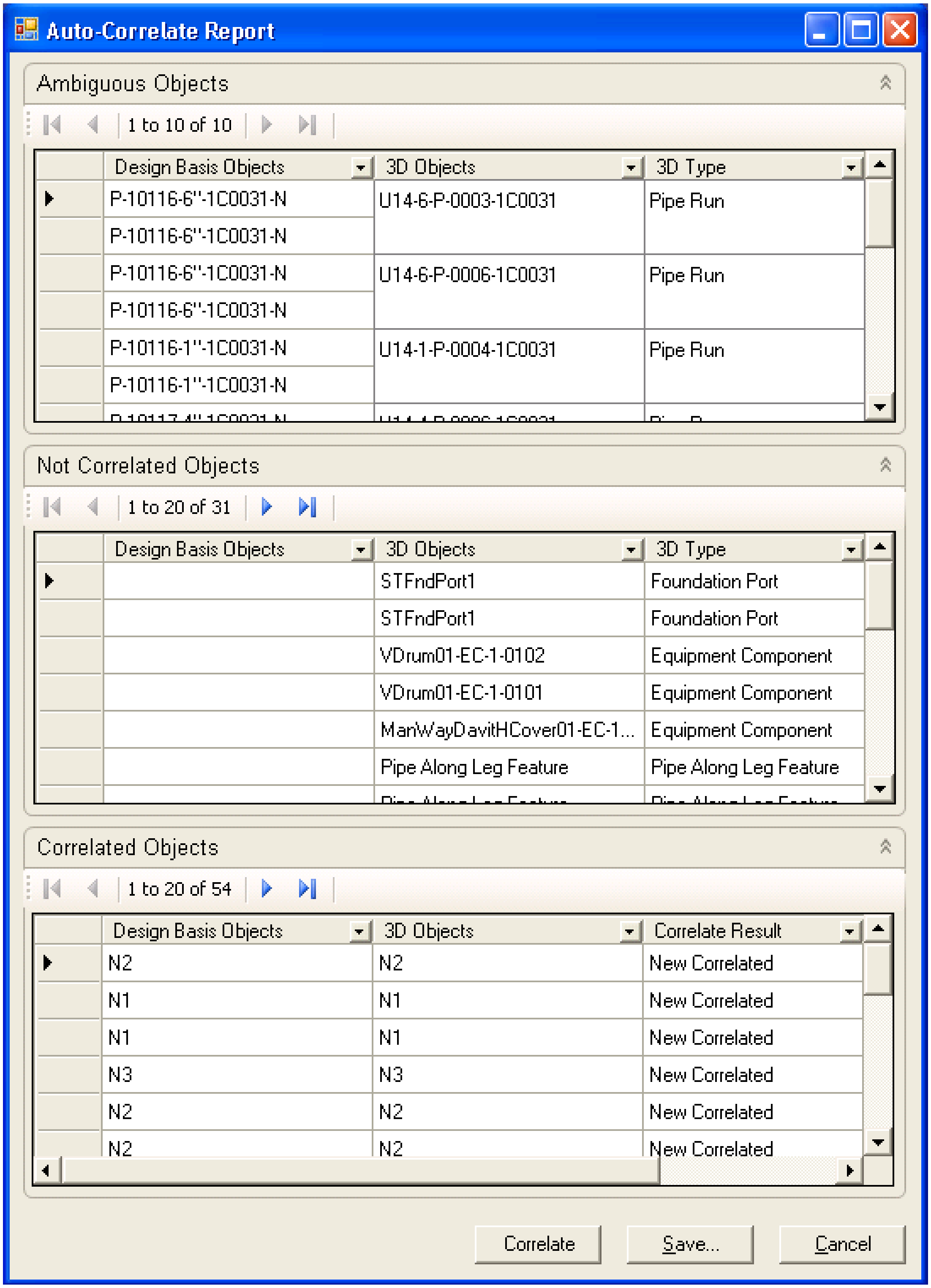
In the next steps we will start using this command to correlate with the P&ID

5. Select **SmartPlant > Correlate Automatically…**
6. Select **PID1004**, and click **OK**.



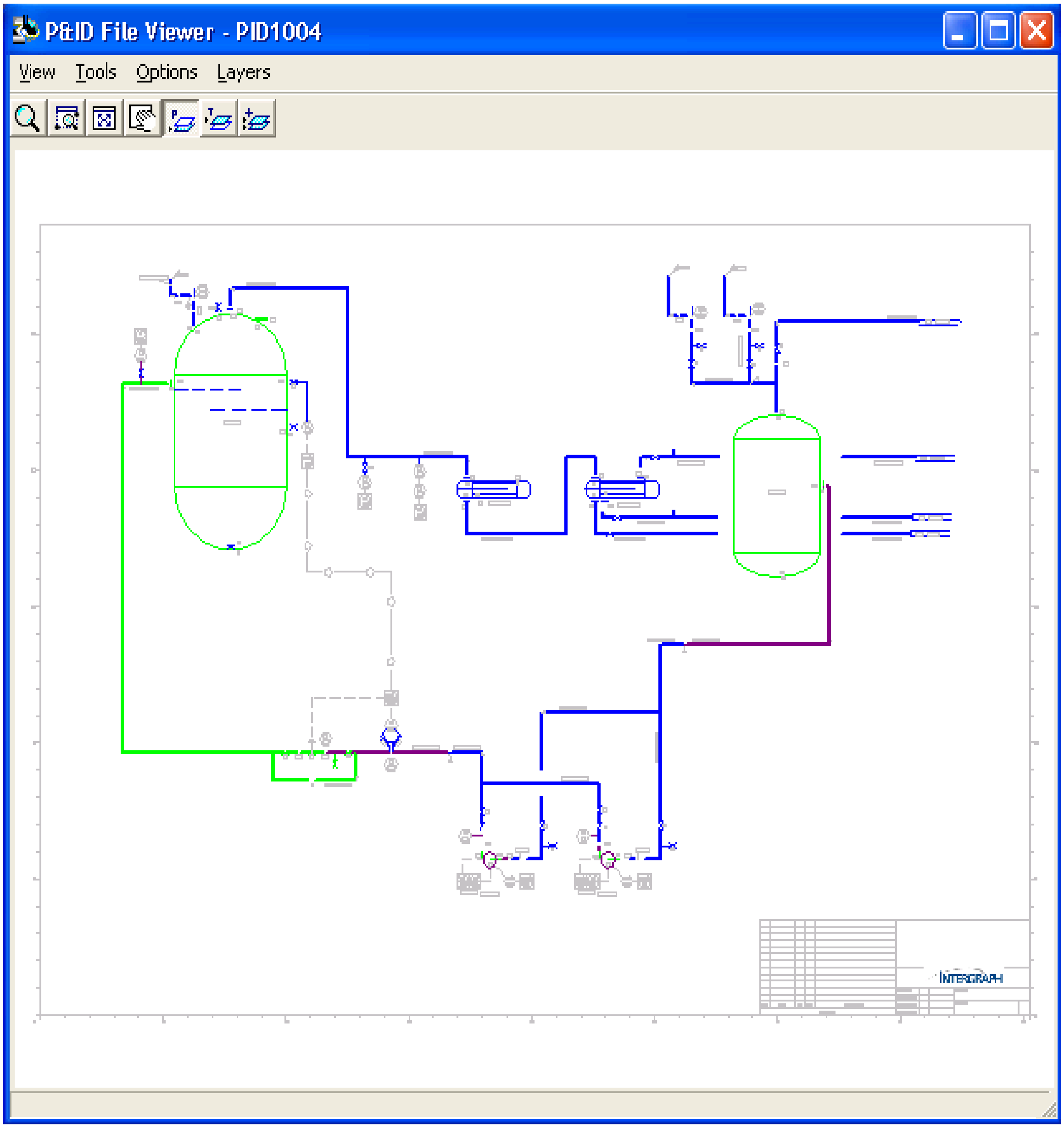
NOTE You can choose to select the check box to "Copy Properties from Design Basis." This will update all the properties in Smart 3D with the ones that are coming over from P&ID.

After the software finishes processing you will get the window shown below:



7. From this window you can hover your mouse over the objects to see them in the Smart 3D workspace.and the P&ID viewer. This window will allow you to resolve the ambiguities that exist by correlating the correct object.

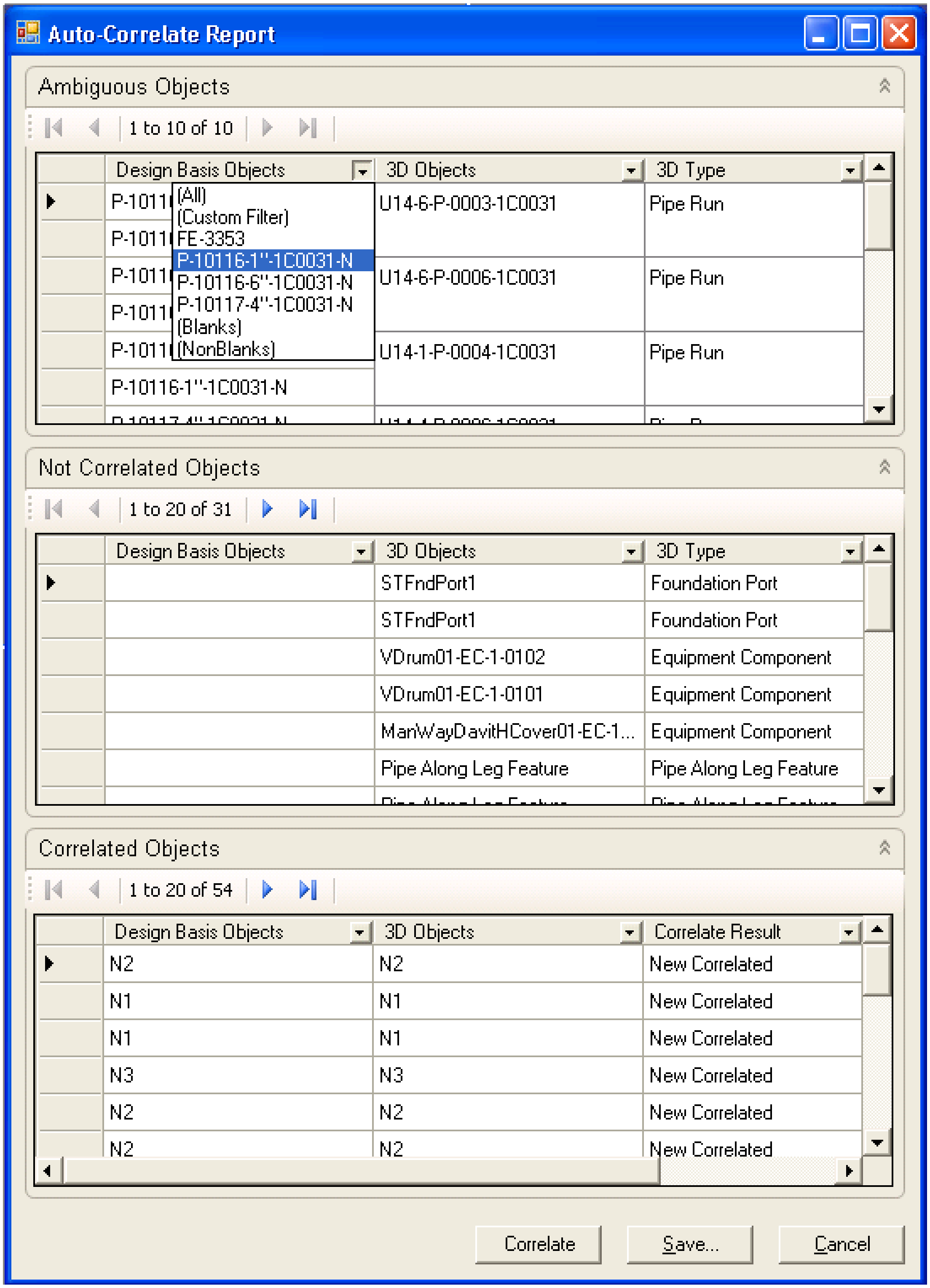
8. When you view the P&ID you will see some of the objects that were correlated, some that have inconsistent topologies, and some that are not yet correlated:



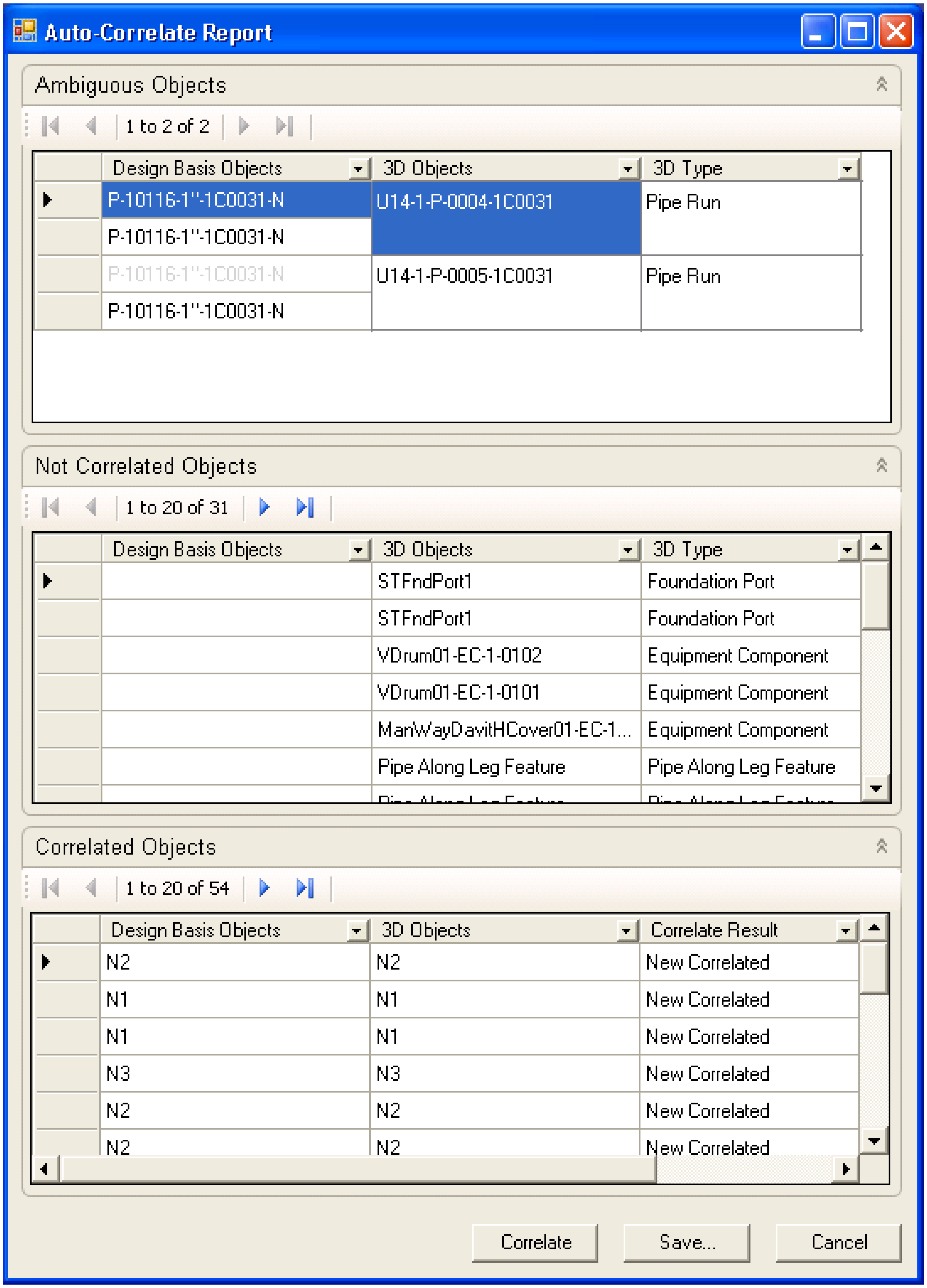
NOTE 54 objects were correlated (See Figure 7.7). You will now use the Auto-Correlate window in order to resolve the ambiguities as shown in the following steps.

The Auto correlate window has many useful features. One important feature is the ability to filter what objects you would like to see. This can be very helpful if you have many different ambiguous objects:

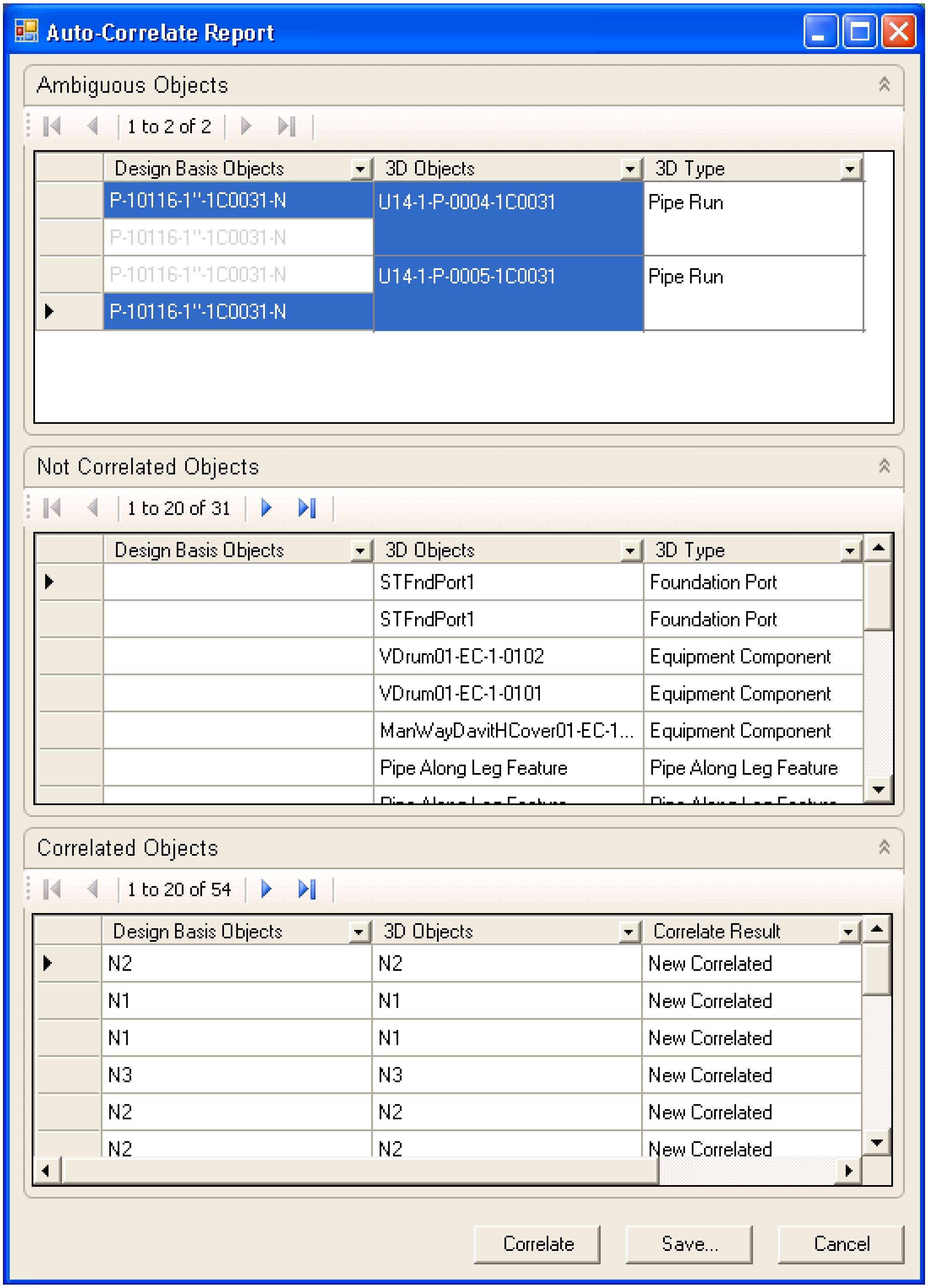
9. Click the drop down by the **Design Basis Objects**, and select **P-10116-1"-1C0031-N**



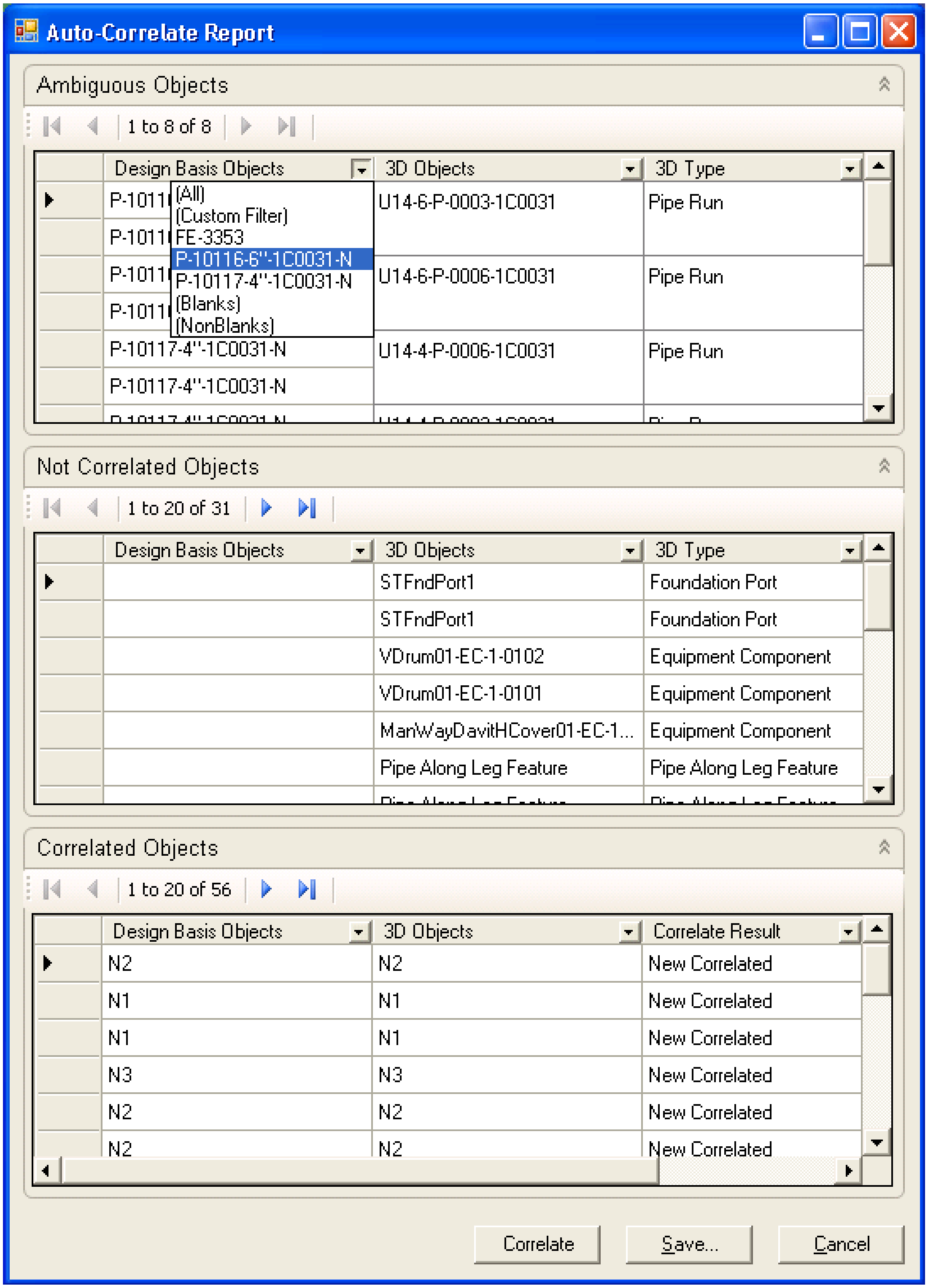
Now it makes it easier to see the the ambiguities for the design basis object that we are interested in. You will also see that when you click the correct corresponding object the software automatically resolves the other ambiguity:



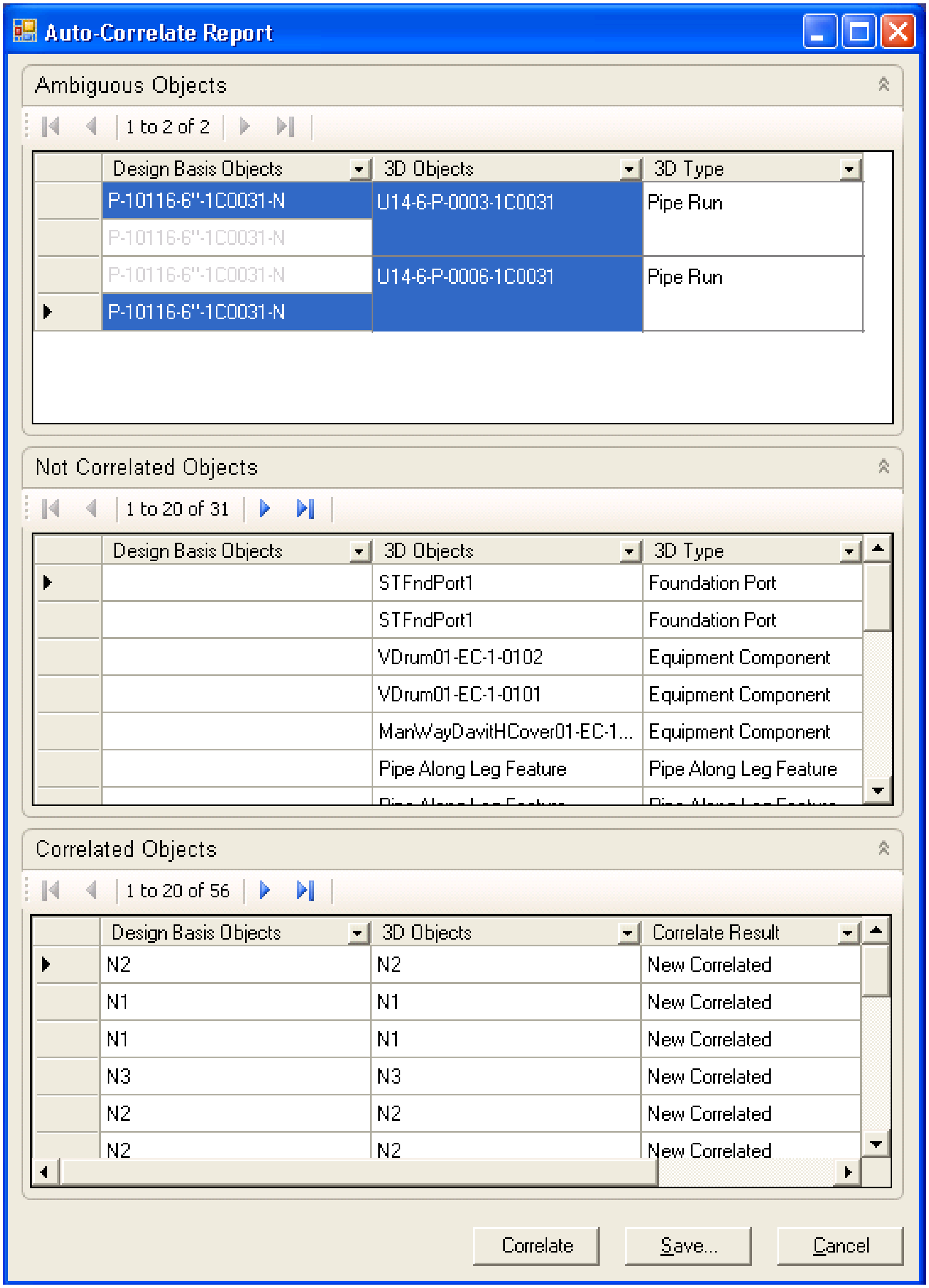
10. Select the following two objects, and click **Correlate** as shown below to resolve those particular ambiguities:



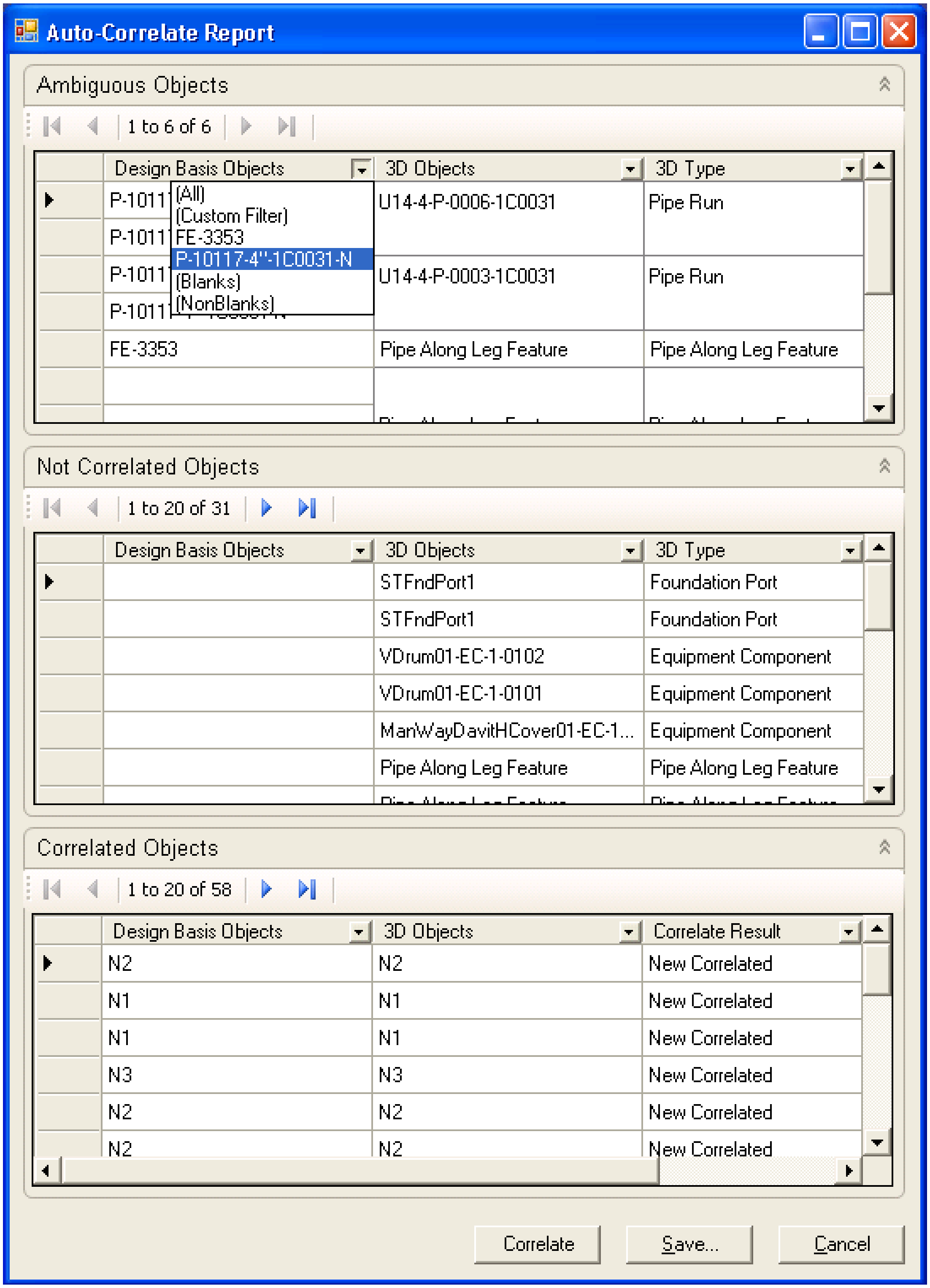
11. Select **P-10116-6"-1C0031-N**, from the **Design Basis Objects** drop down:



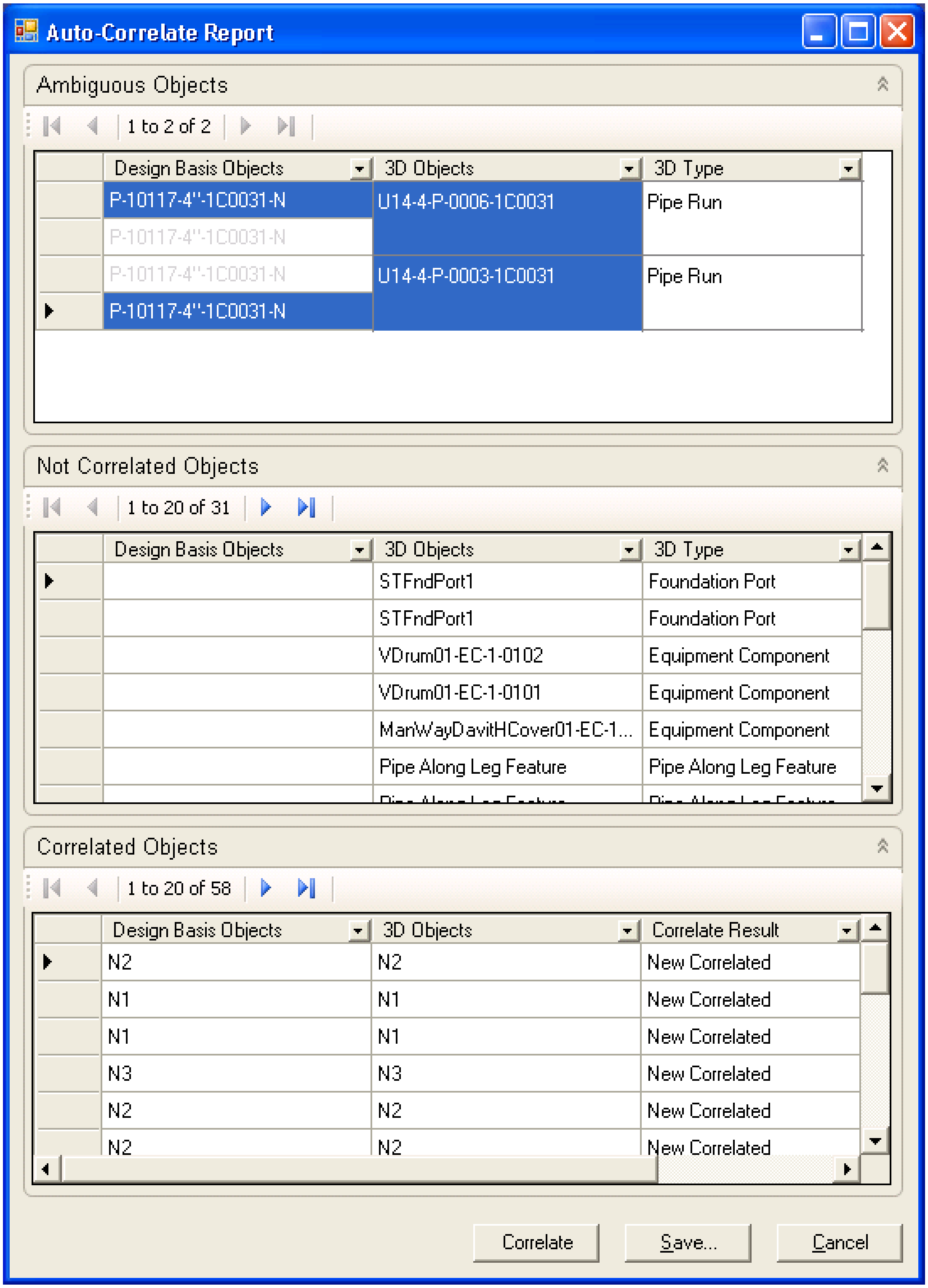
12. Select the following two objects, and click **Correlate**:



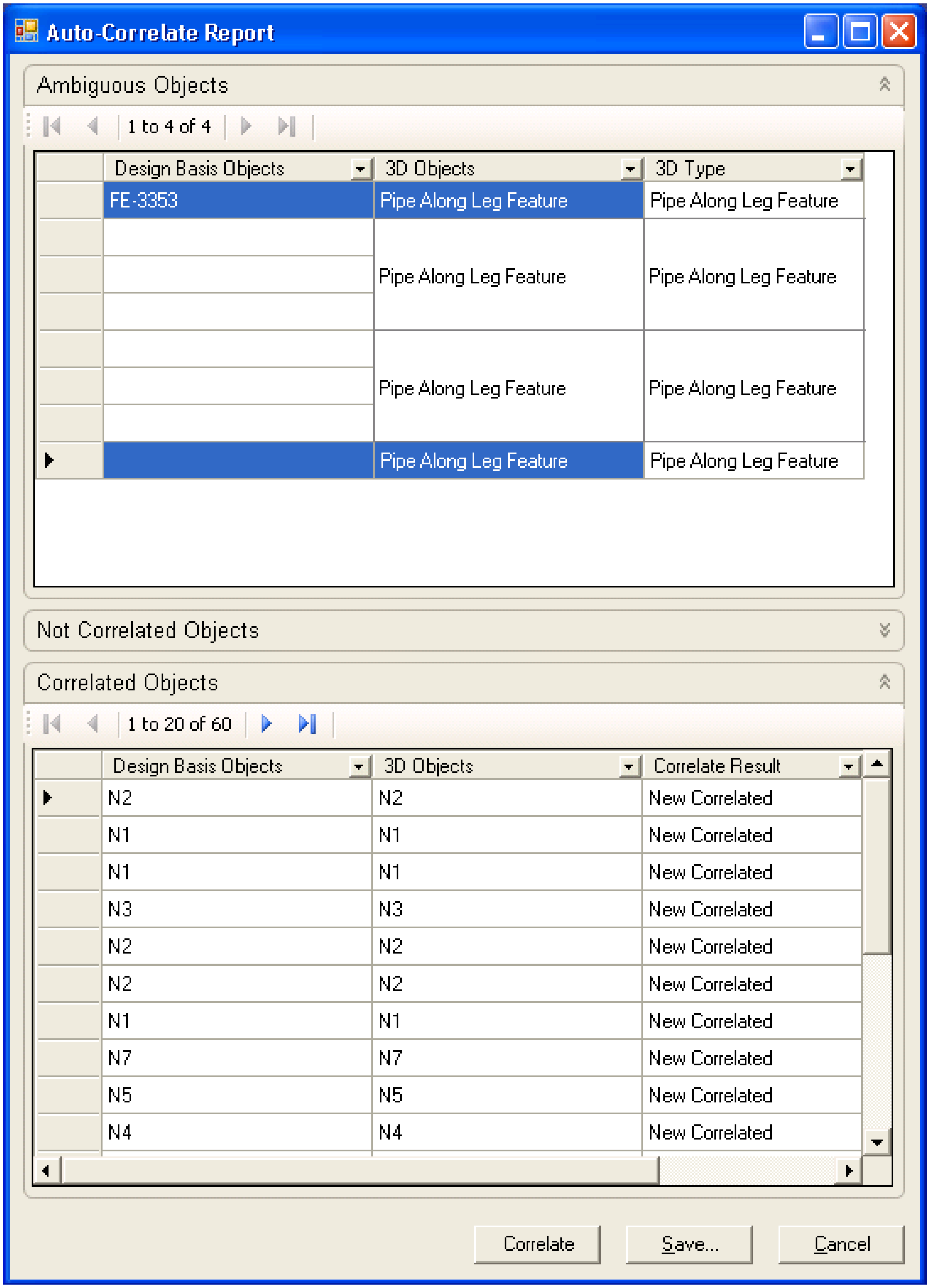
13. Select **P-10117-4"-1C0031-N** from the drop down as shown below:



14. Select the two objects, and click **Correlate** again as shown below:

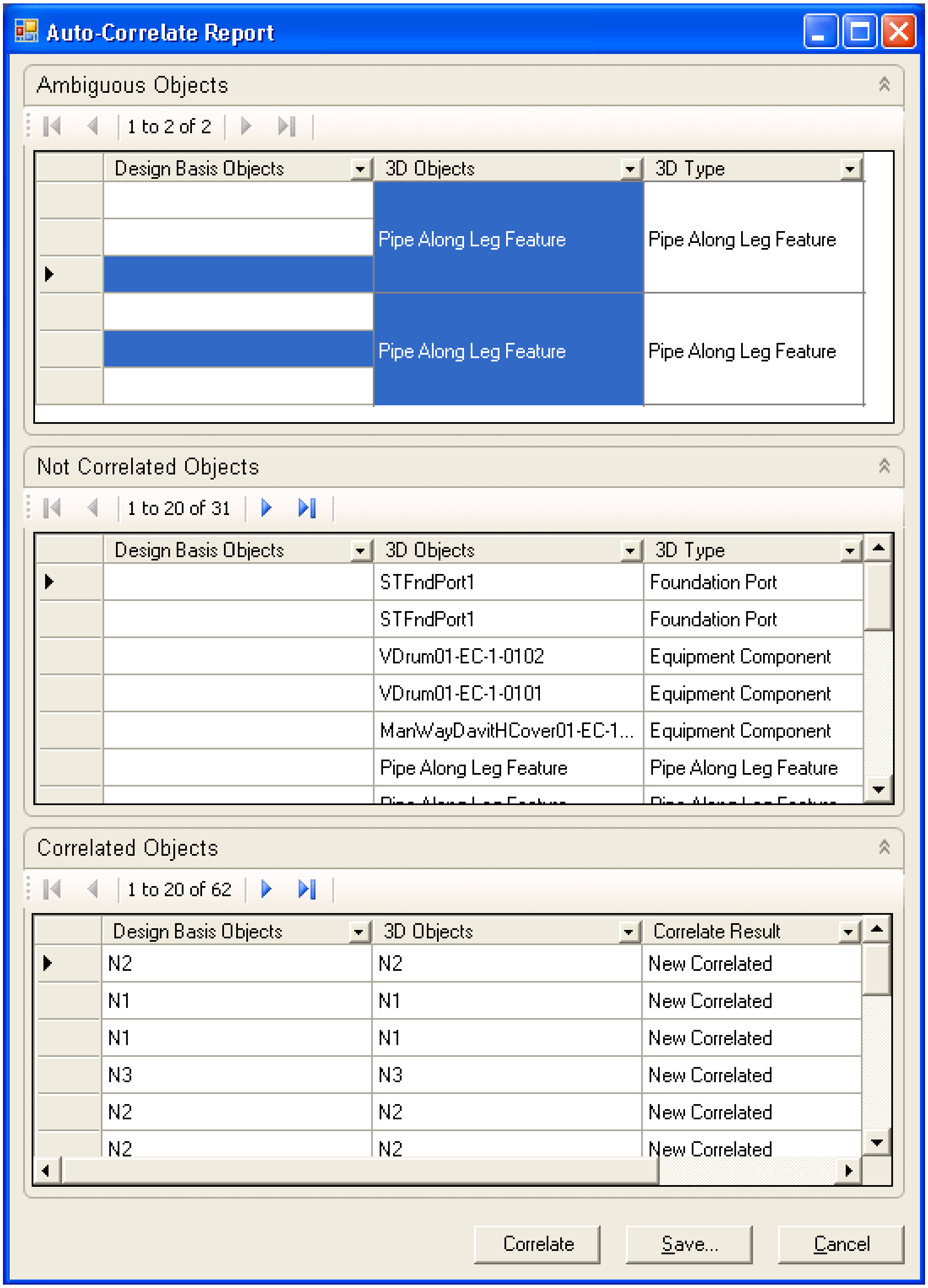


15. Find the two features, and click **Correlate** as shown below:



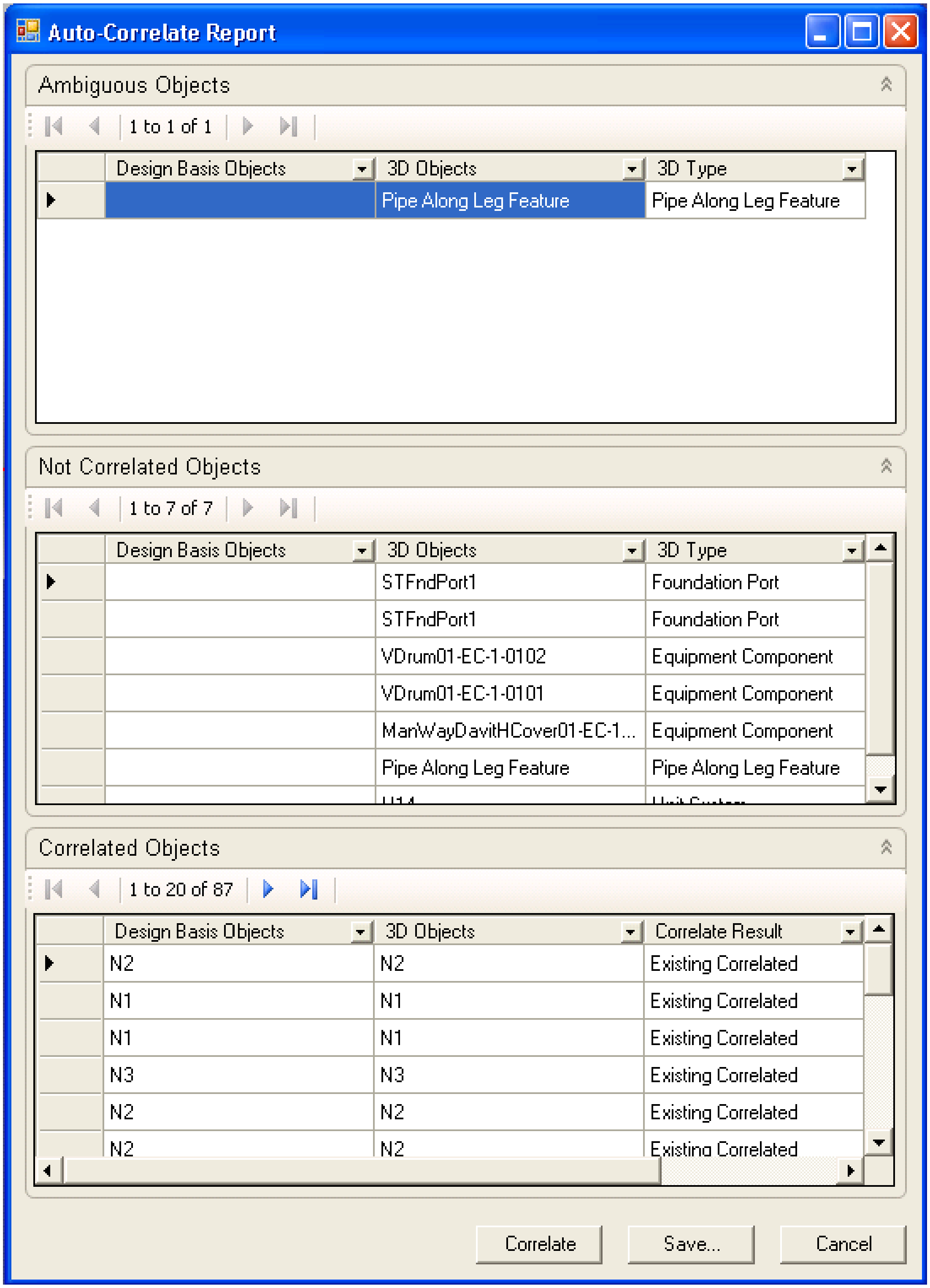
NOTE This window allows you to click on the different section to minimize and maximize.

16. Select the following two objects, and click **Correlate**:

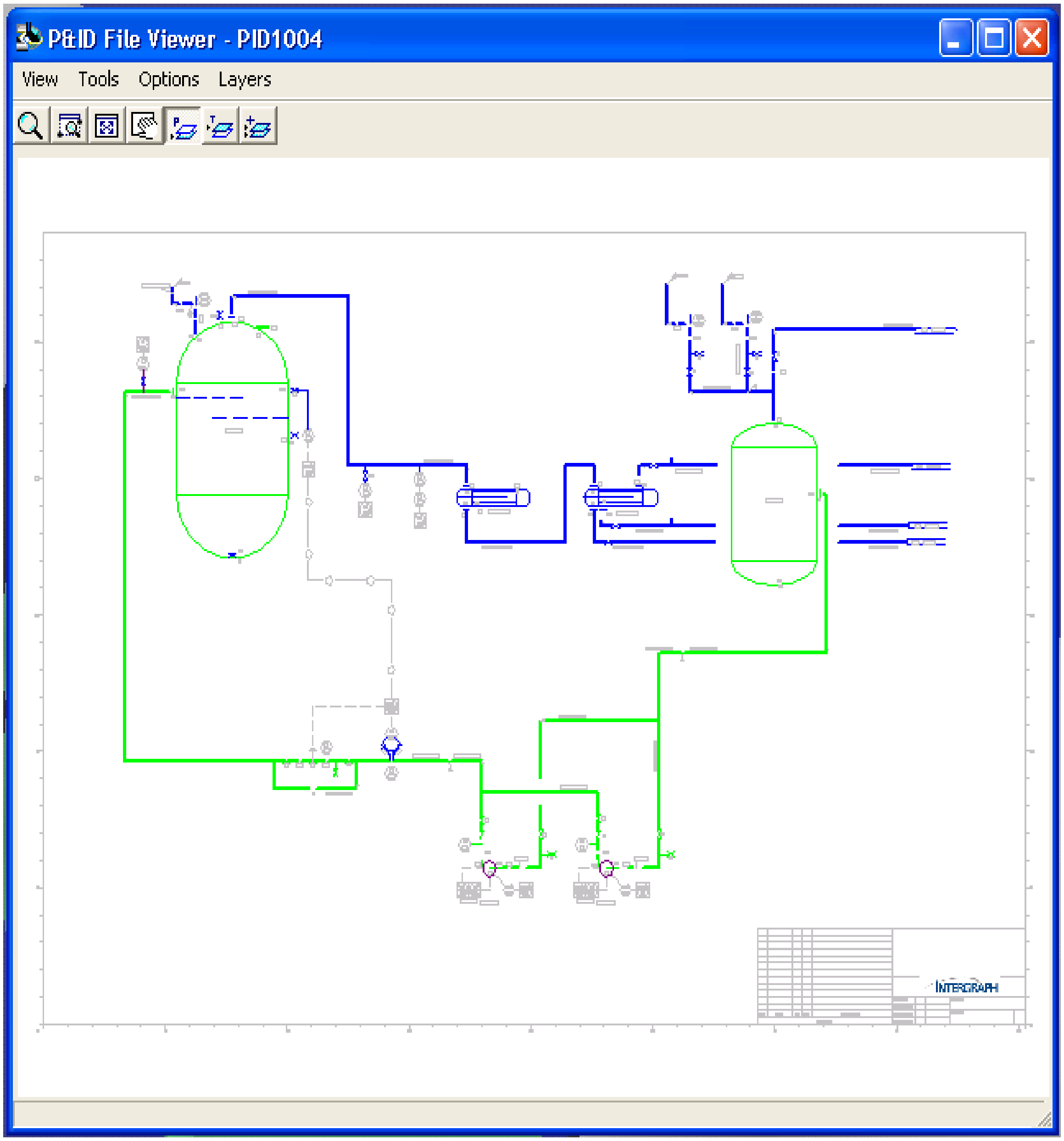


NOTE You should now see no Ambiguous objects in the Auto-Correlate Report.

17. Click **Cancel** and repeat the steps again. This will be our second pass of Auto-Correlation. You will see one Ambiguous object in the Auto-Correlate Report. Select the object and click **Correlate** as shown below:



18. You should now see no objects in the ambiguous object section for the second time, and your P&ID should look like the one shown below:



The two features towards the bottom are purple because of topology mismatches. If you use the **P&ID Viewer** to zoom in you will see that these features have some attached features that have not been correlated because they do not exist in the Smart 3D model.

SESSION 15

Approving Inconsistencies and Data Mismatches

Objective

By the end of this session, you will be able to:

- Approve inconsistencies and data mismatches from the **Compare Design Basis** window.

Overview

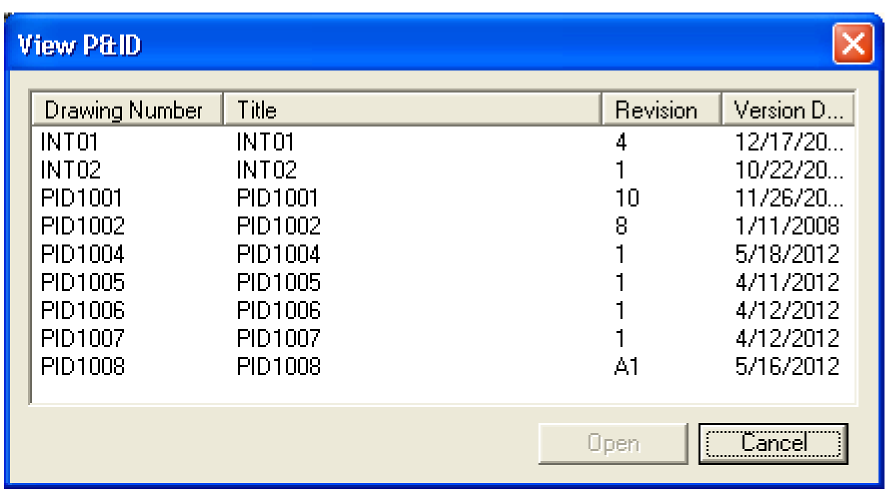
Smart 3D supports the ability to approve property and topological inconsistencies between S3D objects and their correlated design basis objects. A new property "Correlation Approval Status" will hold this override and the Compare with Design Basis dialog will show these approval records with a yellow fill. The P&ID File Viewer has the option to display such overrides with different optional color.

The P&ID's that were retrieved from the central repository will be used for approving inconsistencies.

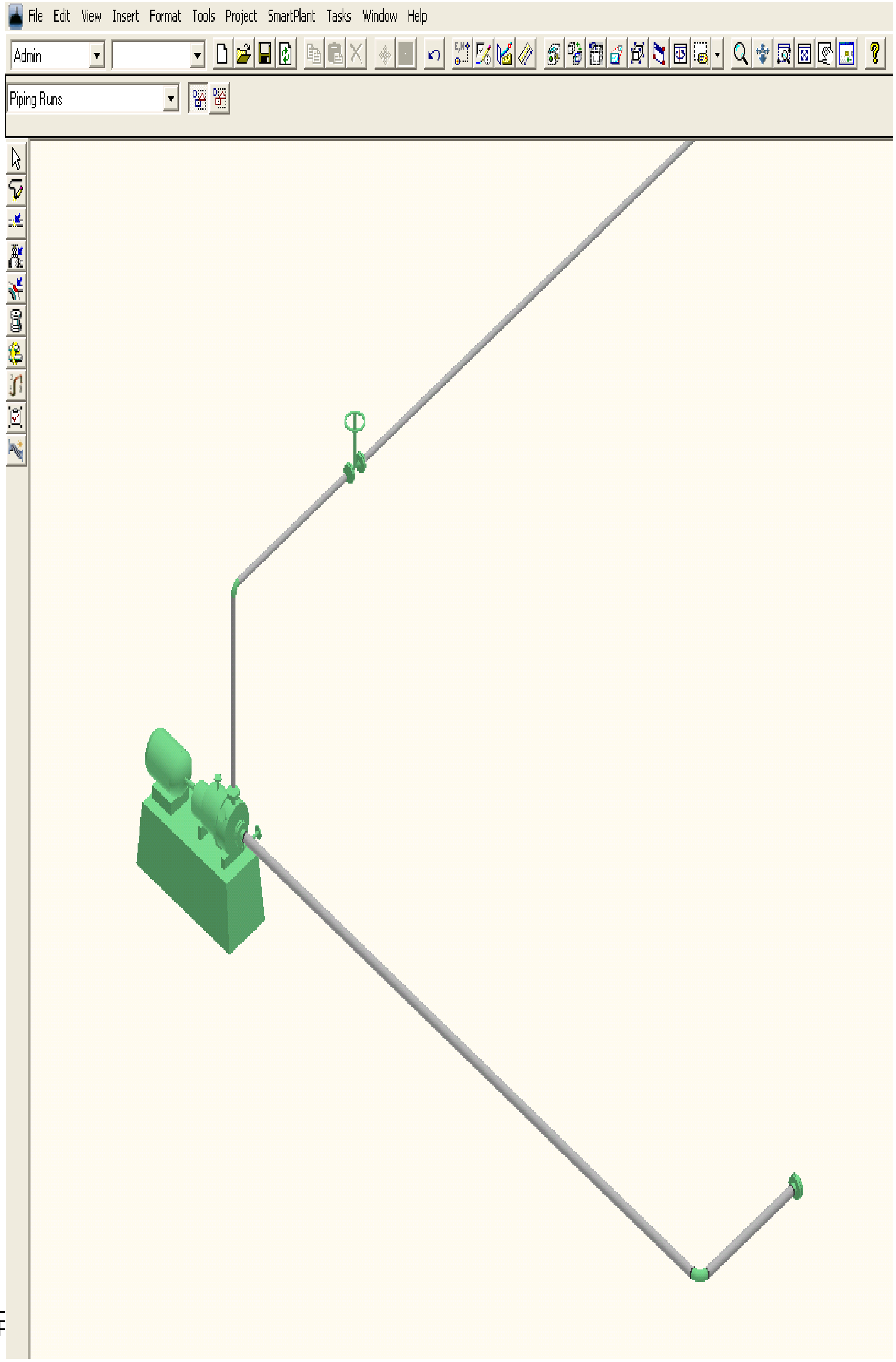
Before Starting this Procedure

- Define your workspace to display Unit **U15** from **A3**. In your training plant, select **U15** from **Plant Filters > Training Filters** in the the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

1. Select **SmartPlant > View P&ID**.
2. Select **PID1008**, and click **Open**.



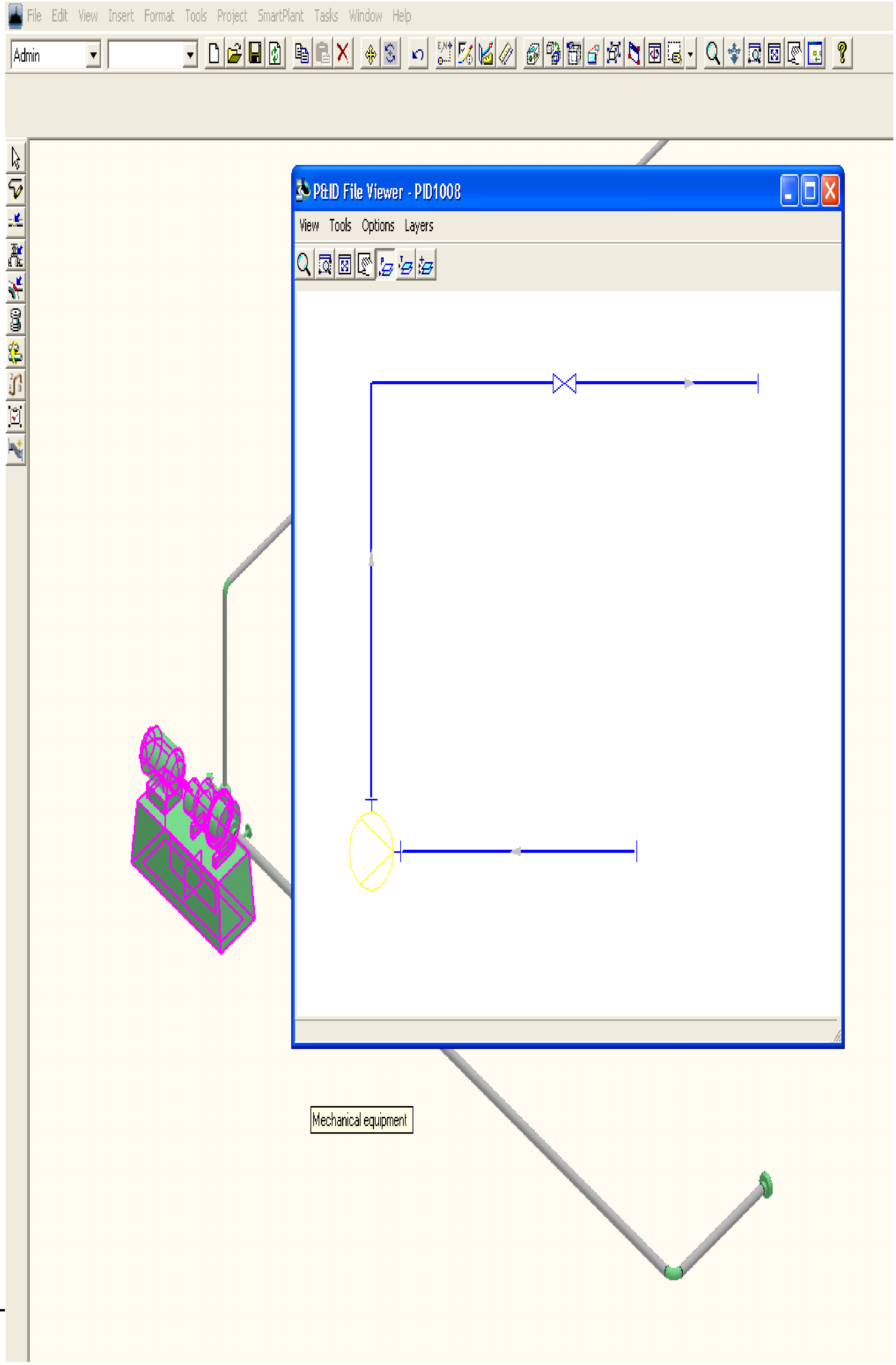
NOTE This is the P&ID we will be working with during this exercise. The filter that should be defined in our workspace should be P&ID1008.


File Edit View Insert Format Tools Project SmartPlant Tasks Window Help
Admin
Piping Runs

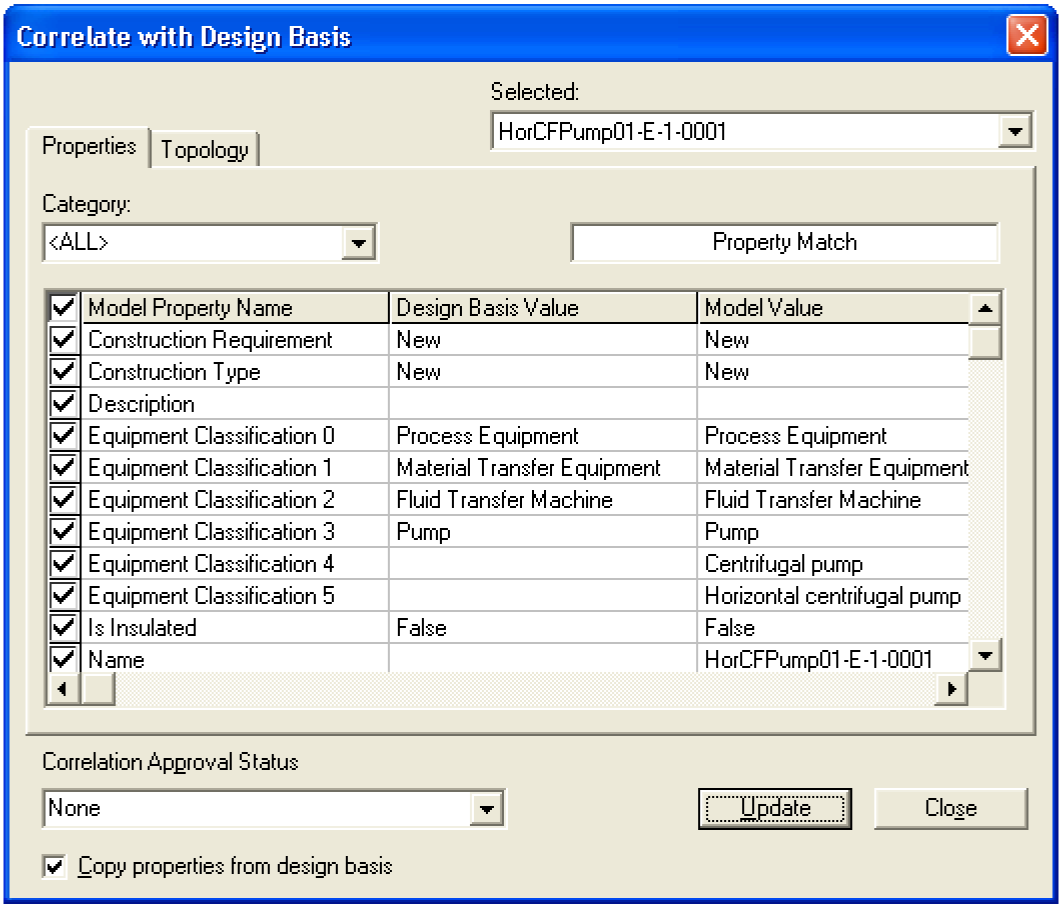
3. Select the **Equipment in SP3D Model Space**. Then, select **SmartPlant > Correlate with Design Basis…**

   You are prompted to "Select P&ID item to be correlated."

4. Click on the P&ID from your Windows taskbar that you opened earlier, and select the equipment on the P&ID.

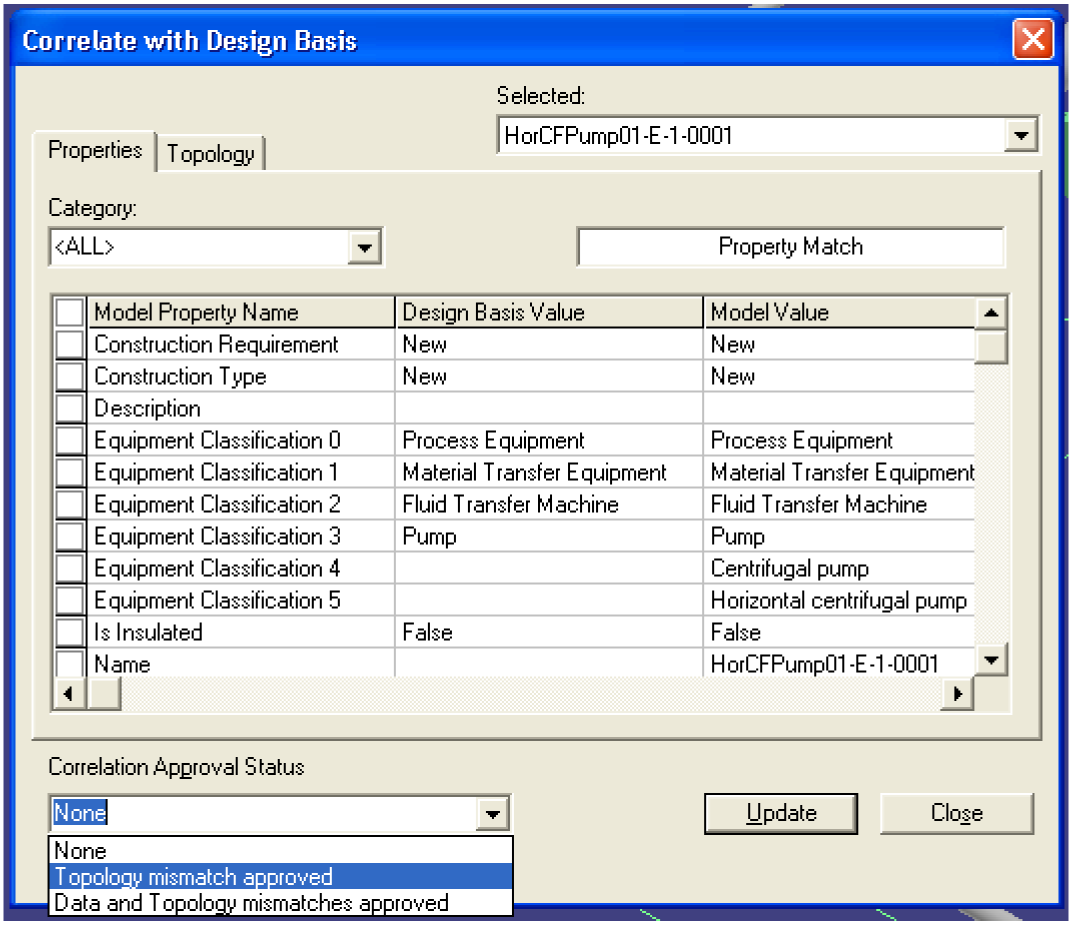

File Edit View Insert Format Tools Project SmartPlant Tasks Window Help
Admin
P&ID File Viewer - PID1008
View Tools Options Layers
Mechanical equipment

You will now see the following window pop up:

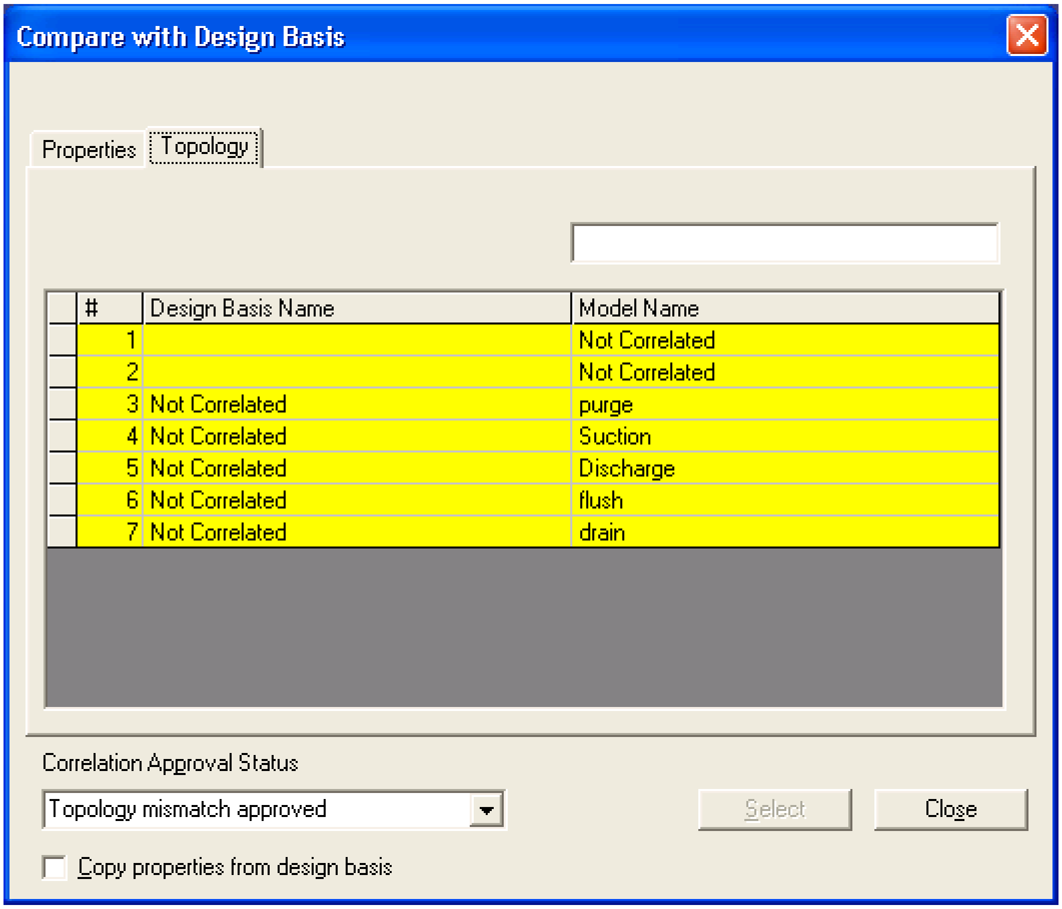


NOTE You will see that clicking update here will not update the topology.

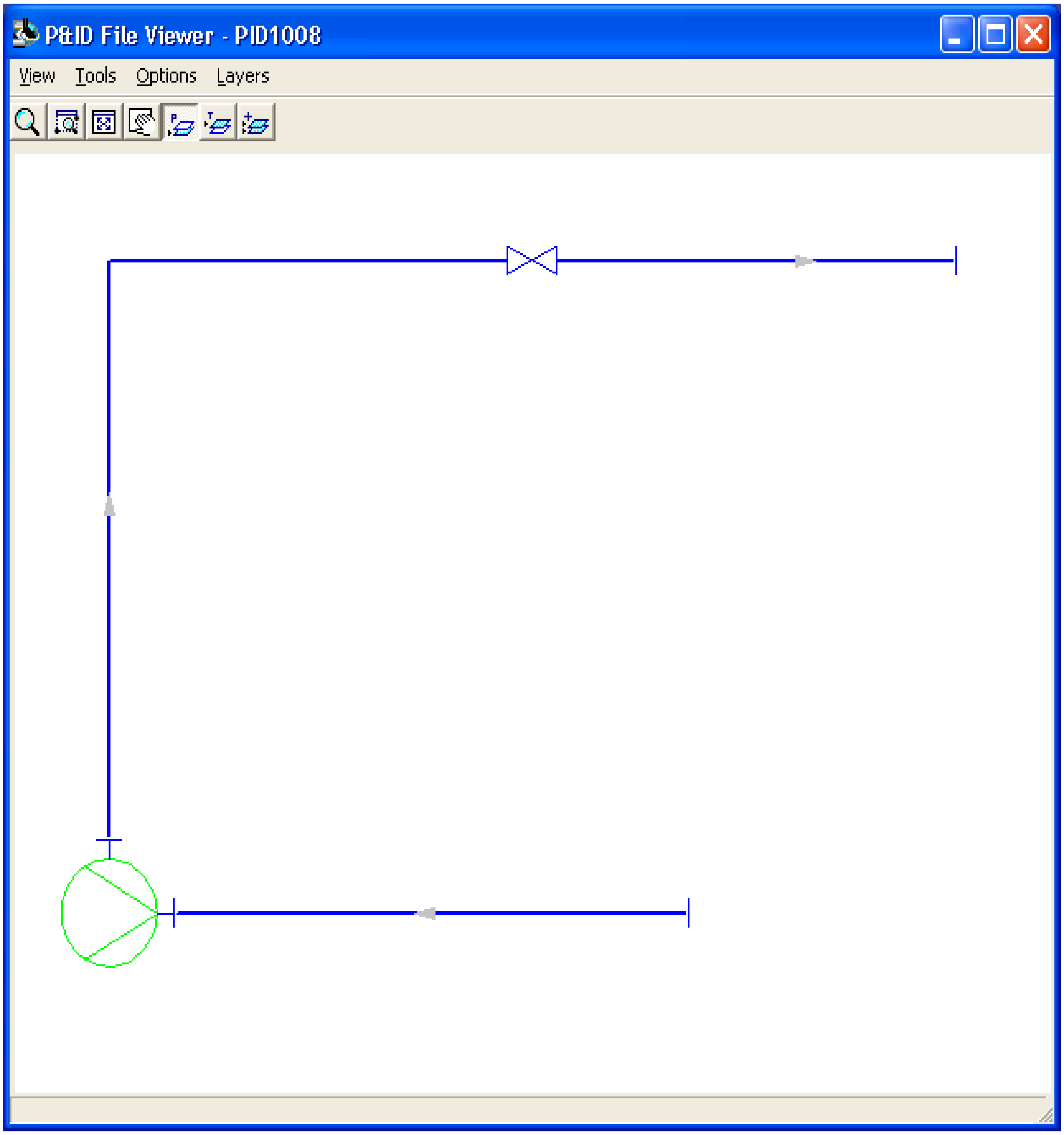
5. From the pull-down list under the **Correlation Approval Status** select **Topology mismatch approved** as shown below and click **Update**.



6. Repeat steps 4, 5, and 6 until all the rows from your **Topology** tab turn yellow as shown below:



7. Now when you view the P&ID you will see that the equipment turns from Purple to Green as shown below:



SESSION 16

One-to-Many Correlation

Objective

By the end of this session, you will be able to:

- Correlate one P&ID pipe run to many pipe runs in Smart 3D

Overview

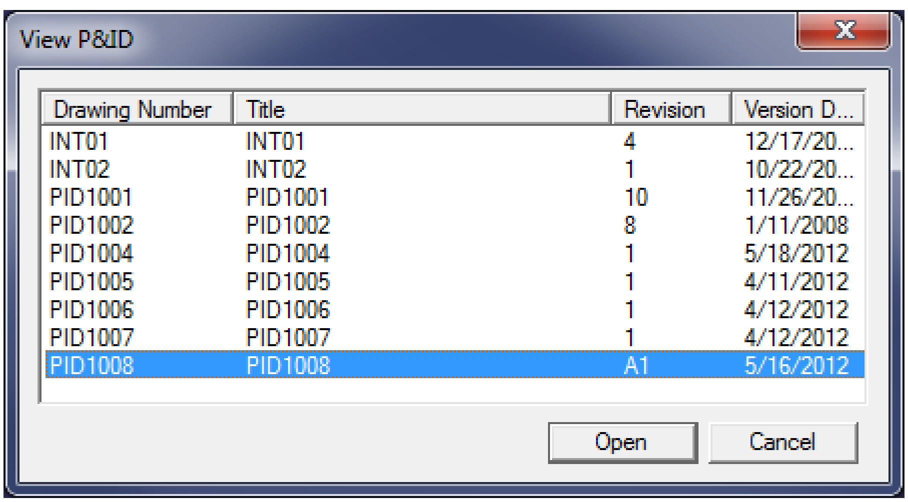
Sometimes, a pipeline is split into multiple pipelines at construction boundaries in the 3D model while complying with one logical pipeline specified in the P&ID's. To establish correlation for these cases, Smart 3D supports the ability to correlate many Smart 3D Pipeline/Runs to one P&ID Pipeline/Run.

The P&ID's that were retrieved from the central repository will be used for one-to-many

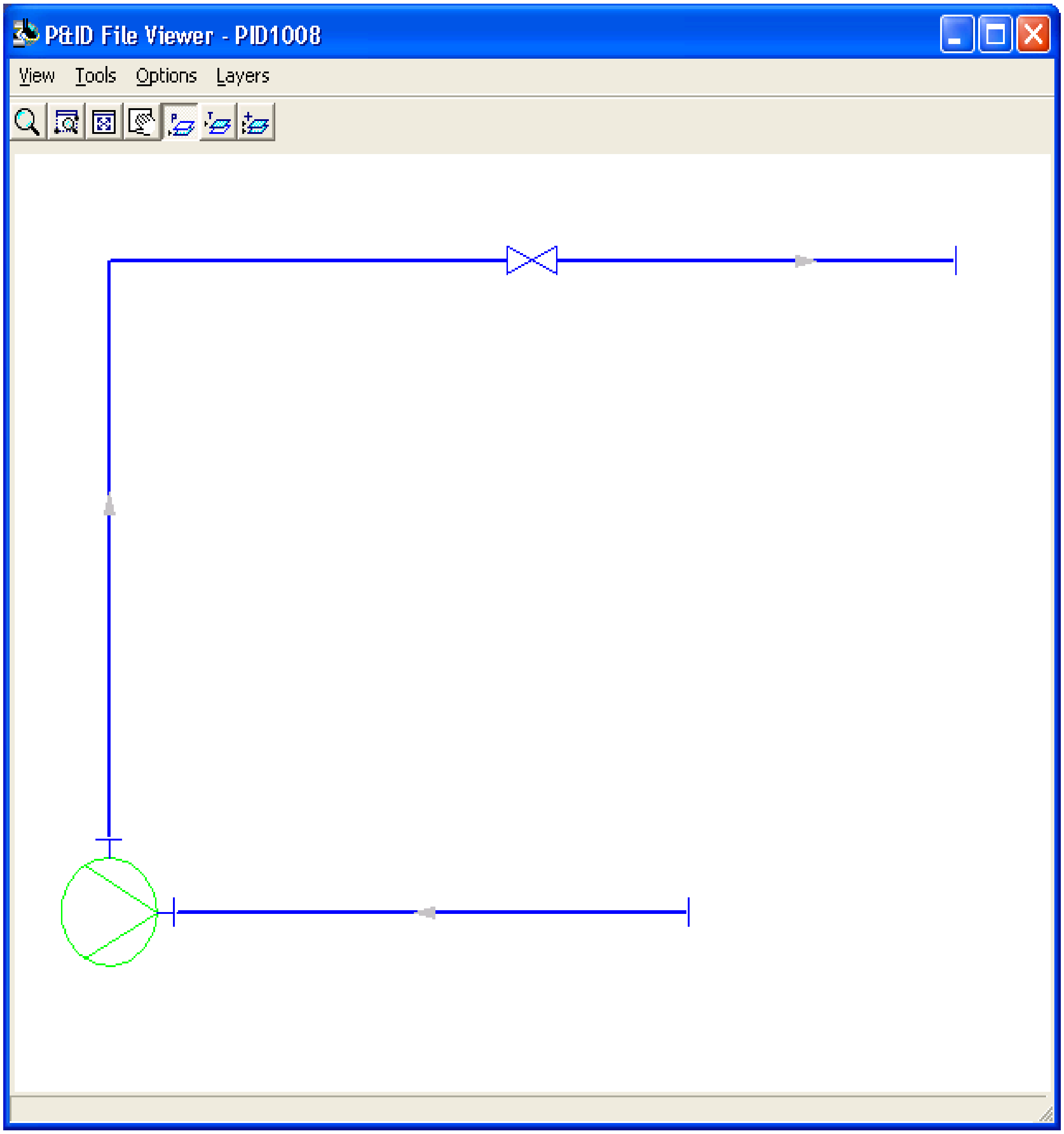
Before Starting this Procedure

- Define your workspace to display Unit **U15** from **A3**. In your training plant, select **U15** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.
- Make sure you are in the **Piping** task and the **Active Permission Group** is set to **Piping**.

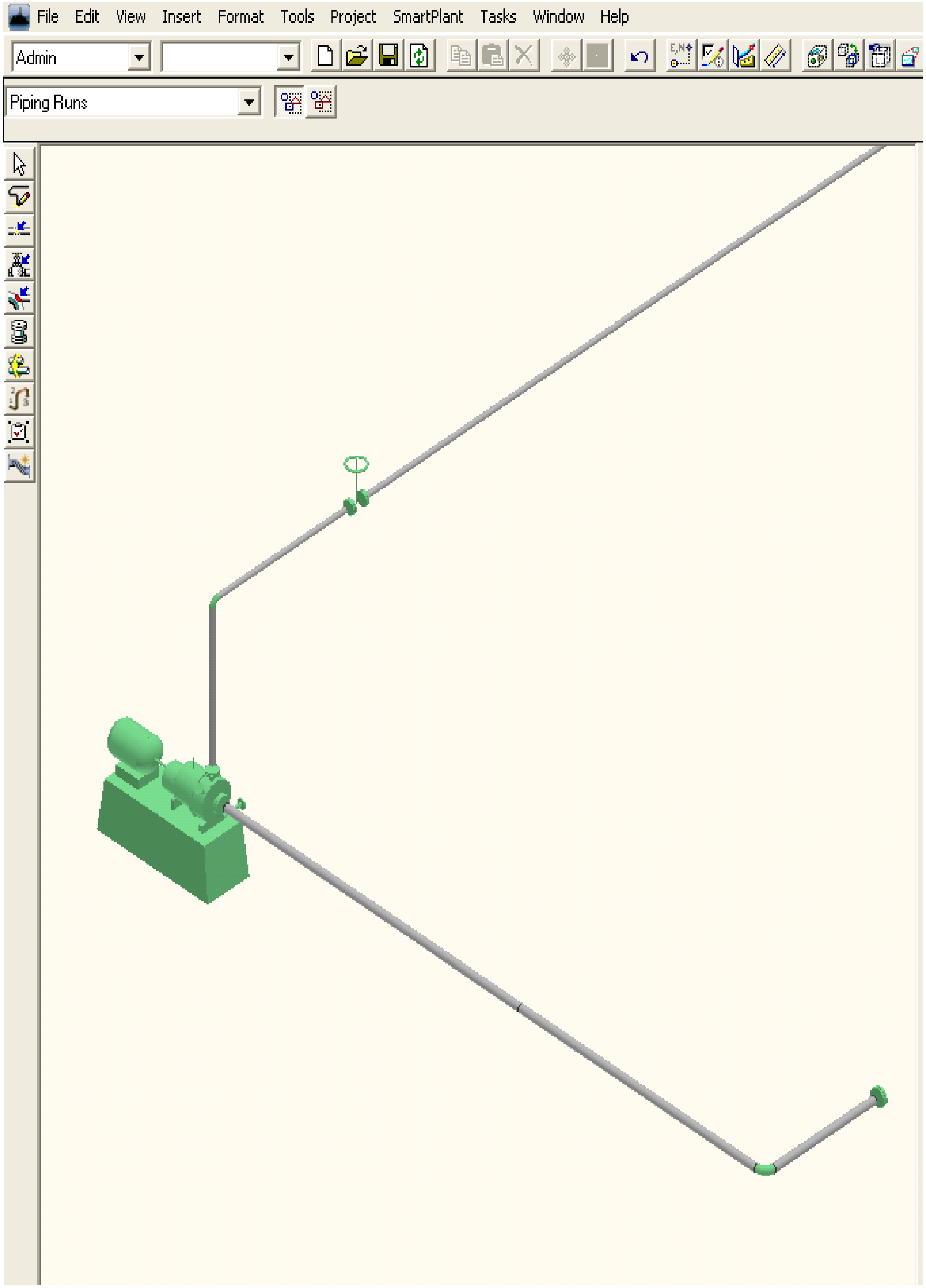
1. Select **SmartPlant > View P&ID**.
2. Select **PID1008**, and click **Open**.



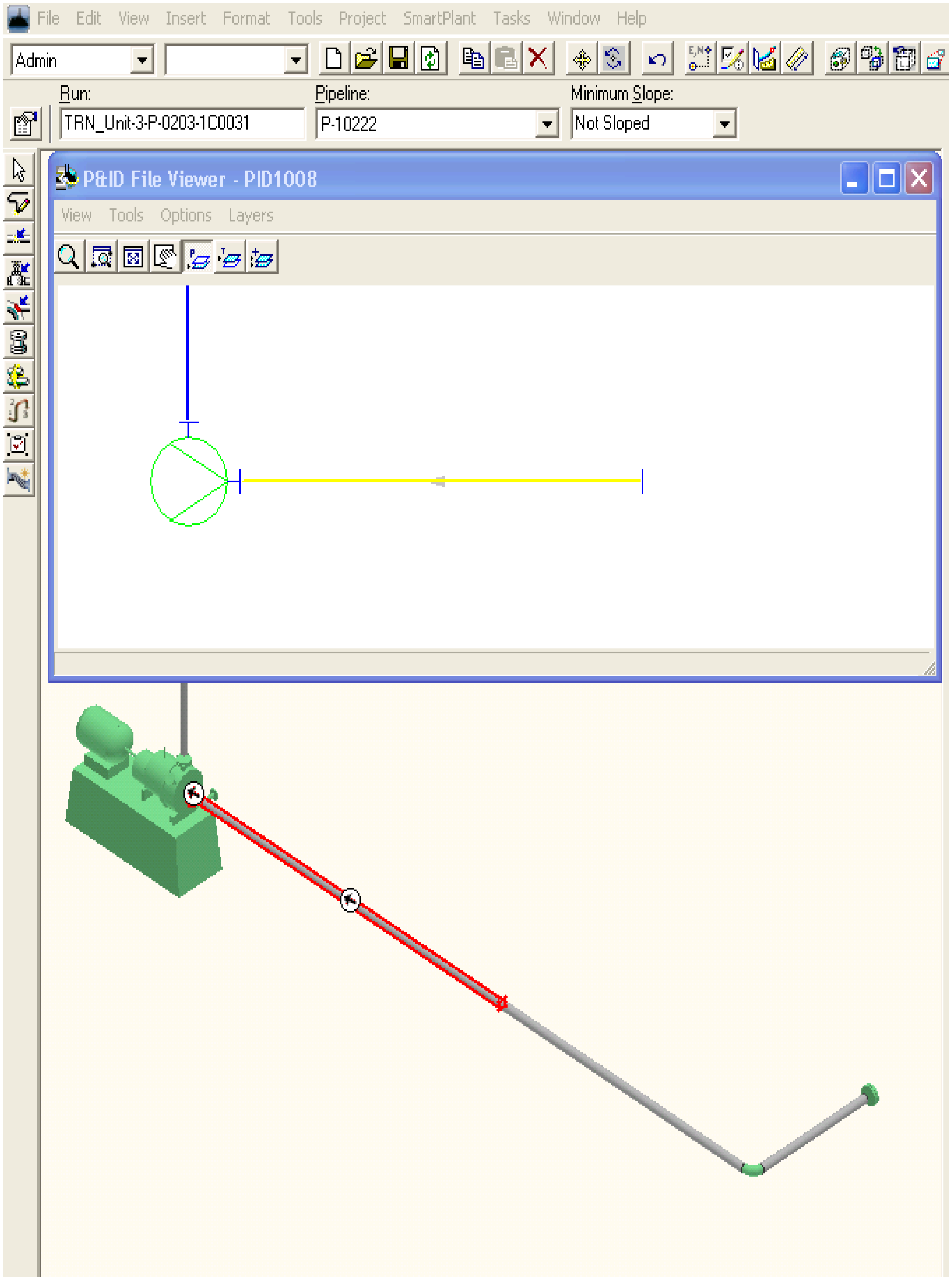
NOTE This is the PID we will be working with during this exercise, and the same one we used earlier.



3. The workspace you defined earlier should look like the one below.

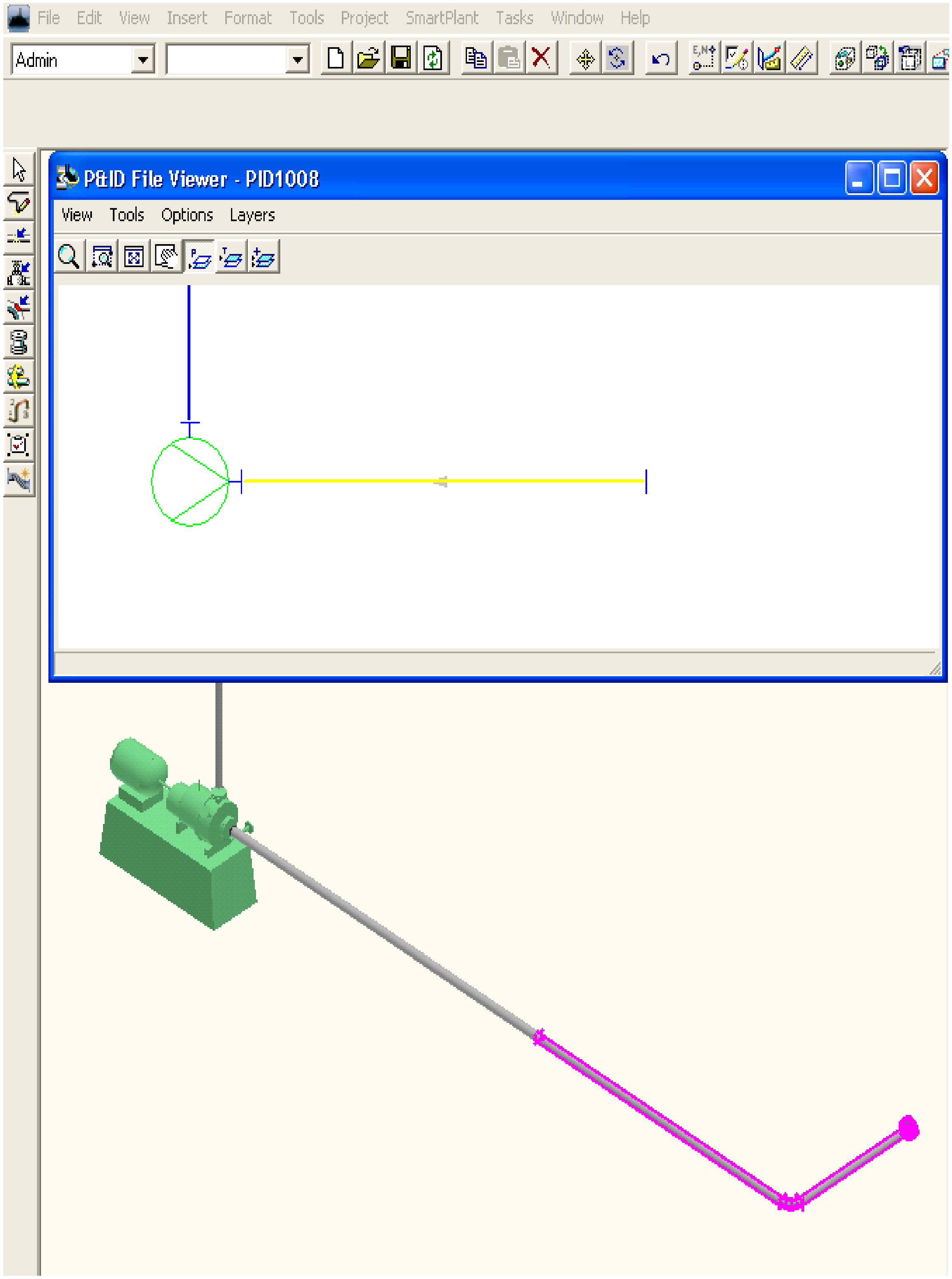

File
Edit
View
Insert
Format
Tools
Project
SmartPlant
Tasks
Window
Help
Admin
Piping Runs

4. Select the following **PipeRun and Correlate with Design Basis** and then select the pipe run shown in the P&ID Viewer:

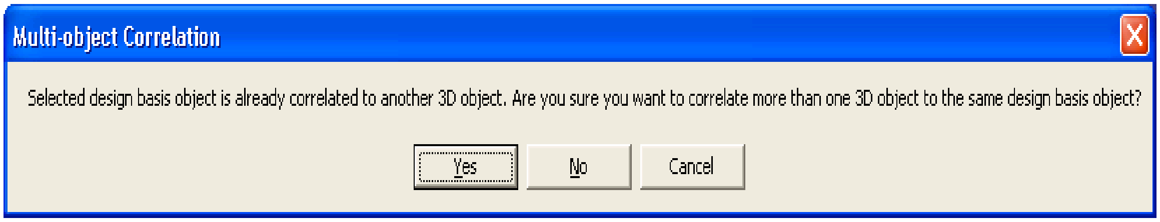


5. When you see the **Correlate with design basis** window, click **Update**.

6. Select the pipe run beside the one that was selected earlier and select **Correlate with Design Basis** again and select the same P&ID pipe run that was selected in the previous step:


File Edit View Insert Format Tools Project SmartPlant Tasks Window Help
Admin
P&ID File Viewer - PID1008
View Tools Options Layers

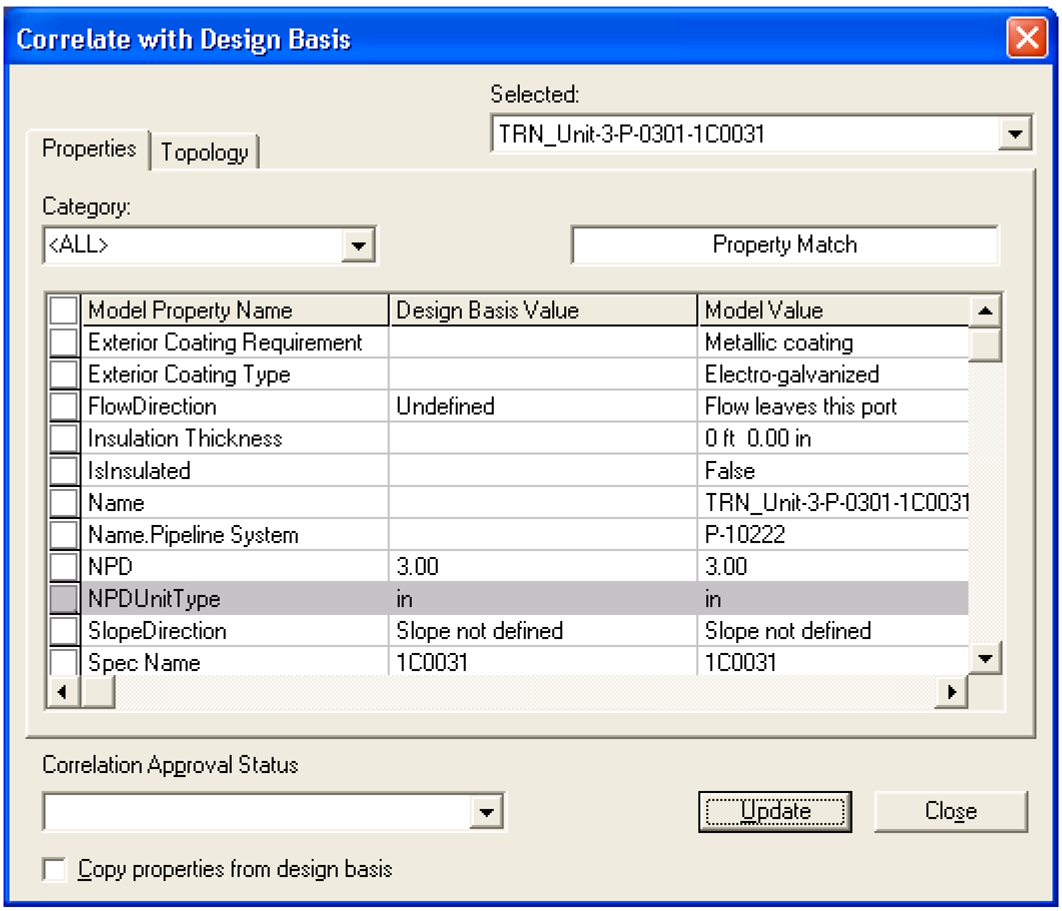
A message box appears.



7. Click **Yes**.

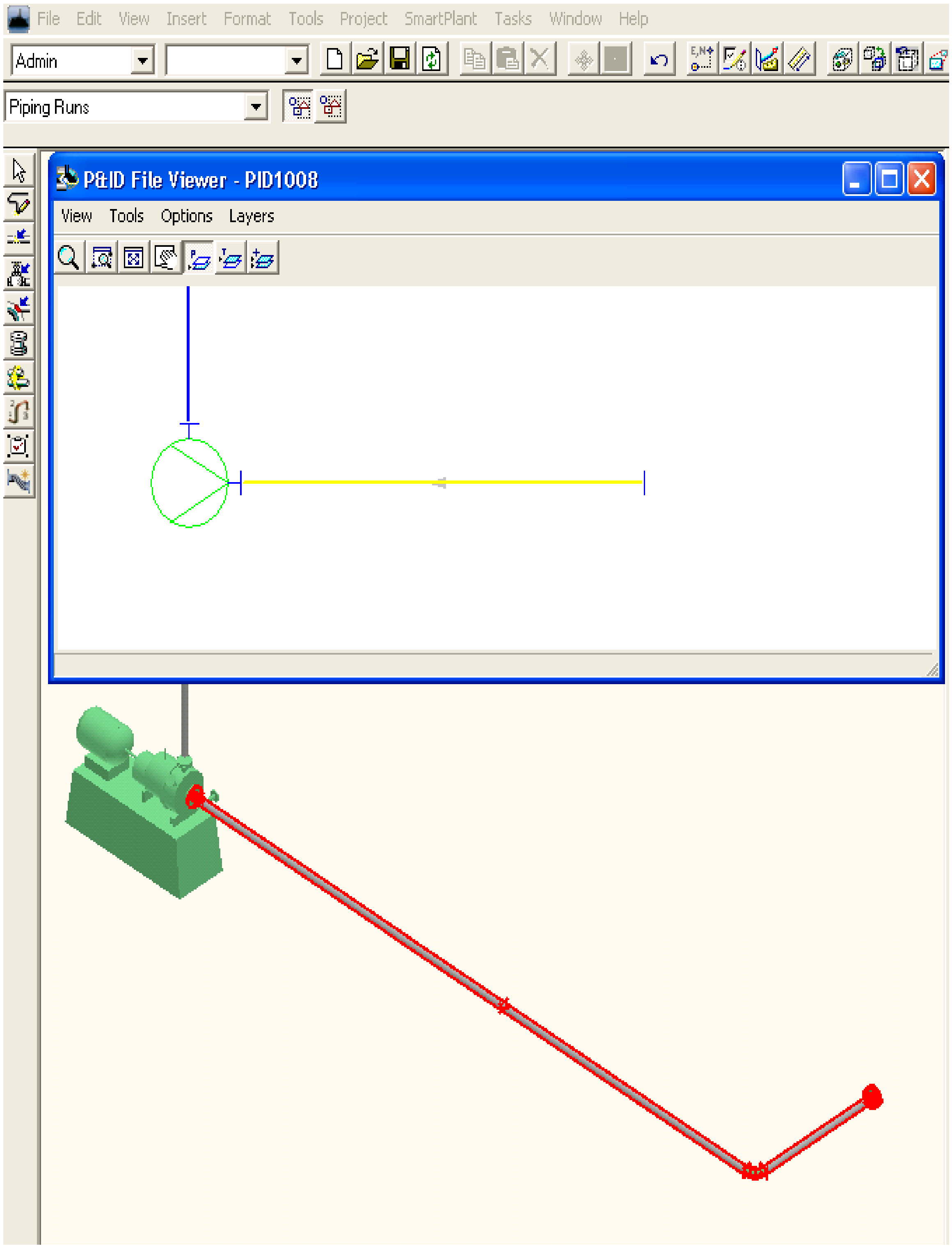
   *The **Correlate with Design Basis** dialog appears again.*

8. Click **Update** as shown below:



9. When you see the **Compare Design Basis** dialog, click **Update**.

10. Now when you open the P&ID and select the one pipe run that we correlated to multiple Smart 3D pipe runs, you will see two Smart 3D highlighted, as shown below:



Index