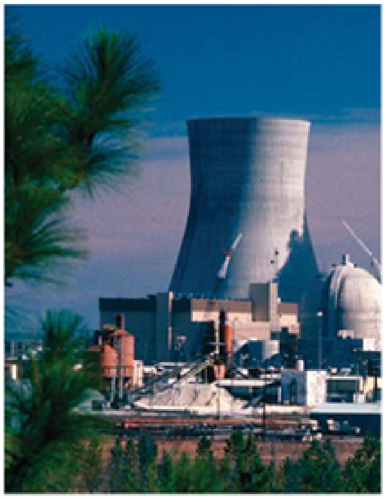
SmartPlant 3D

Tutorials for Piping

Process, Power & Marine



INTERGRAPH

Version 2009 SP1 April 2009 DSP3D-TP-100011A

Table of 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:

- SP3D Overview

Overview:

You use the **Piping** task of SP3D 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 SP3D, click the **Task** > **Piping** command.

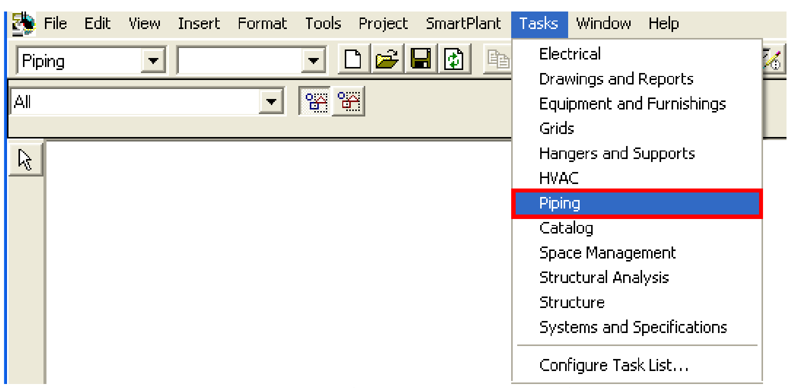


Figure 1: Tasks > Piping Command

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. Figure 2 shows a piping feature model and the relationship among the features, which represent a section of the piping system.

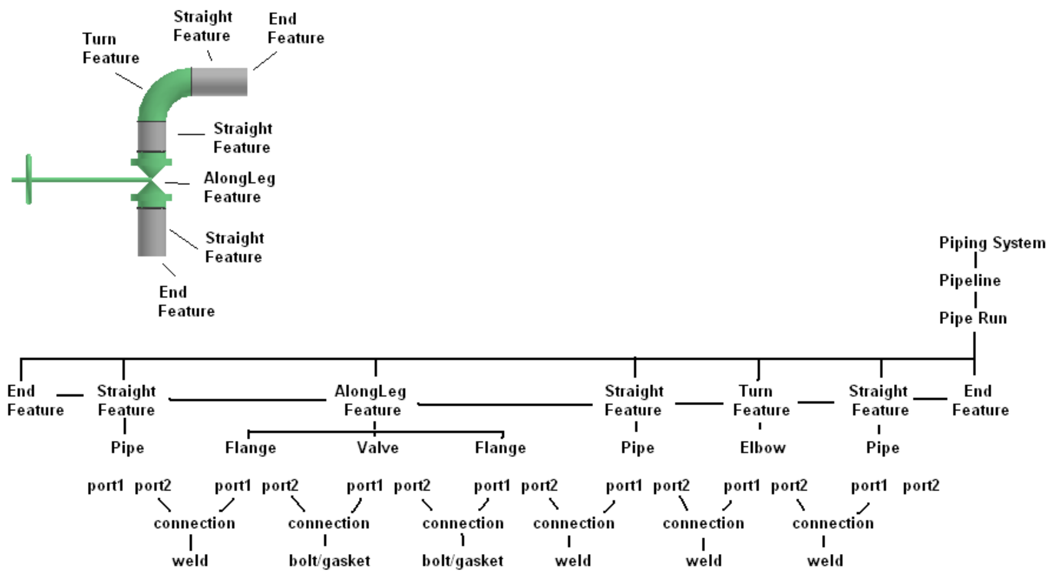


Figure 2: Piping Feature Model

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. Figure 3 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.

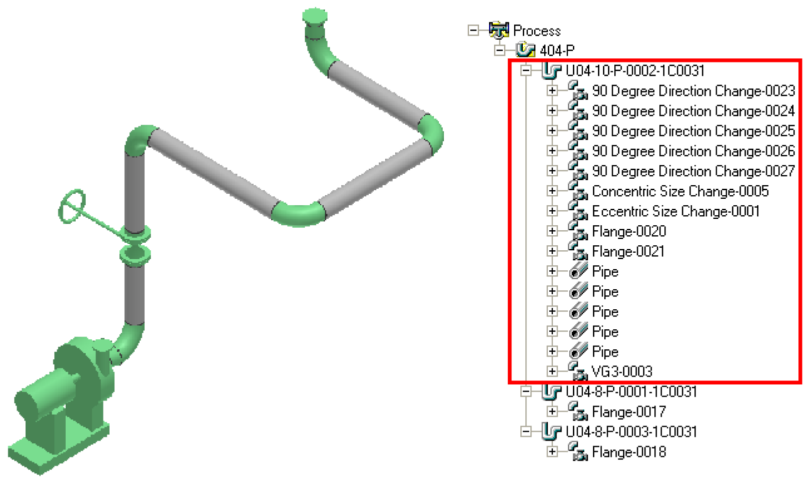


Figure 3: Pipe Parts in the System Hierarchy

- **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 the Piping Task:

When in the **Piping** task, you can perform the following tasks by using the commands available on the vertical toolbar, as shown in Figure 4.

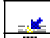
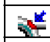
Symbol	Command Name	Function
	Select	Selects objects in the model
	Route Pipe	Places pipe runs in the model
	Insert Split	Places welds or takedown joints in pipe runs
	Insert Component	Places pipe components, instruments, and specialty items in pipe runs
	Insert Tap	Places taps in the selected component
	Generate Spools	Generates piping spools to be fabricated as a unit
	Create Penetration Spools	Generates penetration spools for pipes with penetration plates placed in the model
	Sequence Objects	Renames the welds on pipelines or spools 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

Figure 4: Piping Commands on the Vertical Toolbar

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, SP3D 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.

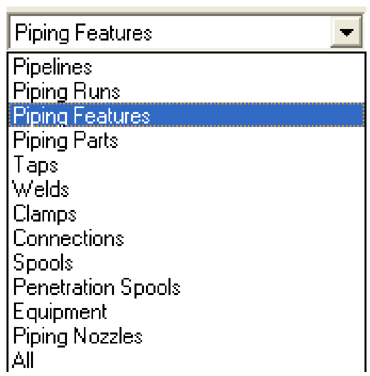


Figure 5: Locate Filter Drop-Down List in the Piping Task

Quiz:

1. What are pipe runs?
2. What are pipe features?
3. Define the following:
    - Piping parts
    - Pipe ports
    - Connections

Session 2: Routing Pipes

Objectives:

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 Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview

Overview:

In SP3D, 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 SP3D.

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 in Figure 1.

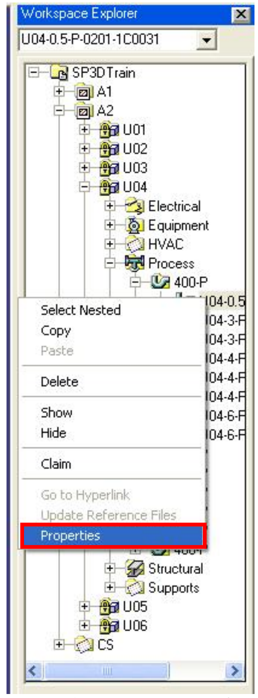


Figure 1: Accessing the Properties of a Piping System

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

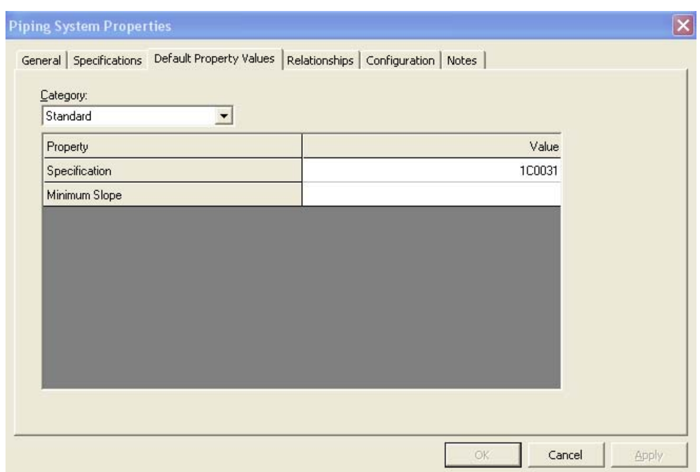


Figure 2: Piping System Properties Dialog Box

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.

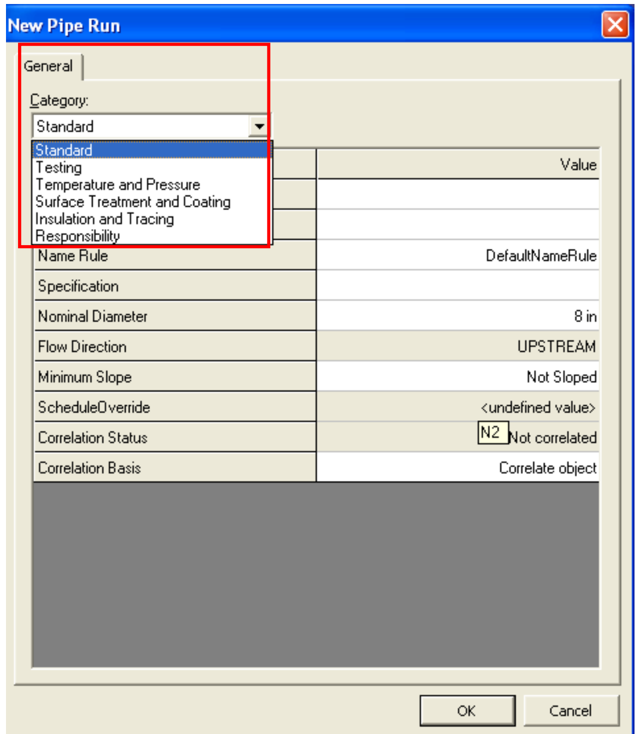


Figure 3: New Pipe Run Dialog Box

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 in Figure 3.

Pipe run properties are divided into different categories as follows:

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

Standard Category

Within the **Standard** category, you can define the following values, as shown in Figure 4:

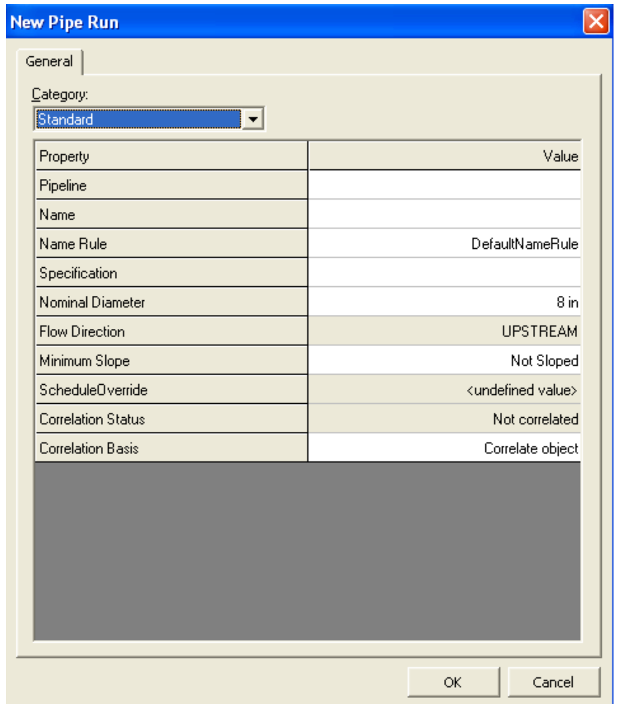


Figure 4: Standard Category in the New Pipe Run Dialog Box

- **Pipeline** - specifies the pipeline system to which the pipe run will belong.
- **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** - Specifies the naming rule to name the pipe run. You can select one of the listed rules or specify the pipe run name in the **Name** box.
- **Specification** - Selects the pipe specification which will control the pipe run.
- **Nominal Diameter** - Selects the NPD to use for this 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 be displayed in a session file basis by using the **Tools > Options** command on the **Units of Measure** tab.
- **Flow Direction** - Selects the direction of flow 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.
- **Minimum Slope** - Specifies the slope for the pipe run. You can specify the slope as a ratio, as a percentage, or in degrees.
    - As a ratio, if a run drops ¼ in for every foot of horizontal distance, specify the slope as **¼ inch/1 ft**.
    - As a percentage, if a run drops 1 in for every 10 inches of horizontal distance, specify the slope as **10%**.
    - In degrees, a 5 degree slope can be specified as **5 deg**.
- **Schedule Override** - Specifies 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** - Specifies if the pipe run needs to be correlated to a P&ID pipe run.

Testing Category

Within the **Testing** category, you can define the following values, as shown in Figure 5:

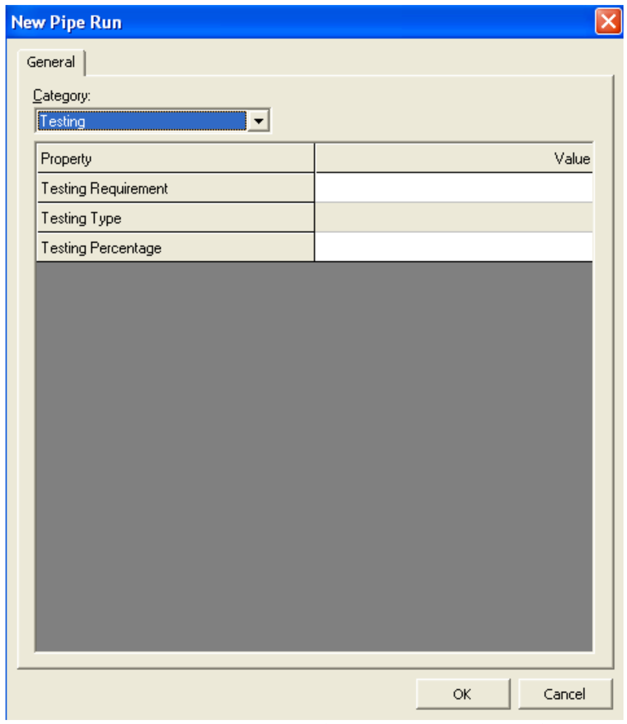


Figure 5: Testing Category in the New Pipe Run Dialog Box

- **Testing Requirements** - Specifies whether non-destructive testing of welds is required.
- **Testing Type** – Selects the type of non-destructive weld testing for the welds.
- **Testing Percentage** - Specifies the percentage of the welds on the pipe run that needs to be tested. This option is available only if you are viewing pipe run properties.

Temperature and Pressure Category

Within the **Temperature and Pressure** category, you can define the following maximum and minimum temperatures and pressures for the object, as shown in Figure 6:

- Design Temperature or Pressure
- Operating Temperature or Pressure
- Testing Temperature or Pressure

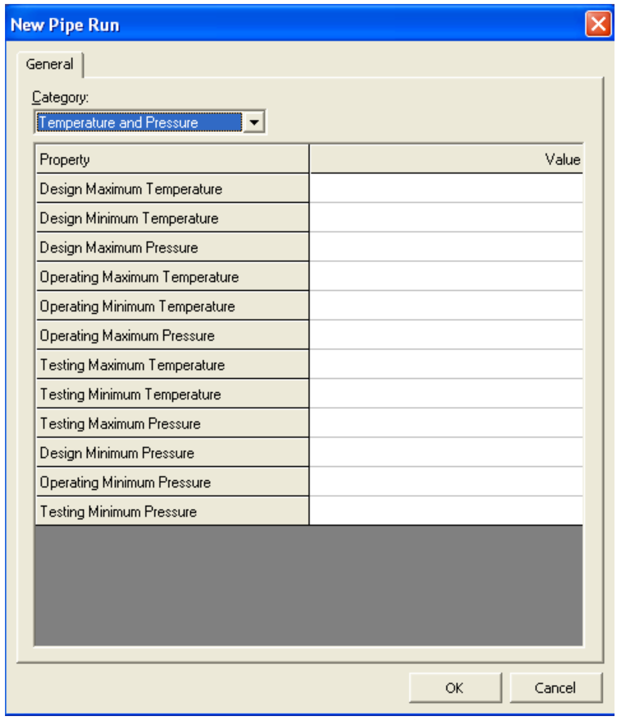


Figure 6: Temperature and Pressure Category in the New Pipe Run Dialog Box

Temperature and pressure values are controlled by a **Project** option. The **Project** option can be set to **On,** and you should key in the temperature and pressure value for all the pipe runs. This option can be set to **Off**; in which case you will receive a warning "The temperature and pressure values are ignored."

Surface Treatment and Coating Category

Within the **Surface Treatment and Coating** category, you can define the following properties for the object, as shown in Figure 7.

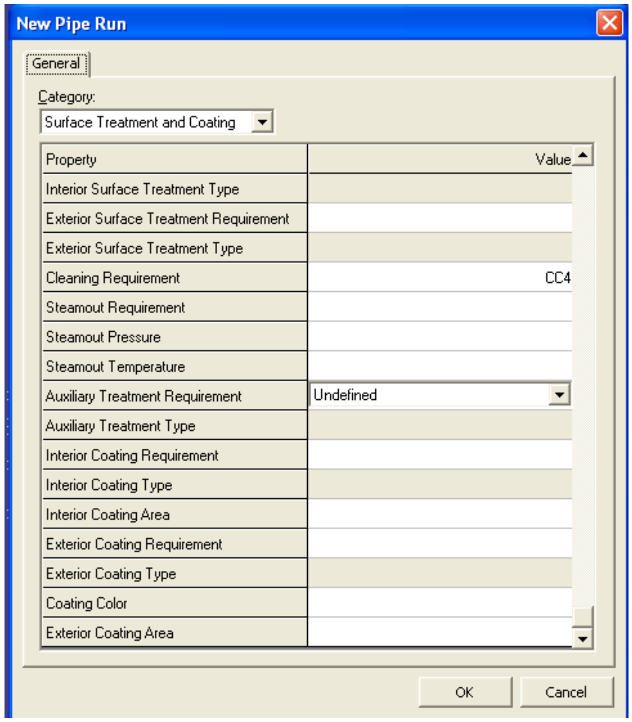


Figure 7: Surface Treatment and Coating Category in the New Pipe Run Dialog Box

- Requirement and Type of Interior and Exterior Surface Treatment
- Requirement for cleaning, Steamout, Auxiliary Treatment, Interior and Exterior Coating
- Temperature and pressure for Steamout, if required
- Type of auxiliary treatment, if required
- Type and area of Interior and Exterior Coating, if required
- Color of Coating

Insulation and Tracing Category

Within the **Insulation and Tracing** category, you can define the specifications for insulation and heat tracing, as shown in Figure 8. For example, you can specify the purpose, thickness, and temperature of the insulation and the material to be used for it. You can also specify the type and medium of heat tracing for the pipe run.

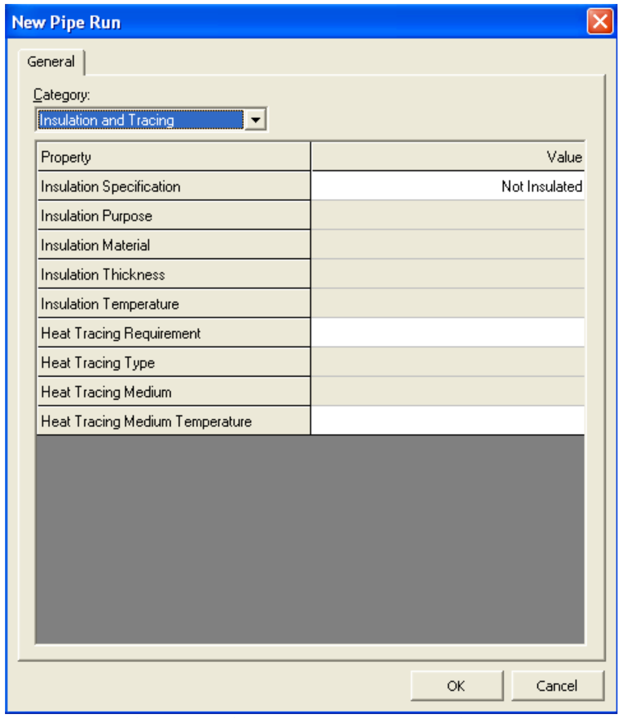


Figure 8: Insulation and Tracing Category in the New Pipe Run

Responsibility Category

Within the **Responsibility** category, you can select the party responsible for the following tasks, as shown in the Figure 9:

- Cleaning Responsibility
- Design Responsibility
- Fabrication Responsibility
- Installation Responsibility
- Painting Responsibility
- Requisition Responsibility
- Supply Responsibility
- Testing Responsibility

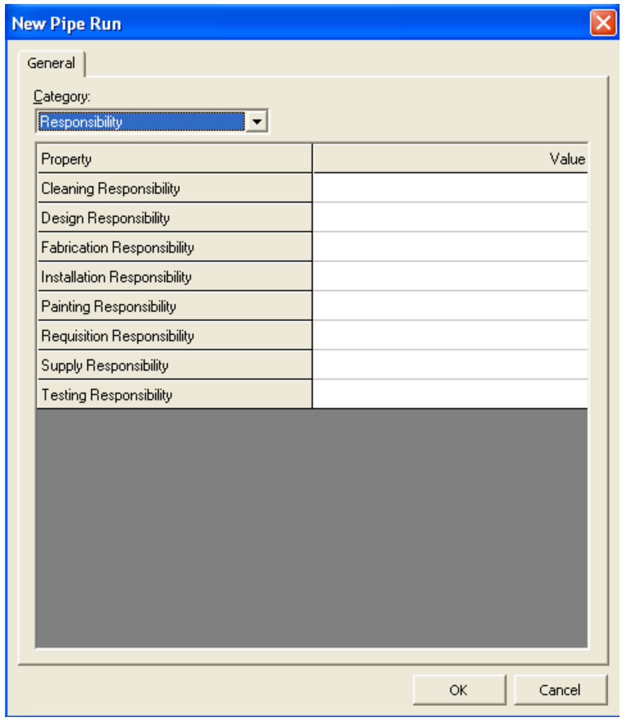


Figure 9: Responsibility Category in the New Pipe Run Dialog Box

This session will cover the procedure for routing pipelines in SP3D.

Before going through this procedure and the subsequent SP3D Piping sessions, define your workspace to include all objects in the SP3Dtrain model database:

1. Start **SP3D** software by using the command **Start > Programs > Intergraph SmartPlant 3D > SmartPlant 3D**.

2. In the **New** dialog box, select the **EnglishUnits** or **MetricUnits** template and then, click **OK**.

3. Click 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 SP3D 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 the **Delete** command 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 Figure 10 in the graphic view.

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**

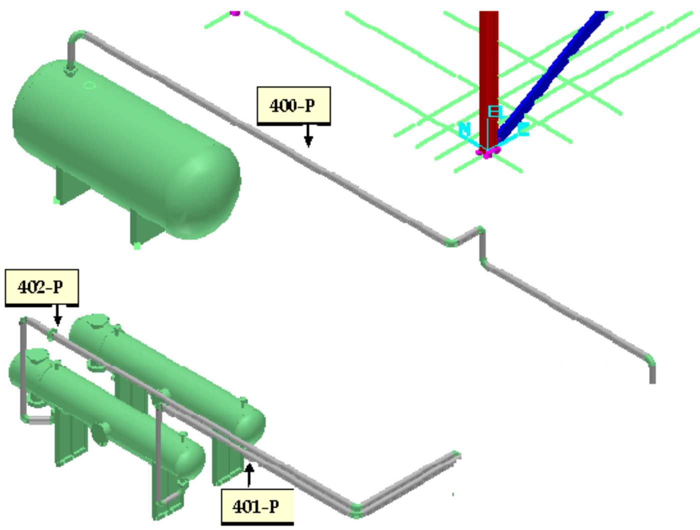


Figure 10: Routed Pipelines 400-P, 401-P, and 402-P

Before beginning the 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 the **Route Pipe** button on the vertical toolbar.

Figure 11: Route Pipe Button

2. Select the nozzle **40V-101/A** as the starting point, as shown in Figure 12. The starting location can be an existing pipe run, a nozzle, a point in space or a piping component.

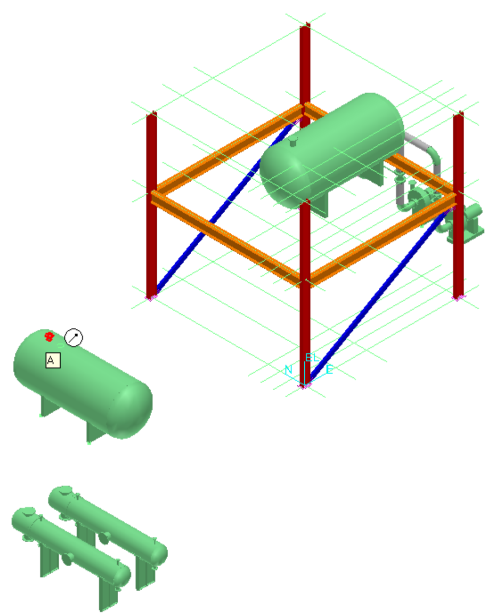


Figure 12: Selecting the Starting Point

3. The **New Pipe Run** dialog box appears. Select the **More…** option in the **Pipeline** drop-down list in the dialog box.

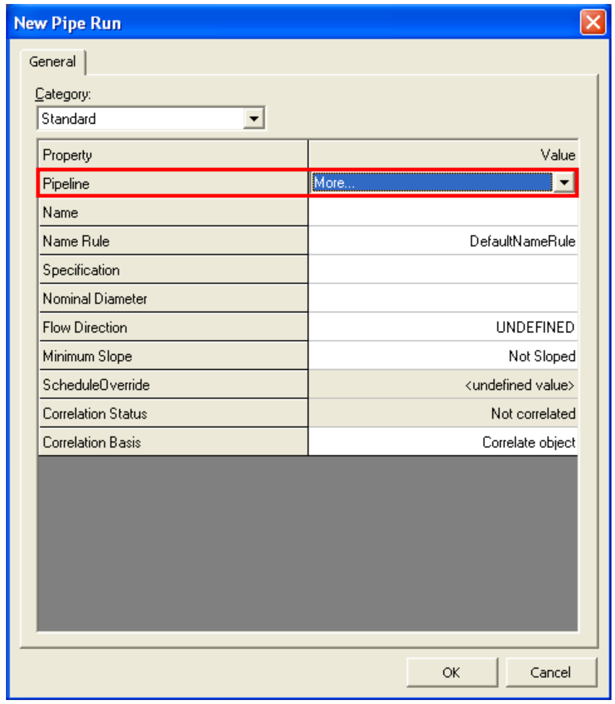


Figure 13: New Pipe Run Dialog Box

4. The **Select System** dialog box appears. 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, as shown in Figure 14. Then, click **OK**.

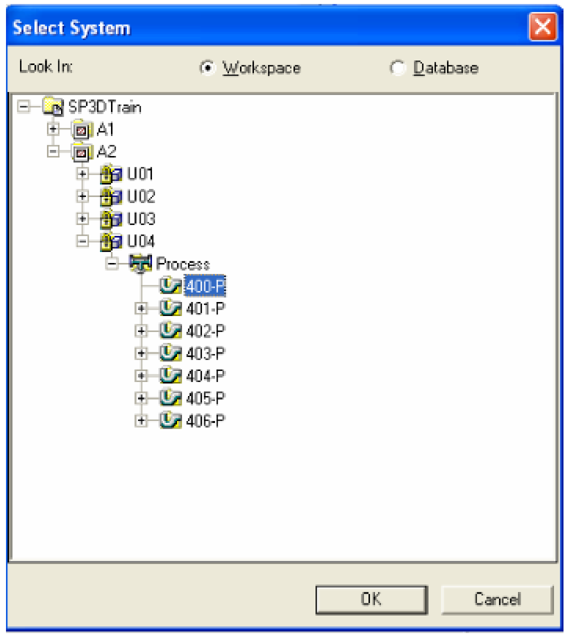


Figure 14: Selection of Pipeline

SP3D 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.

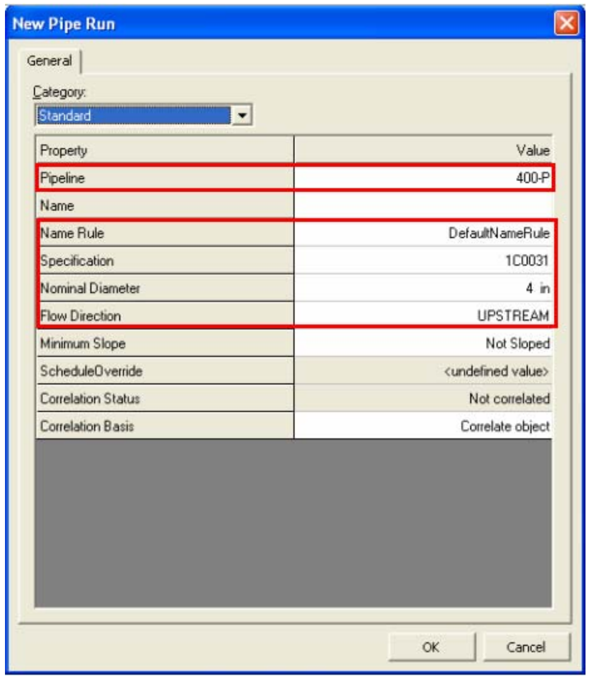


Figure 15: Standard Properties of a Pipeline

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.

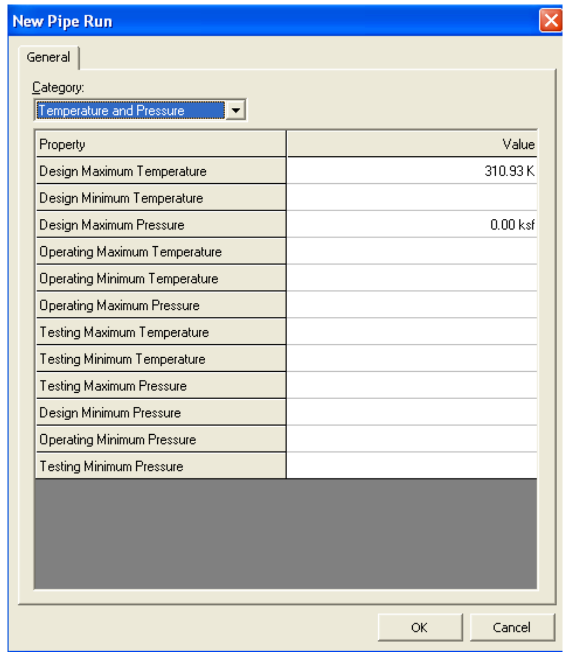


Figure 16: Temperature and Pressure Category in the New Pipe Run Dialog Box

6. Click **OK**.

7. SP3D displays a warning **Temperature/Pressure** note on a message box. Check the **Do not show this message again** option and click **OK**.

   An outline of a pipe appears in the graphic view. SP3D locks the angle at **0 deg**. As a result, you can only route the pipe in the upward direction, as shown in the Figure 17.

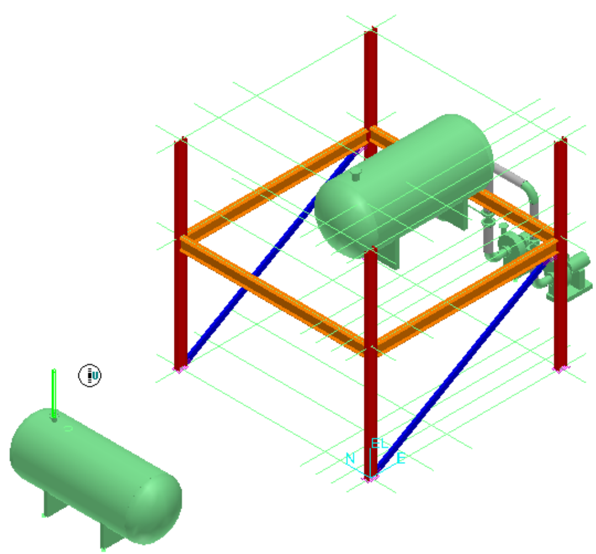


Figure 17: Pipe Connected to a Vertical Nozzle

8. 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.

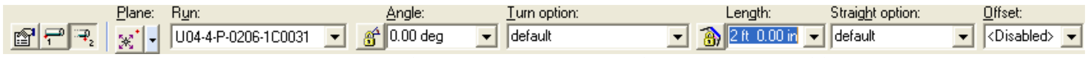


Figure 18: Length Drop-Down List on the Route Pipe Ribbon

9. Position the cursor on top of the pipe. SmartSketch will display a **U** glyph. This glyph depicts that you are going in Up-Down plane, as shown in Figure 19.

   The SmartSketch glyph **U** indicates that the projection for the routing of a pipe is going in the vertical direction.

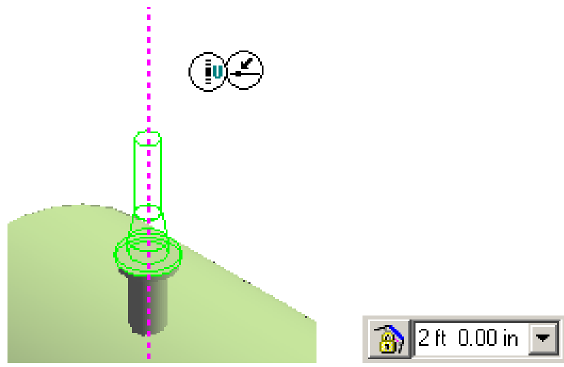


Figure 19: Routing Pipe in the Vertical Direction

10. **Left Mouse 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.

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

12. Position the cursor in the east direction until SmartSketch displays the **E** glyph, as shown in Figure 20, which indicates that you are in the East-West plane.

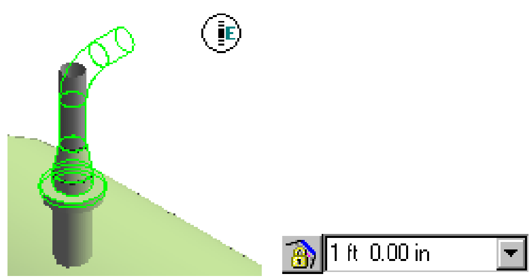


Figure 20: Routing Pipe in the East-West Plane

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

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

15. Position the cursor in the south direction until SmartSketch displays the **N** glyph, as shown in Figure 21, which depicts that you are in North-South plane.

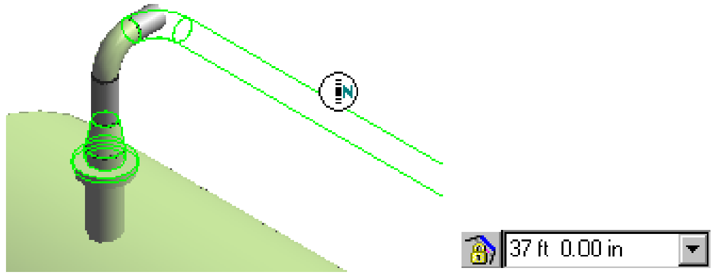


Figure 21: Routing Pipe in the North-South Plane

16. 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.

17. Key in **3 ft** in the **Length** drop-down list on the **Route Pipe** ribbon.

18. Position the cursor in the east direction until SmartSketch displays the **E** glyph, as shown in Figure 22.

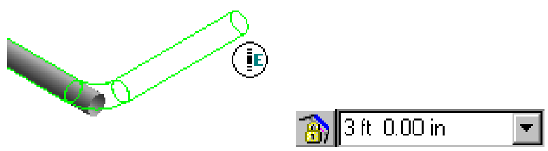


Figure 22: Routing Pipe in East Direction

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

20. Key in **3 ft** in the **Length** drop-down list on the **Route Pipe** ribbon.

21. Position the cursor down until SmartSketch displays the **U** glyph, as shown in Figure 23.

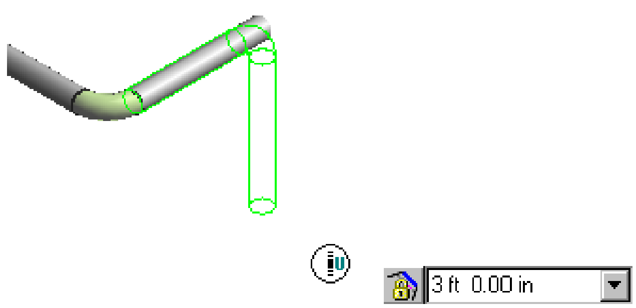


Figure 23: Routing Pipe in the Vertical Direction

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

23. 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, as shown in Figure 24.

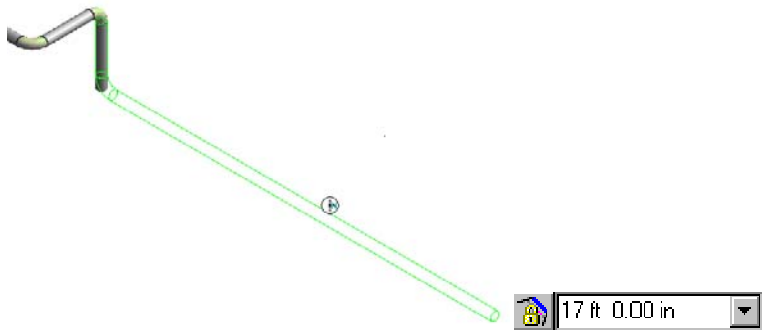


Figure 24: Routing Pipe in the North Direction

24. Click in the graphic view to accept the placement of this pipe.
25. 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.

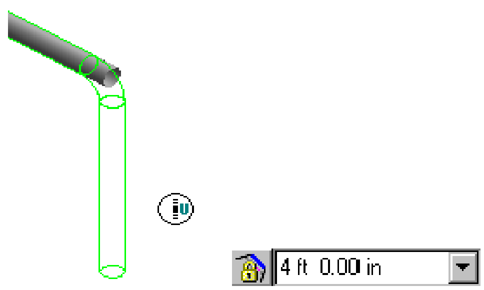


Figure 25: Routing Pipe in the Vertical Direction

26. Click in the graphic view to accept the placement of this pipe.
27. Right-click in the graphic view to terminate the **Route Pipe** command. The routed pipeline **400-P** should resemble Figure 26.

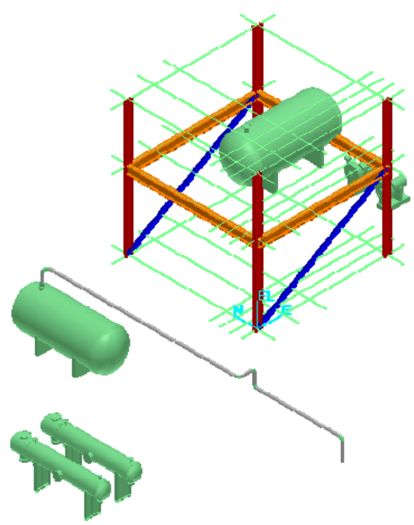


Figure 26: Routed 400-P Pipe

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 in Figure 27.

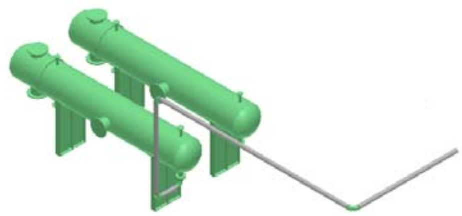


Figure 27: Routed 401-P Pipeline

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

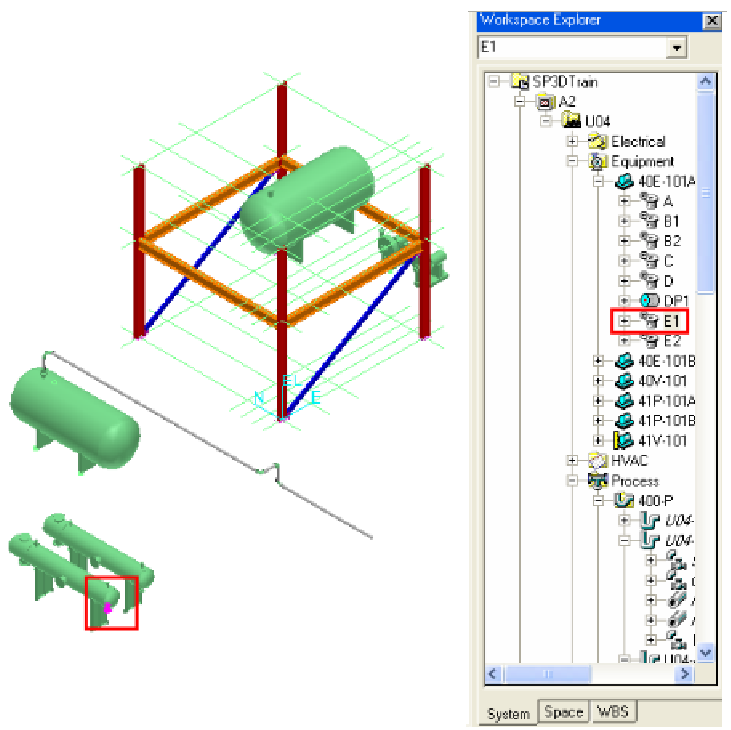


Figure 28: Locating the Nozzle 40E-101A/E1 in the Workspace Explorer

29. Click the **Route Pipe** button on the vertical toolbar.

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

31. The **New Pipe Run** dialog box appears. Set the following parameters in the **New Pipe Run** dialog box, as shown in Figure 29, and click **OK**:

Pipeline: 401-P

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

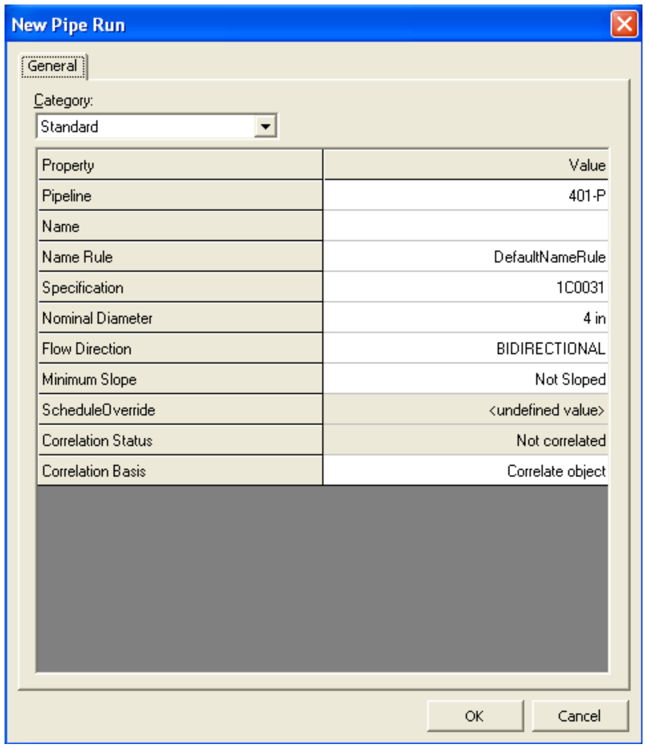


Figure 29: New Pipe Run Dialog Box

32. An outline of a pipe will appear in the graphic view. 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, as shown in Figure 30.

Figure 30: Length and Angle Specifications on the Route Pipe Ribbon

33. 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.

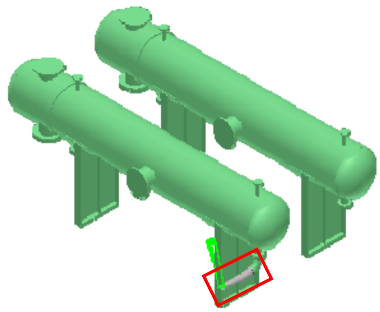


Figure 31: Routed Pipe in the East Direction

34. Now, on the **Route Pipe** ribbon, key in **9 ft** in the **Length** drop-down list and use SmartSketch to locate the **U** glyph. Click in the graphic view to place the pipe, as shown in Figure 32.

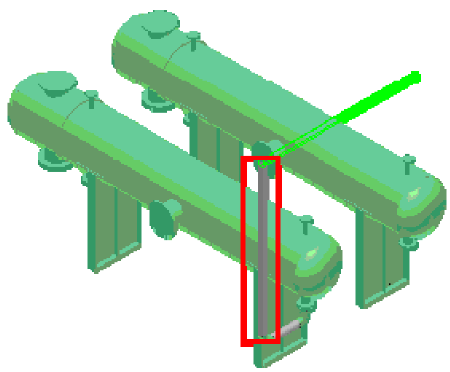


Figure 32: Routed Pipe in the Vertical Direction

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

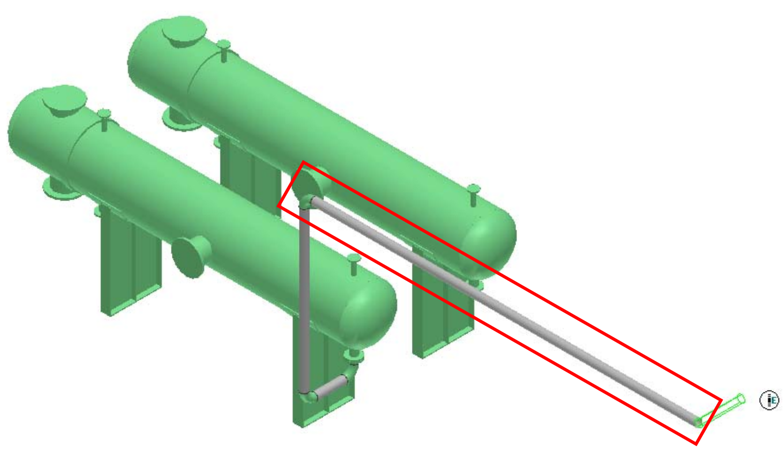


Figure 33: Placed Pipe

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

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

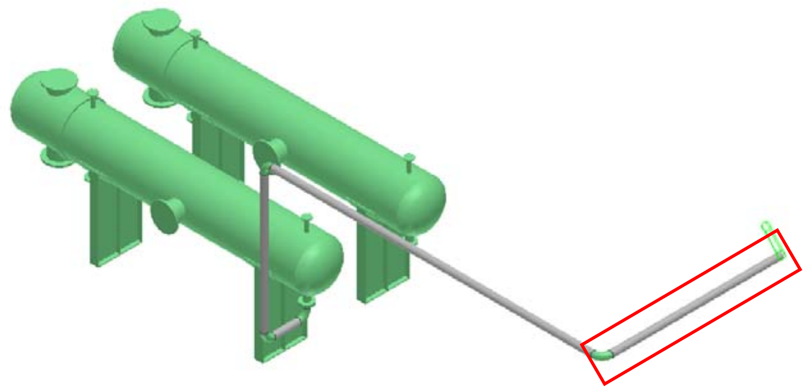


Figure 34: Routed 402-P Pipeline From the Nozzle 40E-101A/E1

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 in Figure 35.

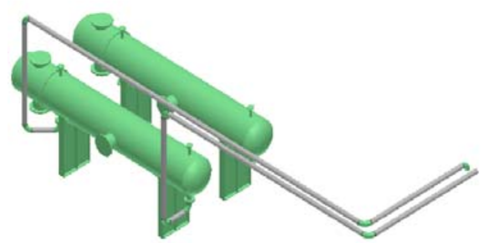


Figure 35: Routed 402-P Pipeline From the Nozzle 40E-101A/E2

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

39. Click the **Pinpoint** button on the **Common** toolbar and then click the **Relative Tracking** button on the **PinPoint** ribbon, as shown in Figure 36.

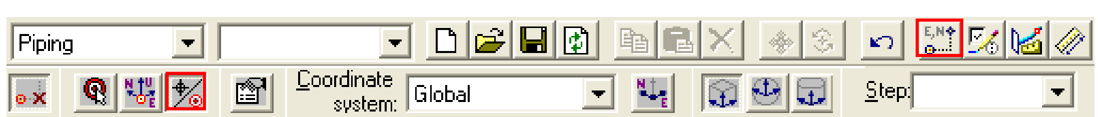


Figure 36: PinPoint and Relative Tracking Buttons

40. Click the **Route Pipe** button on the vertical toolbar and select the equipment nozzle **E2**, as shown in Figure 37.

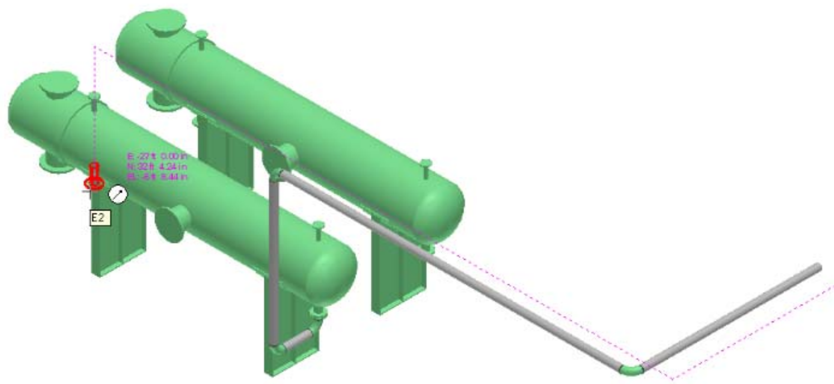


Figure 37: Nozzle E2 in the Graphic View

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

    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.

Figure 38: East and North Coordinates on the PinPoint Ribbon

41. Notice the Pinpoint target is automatically placed, as shown in Figure 4, when you identify the equipment nozzle **E2**. Select the **Zoom Window** command to ensure target has been placed correctly and then **Left-Click** in the graphic view to place the pipe.

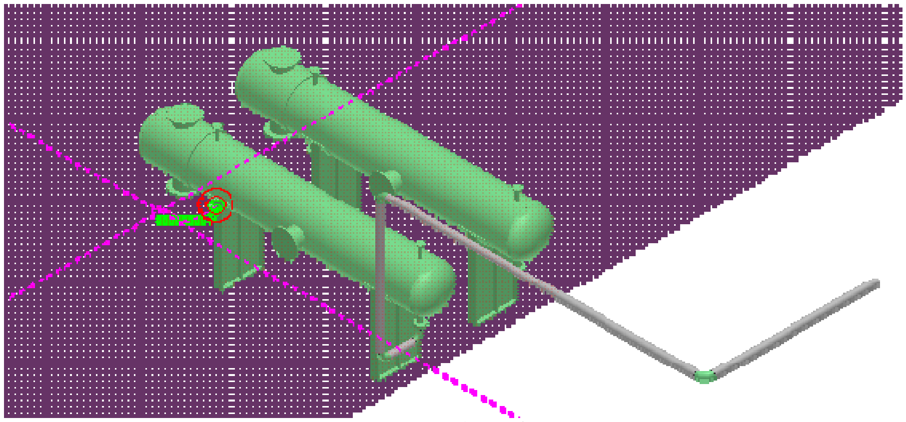


Figure 39: Outline of a Pipe

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

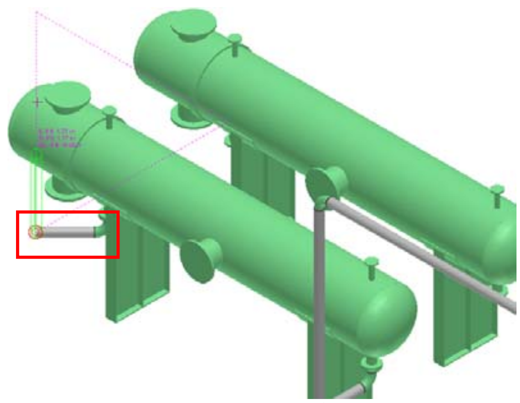


Figure 40: Routed Pipe

45. Now, on the **Route Pipe** ribbon, key in **9 ft** in the **Length** drop-down list and use SmartSketch to locate the **U** glyph. Click in the graphic view to place the pipe, as shown in Figure 41.

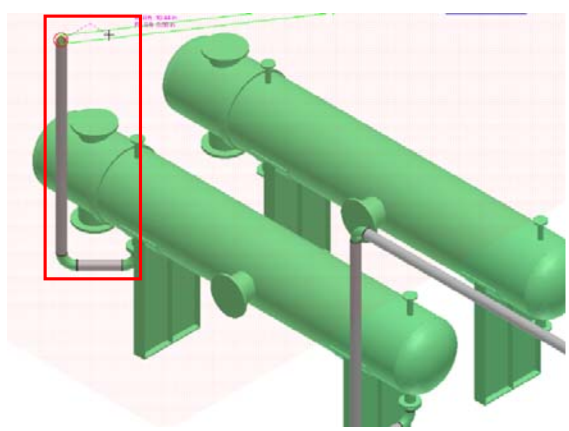


Figure 41: Routed Pipe in the Vertical Direction

46. 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. Click in the graphic view to place the pipe, as shown in Figure 42.

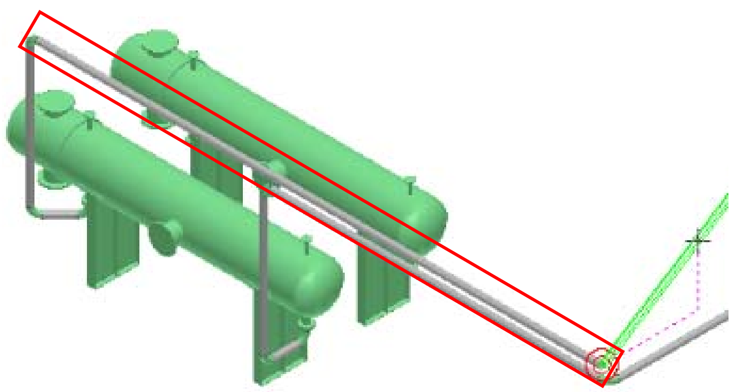


Figure 42: Routed Pipe in the North Direction

47. Now, on the **Route Pipe** ribbon, key in **10 ft** in the **Length** drop-down list and use SmartSketch to locate the **E** glyph. Click in the graphic view to place the pipe, as shown in Figure 43.

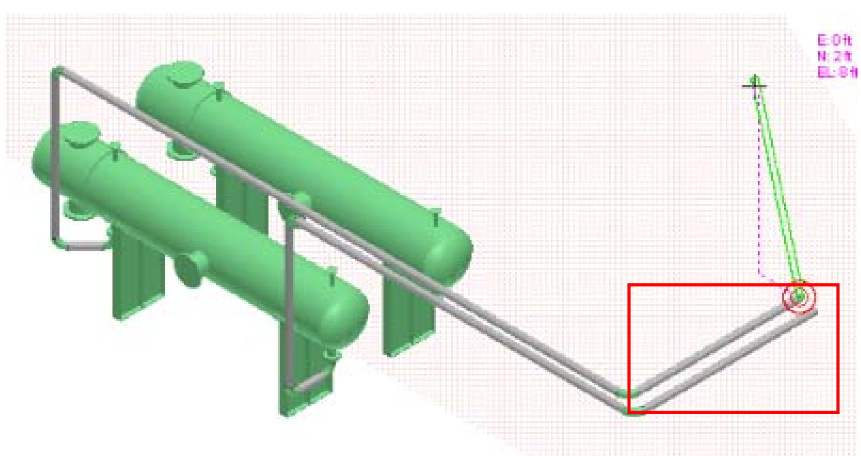


Figure 43: Routed Pipe in the East Direction

48. 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. Click in the graphic view to place the pipe, as shown in Figure 44.

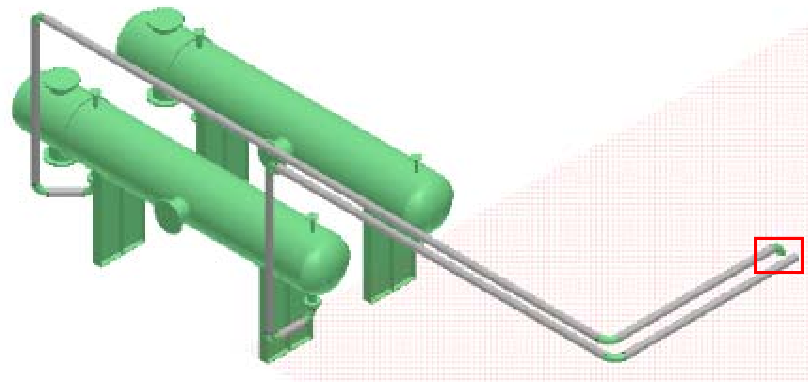


Figure 44: Routed Pipe 402-P

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

For additional information about default run properties, refer to the *New Pipe Run Dialog Box* topic in the user guide *PipingUsersGuide.pdf*.
For information about routing pipes, refer to the *Routing a Pipe Run: An Overview* topic in the user guide.

Session 3: Inserting Components in a Pipe Run

Objective:

By the end of this topic, 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.

Prerequisite Sessions:

- SP3D Overview
- SP3D 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.

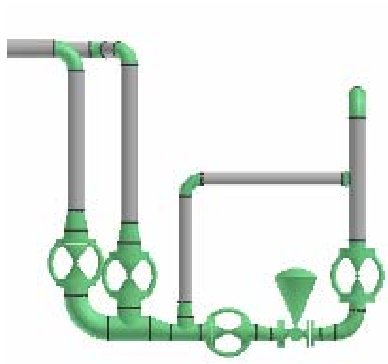


Figure 1: Placed Piping Components

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.

Steps for Inserting Components and Routing a Pipe Run:

The following example shows a typical workflow to insert components in a pipe run.

Exercise Objective: 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 by 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 the highlighted area in Figure 2.

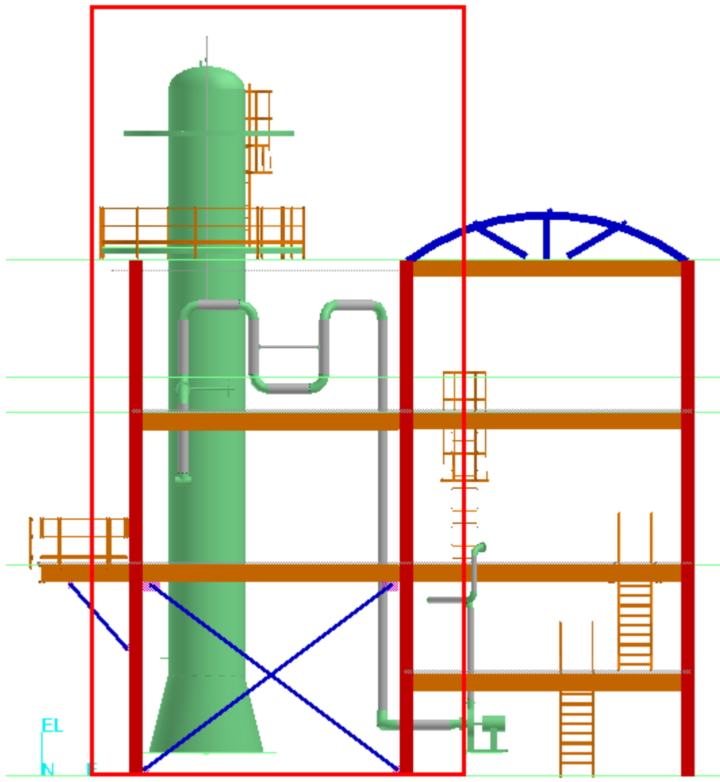


Figure 2: Final Output: Pipe Routed with Components

Before beginning the 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. Activate the **PinPoint** command by clicking the **PinPoint** button on the **Common** toolbar and then click the **Relative Tracking** on the **PinPoint** ribbon.

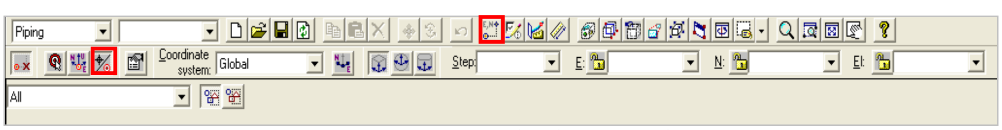


Figure 3: PinPoint Button and Relative Tracking Option

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

Inserting a Flange at an Equipment Nozzle

3. Now, click the **Insert Component** button on the vertical toolbar.

Figure 4: Insert Component Button on the Vertical Toolbar

4. Use the SmartSketch options of SP3D to locate the suction nozzle of **Pump P-101** and click the nozzle.

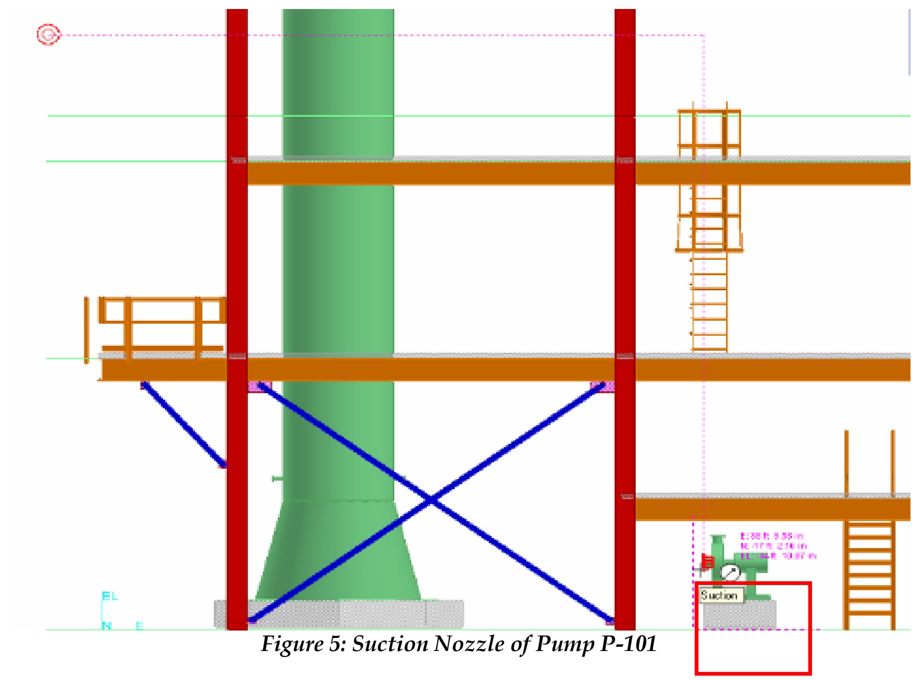


Figure 5: Suction Nozzle of Pump P-101

5. The **New Pipe Run** dialog box appears. Select **300-W** in the **Pipeline** field, as shown in Figure 6.

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

ScheduleOverride: <undefined value>
Correlation Basis: Correlate object

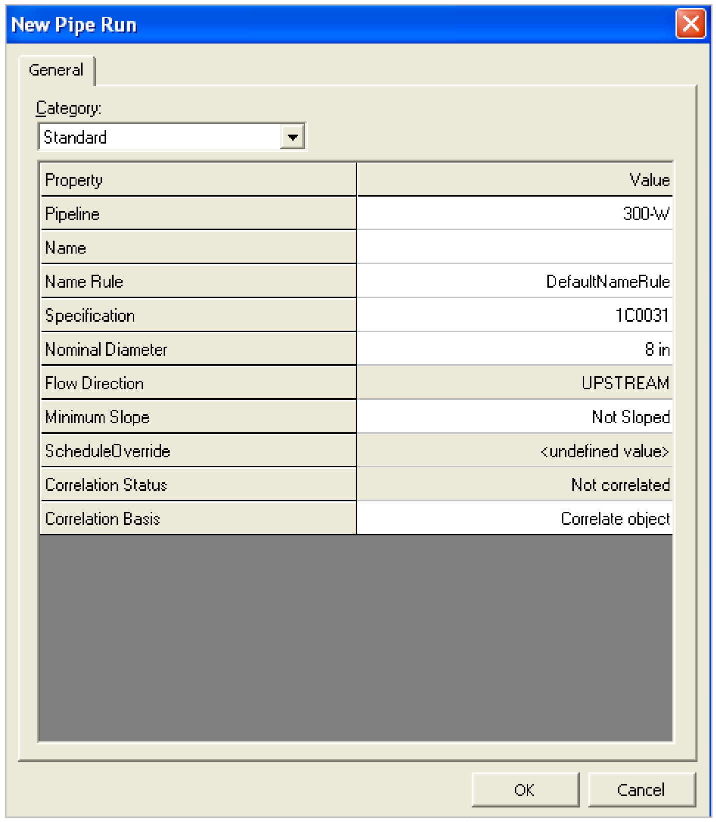


Figure 6: New Pipe Run Dialog Box

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

   Placement operations are specification-driven. SP3D 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.

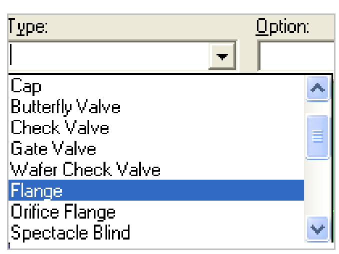


Figure 7: Type Drop-Down List

8. You will now see an outline of a flange on the suction nozzle of **Pump P-101**. Click **Finish** on the **Insert Component** ribbon to place the flange.

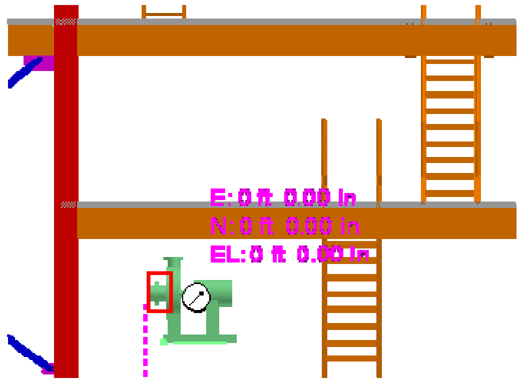


Figure 8: Placed Flange

9. Now, select the **Eccentric Size Change** option in the **Type** drop-down list on the **Insert Component** ribbon. This would now require you to place a new pipe run. So, select the **New Pipe Run** option in the **Run** drop-down list on the **Insert Component** ribbon.

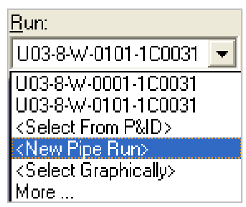


Figure 9: Run Drop-Down List

10. The **New Pipe Run** dialog box appears. Change the **Nominal Diameter** to **10 in** and click **OK**.
The outline of an eccentric reducer will appear in the graphic view, as shown in Figure 10.

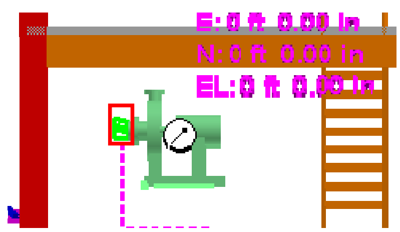


Figure 10: Outline of an Eccentric Reducer

11. The flat side of the eccentric reducer will point downwards. In the **Angle** drop-down list on the ribbon, key in **180 deg** to rotate the eccentric reducer. Then, click **Finish** on the ribbon to place the reducer.

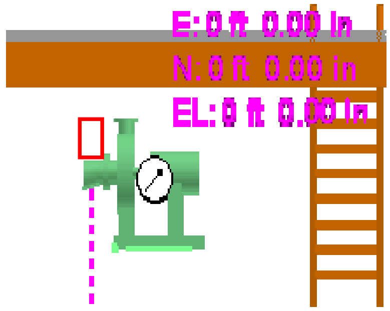


Figure 11: Placed Eccentric Reducer

Basic Routing Techniques by Using the PinPoint Tool and Offset Method

12. Now, click the **Route Pipe** button on the vertical toolbar.

13. Select the **Elevation Plane: East-West** option in the **Plane** drop-down list on the Route Pipe ribbon.

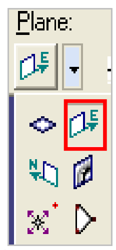


Figure 12: Plane Drop-Down Arrow

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

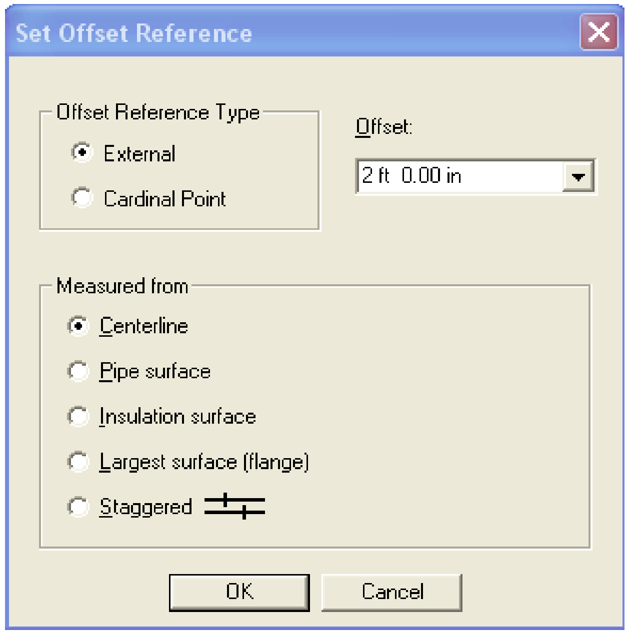


Figure 13: Set Offset Reference Dialog Box

Note:

- The following types of offset references are available while routing a pipe run:
  - **External** - Routes a pipe run at a specified distance from another object, such as a pipe run running parallel to the pipe run you are placing
  - **Cardinal Point** - Routes a pipe run by the top, sides, bottom, or invert elevation of the pipe run instead of the pipe run centerline

15. Move the cursor over the **Column** ***(Column shown below in Figure 14)*** until the offset glyph appears, as shown in Figure 14 and move slowly away from **Column** until perpendicular **projection line** is displayed .

The system will display a projection line that indicates 2 ft offset from Column as Shown Below.

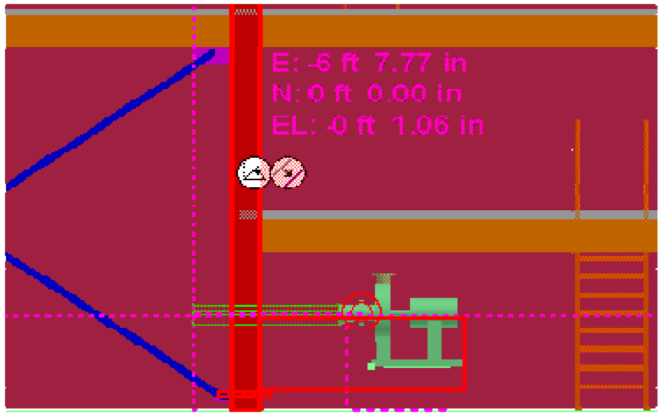


Figure 14: Projection Line Indicating 2 ft Offset from Column

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

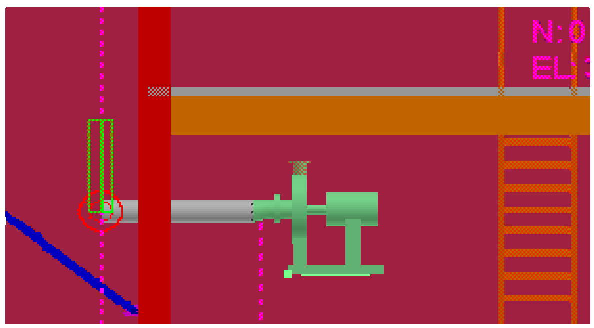


Figure 15: Placed Pipe

17. 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.

18. Now move the cursor in the north direction. You will see an outline of the pipe. Move the cursor to the grid line highlighted in Figure 16 to add it to SmartSketch.

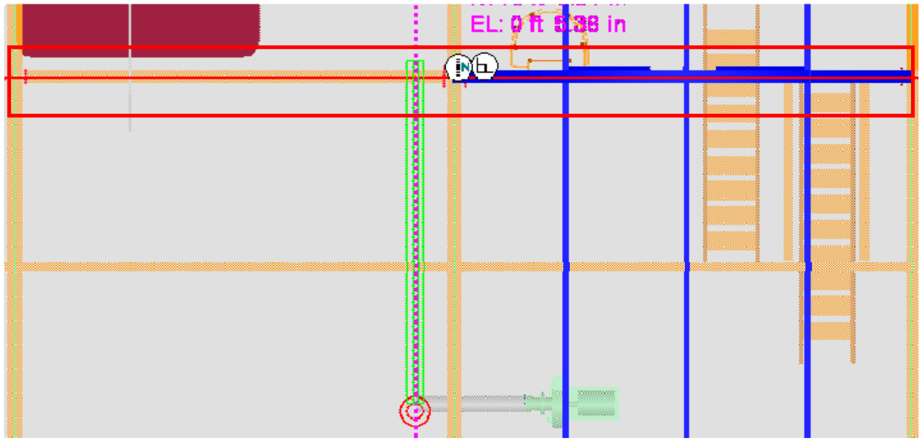


Figure 16: Grid Line Added to SmartSketch

19. 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.

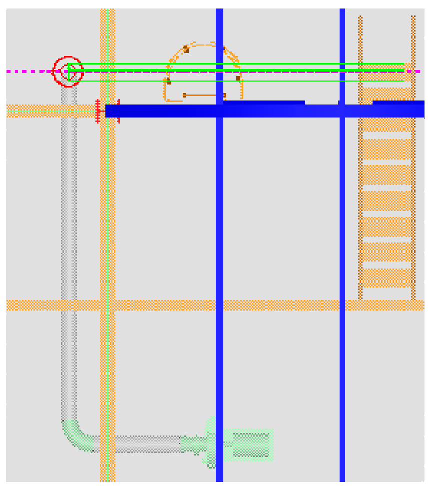


Figure 17: Placed Pipe

20. Now, you need to route the next segment of the pipe run to the roof of the model. Click the **Add to SmartSketch List** button on the **Common** toolbar and then click the roof surface to add it to the SmartSketch list. Click **Finish** to close the SmartSketch list ribbon.

21. Click the **Plane** drop-down list on the ribbon and select **Elevation Plane: East-West** to route the pipe. Key in **3 ft** in the **Offset** drop-down list.

22. Use the **Common Views** button to change the view **to Looking North**.

23. Now, move the cursor upwards and locate the projection line displayed by the system at **3 ft** offset from the roof surface. Click in the graphic view to place the pipe.

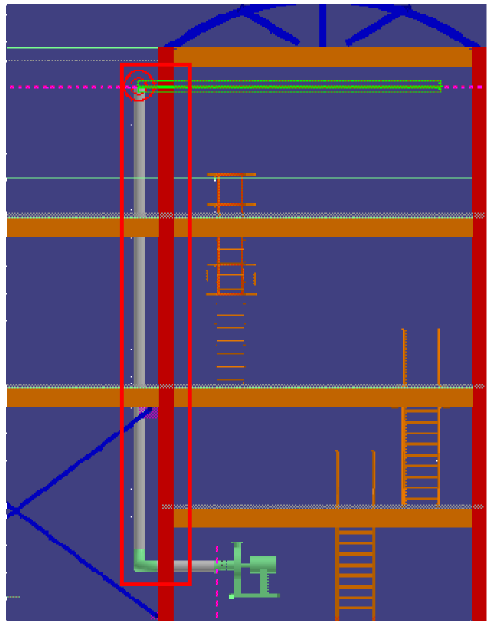


Figure 18: Placed Pipe

24. 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**.

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

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

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

Figure 19: Spherical Coordinate Button and Distance and Horizontal Drop-Down Lists

28. The system will now define the constraints within which the pipe can be placed, as shown in Figure 20.

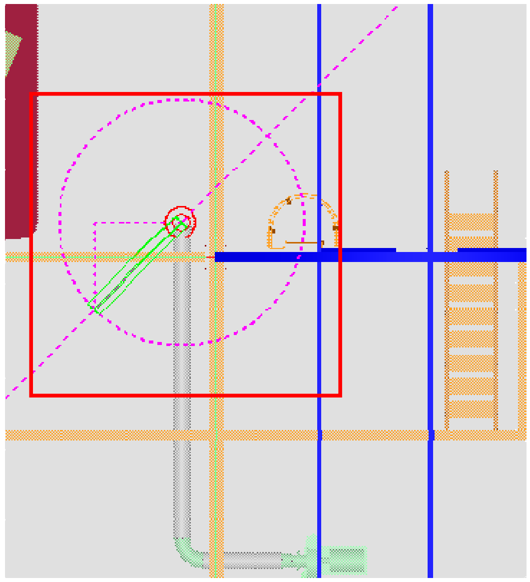


Figure 20: Pipe Constraints

29. Click in the graphic view to place the pipe. The view of your model should resemble Figure 21.

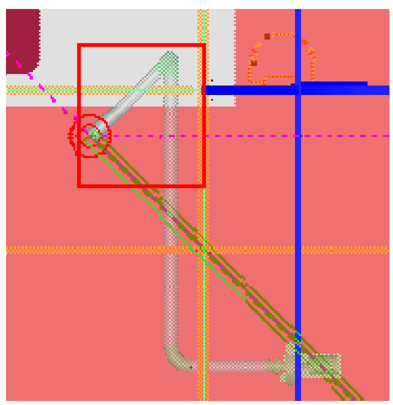


Figure 21: Placed Pipe

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

31. Select the **Elevation Plane: East-West** option in the **Plane** drop-down list to route the pipe. Key in **2 ft** in the **Offset** drop-down list.

32. Move the cursor to **Beam** *as shown above in Figure 21* to add it to SmartSketch.

33. Now move the cursor upwards to locate the projection line displayed by the system at **2 ft** offset from **Beam** *as shown above in Figure 21*. Click in the graphic view to place the pipe.

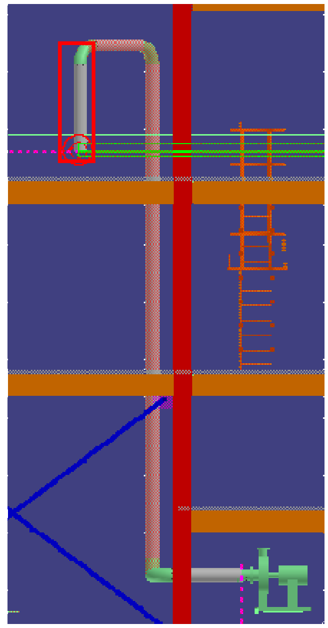


Figure 22: Placed Pipe

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.

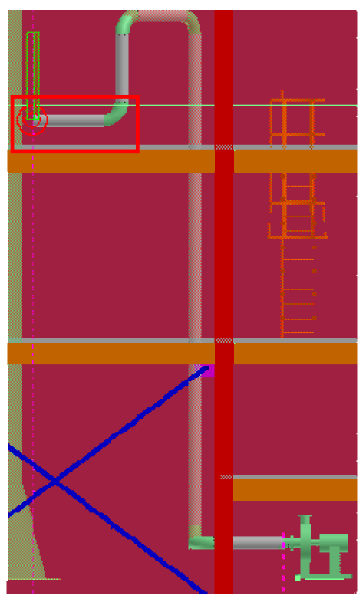


Figure 23: Placed Pipe

34. Now, you will route the next segment of the pipe upwards. Use the **Pipe Straight Feature** highlighted in Figure 24 to find the intersection point.

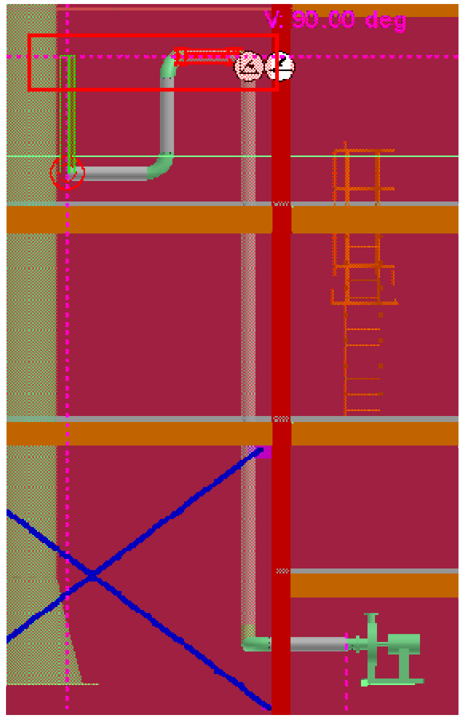


Figure 24: Using The Pipe Straight Feature to Find the Intersection Point

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

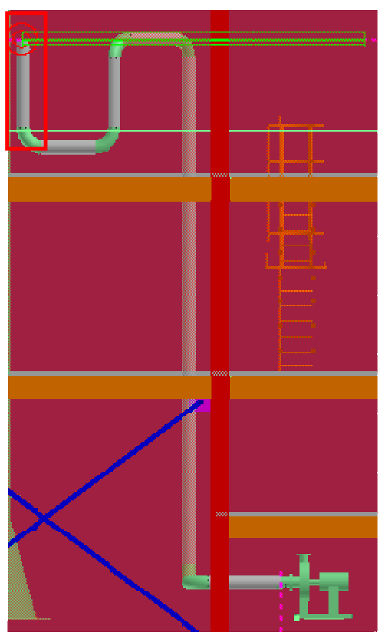


Figure 25: Placed Pipe

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 in Figure 25 to locate the intersection point. Click in the graphic view to place the pipe. The view of your model should resemble Figure 26.

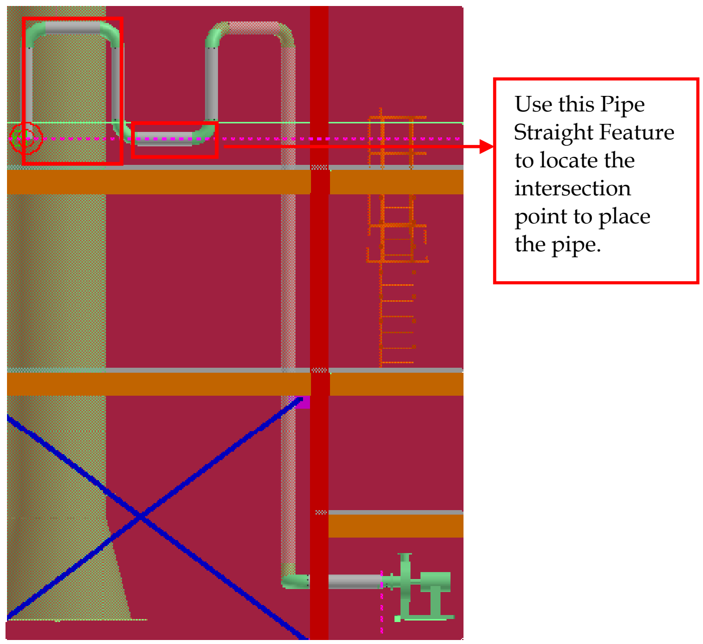


Figure 26: Placed Pipes

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 in Figure 27 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.

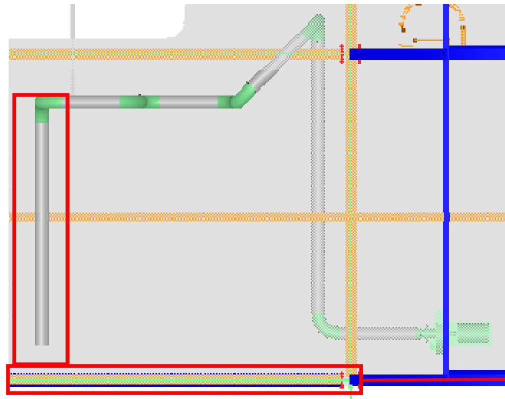


Figure 27: Placed Pipe

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 highlighted in Figure 28. Use the **Common Views** button to change the view to **Looking North**.

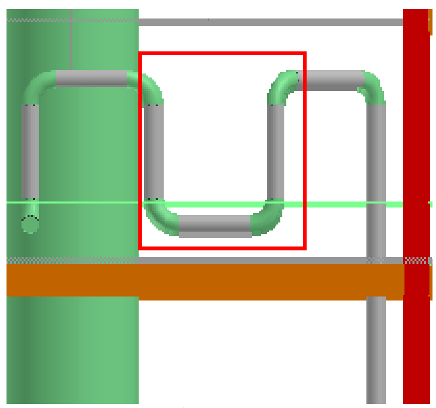


Figure 28: Area for the Pipe to be Placed

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

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

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

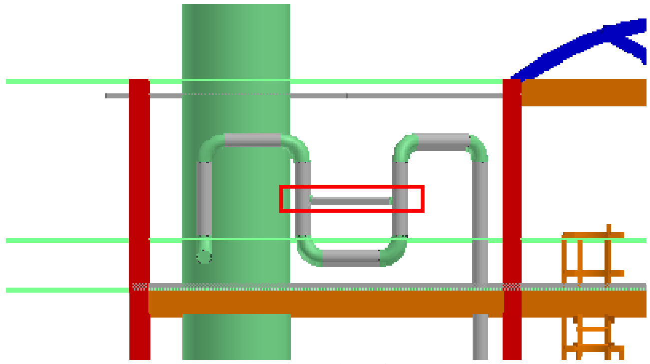


Figure 29: Placed Pipe

48. Now, you will route a pipe from the nozzle at the top of the tower **T-101**. Click the **Route Pipe** button on the vertical toolbar and then click the nozzle at the top of the tower **T-101**.

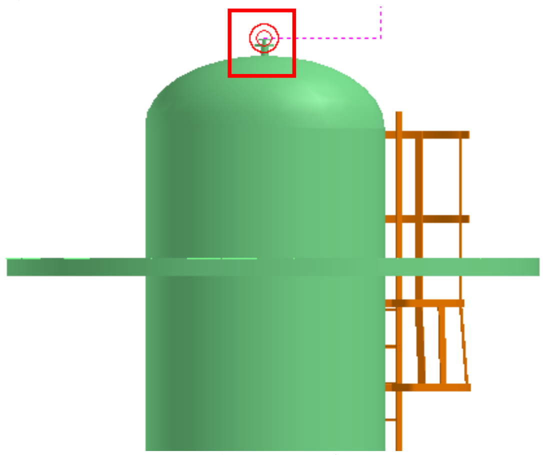


Figure 30: Nozzle on Top of Tower T-101

49. The **New Pipe Run** dialog box appears. Change the **Pipeline** to **301-W** and click **OK**.

50. On the **PinPoint** ribbon, click the **Rectangular Coordinates** option and key in **2 ft** in the **El** drop-down list.

Figure 31: Rectangular Coordinates Button and El Drop-Down list

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

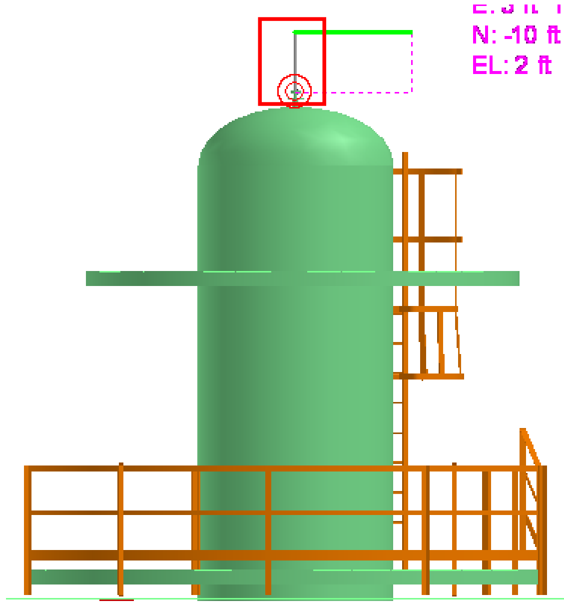


Figure 32: Placed Pipe

52. 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**.

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

54. 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.

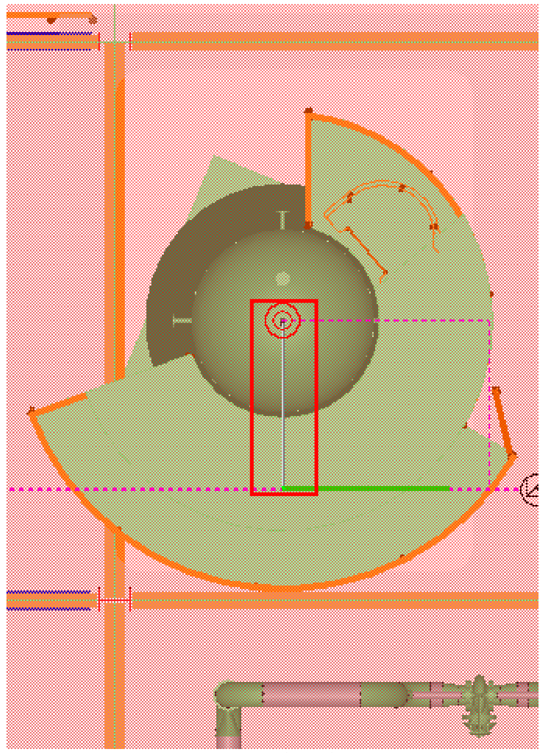


Figure 33: Placed Pipe

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

56. 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.

57. 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.

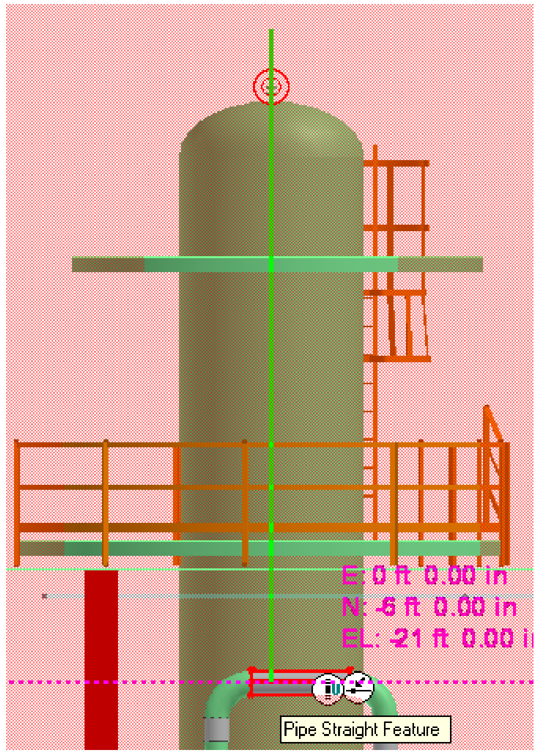


Figure 34: Pipe Straight Feature and Corresponding Intersection Point

58. Now, use the **Common Views** button to change the view to **Looking Plan**.

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

60. 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.

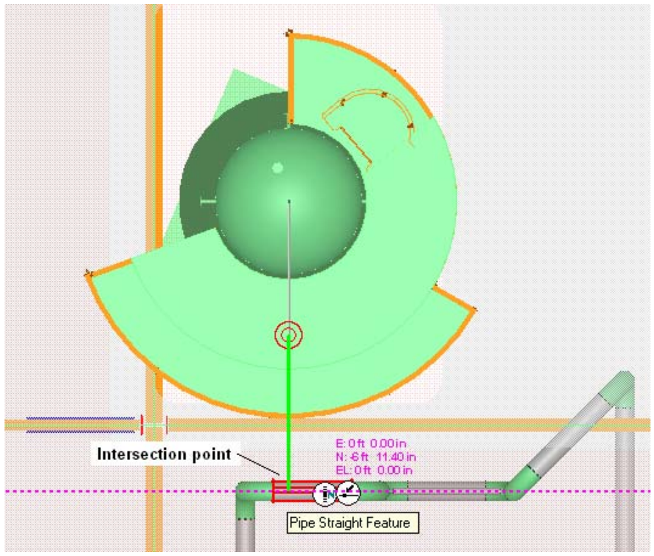


Figure 35: Pipe Straight Feature and Corresponding Intersection Point

The view of your model should resemble Figure 36.

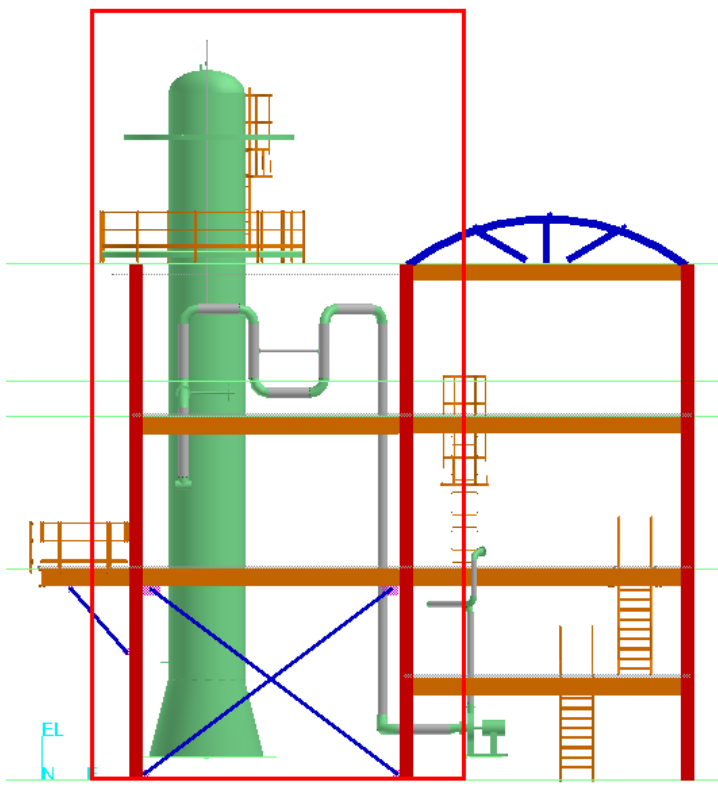


Figure 36: Final Output

Inserting a Gate Valve at a Known Distance from the End of a Pipe

61. Locate the free end of the pipeline **300-W**, as shown in Figure 37. Click the **Reposition Target** option on the **PinPoint** ribbon and put the target at the free end of the pipe.

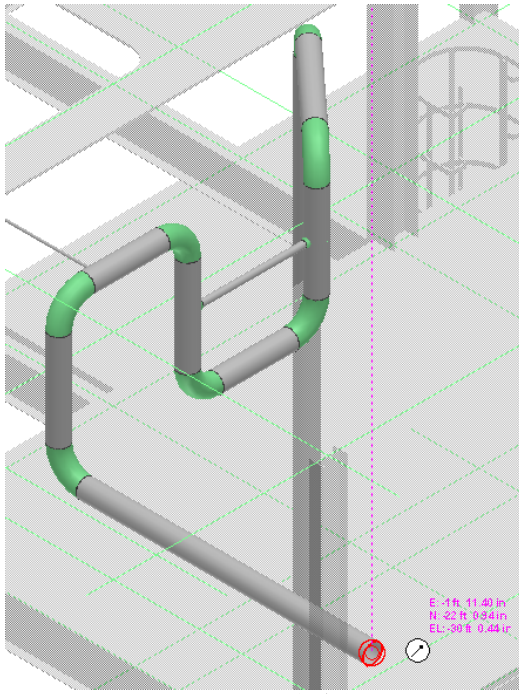


Figure 37: Free End of the Pipe

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

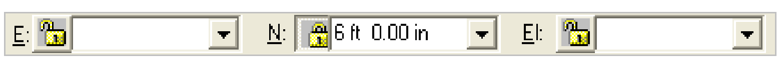


Figure 38: Distance Specification on the PinPoint Ribbon

63. Click the **Insert Component** button on the vertical toolbar.

64. Now, position the cursor along the **Pipe Straight Feature**, as shown in Figure 39, and click to define the active placement point.

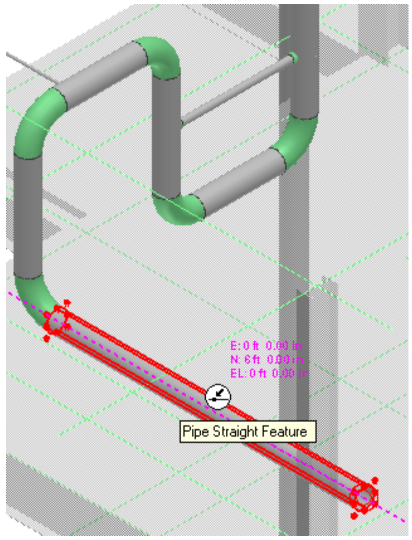


Figure 39: Pipe Straight Feature

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 SP3D finds the intersection point between the plane and the line.

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

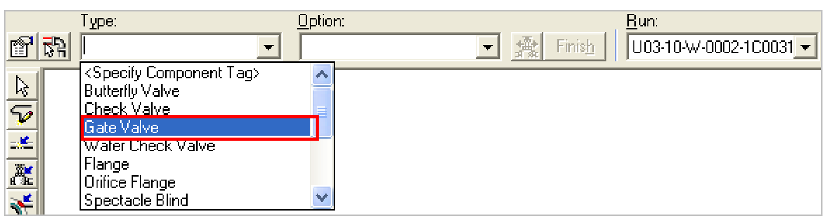


Figure 40: Gate Valve Option in the Type Drop-Down List

66. You will now see an outline of a valve (base part) with mating flanges at the active placement point, as shown in Figure 41.

    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.

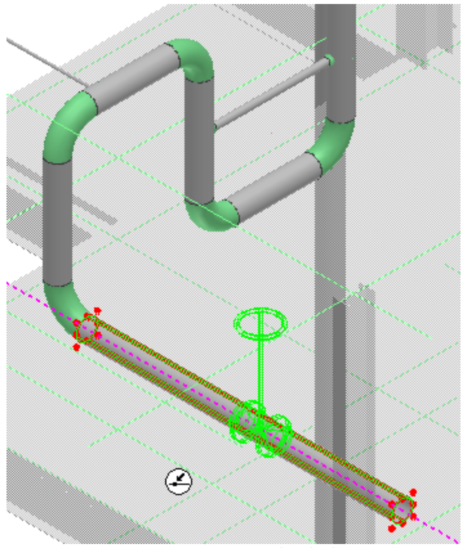


Figure 41: Gate Valve at a Distance of 6 ft from the End of the Pipe

Notice that SP3D automatically selects the **Default** option in the **Option** drop-down list on the ribbon. The option selected in the **Option** drop-down list instructs SP3D 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, SP3D prompts you to select an option in the **Option** drop-down list.

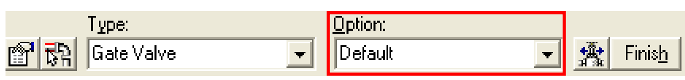


Figure 42: Default Option in the Option Drop-Down List

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.

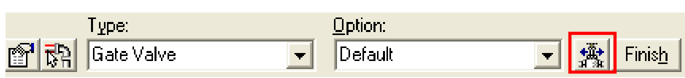


Figure 43: Insert Point step

67. 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 in Figure 44.

Figure 44: Angle Control in the Route Pipe Ribbon

68. Click the **Finish** button on the **Insert Component** ribbon to place the valve.

69. Right-click in the graphic view to terminate the **Insert Component** command.

    You have now placed a gate valve in your model.

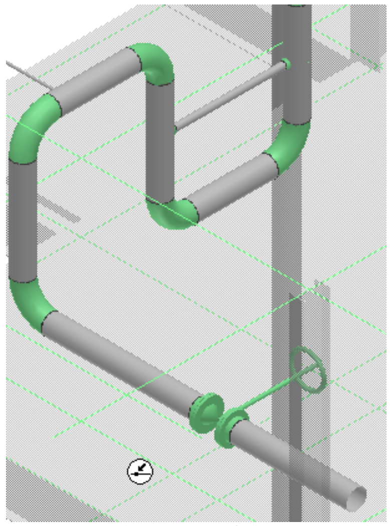


Figure 45: Placement of the Gate Valve

Inserting a 90 deg Elbow at the Free End of a Pipe

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

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

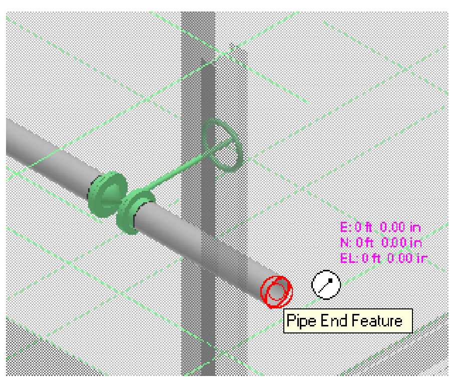


Figure 46: Pipe End Feature

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

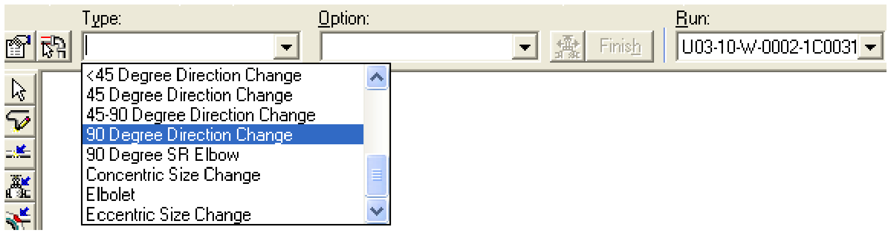


Figure 47: 90 degree Direction Change Option in the Type Drop-Down List

72. You will now see the outline of an elbow at the active placement point, as shown in Figure 48.

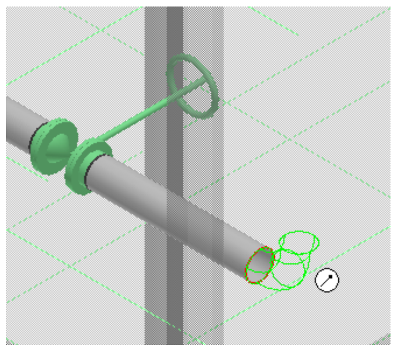


Figure 48: 90 Deg Elbow at the End of the Pipe

73. 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.

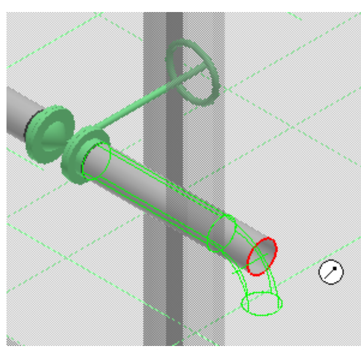


Figure 49: 90 Deg Elbow Rotated by 180 Deg and Positioned by Its Origin

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

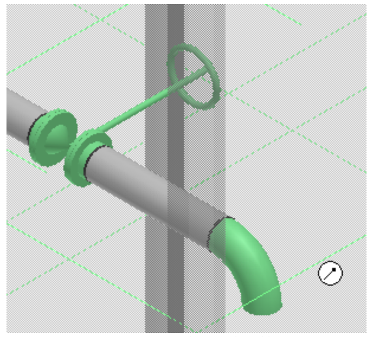


Figure 50: Placed Elbow

Inserting a Tee by the Branch Point at the Free End of a Pipe

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:

75. 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, SP3D 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 in Figure 51.

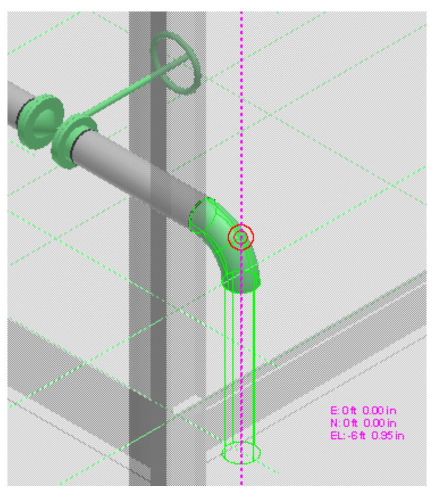


Figure 51: Pipe Straight Feature

76. 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.

Figure 52: Length Field on the Route Pipe Ribbon

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

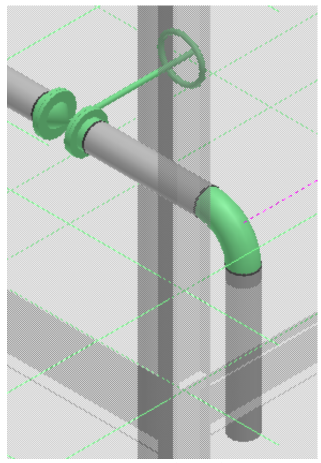


Figure 53: Placed Pipe

78. Now, click the **Insert Component** button on the vertical toolbar.

79. Select the **Pipe End Feature** of the pipe that you routed.

80. Select the **Tee** option in the **Type** drop-down list on the ribbon.

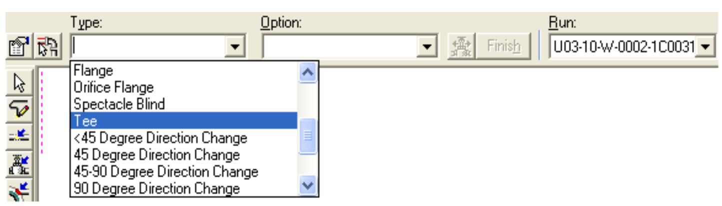


Figure 54: Tee Option in the Type Drop-Down List

81. 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 in Figure 55.

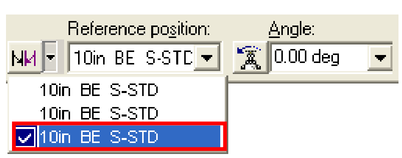


Figure 55: Branched Port Option in the Flip Drop-Down List

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

ribbon.

83. The **New Pipe Run** dialog box appears. Click **OK** to accept the default values of the new pipe run.

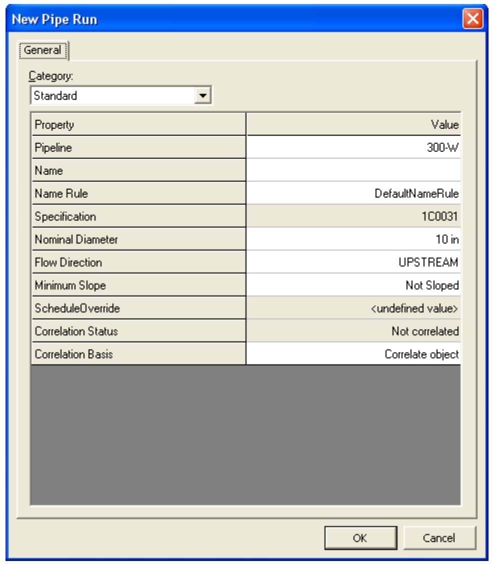


Figure 56: New Pipe Run Dialog Box

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

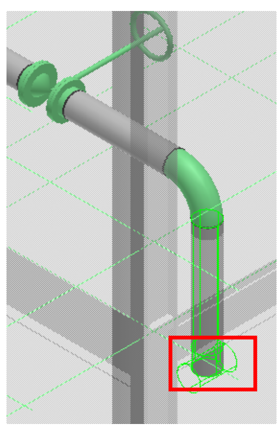


Figure 57: Tee at the End of the Pipe

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

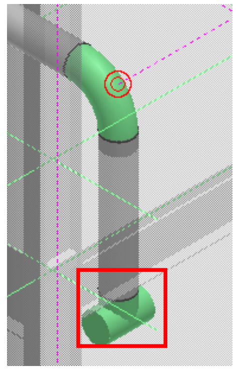


Figure 58: Placed Tee

Steps for Creating the Control Valve Station by Inserting Components:

Exercise Objective: In this exercise you will be creating a **control valve station** by using the **Insert Component** command on the pipeline **400-P** in Unit **U04** of your workspace. After inserting the components, the control valve station will look like the highlighted section of Figure 59.

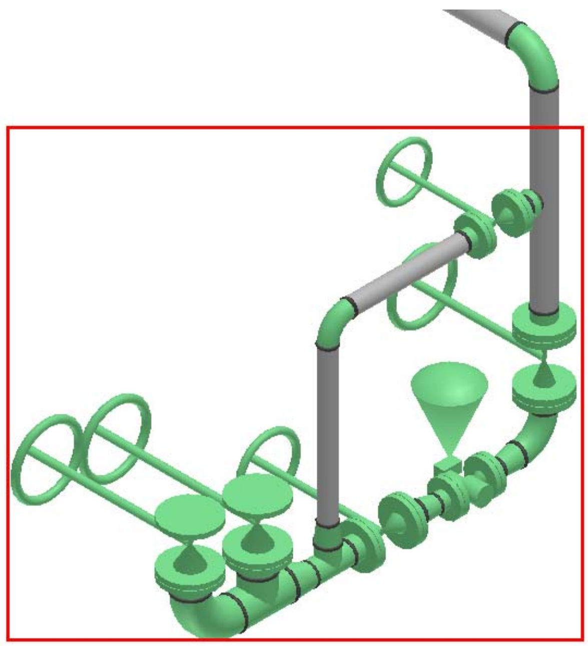


Figure 59: Control Valve Station

Before beginning the 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 the **Insert Component** button on the vertical toolbar.

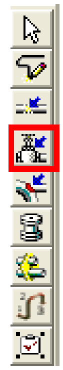


Figure 60: Insert Component Button on the Vertical Toolbar

2. Select the **Pipe End Feature** of the pipeline **400-P** by using the SmartSketch options of SP3D.

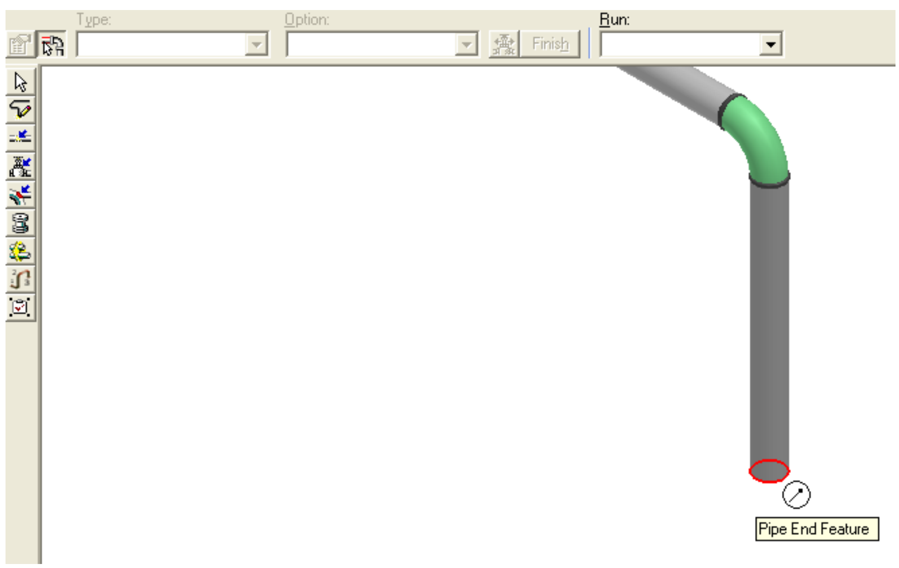


Figure 61: Pipe End Feature

3. 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**. Select **Flange** option in the **Type** drop-down list, as shown in Figure 62.

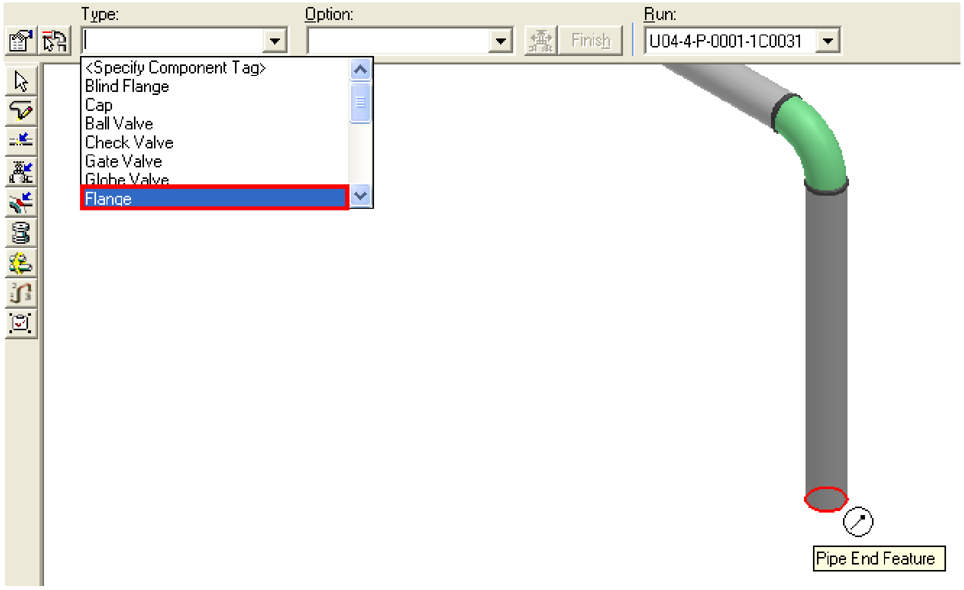


Figure 62: Type Drop-down List on the Insert Component Ribbon

4. An outline of a flange at the end of the pipeline will appear in the graphic view. Select **300** option in the commodity **Option** drop-down list and click **Finish** on the **Insert Component** ribbon to place the selected flange.

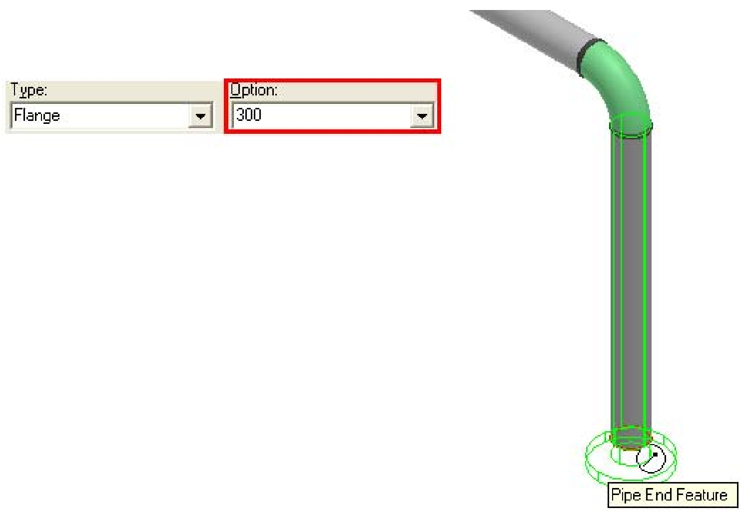


Figure 63: Outline of a Flange and the Option Drop-Down List

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

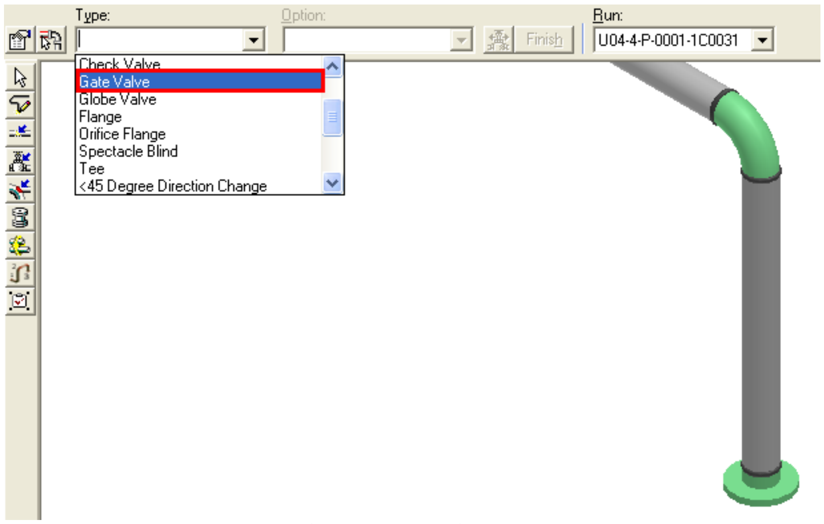


Figure 64: Gate Valve Option in the Type Drop-Down List

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 in Figure 65.

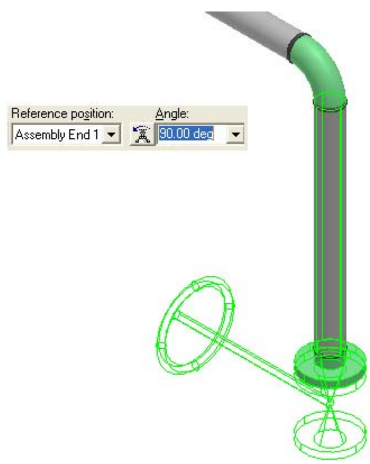


Figure 65: Gate Valve Facing the North Direction

7. An outline of the gate valve will appear in the graphic view. Click **Finish** on the **Insert Component** ribbon to place the gate valve.

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

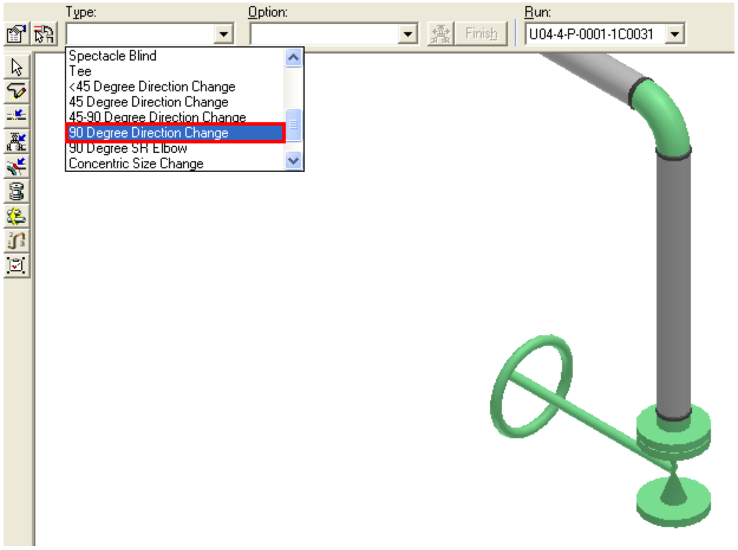


Figure 66: 90 *Degree Direction Change Option on the Type Drop-Down List*

9. An outline of the elbow and mating flange will appear in the graphic view. 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 in Figure 67.

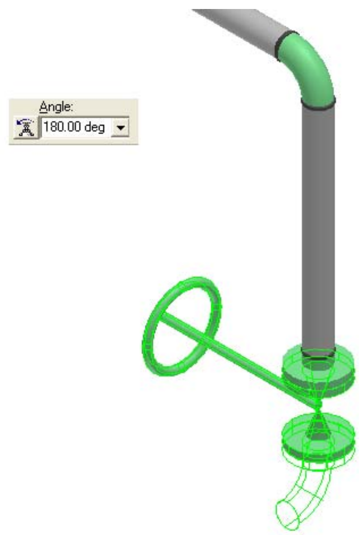


Figure 67: Elbow Rotated by 180 Deg

10. Click **Finish** on the **Insert Component** ribbon to place the 90-degree elbow.

11. Select the **Eccentric Size Change** option in the **Type** drop-down list on the **Insert Component** ribbon.

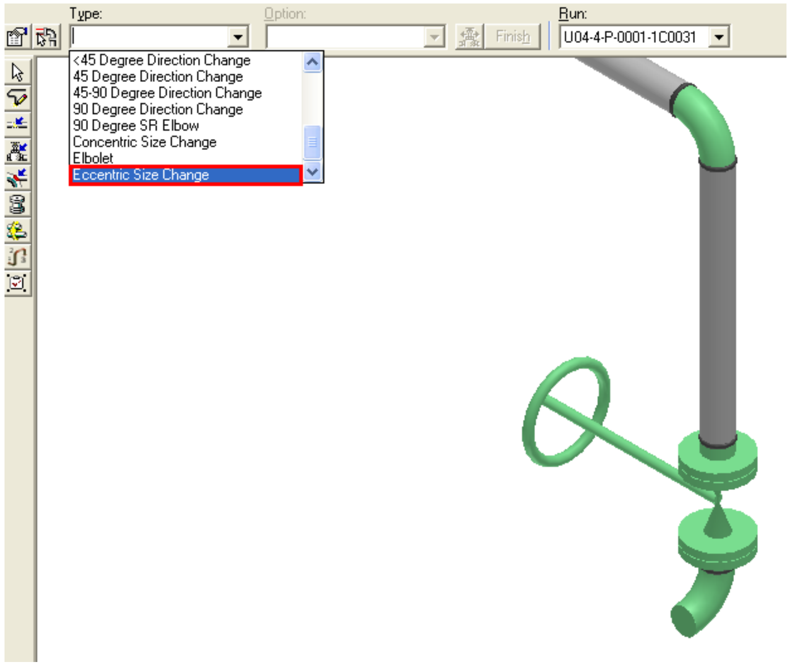


Figure 68: Eccentric Size Change in the Type Drop-Down List

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

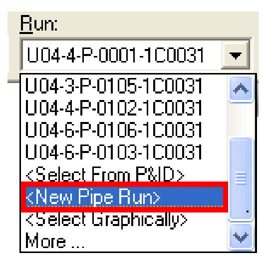


Figure 69: <New Pipe Run> Option in the Run Drop-Down List

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

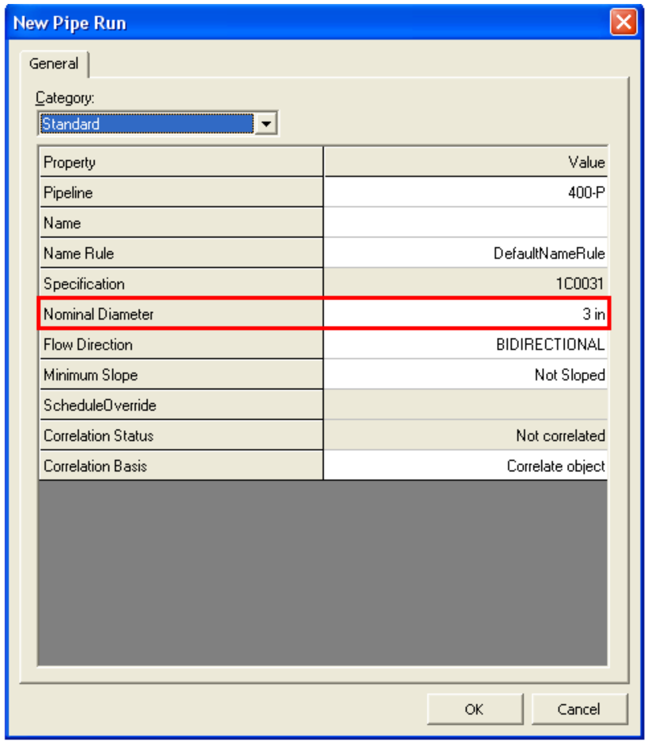


Figure 70: New Pipe Run Dialog Box

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

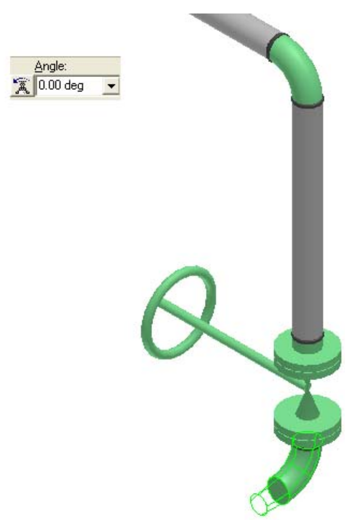


Figure 71: Eccentric Reducer in the Graphic View

15. Click **Finish** on the **Insert Component** ribbon to place the eccentric reducer.

16. Select **Flange** option in the **Type** drop-down list on the **Insert Component** ribbon.

17. An outline of the flange will appear in the graphic view, as shown in Figure 72. Click **Finish** on the **Insert Component** ribbon to place the flange.

Figure 72: Outlined Flange in the Graphic View

18. Select the **<Specific Component Tag>** option in the **Type** drop-down list on the **Insert Component** ribbon, as shown in Figure 73.

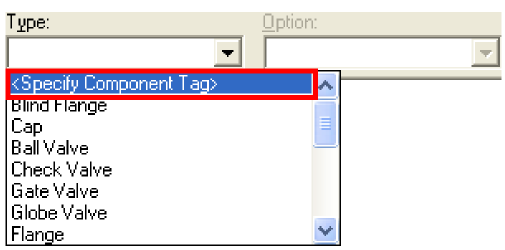


Figure 73: <Specific Component Tag> Option in the Type Drop-Down List

19. The **Specific Component Tag** dialog box appears. Select the **Browse Instruments…** option to select the instrument from the catalog.

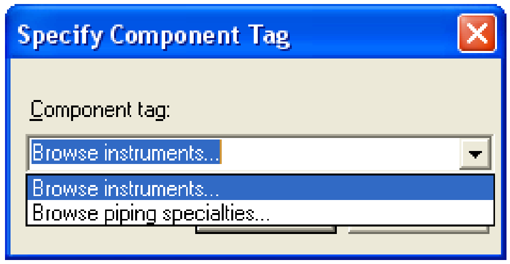


Figure 74: Specify Component Tag Dialog Box

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.

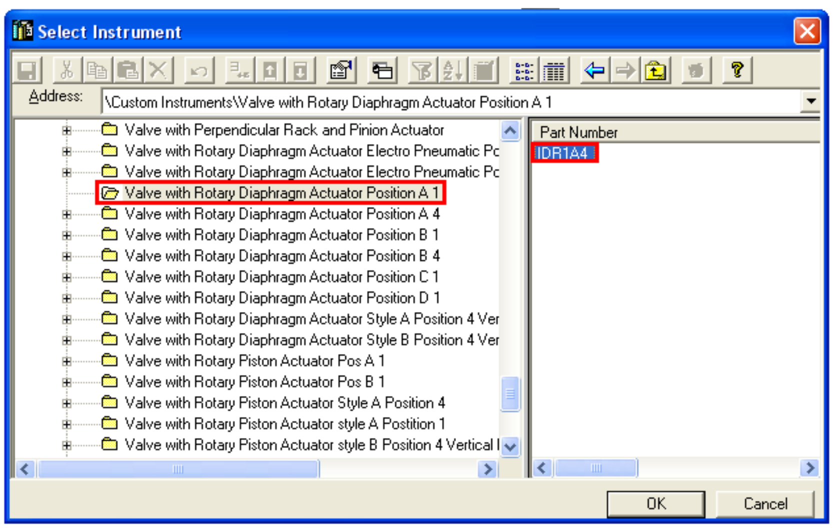


Figure 75: Select Instrument Dialog Box

21. An outline of the instrument will appear in the graphic view. Click the **Properties** option on the **Insert Component** ribbon to change the properties of the valve actuator.
22. The **Pipe Component Feature Properties** dialog box appears. 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

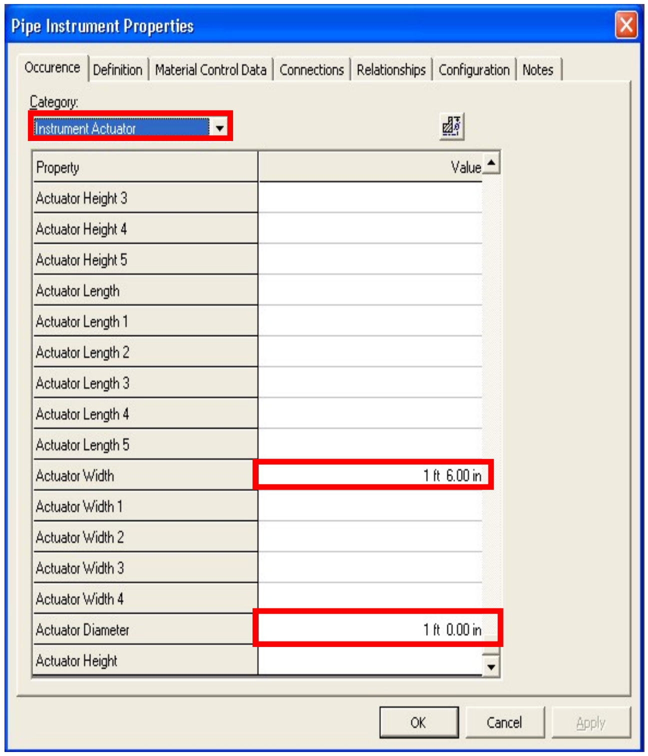


Figure 76: Pipe Component Feature Properties Dialog Box

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 Erector
    Fabrication Type: Contractor field 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 erector
    Fabrication Type: Contractor fabricated

    Bolting Requirements: bolting required
    Gasket Requirements: Gasket required at each bolted end

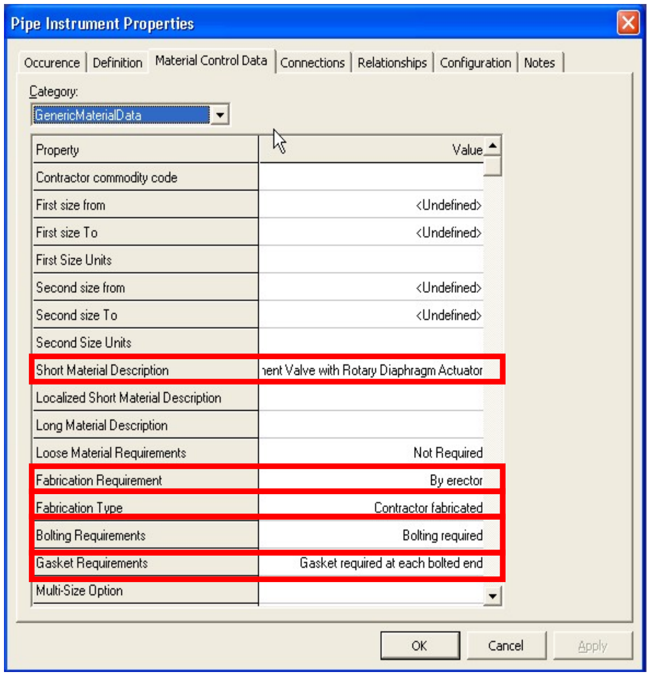


Figure 77: Pipe Component Feature Properties

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

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

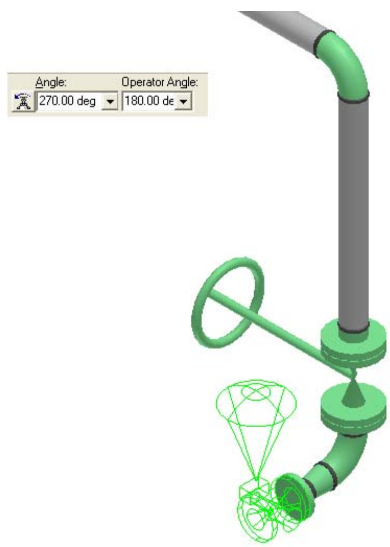


Figure 78: Rotation of the Instrument and Actuator

28. Click **Finish** on the **Insert Component** ribbon 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.

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

31. An outline of the eccentric reducer and mating flange will appear in the graphic view, as shown in Figure 79. 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.

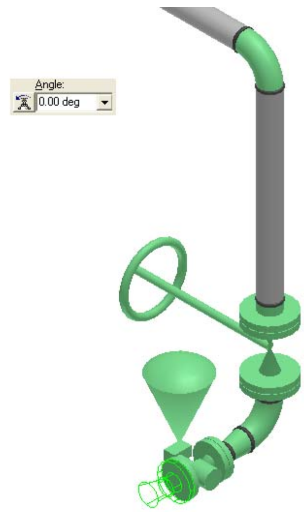


Figure 79: Outlined Eccentric Reducer in the Graphic View

32. Click **Finish** on the **Insert Component** ribbon 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 in Figure 80.

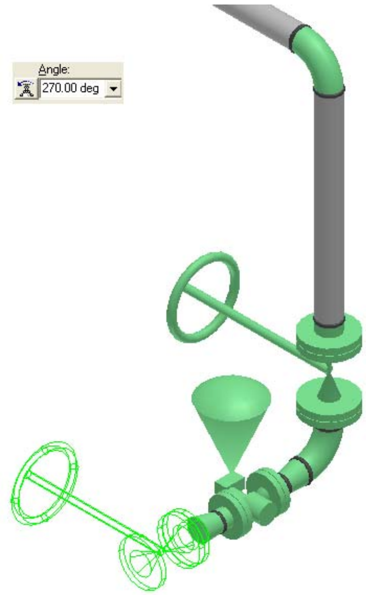


Figure 80: Outlined Gate Valve in the Graphic View

34. Click **Finish** on the **Insert Component** ribbon 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 in Figure 81.

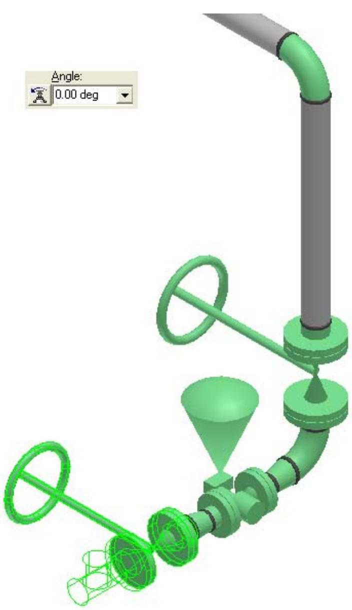


Figure 81: Tee and Mating Flanges in the Graphic View

36. Click **Finish** on the **Insert Component** ribbon to place the tee.

37. Select the **Eccentric Size Change** option in the **Type** drop-down list on the **Insert Component** ribbon, as shown in Figure 82.

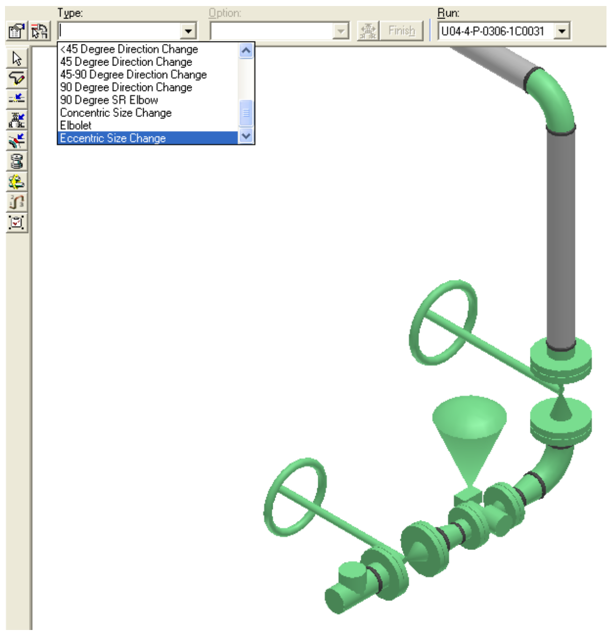


Figure 82: Eccentric Size Change Option in the Type Drop-Down List

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

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

40. An outline of the eccentric reducer will appear in the graphic view. Click **Finish** on the **Insert Component** ribbon to place the eccentric reducer, as shown in Figure 83.

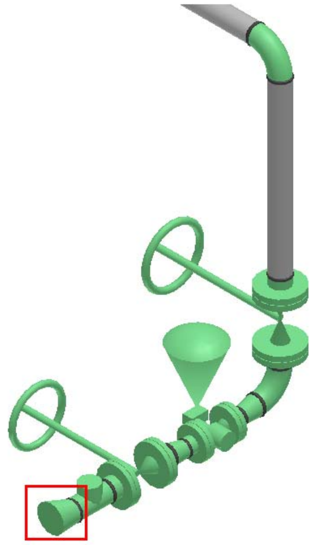


Figure 83: Placed Eccentric Size Change

41. Now insert a tee and an elbow **90 Degree Direction Change**, as shown in Figure 84, by selecting them in the **Type** drop-down list on the **Insert Component** ribbon.

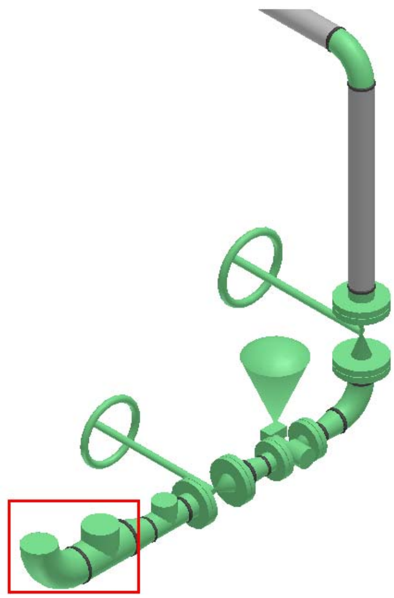


Figure 84: Placed Tee and Elbow

42. Now insert a gate valve by rotating it by **270 deg** so that the gate valve operator is facing the north direction, as shown in Figure 85.

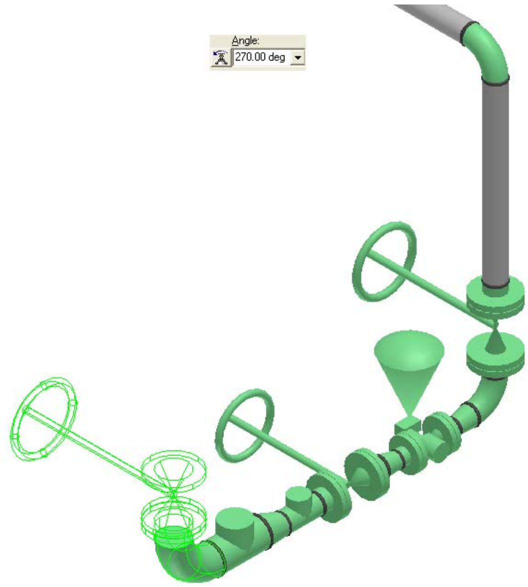


Figure 85: Gate Valve in the Graphic View

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

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

45. Select **port 3** of the 6 inch tee, as shown in Figure 86, by using the SmartSketch options of SP3D.

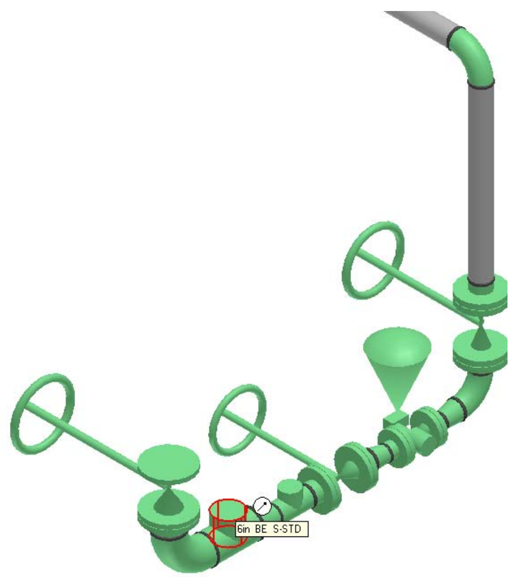


Figure 86: Port 3 of a Tee

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

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

48. An outline of the gate valve and the mating flange will appear in the graphic view. Key in **270 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the gate valve so that the valve operator is facing north, as shown in Figure 87.

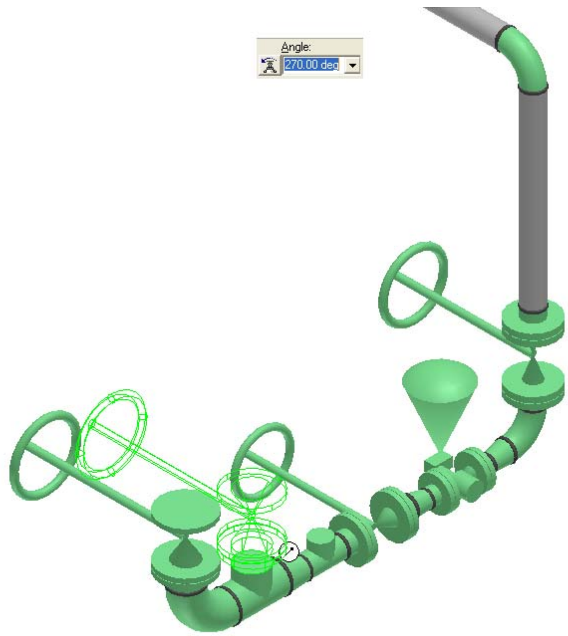


Figure 87: Gate Valve in the Graphic View

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

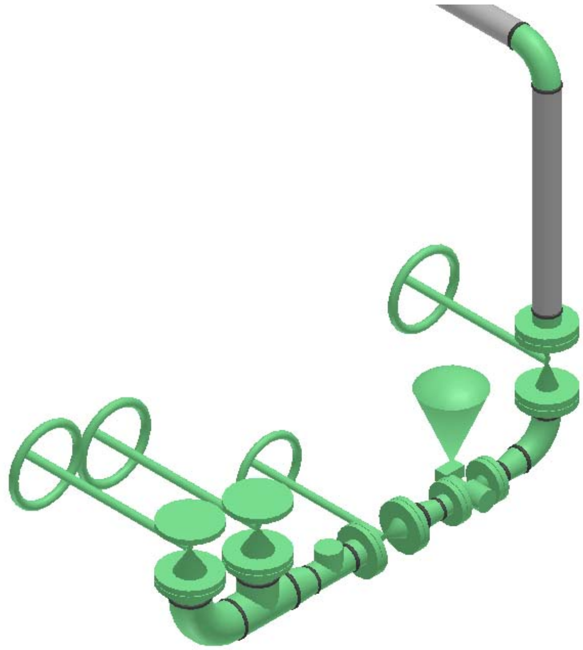


Figure 88: Placed Gate Valve in the Graphic View

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 in Figure 89.

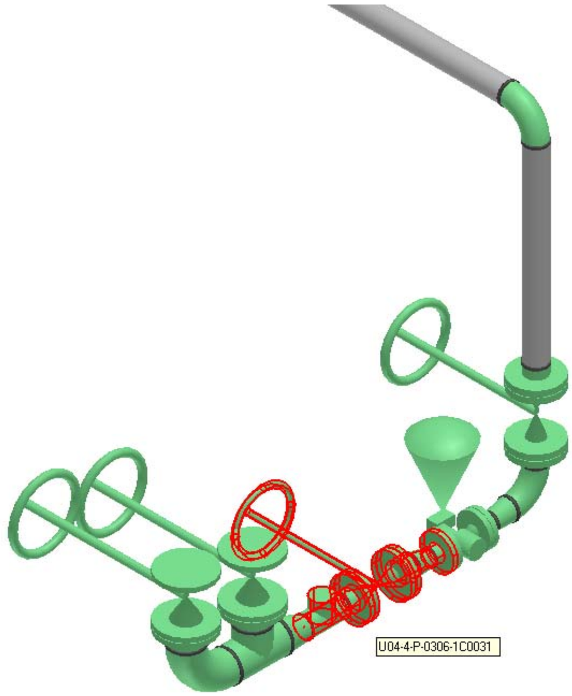


Figure 89: Selected Pipe Run Along With the Tee

52. Select **port 3** of 4 inch the tee in the graphic view by using the SmartSketch options of SP3D, as shown in Figure 90.

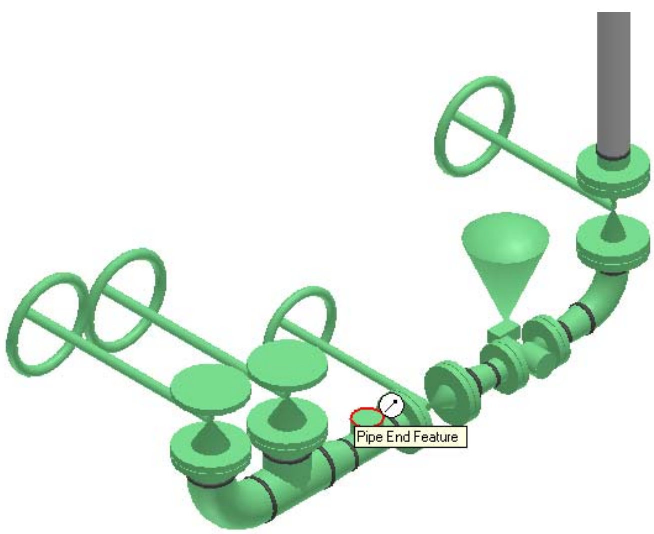


Figure 90: Selected Port 3 of the Tee

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.

54. The **New Pipe Run** dialog box appears. 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 in Figure 91.

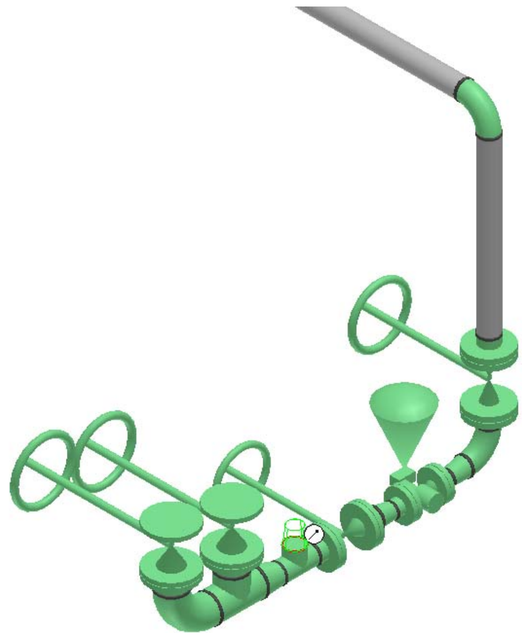


Figure 91: Outlined Concentric Reducer in the Graphic View

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

56. Click the **Route Pipe** button on the vertical toolbar and locate the end of the concentric reducer, as shown in Figure 92.

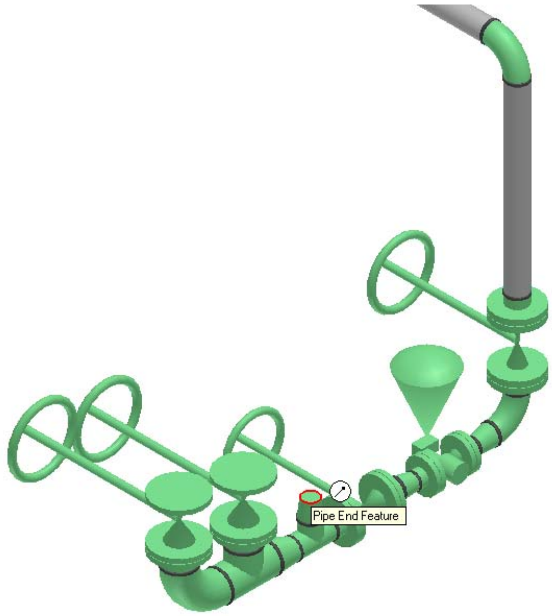


Figure 92: End Point of the Concentric Reducer

57. Route a **Pipe Straight Feature** to the same elevation as the mid point of the vertical straight feature, as shown in Figure 93.

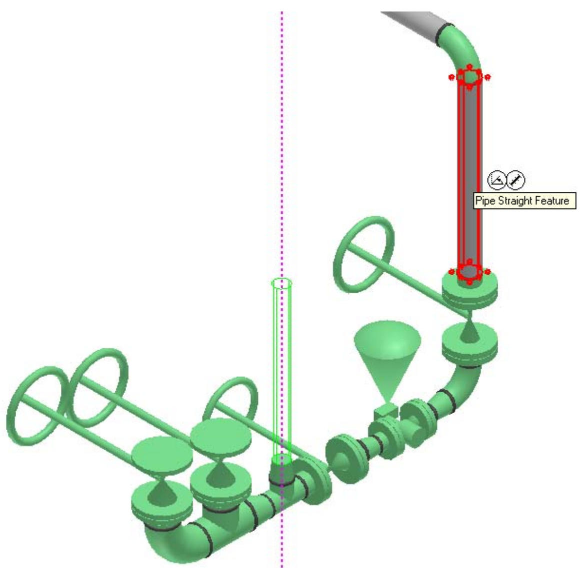


Figure 93: Routing a Pipe Straight Feature

58. Make a 90-degree turn and route into the vertical straight feature to create a by pass line, as shown in Figure 94.

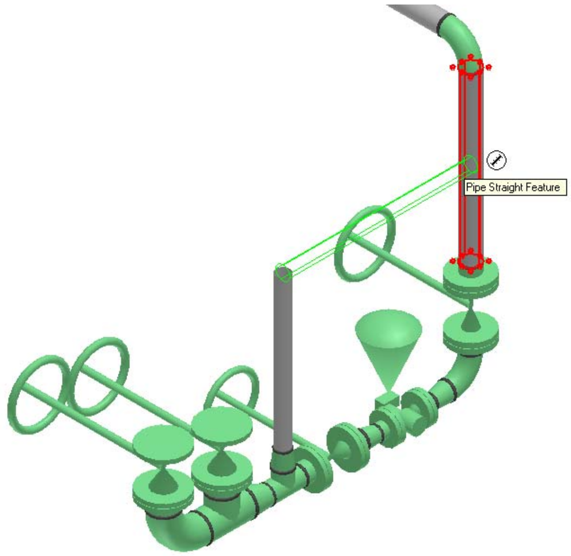


Figure 94: Routing the Pipe

59. Click the **Insert Component** button again on the vertical toolbar.

60. Specify the end of the by pass line by using the SmartSketch options of SP3D, as shown in Figure 95.

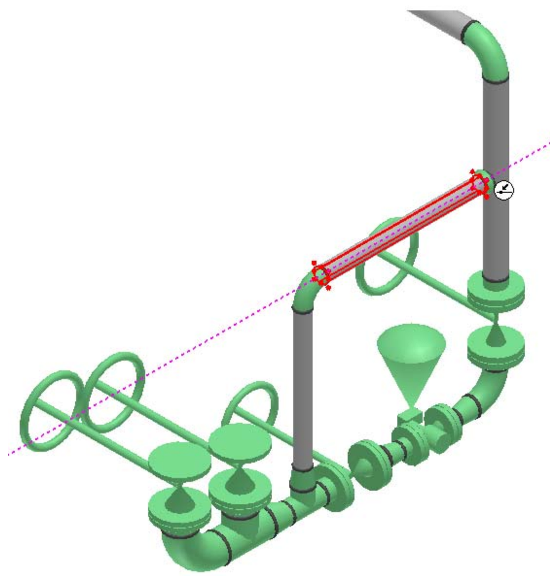


Figure 95: End of the By Pass Line

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

62. An outline of the gate valve and the mating flange will appear in the graphic view. Key in **90 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the gate valve so that the valve operator faces north, as shown in Figure 96.

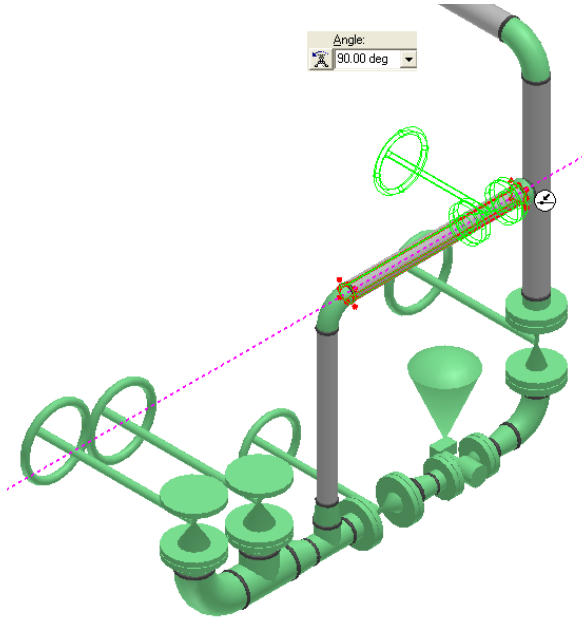


Figure 96: Gate Valve Rotated by 90 Deg

63. Click **Finish** on the **Insert Component** ribbon to place the gate valve, as shown in Figure 97.

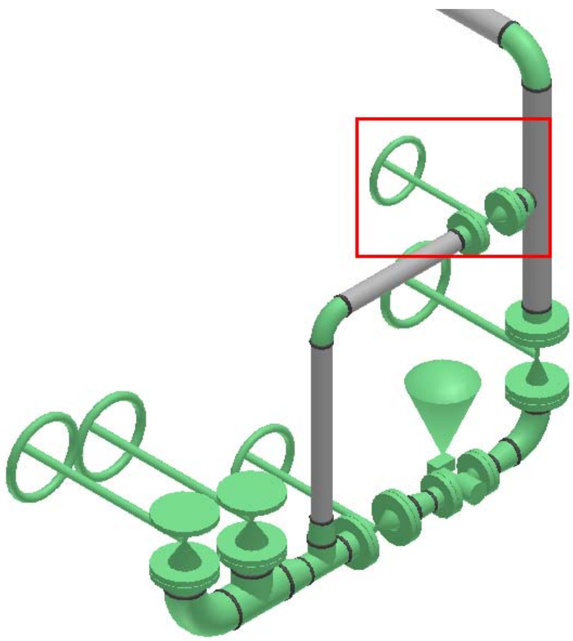


Figure 97: Placed Gate Valve

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

Session 4: Routing a Sloped Pipe

Objective:

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

- Route a sloped pipe.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run

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 the **Route Pipe** button on the vertical toolbar 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, as shown in Figure 1. You define the slope by either defining the angle of the slope or the fraction to which you want the pipe to be sloped.

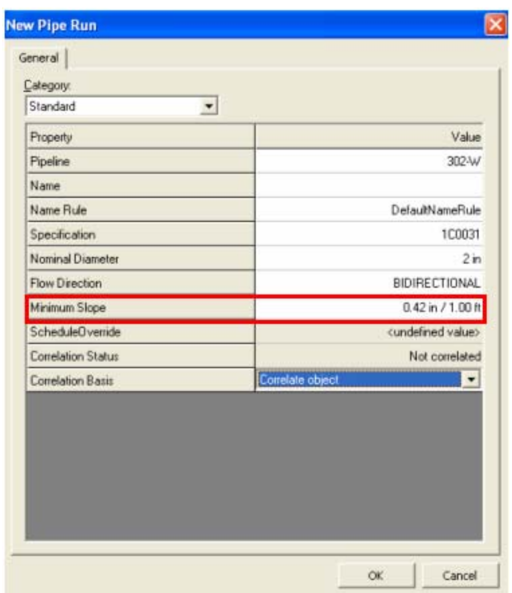


Figure 1: Minimum Slope Field in the New Pipe Run Dialog Box

This session covers the procedure for routing a sewer line to a catch basin with laterals and cleanouts.

Steps for Routing a Sewer Line to a Catch Basin with Laterals and Cleanouts:

Exercise 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 of Figure 2.

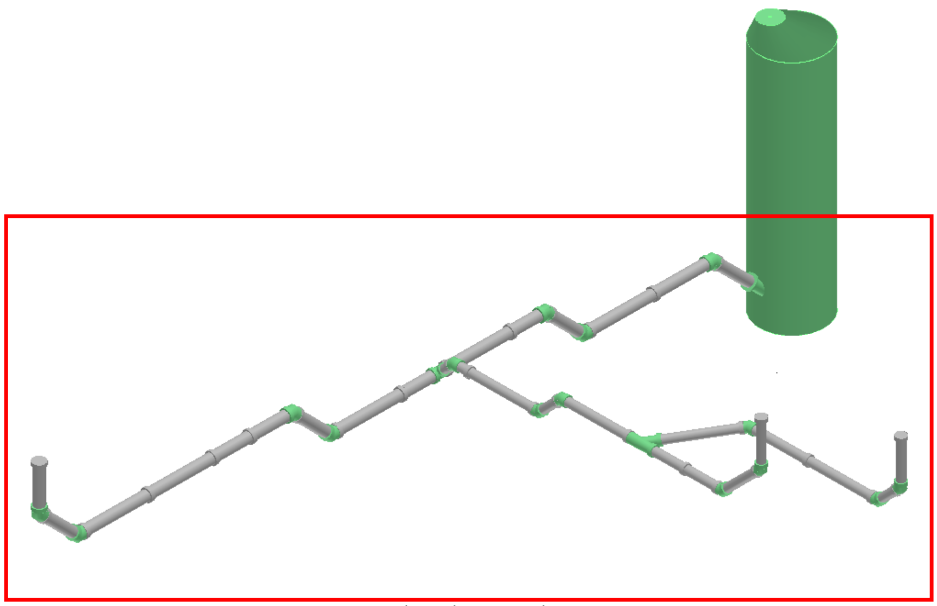


Figure 2: Routed Underground Piping System

Before beginning the 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 the **PinPoint** command by using the **Tools > PinPoint** menu command.

3. Click the **Route Pipe** button on the vertical toolbar to start routing the main sewer pipe.

Figure 3: Route Pipe Button on the Vertical Toolbar

4. Select the **Reposition Target** option from 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, as shown in Figure 4.

Figure 4: Repositioning of the Reference Point

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.

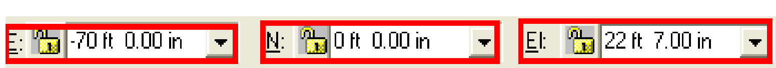


Figure 5: Starting Points Defined on the PinPoint Ribbon

7. The **New Pipe Run** dialog box appears. 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
 Minimum Slope: 0.0625 in /1.0 ft
 ScheduleOverride: <undefined value>
 Correlation Status: Not correlated
 Correlation Basis: Correlate object

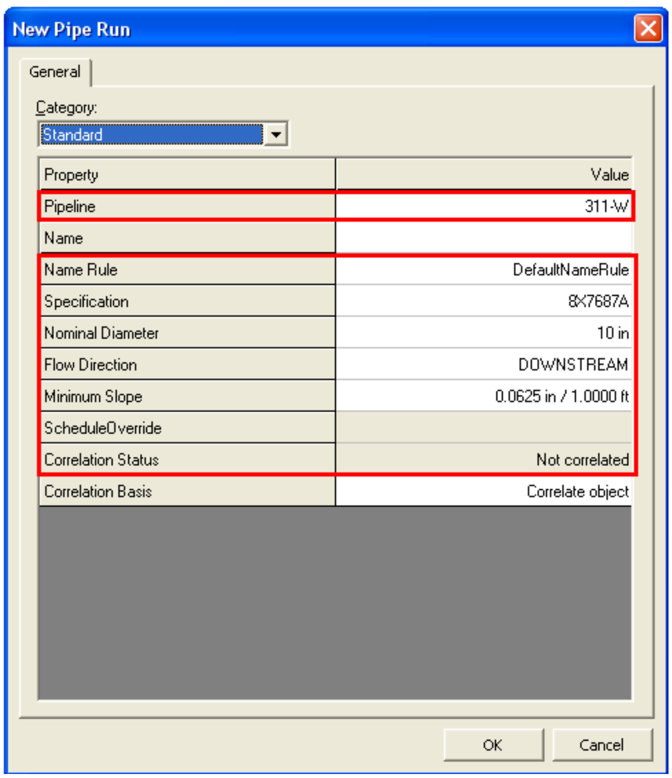


Figure 6: Pipeline Specifications on the New Pipe Run Dialog Box

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.

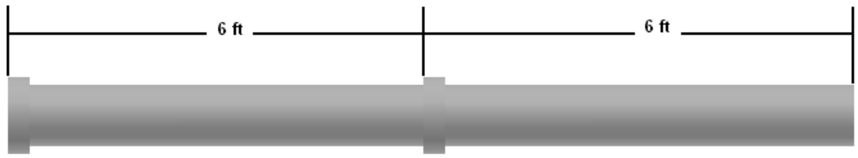


Figure 7: Bell and Spigot Joints in the Underground Piping System

8. The **Specify Slope Direction** dialog box appears, as shown in Figure 8. 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. Then, click **OK** to close the **Specify Slope Direction** dialog box.

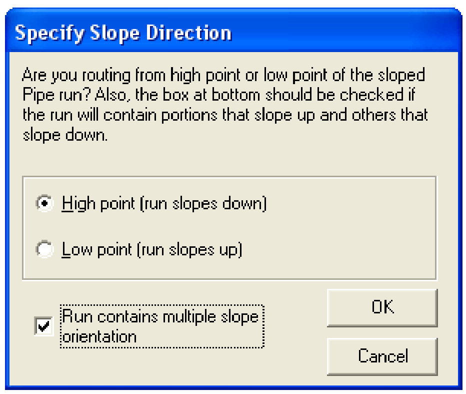


Figure 8: Specify Slope Direction Dialog Box

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. Key in **4 ft** in the **Length** drop-down list on the **Route Pipe** ribbon to define the length of the pipe. Make sure the **Plane** constraint is set to **No Plane**.

Figure 9: Length Drop-Down List on the Route Pipe Ribbon

10. 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, as shown in Figure 10.

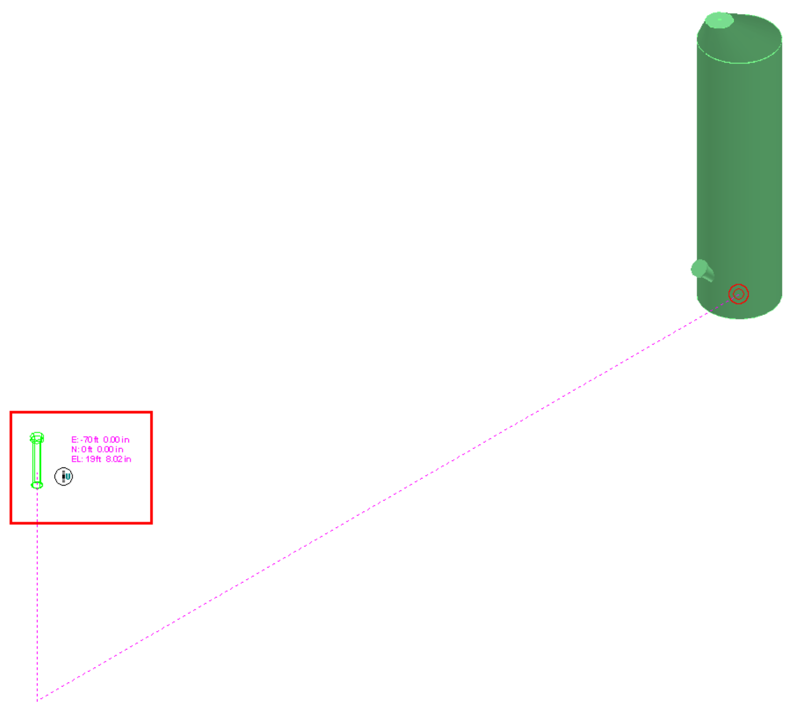


Figure 10: SmartSketch Indicator Showing the Downward Direction

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

12. Route a pipe, as shown in Figure 11, 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

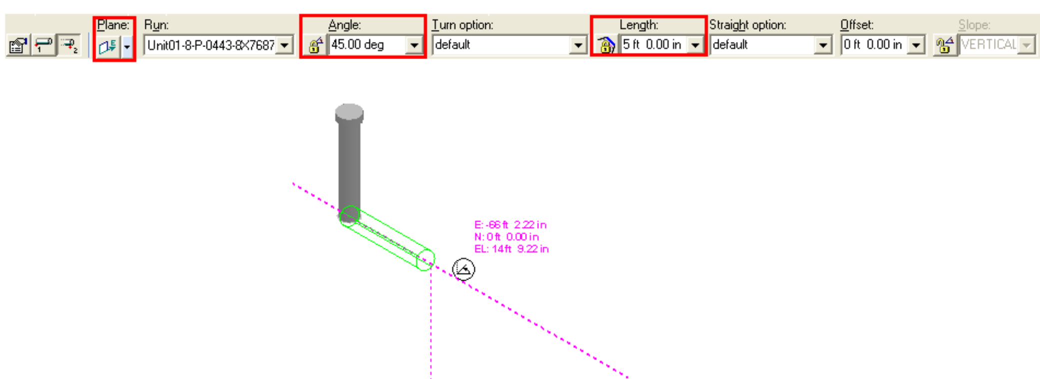


Figure 11: Specifications to Route a Pipe on the Route Pipe Ribbon

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

14. Now, set the following specifications on the **Route Pipe** ribbon:
    Length: 20 ft
    Plane: Plan-Plane

15. 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**, as shown in Figure 12.

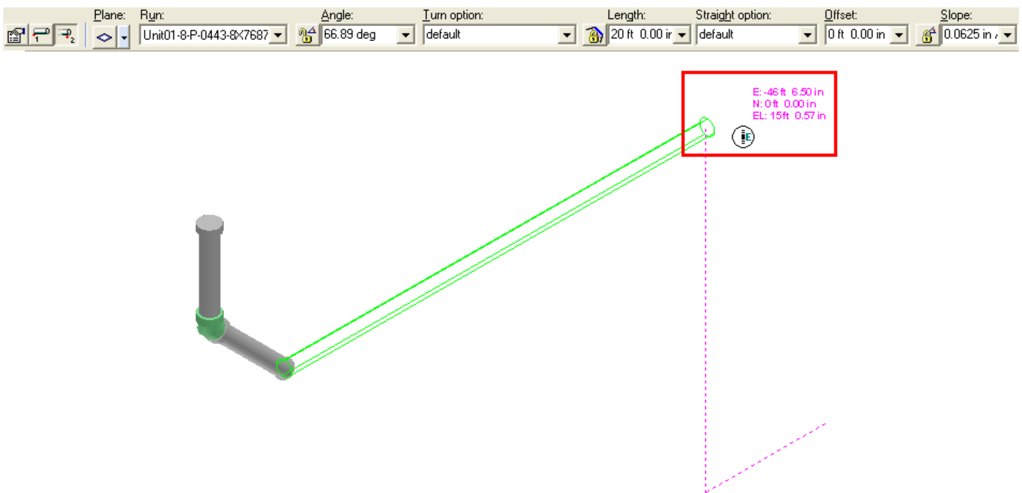


Figure 12: SmartSketch Indicator Specifying the East Direction of Routing

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

17. Set the following specifications on the **Route Pipe** ribbon:
    Length: 5 ft
    Plane: Elevation Plane: East-West
    Angle: 45 deg

18. Select the **Lock Slope** option on the **Route Pipe** ribbon to unlock the slope constraint to route the sloped pipe, as shown in Figure 13.

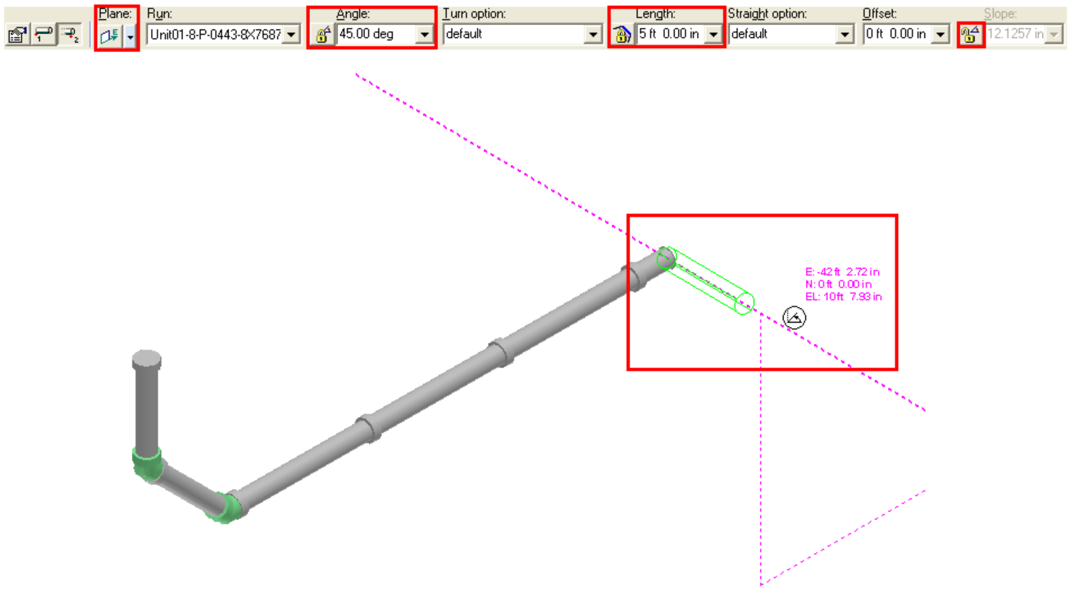


Figure 13: 5 Ft Pipe Routed

19. Click in the graphic view to place the 45 deg elbow and a piece of the pipe.

20. Now, set the following specifications on the **Route Pipe** ribbon to route a pipe in the easting direction:
    Length: 20 ft
    Plane: Plan-Plane

21. 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, as shown in Figure 14.

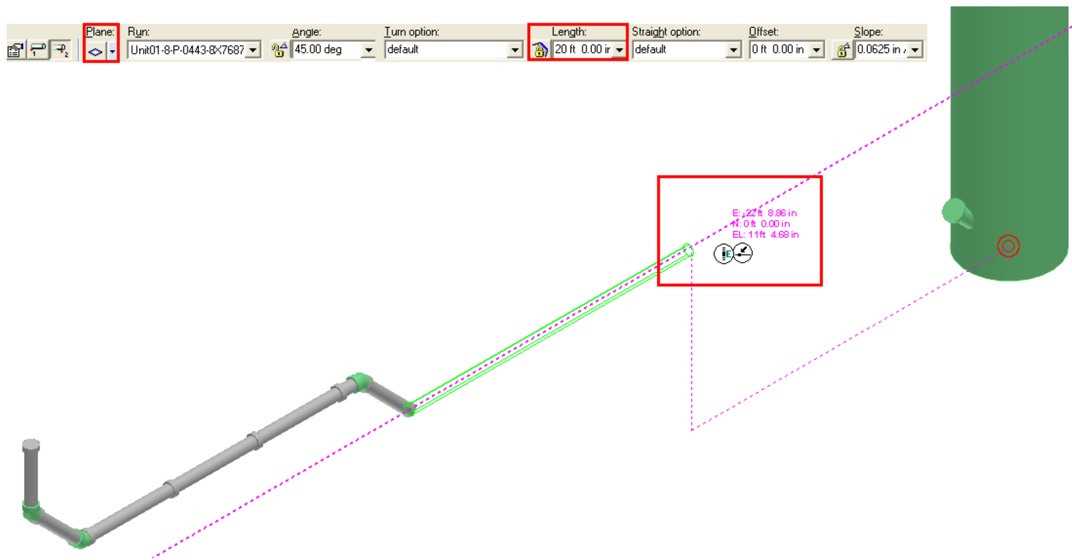


Figure 14: 20 Ft Pipe Routed

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

23. Now, route a **5 ft** by using the following specifications on the **Route Pipe** ribbon:
    Plane: Elevation Plane: East-West
    Angle: 45 deg

24. Select the **Lock Slope** option on the **Route Pipe** ribbon to unlock the slope constraint and route the sloped pipe, as shown in Figure 15.

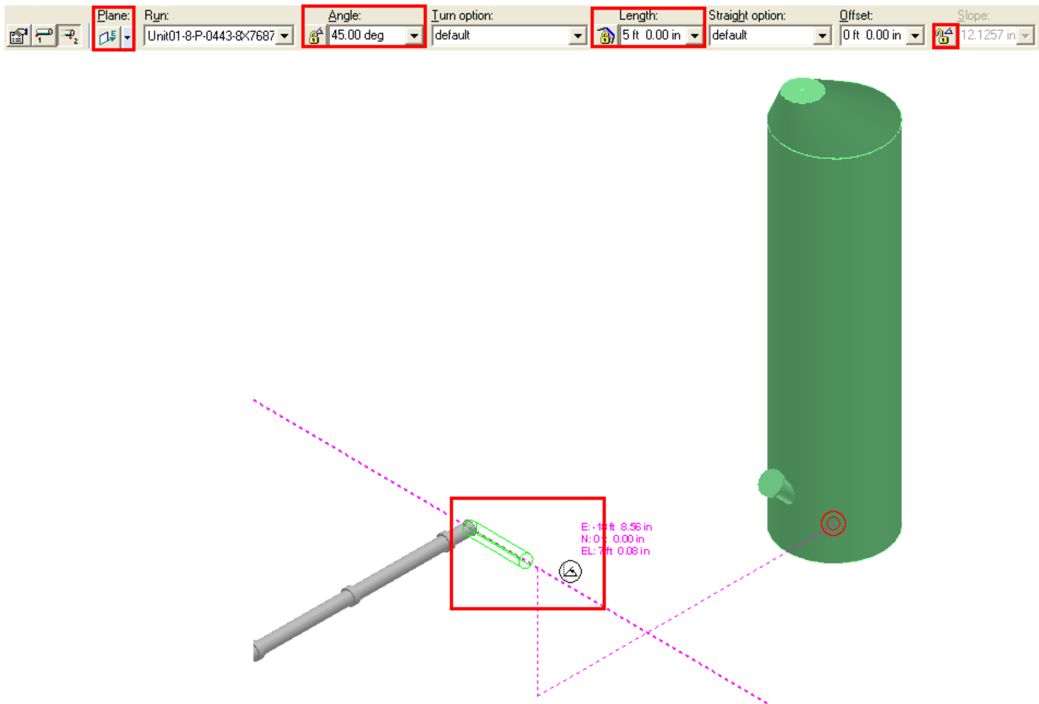


Figure 15: 5 Ft Pipe Routed

25. Click in the graphic view to place the 45 deg elbow and a piece of the pipe.

26. 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.

27. 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, as shown in Figure 16, to add it to the SmartSketch stack.

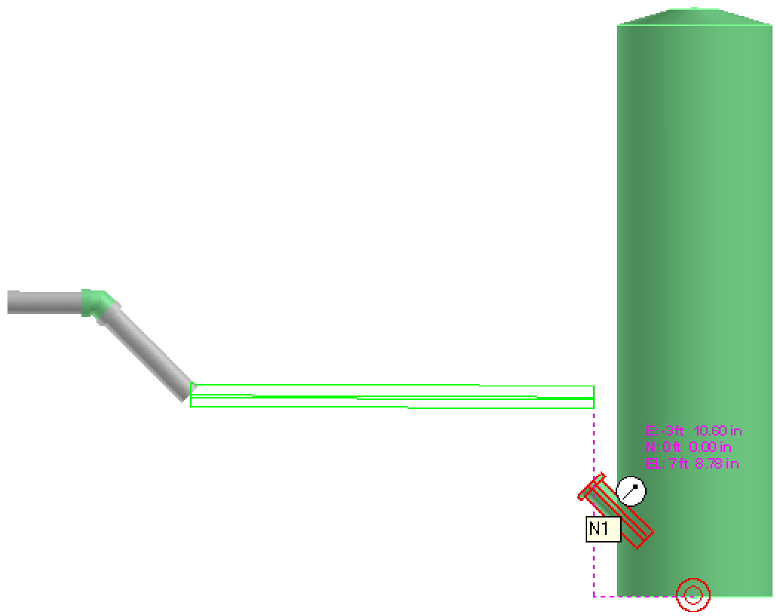


Figure 16: Pipe Extended Until the Nozzle of the Catch Basin

28. The system will display a projection line that indicates the centerline of the piping nozzle. 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, as shown in Figure 17.

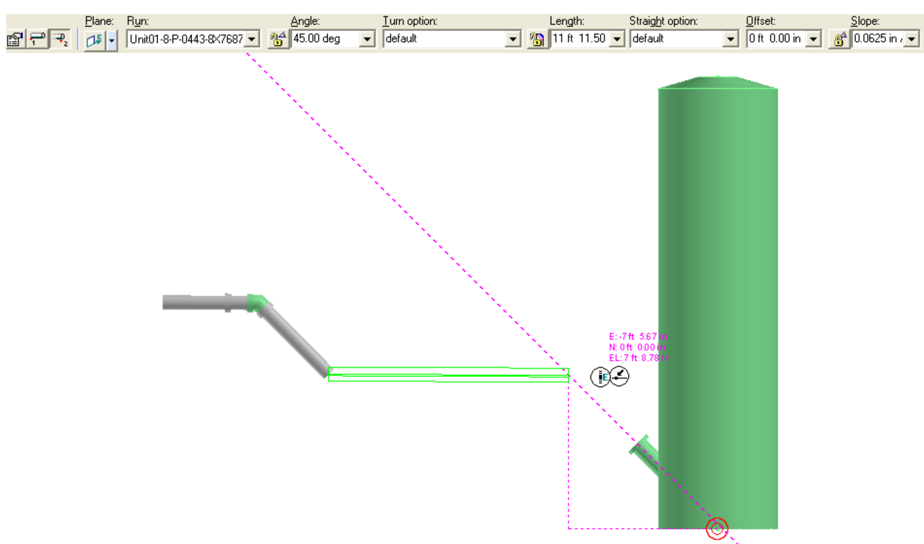


Figure 17: Projection Line Indicating the Centerline of the Piping Nozzle

29. Click to accept the placement of this pipe.

30. Now, select **No Plane** 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.

31. Locate the piping nozzle **N1** until SmartSketch displays point port glyph, as shown in Figure 18.

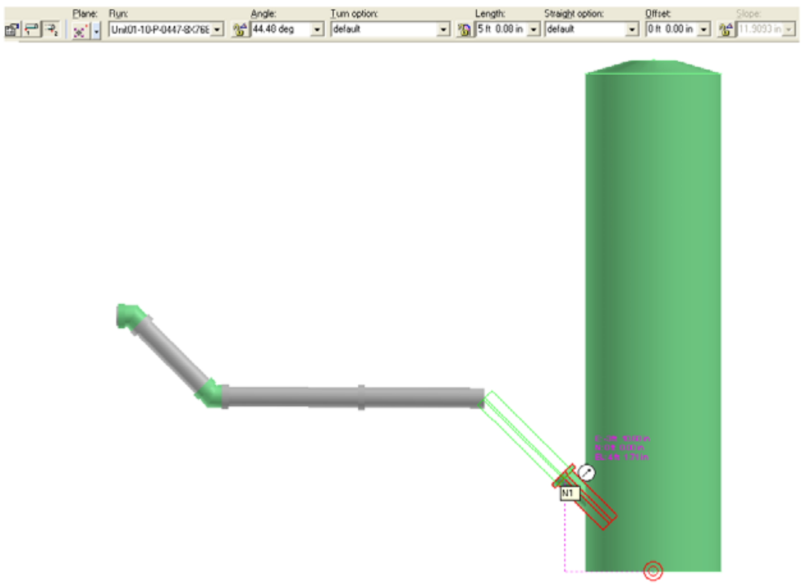


Figure 18: Locating the Piping Nozzle By Using the Point Port Glyph

32. Click in the graphic view to finish routing the main sewer line.

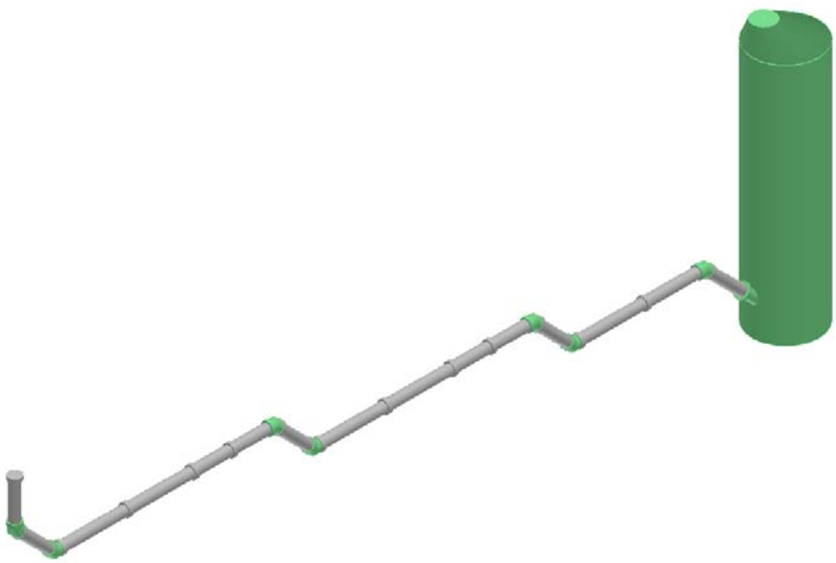


Figure 19: Routed Main Sewer Line

33. Now, click the **Route Pipe** button on the vertical toolbar to start routing the lateral sewer line.

34. Select the **Reposition Target** option on the **PinPoint** ribbon to define a reference point to route the lateral sewer line. Place the PinPoint target at the midpoint of the sloped pipe, as shown in Figure 20.

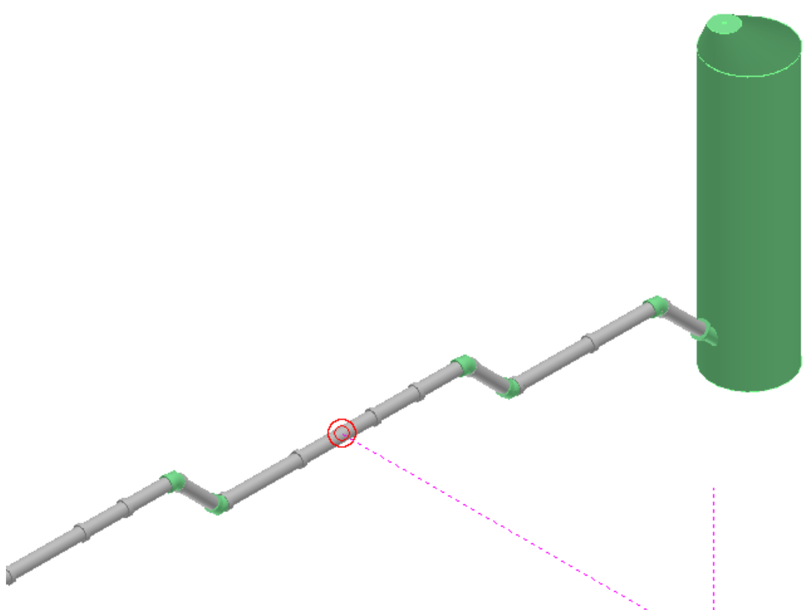


Figure 20: Mid Point of the Sloped Pipe as the Reference Target for Routing the Lateral Sewer Line

35. 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, as shown in Figure 21. Press the function key **F8** to lock the **El** drop-down list by using the cleanout open port.

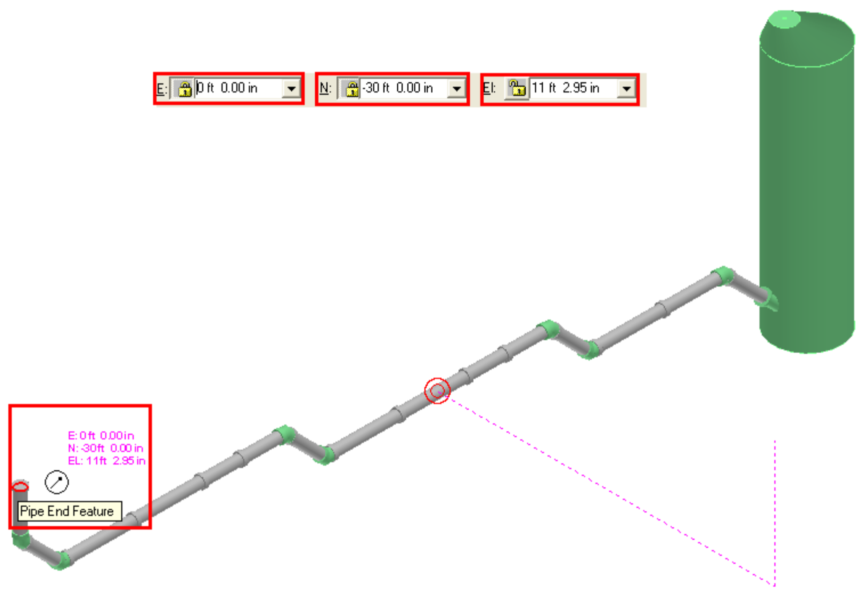


Figure 21: Mid Point of the Sloped Pipe

36. The **New Pipe Run** dialog box appears. Select the following specifications in the **New Pipe Run** dialog box, as shown in Figure 22, 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
Correlation Status: Not correlated
Correlation Basis: Correlate object

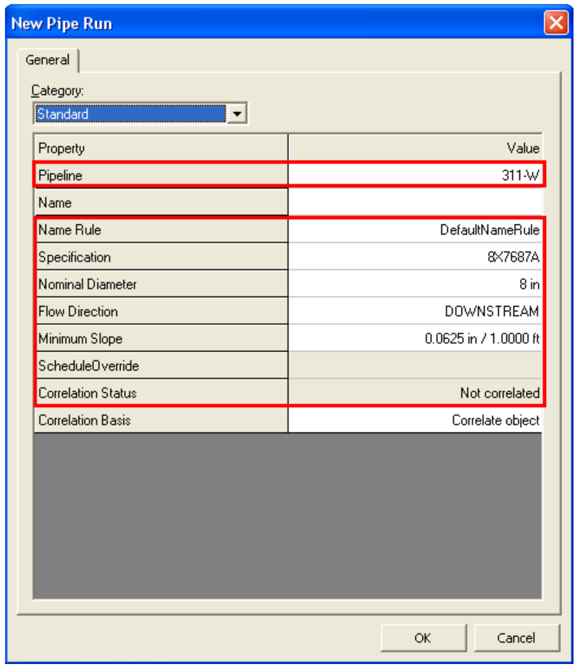


Figure 22: New Pipe Run Dialog Box

37. The **Specify Slope Direction** dialog box appears. 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, as shown in Figure 23.

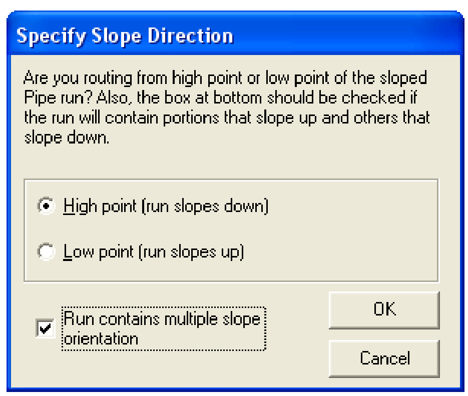


Figure 23: Specify Slope Direction Dialog Box

38. Click **OK** to close the **Specify Slope Direction** dialog box.

39. Now, set the following specifications on the **Route Pipe** ribbon:
    Length: 4 ft
    Plane: No Plane

40. Also, click the **Lock Slope** icon 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, as shown in Figure 24.

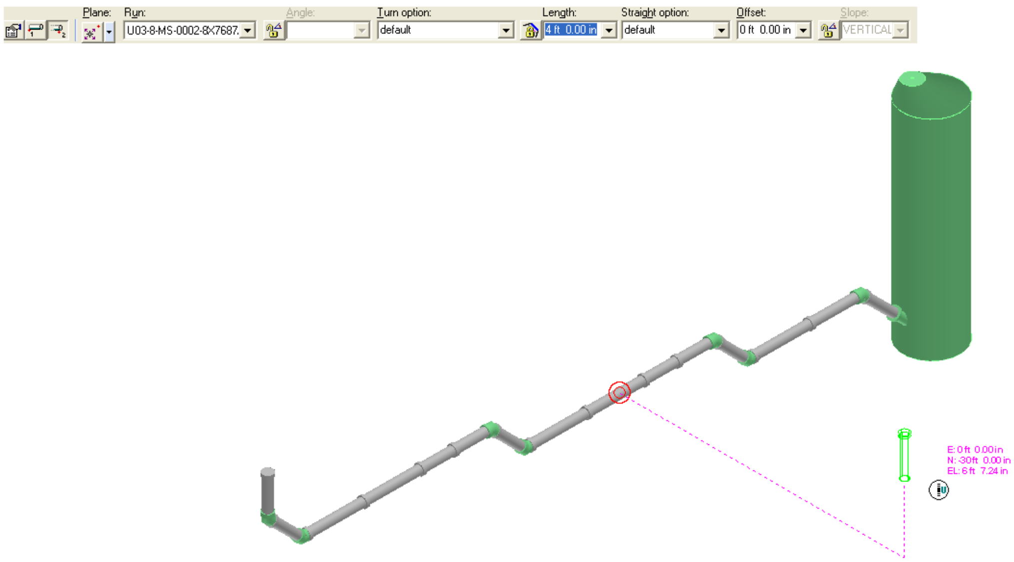


Figure 24: Pipe Routed in the Downward Direction

41. Click in the graphic view to accept the placement of the pipe.

42. 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
    Then move the cursor in the north-south direction, as shown in Figure 25.

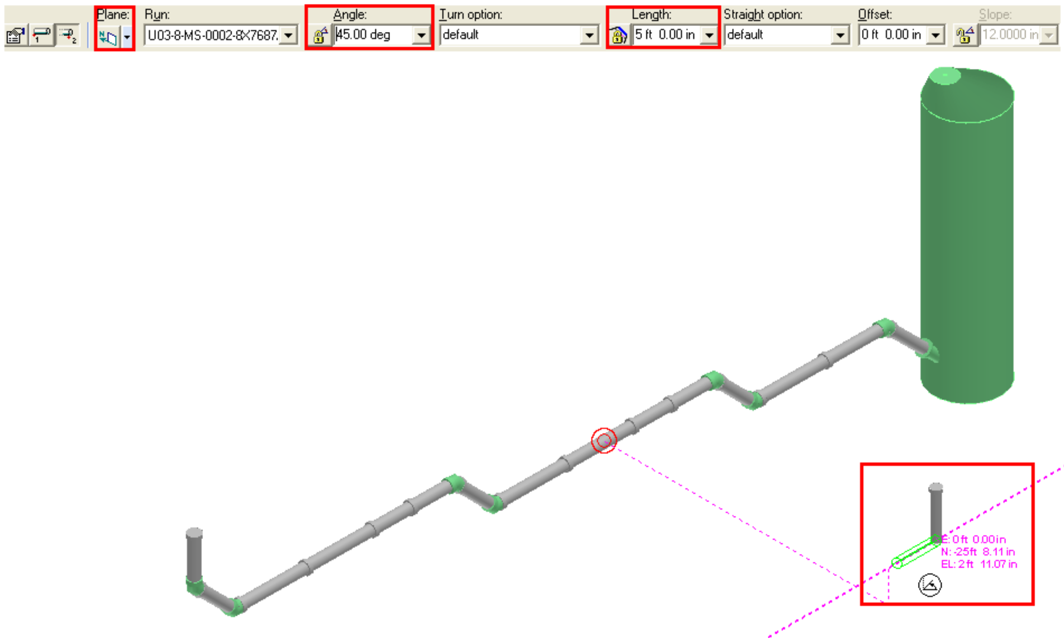


Figure 25: Pipe Routed in the North-South Direction

43. Click in the graphic view to place the 45 deg elbow and a piece of the pipe.

44. 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

45. 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 **N** glyph. This glyph indicates that you are going in the northing direction, as shown in Figure 26.

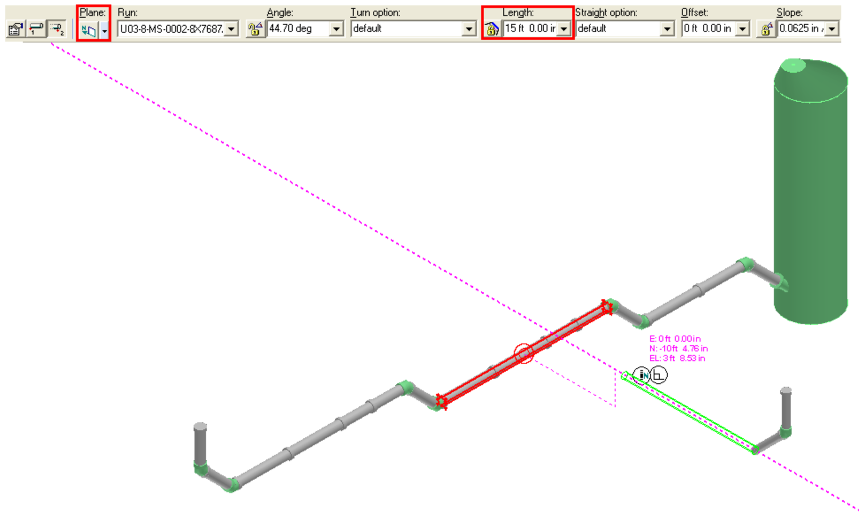


Figure 26: Pipe Routed in the Northing Direction

46. Click in the graphic view to accept the placement of the pipe.

47. Now, route a **3 ft** pipe, as shown in Figure 27, by using the following specifications on the **Route Pipe** ribbon:
    Plane: Elevation Plane: North-South
    Angle: 45 deg
    Lock Slope option: Unlock

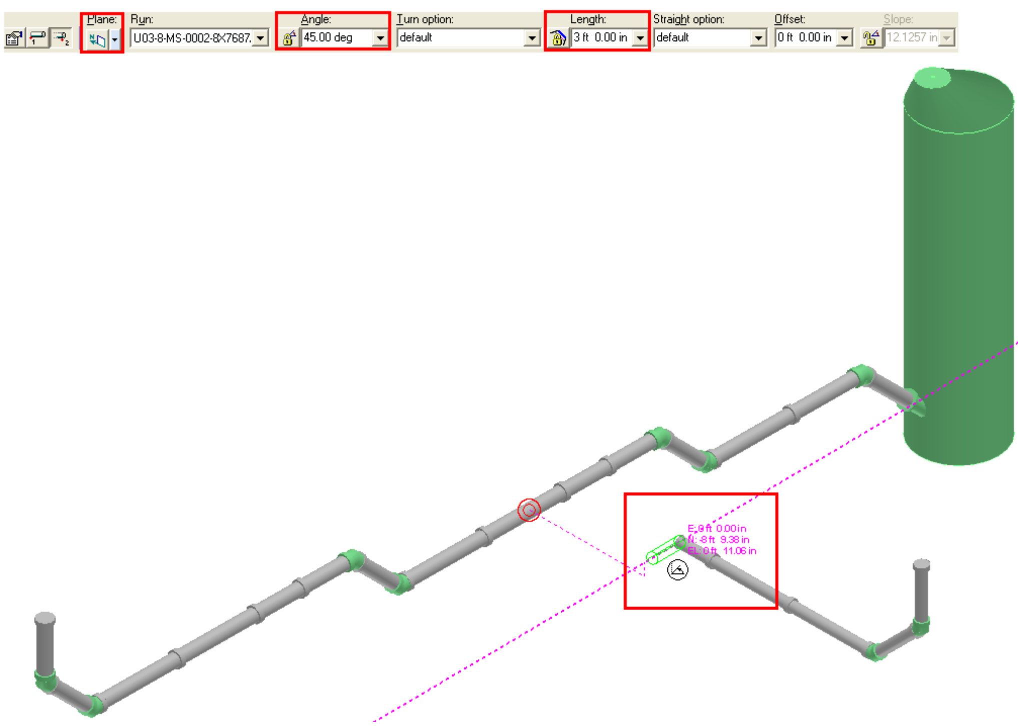


Figure 27: Routing the Pipe

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

49. Now, route another pipe by using the following specifications on the **Route Pipe** ribbon:
    Length: 3 ft
    Plane: Elevation Plane: No Plane

50. Select the **Lock Slope** icon to constrain the routing to the minimum slope value.

51. Click in the graphic view to accept the placement of this small sloped pipe, as shown in Figure 28. Right-click to terminate the **Route Pipe** command.

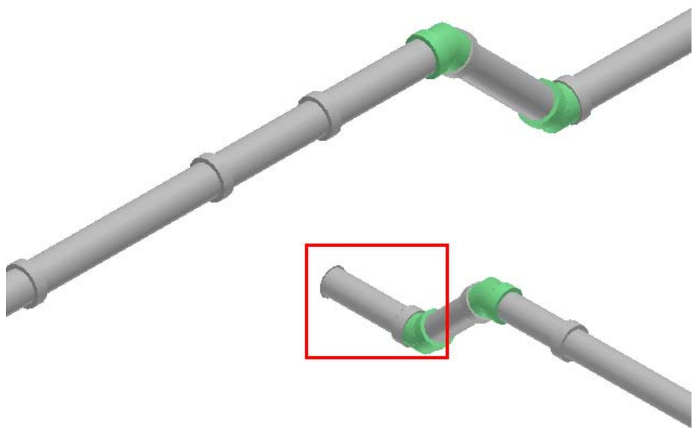


Figure 28: 3 Ft Pipe Routed

52. Select the **Reposition Target** option on the **PinPoint** ribbon and place the target at the end of the sloped pipe, as shown in Figure 29.

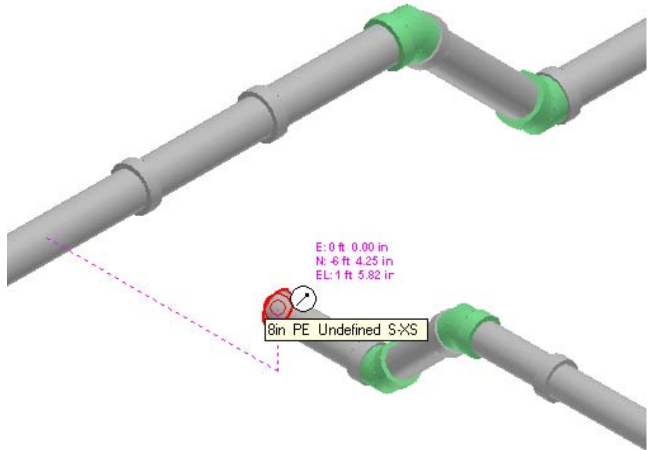


Figure 29: Repositioning of the Target

53. Select the **Relative Tracking** option and key in **0 ft** for **E** on the **PinPoint** ribbon.

Figure 30: PinPoint Ribbon

54. Now, click the **Route Pipe** button on the vertical toolbar and position the cursor on the **Pipe Straight Feature** of the main sewer line to start routing the lateral sewer line, as shown in Figure 31.

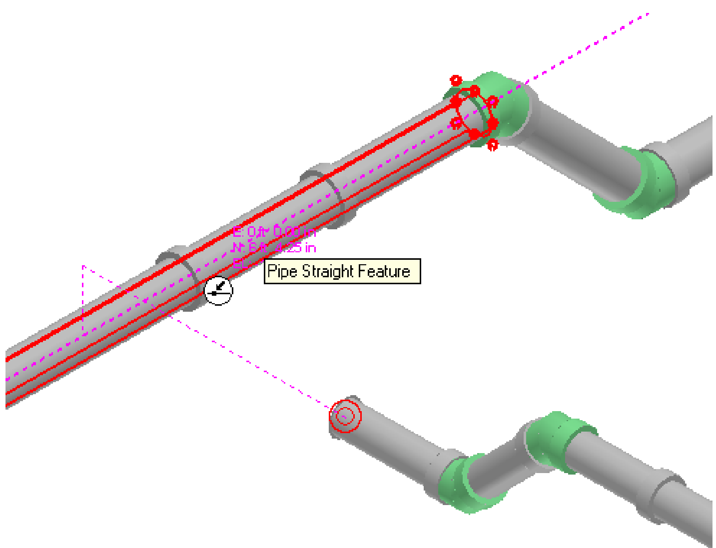


Figure 31: Pipe Straight Feature of the Main Sewer Line

55. The system displays a projection line and locates the intersection point between the sloped pipe and the main sewer line. Left-click to start routing the pipe.

56. The **New Pipe Run** dialog box appears. 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
    ScheduleOverride: <undefined value>
    Correlation Basis: Correlate object

57. The **Specify Slope Direction** dialog box appears. 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 on the **Specify Slope Direction** dialog box. Click **OK** to close the **Specify Slope Direction** dialog box.

58. 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.

59. On the **PinPoint** ribbon, select the **Spherical Coordinates** option and select the **Down - 45 deg** option in the **Vertical** drop-down list.

60. 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, as placed in Figure 32.

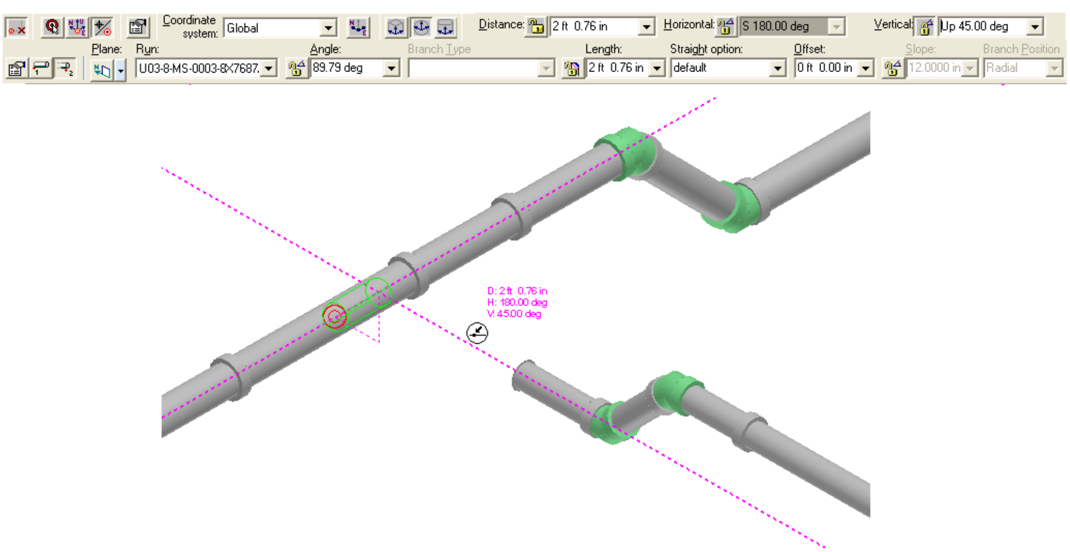


Figure 32: SmartSketch Indicator Showing the Intersection Point

61. Click in the graphic view to place the data point. The system places a reducing tee and a small pipe, as shown in Figure 33. Right-click to terminate the **Route Pipe** command.

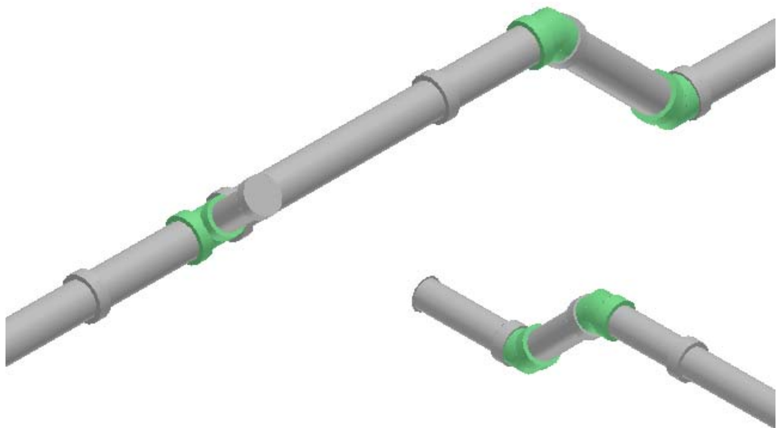


Figure 33: Placed Pipe

62. Now, click the **Route Pipe** button on the vertical toolbar and position the cursor on the **End Feature** of the small sloped pipe to continue routing the lateral sewer line, as shown in Figure 34.

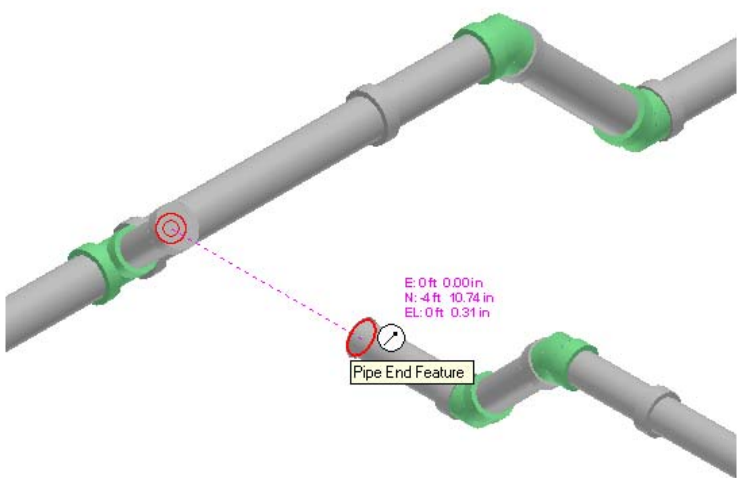


Figure 34: Pipe End Feature

63. The **Specify Slope Direction** dialog box appears, as shown in Figure 35. You need to

route a pipe that slopes downwards. Select the **High point (run slopes down)** option. Then, click **OK** to close the **Specify Slope Direction** dialog box.

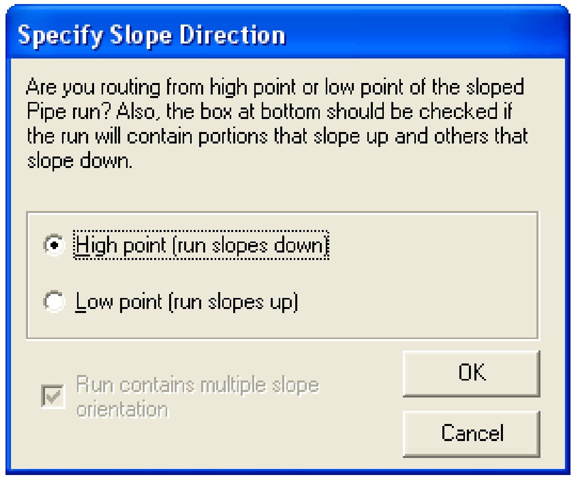


Figure 35: Specify Slope Direction Dialog Box

64. Locate the **End Feature** on the pipe coming out from the reducing tee, as shown in Figure 36.

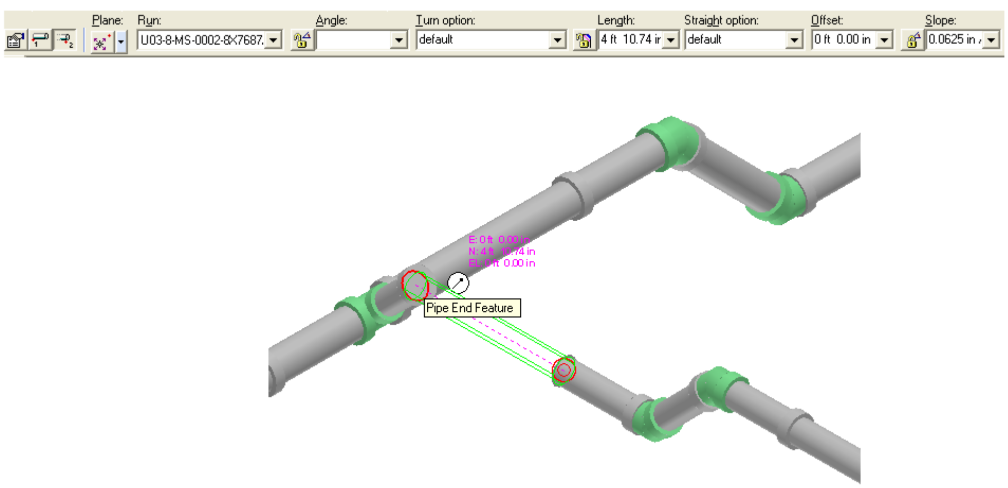


Figure 36: Pipe End Feature

65. Click in the graphic view to finish routing the lateral sewer line, as shown in Figure 37.

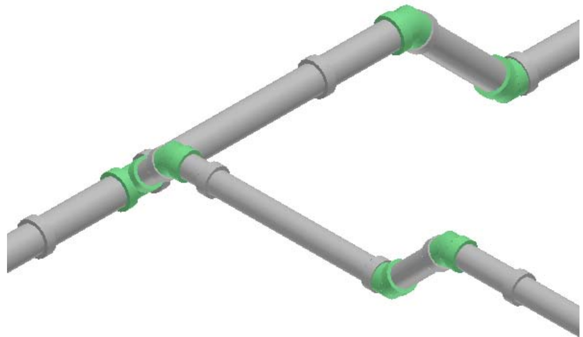


Figure 37: Placed Lateral Sewer Line

66. Now, click the **Route Pipe** button on the vertical toolbar to start routing another cleanout line.

67. Set the PinPoint mode to **Rectangular Coordinates**. Select the **Reposition Target** option on the **PinPoint** ribbon and place the target at the cleanout located in the lateral sewer line, as shown in Figure 38.

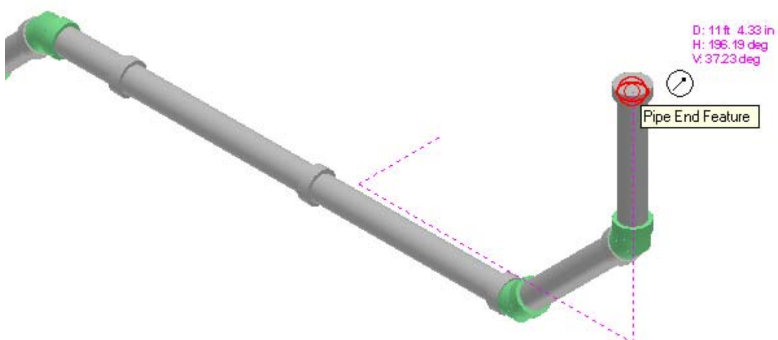


Figure 38: Repositioning of the Target Point

68. Key in **5 ft** for **E**, **-8 ft** for **N**, and **0 ft** for **El** on the **PinPoint** ribbon.

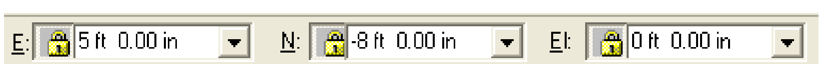


Figure 39: Specifying Coordinates on the PinPoint Ribbon

69. The **New Pipe Run** dialog box appears. 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
ScheduleOverride: <undefined value>
Correlation Basis: Correlate object

70. The **Specify Slope Direction** dialog box appears. You need to route a pipe that slopes downwards. So, select the **High point (run slopes down)** option. Click **OK** to close the **Specify Slope Direction** dialog box.

71. Now, specify the following specifications on the **Route Pipe** ribbon:
    Length: 4 ft
    Plane: No Plane

72. Select the **Lock slope** option 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, as shown in Figure 40.

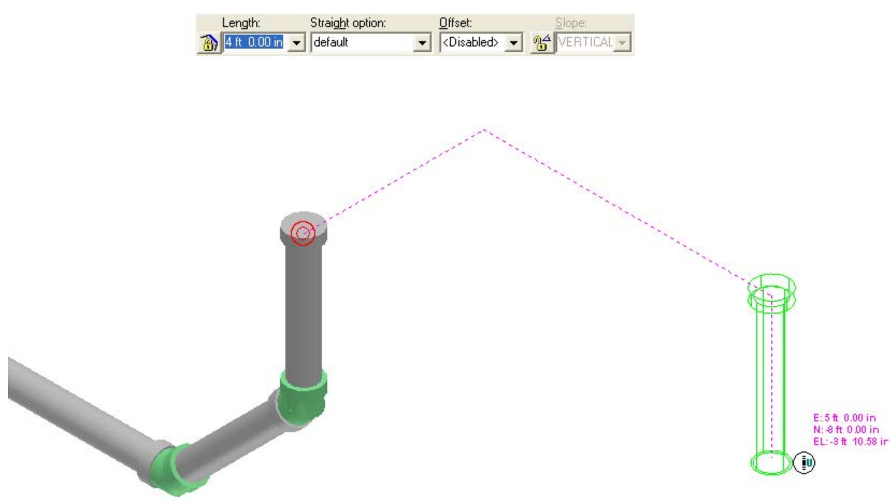


Figure 40: Routing the Pipe in the El Direction

73. Click in the graphic view to accept the placement of the pipe.

74. Route a pipe, as shown in Figure 41, by using the following specifications on the **Route Pipe** ribbon:
    Length: 3 ft
    Plane: Elevation Plane: North-South
    Angle: 45 deg

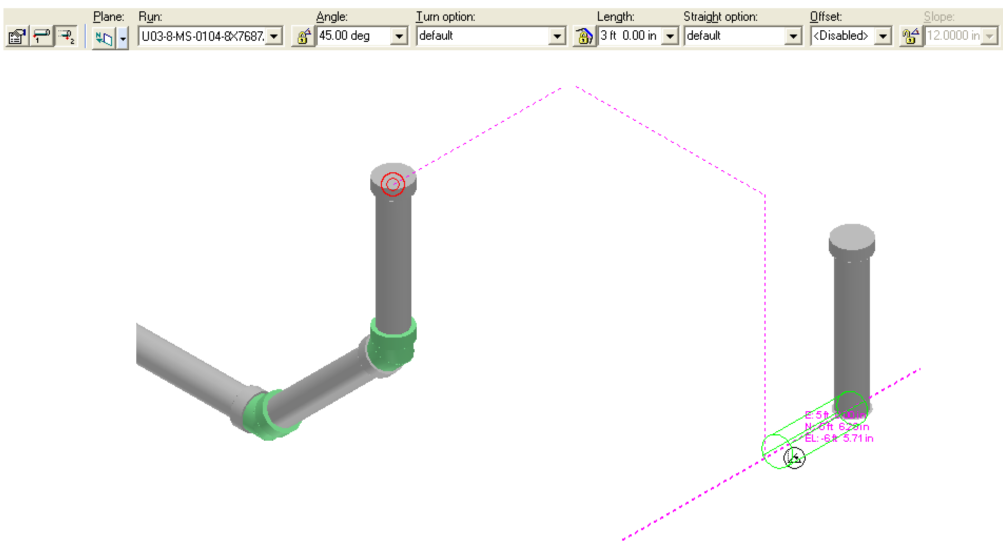


Figure 41: Routing the Pipe in the North-South Direction

75. 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

76. 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, as shown in Figure 42. Click in the graphic view to accept the placement of the pipe.

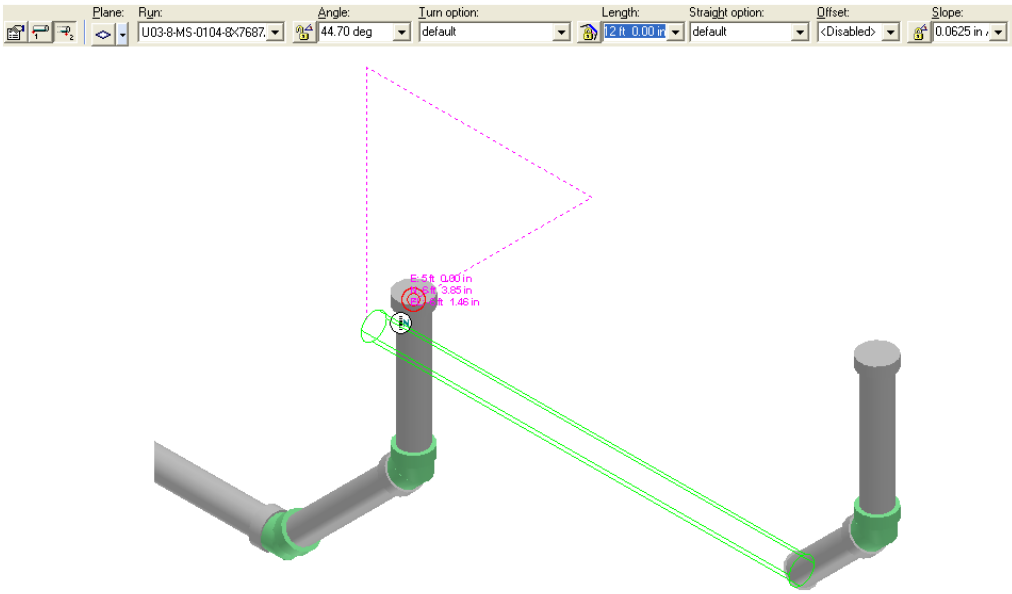


Figure 42: Routing the Pipe in the North Direction

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.

77. Select the **Plane by Three Points** option from the **Plane** drop-down list on the **Route Pipe** ribbon**.**

78. The system prompts you to specify the first point of the plane. Select a port on the lateral pipeline to define the first point of the plane, as shown in Figure 43.

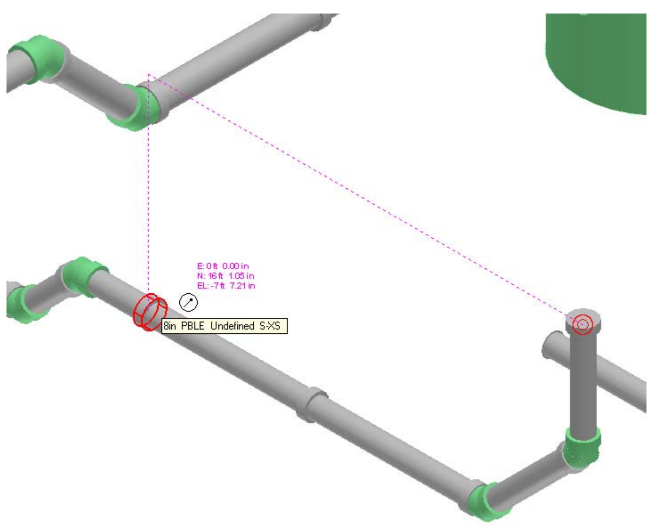


Figure 43: Defining the First Point on the Plane

79. The system prompts you to specify the second point of the plane. Select another port on the lateral pipeline, as shown in Figure 44, to define the second point of the plane.

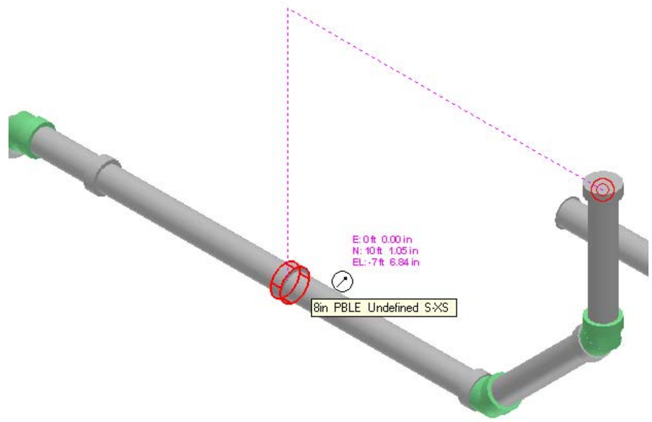


Figure 44: Defining the Second Point on the Plane

80. The system prompts you to specify the third point of the plane. Select the turning point of the routing line, as shown in Figure 45, to define the third point of the plane.

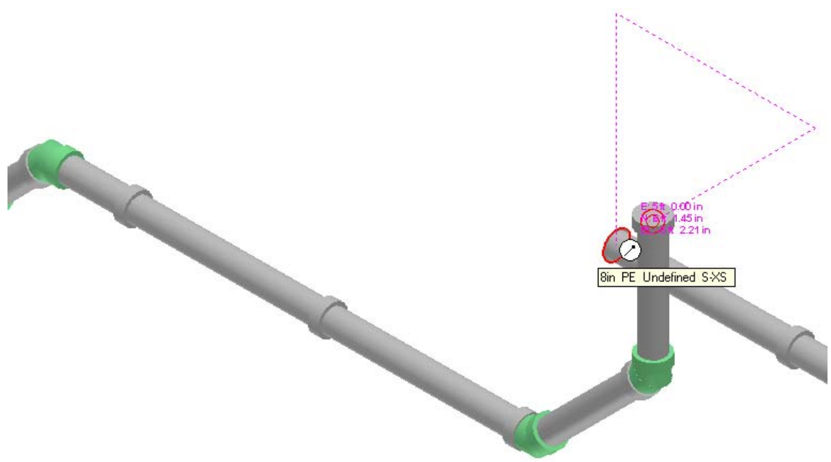


Figure 45: Defining the Third Point on the Plane

81. 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.

82. Position the cursor point along the lateral pipeline. SmartSketch will display a point on curve glyph, as shown in Figure 46.

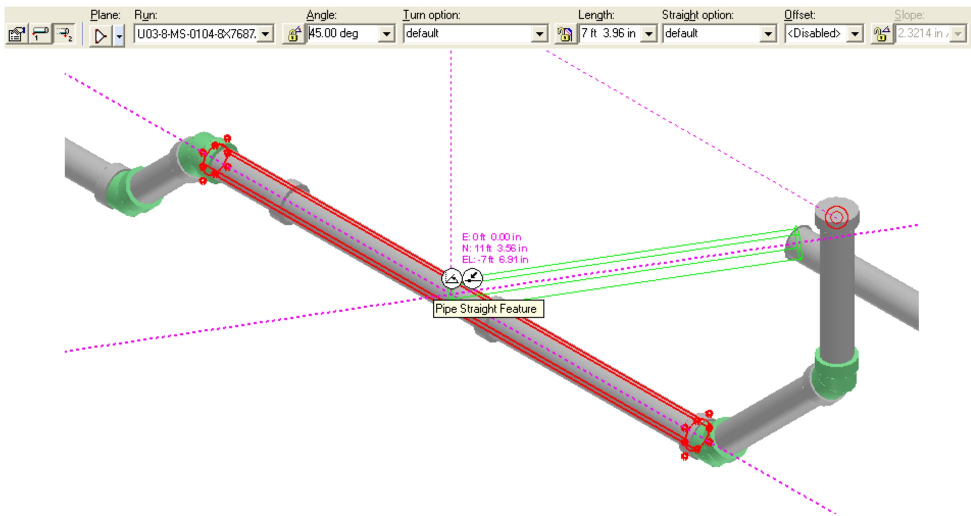


Figure 46: Point on Curve Glyph

83. 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.

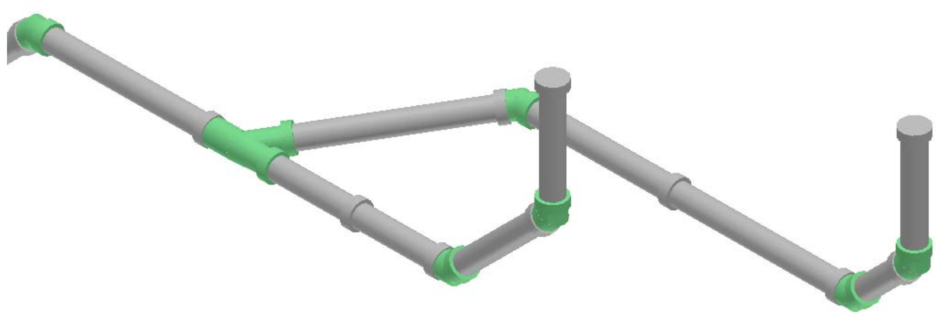


Figure 47: Routed Underground Sloped Pipe

Steps for Routing a Sloped Pipe Run:

Exercise 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 in Figure 48.

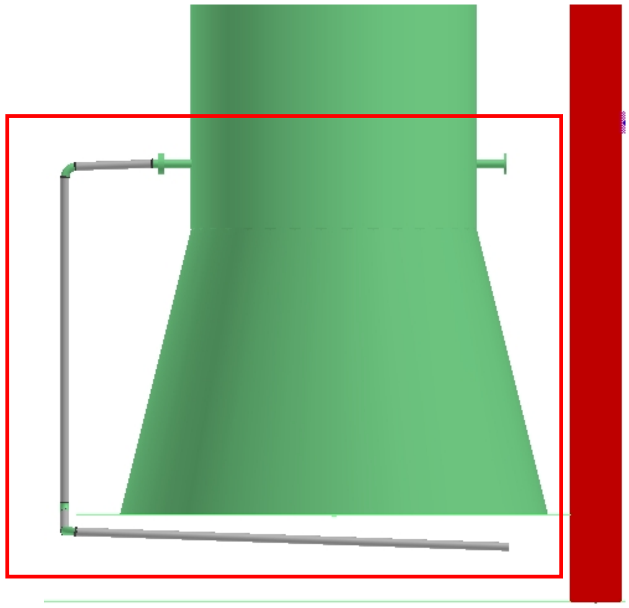


Figure 48: Final Output: Routed Sloped Pipe Run

Before beginning the 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 the **Common Views** button 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. Now, click the **Route Pipe** button on the vertical toolbar to start routing the pipe.

Figure 49: Route Pipe Button on the Vertical Toolbar

4. In the graphic view, locate and click the equipment nozzle **D1**, as shown in Figure 50.

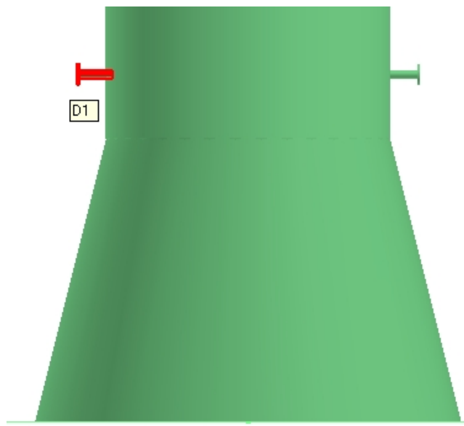


Figure 50: Located Nozzle D1

5. The **New Pipe Run** dialog box appears, as shown in Figure 51. 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 as follows:

    Pipeline: 302-W
    Name Rule: DefaultNameRule
    Specification: 1C0031
    Nominal Diameter: 2 in
    Flow Direction: BIDIRECTIONAL
    Minimum Slope: 2 deg
    ScheduleOverride: <undefined value>
    Correlation Status: Not Correlated
    Correlation Basis: Correlate object

6. The software changes the **Minimum Slope** values to the current display unit of measurement 0.42 in/1.00 ft. Click **OK** to close the **New Pipe Run** dialog box.

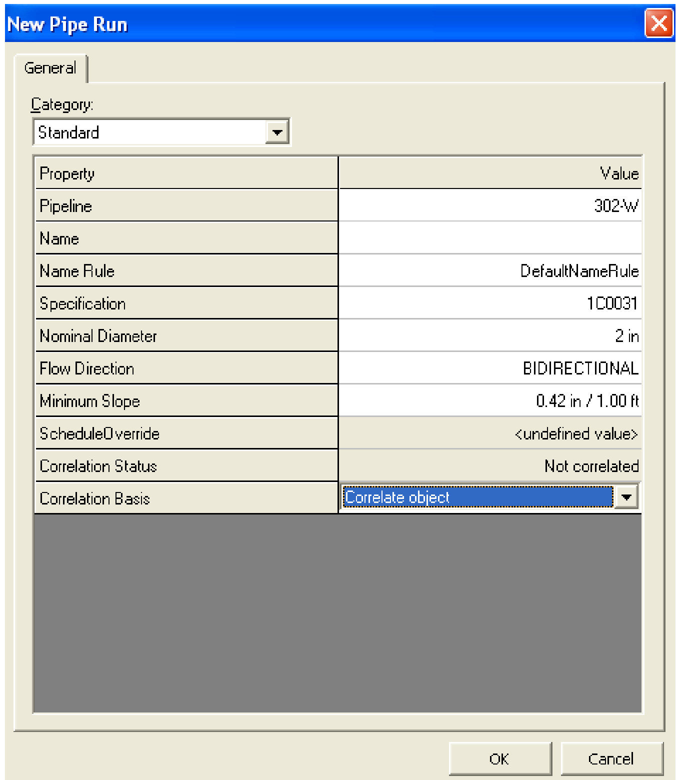


Figure 51: New Pipe Run Dialog Box

7. The **Specify Slope Direction** dialog box appears, as shown in Figure 52. You need to route a pipe that slopes downwards. Select the **High point (run slopes down)** option and click **OK** to close the dialog box.

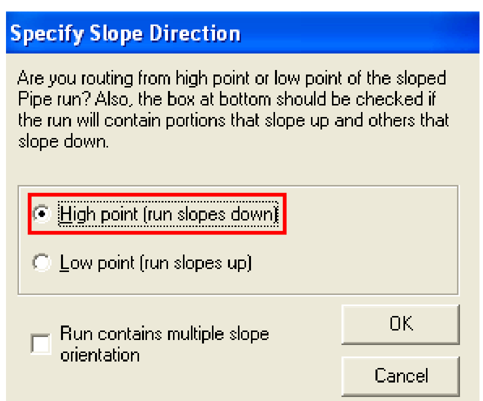


Figure 52: Specify Slope Direction Dialog Box

8. You now see an outline of a pipe in your model. 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. Click in the graphic view to place the pipe, as shown in Figure 53.

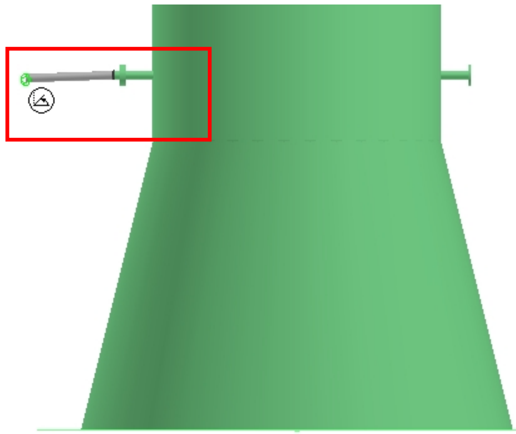


Figure 53: Placed Pipe

9. On the **Route Pipe** ribbon, unlock the **Angle**, **Slope**, and **Length** fields. Key in **2 ft** in the **Offset** field.

Figure 54: Route Pipe Ribbon

10. Click the **Plane** drop-down arrow on the **Route Pipe** ribbon and select **Elevation Plane: East-West** to route the pipe.

11. Move your cursor downwards to locate the grid line highlighted in Figure 55 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.

12. Move your cursor straight upwards. The system displays a projection line that indicates **2 ft** offset from the selected grid line. Then, click in the graphic view to place the pipe, as shown in Figure 55.

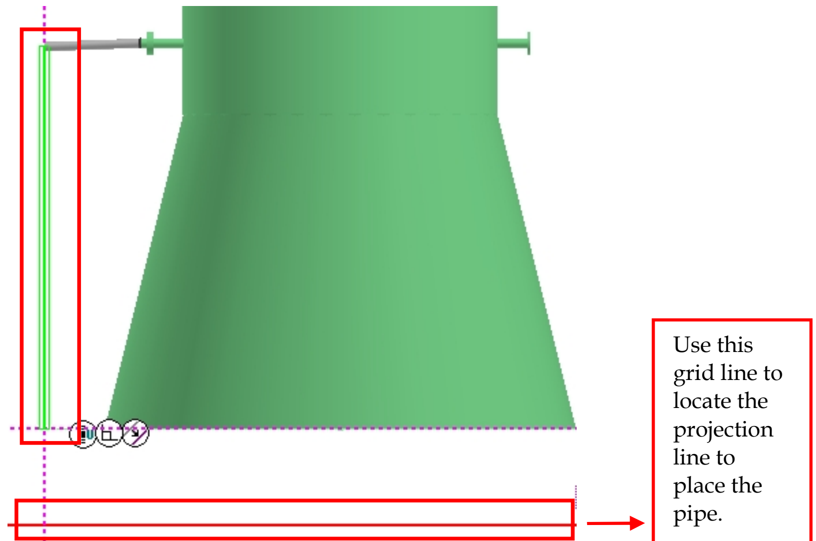


Figure 55: Placed Pipe

13. Click the **Common Views** button on the toolbar to switch to the **Looking Plan** view. On the **Route Pipe** ribbon, lock the **Slope** field again.

Figure 56: Slope Field on the Route Pipe Ribbon

14. Click the **Plane** drop-down arrow on the **Route Pipe** ribbon and select **Plan Plane** to route the pipe.

15. Move the cursor upwards to locate the grid line highlighted in Figure 57 and add it to the SmartSketch list.

16. Move your cursor straight upwards. The system displays a projection line that indicates a 2 ft offset from the selected grid line. Then, click in the graphic view to place the pipe, as shown in Figure 57.

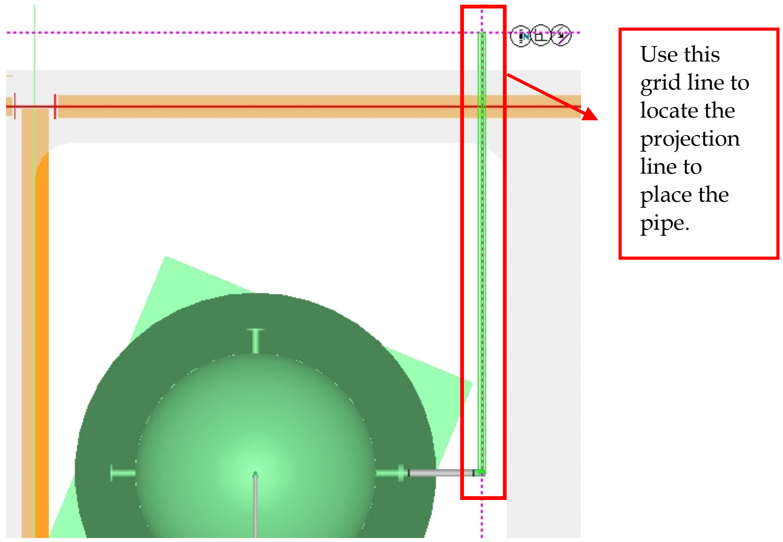


Figure 57: Placed Pipe

17. Move the cursor in the west direction to locate the grid line highlighted in Figure 58 and add it to the SmartSketch list.

18. Move your cursor straight towards the right. The system displays a projection line that indicates a **2 ft** offset from the selected grid line. Then, click in the graphic view to place the pipe, as shown in Figure 58.

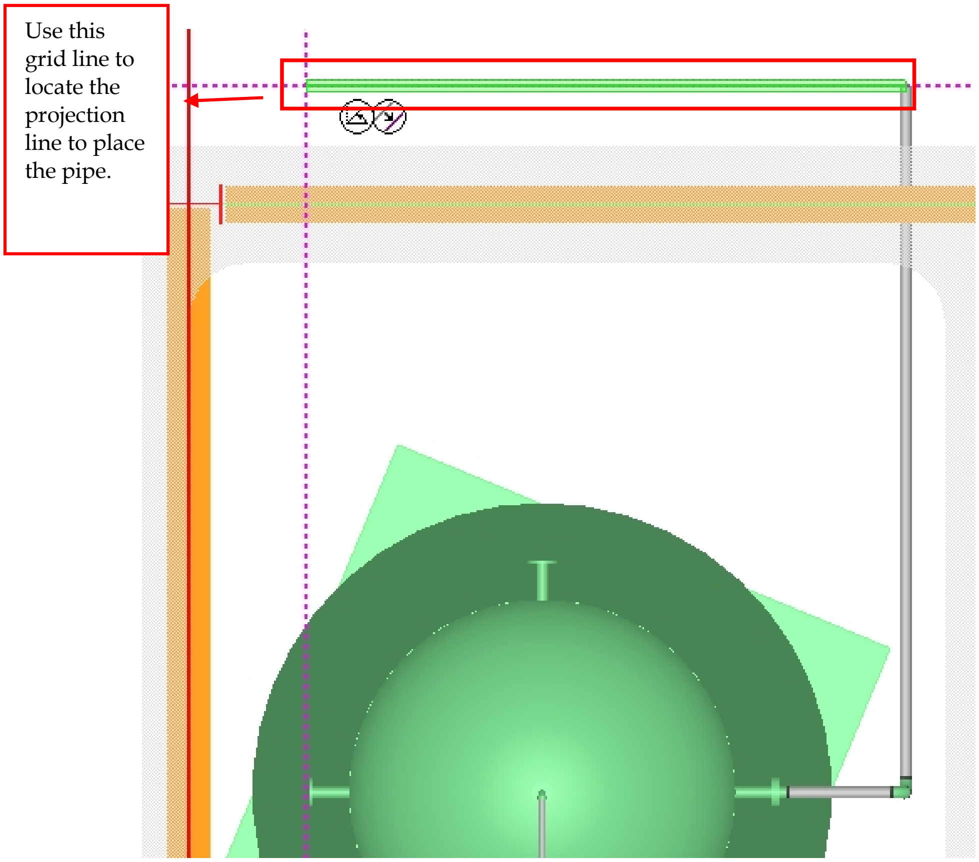


Figure 58: Placed Pipe

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

20. Click the **Common Views** button on the toolbar 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 Figure 59.

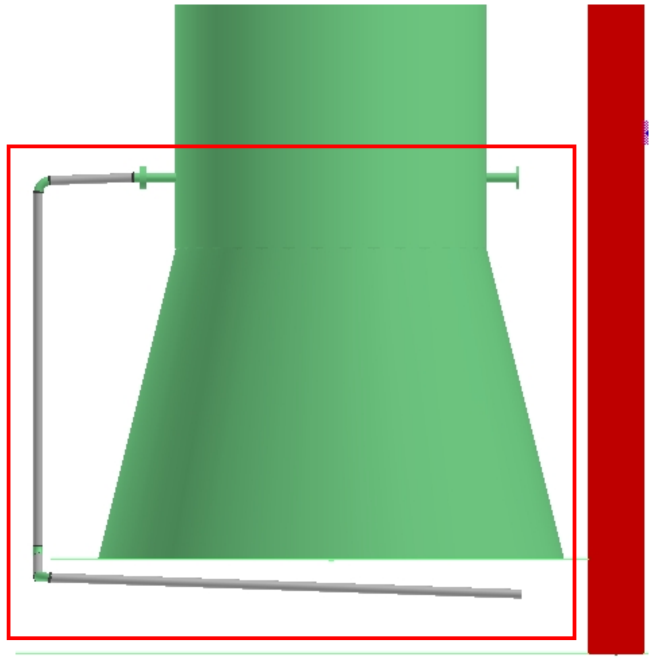


Figure 59: Final Output

After routing the sloped pipe, you use the **Insert Component** button on the vertical toolbar to insert components, where required, in the pipe. For steps to insert components in a pipe run, refer to the tutorial for the session *Inserting Components in a Pipe Run*.

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 5: Routing Pipes from P&ID

Objectives:

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

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

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run

Overview:

When designing or creating a plant in SP3D, 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, SP3D, 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 SP3D 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 SP3D, as shown in Figure 1.

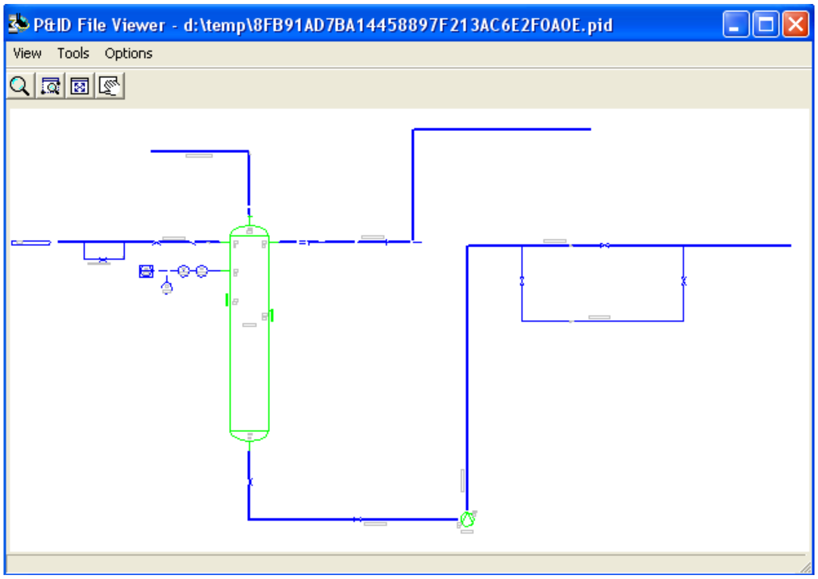


Figure 1: P&ID File Viewer Window

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 SP3D. 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 SP3D. 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 SP3D database. Once the data is stored in the SP3D database, all users of SP3D can access this data.

For more information, refer to the *Design Basis Explanation* topic in *SmartPlant 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:

Exercise 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 Figure 2.

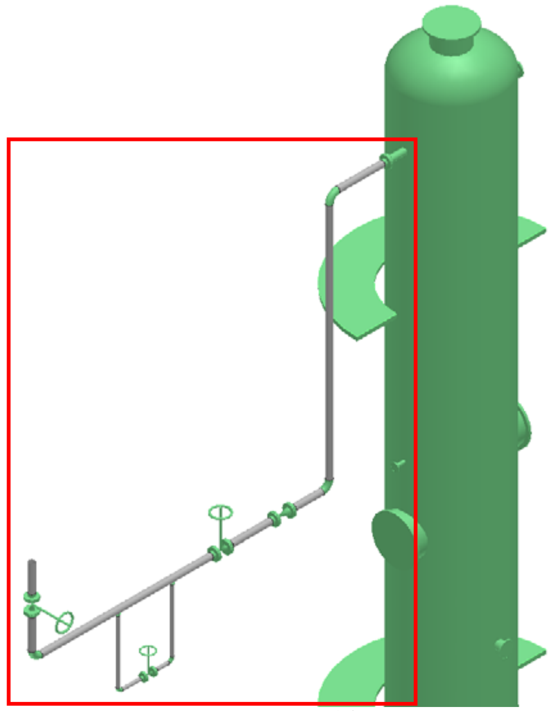


Figure 2: Pipe Runs Routed from P&ID

Steps for Routing a Pipeline from P&ID:

Route a pipeline from an equipment nozzle **A5** in Unit **U02** of your workspace by using P&ID.

Before beginning the 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.**

34. Open the P&ID file that contains the design basis of the pipeline you want to model. Click the **SmartPlant > View P&ID…** command, as shown in Figure 3.

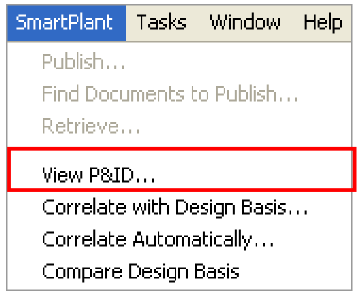


Figure 3: SmartPlant > View P&ID

35. The **View P&ID** dialog box appears. This dialog box displays a list of the P&IDs available for the model. Select the **INT01** drawing and click **Open**.

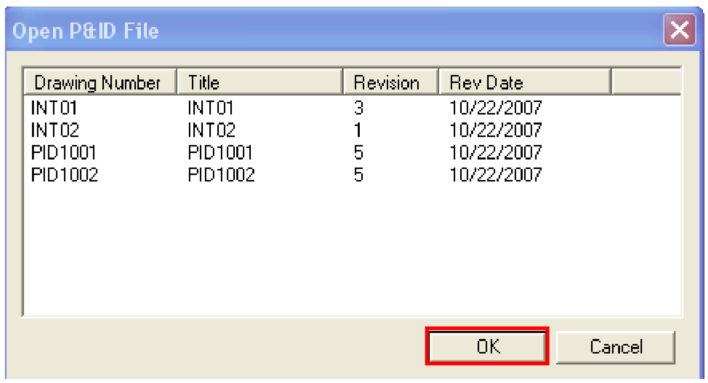


Figure 4: Open P&ID File Dialog Box

The **P&ID File Viewer** window appears.

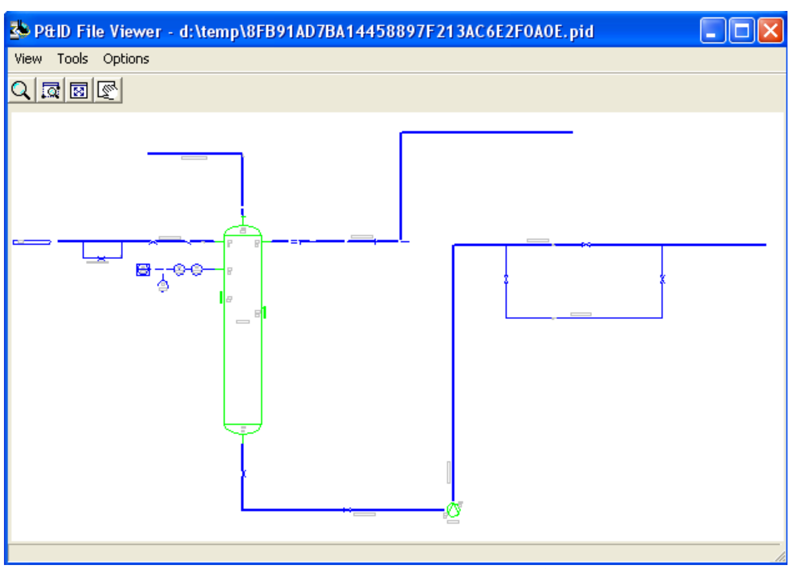


Figure 5: P&ID File Viewer Window

36. Click the **Options > Set Correlation Status Colors** command 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.

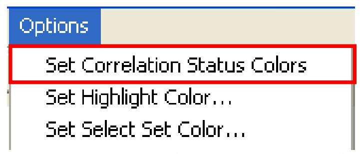


Figure 6: Options > Set Correlation Status Colors Command

37. 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 Figure 7, 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.

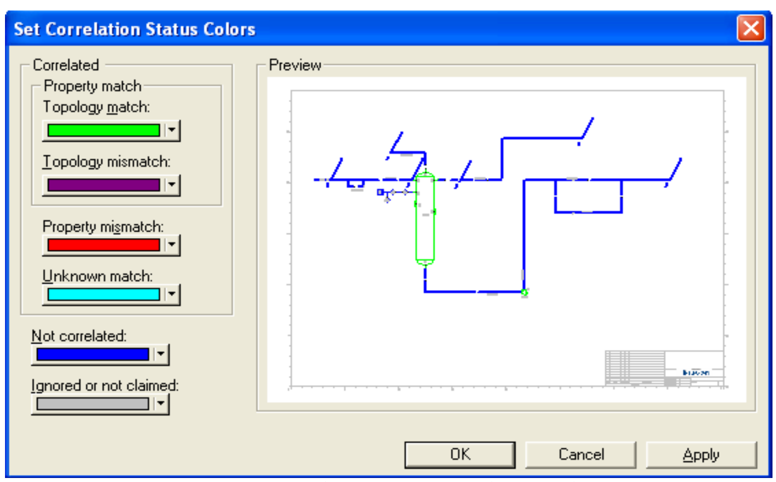


Figure 7: Set Correlation Status Colors Dialog Box

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.

38. Click **Cancel** in the **Set Correlation Status Colors** dialog box to close it.

39. Select the **All** option in the **Locate Filter** drop-down list to select all the objects in the model.

40. In the **P&ID File Viewer** window, click the equipment nozzle **A5**. The nozzle **A5** will be highlighted in the graphic view, as shown in Figure 8.

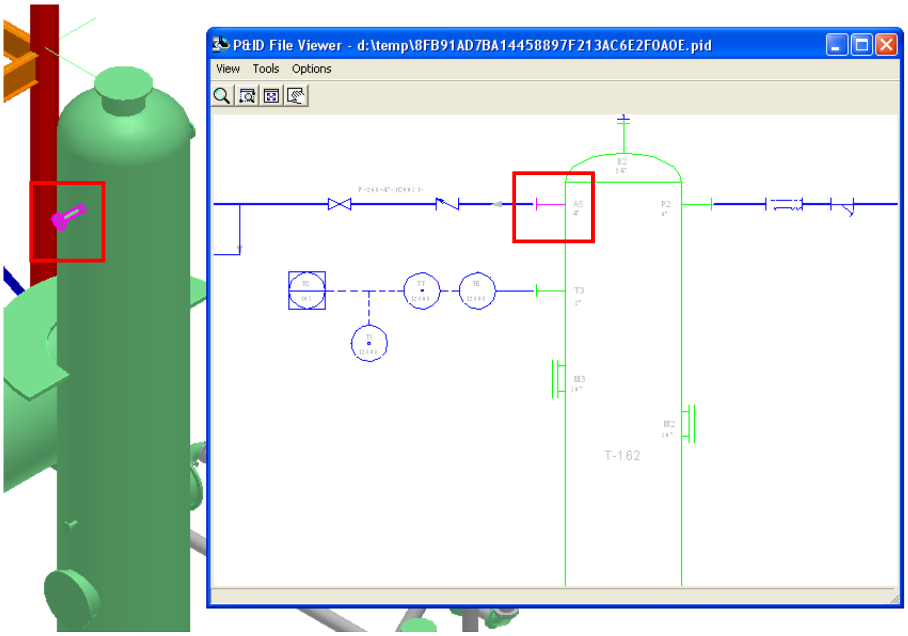


Figure 8: Selected Equipment Nozzle A5

41. Click the **Route Pipe** button on the vertical toolbar to start routing the pipe run.

42. 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. Ensure that the **New Pipe Run** dialog box has the following parameters and then click **OK** to close the dialog box:
    Pipeline: P-268
    Name: P-268-4"-1C0031
    Name Rule: User Defined
    Specification: 1C0031
    Nominal Diameter: 4 in
    Flow Direction: DOWNSTREAM
    Minimum Slope: Not Sloped
    ScheduleOverride: <undefined value>
    Correlation Status: Correlation with inconsistent data
    Correlation Basis: Correlate object

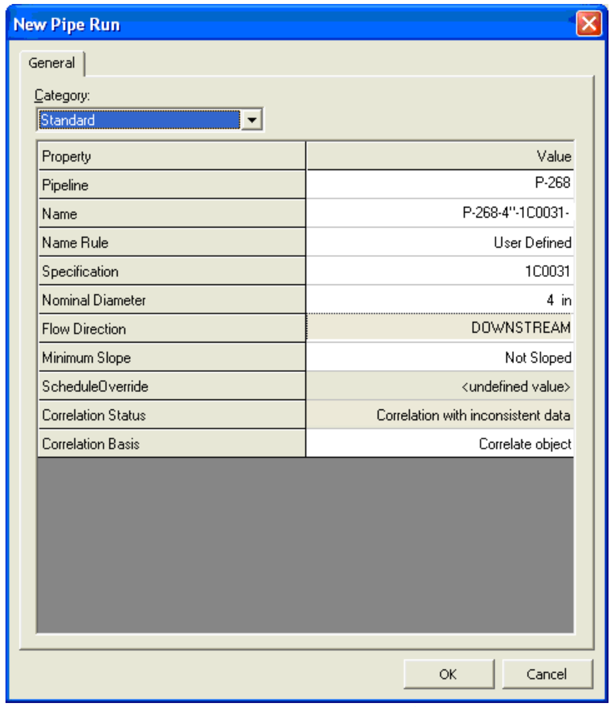


Figure 9: New Pipe Run Dialog Box

43. An outline of a pipe run will appear in the graphic view starting at pipe nozzle **A5**. Key in **5 ft** in the **Length** drop-down list on the **Route Pipe** ribbon to lock the length of the pipe run.

44. Move the cursor towards the left until SmartSketch displays the **E** glyph, as shown in Figure 10. The **E** glyph indicates that you are routing the pipe in the West direction. Click in the graphic view to place the pipe.

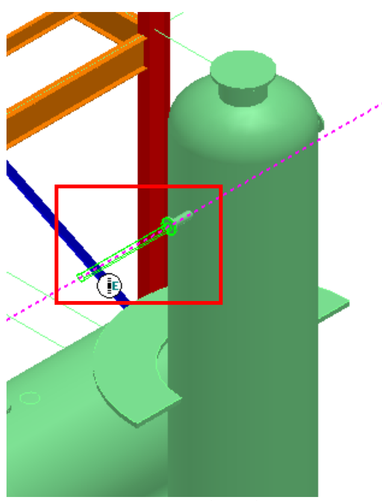


Figure 10: Placed Pipe

45. Key in **15 ft** in the **Length** drop-down list on the **Route Pipe** ribbon. Move the cursor downwards until SmartSketch displays the **U** glyph, as shown in Figure 11. If you can not position the cursor downwards, then select **No Plane** in the **Plane** drop-down list on the **Route Pipe** ribbon. Click in the graphic view to place the pipe.

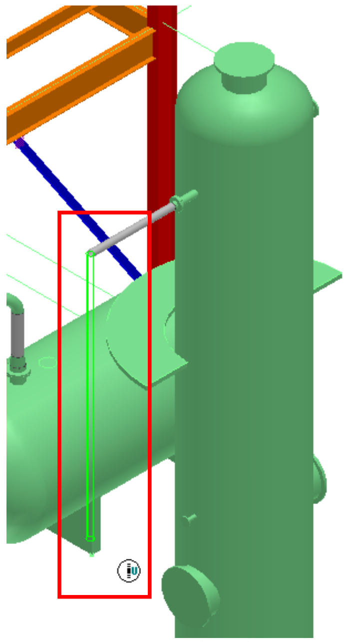


Figure 11: Placed Pipe

46. Key in **20 ft** in the **Length** drop-down list on the **Route Pipe** ribbon. Move the cursor towards the left (west direction) until SmartSketch displays the **E** glyph, as shown in Figure 12 and click in the graphic view to place the pipe.

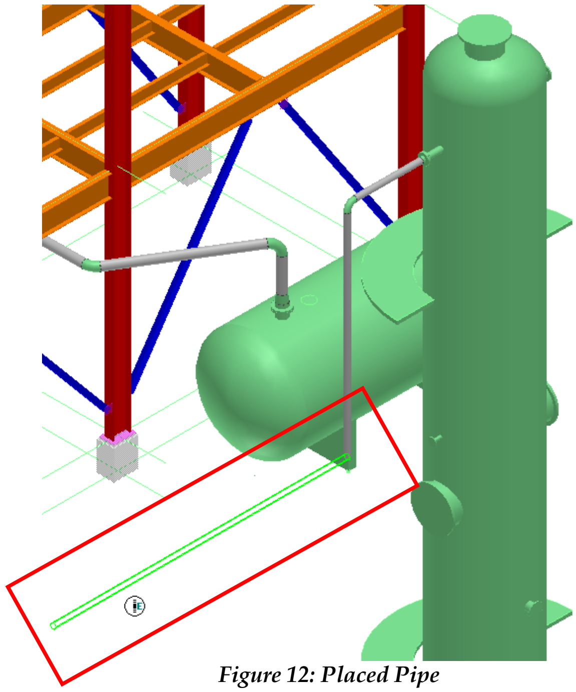


Figure 12: Placed Pipe

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

Steps to Match 3D Model Items with the P&ID Design Basis:

Open the **P&ID File Viewer** window. The pipe run you have just routed will be highlighted in red, as shown in Figure 13. 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:

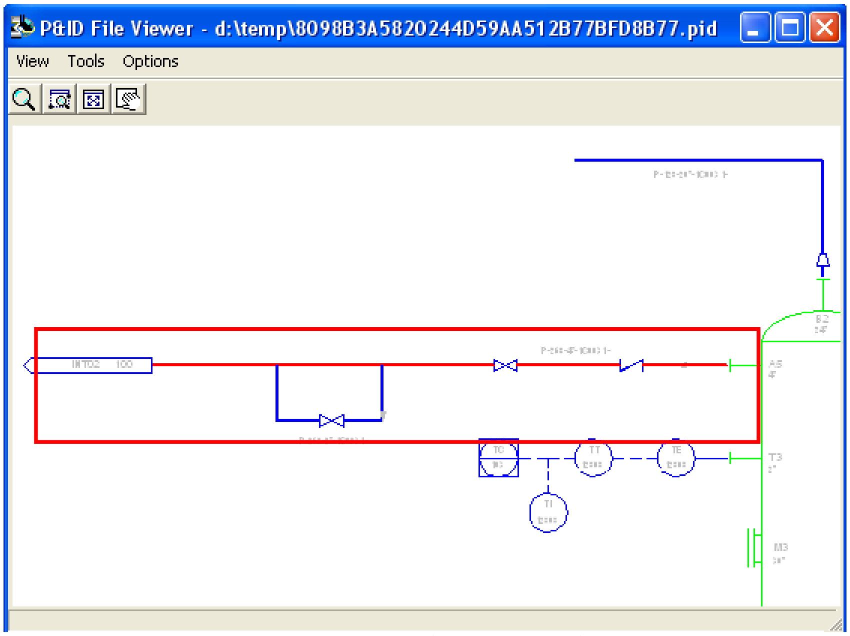


Figure 13: P&ID File Viewer Window

1. In the graphic view, select the pipe run that you have just routed. Click the **SmartPlant > Compare Design Basis** command 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, as shown in Figure 14. This indicates that the flow direction property of the pipe run does not match the design basis.

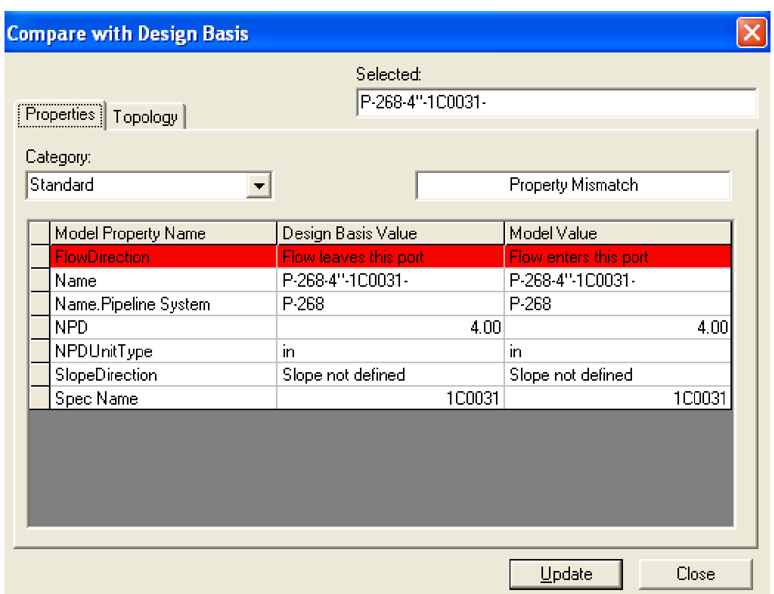


Figure 14: Compare with Design Basis Dialog Box

2. Click **Update** in the **Compare with Design Basis** dialog box 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.

Steps to Correlate and Compare the model items with the Design Basis:

Exercise Objective: in this exercise you will click to view the **Topology** tab of the **Compare with Design Basis** dialog box. A couple of columns under the **Topology** tab are highlighted in red, as shown in Figure 15. 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.

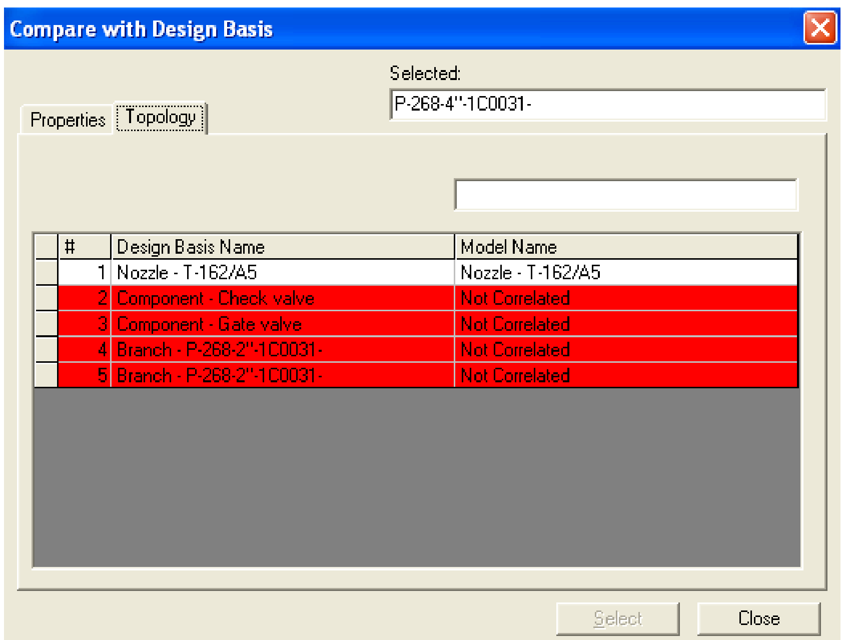


Figure 15: Topology Tab in the Compare with Design Basis Dialog Box

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. Activate the **PinPoint** ribbon by selecting the **PinPoint** button on the **Common** toolbar.

3. Select the **Reposition Target** option on the **PinPoint** ribbon. 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 the **Route Pipe** button on the vertical toolbar.

6. Position the cursor along the **Pipe Straight Feature,** as shown in Figure 16 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.

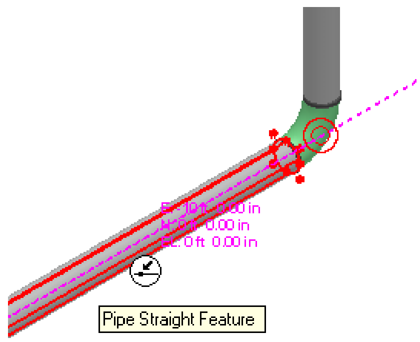


Figure 16: Pipe Straight Feature

7. The **P&ID File Viewer** window appears. In the **P&ID File Viewer** window click the bypass line, as shown in Figure 17.

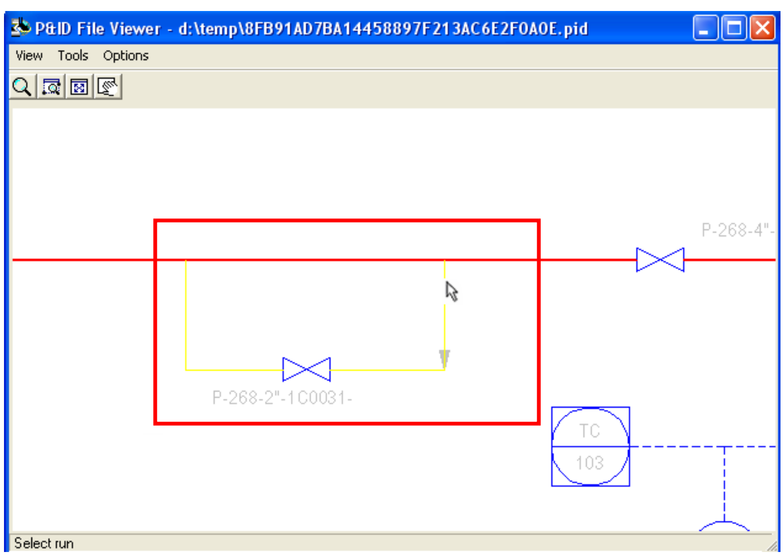


Figure 17: Bypass Line in the P&ID File Viewer Dialog Box

8. The **New Pipe Run** dialog box appears. The system populates the **New Pipe Run** dialog box automatically by using P&ID and design basis. Click **OK** to accept the default values and close the dialog box.

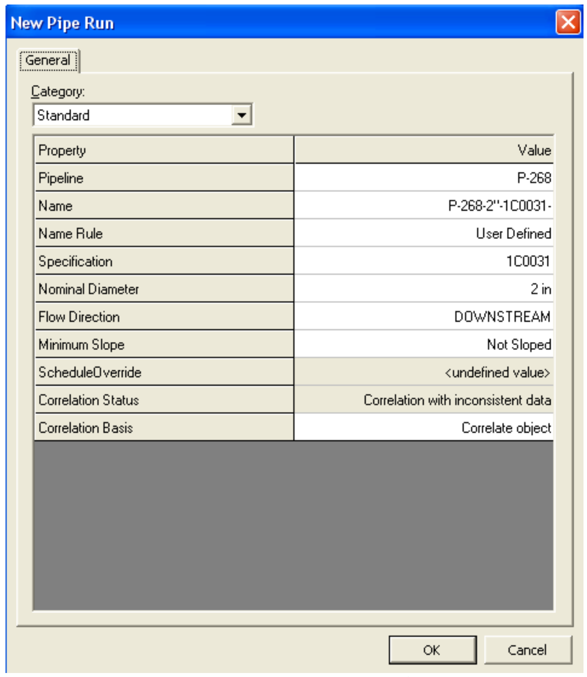


Figure 18: New Pipe Run Dialog Box

9. You now see an outline of the pipe run in the graphic view. Click to place the pipe run, as shown in Figure 19.

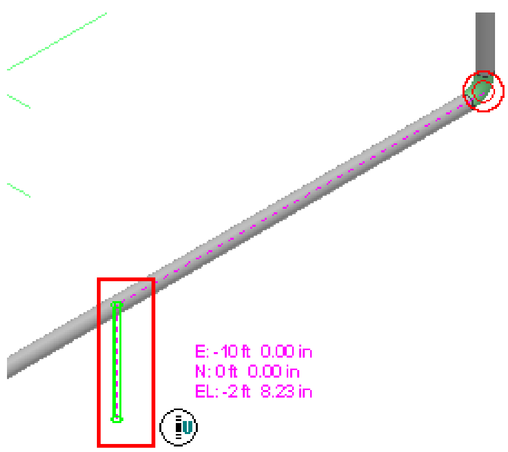


Figure 19: Placed Pipe Run

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 in Figure 20.

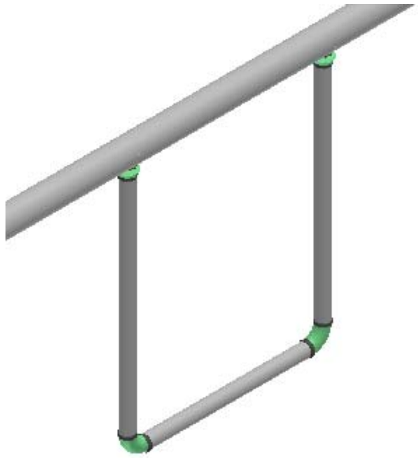


Figure 20: Placed Bypass line

10. Now, place a gate valve on the bypass line you have just placed. Click the **Insert Component** button on the vertical toolbar and use SmartSketch to locate the midpoint glyph of the pipe you have just placed, as shown in Figure 21. Use the midpoint as the active placement point for the component.

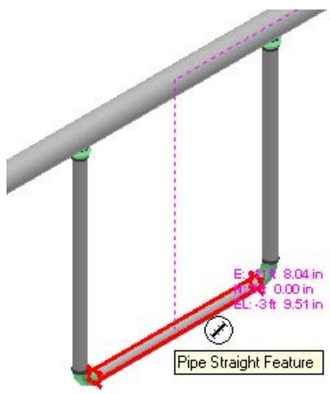


Figure 21: Midpoint of the pipe

11. Click on the active placement point of the pipe. The **P&ID File Viewer** window appears. In the **P&ID File Viewer** window, click the gate valve located in the bypass line, as shown in Figure 22.

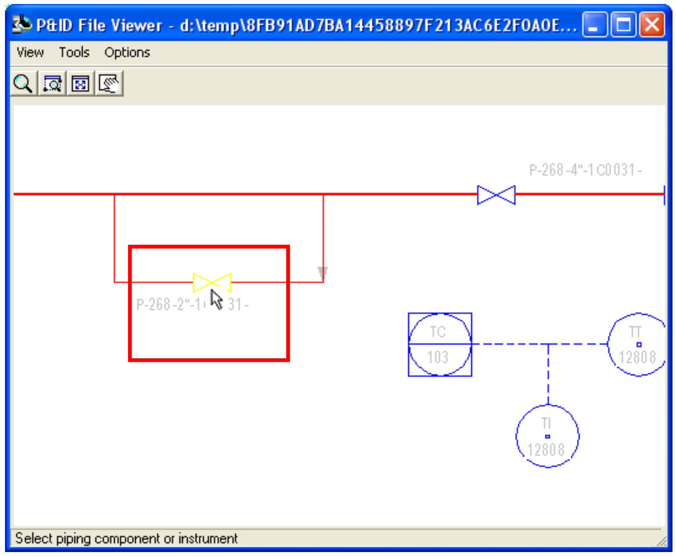


Figure 22: Gate valve on the Bypass Line in the P&ID File Viewer Window

12. The software uses P&ID and design basis to select the appropriate pipe component. Click **Finish** on the **Insert Component** ribbon to place the gate valve, as shown in Figure 23.

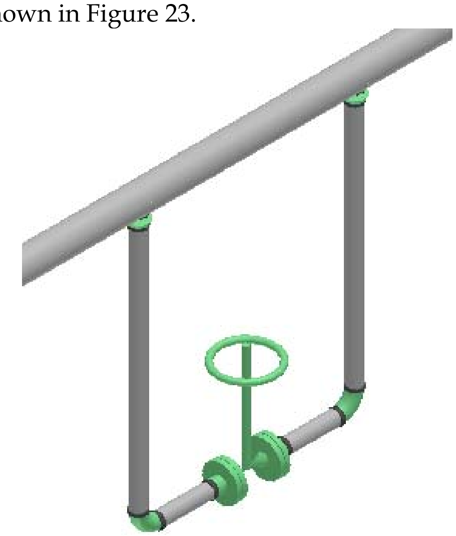


Figure 23: Placed Bypass Line

13. Open the **P&ID File Viewer** window. The bypass line you just placed is highlighted in green, as shown in Figure 24. This indicates that the bypass line is correlated and its properties and topology matches the design basis.

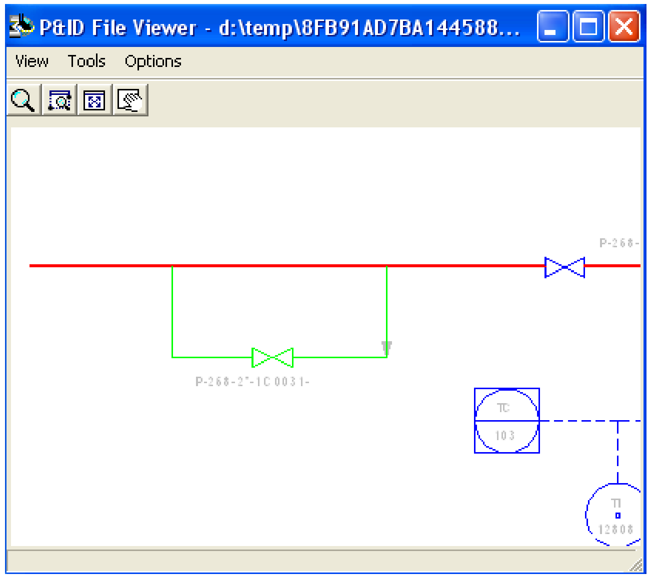


Figure 24: Correlated Bypass Line in the P&ID File Viewer Window

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

14. Key in **-3 ft** for **E** on the **PinPoint** ribbon to define the distance of **3 ft** from the target.

15. Click the **Insert Component** button on the vertical toolbar.

16. Now, position the cursor along the **Pipe Straight Feature,** as shown in Figure 25, and click to define the active placement point.

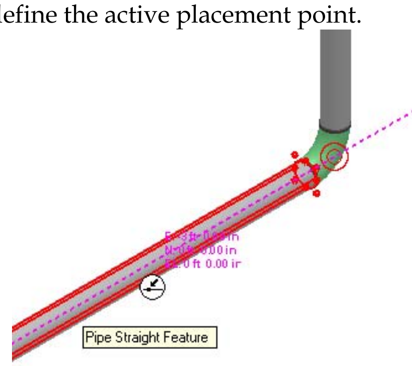


Figure 25: Pipe Straight Feature

17. The **P&ID File Viewer** window appears. In the **P&ID File Viewer** window, click the check valve, as shown in Figure 26.

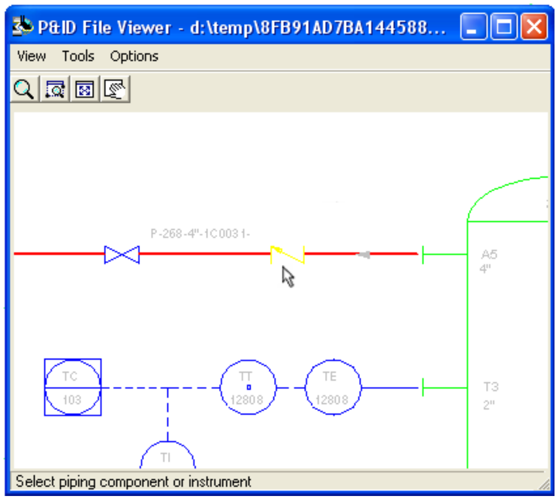


Figure 26: Check Valve in the P&ID File Viewer Window

18. 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. Click **Finish** on the **Insert Component** ribbon to place the pipe component, as shown in Figure 27.

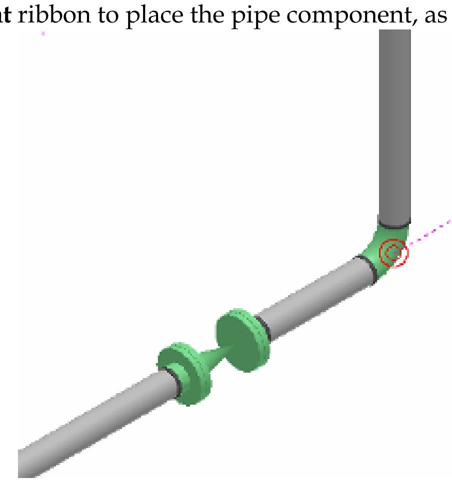


Figure 27: Placed Check Valve

19. 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, as shown in Figure 28.

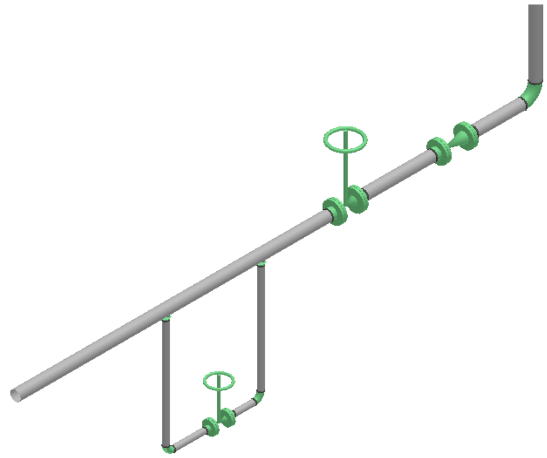


Figure 28: Placed Component

20. Now, continue routing the pipeline using the off-page connector (OPC). Open the continuation P&ID **INT02**. Click **SmartPlant > View P&ID…** command.

21. The **View P&ID** dialog box appears. Select the drawing **INT02** and click **Open**.

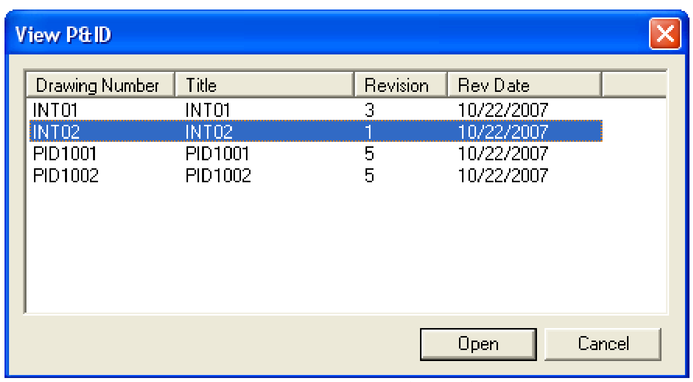


Figure 29: Open P&ID File Dialog Box

The **P&ID File Viewer** window appears.

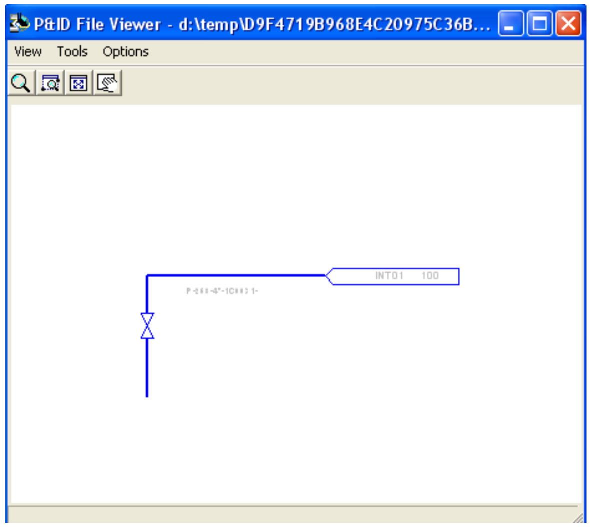


Figure 30: P&ID File Viewer Window

22. Now, click the **Route Pipe** button on the vertical toolbar.

23. Select the **Select From P&ID** option in the **Run** drop-down list on the **Route Pipe** ribbon.

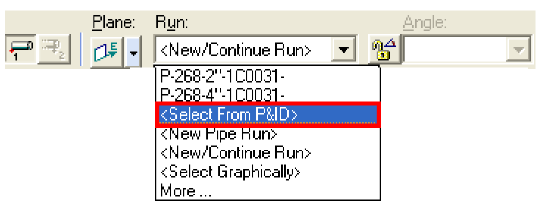


Figure 31: Route Pipe Ribbon

24. The **P&ID File Viewer** window appears. In the **P&ID File Viewer**, click the **P-268** pipeline, as shown in Figure 32.

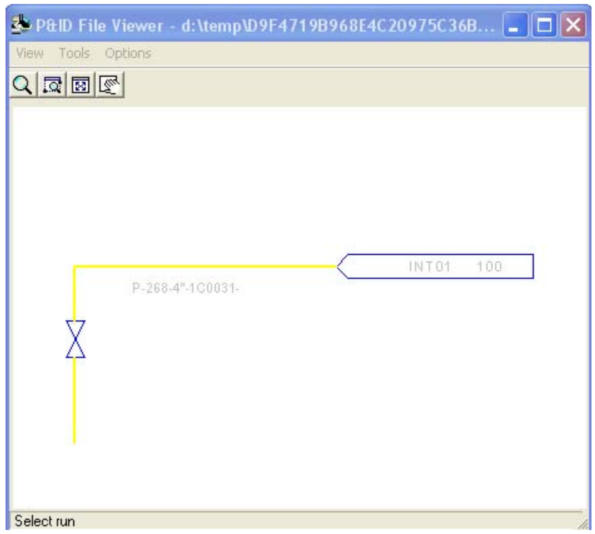


Figure 32: P-268 Pipeline in the P&ID INT02 Drawing

25. The **New Pipe Run** dialog box appears. The system populates the **New Pipe Run** dialog box automatically by using P&ID and design basis. Click **OK** to accept the default values and close the dialog box.

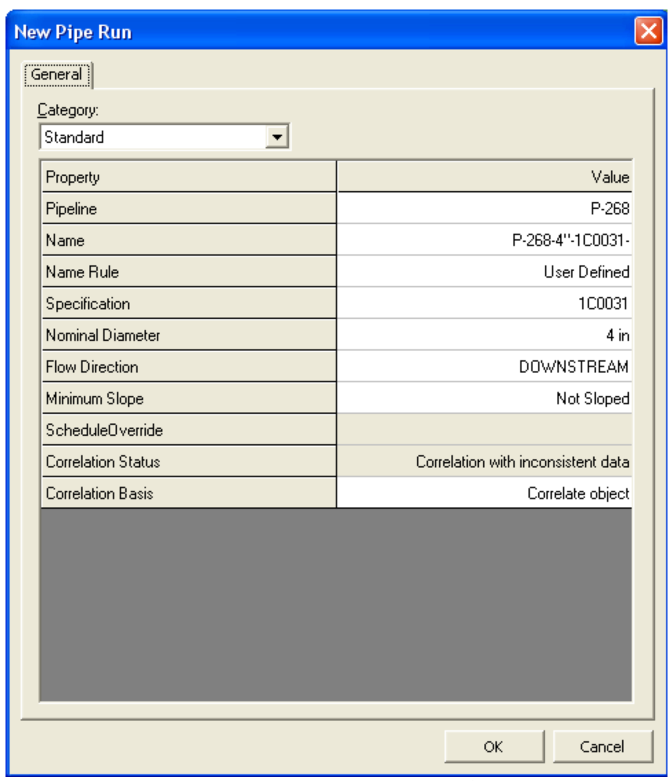


Figure 33: New Pipe Run Dialog Box

26. Select the end feature of the pipe to start the routing. You now see an outline of the pipe run in your model now. Click in the graphic view to place the pipe run, as shown in Figure 34.

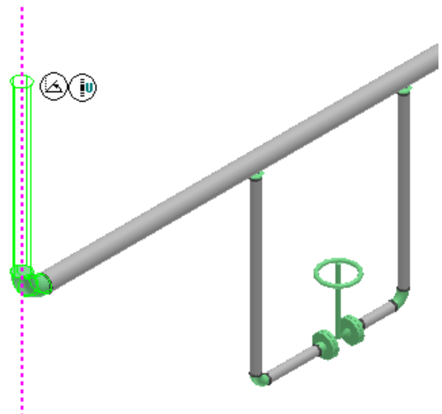


Figure 34: Placed Pipe Run

27. Click the **Insert Component** button on the vertical toolbar.

28. Now, position the cursor along the **Pipe Straight Feature,** as shown in Figure 35 and click to define the active placement point.

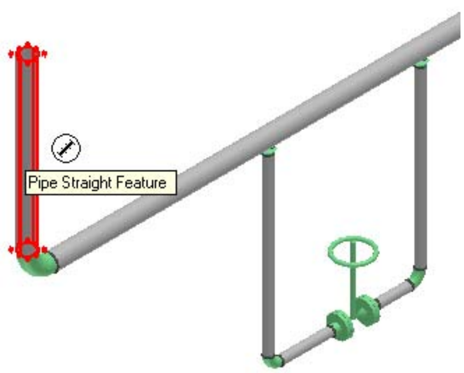


Figure 35: Pipe Straight Feature

29. The **P&ID File Viewer** window appears. In the **P&ID File Viewer**, click the gate valve, as shown in Figure 36.

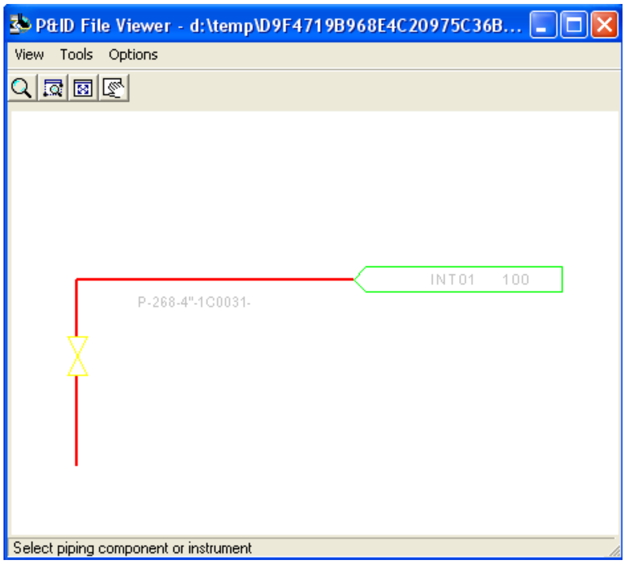


Figure 36: P&ID File Viewer Window

30. 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. Key in **90 deg** in the **Angle** drop-down list to rotate the valve. Click **Finish** on the **Insert Component** ribbon to place the pipe component, as shown in Figure 37.

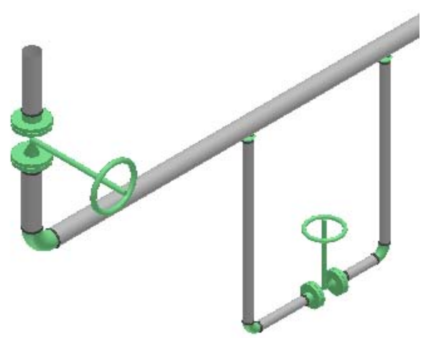


Figure 37: Placed Component

31. Place a **2 ft** pipe to complete the route. Right-click to terminate the **Route Pipe** command.

32. Now, open the **P&ID File Viewer** window. The pipe section you placed is highlighted in green, as shown in Figure 38, which indicates that the pipe section and the off-page connector now are correlated and its properties and topology match the design basis.

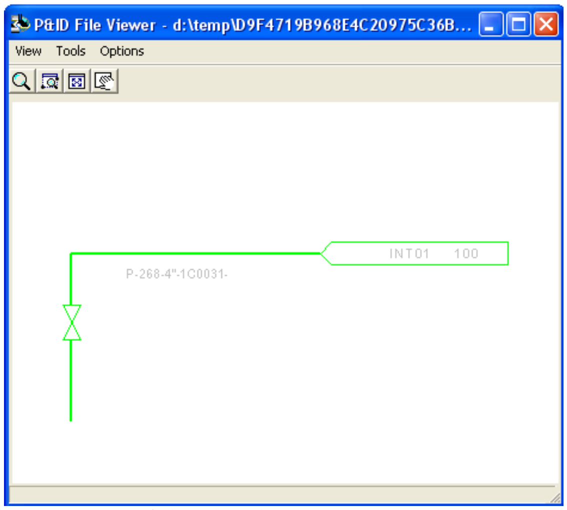


Figure 38: Correlation Status of the Placed Pipe Section in the P&ID File Viewer Window

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.

33. Now, open the **P&ID INT01** drawing. The pipe section you placed is highlighted in green, as shown in Figure39, which indicates that the pipe section and the OPC now are correlated and its properties and topology match the design basis.

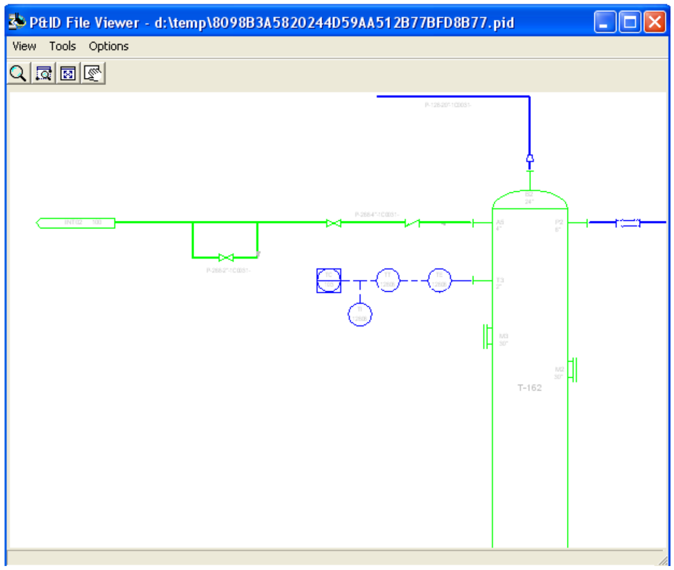


Figure39: Correlation Status of the Placed Pipe Section in the P&ID File Viewer Window

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:*

Session 6: Placing Instruments

Objective:

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

- Place an instrument in a pipeline.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run
- Routing Pipes from P&ID

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, as shown in Figure 1.

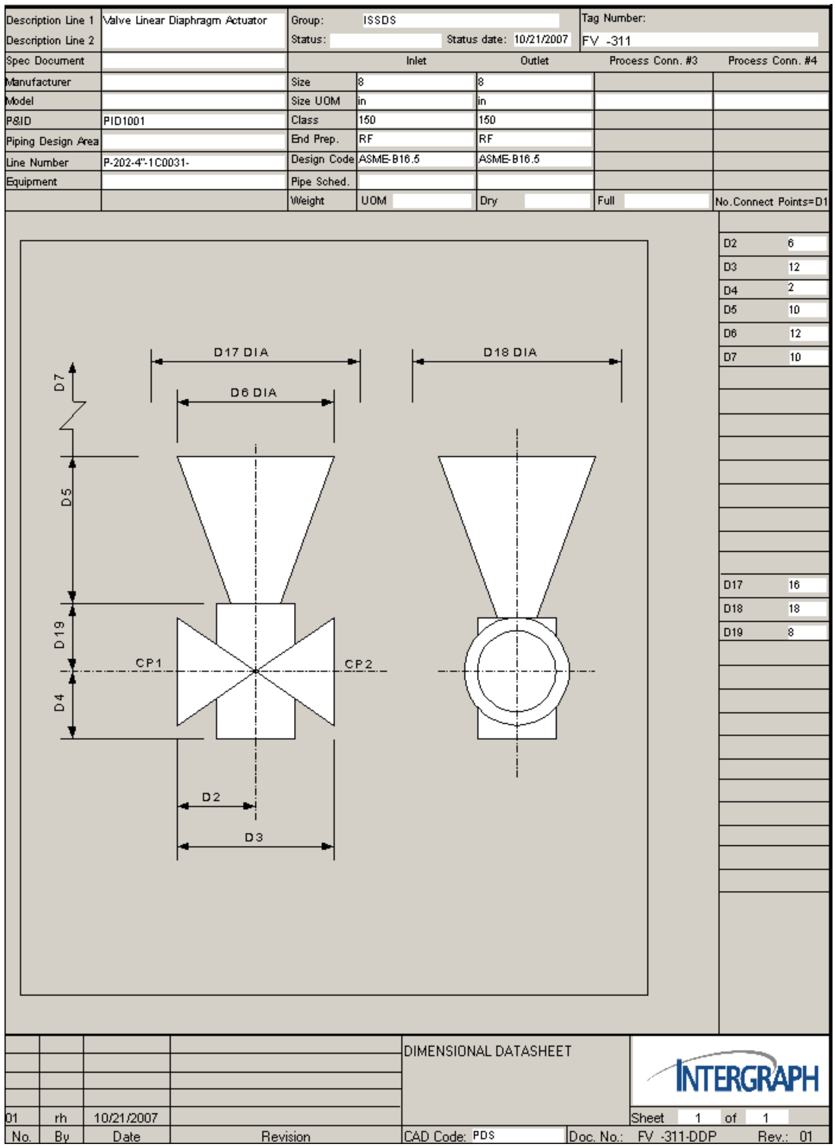


Figure 1: FV-311 Instrumentation Dimensional Datasheet (DDP)

You use the **Insert Component** button 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, as shown in Figure 2.

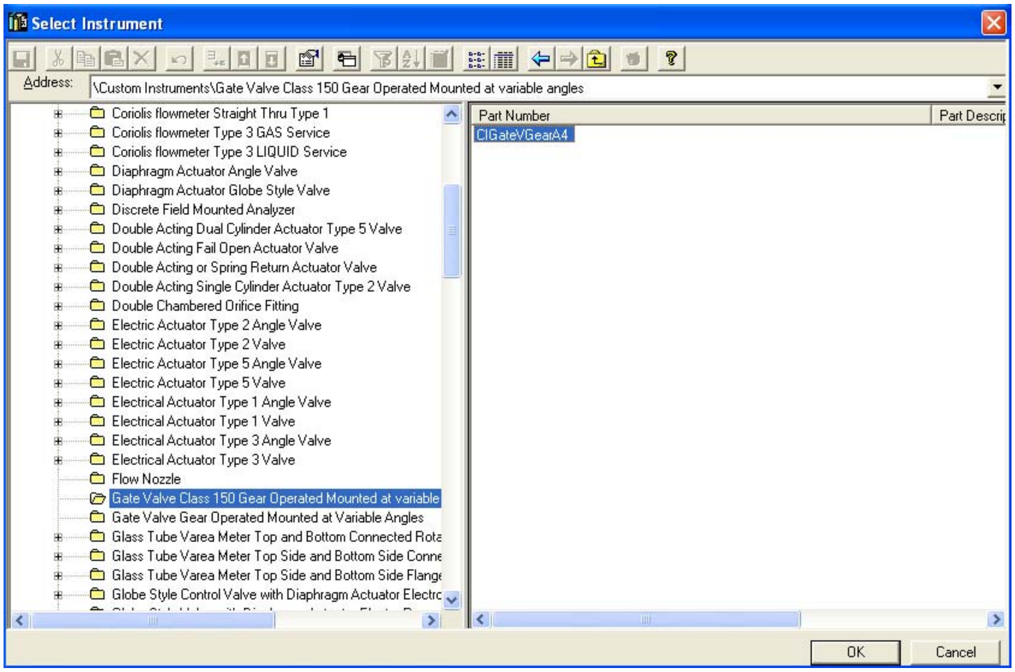


Figure 2: Select Instrument Dialog Box

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.

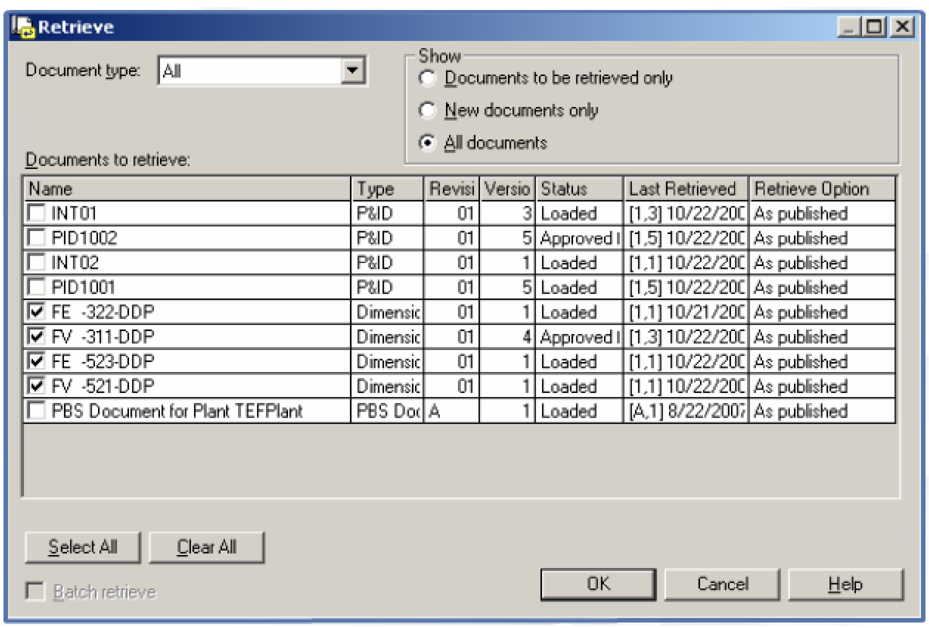


Figure 3: Retrieve Dialog Box

Placing Instruments in a Pipeline:

Exercise Objective: In this exercise you will be inserting instruments **CIGateVGearA4, VL2** and a custom instrument **FV-311** 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 Figure 4.

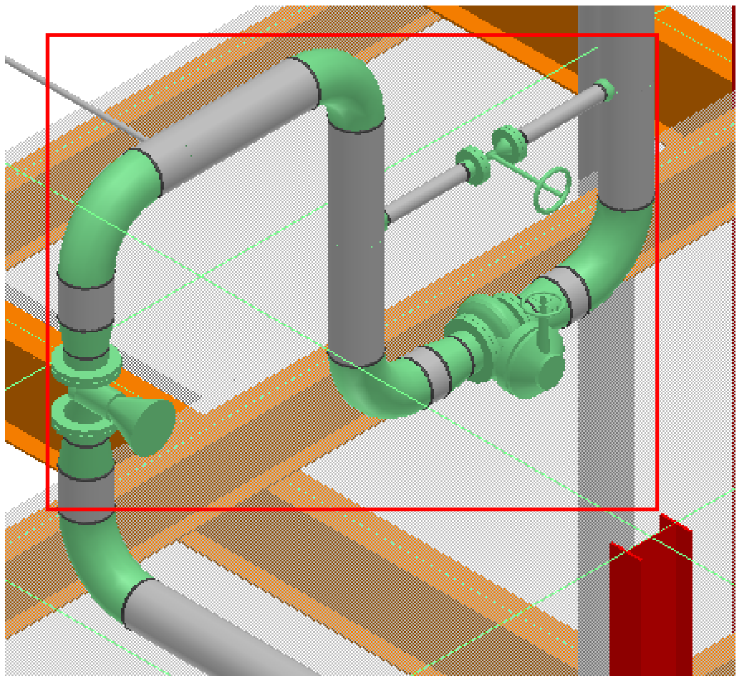


Figure 4: Placed Instruments

Steps for Placing Instruments from the Catalog:

Insert the instruments **CIGateVGearA4** and **VL2** from the Catalog along the pipeline **300-W** in Unit **U03** of your workspace.

Before beginning the 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**

- Click the **Zoom Area** button on the **Common** toolbar and zoom in to the area, as shown in Figure 5.

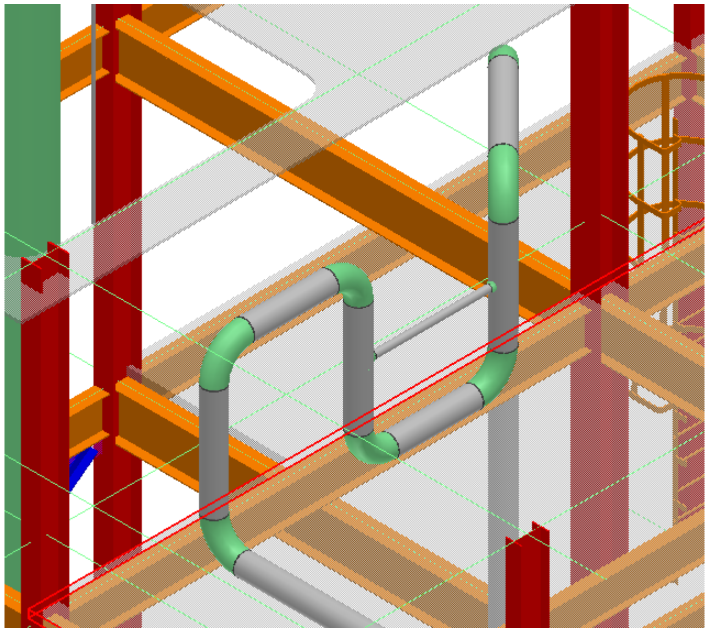


Figure 5: Zoomed Area of the Model

- Now, click the **Insert Component** button on the vertical toolbar to start placing the instrument.

Figure 6: Insert Component Button on the Vertical Toolbar

- By using SmartSketch locate the midpoint of the pipe, as shown in Figure 7, and click on it.

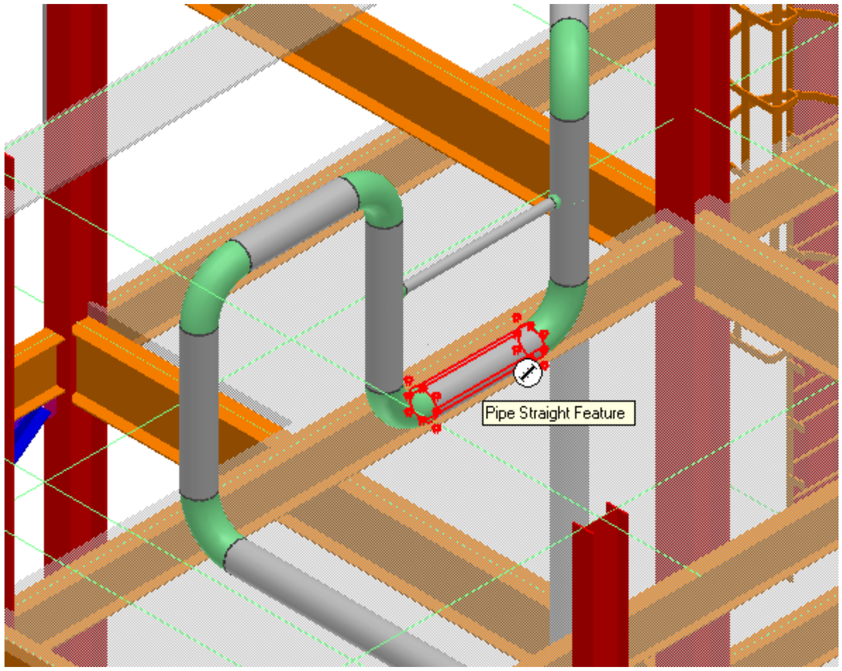


Figure 7: Located Midpoint of the Pipe

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

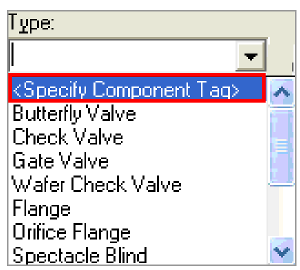


Figure 8: Type Drop-Down Field

- The **Specify Component Tag** dialog box appears, as shown in Figure 9. Select the **Browse instruments…** option in the **Component tag** drop-down list and click **OK**.

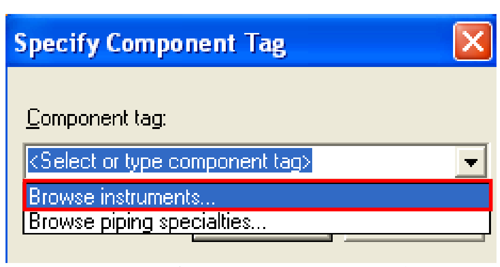


Figure 9: Specify Component Tag Dialog Box

- 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. Navigate to **Gate Valve Class 150 Gear Operated Mounted at variable angles** and click the part number **CIGateVGearA4** in the right pane. Then, click **OK** to close the **Select Instrument** dialog box.

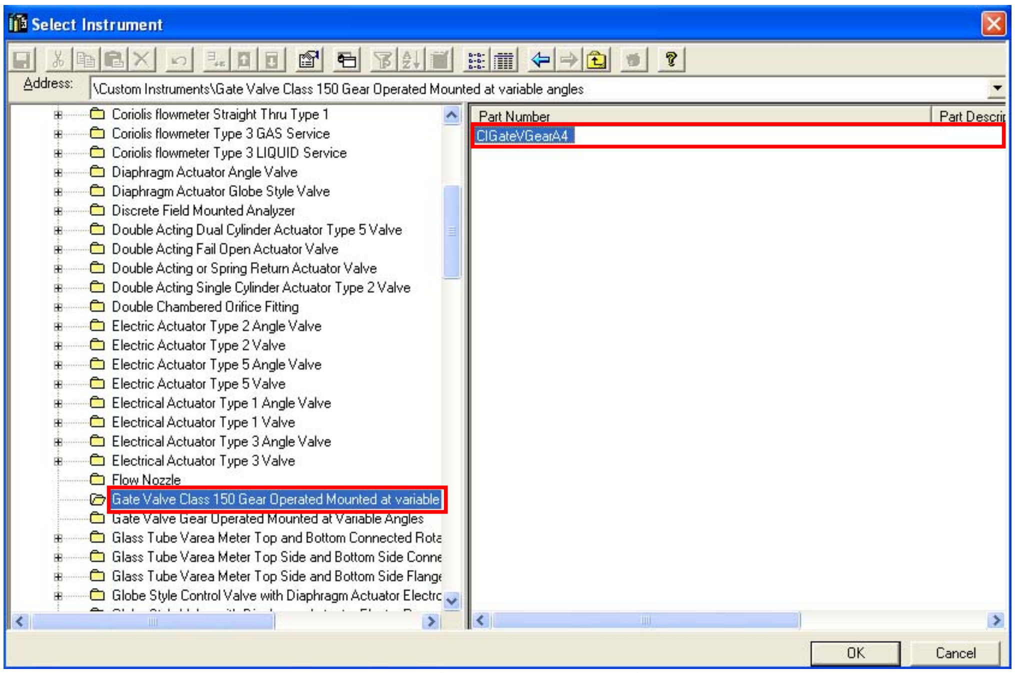


Figure 10: Select Instrument Dialog Box

Tip:

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

You will now see an outline of the valve in your model, as shown in Figure 11.

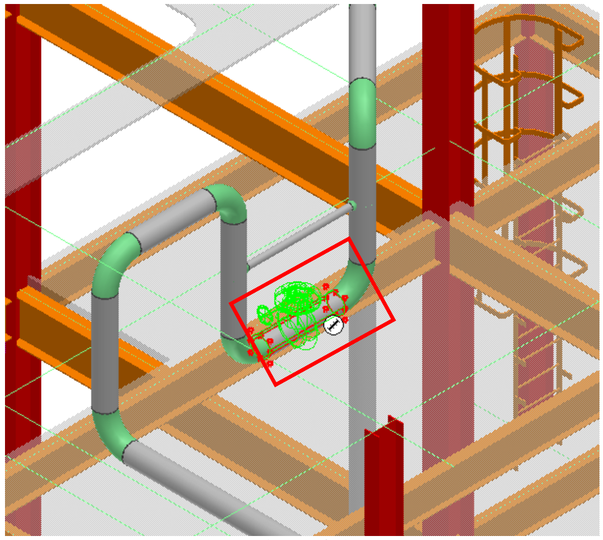


Figure 11: Outline of the Instrument

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

- Click the **Common Views** button 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.
- Click the **Properties** button on the **Insert Component** ribbon.
- The **Pipe Component Feature Properties** dialog box appears. This dialog box has specifications automatically populated by the system, as shown in Figure 12.

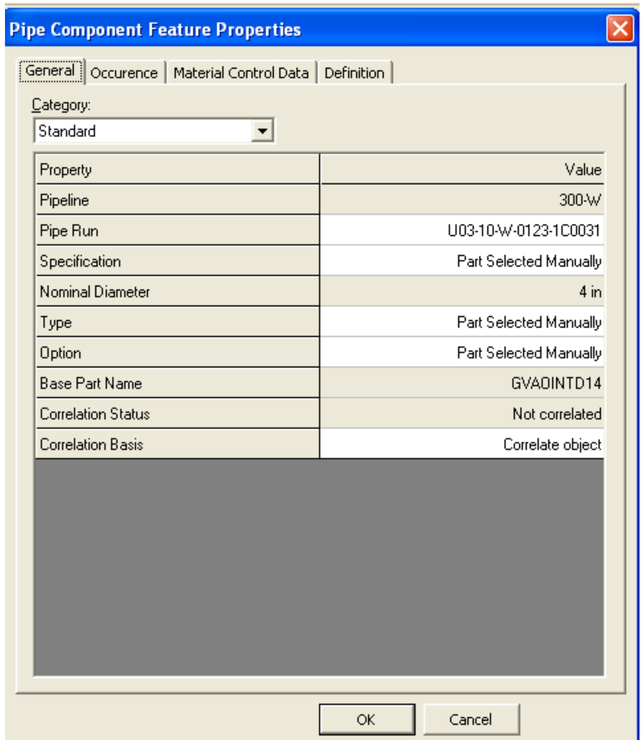


Figure 12: Pipe Component Feature Properties Dialog Box

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

- 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: US Practice
Schedule Thickness 1: Standard Weight
End Practice 1: US Practice
End Standard 1: Default
Rating Practice 1: US Practice
Pressure Rating 1: CL150
Flow Direction 1: Flow may enter or leave this port
Piping Point Basis 1: <undefined value>

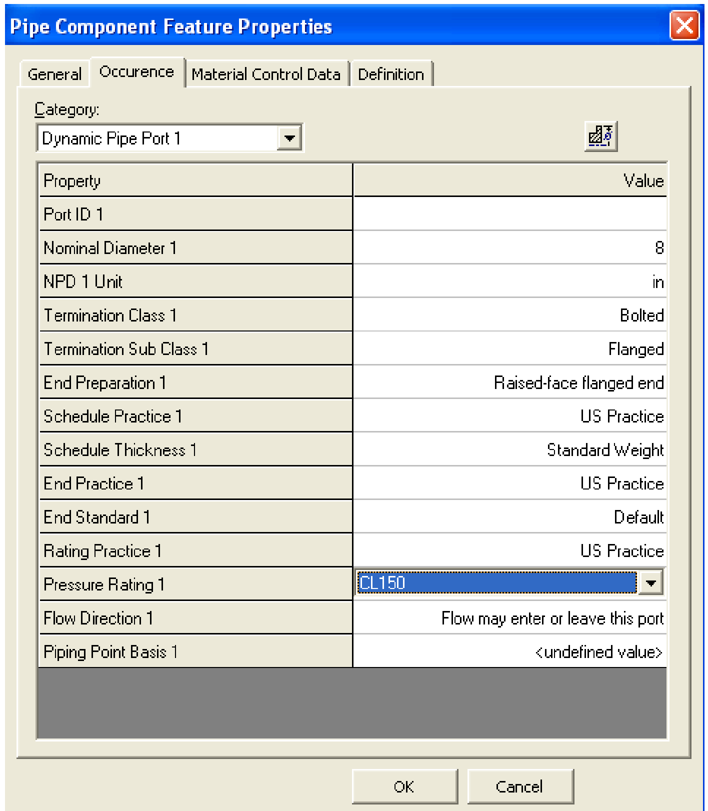


Figure 13: Dynamic Pipe Port 1 Properties in the Pipe Component Feature Properties Dialog Box

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

- 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 Erector
  Fabrication Type: Contractor field fabricated
  Construction Requirement: New
  Construction Type: New

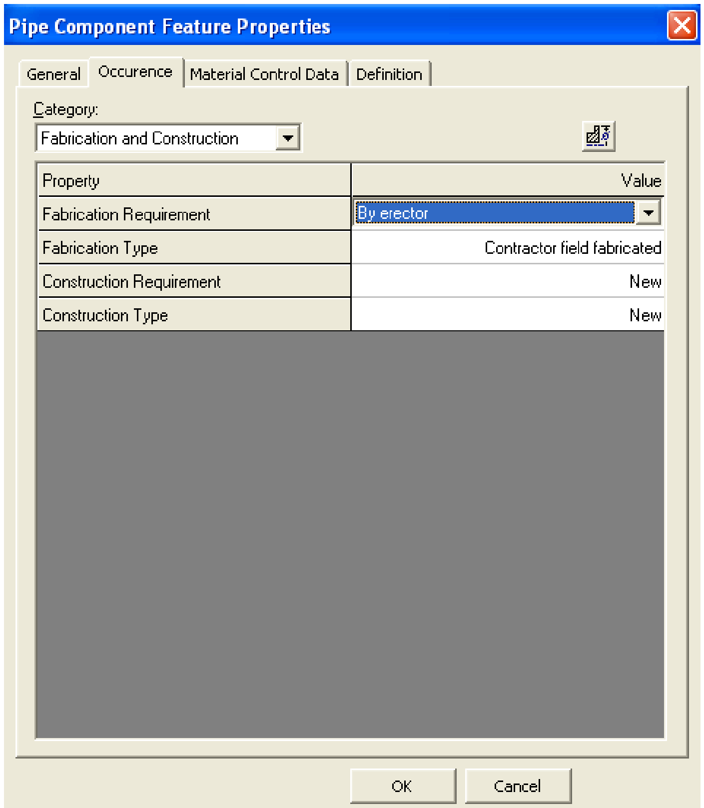


Figure 14: Fabrication and Construction Properties in the Pipe Component Feature Properties Dialog Box

- In the **Category** drop-down list, select the **Standard** option.
- 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, as shown in Figure 15.

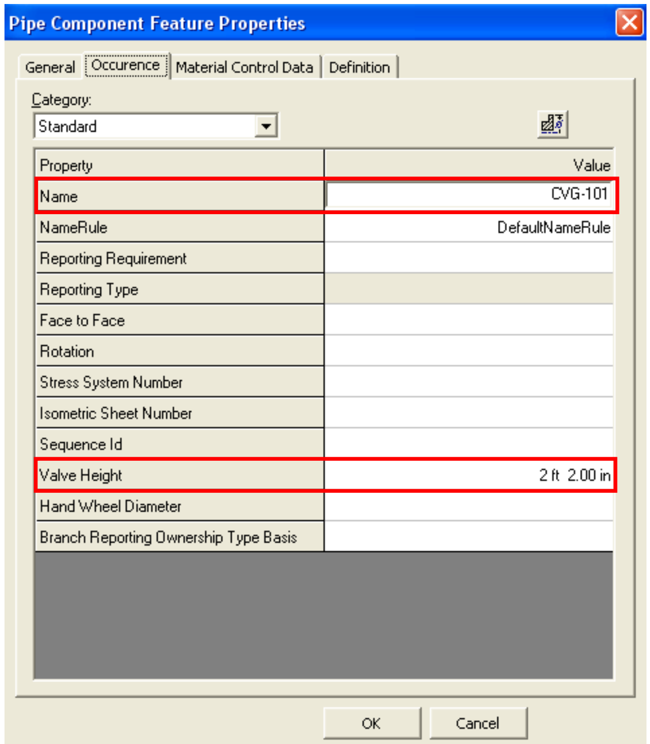


Figure 15: Standard Properties in the Pipe Component Feature Properties Dialog Box

- 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 erector
  Fabrication Type: Contractor field fabricated
  Bolting Requirements: Reportable bolts required
  Gasket Requirements: Gasket required at each bolted end

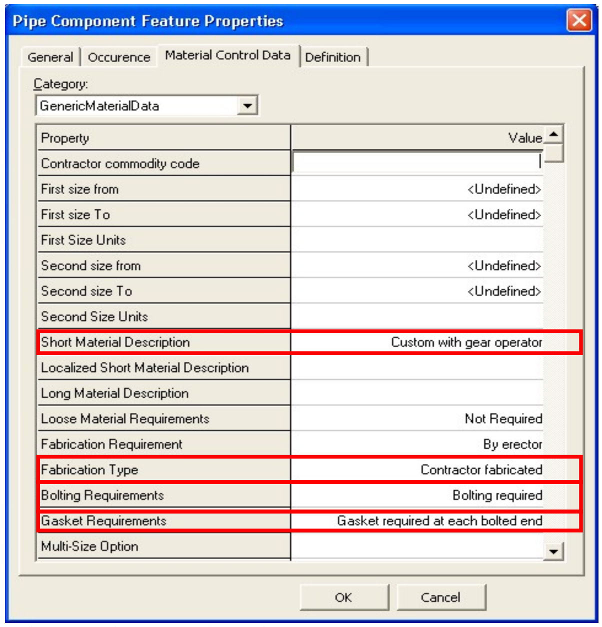


Figure 16: Material Control Data Tab in the Pipe Component Feature Properties Dialog Box

- 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, as shown in Figure 17.

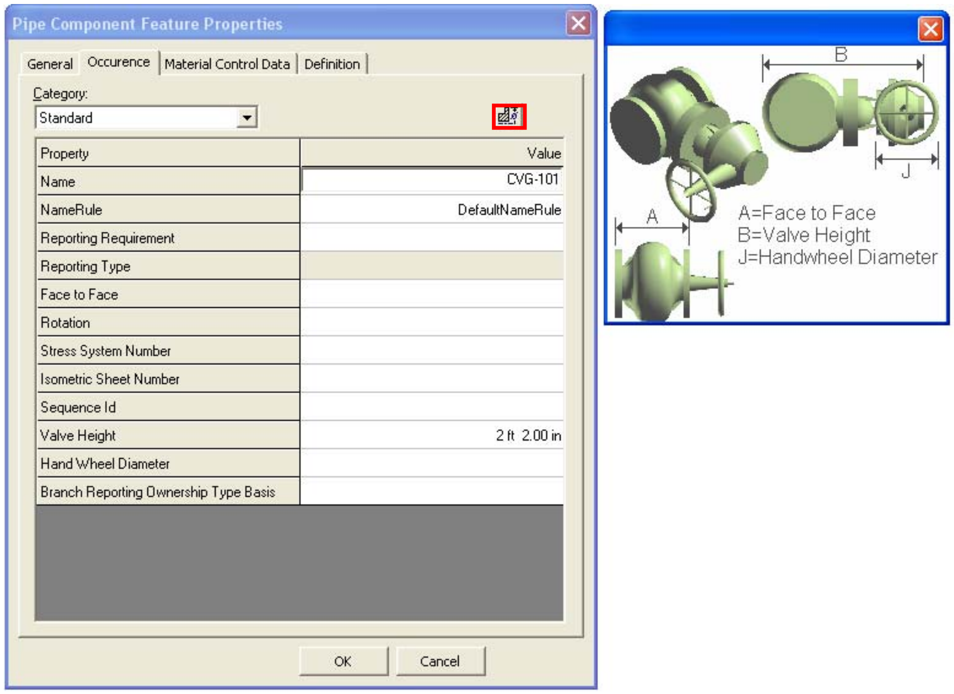


Figure 17: Show Dimensional Legend Button on the Pipe Component Feature Properties Dialog Box

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.

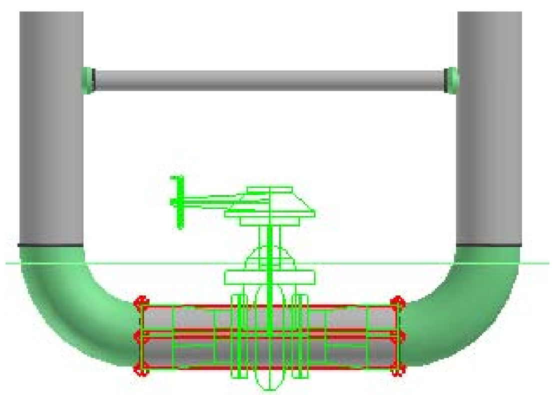


Figure 18: Mating Components on the Valve

- 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.

Figure 19: Angle and Operator Angle Drop-Down Fields on the Insert Component Ribbon

- Click the **Finish** button on the **Insert Component** ribbon to place the valve, as shown in Figure 20.

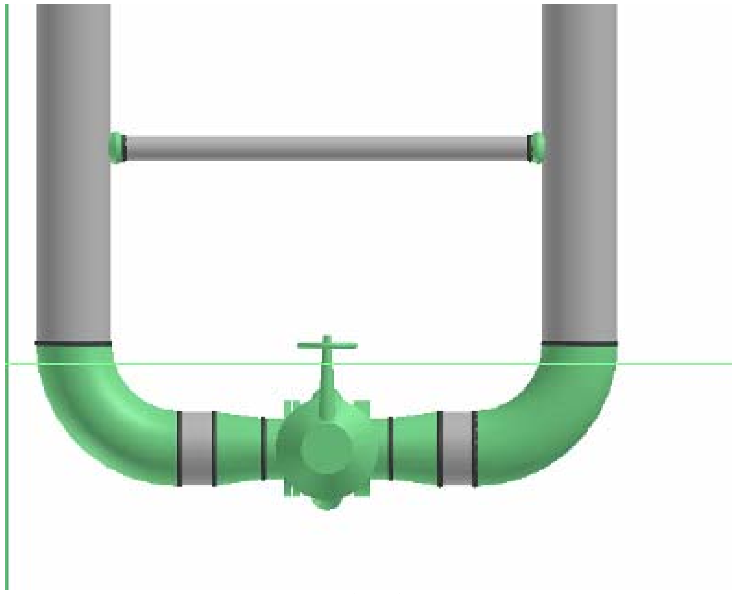


Figure 20: Placed Instrument

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

- Next, place a stock instrument on the pipe highlighted in Figure 21. Click the **Insert Component** button on the vertical toolbar and use SmartSketch to locate the midpoint of the pipe, as shown in Figure 21.

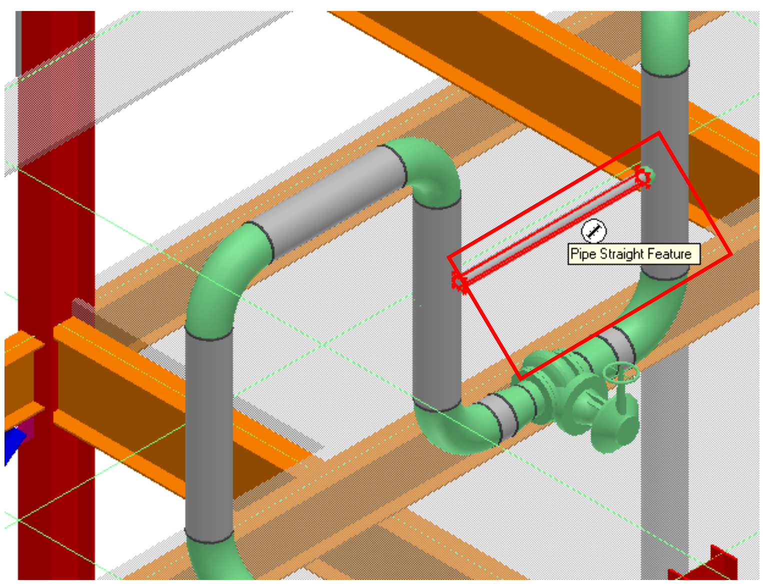


Figure 21: Located Midpoint of the Pipe

- 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. **VL2** is the tag number for a global valve. Key in **VL2** in the **Component tag** drop-down field and click **OK** to close the **Specify Component Tag** dialog box.

- An outline of the instrument will appear in the graphic view. On the **Insert component** ribbon, key in **270 deg** in the **Angle** drop-down list to rotate the instrument towards the south direction.

- Click **Finish** on the **Insert component** ribbon to place the instrument, as shown in Figure 22.

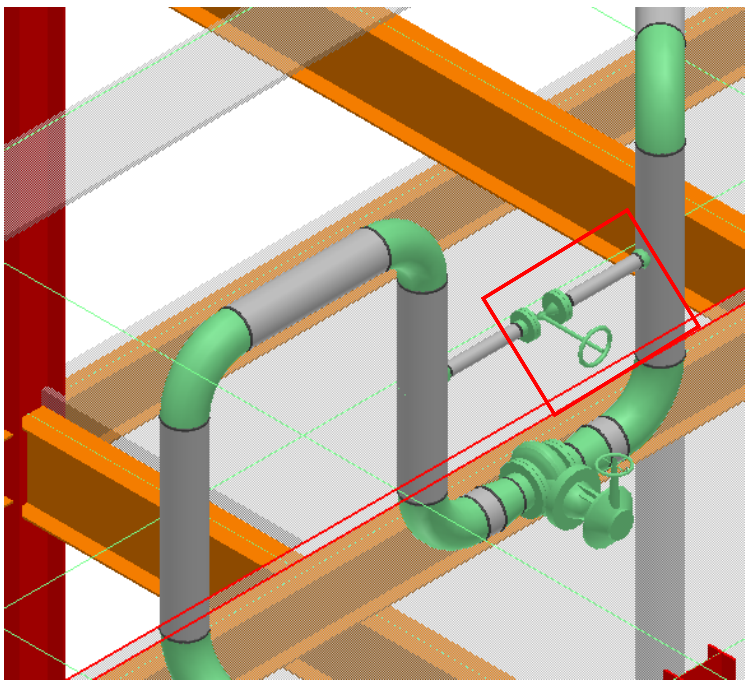


Figure 22: Placed Instrument

Steps for Placing Instrument by using DDP:

Now, you need to place another custom instrument **FV-311** by using the retrieved DDP on the pipeline **300-W** highlighted in Figure 23.

- Click the **Insert Component** button on the vertical toolbar and use SmartSketch to locate the midpoint of the pipe as shown in Figure 23.

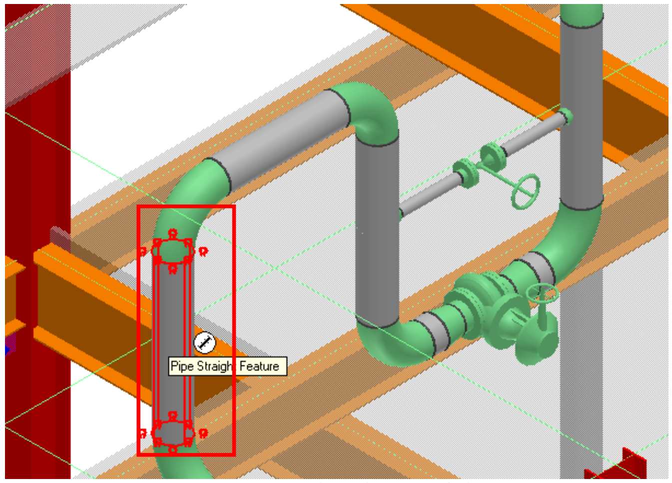


Figure 23: Located Midpoint of the Pipe

- Follow steps 23 to 26 to place the instrument. Type **FV-311** in the **Component tag** drop-down field of the **Specify Component Tag** dialog box.

- The placed **FV-311** instrument should resemble the highlighted part in Figure 24. Key in **-90 deg** in the **Angle** drop-down list to rotate the instrument.

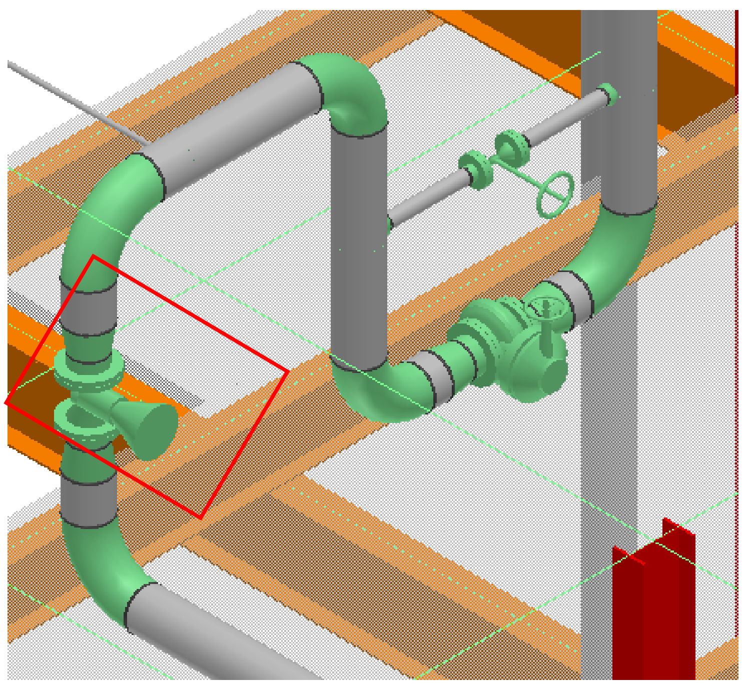


Figure 24: Placed Instrument

Note:

- SP3D 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.

Steps for Placing Instruments from a P&ID:

Exercise 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 Figure 25.

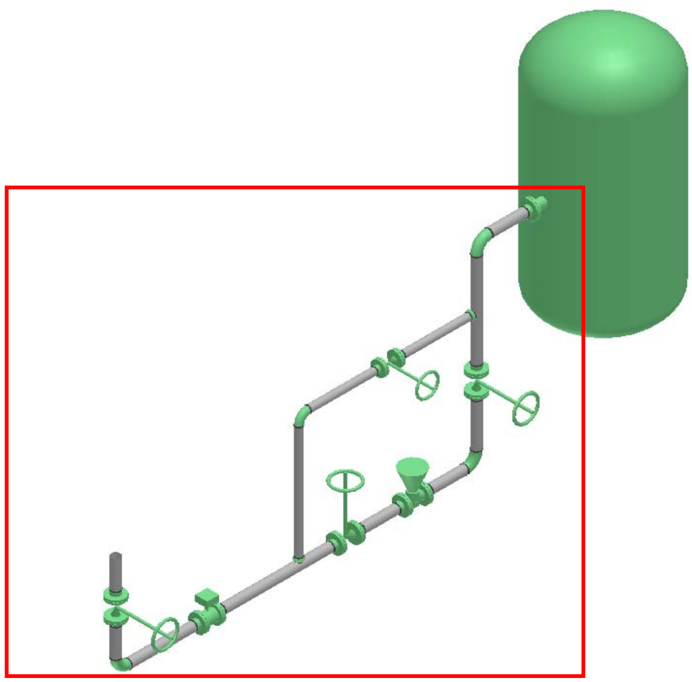


Figure 25: Placed Instruments From P&ID

Before beginning the 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. Open the P&ID that contains the design basis of the pipeline you want to model. Select the **SmartPlant > View P&ID** command.

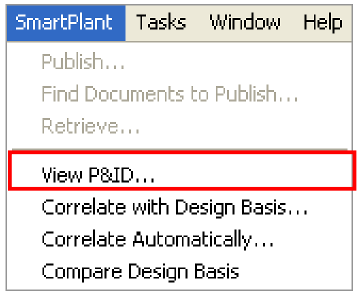


Figure 26: SmartPlant > View P&ID Command

2. The **View P&ID** dialog box appears. The **View P&ID** dialog box displays a list of the P&IDs available for the model. Select **PID1001** drawing and click **Open**.

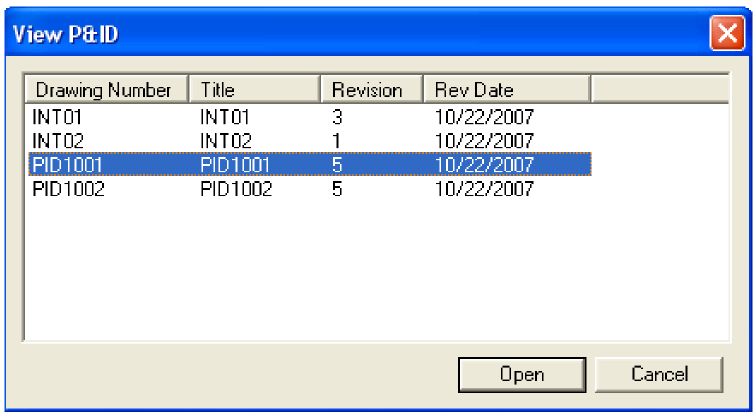


Figure 27: Open P&ID File Dialog Box

The **P&ID File Viewer** window appears.

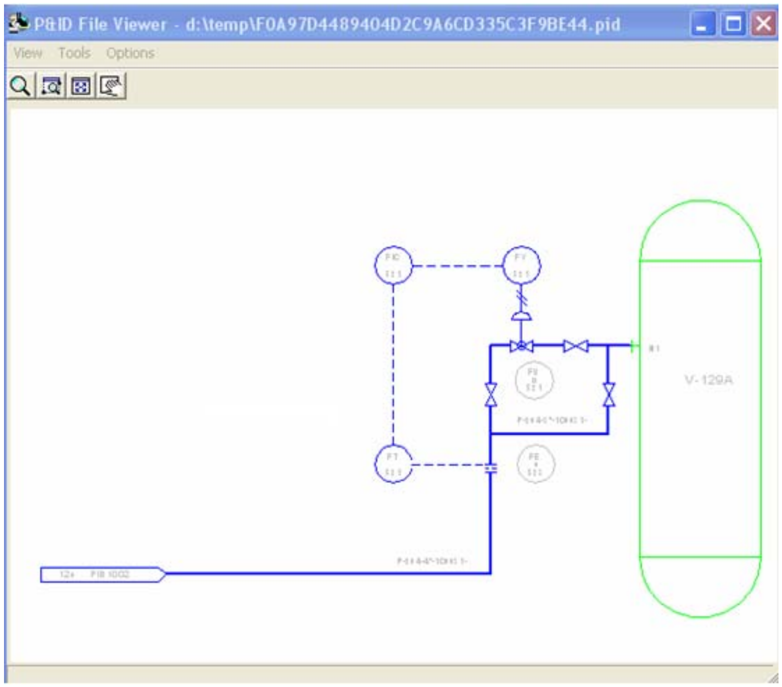


Figure 28: P&ID File Viewer Window

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. Select the equipment nozzle **N1** from the **P&ID File Viewer**.

4. Click the **Route Pipe** button on the vertical toolbar to start routing the pipe run on the equipment nozzle **N1**, as shown in Figure 29.

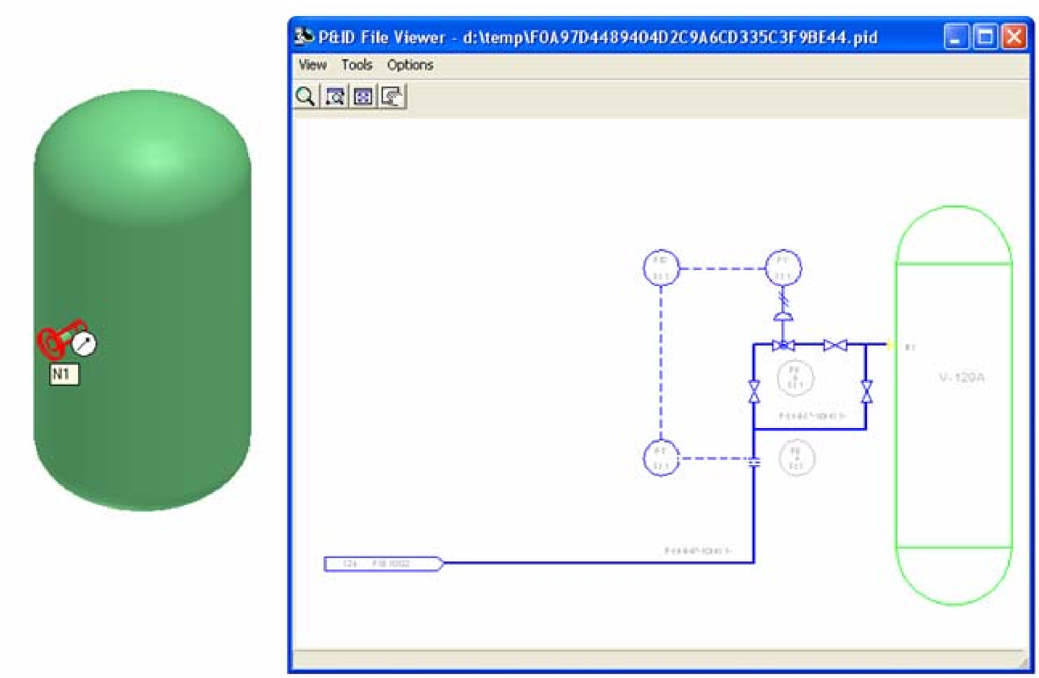


Figure 29: Selected Equipment Nozzle N1

5. 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. Ensure 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
   Minimum Slope: Not Sloped
   ScheduleOverride: <undefined value>
   Correlation Status: Correlation with inconsistent data
   Correlation Basis: Correlate object

6. Click **OK** to close the dialog box:

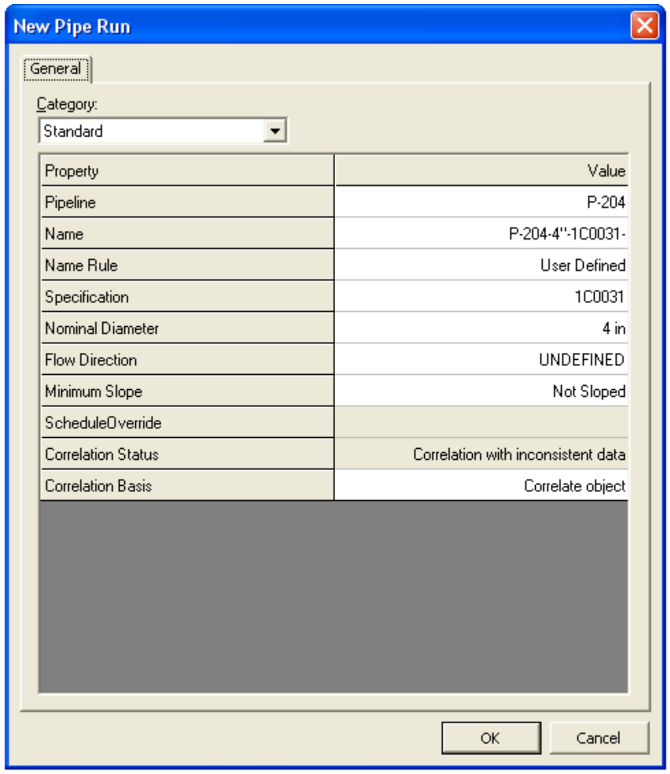


Figure 30: New Pipe Run Dialog Box

7. An outline of a pipe run appears in the graphic view starting at pipe nozzle **N1**. Key in **2 ft** in the **Length** drop-down list on the **Route Pipe** ribbon to lock the length of the pipe run.

8. Move the cursor towards the left until SmartSketch displays the **E** glyph, as shown in Figure 31. The **E** glyph indicates that you are routing the pipe in the west direction. Click in the graphic view to place the pipe.

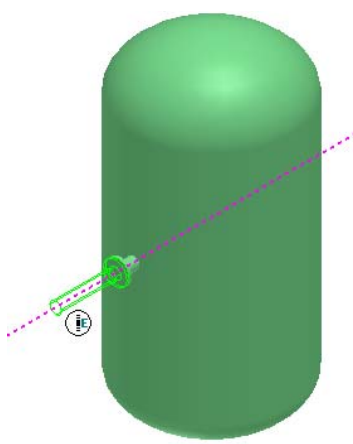


Figure 31: Placed Pipe

9. Now, key in **8 ft** in the **Length** drop-down list on the **Route Pipe** ribbon. Move the cursor downwards until SmartSketch displays the **U** glyph, as shown in Figure 32. Click in the graphic view to place the pipe.

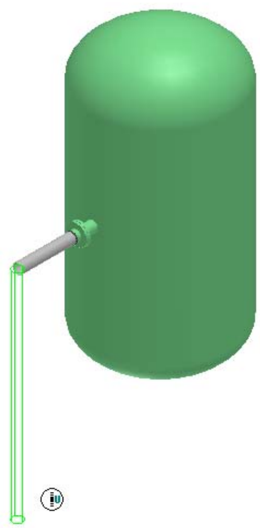


Figure 32: Placed Pipe

10. Key in **15 ft** in the **Length** drop-down list on the **Route Pipe** ribbon. Move the cursor towards the left until SmartSketch displays the E glyph as shown in Figure 33 and click in the graphic view to place the pipe.

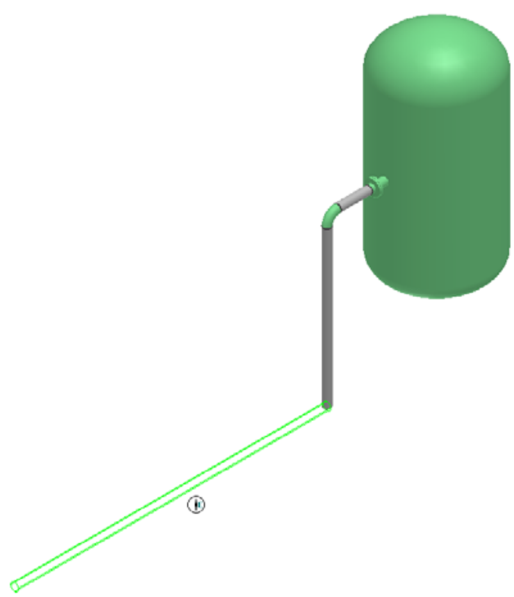


Figure 33: Placed Pipe

11. 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:

12. Click the **Route Pipe** button on the vertical toolbar. Position the cursor along the **Pipe Straight Feature,** as shown in Figure 34 and click to define the active placement point.

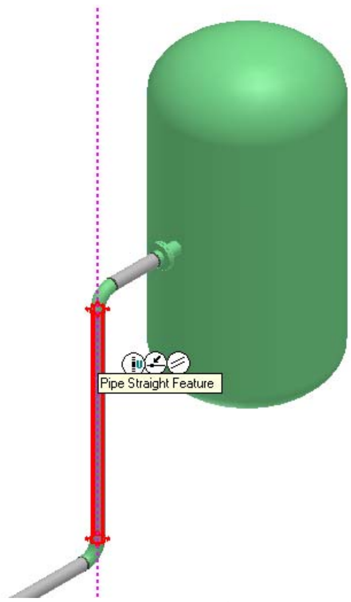


Figure 34: Pipe Straight Feature

13. The **P&ID File Viewer** window appears. In the **P&ID File Viewer**, select the bypass line, as shown in Figure 35.

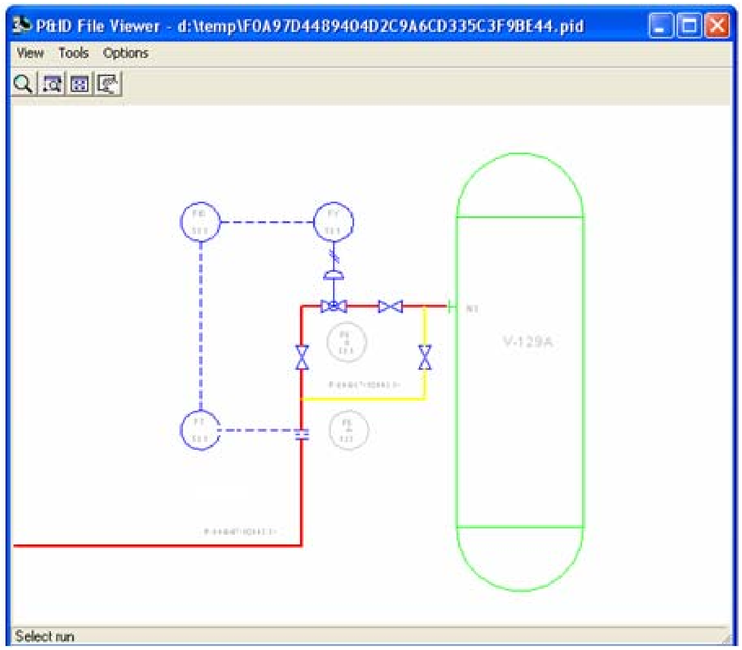


Figure 35: Selected bypass line

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

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

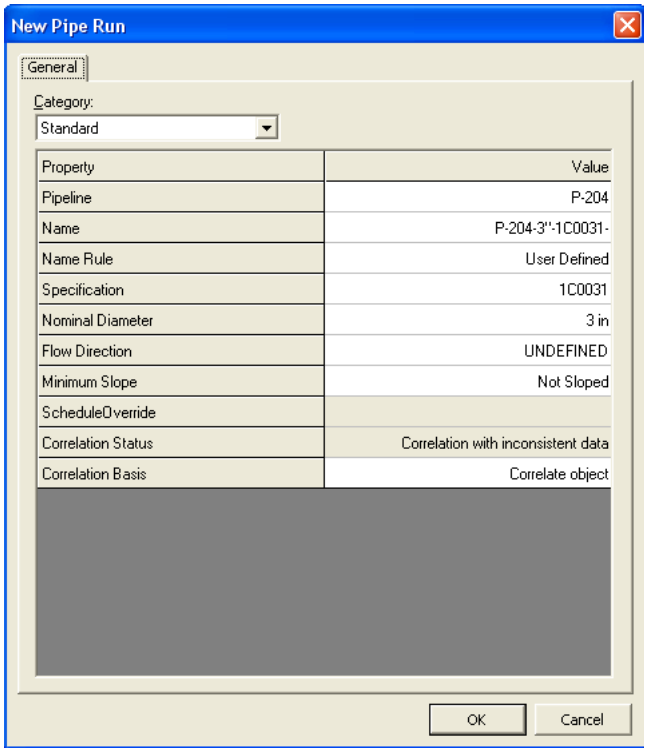


Figure 36: New Pipe Run Dialog Box

16. You now see an outline of the pipe run in your model. Route the pipe run to complete the bypass line, as shown in Figure 37.

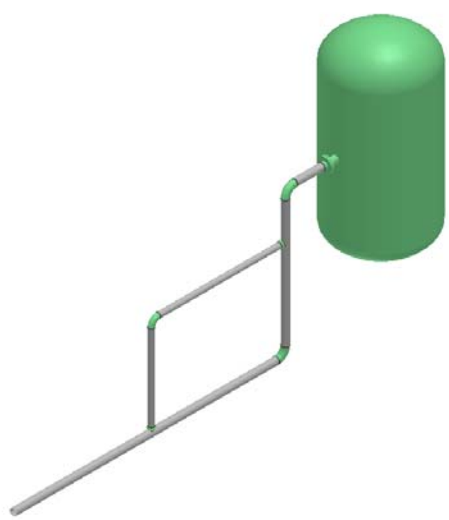


Figure 37: Bypass line

17. Now, place a gate valve on the bypass line you have just placed. Click the **Insert**

Component button on the vertical toolbar 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.

18. Click on the active placement point of the pipe. The **P&ID File Viewer** window appears.

19. In the **P&ID File Viewer** window, select the gate valve located in the bypass line, as shown in Figure 38.

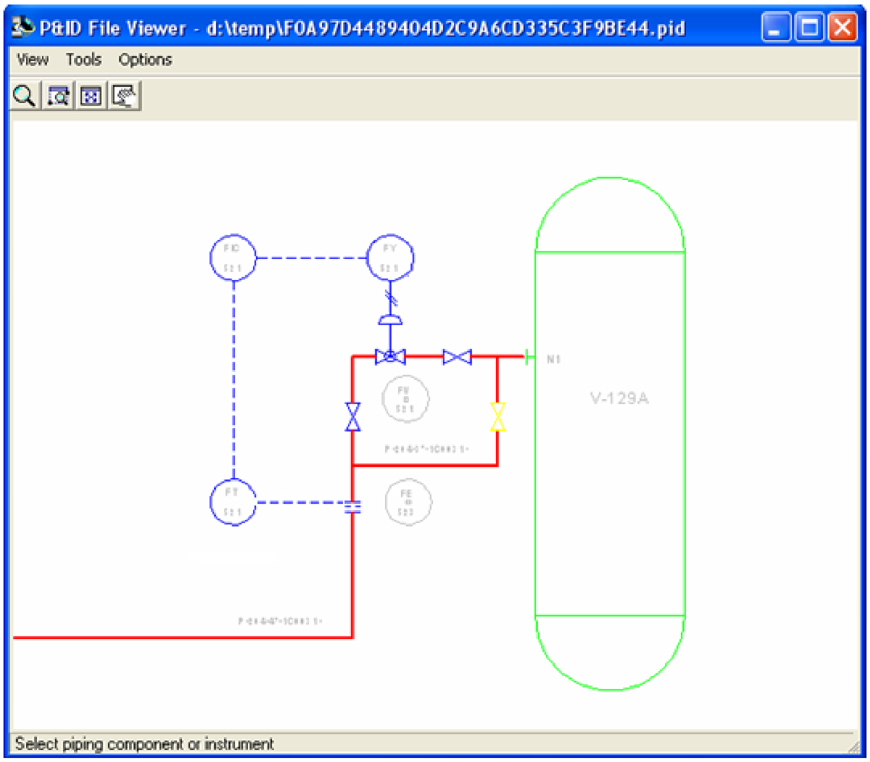


Figure 38: Selected Gate Valve

20. The software uses P&ID and design basis to select the appropriate pipe component. Key in **90 deg** in the **Angle** drop-down list to rotate the valve.

21. Click **Finish** on the **Insert Component** ribbon to place the gate valve, as shown in Figure 39.

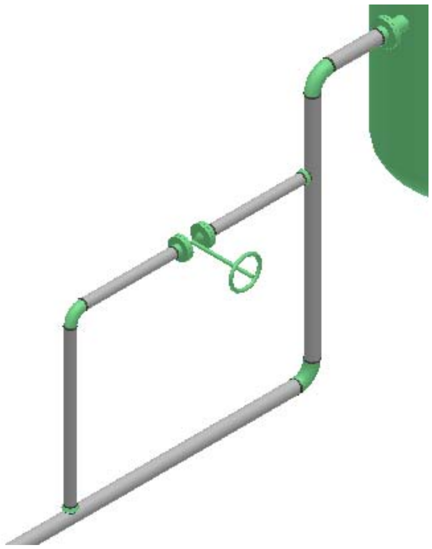


Figure 39: Placed Gate Valve

22. Repeat the above steps to place two gate valves, as shown in Figure 40.

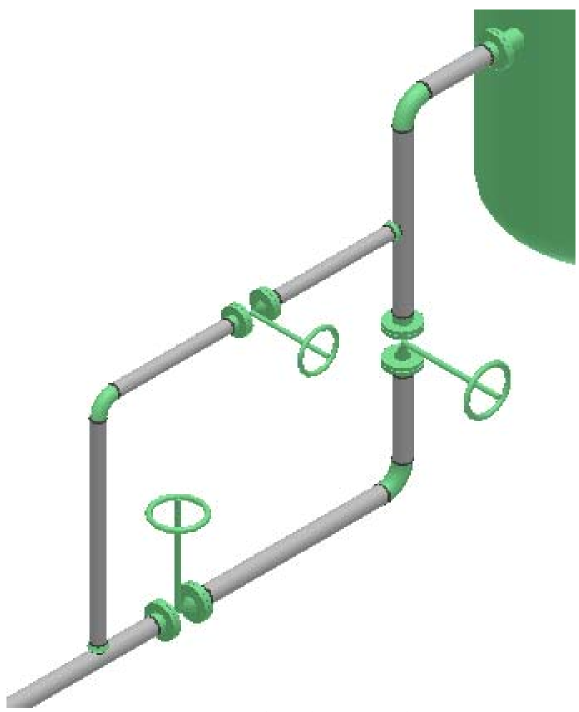


Figure 40: Placed Gate Valves

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

23. Click the **Insert Component** button on the vertical toolbar. Use the midpoint of the pipeline **P-204** as the active placement point for the instrument.

24. Click on the active placement point of the pipe, as shown in Figure 41.

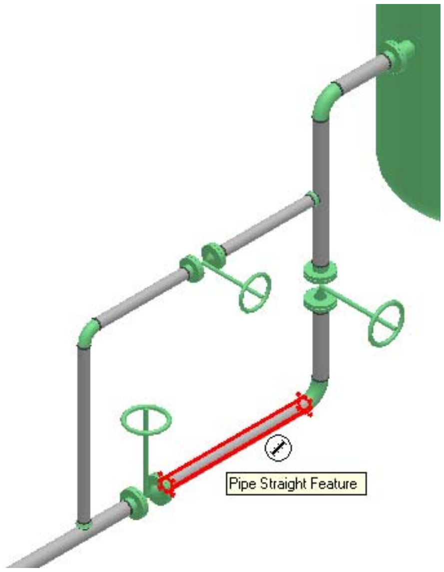


Figure 41: Pipe Straight Feature

25. The **P&ID File Viewer** window appears. In the **P&ID File Viewer** window, select the instrument located in the pipeline, as shown in Figure 42.

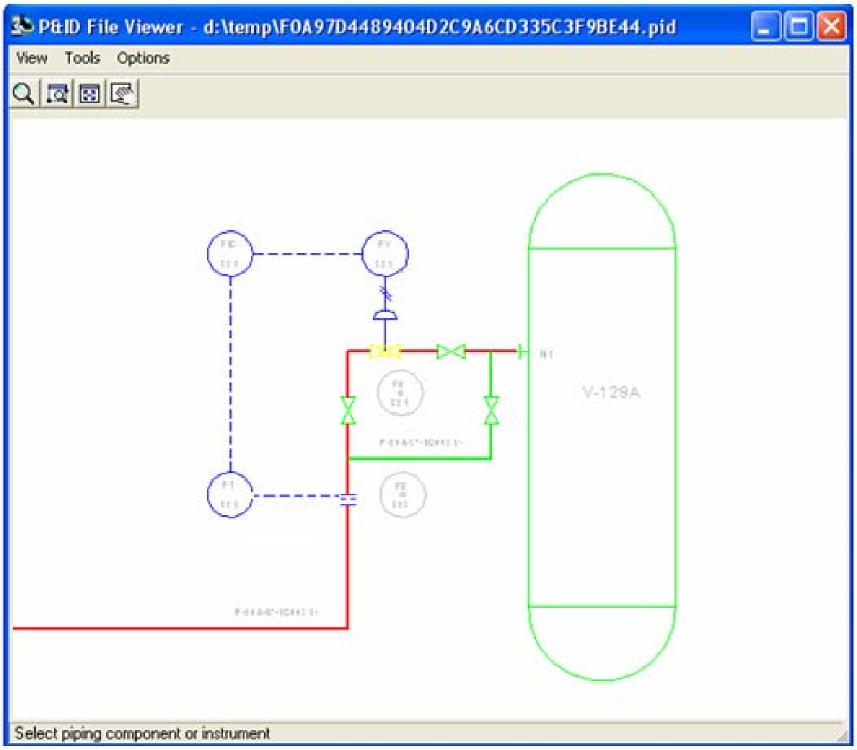


Figure 42: Selected Instrument

26. Click **Finish** on the **Insert Component** ribbon to place the instrument, as shown in Figure 43.

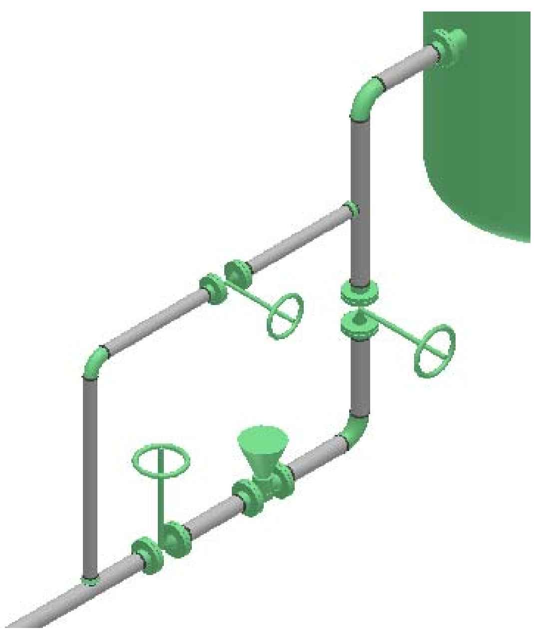


Figure 43: Placed Instrument

27. Now, place a flow instrument on the pipeline **P-204**. Click the **Insert Component** button on the vertical toolbar. Use the midpoint as the active placement point for the flow instrument.

28. Click on the active placement point of the pipe, as shown in Figure 44.

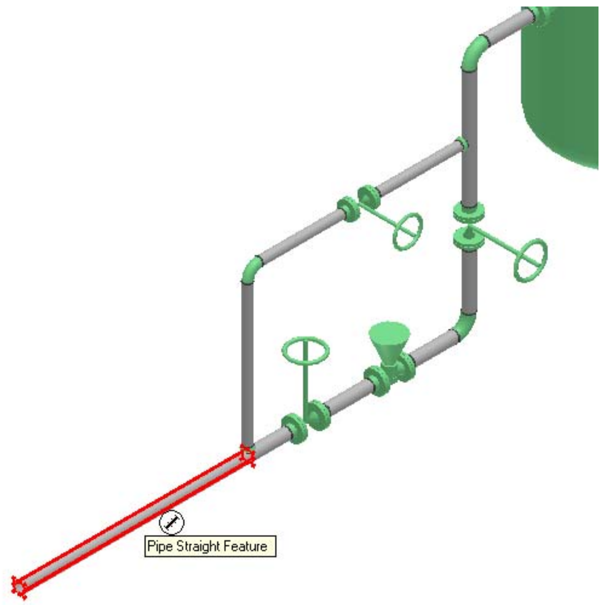


Figure 44: Pipe Straight Feature

29. The **P&ID File Viewer** window appears. In the **P&ID File Viewer** window, select the instrument located in the pipeline, as shown in Figure 45.

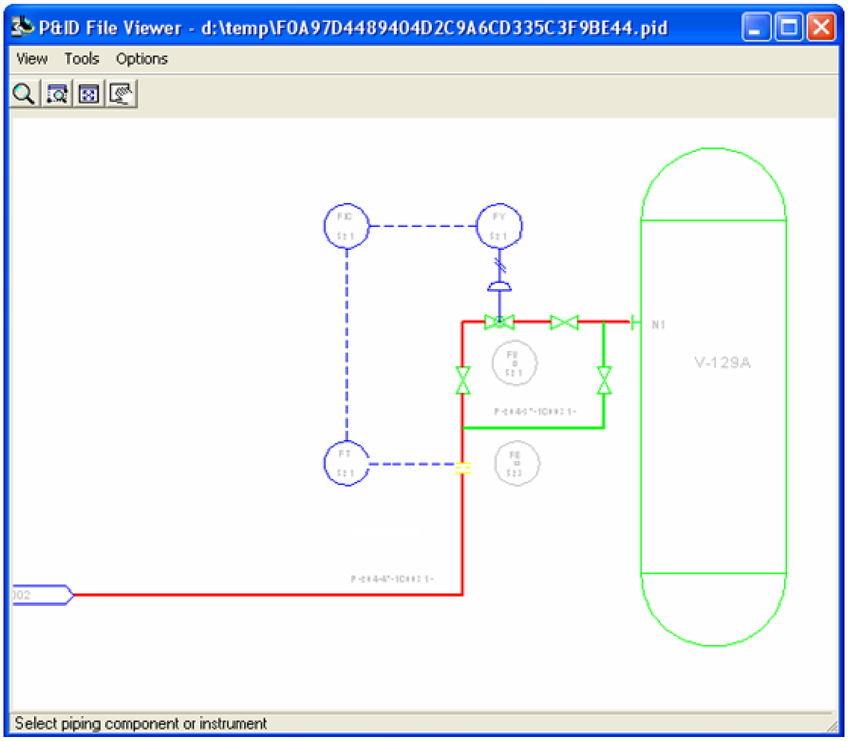


Figure 45: Selected Instrument

30. Click **Finish** on the **Insert Component** ribbon to place the instrument, as shown in Figure 46.

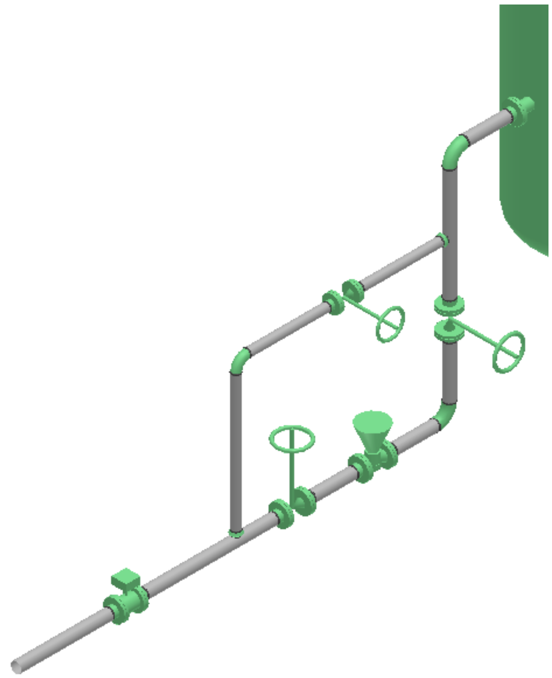


Figure 46: Placed Instrument

31. Now, continue routing the pipeline using the off-page connector (OPC). Open the continuation P&ID drawing. Click the **SmartPlant > View P&ID** command.

32. The **View P&ID** dialog box appears. Select **PID1002** drawing and click **Open.**

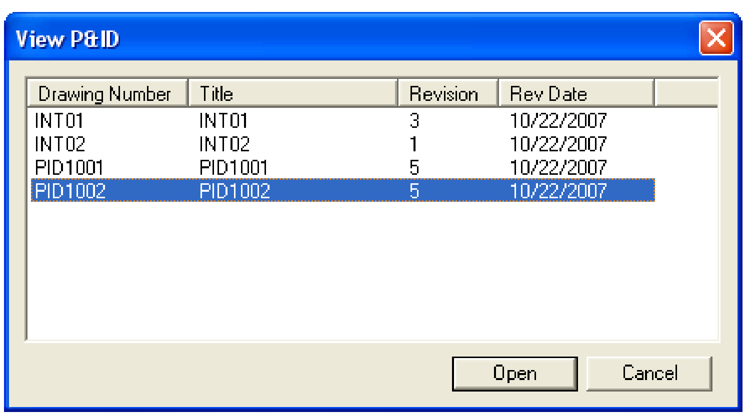


Figure 47: View P&ID Dialog Box

The **P&ID File Viewer** window appears.

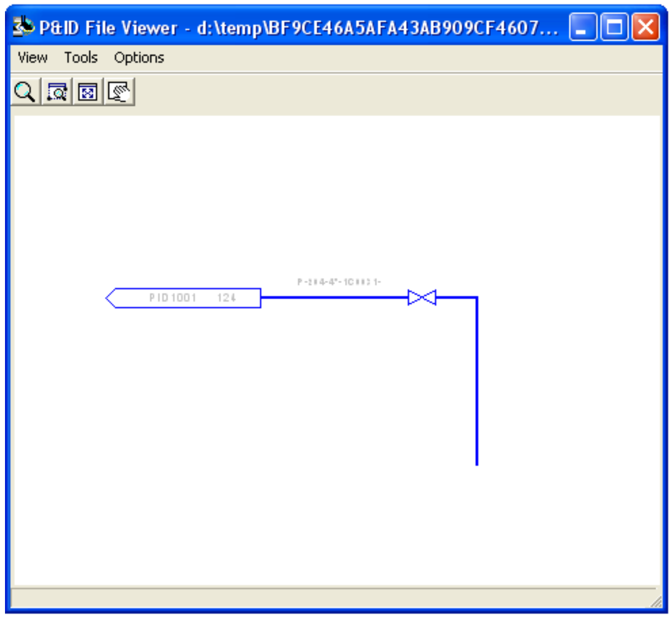


Figure 48: P&ID File Viewer dialog

33. Now, click the **Route Pipe** command on the vertical toolbar. Select the **<Select From P&ID>** option on the **Run** drop-down list on the **Route Pipe** ribbon.

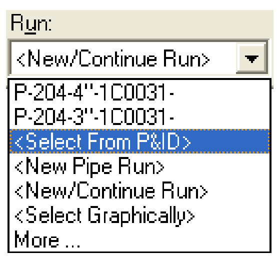


Figure 49: Route Pipe ribbon

34. The **P&ID File Viewer** window appears. In the P&ID File Viewer, select the pipeline **P-204**, as shown in Figure 50.

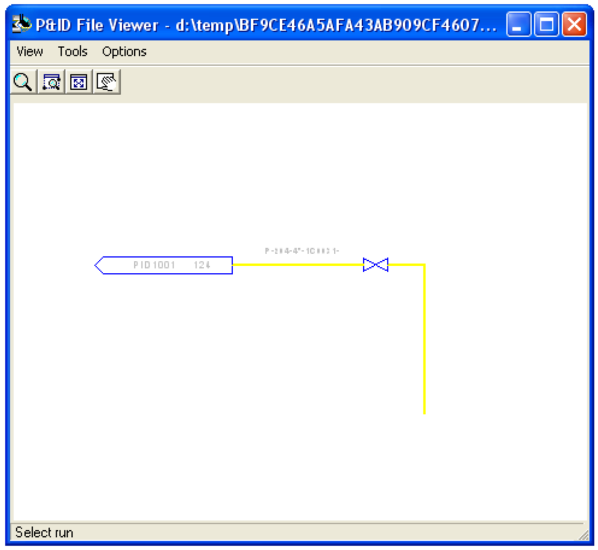


Figure 50: P-204 Pipeline in the P&ID PID1002 Drawing

35. The **New Pipe Run** dialog box appears. The system populates the **New Pipe Run** dialog box automatically by using P&ID and design basis. Click **OK** to accept the default values and close the dialog box.

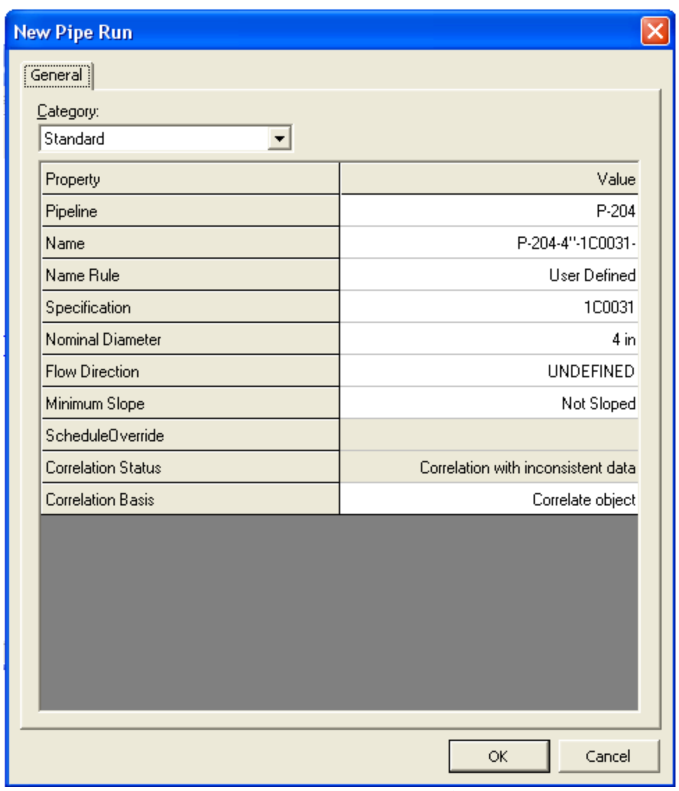


Figure 51: New Pipe Run Dialog Box

36. Select the end of the pipe to start the routing. You now see an outline of the pipe run in your model. Click to place the pipe run, as shown in Figure 52.

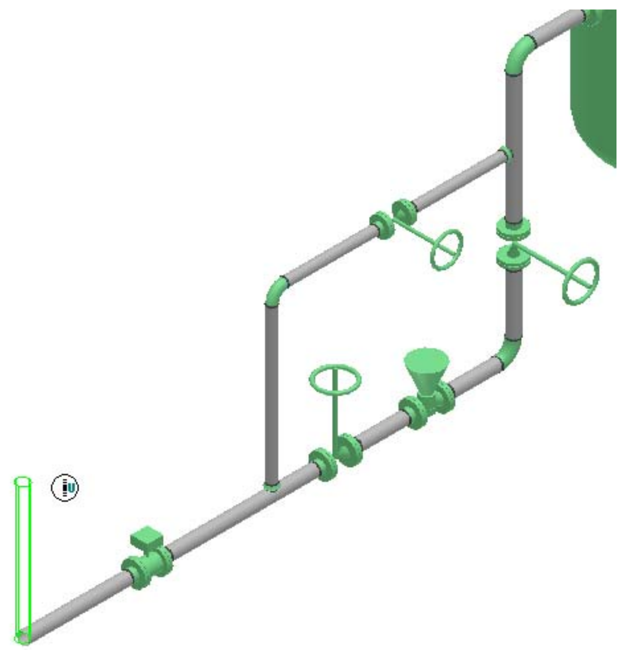


Figure 52: New Pipe Run Dialog Box

37. Click the **Insert Component** button on the vertical toolbar.

38. Now, position the cursor along the **Pipe Straight Feature**, as shown in Figure 53, and click to define the active placement point.

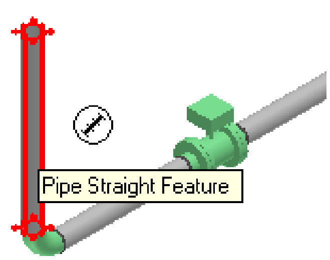


Figure 53: Pipe Straight Feature

39. The **P&ID File Viewer** window appears. In the **P&ID File Viewer**, select the gate valve, as shown in Figure 54.

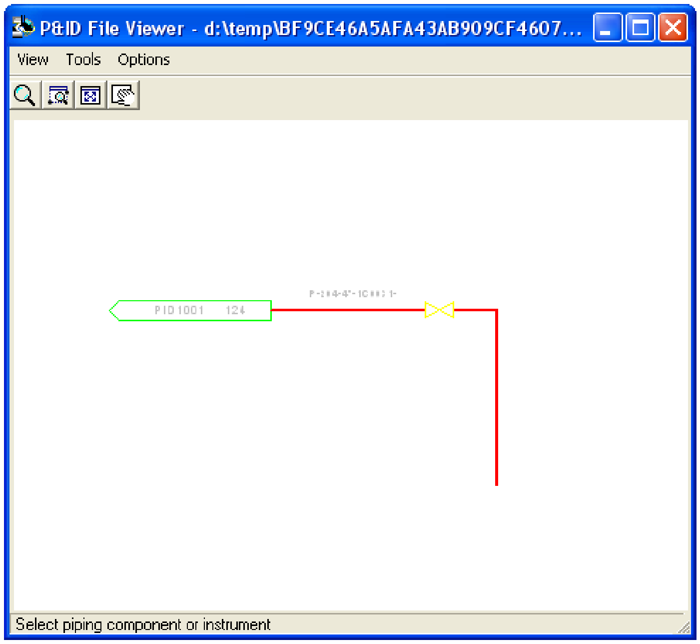


Figure 54: P&ID File Viewer Window

40. 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. Key in **90 deg** in the **Angle** drop-down list to rotate the valve.

41. Click **Finish** on the **Insert Component** ribbon to place the pipe component, as shown in Figure 55.

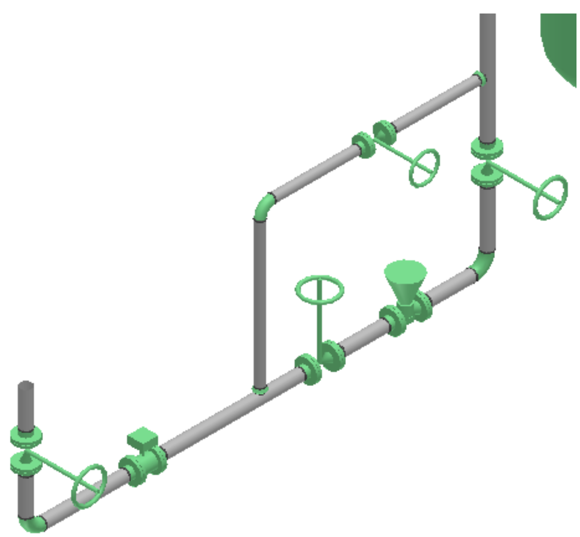


Figure 55: Placed Gate Valve

42. Now, open the **P&ID PID1001 drawing**. The pipe section you placed is highlighted in

green, as shown in Figure 56, which indicates that the pipe section and the OPC are correlated now and its properties and topology matches the design basis.

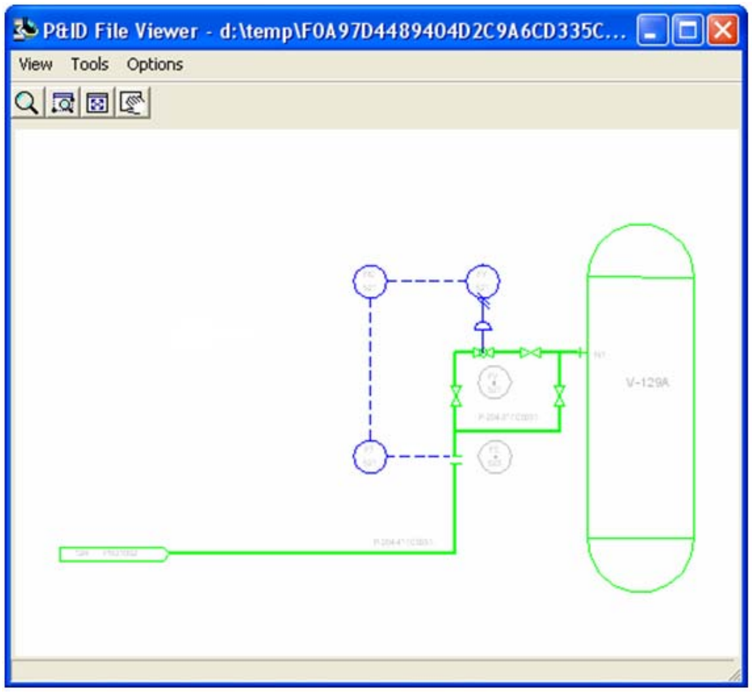


Figure 56: Correlation status of the placed pipe section in the P&ID File Viewer Window

For more information related to placing instrument, refer to the *Insert an Instrument* topic in the user guide *PipingUsersGuide.pdf:*

Session 7: 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:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run

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.

Steps for Placing Piping Specialty Items:

Exercise 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 Figure 1.

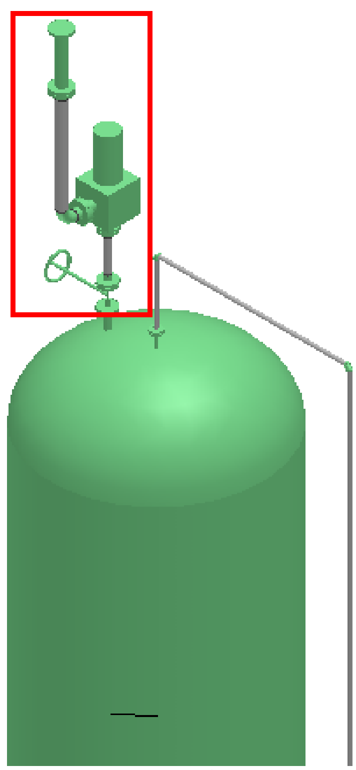


Figure 1: Placed Components and Specialty Item Silencer

Before beginning the 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.**

Steps for Placing Isolating Gate Valve:

1. Locate nozzle **F** on equipment **T-101** in Unit **U03** on which you will insert components and specialty items, as shown in Figure 2.

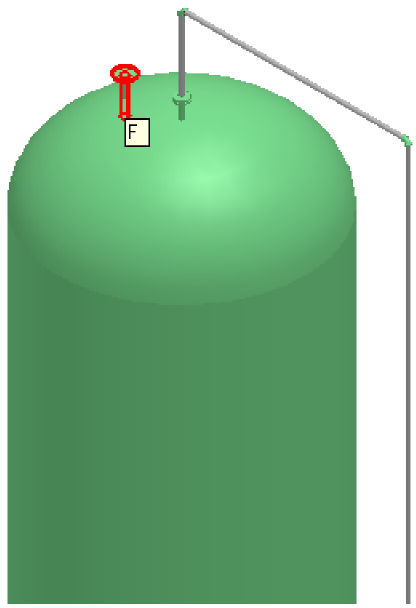


Figure 2: Nozzle F on Equipment T-101

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

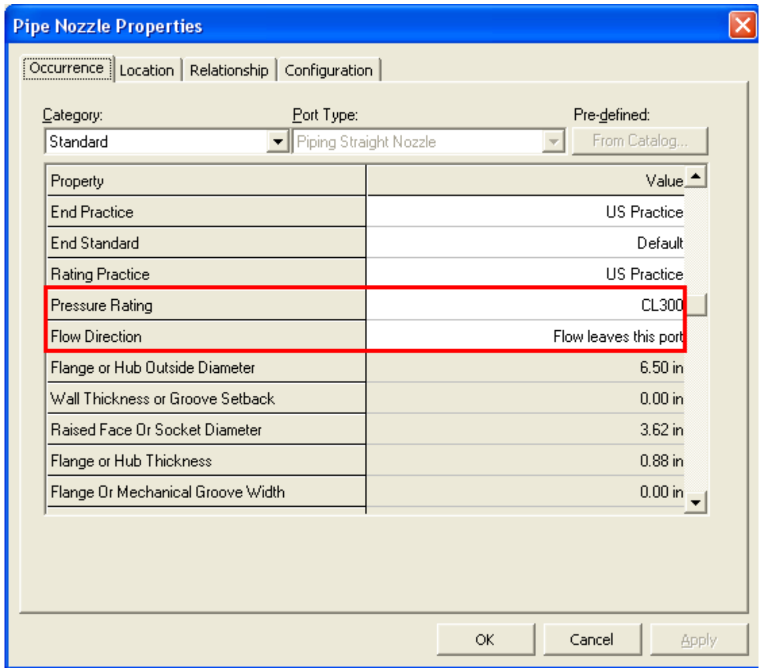


Figure 3: Pipe Nozzle Properties

4. Click the **Insert Component** button on the vertical toolbar.

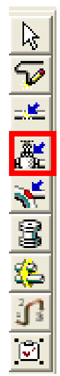


Figure 4: Insert Component Button on the Vertical Toolbar

5. If you deselect the pipe nozzle **F** from the previous step, then select the pipe nozzle **F** for placing the components.

6. The **New Pipe Run** dialog box appears as shown in Figure 5. Change the following specifications in the dialog box and click **OK** :
   Pipeline: 302-W
   Specification: 2C0032

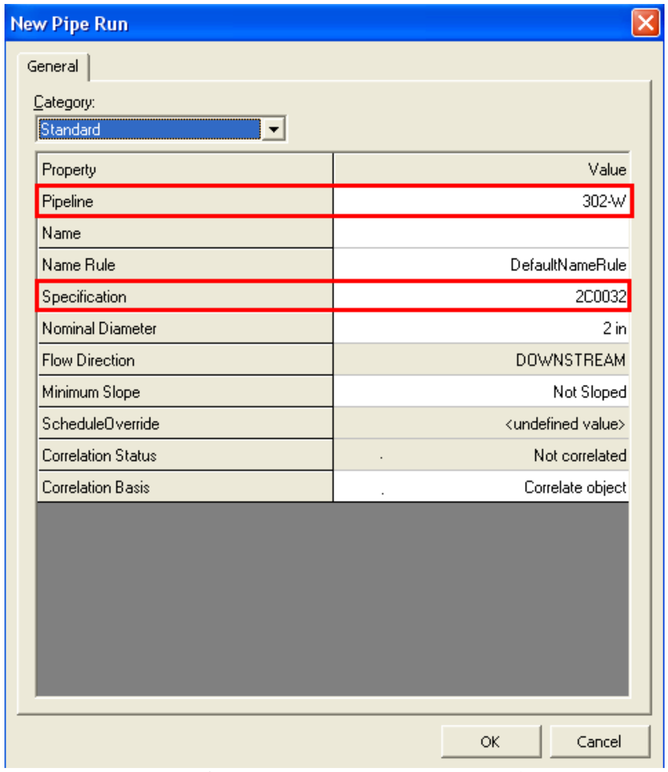


Figure 5: Specifications on New Pipe Run Dialog Box

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 as shown in Figure 6.

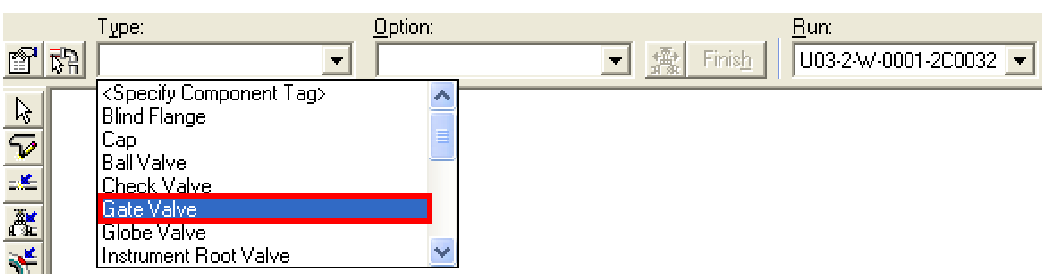


Figure 6: Gate Valve Option on the Insert Component Ribbon

8. The **Gate Valve** in the graphic view before accepting the placement appears as shown in Figure 7.

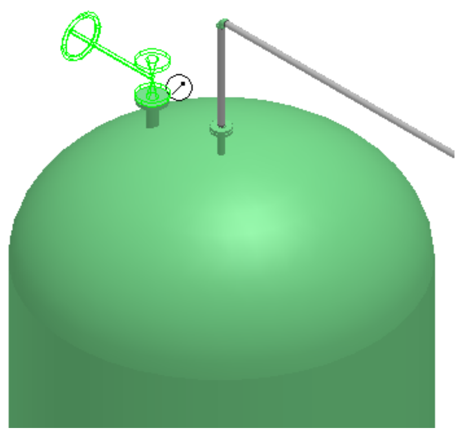


Figure 7: Gate Valve in the Graphic View

9. Key in **270 deg** in the Angle box on the **Insert Component** ribbon to rotate the gate valve to set the operator of the valve in right orientation.

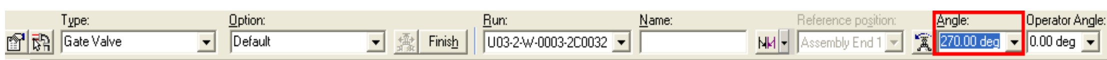


Figure 8: Angle Specified on the Insert Component Ribbon

10. Click **Finish** on the **Insert Component** ribbon to accept the placement of the gate valve.

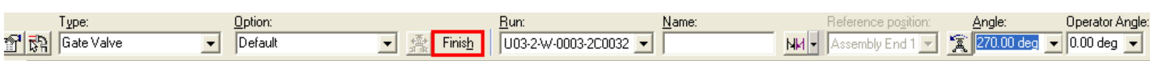


Figure 9: Finish Option on the Insert Component Ribbon

The placed gate valve should resemble Figure 10.

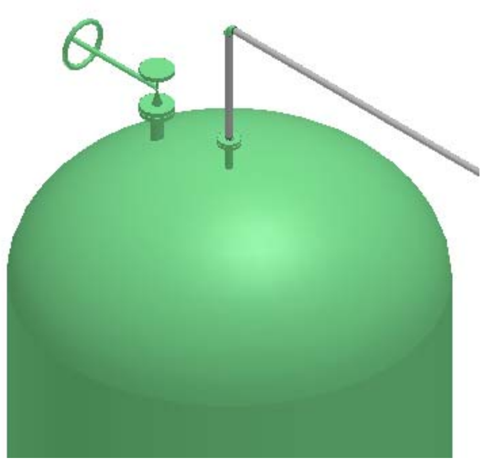


Figure 10: Gate Valve Placed on the Nozzle

11. Select **Flange** option from the **Type** drop-down list on the **Insert Component** ribbon to place a flange on top of the gate valve as shown in Figure 11.

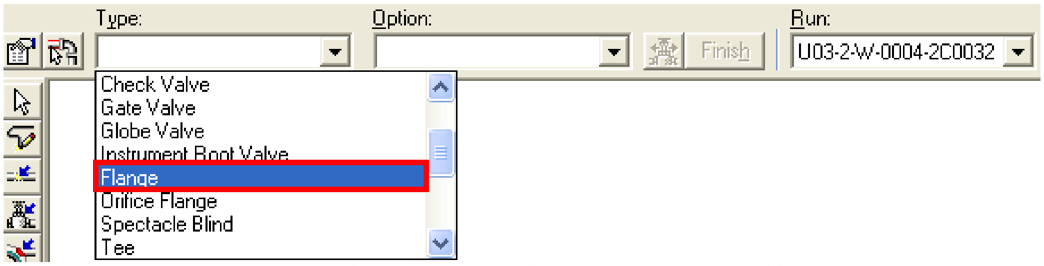


Figure 11: Flange Option on the Insert Component Ribbon

12. Click **Finish** on the **Insert Component** ribbon to accept the placement of the flange. The flange placed on the gate valve should resemble Figure 12.

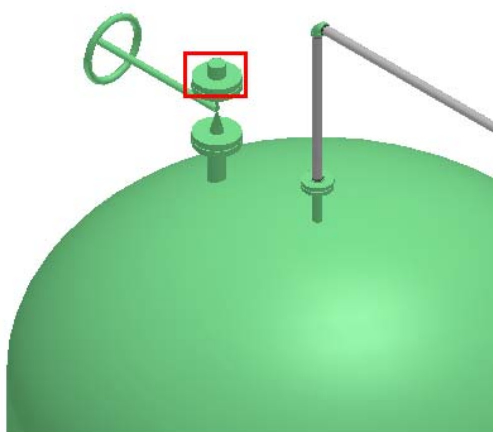


Figure 12: Flange Placed on the Gate Valve

13. 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 the **Route Pipe** command as shown in Figure 13.

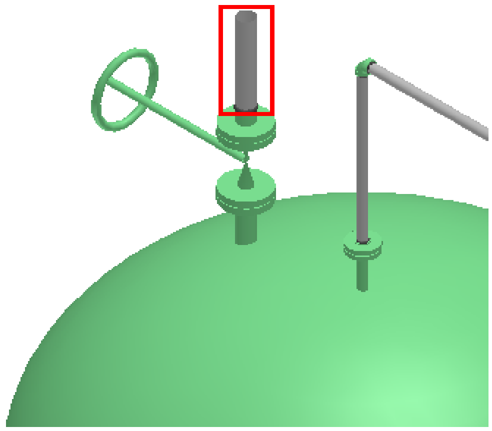


Figure 13: Pipe Routed From the Flange

14. Right-click to terminate the **Route Pipe** command.

15. After routing the pipe, you cover the top of the pipe with a flange by using the **Insert Component** command. Click the **Insert Component** button on the vertical toolbar and select the end feature of the pipe as shown in Figure 14.

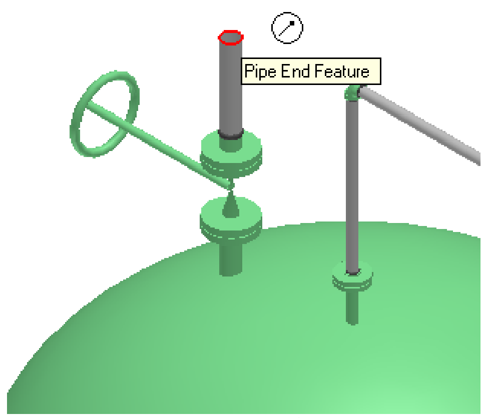


Figure 14: Pipe End Feature

16. SP3D prompts you to select type of the component to place. Select the **Flange** option from the **Type** drop-down list on the **Insert Component** ribbon and click **Finish** on the **Insert Component** ribbon to accept the placement of the flange. After placing the flange, the view of the model should resemble Figure 15.

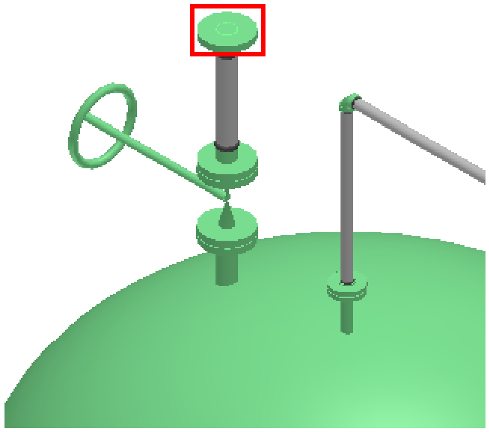


Figure 15: Flange Placed on the Pipe

Steps for Placing Pressure Relief Valve (PSV):

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 the **Specify Component Tag** from the **Type** drop-down list on the **Insert Component** ribbon to select the item from the catalog as shown in Figure 16.

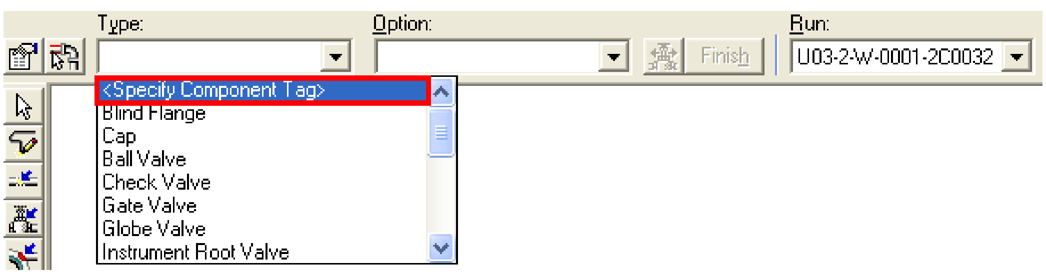


Figure 16: Specify Component Tag on the Insert Component Ribbon

2. The **Specify Component Tag** dialog box appears. Select **Browse instruments…** option from the **Component Tag** drop-down list as shown in Figure 17.

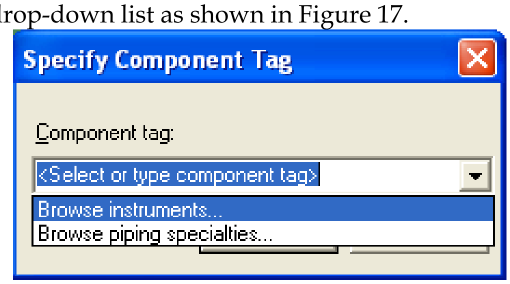


Figure 17: Specify Component Tag Dialog Box

3. The **Select Instrument** dialog box displays the instrument items from the catalog. Expand **Custom Instrument>Relief Valve Type 3** and select the part **IRVT 34** as shown in Figure 18**.** Then, click **OK.**

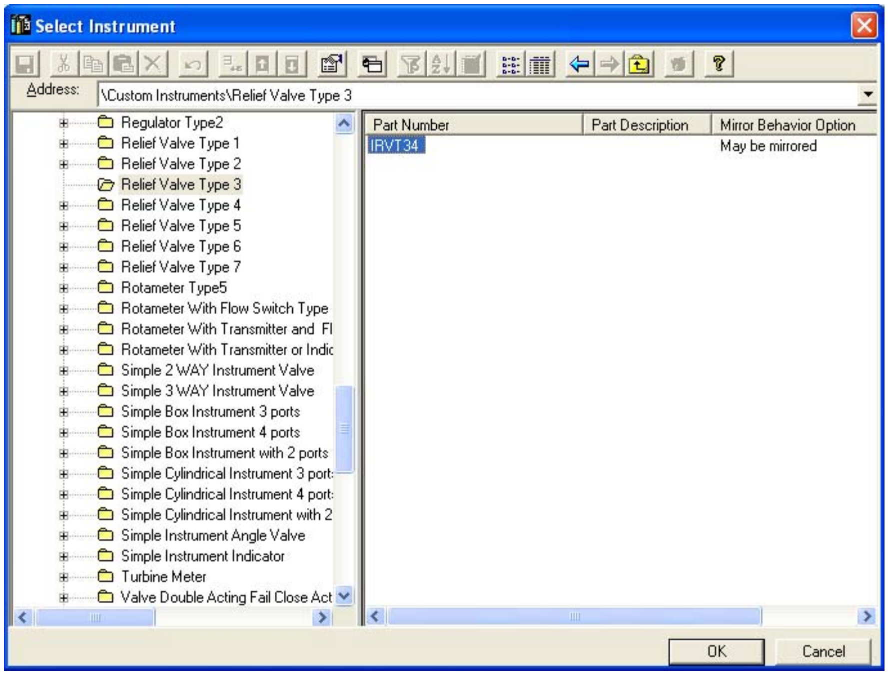


Figure 18: Select Instrument Dialog Box

4. The valve appears in the graphic view. Select the **Flip** option to change the port of the valve to **2** on the **Insert Component** ribbon, as shown in Figure 19.

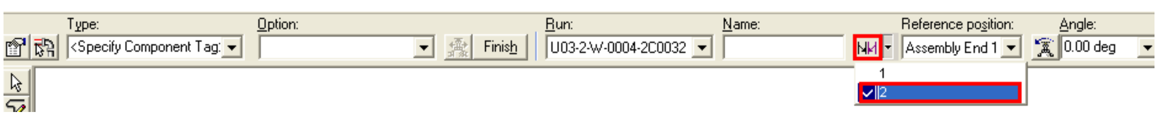


Figure 19: Changing the Port of the Relief Valve

The valve in the graphic view appears as shown in Figure 20.

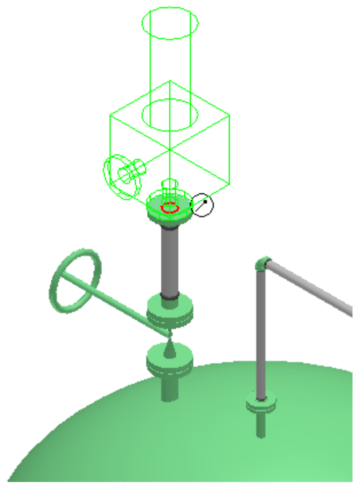


Figure 20: Relief Valve in the Graphic View

5. After you change the port of the valve to be placed, change the properties of the valve. Select the **Properties** option from the **Insert Component** ribbon as shown in Figure 21.

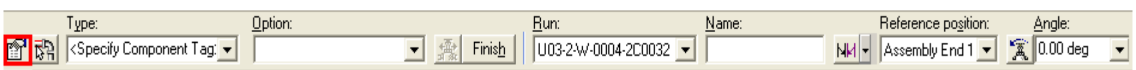


Figure 21: Properties Option on the Insert Component Ribbon

6. The **Insert Component Settings** dialog box appears. Switch to the **Dynamic Pipe Port 1** category under 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** as shown in Figure 22.

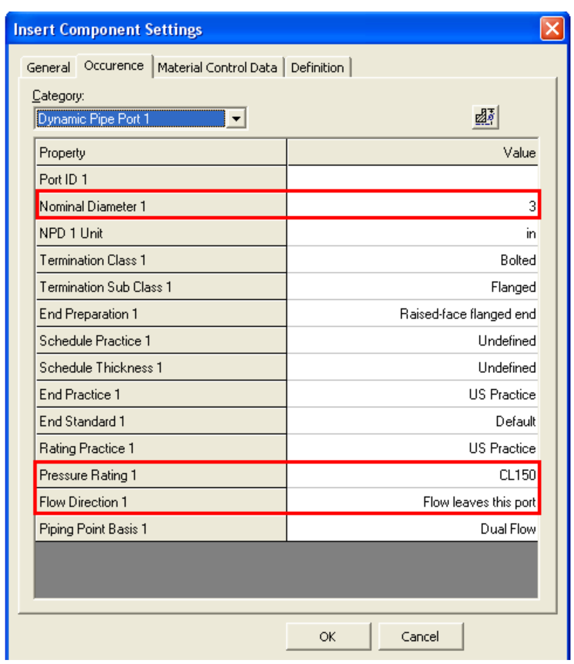


Figure 22: Insert Component Settings Dialog Box

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

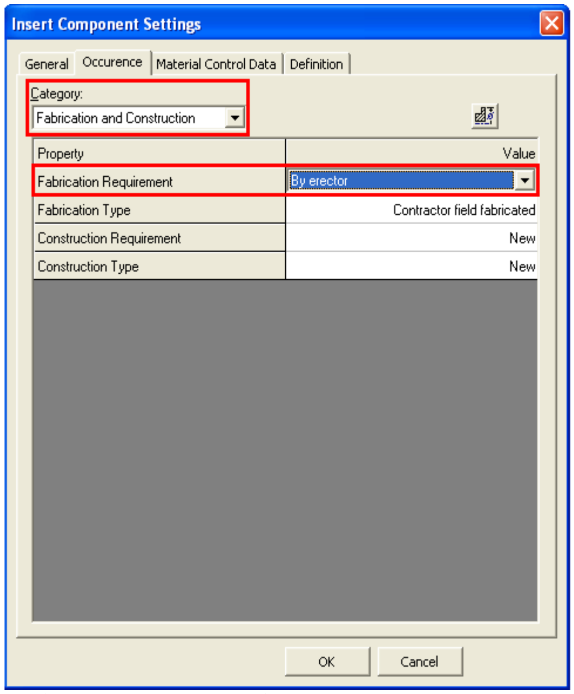
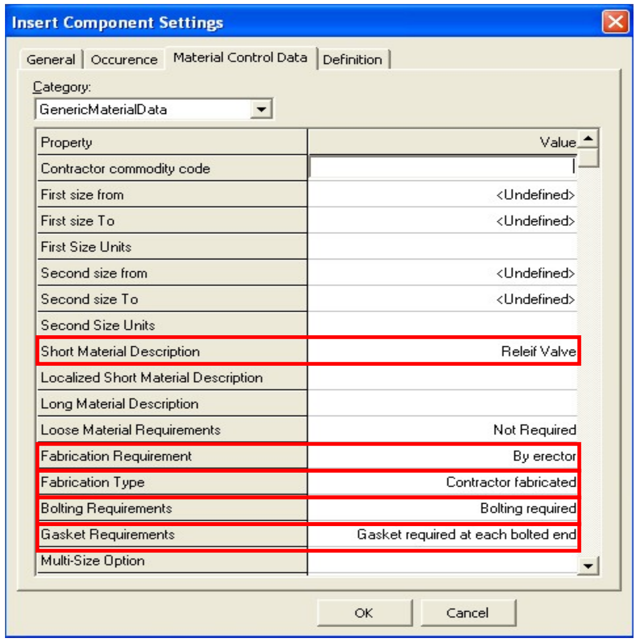


Figure 23: Setting Fabrication Settings in the Insert Component Settings Dialog Box

8. Switch to the **Material Control Data** tab and then select the **GenericMaterialData** option under the **Category** drop-down list. and change the following specifications and click **OK**:

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


Insert Component Settings

General | Occurence | Material Control Data | Definition

Category:

GenericMaterialData

Property	Value
Contractor commodity code	
First size from	<Undefined>
First size To	<Undefined>
First Size Units	
Second size from	<Undefined>
Second size To	<Undefined>
Second Size Units	
Short Material Description	Releif Valve
Localized Short Material Description	
Long Material Description	
Loose Material Requirements	Not Required
Fabrication Requirement	By erector
Fabrication Type	Contractor fabricated
Bolting Requirements	Bolting required
Gasket Requirements	Gasket required at each bolted end
Multi-Size Option	

OK | Cancel

Figure 24: Setting Material Control Data in the Insert Component Settings Dialog Box

9. Click **Finish** on the **Insert Component** ribbon to accept the placement of relief valve. The relief valve will resemble Figure 25 in the graphic view. Right-click to terminate the **Insert Component** command.

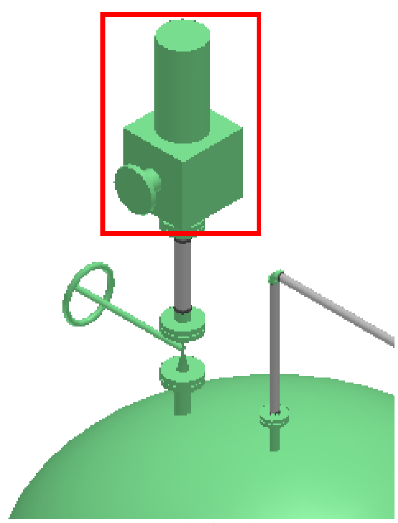


Figure 25: Relief Valve

10. Select the end feature of the valve, as shown in Figure 26, and open the **Pipe End Feature Properties** dialog box to reduce the pressure and change the diameter.

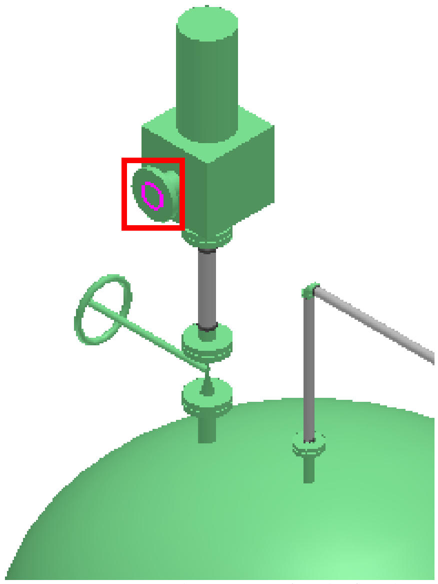


Figure 26: End Feature of Relief Valve

11. Change the following specifications in the **Pipe End Feature Properties** dialog box, as shown in Figure 27, and click **OK**.

Specification: 1C0031
Nominal Diameter: 3 in

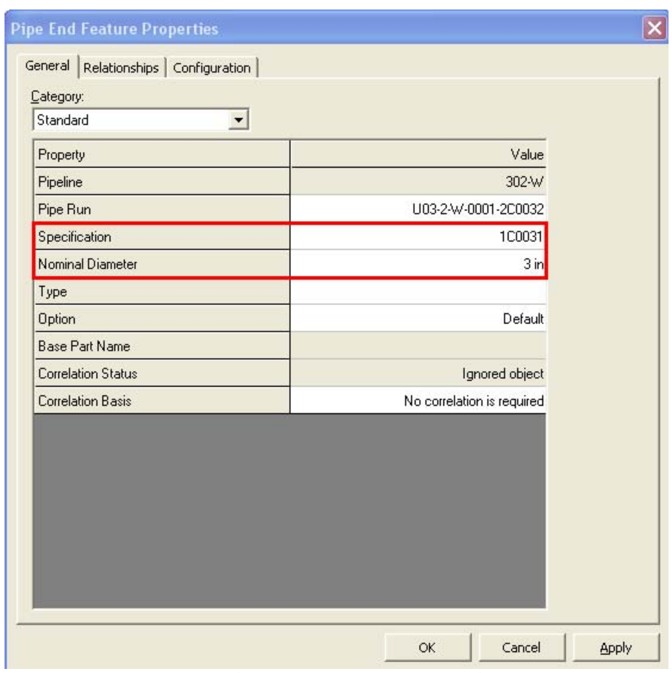


Figure 27: Pipe End Feature Properties Dialog Box

12. Select **Route Pipe** Command to begin routing from Pipe end feature.

13. Select the **New Pipe Run** option in the **Run** drop-down list on the **Route Pipe** ribbon to create a new pipe run as shown in Figure 28.

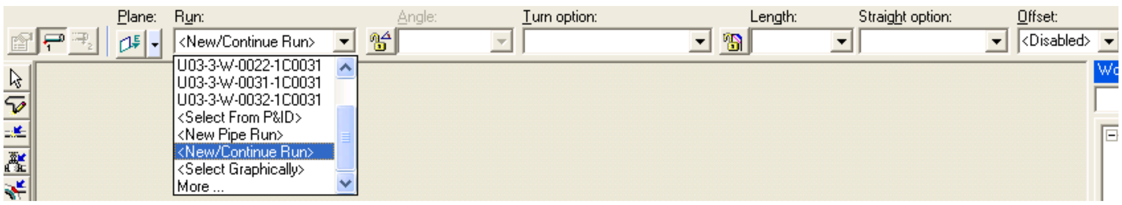


Figure 28: Run Drop-Down List on the Route Pipe Ribbon

14. Change the following specifications in the **New Pipe Run** dialog box, as shown in Figure 29, and click **OK**.

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

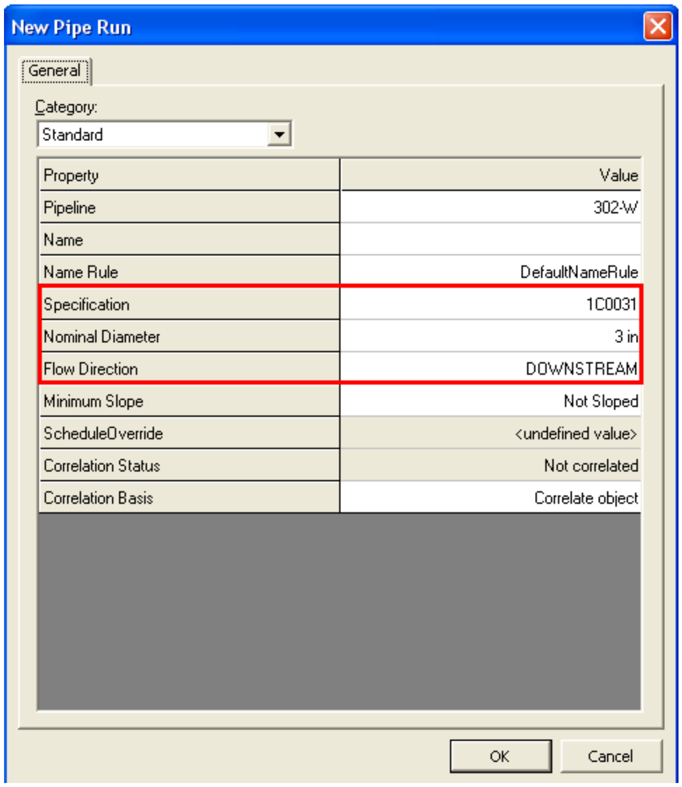


Figure 29: New Pipe Run Dialog Box

15. While still on the Route pipe Ribbon lock the **Angle** at 90 deg. and lock **Length** at 3′ *as shown in figure 30.*

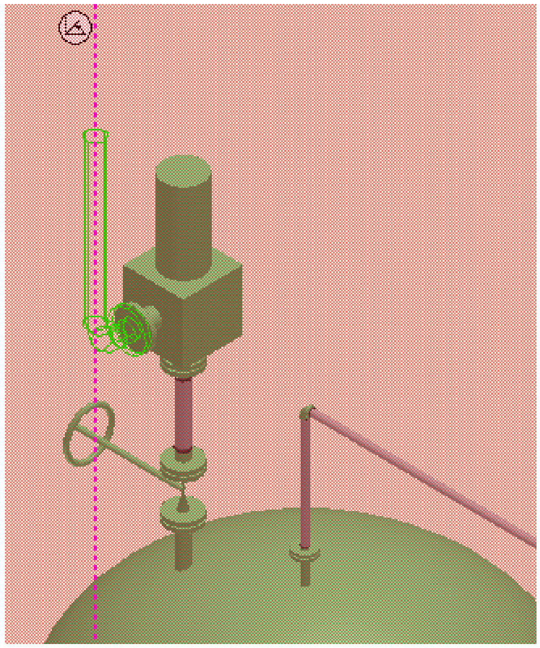


Figure 30: Pipe routing from modified pipe End Feature

16. Click in the graphic View to place pipe. Pipe run should resemble Figure 30.

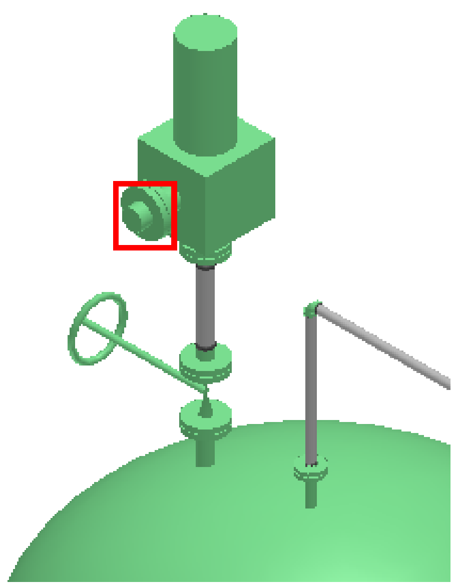


Figure 30: Flange Placed on the Relief Valve

Steps for 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. Place an elbow and then place a vent pipe on it.

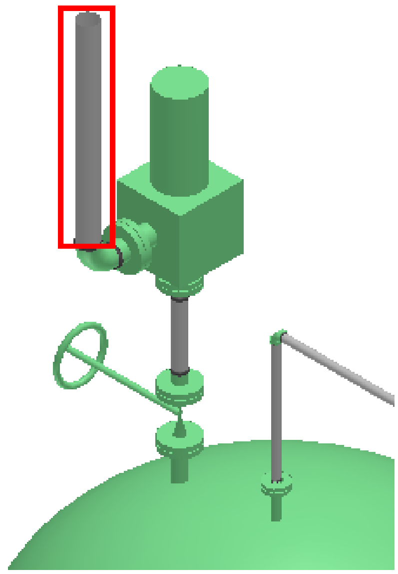


Figure 32: Vent Pipe

1. Place a flange at the top of the pipe that you routed above by using the **Insert Component** command. The flange after placement will resemble Figure 33 in the graphic view.

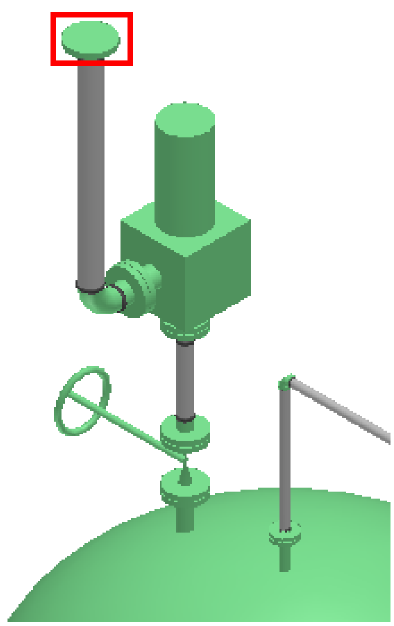


Figure 33: Flange Connected With the Vent Pipe

Steps for 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 the **Specify Component Tag** from the **Type** drop-down list on the **Insert Component** ribbon as shown in Figure 34.

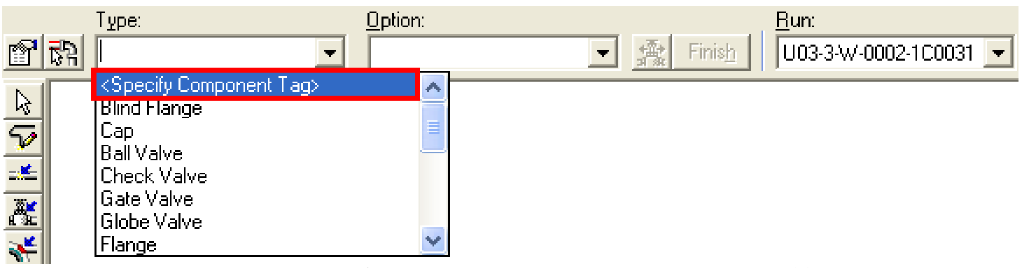


Figure 34: Specify Component Tag on Insert Component Ribbon

2. The **Specify Component Tag** dialog box appears. Select **Browse piping specialties…** option from the **Component Tag** drop-down list as shown in Figure 35.

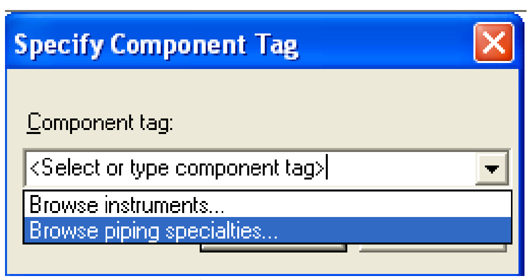


Figure 35: Specify Component Tag Dialog Box

3. The **Select Pipe Specialty** dialog box displays the specialty items from the catalog. Expand **Custom Specialties > Custom Specialty Inline Silencer** and select the part **CSInlineSilencer4** as shown in Figure 36**.** Then click **OK.**

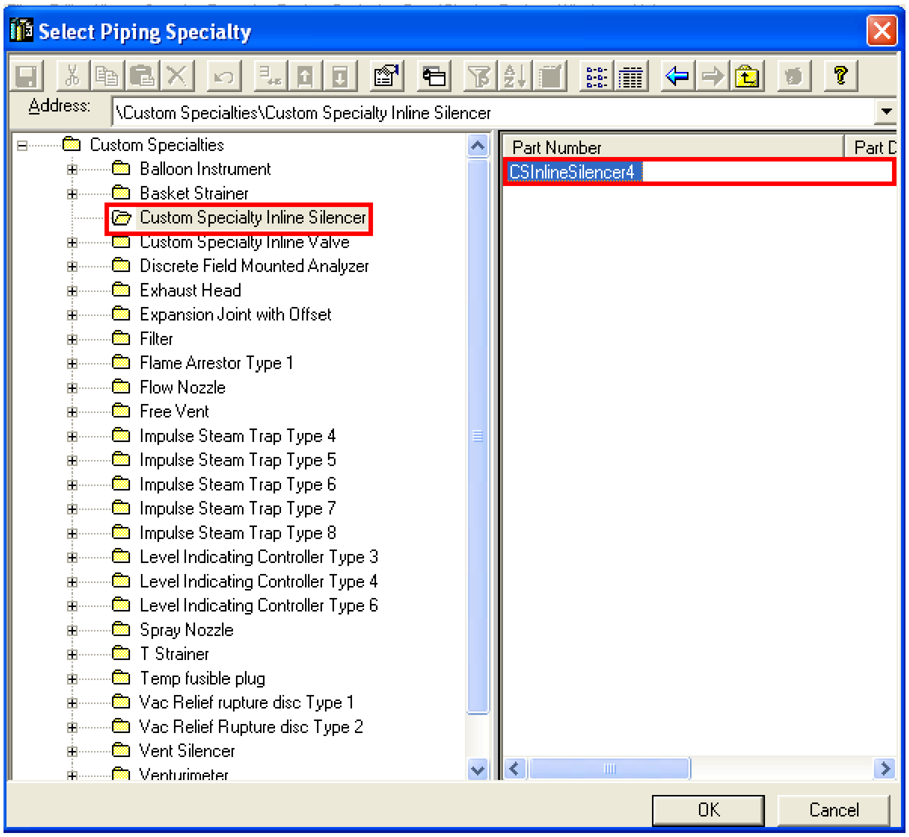


Figure 36: Select Piping Specialty Dialog Box

The specialty item silencer will resemble Figure 37 in the graphic view.

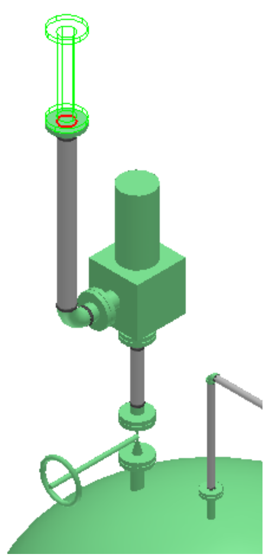


Figure 37: Silencer in the Graphic View

4. 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 the **Properties** option on the **Insert Component** ribbon and open the **Pipe Component Feature Properties** dialog box.

5. Switch to **Fabrication and Construction** category under **Occurrence** tab in the **Pipe Component Feature Properties** dialog box.

6. Select **By erector** from the **Fabrication Requirement** drop-down list in the **Pipe Component Feature Properties** dialog box, as shown in Figure 38.

7. Select **Contractor field fabricated** from the **Fabrication Type** drop-down list in the **Pipe Component Feature Properties** dialog box, as shown in Figure 38.

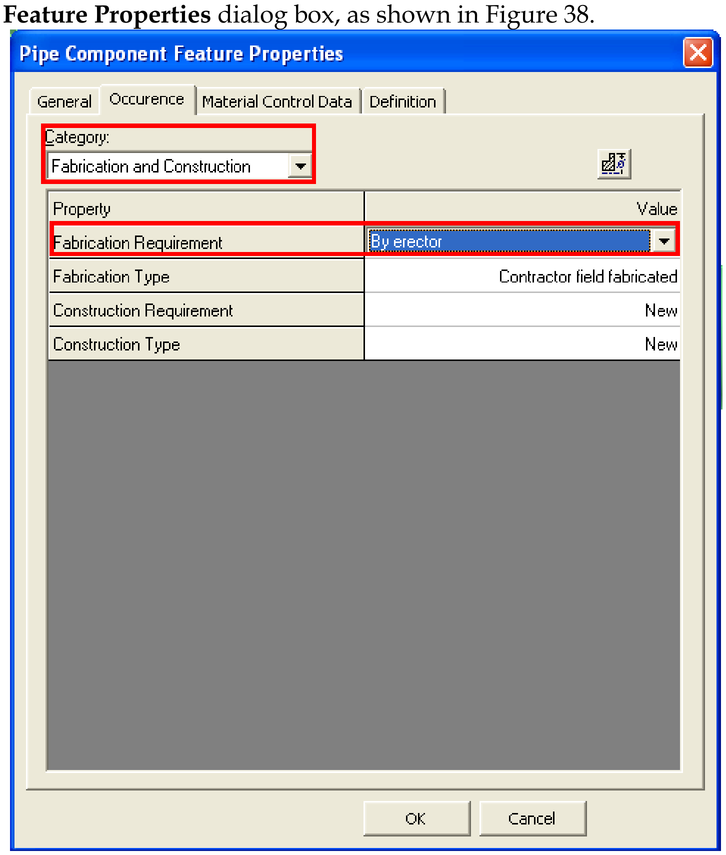


Figure 38: Fabrication Settings on Pipe Component Feature Properties Dialog Box

8. Switch to **Material Control Data** tab and then select the **GenericMaterialData** option under the **Category** drop-down list .Change the following specifications and click **OK**.

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

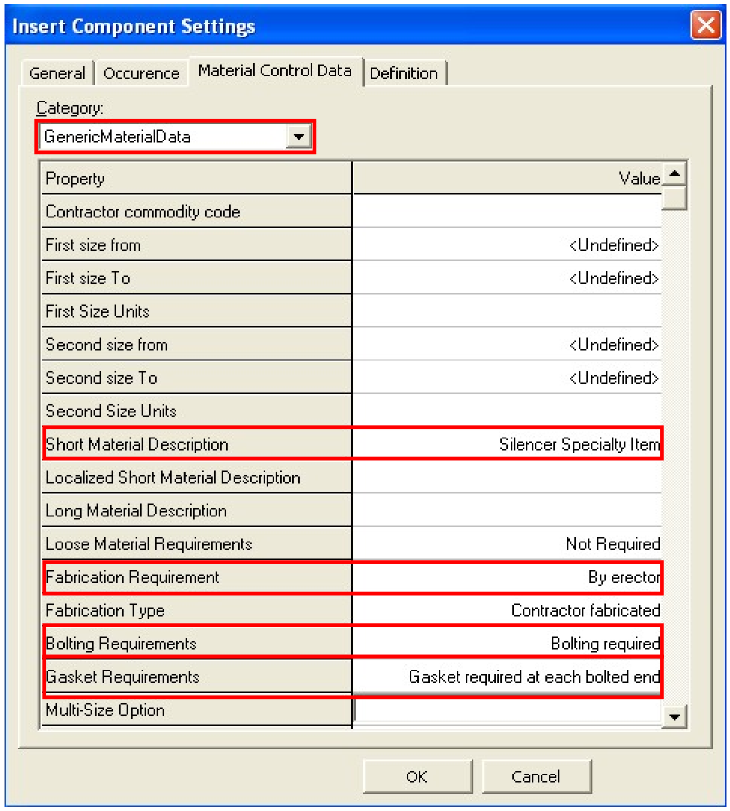


Figure 39: Pipe Component Feature Properties Dialog Box

9. Click **Finish** on the **Insert Component** ribbon to accept the placement of the silencer and right-click in the graphic view to terminate the command. The silencer will resemble Figure 40.

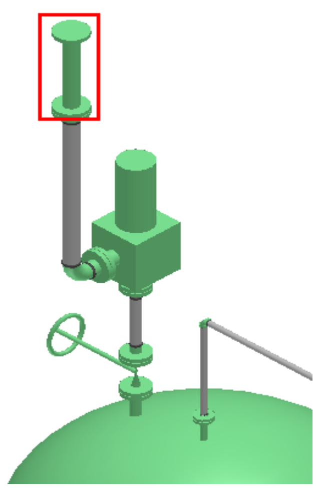


Figure 40: Specialty Item: Silencer

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

Session 8: Placing Taps

Objective:

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

- Place a tap on pipe parts.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run

Overview:

In SP3D taps are placed on pipe or piping components. You use the **Insert Tap** command 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.

Steps for Inserting a Tap to Place a Pipe Trunnion:

Exercise Objective: In this exercise you will be inserting a tap on the elbow **90 Degree Direction Change-0034** of the pipeline **300-W** by 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 of Figure 1.

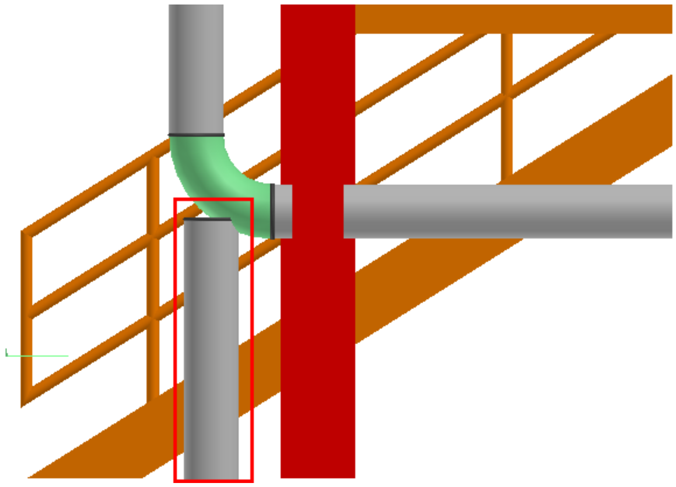


Figure 1: Pipe Trunnion

Before beginning the 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.**

17. Change the view of the model to **Looking East** by using the **Common Views** dialog box to focus on the piping components.

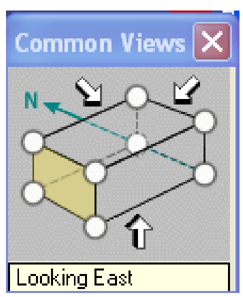


Figure 2: Common Views Dialog Box

18. Locate the elbow **90 Degree Direction Change-0034** from the **Workspace Explorer**, as shown in Figure 3, to highlight it in the graphic view.

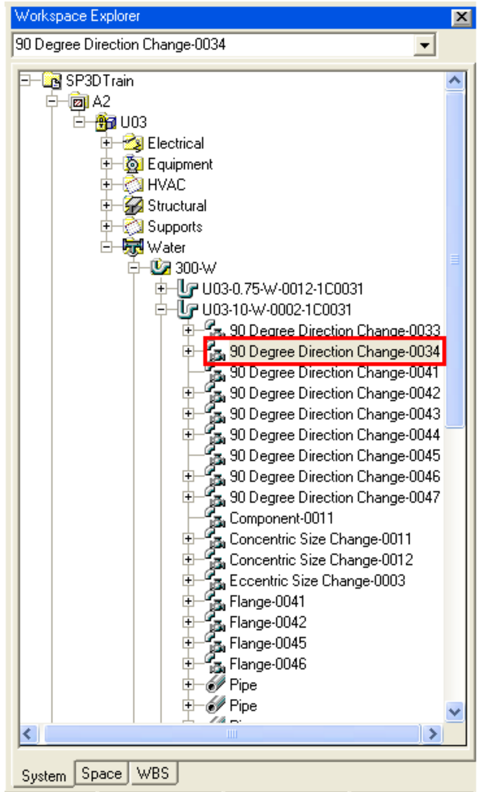


Figure 3: Elbow Located in the Workspace Explorer

19. Click the **Insert Tap** button on the vertical toolbar to activate the **Insert Tap** ribbon.

Figure 4: Insert Tap Button on the Vertical Toolbar

20. Select the elbow **90 Degree Direction Change** ***(as shown below)*** of pipeline **300-W** in the graphic view, as shown in Figure 5. This is the portion of the pipeline where you will place the tap.

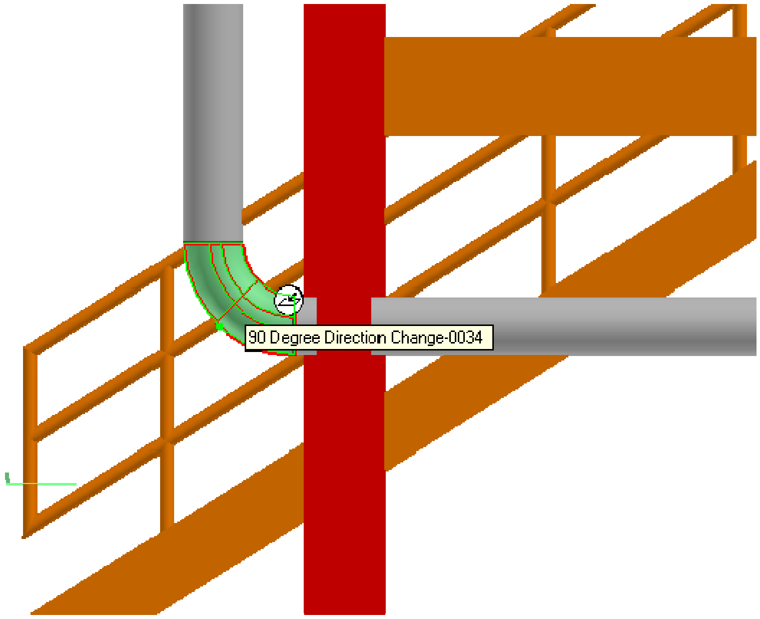


Figure 5: Selection of Elbow in the Graphic View

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

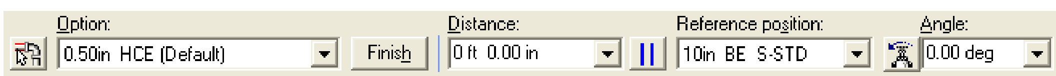


Figure 6: Insert Tap Ribbon

21. Select the **10in HCE** option in the **Option** drop-down list on the **Insert Tap** ribbon to specify a 10 inch Hole Circular End tap to be placed in the selected elbow. SP3D generates this list based on the pipe specification of the component into which you want to insert the tap.

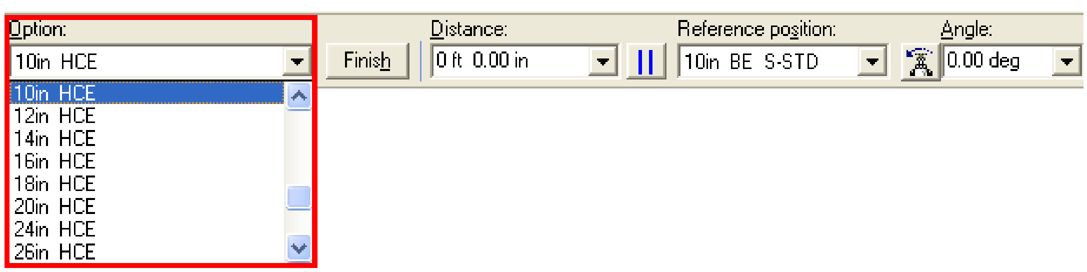


Figure 7: Option Drop-Down List on the Insert Tap Ribbon

22. After defining the tap ID to place the tap perpendicular to the elbow, set the **Orientation** option to perpendicular on the **Insert Tap** ribbon. This will measure the distance from the reference port to the tap location along the arc of the turn feature.

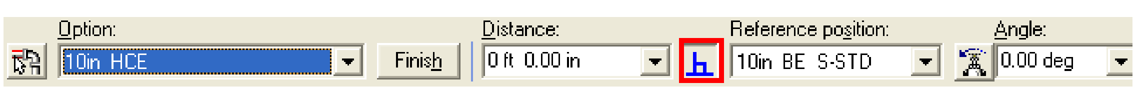


Figure 8: Orientation Option on the Insert Tap Ribbon

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.

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

Figure 9: Distance Defined on the Insert Tap Ribbon

24. Click the **Finish** button on the **Insert Tap** ribbon to accept the placement of the tap.

Figure 10: Finish Button on the Insert Tap Ribbon

The tap placed on the elbow should resemble the highlighted section of Figure 11.

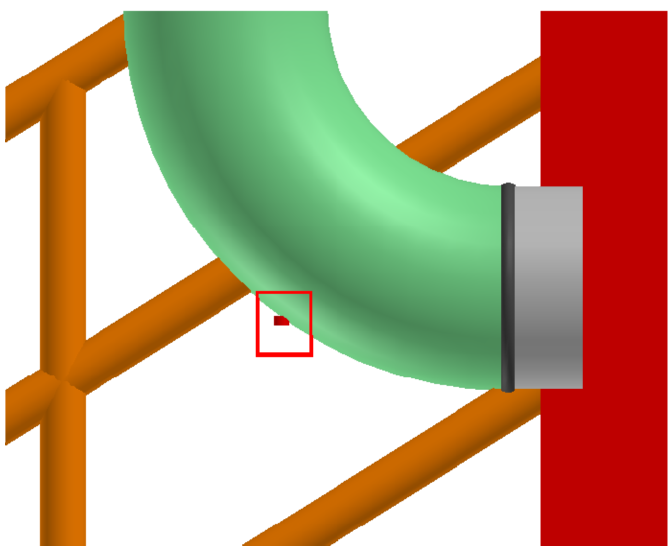


Figure 11: Tap Placed on the Elbow

25. After placing the tap you next route a pipe starting from the end of the tap port to support the pipeline. Click the **Route Pipe** button on the vertical toolbar and select the tap point.

26. The **New Pipe Run** dialog box appears. Click **OK** to accept the default pipe run properties.

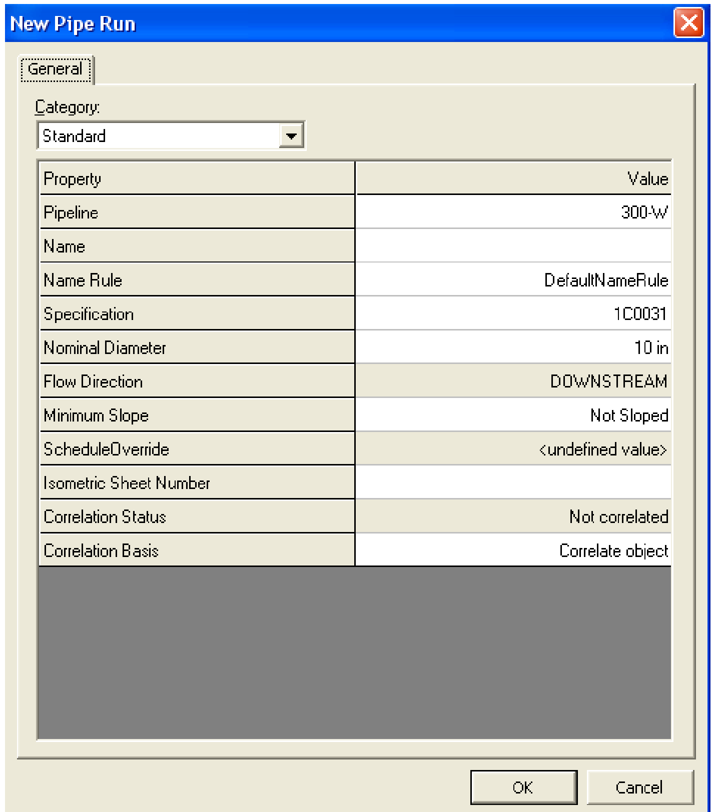


Figure 12: New Pipe Run Dialog Box

27. Define the length of the pipe trunnion by moving the cursor and find the intersection between the grid line and the steel column, as shown in Figure 13.

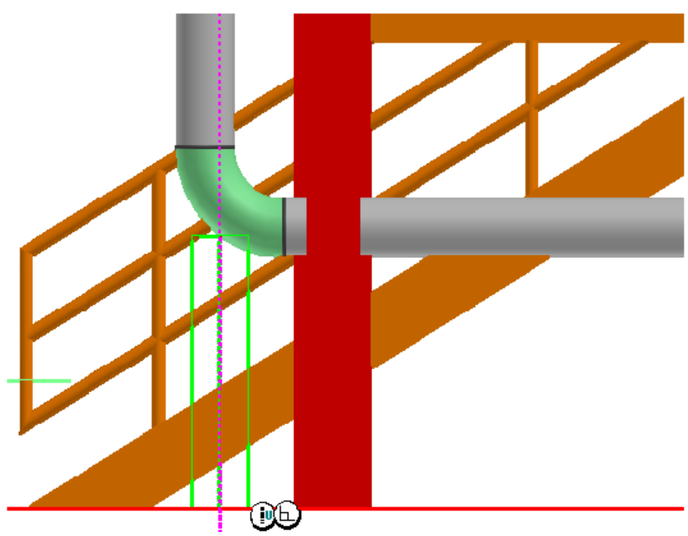


Figure 13: Defining the Length of the Pipe Trunnion

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

Steps for Inserting Tap to Connect Components:

Exercise 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 of Figure 14.

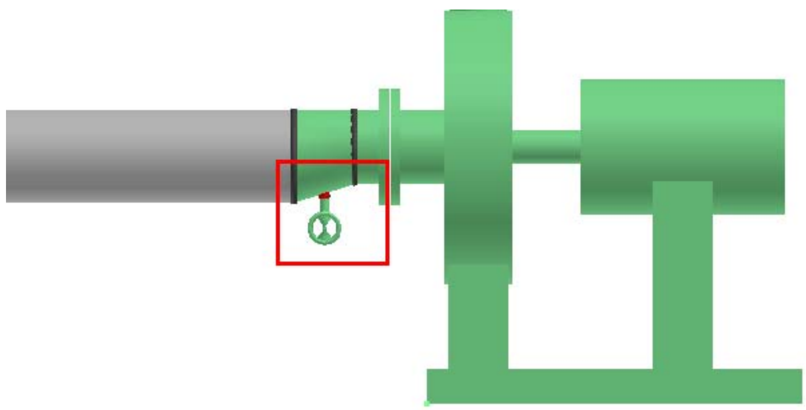


Figure 14: Instrument Root Valve Connected to Eccentric Reducer

Before beginning the 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 view of the model to **Looking North** by using the **Common Views** dialog box to focus on piping components.

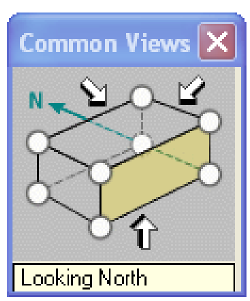


Figure 15: Common Views: Looking North

2. Locate the **Eccentric Size Change-0003** from the **Workspace Explorer**, as shown in Figure 16, to highlight it in the graphic view.

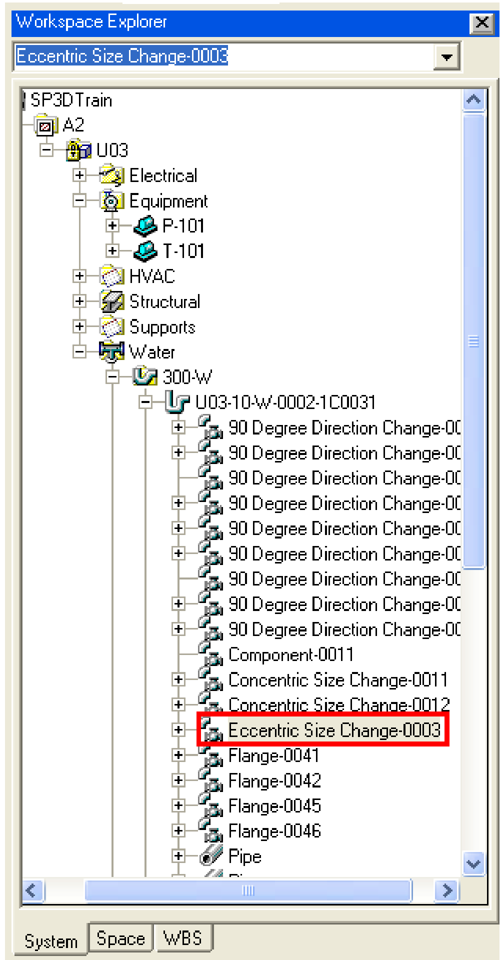


Figure 16: Eccentric Reducer Located on the Workspace Explorer

3. Click the **Insert Tap** button on the vertical toolbar to activate the **Insert Tap** ribbon.

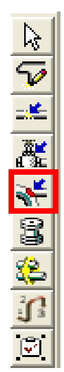


Figure 17: Insert Tap Button on the Vertical Toolbar

4. Select the eccentric reducer **Eccentric Size Change**, as shown in Figure 18.

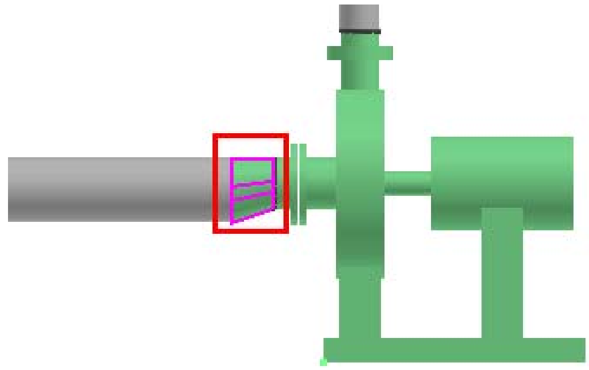


Figure 18: Selected Eccentric Reducer

5. Select the **0.75in SWE 3000** option in the **Option** drop-down list on the **Insert Tap** ribbon to specify the tap ID to be placed.

6. Click the **Finish** button to accept the placement of the tap. The tap placed on the eccentric reducer should resemble Figure 19.

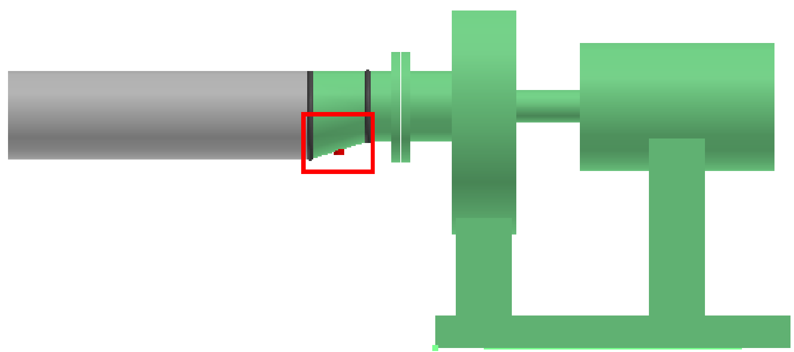


Figure 19: Tap Placed on the Eccentric Reducer

7. Now place a nipple at the end of the tap point to connect the instrument root valve with the tap. Click the **Insert Component** button on the vertical toolbar and select the tap point in the graphic view.

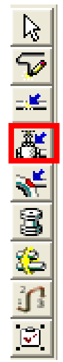


Figure 20: Insert Component Button on the Vertical Toolbar

8. The **New Pipe Run** dialog box appears. Click **OK** in the dialog box to accept the default pipe run properties.

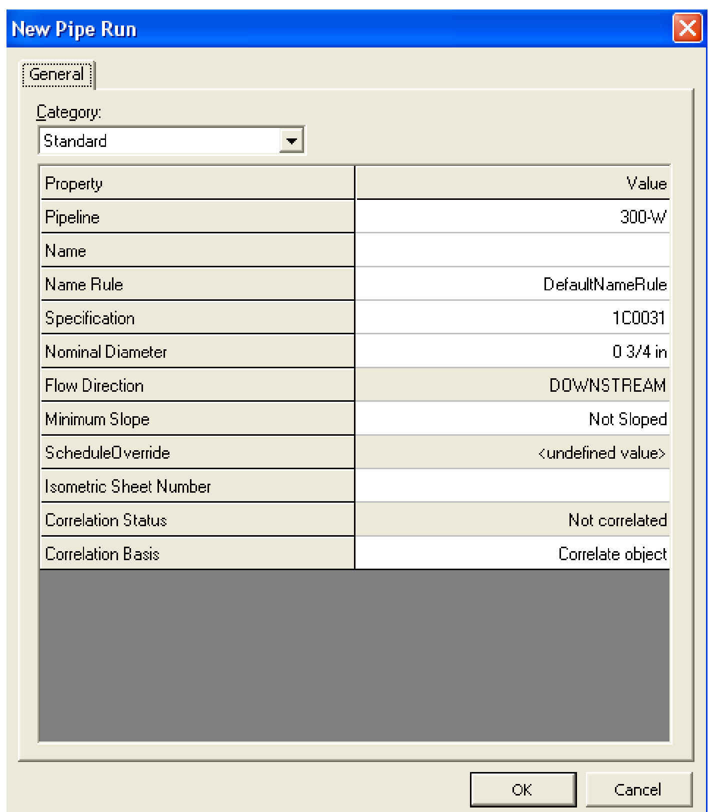


Figure 21: New Pipe Run Dialog Box

9. The **Insert Component** ribbon appears. Select the **Nipple** option in the **Type** drop-down list on the **Insert Component** ribbon to insert the nipple.

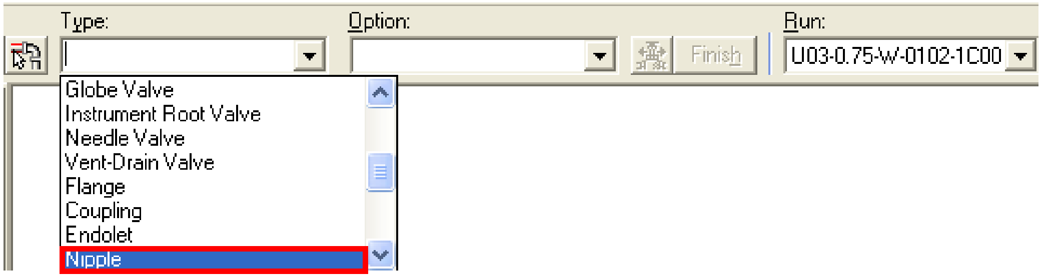


Figure 22: Specify Component Type for Placing Nipple

The nipple appears in the graphic view, as shown in Figure 23.

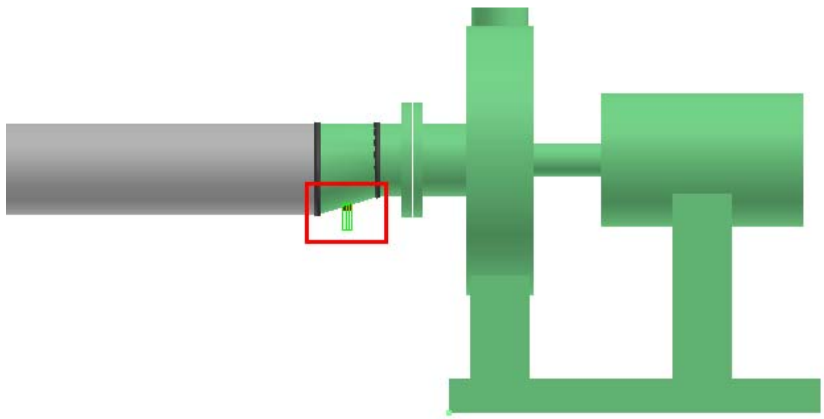


Figure 23: Nipple on the Tap in the Graphic View

10. Click the **Finish** button on the **Insert Component** ribbon 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 of Figure 24.

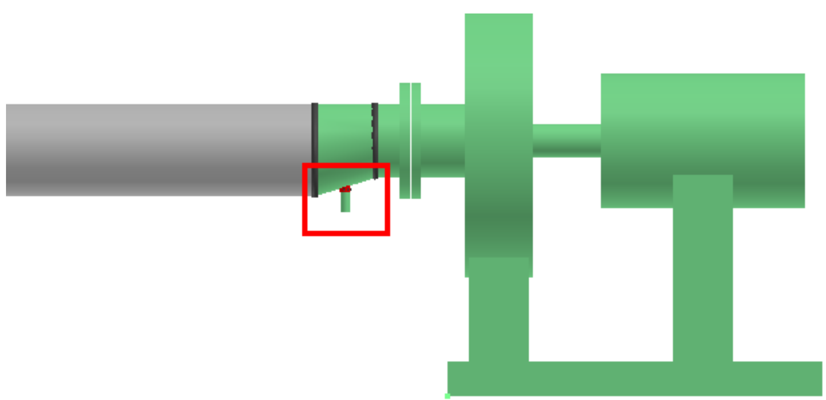


Figure 24: Nipple Placed on the Tap

11. Now again select the **Instrument Root Valve** option in the **Type** drop-down list on the **Insert Component** ribbon to connect the root valve with the eccentric reducer.

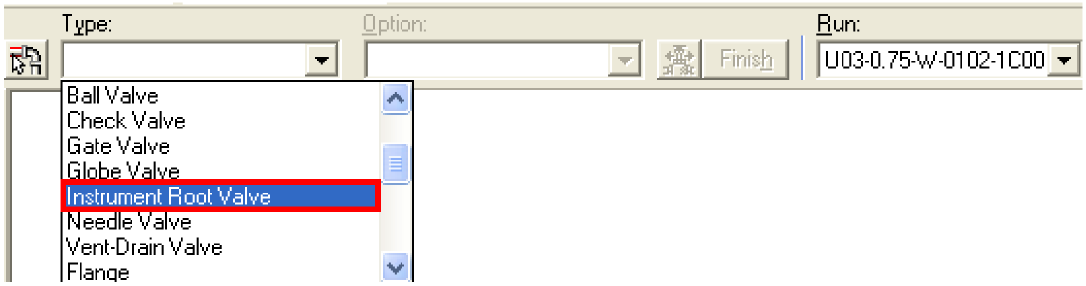


Figure 25: Specifying the Component Type for Instrument Root Valve

The **Instrument Root Valve** appears in the graphic view, as shown in Figure 25.

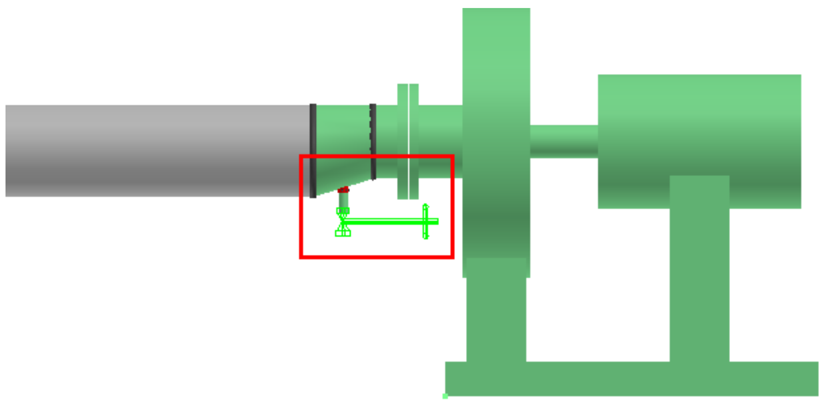


Figure 26: Instrument Root Valve Connected to the Nipple

12. Key in **270 deg** in the **Angle** drop-down list on the **Insert Component** ribbon to rotate the valve.

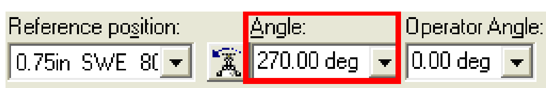


Figure 27: Specifying Angle for Rotating the Root Valve

13. Click **Finish** on the **Insert Component** ribbon to accept the placement of the **Instrument Root Valve.**

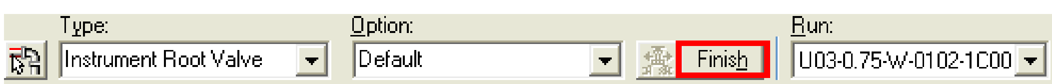


Figure 28: Finish Button on the Insert Component Ribbon

For more information related to sequencing objects, refer to *Inserting Taps: An Overview,* topic in the user guide *PipingUsersGuide.pdf.*

Session 9: Inserting Splits

Objective:

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

- Insert splits in a pipe run.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run

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. The **Insert Split** command divides a pipeline into sections. You can select the **Insert Split** command 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 by using **Insert Split** command:

- 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.

Steps for Inserting Weld Joint and Takedown Joint Splits in a Pipe Run:

Exercise 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 Figure 1.

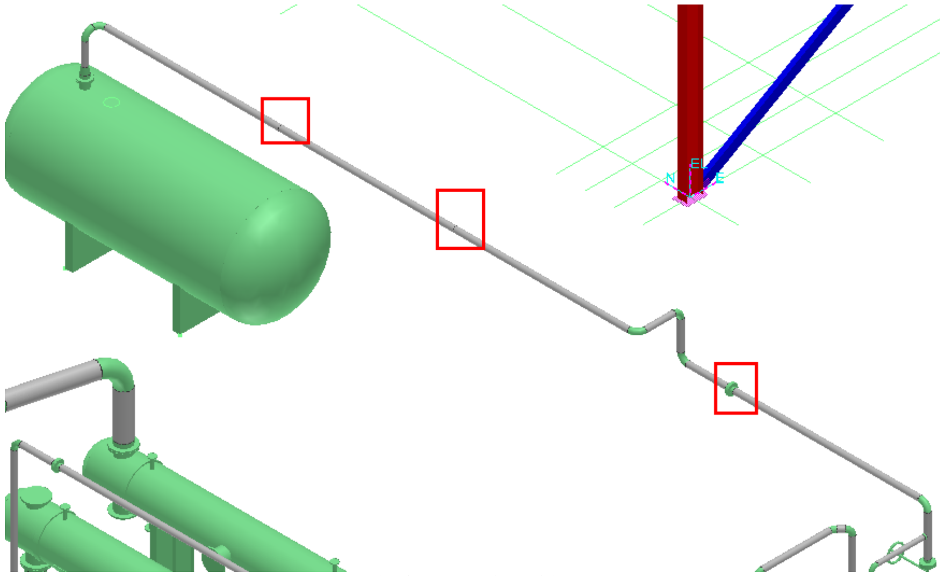


Figure 1: Takedown and Weld Joints in a Pipe Run

Before beginning the 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.**

29. Activate **PinPoint** ribbon from the **Tools>PinPoint** command.

30. Set the active coordinate system to **U04 CS** and select the **Set Target to Origin** option on the **PinPoint** ribbon.

31. Click the **Insert Split** button on the vertical toolbar to insert takedown joint.

Figure 2: Insert Split Button on the Vertical Toolbar

When you click the **Insert Split** command, the **Insert Split** ribbon appears to set the options for adding a break into a pipe run.

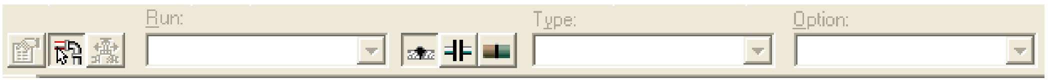


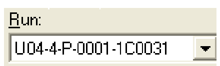
Figure 3: Insert Split Ribbon

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

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

- **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.

- **Weld Joint -** Defines split type to be a welded split.

- **Takedown Joint –** Defines split type to be a takedown joint.

- **Feature Break -**Defines split type to be a feature break.

- 

Type - Selects the type of split component to place. SP3D generates this list based on the components allowed by the pipe specification for the selected pipe.

- 

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.

- **Split Mode** – Selects single or multiple splits to be placed on a straight feature or pipe run.

32. Select the **Takedown Joint** option on the **Insert Split** ribbon to place takedown split in a pipe run.

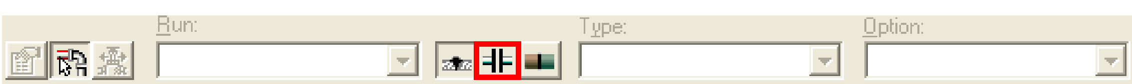


Figure 4: Takedown Joint Option on the Insert Split Ribbon

33. The system prompts to select either a Pipe Straight feature. Select the pipe run **U04-4-P-0203-1C0031**, to specify the pipe where you want to insert the takedown joint, as shown in Figure 5.

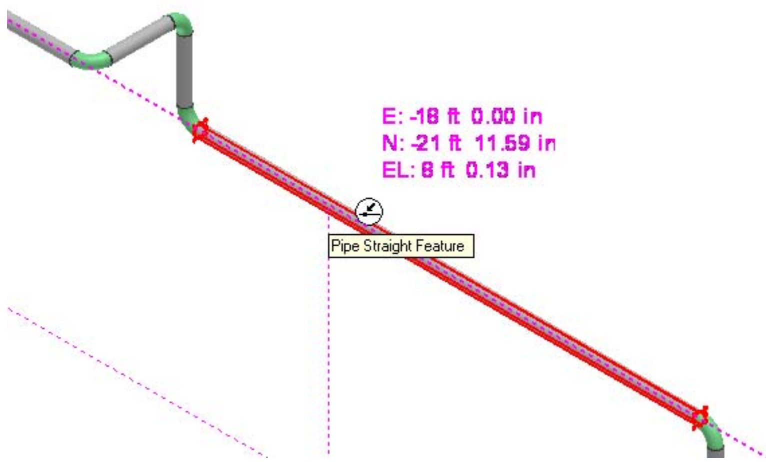


Figure 5: Pipe Straight Feature of 400-P Pipeline

34. Select the **Reposition Target** option on the **PinPoint** ribbon to change the target origin.

35. 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, as shown in Figure 6.

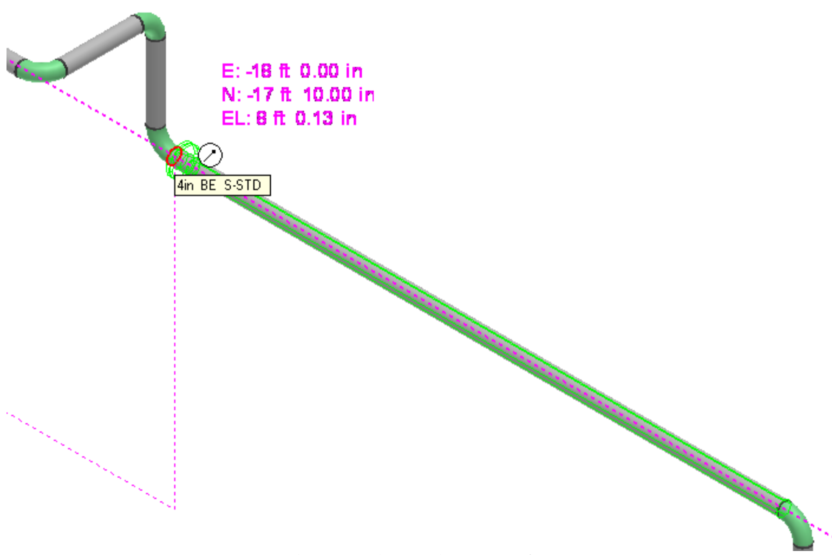


Figure 6: Selecting the End Point of a Pipe Run

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

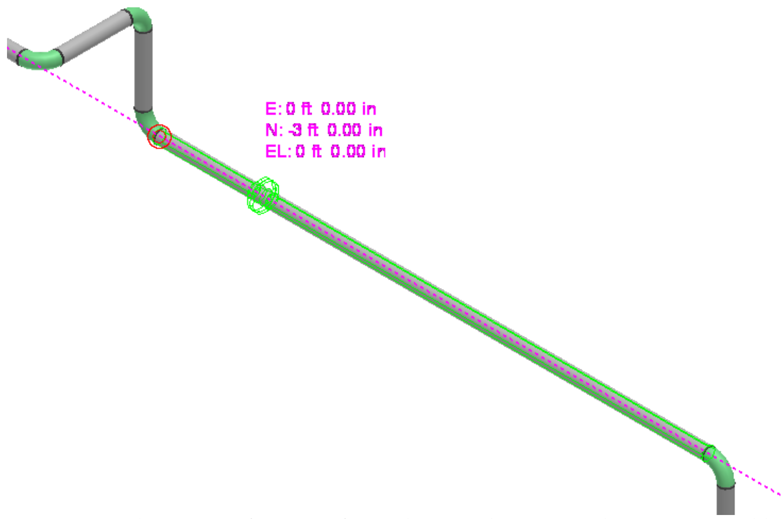


Figure 7: Graphic View After Specifying the Coordinates on the PinPoint Ribbon

37. Click on the graphic view to accept the position of a split on the pipe run. The highlighted portion in Figure 8 shows takedown joint (flange set) placed in a pipe run.

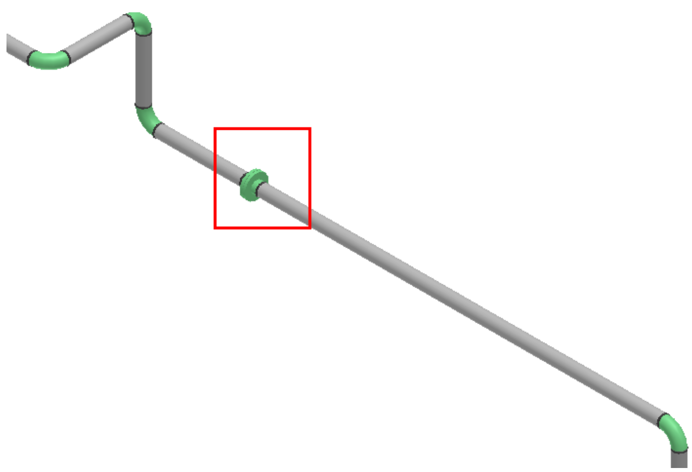


Figure 8: Placed Takedown Joint in a Pipe Run

38. Click the **Insert Split** button on the vertical toolbar to insert weld joint.

39. Select the **Weld Joint** option on the **Insert Split** ribbon to place welds in a pipe run.

Figure 9: Weld Joint Option on the Insert Split Ribbon

40. Select the **Pipe Straight Feature** of a pipe run to specify the pipe where you want to insert splits, as shown in Figure 10.

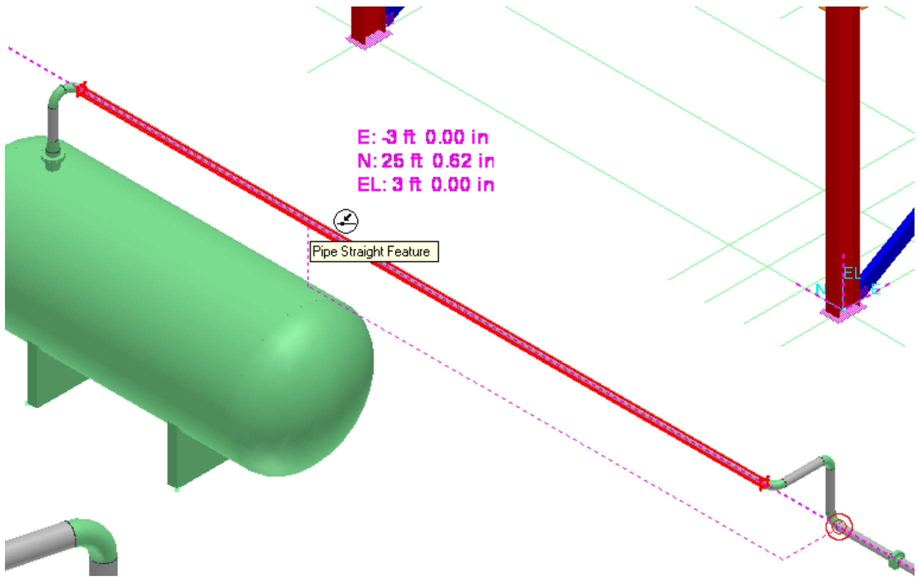


Figure 10: Pipe Straight Feature of 400-P Pipeline

41. 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**

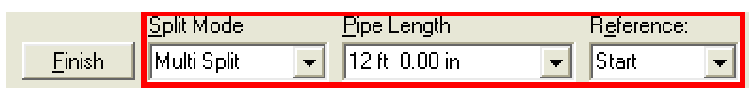


Figure 11: Weld Joint Options on the Insert Split Ribbon

SP3D places splits at every 12 ft distance from the start end of the pipe.

42. Click the **Finish** button on the **Insert Split** ribbon to accept the placement of weld joints.

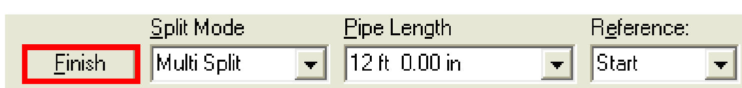


Figure 12: Finish Button on the Insert Split Ribbon

The highlighted section in figure 13 shows multiple weld joints in a pipe run.

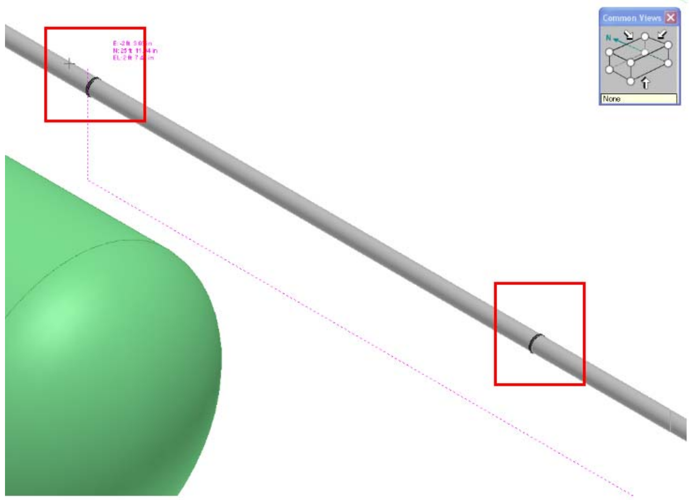


Figure 13: Weld Joints

Steps for Inserting Feature Break Split in a Pipe Run:

Exercise 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 Figure 14.

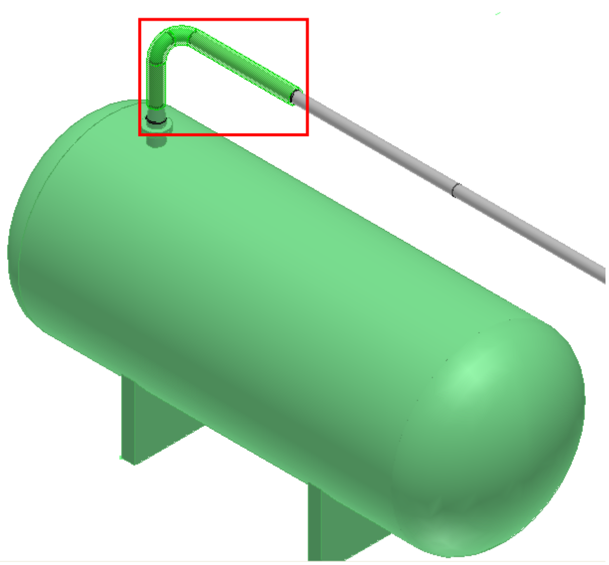


Figure 14: Insulated Pipe Run After Inserting Feature Break

1. Click the **Insert Split** button on the vertical toolbar to activate the **Insert Split** ribbon.

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

Figure 15: Feature Break Option on the Insert Split Ribbon

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 using the **Quick Pick** tool so that SP3D 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**.

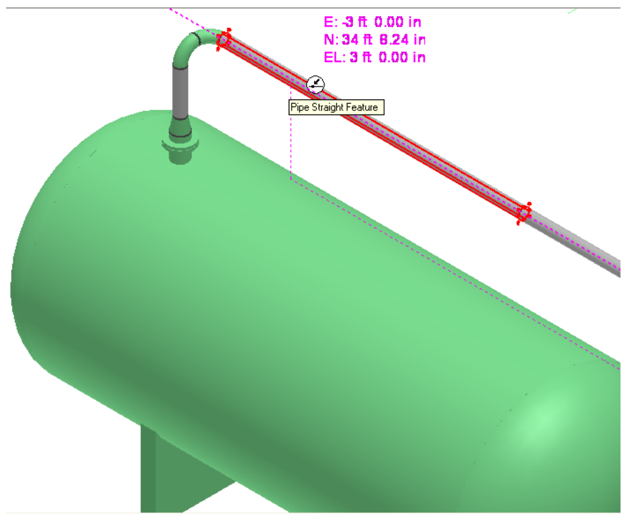


Figure 16: Pipe Straight Feature

4. Select the **Reposition Target** option on the **PinPoint** ribbon.

5. Select the end point of the **Pipe Straight Feature** of the pipeline **400-P** to specify the target origin.

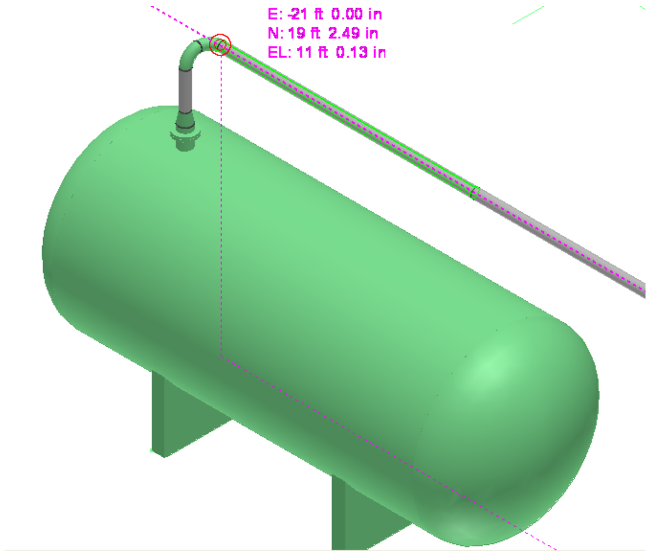


Figure 17: Repositioned Target

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

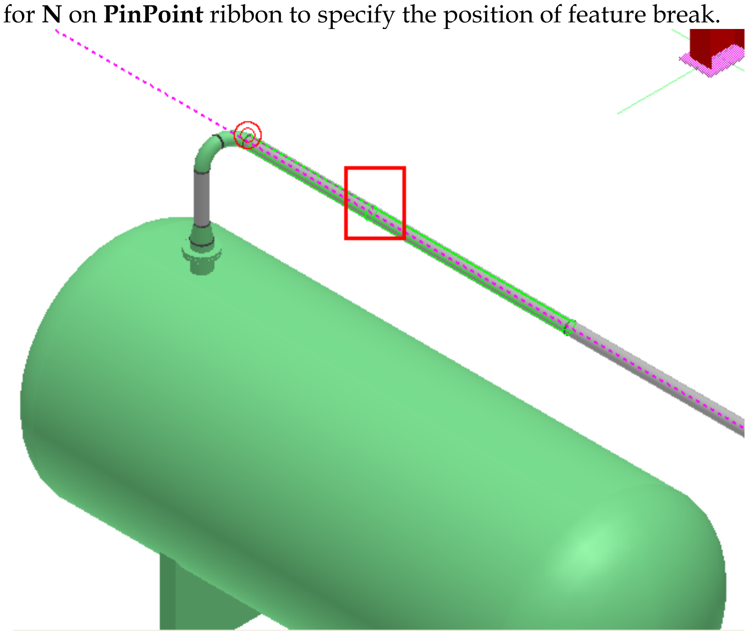


Figure 18: Position of Feature Break

7. Click on the graphic view to accept the placement of feature break. Placement of this split will not be visible in graphic view until you select the pipe run or the adjacent features.

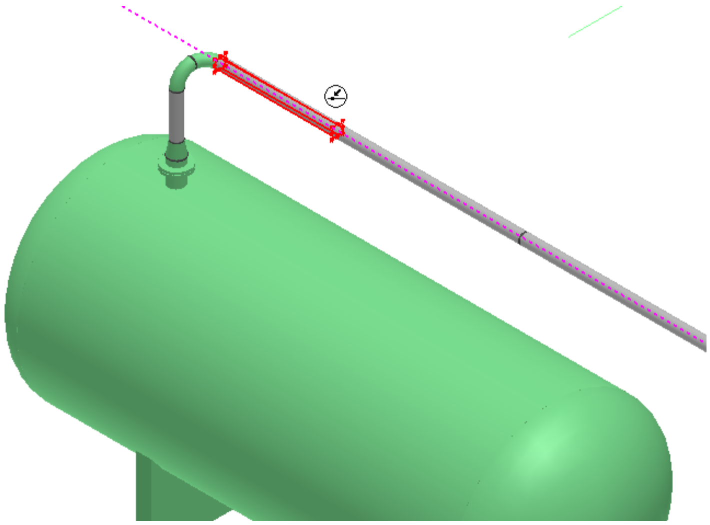


Figure 19: Feature Break Placed

8. Select all straight and turn features between the equipment nozzle and the feature break by holding the Shift key, as shown in Figure 20.

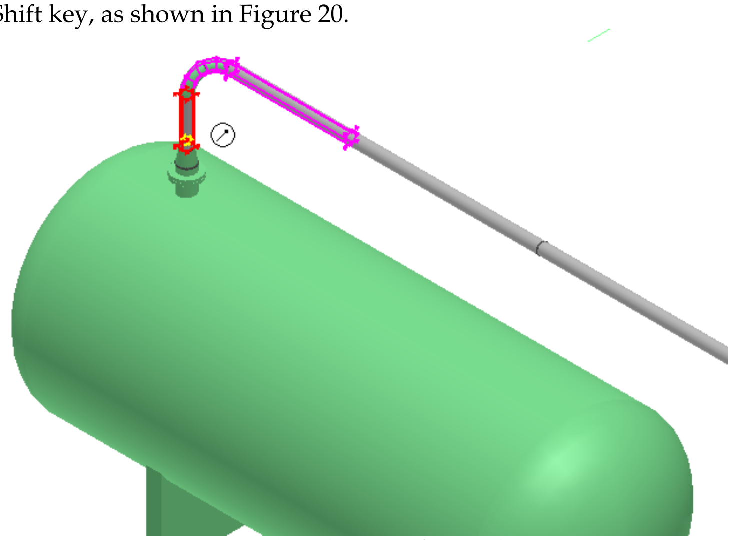


Figure 20: Selected Section for Insulation

9. The **Edit** ribbon will appear. Select the **New Pipe Run** option from the **Run** drop-down list on the **Edit** ribbon.

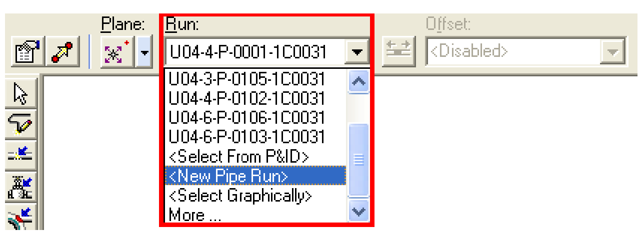


Figure 21: Run Drop-Down List on Edit Ribbon

10. The **New Pipe Run** dialog box will open. Select **400-P** for **Pipeline** and **4 in** for **Nominal Diameter** under the **Standard** category.

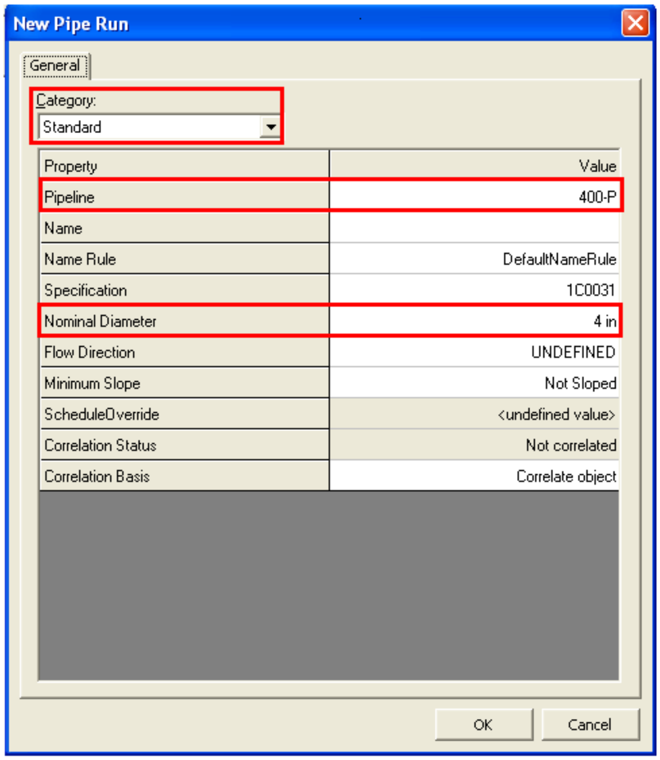


Figure 22: New Pipe Run Dialog Box

11. Switch to the **Insulation and Tracing** category. Select the **More…** option in the **Insulation Specification** drop-down list to specify the insulation type for the pipe run.

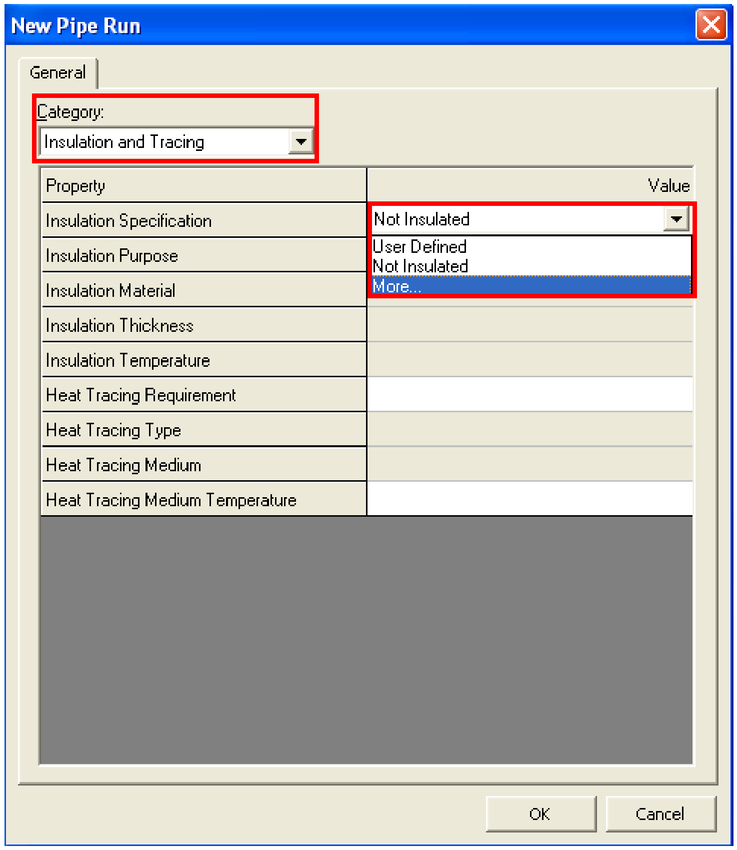


Figure 23: Insulation and Tracing Category in New Pipe Run Dialog Box

12. The **Insulation Specification** dialog box will appear to select the type of insulation for the pipe run. Select the **Fiberglas**s option and click **OK**.

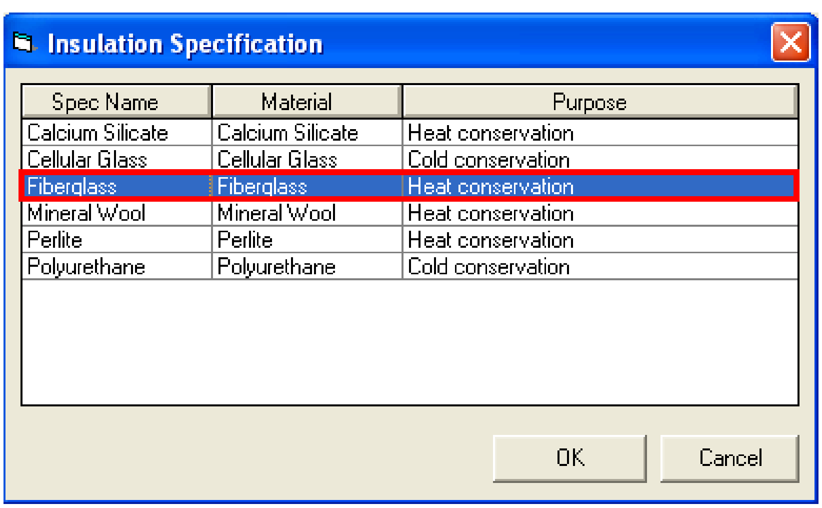


Figure 24: Insulation and Tracing Dialog Box

13. 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

    Then click **OK** on the **New Pipe Run** dialog box.

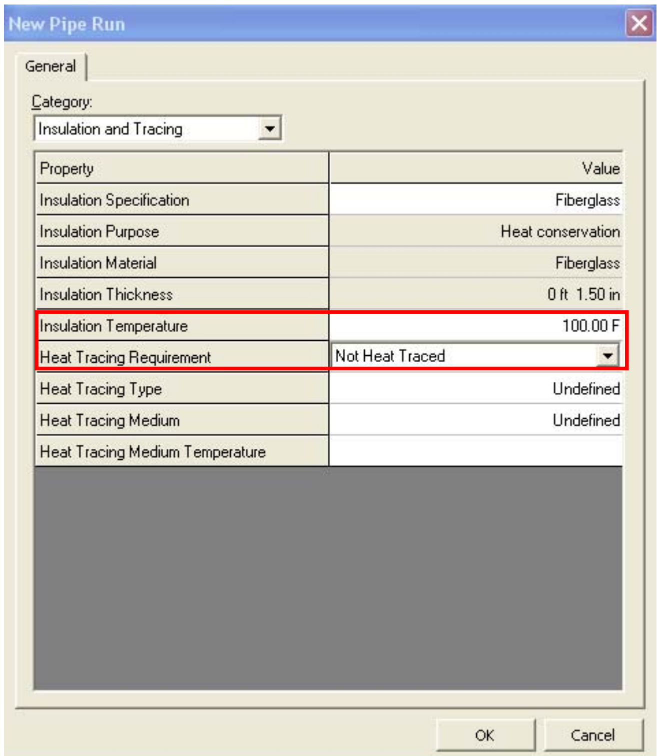


Figure 25: Insulation specifications

14. Insulation will be applied on the selected section of pipe run. To display the insulated section change the view of the section by the **Format>View** command. Select the **Insulation** and **Simple physical** in **Selected Aspects** in the **Format View** dialog box. Then click **OK** on the **Format View** dialog box.

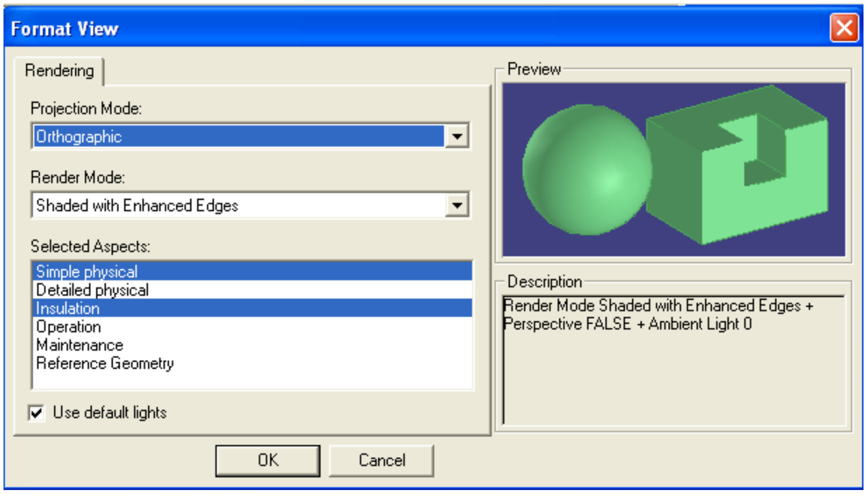


Figure 26: Format View Dialog Box

15. Now apply surface style rule on the insulated section by the **Format>Surface Style Rules** command. Select the **Piping Insulation – Delivered** rule and click **OK** to apply the rule to the insulated section.

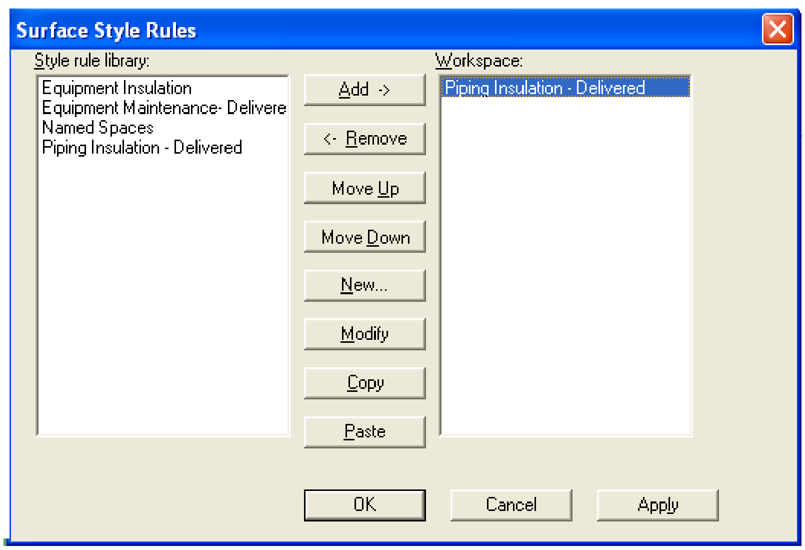


Figure 27: Surface Style Rule For Piping Insulation

For more information related to sequencing objects, refer to *Inserting Splits: An Overview,* topic in the user guide *PipingUsersGuide.pdf.*

Session 10: Manipulating Piping Objects

Objectives:

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

- Modify pipes, their features, and piping objects.
- Update Custom Instrument after Instrumentation Dimensional Datasheet (DDP) is changed in SmartPlant Instrumentation.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run

Overview:

SP3D 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 the **Select** button 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 SP3D. 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 SP3D after Instrumentation Dimensional Datasheet (DDP) is changed in SmartPlant Instrumentation.

Steps for Moving Pipe Straight Feature:

Exercise 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 Figure 1.

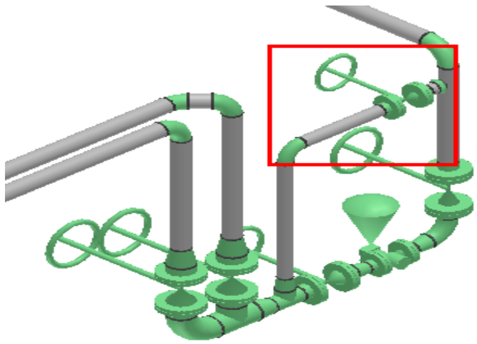


Figure 1: Moved Pipe Straight Feature of Pipeline 400-P

Before beginning the 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.**

43. Activate the **PinPoint** ribbon from the **Tools>PinPoint** command.

44. Select the **Piping Features** option in the **Locate Filter** drop-down list to select only piping features in the graphic view.

45. Select the **Reposition Target** option on the **PinPoint** ribbon to change the target origin.

46. Select the **Pipe Straight Feature** of the pipeline **400-P**, as shown in Figure 2, to specify the centerline of the pipe as origin.

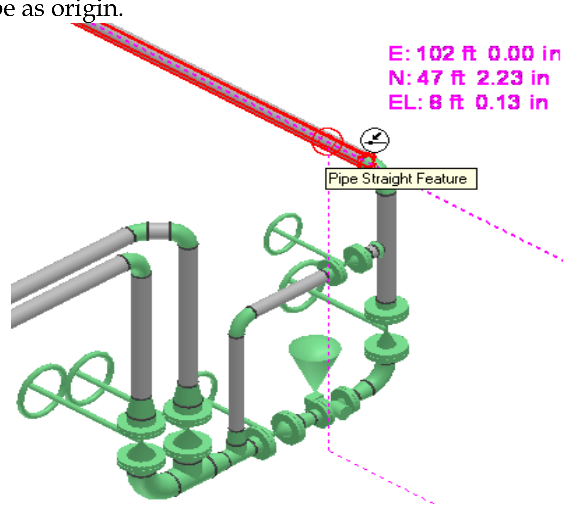


Figure 2: Repositioning the Target

47. Select the **Pipe Straight Feature**, as shown in Figure 3, to specify the piping feature that you will move with respect to **400-P**.

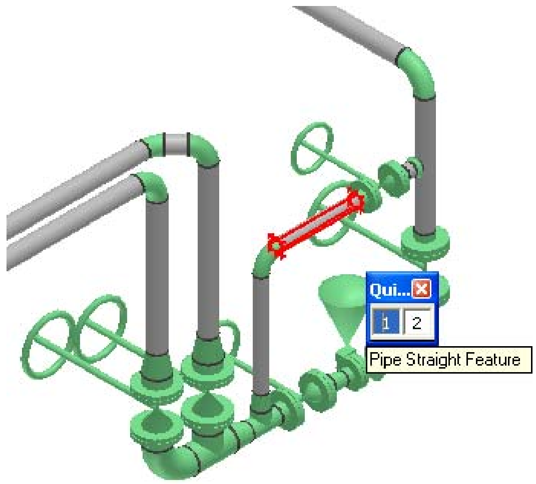


Figure 3: Pipe Straight Feature

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.

48. When you select the **Pipe Straight Feature**, an **Edit** ribbon appears. Select the **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.

Figure 4: Move From Option on the Edit Ribbon

49. Select the **Pipe Straight Feature** to be moved. The selected pipe appears with bordered outline in the graphic view, as shown in Figure 5.

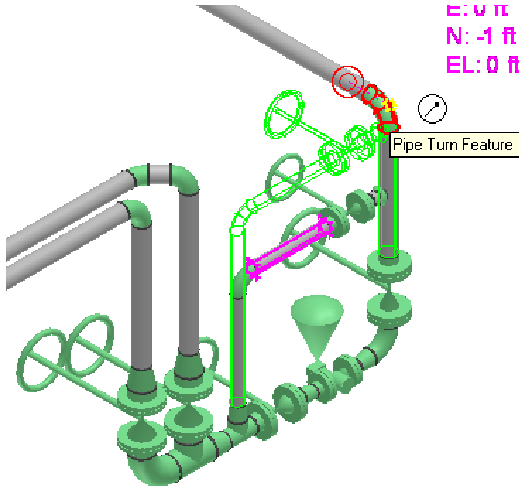


Figure 5: Pipe With Bordered Outline in the Graphic View

50. 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 Figure 6.

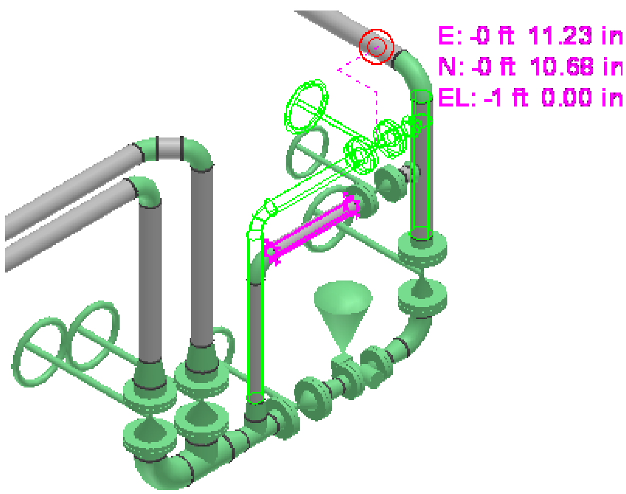


Figure 6: Moved Pipe Feature

51. Click in the graphic view to accept the position of the moved pipe feature, as shown in Figure 7.

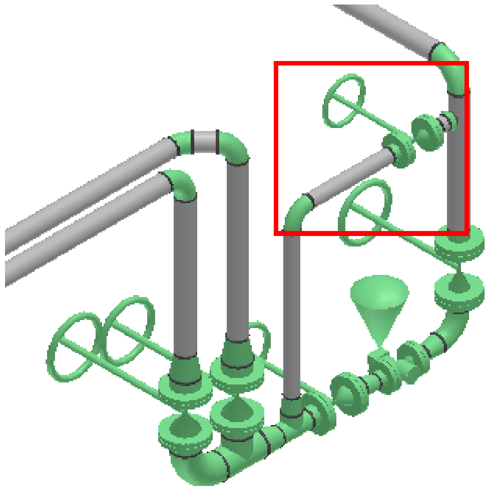


Figure 7: Pipe Straight Feature After Moving to a New Position

Steps for Moving Pipe End Feature:

Exercise 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 Figure 8.

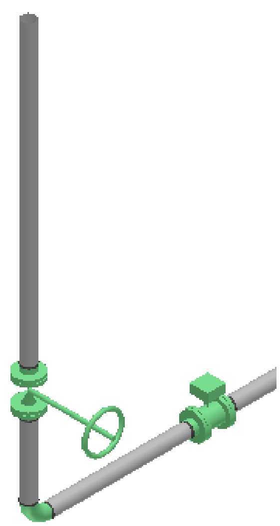


Figure 8: Moved Pipe End Feature

Before beginning the 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 in Figure 9. You can use the quick pick tool to help you locate the end feature of the pipe.

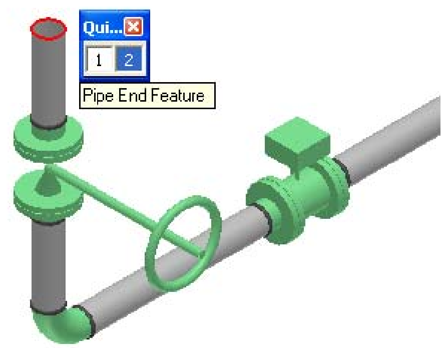


Figure 9: Pipe End Feature

The **Edit End Feature** ribbon appears. The end feature can be moved by using the smartstep options available on the **Edit End Feature** ribbon. You can use the **Length** drop-down list to type a new length for the pipe. SP3D moves the end feature to the specified length.

Figure 10: Edit End Feature Ribbon

3. Key in **6 ft** in the **Length** drop-down list on the **Edit End Feature** ribbon to extend the pipe.

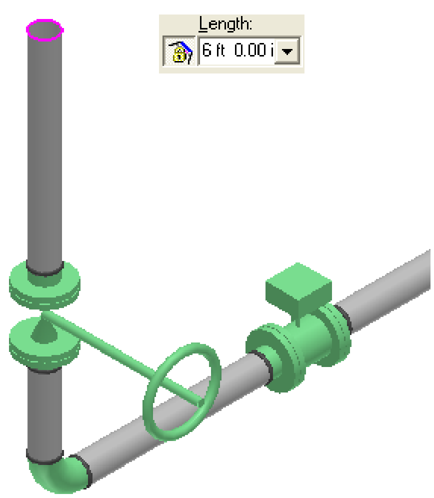


Figure 11: Moved Pipe End Feature

Steps for Moving Multiple Pipe Features:

Exercise 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 Figure 12.

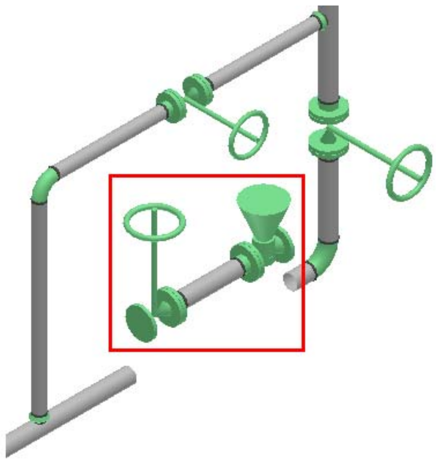


Figure 12: Contiguous Pipe Parts After Moving to a New Position

1. Select the features that generate the **FV-521** instrument, pipe and the gate valve located in line **P-204**, as shown in Figure 13. You can use Shift+Select method to help you select all the features.

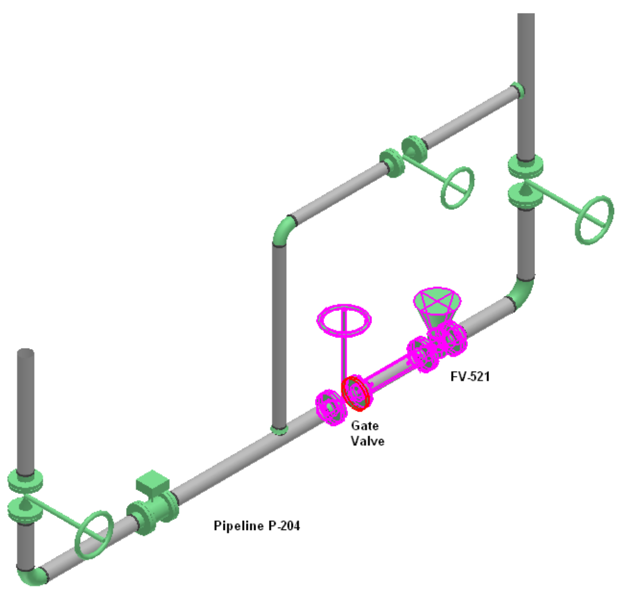


Figure 13: Selected Features to be Moved

The **Multi-Select** ribbon appears when the select set contains more than one feature.

Figure 14: Multi-Select Ribbon

2. Select the **Move** smartstep on the Multi-Select ribbon to start moving the pipe parts.

Figure 15: Move Smartstep on the Multi-Select Ribbon

3. Activate the **PinPoint** ribbon by clicking the **PinPoint** button on the **Common** toolbar and then select the **Relative Tracking** option on the **PinPoint** ribbon.

Figure 16: PinPoint Ribbon

4. Select the buttweld end port of the flange, as shown in Figure 17, to specify the origin for moving the pipe parts.

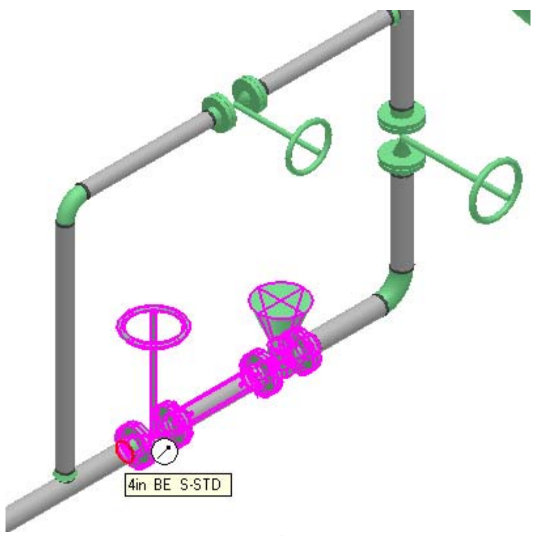


Figure 17: Origin of the Move Vector

5. On the **PinPoint** ribbon, key in **1ft** for easting **E** drop-down list. Move the cursor along the path and SP3D constraints your movement along the path, as shown in Figure 18.

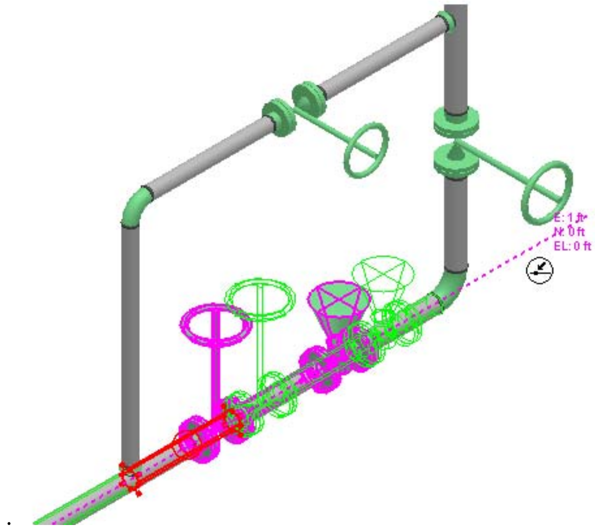


Figure 18: Move Constraint

6. Click in the graphic view to finish the move operation.

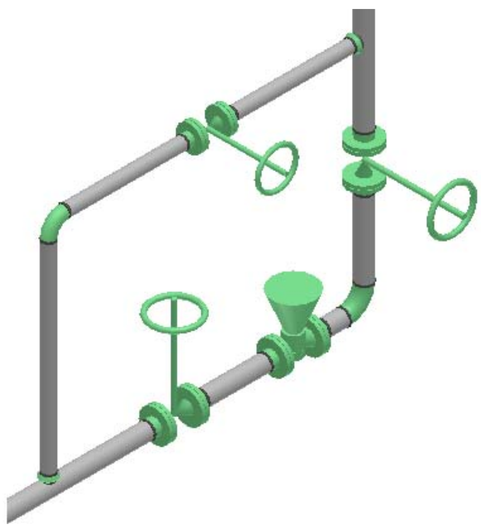


Figure 19: Selected Features After Moving to a New Position

The **Multi-Select** ribbon provides an option to move the selected contiguous features in any direction and create a break on the pipe run.

7. Select the three features again to open the **Multi-Select** ribbon and de-select the **Move Along Leg** option.

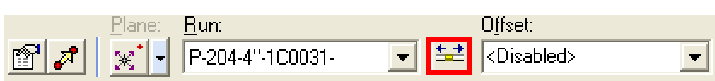


Figure 20: Move Along Leg Option on Multi-Select Ribbon

8. Select the **Move** smartstep on the **Multi-Select** ribbon.

9. Select the gate valve **port1** as the origin of the move vector, as shown in Figure 21.

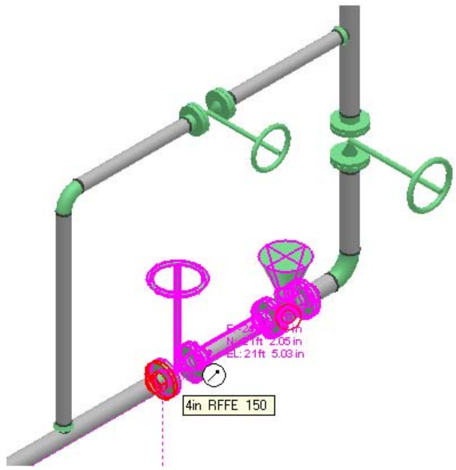
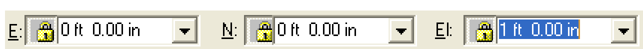


Figure 21: Origin of the Move Vector

10. Key in **0 ft** for **E**, **0 ft** for **N** and **1 ft** for **El** on the **PinPoint** ribbon to move the selected features **1ft** up. Before you accept the placement position, the selected features after moving should resemble Figure 22.



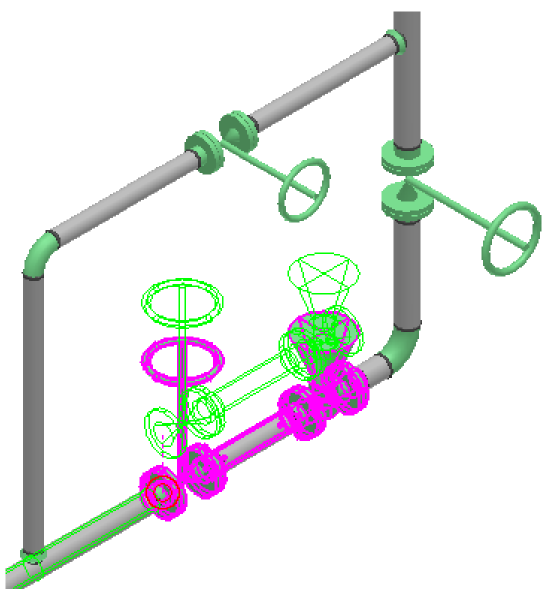


Figure 22: Selected Features Outline in the Graphic View

11. Click in the graphic view to accept the position of the moved features, as shown in Figure 23.

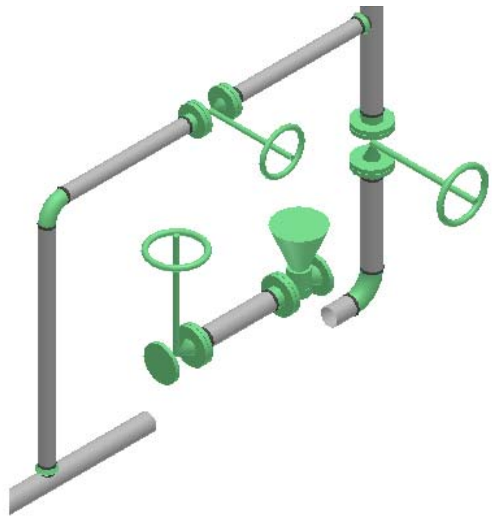


Figure 23: Selected Features After Moving to a New Position

12. Select the **Undo** command to undo the move operation.

Steps for Copying and Pasting a Pipe Run:

Exercise Objective: In this exercise you will be copying a pipe run in Pipeline **403-P** along with the equipments **40E-101A** and **40E-101B** to which the pipe run is connected. After moving these pipe runs the graphic view should resemble Figure 1.

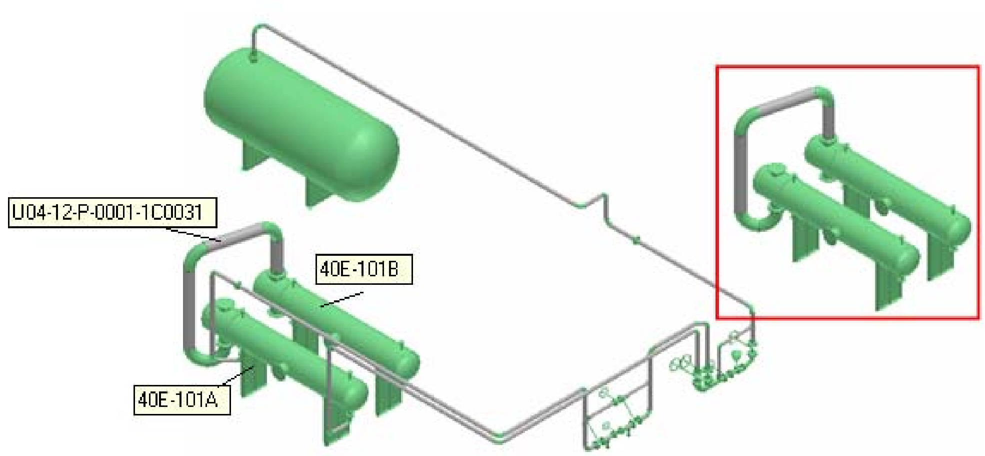


Figure 24: View of the Model After Copying and Pasting Pipe Run

Before beginning the 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 in Figure 25. This pipe run name is set to **User Defined** to demonstrate the behavior on object names after the Copy/Paste operations.

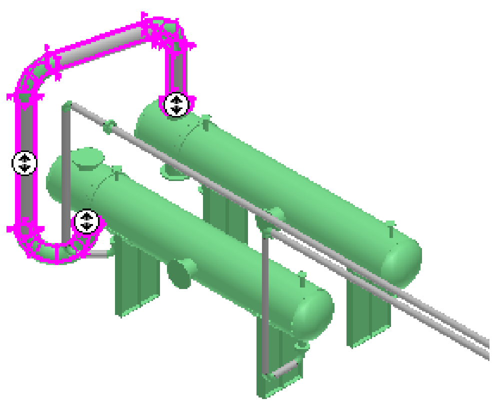


Figure 25: Pipe Run

3. Select both the equipments to which the pipe run is connected. Click the **Edit>Locate Filter** command to set the locate filter to equipment along with the pipe run.

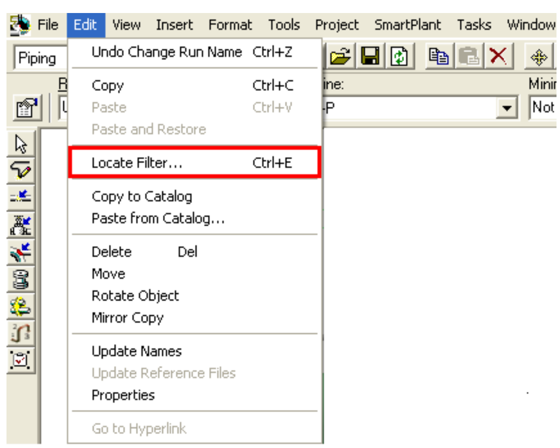


Figure 26: Edit> Locate Filter Command

4. The **Locate Filter** dialog box appears. Select the **Equipment** option in the **Select Locate Filter** drop-down list and click **OK**.

Figure 27: Equipment Option in the Locate Filter Dialog Box

5. While holding the CTRL key, select the equipments **40E-101A** and **40E-101B** along with the pipe run, as shown in Figure 28.

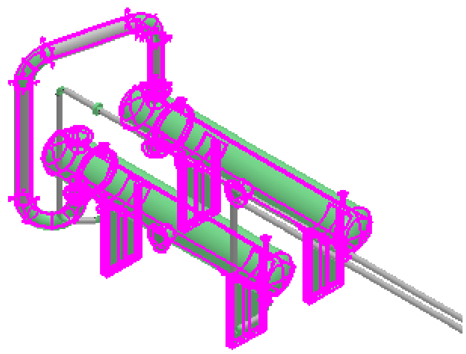


Figure 28: Selected Pipe Run and Equipments

6. Click the **Edit>Copy** command to copy the objects from the graphic view.

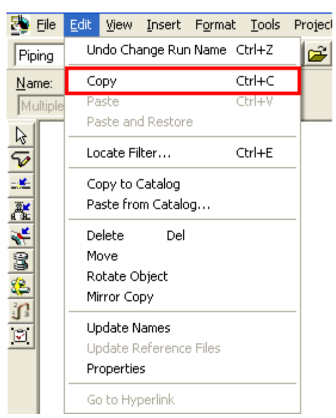


Figure 29: Edit> Copy Command

7. SP3D prompts you to select the reference point within the selected set of objects. Select the pipe nozzle **D** on equipment **40E-101A**, as shown in Figure 30.

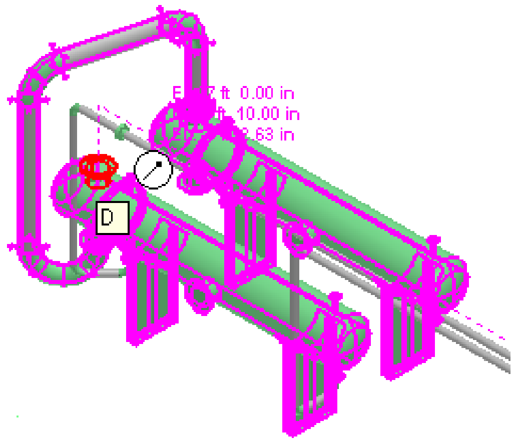


Figure 30: Reference Point for Pasting the Objects

8. Click the **Edit>Paste** command to paste the objects.

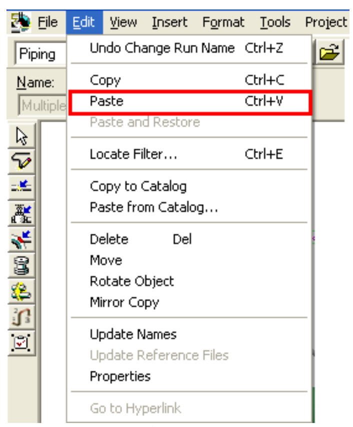


Figure 31: Edit> Paste Command

9. 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.

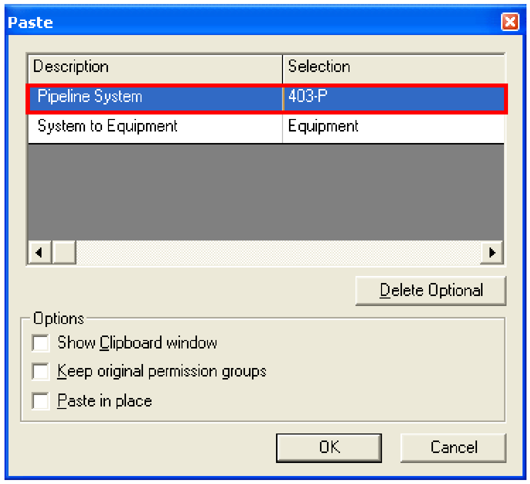


Figure 32: Piping System in the Paste Dialog Box

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 in Figure 33. Similarly the parent system for copied equipment can also be changed from the **Workspace Explorer.**

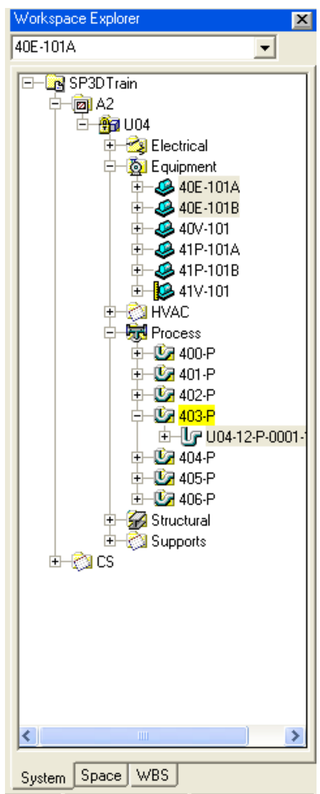


Figure 33: Piping Parent System Highlighted in Workspace Explorer

Keep the parent system for piping as **403-P** and equipment system for equipments **40E-101A** and **40E-101B** from where they have been copied. 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 in Figure 34.

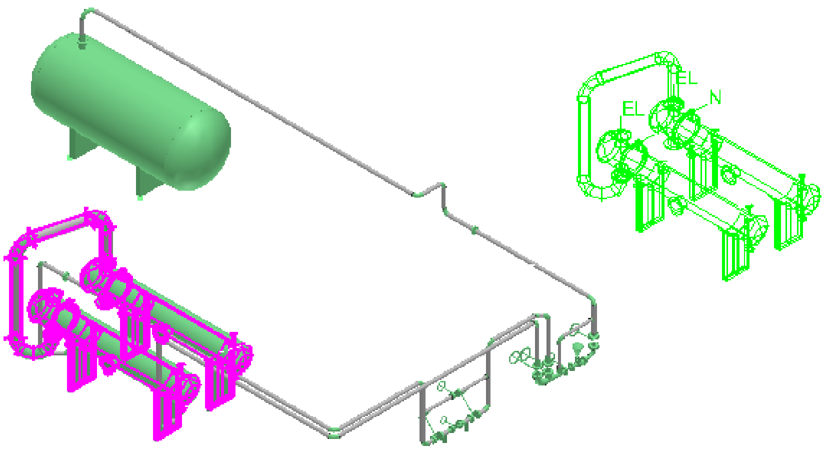


Figure 34: Copied Objects in the Graphic View

10. Click in the graphic view to paste the objects as shown in Figure 35.

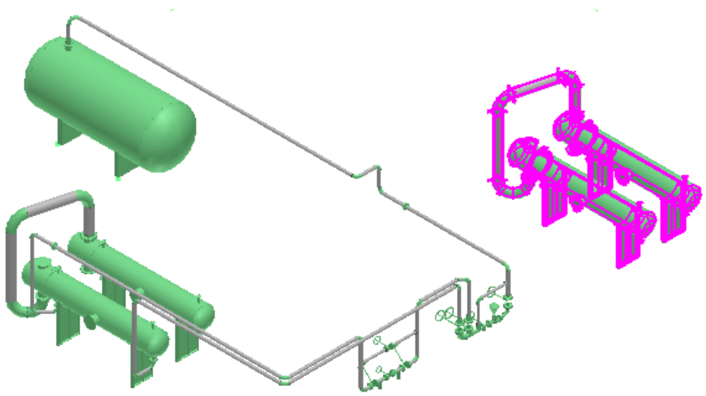


Figure 35: Objects After Pasting on the Graphic View

The pasted objects also appear in the **Workspace Explorer**, as shown in the highlighted sections of Figure 36.

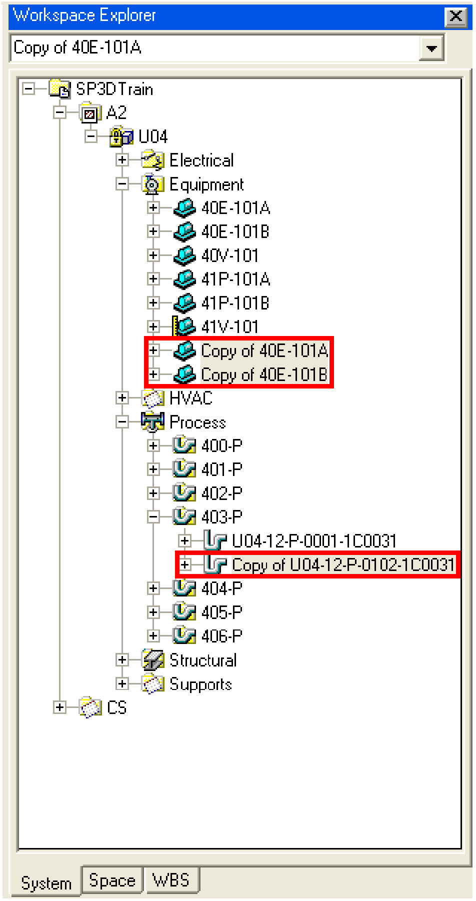


Figure 36: Pasted Objects in Workspace Explorer

Steps for Modifying the Properties of a Pipe Run:

Exercise Objective: In this exercise you will be applying the default name rule of the pipe run that was previously copied *(refer to previous lab)***.** Remember, the Pipe run name was intentionally set to User Defined in order to see the system behavior during the COPY/PASTE operation. After applying the name rule of the object, the name of the pipe run will change in the **Workspace Explorer** highlighted in Figure 37.

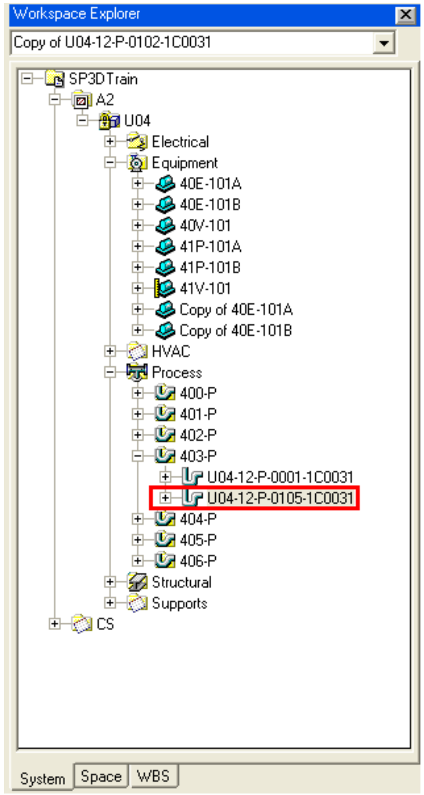


Figure 37: Modified Pipe Run Name

1. Select the **Piping Runs** option in the **Locate Filter** drop down list to select only pipe runs from the graphic view.

2. Select the pipe run **Copy of U04-12-P-xxxx-1C0031** *(copied run created in previous lab)* in the graphic view, as shown in Figure 38.

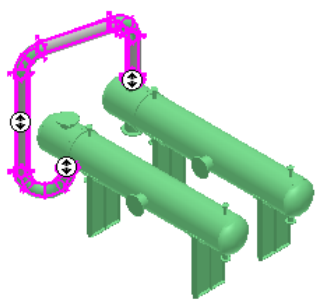


Figure 38: Pipe Run for Modification

3. Right-click the selected pipe run to access the **Pipe Run Properties** dialog box.

4. Select the **DefaultNameRule** option in the **Name Rule** drop-down list on the **Pipe Run Properties** dialog box, as shown in Figure 39.

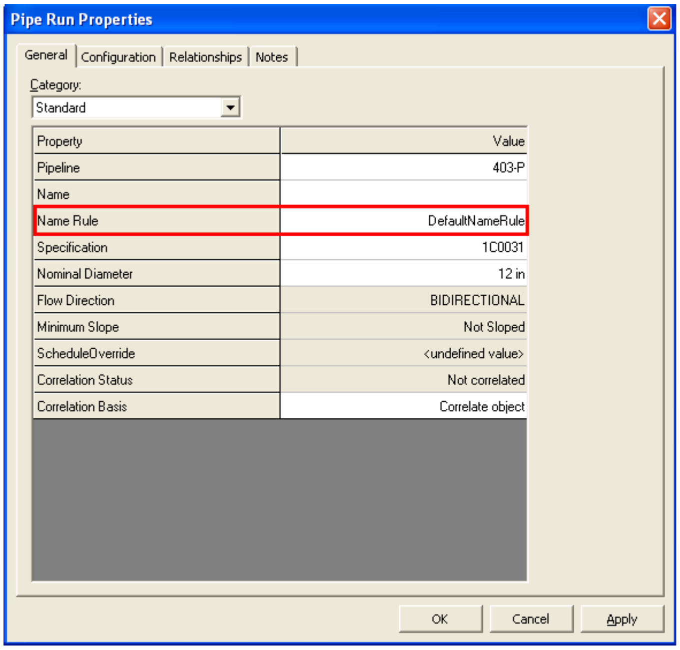


Figure 39: Pipe Run Properties Dialog Box

5. Click **OK** to apply the modified name rule on the pipe run.

Steps for Mirror Copying Pipe Runs/Equipments:

Exercise 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 equipments should resemble Figure 40.

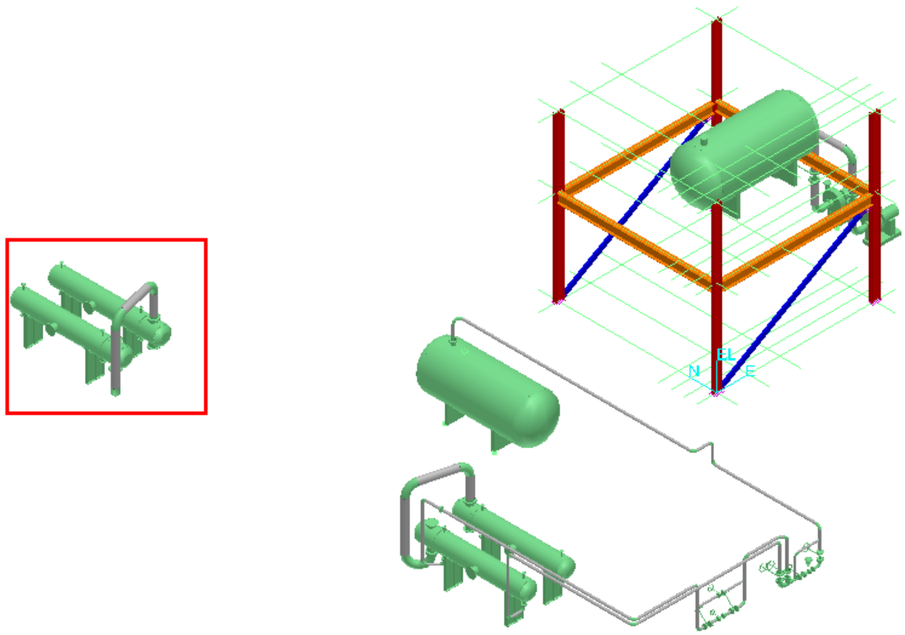


Figure 40: Mirror Copied Pipe Runs and Equipment

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 in Figure 41.

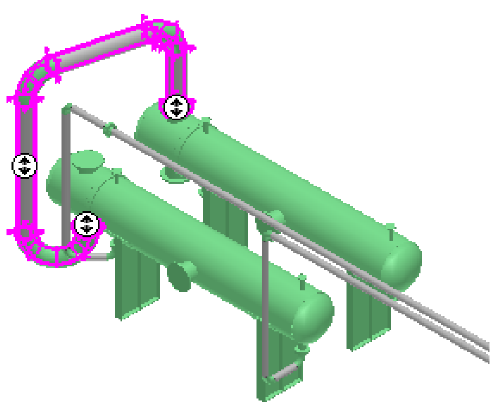


Figure 41: Selected Pipe Run

3. Select both the equipments **40E-101A** and **40E-101B** to which the pipe run is connected. Click the **Edit>Locate Filter** command to set the locate filter to equipment along with the pipe run.

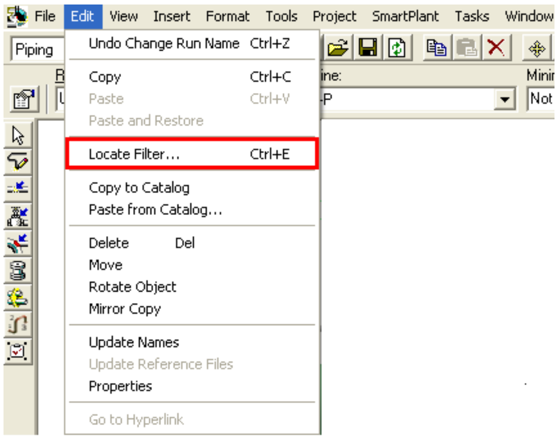


Figure 42: Edit> Locate Filter Command

4. The **Locate Filter** dialog box appears. Select the **Equipment** option from the **Select Locate Filter** drop-down list and click **OK**.

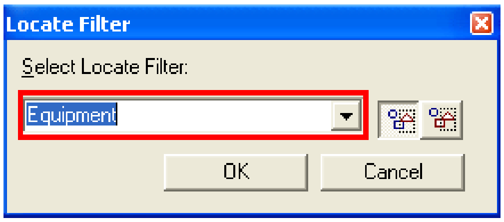


Figure 43: Equipment Option in the Locate Filter Dialog Box

5. While holding the CTRL key, select the equipments **40E-101A** and **40E-101B** along with the pipe run, as shown in Figure 44.

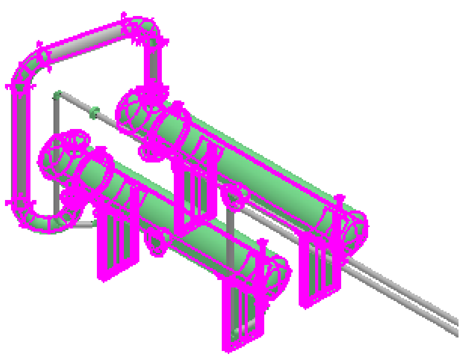


Figure 44: Selected Pipe Run and Equipments

6. Click the **Edit> Mirror Copy** command to mirror copy the selected objects from the graphic view.

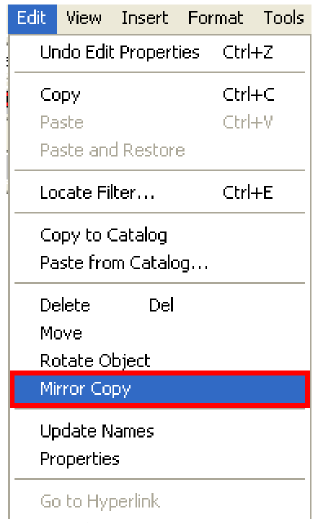


Figure 45: Edit> Mirror Copy Command

7. The **Mirror Copy** ribbon appears. 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**.

Figure 46: Point to Mirror About Destination Mode

8. Select the end of the column as the **Point to Mirror About**, as shown in Figure 47.

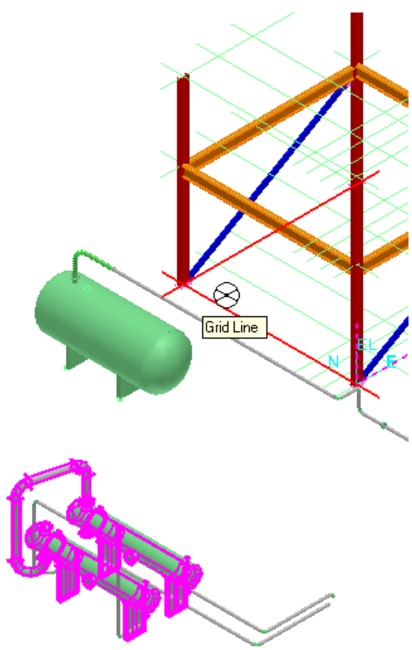


Figure 47: Point to Mirror About

9. 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.

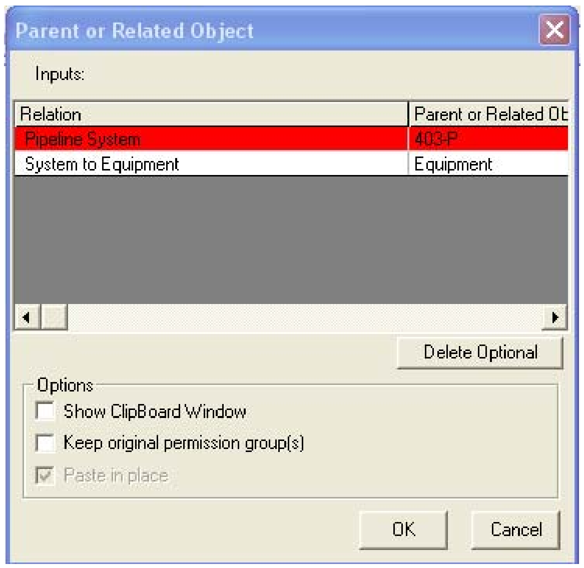


Figure 48: Parent or Related Object Dialog Box

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 in Figure 49. Similarly the parent system for copied equipment can be changed from the **Workspace Explorer.**

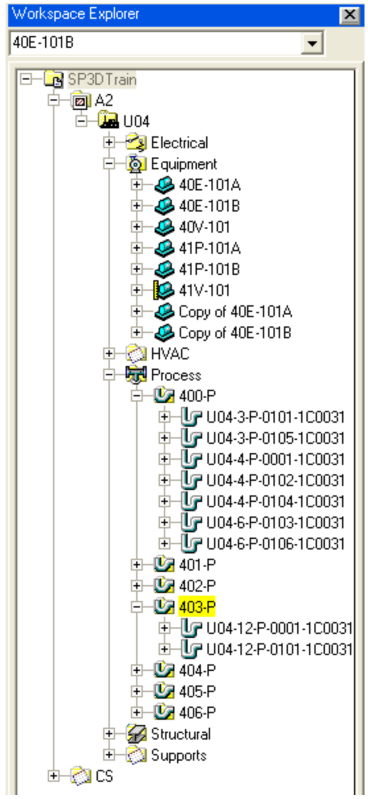


Figure 49: Piping Parent System Highlighted in Workspace Explorer

10. Keep the parent system for piping as **403-P** and equipment system for the equipments **40E-101A** and **40E-101B** from where they have been copied and click **OK**.

11. The mirrored objects appear with green bordered outline in the graphic view, as shown in Figure 50.

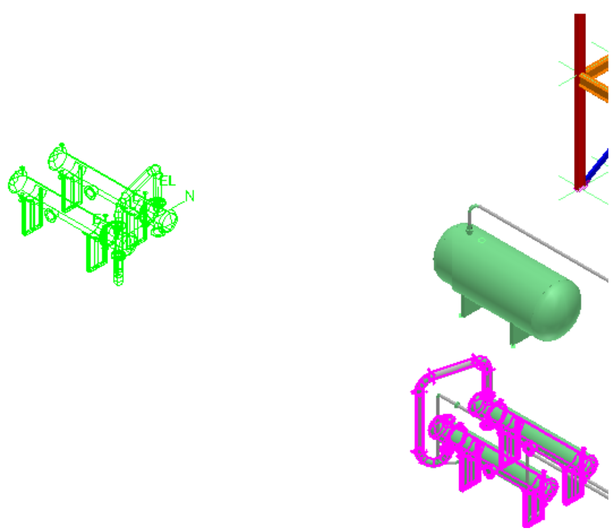


Figure 50: Mirrored Objects in the Graphic View

12. Click the **Finish** button on the **Mirror Copy** ribbon. The mirrored objects will appear in the graphic view, as shown in Figure 51 and 52.

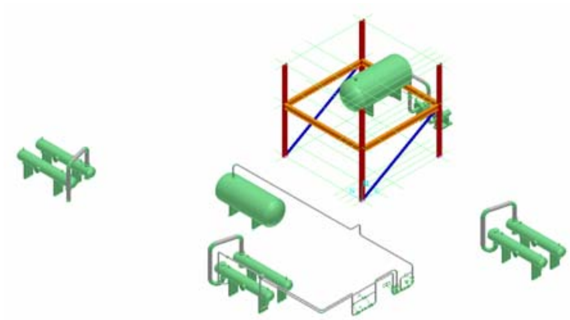


Figure 51: Isometric View

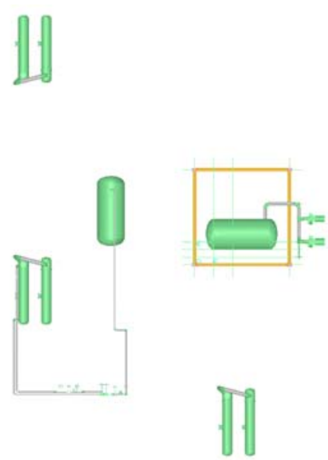


Figure 52: Plan View

Steps for Rotating Pipe Runs/Equipments:

Exercise 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 Figure 53.

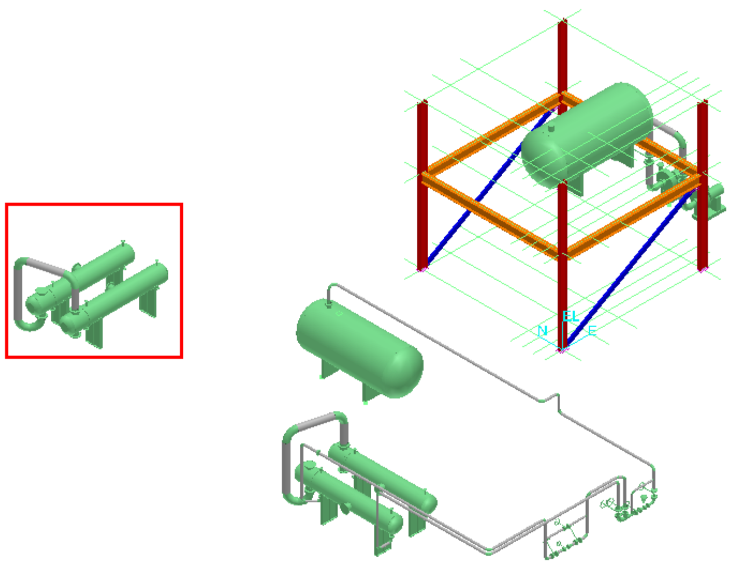


Figure 53: Isometric view

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 mirrored pipe run in the pipeline **403-P**, as shown in Figure 54.

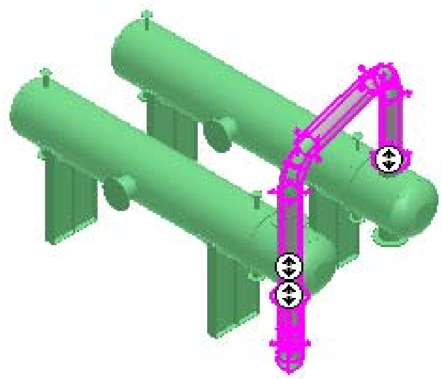


Figure 54: Mirrored Pipe Run

3. Select both the equipments to which the pipe run is connected.

4. Click the **Edit>Locate Filter** command to set the locate filter to equipment along with pipe run.

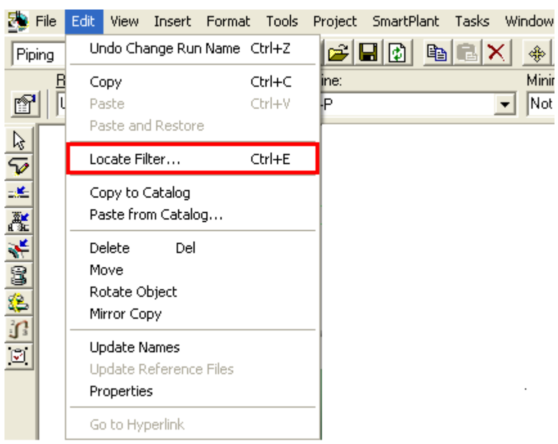


Figure 55: Edit> Locate Filter Command

5. The **Locate Filter** dialog box appears. Select the **Equipment** option in the **Select Locate Filter** drop-down list and click **OK**.

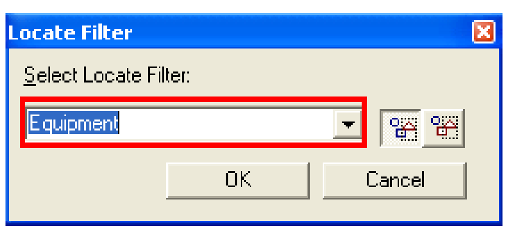


Figure 56: Equipment Option in the Locate Filter Dialog Box

6. While holding the CTRL key, select the equipments **Copy of 40E-101A** and **Copy of 40E-101B** along with the mirrored pipe run, as shown in Figure 57.

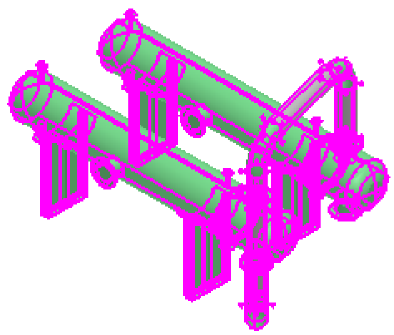


Figure 57: Selected Pipe Run and Equipments

7. Click the **Edit>Rotate Object** command to rotate the selected objects in the graphic view.

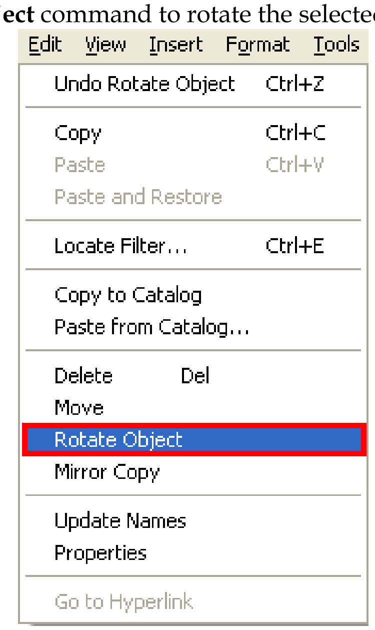
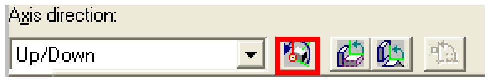


Figure 58: Edit> Rotate Object Command

8. The **Rotate Object** ribbon appears. In this ribbon define the axis direction and the origin point for the rotation. Select the **Up/Down** option in the **Axis Direction** drop-down list.

Figure 59: Rotate Object Ribbon

9. Select the **Axis Position Point** option on the **Rotate Object** ribbon and select the midpoint of the pipe run as the origin point for the axis of rotation.



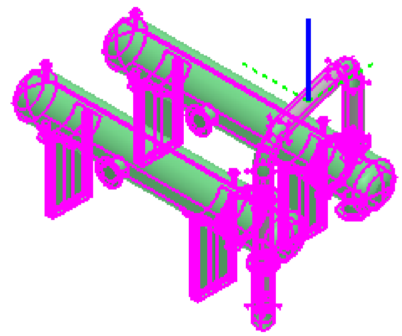


Figure 60: Origin Point for the Axis of Rotation

10. Key in **90 deg** in the **Angle** drop-down list on the **Rotate Object** ribbon. The selected objects will rotate in the graphic view, as shown in Figure 61.

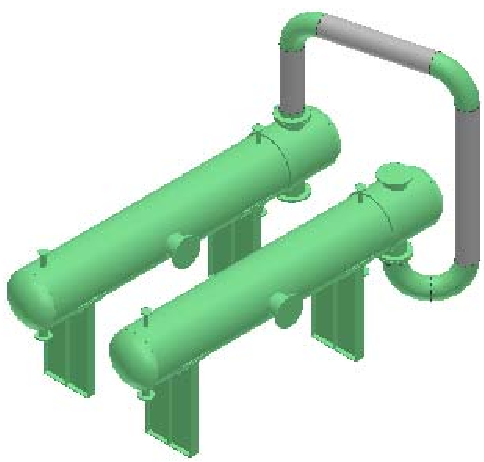


Figure 61: Selected Objects After Rotation

Steps for Deleting the Pipelines:

Exercise 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** button 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**, as shown in Figure 62.

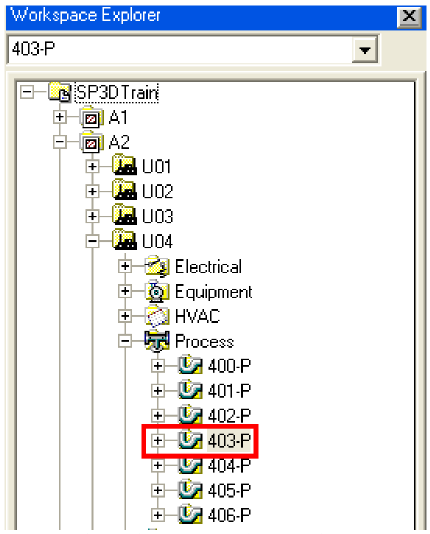


Figure 62: Selected 403-P in the WorkSpace Explorer

5. Click the **Delete** command.

6. Select Undo command. 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.

Steps for Deleting Pipe Straight Feature:

Exercise 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 Figure 63.

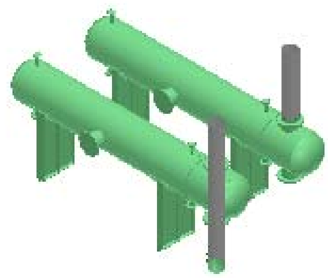


Figure 63: Deleted Pipe Straight Feature

1. Select the **Piping Features** option in the **Locate Filter** drop-down list.

2. Select the **Pipe Straight Feature** of pipeline **403-P**.

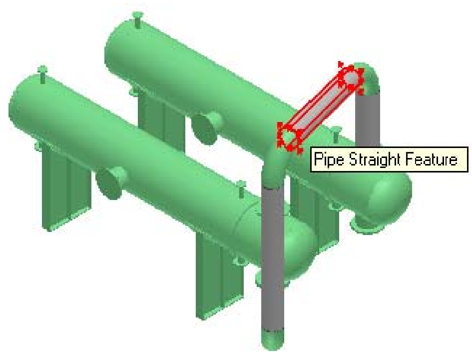


Figure 64: Pipe Straight Feature

3. Click the **Delete** button on the **Common** toolbar to delete the **Pipe Straight Feature**.

Figure 65: Delete Option on the Common Toolbar

Deleting a pipe straight feature does not remove connected features. The pipe straight feature is not connected by the associated other existing features. If the straight feature is connected to the third port of a tee-type branching (making the straight feature the defining feature for the branch point), deleting it will result in the tee type branch part remaining constant and the owning branch feature.

Deleting Pipe Run:

Exercise Objective: In this exercise you will be deleting the pipe runs of the mirrored/pasted pipeline **403-P** in Unit **U04** of your workspace. The view of the model after deleting the pipe runs will resemble Figure 66.

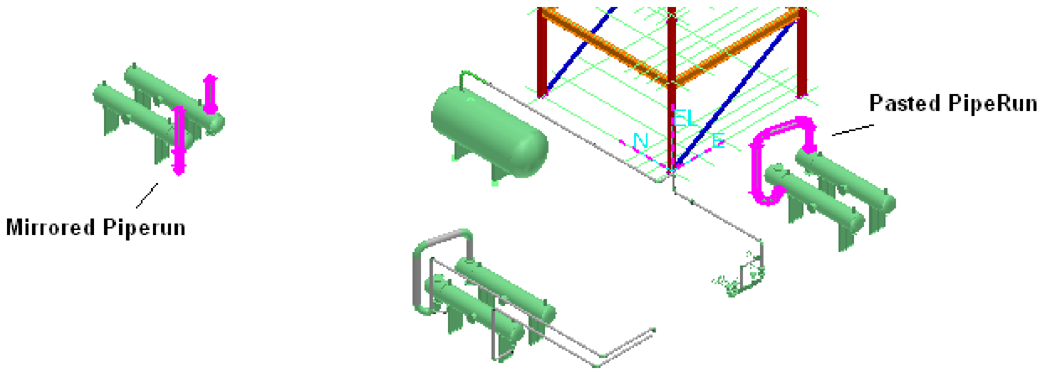


Figure 66: Deleted Pipe Run

Steps:

1. Select the **Piping Runs** option in the **Locate Filter** drop-down list.
2. Select **Mirrored/ Pasted Pipe Runs** of pipeline **403-P**.
3. Click the **Delete** button on the **Common** toolbar to delete the selected pipe runs.

Steps for Changing Mating Part to Base Part

Exercise 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 beginning the 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 in Figure 67.

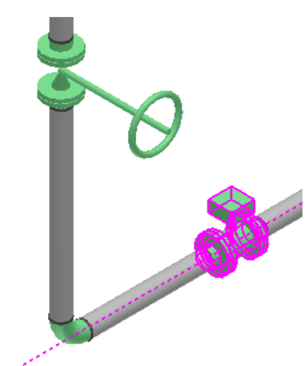


Figure 67: Mating Part

3. Select the **Delete** command. See how both the flanges and the instrument get deleted.

4. Select the **Undo** command.

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**.

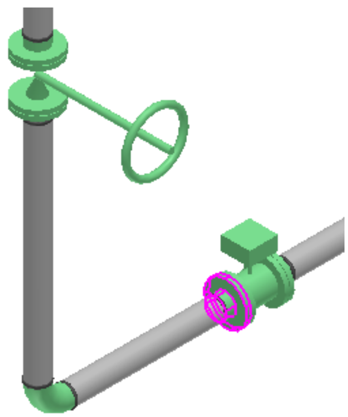


Figure 68: Mating Part

The **Edit Pipe Part** ribbon bar appears, as shown in Figure 69.

Figure 69: Edit Pipe Part Ribbon

7. The **Base/Mating Part** drop-down list displays the selected object, which has the mating part. Select the **Base Part** option in the drop-down list.

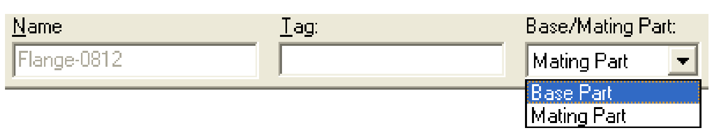


Figure 70: Base/Mating Part Drop-Down List in the Edit Pipe Part Ribbon

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 the **Delete** command. See how the instrument gets deleted, leaving the flanges.

Steps to Update Custom Instrument after the Dimensional Datasheet (DDP) is Changed in SmartPlant Instrumentation:

Exercise Objective: In this exercise you will be retrieving the Updated Dimensional Datasheet into the model to update the custom instrument **FV-311 DDP** on the pipeline **300-W** in Unit **U03** of your workspace by using the **Retrieve** command. After updating the custom instrument the view of the model should resemble Figure 71.

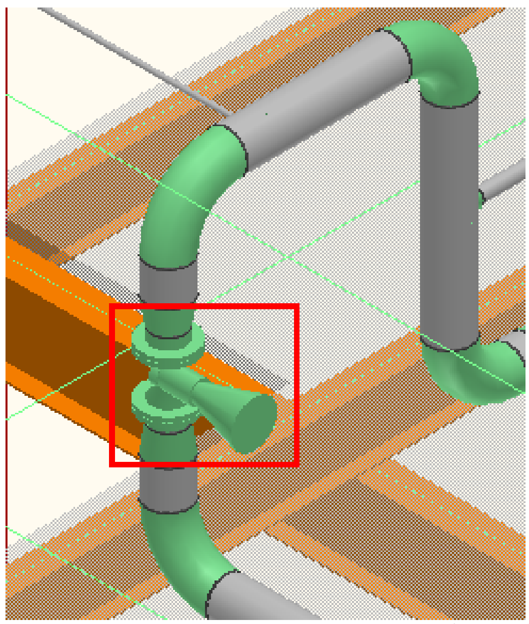


Figure 71: Custom Instrument FV-311 DDP After Updation

Before beginning the procedure:

- Define your workspace to display Unit **U03** and the coordinate system **U03 CS**.

1. Select the **SmartPlant > Retrieve** command to open a **Retrieve** dialog box that assists you in retrieving the Updated Dimensional Datasheet.

2. Select the **FV-311 DDP** document in the **Retrieve** dialog box and click **OK**.

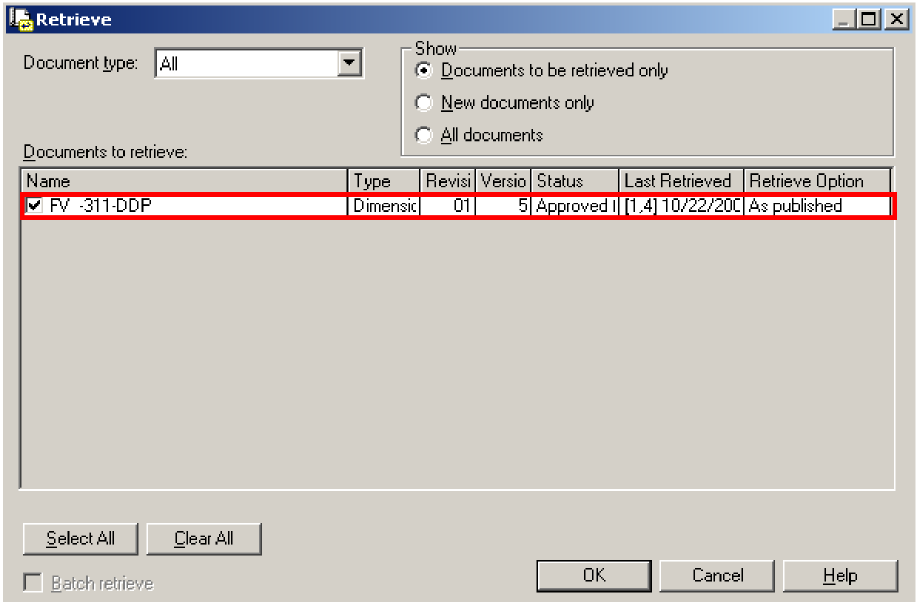


Figure 72: FV-311 DDP Document in Retrieve Dialog Box

3. In the **Locate Filter** list, select the **Pipe Features** option. This helps you select only pipe features in the model.

4. In the graphic view, select the **FV-311** custom instrument located in the pipeline **300-W**, as shown in Figure 73.

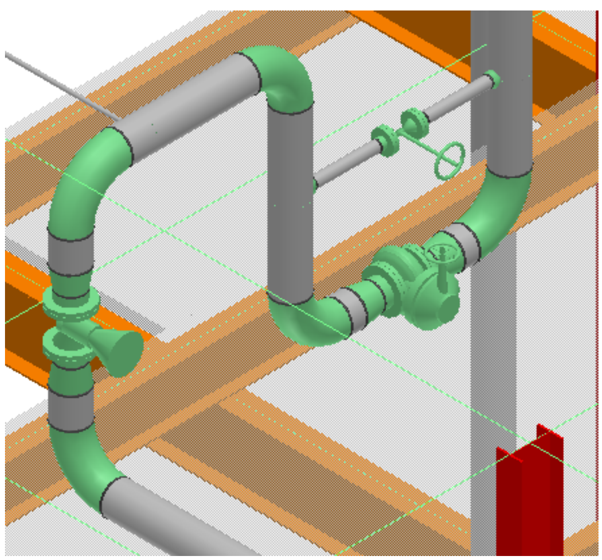


Figure 73: FV-311 Custom Instrument

5. Click the **SmartPlant > Compare Design Basis** command to compare the instrument you have selected with the design basis.

6. The **Compare with Design Basis** dialog box appears. The **Face to Face** row is highlighted in red, as shown in Figure 74. This indicates that the **Face to Face** dimension property of the instrument does not match the design basis.

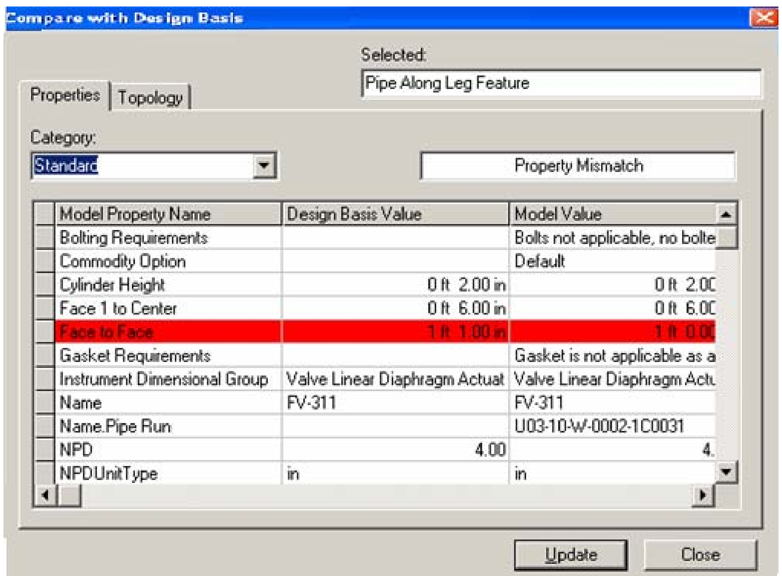


Figure 74: FV-311 Compare with Design Basis Dialog Box

7. Select the **Instrument Actuator** option in the **Category** drop-down list in the **Compare with Design Basis** dialog box. The **Actuator Height** row is highlighted in red, as shown in Figure 75. This indicates that the actuator dimension property of the instrument does not match the design basis.

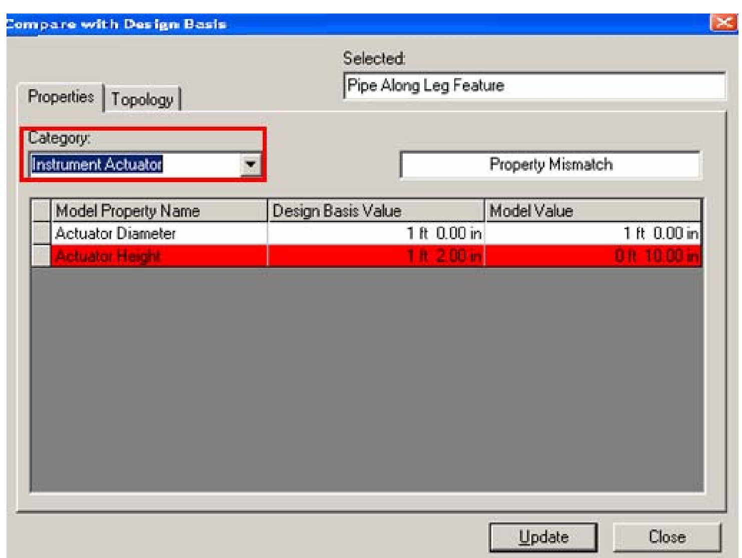


Figure 75: FV-311 Compare with Design Basis Dialog Box

8. Click **Update** in the **Compare with Design Basis** dialog box to update the dimension

properties as per the design basis. The **Face to Face** and **Actuator Height** rows now become white, as shown in Figure 76 and 77, indicating that the dimensions on the modeled object now matches the design basis.

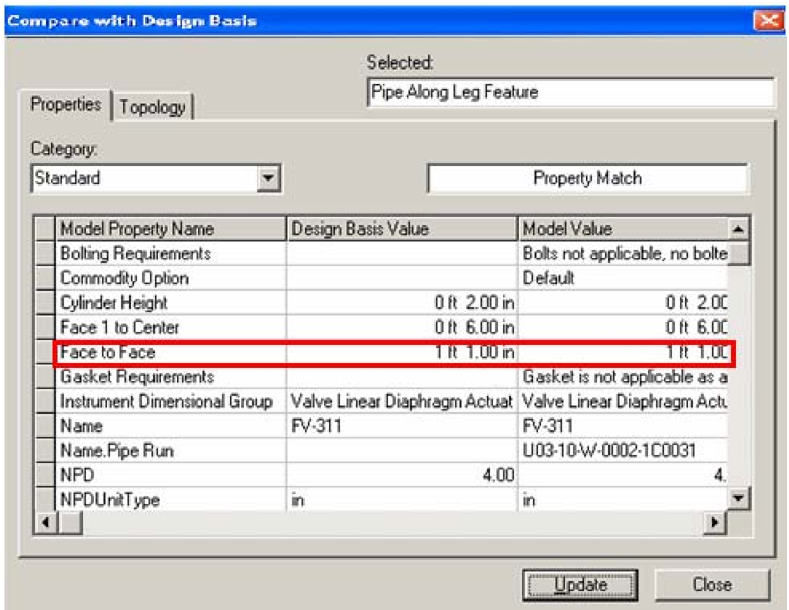


Figure 76: Standard Category of FV-311 Compare with Design Basis Dialog Box

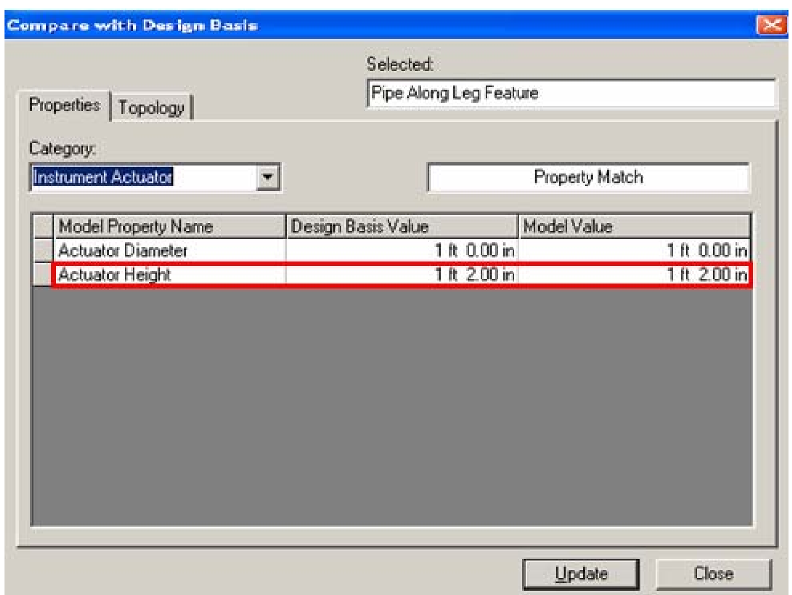


Figure 77: Instrument Actuator Category of FV-311 Compare with Design Basis Dialog Box

9. Select **Close** to close the **Compare with Design Basis** dialog box.

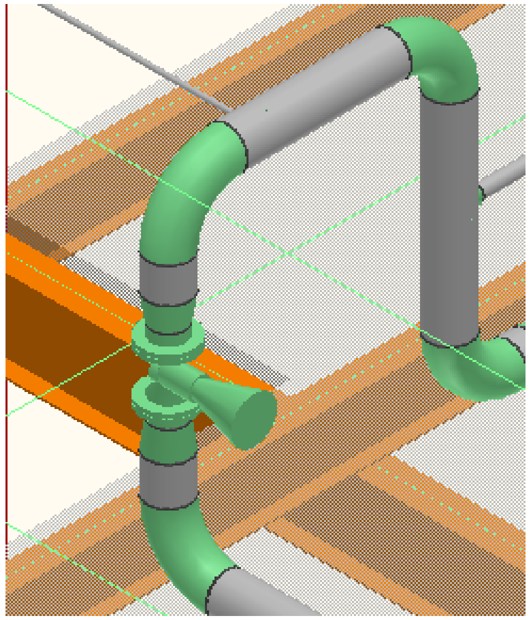


Figure 78: Updated Custom Instrument

For more information related to sequencing objects, refer to the following topics in the user guide *PipingUsersGuide.pdf:*

- *Deleting Feature: An Overview*
- *Moving Pipe Feature: An Overview*
- *Editing Properties: An Overview*

Follow the instructions in the Advanced Lab document in the instructor-led session *Modifying Pipes, Their Features and Piping Objects* to attempt an advanced lab for *Editing of Revised Slop Pipe*

Session 11: Creating Spools

Objective:

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

- Create spool assemblies in a pipe run by using the **Generate Spools** command.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes

Overview:

The **Generate Spools** command breaks pipelines into spools. Spools are collections of piping parts and welds that hold them together. SP3D 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.

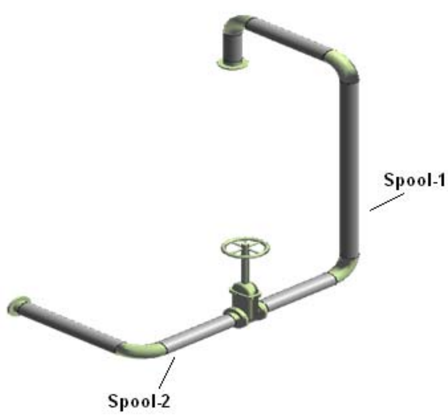


Figure 1: Spools

There are two basic approaches for creating spool assemblies in SP3D.

- **By WBS items method**: SP3D collects all the pipe parts grouped by a WBS item for generating spool assemblies.

- **By Pipeline objects method:** SP3D 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.

Note:

- 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 SP3D. The SP3D administrator can setup to use one or the other.

Steps for Creating Standard Spools from WBS Items:

Exercise 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. 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 that shown in Figure 2.

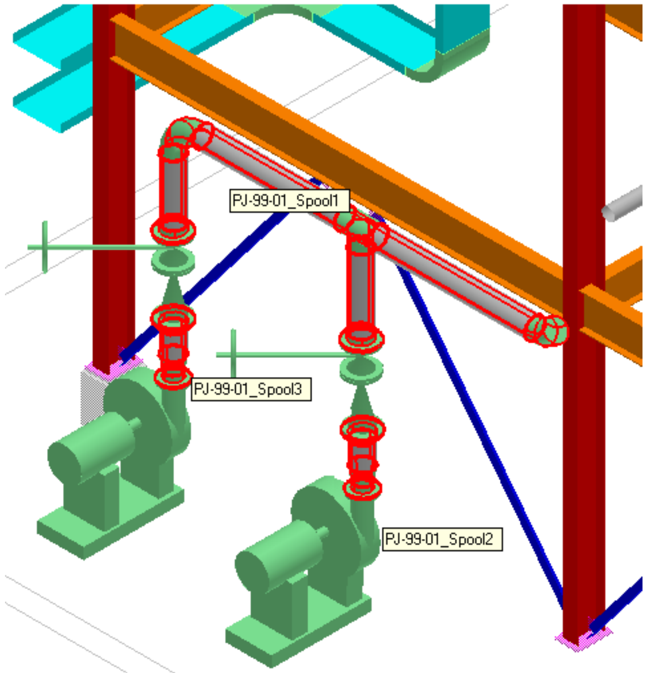


Figure 2: Standard Spools from WBS Item

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 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 the **Piping Parts** option in the **Locate Filter** drop-down list on the **Common** toolbar to select only the piping parts in the graphic view.

   Use the **Inside fence** option on the **Common** toolbar to select all the piping parts of the pipeline **1001-P**, as shown in Figure 3.

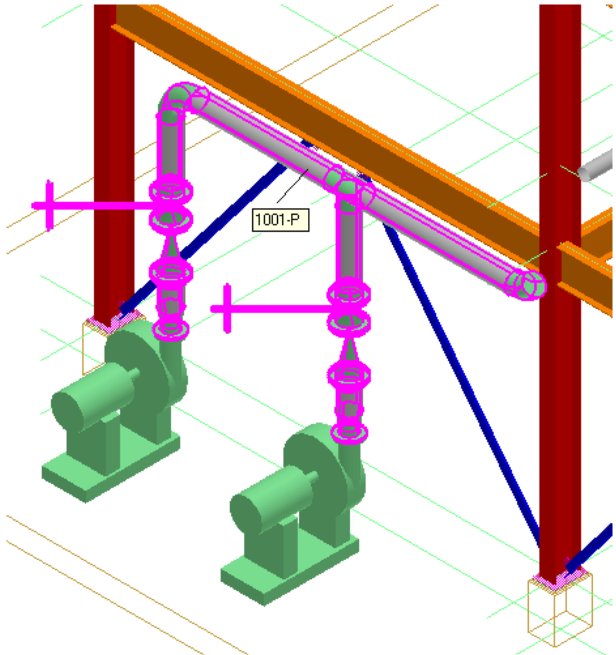


Figure 3: Select Pipe Parts in the Pipeline 1001-P

2. To specify an active project select the **More…** option in the **Active Project** drop-down list on the main toolbar.

3. The **Active Project** dialog box appears. Select the **Database** option to see all the WBS projects in the dialog box. Next, select **PJ-99**, as shown in Figure 4, to specify **PJ-99** as an active project.

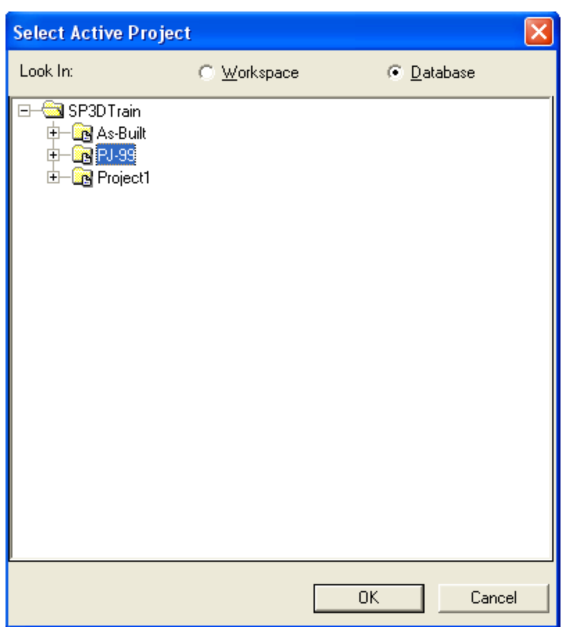


Figure 4: Select the Active Project Dialog Box

4. Click **OK** in the **Active Project** dialog box. The selected active project should appear on the main toolbar, as shown in Figure 5.

Figure 5: Active Project: PJ-99

5. Select the **Project > Claim** command, as shown in Figure 6, to associate the selected piping objects with the active project **PJ-99**.

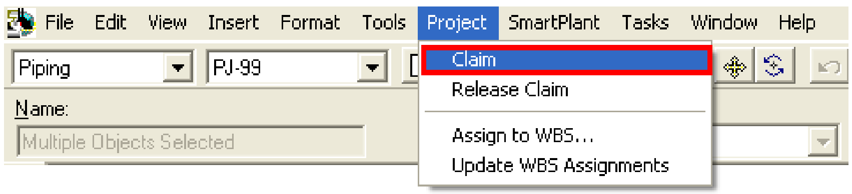


Figure 6: Claim Command

6. After the claim process is complete, SP3D displays the **Claim** dialog box, as shown in Figure 7. Click **Close** to close the dialog box.

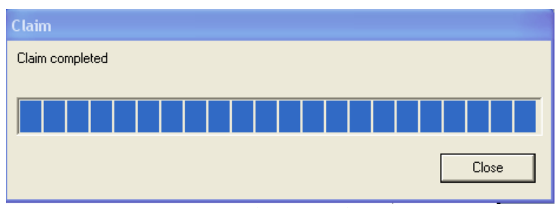


Figure 7: Claim Dialog Box

7. Select the **Project > Assign to WBS…** command, as shown in Figure 8, to associate the selected piping objects as a WBS item.

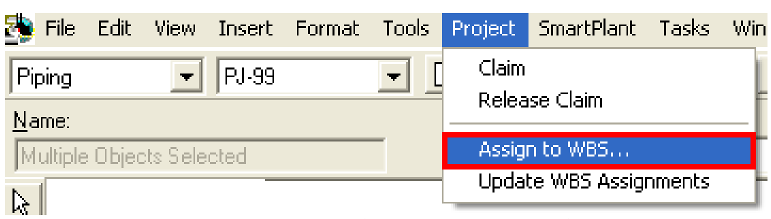


Figure 8: Assign to the WBS… Command

8. The **Assign to WBS** dialog box appears. Select the **Workspace** option and expand the project **PJ-99.**

9. Select the WBS item **PJ-99-01,** as shown in Figure 9 and click **OK** in the **Assign to WBS** dialog box.

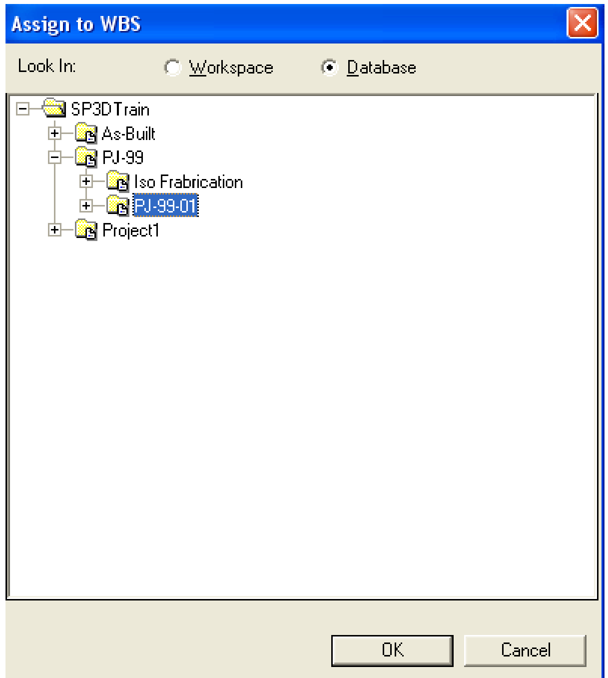


Figure 9: Assign to WBS Dialog Box

Creating Spool Assemblies from a WBS Item:

9. Click the **Generate Spools** button on the vertical toolbar.

Figure 10: Generate Spools Button on the Vertical Toolbar

10. The **Spool Generation** dialog box appears. 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.

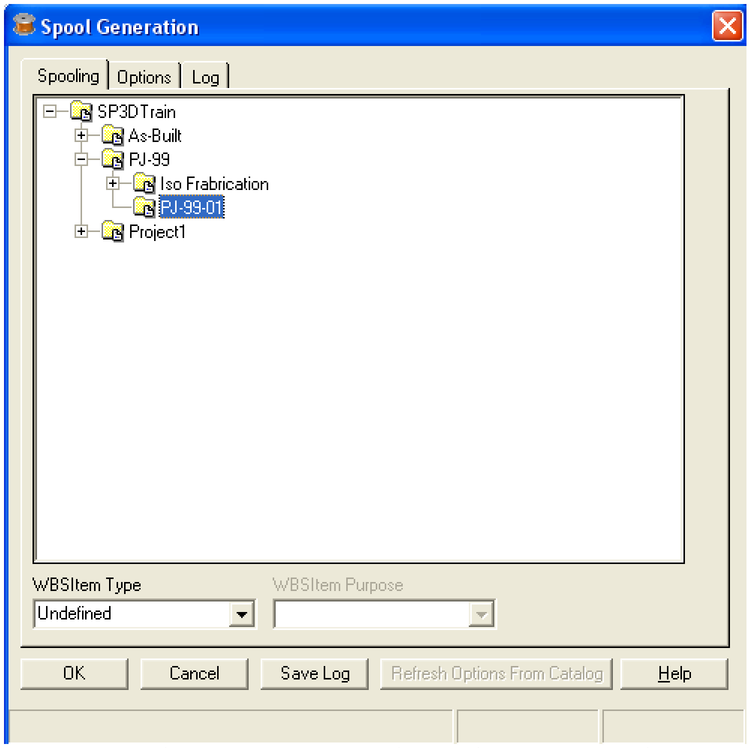


Figure 11: Selecting a WBS Item in the Spool Generation Dialog Box

The **Spool Generation** dialog box contains the following spooling settings on the **Options** tab:

- **Naming Rule** - Specifies the naming rule for spools.
- **Break Spools at Unions** - Specifies that SP3D 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 the Header Spool** - Specifies that a spool can include the stub-in pipe and all the parts of the stub-in branch until the first spool-breaking component appears, as shown in Figure 12.

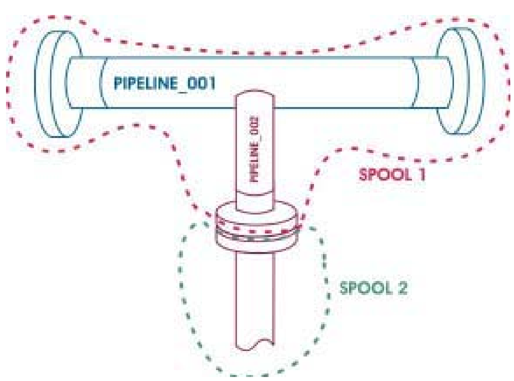


Figure 12: Spools

- **Include Items Welded to Pipe in Spools** - Includes welded objects, such as pipe hangers or support parts, in the same spool to which they are welded.
- **Delete Existing Spools** - Changes the spool numbering only for the modified or added pipes.
- **Ignore Boundaries of the Object Being Spooled** – Specifies whether or not SP3D should cross the boundary of a pipeline or a block for spool generation until an intrinsic spool break is found.
- **Spool Break By Control Point** - Specifies whether or not spools should break at control points, which are used as boundaries to define where one WBS item grouping ends and the next grouping begins. The **Spool Break By Control Point** drop-down list contains the following three options**:**

    - **Ignore Control Points:** Ignores the control points during spooling
    - **Break at Control Points:** Breaks spools at the normal intrinsic line breaks and control points
    - **Break Only at Control Points:** Breaks spools only at control points

11. Select the **Options** tab in the **Spool Generation** dialog box to review the spooling options.

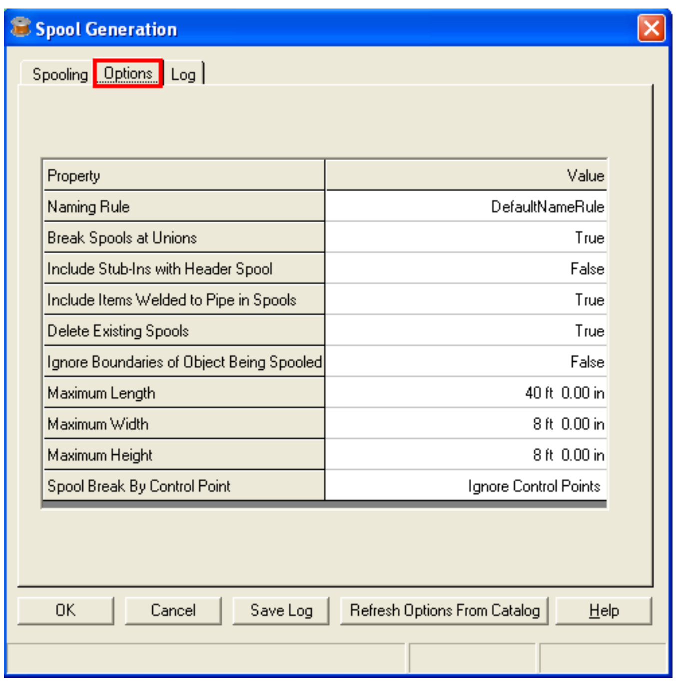


Figure 13: Options Tab of the Spool Generation Dialog Box

12. Click **OK** to apply the default settings to the spool that is generated.

The **Log** tab in the **Spool Generation** dialog box is divided into four parts:

- **Before** - Lists all the spools that existed in the model before you ran the last spooling process. This list of spools is the same as the one that appears if you select this tab before processing spools.

- **ToDoList** - Lists the spools that have objects in the To Do List. You cannot create the spools until you fix the objects in 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.

13. Click the **Log** tab in the **Spool Generation** dialog box to review the spooling log and close the dialog box.

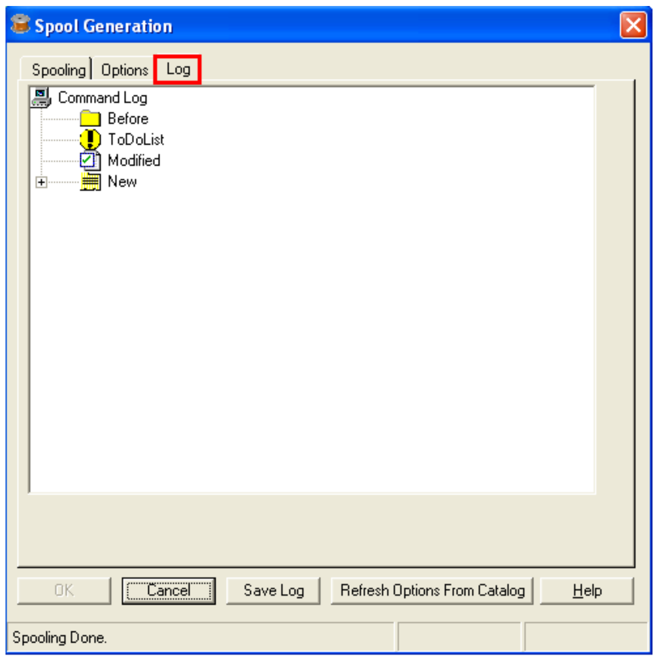


Figure 14: Spooling Log in the Spool Generation Dialog Box

Tips:

- 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.

14. 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, as shown in Figure 15.

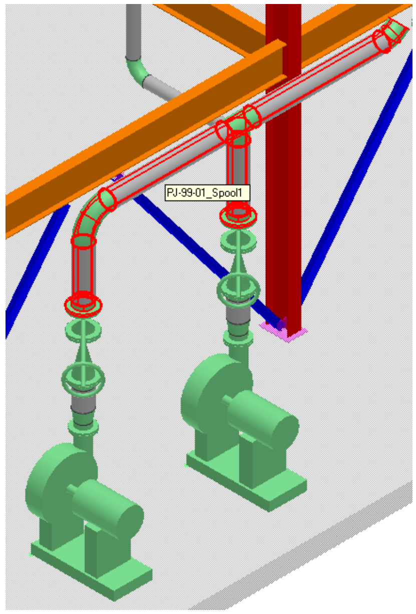


Figure 15: Generated Spool in the Graphic View

You can check the spools on the **Assembly** tab in the **Workspace Explorer**, as shown in Figure 16.

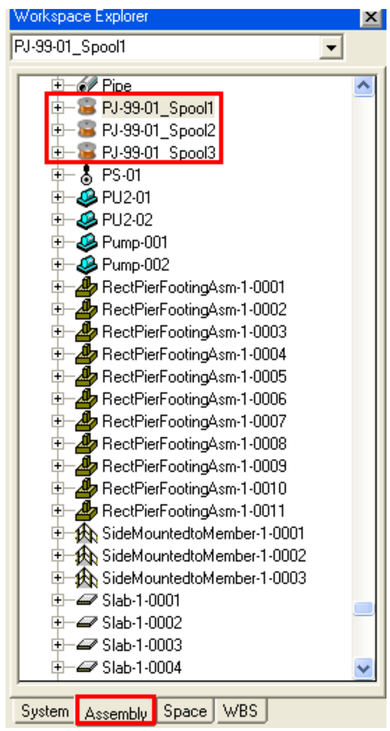


Figure 16: Spools Under the Assembly Tab of the Workspace Explorer

Steps for Creating Standard Spools from Pipeline Objects:

Exercise 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 Figure 17.

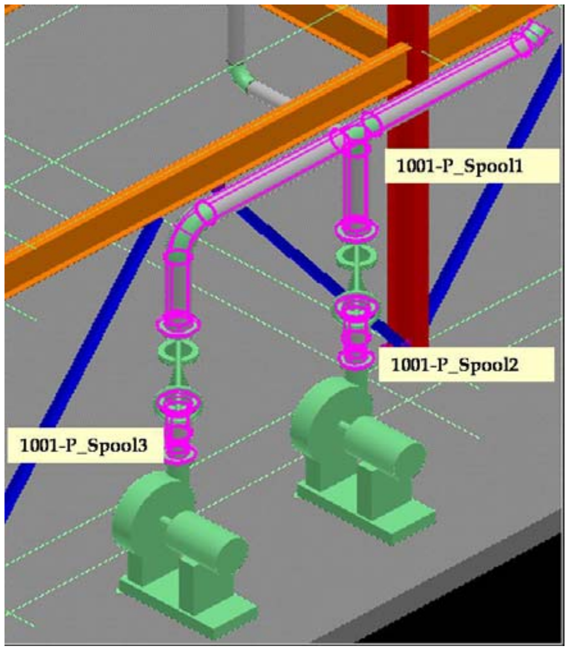


Figure 17: Standard Spools

Before beginning the 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.**

52. Click the **Generate Spools** button on the vertical toolbar.

Figure 18: Generate Spools Button on the Vertical Toolbar

53. The **Spool Generation** dialog box appears. 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 may occur in spooling.

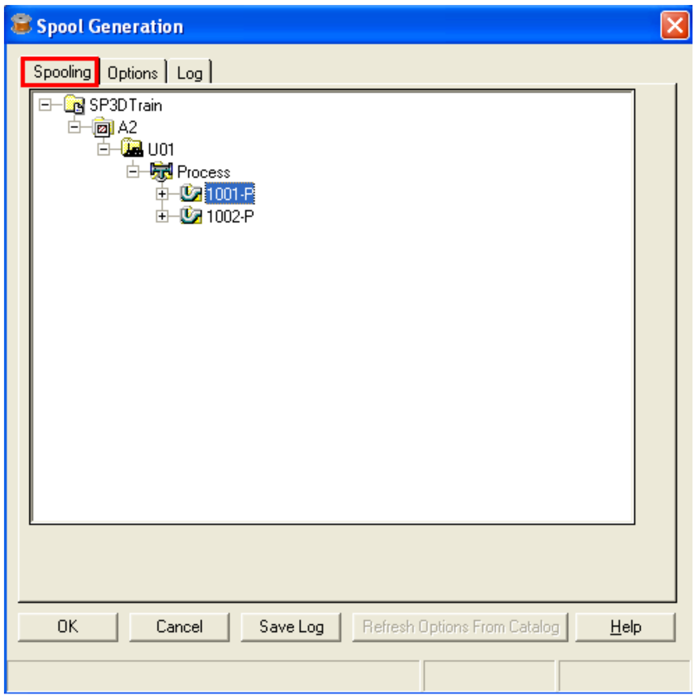


Figure 19: Selecting a Pipeline in the Spool Generation Dialog Box

54. Click the **Options** tab in the **Spool Generation** dialog box to review the spooling options.

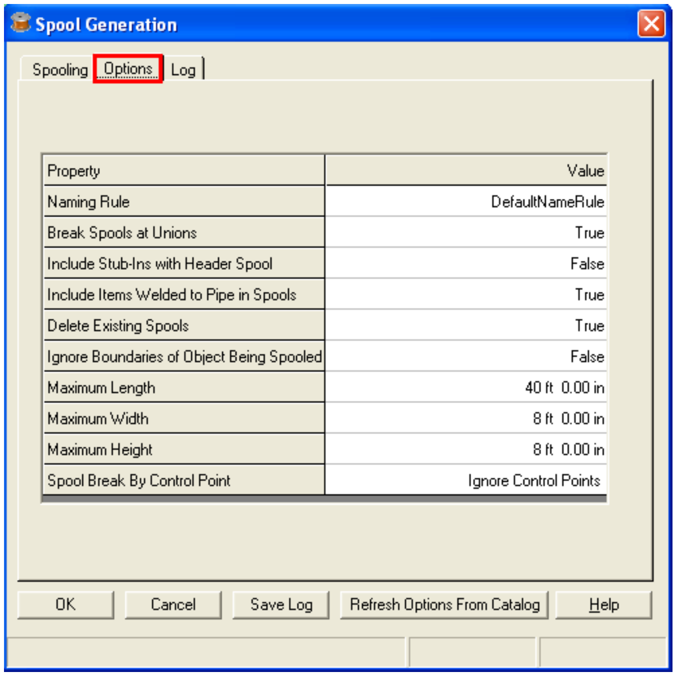


Figure 20: Options Tab in the Spool Generation Dialog Box

55. Click **OK** to apply the default settings to the spool that is generated.

56. Click the **Log** tab in the **Spool Generation** dialog box to review the spooling log and close the dialog box.

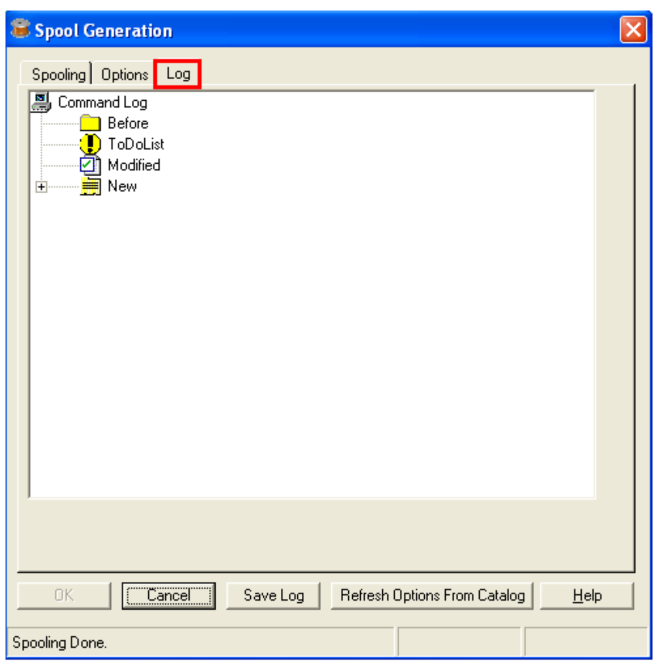


Figure 21: Spooling Log in the Spool Generation dialog box

57. 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, as shown in Figure 22.

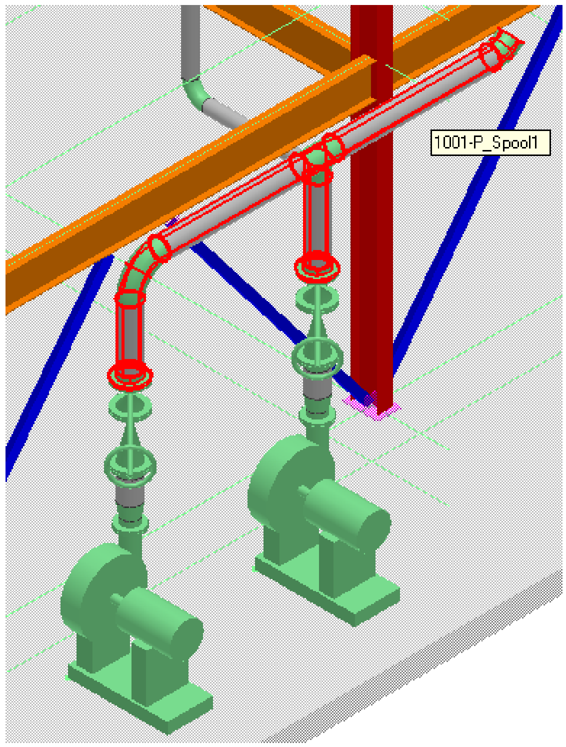


Figure 22: Generated Spool in the Graphic View

While generating spools, SP3D 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, SP3D 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 in Figure 23.

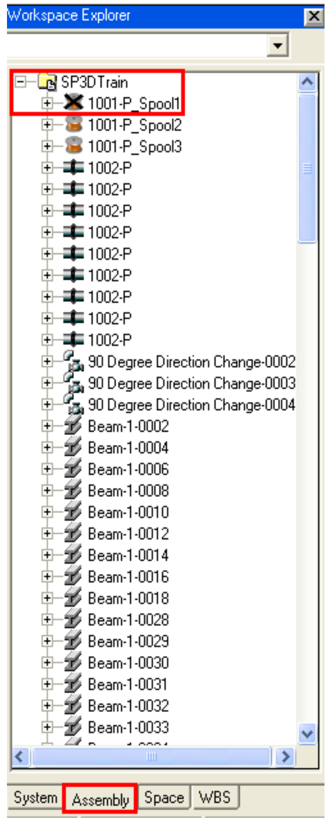


Figure 23: Oversized Spool Under the Assembly Tab of the Workspace Explorer

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.

Steps for Creating Spools by Using Manual Break Points (Control Points):

Exercise Objective: In this exercise you will be inserting control points in the oversized spool **1001-P_Spool1** shown in Figure 9 to manage the size of the spool assembly. The view of the pipe at the end of the procedure should resemble Figure 24.

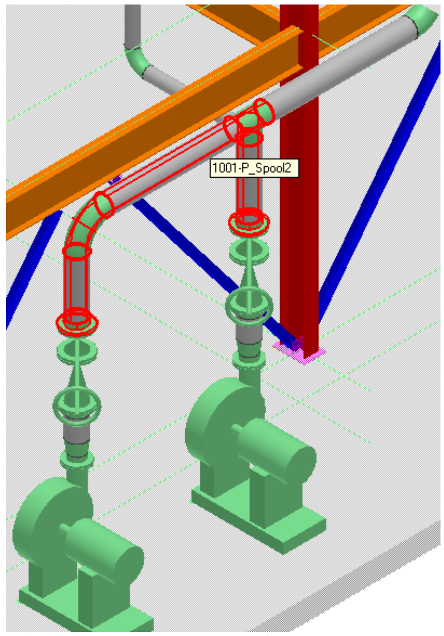


Figure 24: Spool in the Graphic View After Inserting a Control Point

1. Make sure that **Locate Filter** is set to **Spools**.

2. Select the **Insert** > **Control Point** command.

Figure 25: Insert > Control Point Command

3. Select **Distribution Connection** point as **Control Point Parent,** on **1001-P_Spool1**, as shown in Figure 26.

Hint:

- Use the **QuickPick** tool to find the connection objects at busy joints.

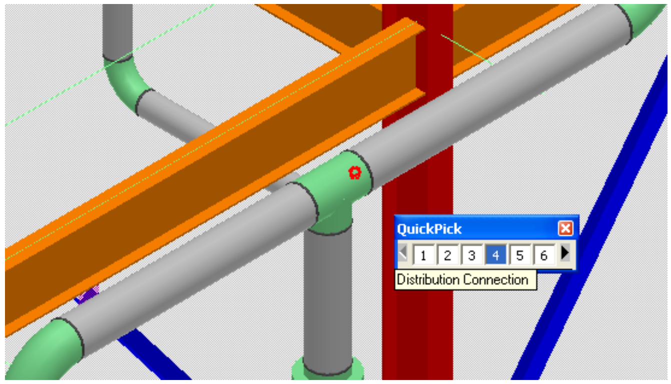


Figure 26: Selecting a Distribution Connection

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 the **Generate Spools** button on the vertical toolbar.

If you have gone through the procedure for Creating Standard Spools From WBS Items in the previous exercise, then you will perform the step 6 and skip step 7 in this exercise. If you did the steps for Creating Standard Spools from Pipeline Objects, then you skip step 6.

6. On the **Spooling** tab in the **Spool Generation** dialog box, expand **PJ-99 folder** and select **PJ-99-01.**

7. On the **Spooling** tab in the **Spool Generation** dialog box, expand **A2 > U01 > Process** and select **1001-P.**

8. 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, as shown in Figure 27.

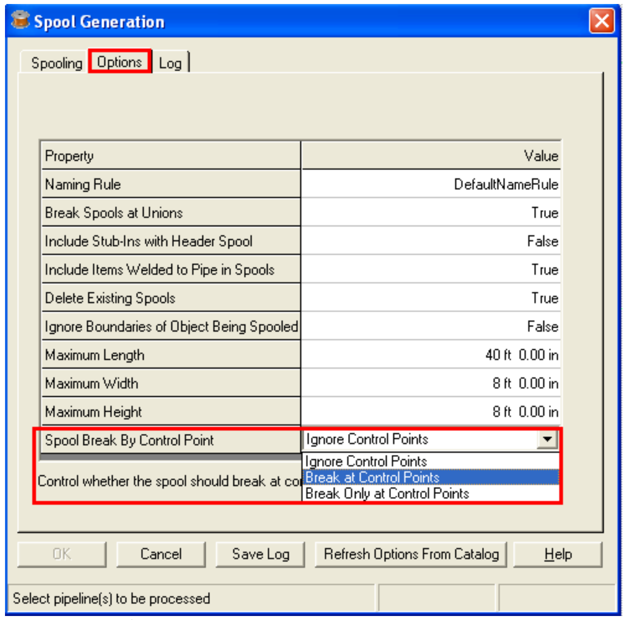


Figure 27: Defining Options in the Spool Generation Dialog Box

9. Click **OK** in the **Spool Generation** dialog box to process the pipelines into spools. This will regenerate spools, and a new spool break will be inserted at the control point.

10. Now click the **Log** tab in the **Spool Generation** dialog box. This will show you the number of spools before and after inserting the control point, as shown in Figure 28.

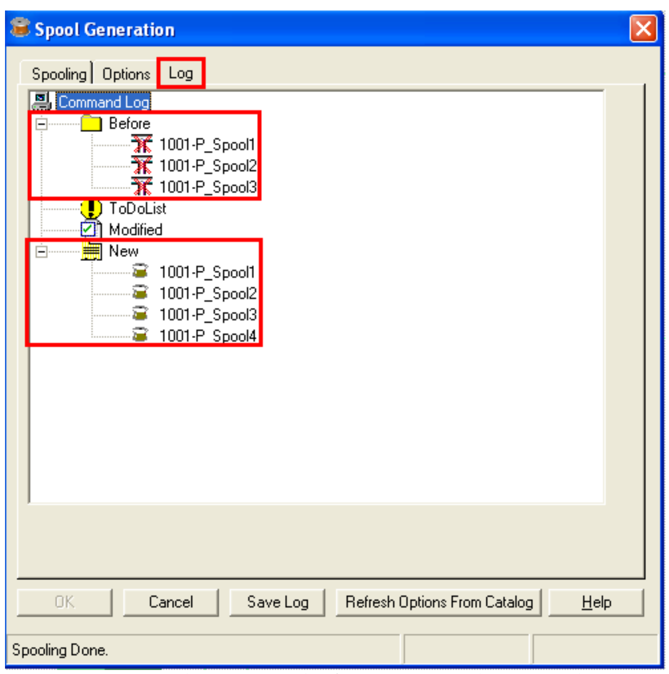


Figure 28: Spools Generated After Inserting the Control Point

11. Click **Can**cel 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 *Spooling: An Overview,* topic in the user guide *PipingUsersGuide.pdf.*

Session 12: Sequencing Objects

Objective:

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

- Sequence piping objects in a model.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Creating Spools

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.

Steps for Sequencing Piping Objects in a WBS Item:

Exercise 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 in Figure 1.

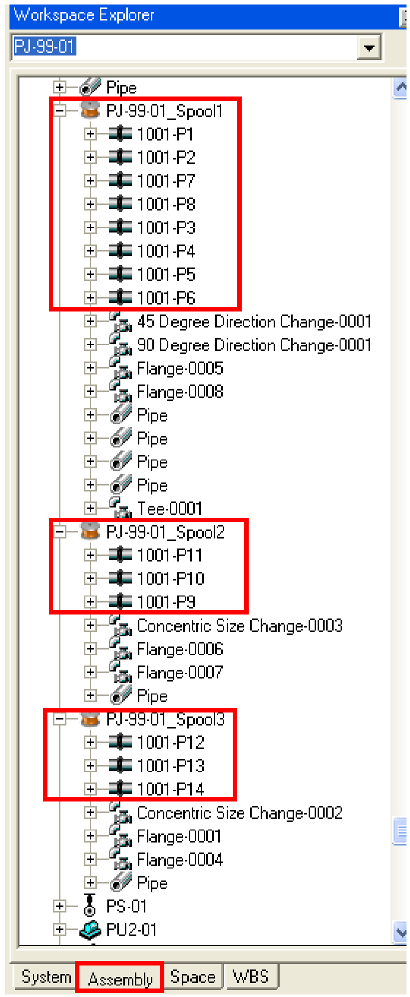


Figure 1: Sequenced Pipe Welds in the Workspace Explorer

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**.

10. Select the **Welds** option in the **Locate Filter** drop-down list on the **Common** toolbar to select only the welds in the graphic view.

    Use the **Inside fence** option on the **Common** toolbar to select all the welds that are part of the pipeline **1001-P**, as shown in Figure 2.

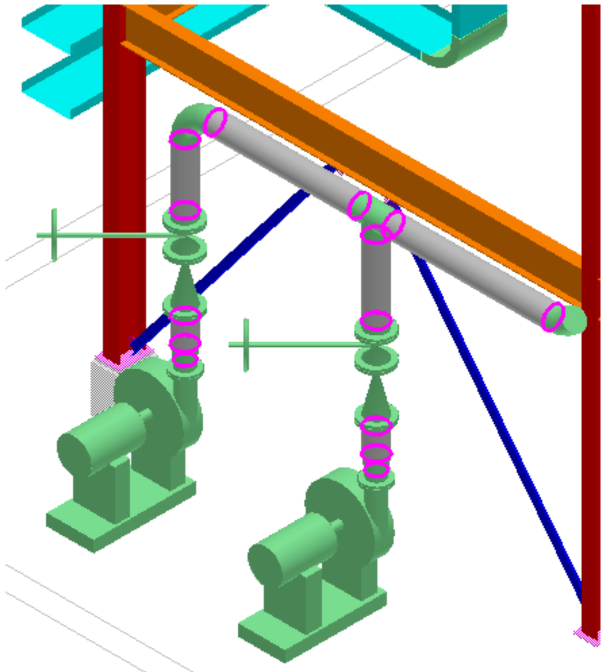


Figure 2: Select Welds in the Pipeline 1001-P

11. Select the **Project > Assign to WBS…** command, as shown in Figure 3, to associate the selected piping objects to a WBS item.

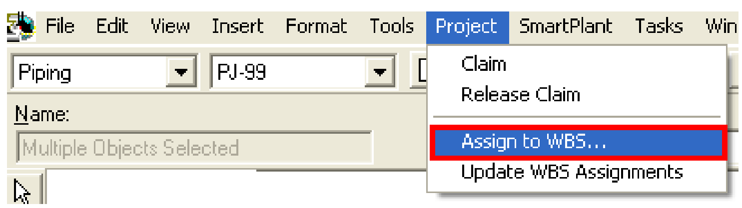


Figure 3: Assign to WBS… Command

12. The **Assign to WBS** dialog box appears. Select the **Database** option in this dialog box.

13. Expand the project **PJ-99** and select the WBS item **PJ-99-01**, as shown in Figure 4**.**

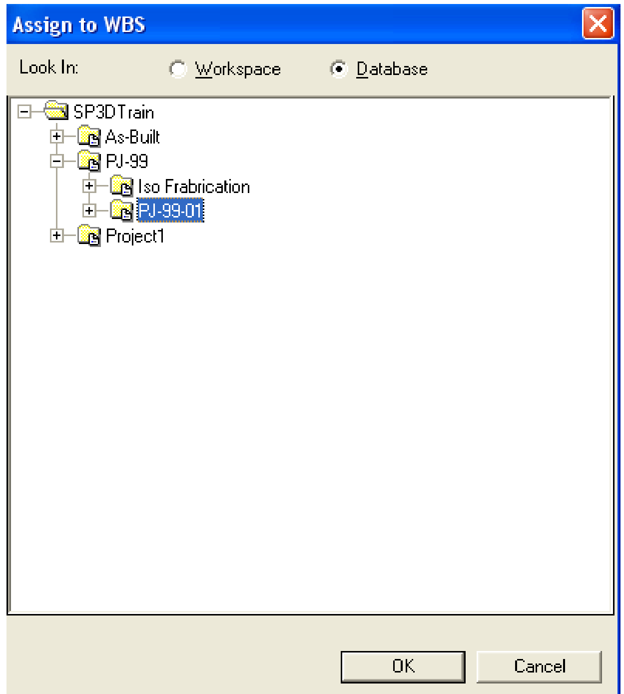


Figure 4: Assign to WBS Dialog Box

14. Click **OK** in the **Assign to WBS** dialog box.

Sequencing Welds from WBS Items:

15. Click the **Sequence Objects** button on the vertical toolbar.

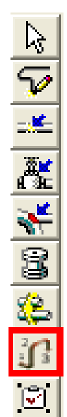


Figure 5: Sequence Objects Button on the Vertical Toolbar

When you click the **Sequence Objects** button, the **Sequence Objects** ribbon appears.

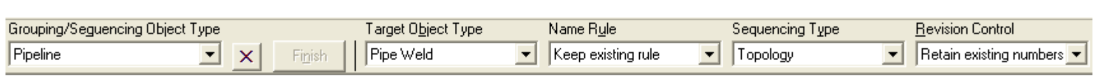


Figure 6: Sequence Objects Ribbon

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.

16. Select the **WBS Item** option in the **Grouping/Sequencing Object Type** drop-down list, as shown in Figure 7.

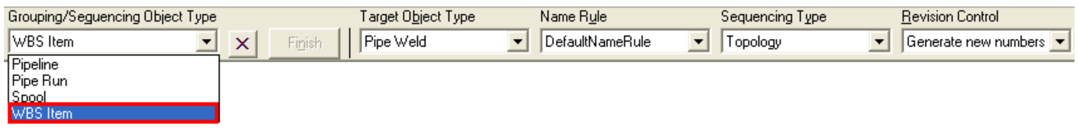


Figure 7: Grouping/Sequencing Object Type Drop-Down List on the Sequence Objects Ribbon

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.

17. Select the **Pipe Weld** option in the **Target Object Type** drop-down list to specify the target object to be renamed on the pipeline, as shown in Figure 8.

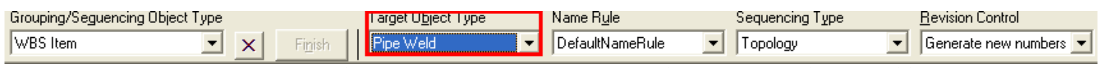


Figure 8: Target Object Type Drop-Down List on the Sequence Objects Ribbon

18. Select the **DefaultNameRule** option in the **Name Rule** drop-down list to specify the naming rule for renaming the target object.

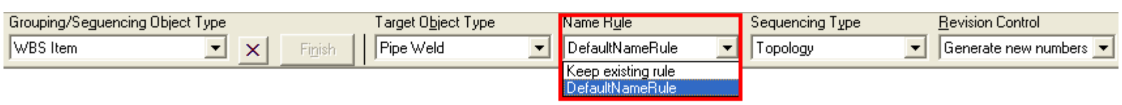


Figure 9: Name Rule Drop-Down List on the Sequence Objects Ribbon

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.

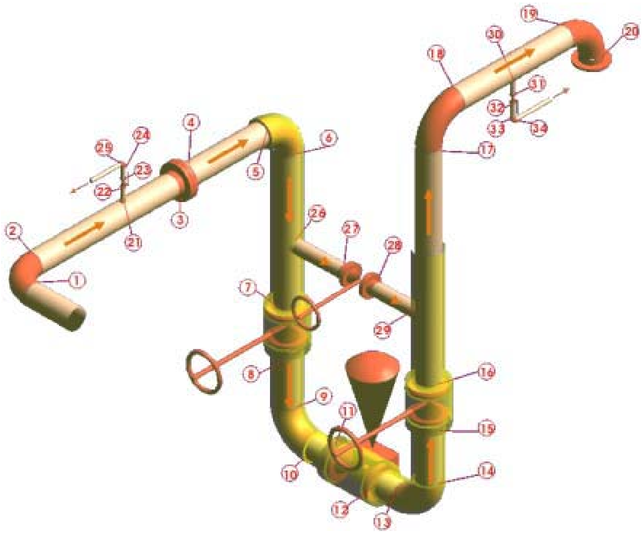


Figure 10: Arrow Shows the Flow Direction

- **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.

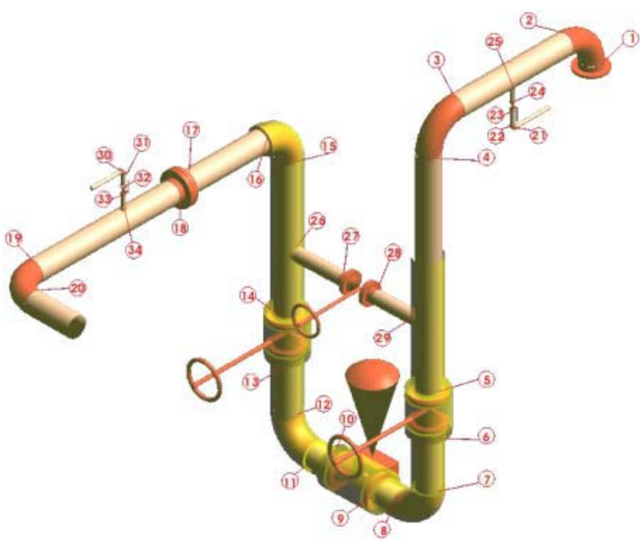


Figure 11: Topology

19. Select the **Topology** option in the **Sequencing Type** drop-down list to define the sequence type for the welds.

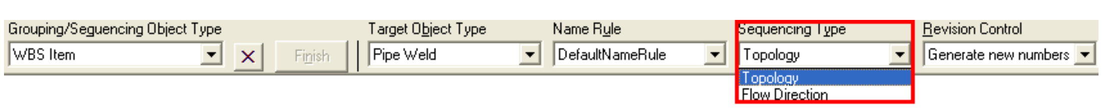


Figure 12: Sequencing Type Drop-Down List on the Sequence Objects Ribbon

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.

20. Select the **Generate new numbers** option in the **Revision Control** drop-down list to generate new numbers for the welds.

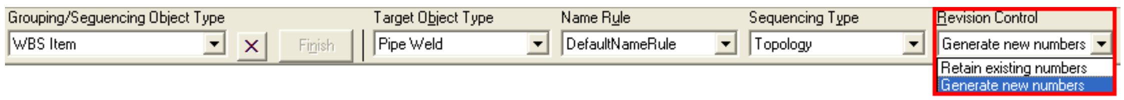


Figure 13: Revision Control Drop-Down List on the Sequence Objects Ribbon

21. Select the WBS item **PJ-99-01** in the **Workspace Explorer**.

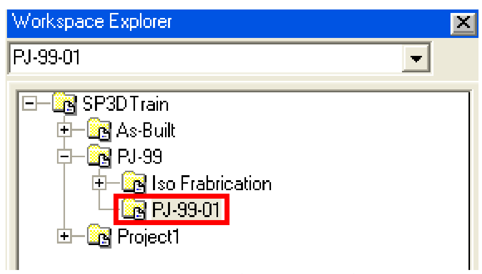


Figure 14: Workspace Explorer

22. Click the **Finish** button on the Sequence Objects Ribbon to execute the command and apply the options set for renaming the pipe welds.

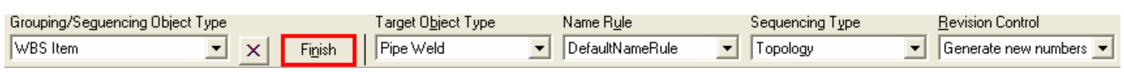


Figure 15: Finish Button on the Sequence Objects Ribbon

Steps for Sequencing the Piping Objects in a Pipeline:

Exercise 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 in Figure 16.

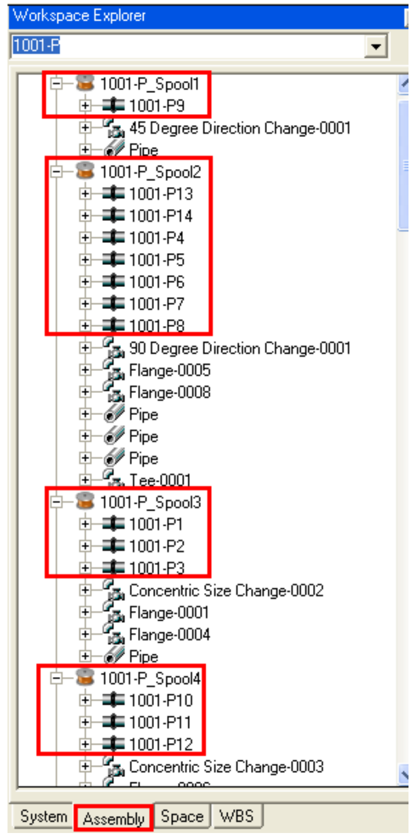


Figure 16: Sequenced Pipe Welds

Before beginning the 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.**

58. Click the **Sequence Objects** button on the vertical toolbar.

Figure 17: Sequence Objects Button on the Vertical Toolbar

When you click the **Sequence Objects** button, the **Sequence Objects** ribbon appears.

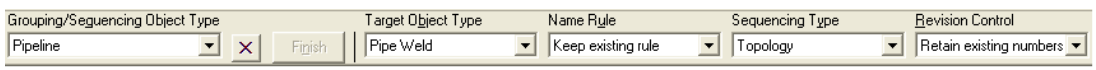


Figure 18: Sequence Objects Ribbon

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 a grouping object. This option defines the collection of target objects to be sequenced and defines the boundaries of the sequencing.

59. Select the **Pipeline** option in the **Grouping/Sequencing Object Type** drop-down list, as shown in Figure 19.

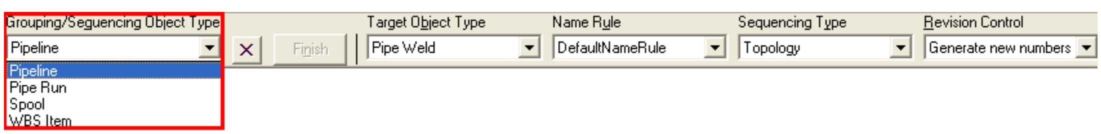


Figure 19: Grouping/Sequencing Object Type Drop-Down List on the Sequence Objects Ribbon

The second option on the **Sequence Objects** ribbon is **Target Object Type.** You use this option to specify the target object that you want to rename on the selected grouping object.

60. Select the **Pipe Weld** option in the **Target Object Type** drop-down list to specify the target object to be renamed on the pipeline, as shown in Figure 20.

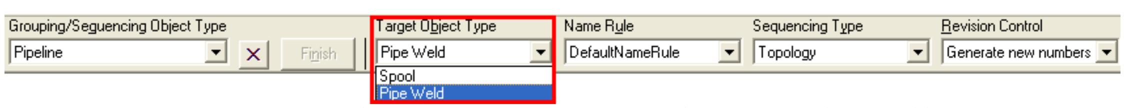


Figure 20: Target Object Type Drop-Down List on the Sequence Objects Ribbon

Tip:

- You can sequence the spools of a pipeline by selecting the **Spool** option in the **Target Object Type** drop-down list.

61. Select the **DefaultNameRule** option in the **Name Rule** drop-down list to specify the naming rule for renaming the target object.

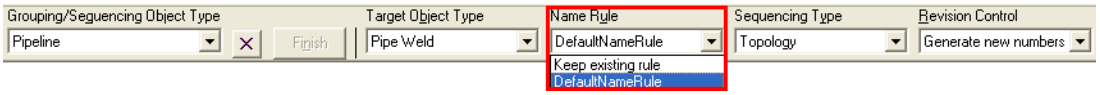


Figure 21: Name Rule Drop-Down List on the Sequence Objects Ribbon

After selecting the target object, you define the sequencing type to specify the logical order of the target object.

62. Select the **Flow Direction** option in the **Sequencing Type** drop-down list to define the sequence type for the welds.

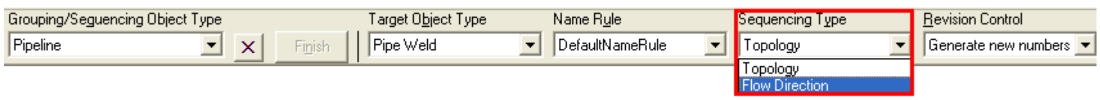


Figure 22: Sequencing Type Drop-Down List on the Sequence Objects Ribbon

Tip:

- In SP3D, spools are sequenced mainly by using the flow direction sequencing type.

63. Select the **Generate new numbers** option in the **Revision Control** drop-down list to generate new numbers for the welds.

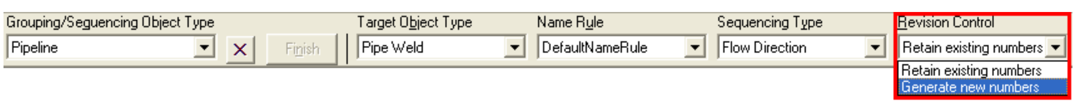


Figure 23: Revision Control Drop-Down List on the Sequence Objects Ribbon

64. Now select the pipeline **1001-P** in the graphic view, as shown in Figure 24.

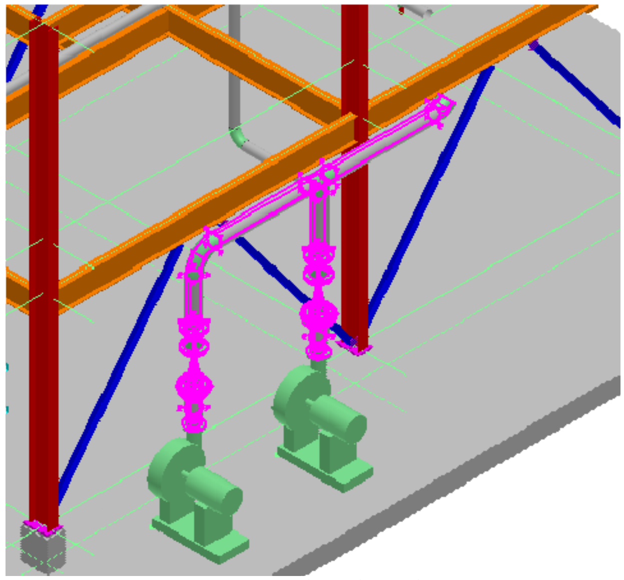


Figure 24: Selected Pipeline

65. Click the **Finish** button on the Sequence Objects Ribbon to execute the command and apply the options set for renaming the pipe welds.

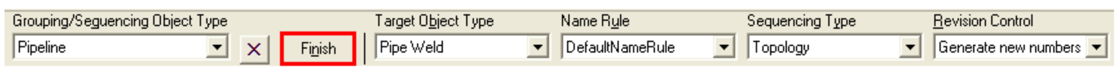


Figure 25: Finish Button on the Sequence Objects Ribbon

For more information related to sequencing objects, refer to *Sequence Objects Command,* topic in the user guide *PipingUsersGuide.pdf.*

Session 13: 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:

- SP3D Overview
- SP3D Common Sessions
- Piping: An Overview
- Routing Pipes
- Inserting Components in a Pipe Run
- Creating Spools
- SP3D 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 SP3D 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 WBS Items:

In SP3D 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.

Steps for Creating Isometric Drawings by Manually Assigning Objects to an Active Project or a WBS Item:

Exercise Objective: In this 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 Figure 1.

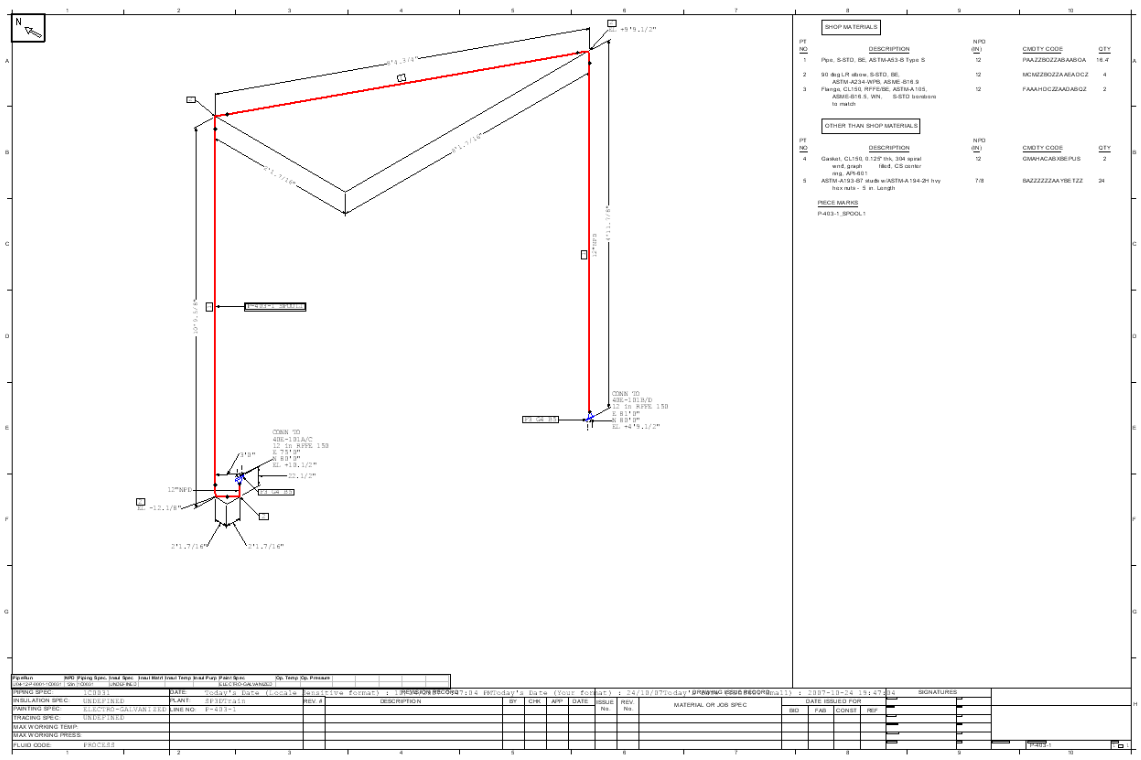


Figure 1: Isometric Drawing for the WBS Group of Pipeline 403-P

Before beginning the 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.

Manually Assigning Piping Objects to an Active Project:

1. Select the **All** option in the **Locate Filter** drop-down list.

2. In the **Workspace Explorer** window expand the hierarchy **A2>U04>Process.**

3. Right-click the pipeline system **403-P** and click the **Select Nested** command in the menu, as shown in Figure 2, to select all the piping objects in the graphic view.

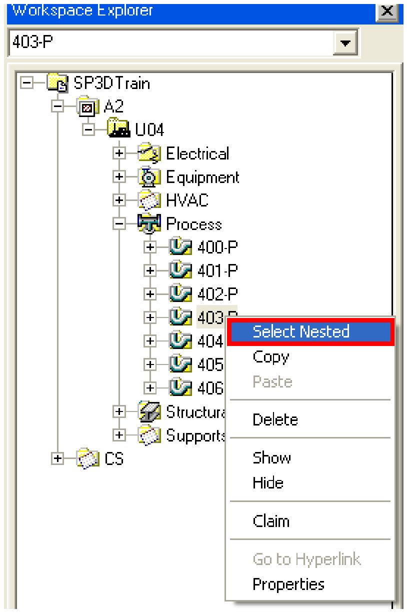


Figure 2: Workspace Explorer Window

4. Select the **More…** option in the **Active Project** drop-down list on the toolbar to specify the active project.

5. Select the **Database** option to see all the WBS projects in the displayed dialog box and then select **PJ-99**, as shown in Figure 3, to specify **PJ-99** as an active project.

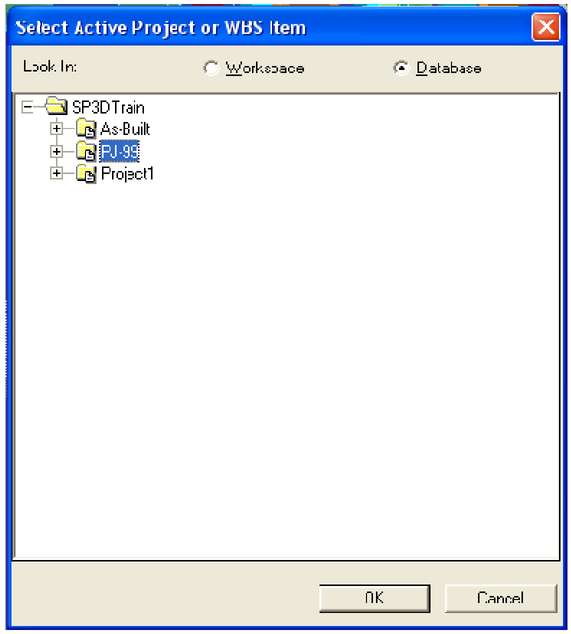


Figure 3: Select Active Project or WBS Item Dialog Box

6. Click **OK** on the dialog box. The selected active project should display, as shown in Figure 4.

Figure 4: Active Project: PJ-99

7. Select the **Project > Claim** command, as shown in Figure 5, to associate the selected piping objects with the active project **PJ-99**.

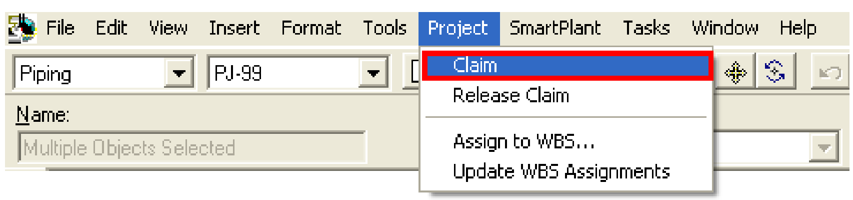


Figure 5: Project>Claim Command

8. After the claim process is complete, SP3D displays the **Claim** dialog box, as shown in Figure 6. Click **Close** to close the dialog box.

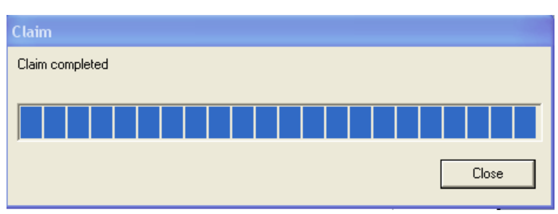


Figure 6: Claim Dialog Box

Manually Assigning Piping Objects to WBS item:

9. In the **Workspace Explorer**, select the **WBS** tab and expand **PJ-99>Iso Fabrication.**

10. Right-click the **Iso Fabrication** system in the **Workspace Explorer** and select the **Create WBS Item** command from the menu, as shown in Figure 7, to create a new WBS item in the system **Iso Fabrication**.

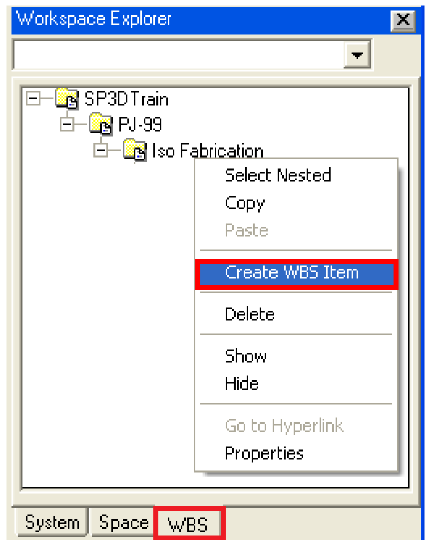


Figure 7: WBS Tab in the Workspace Explorer

11. Select the property specifications, as shown in Figure 8, and Click **OK**.

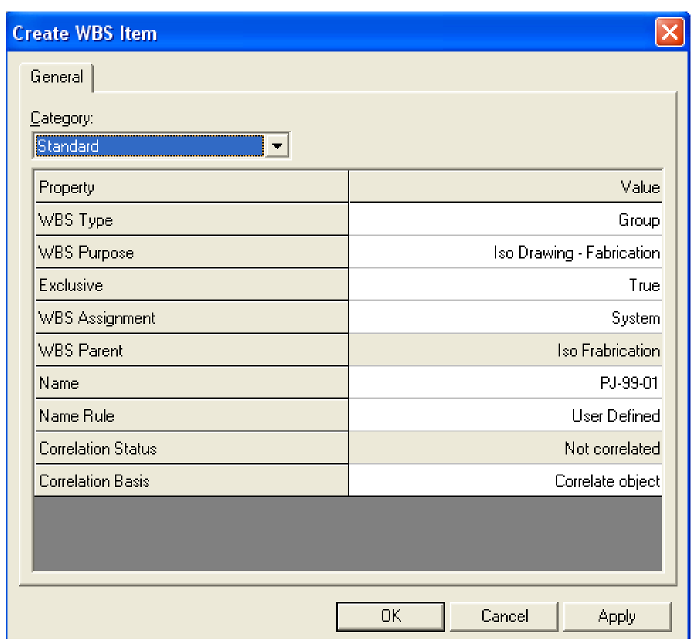


Figure 8: Create WBS Item Dialog Box

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.

12. 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, as shown in Figure 9.

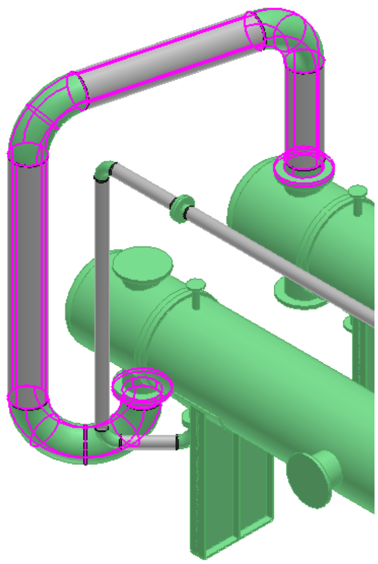


Figure 9: Pipeline 403-P

13. Select the **Project > Assign to WBS…** command. This command creates a relationship between all the components of the selected pipeline system and a selected WBS item.

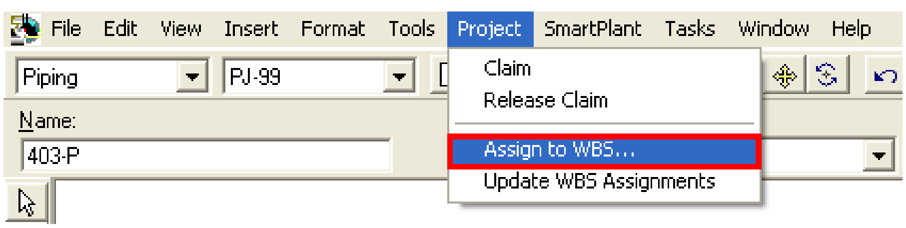


Figure 10: Project>Assign To WBS… Command

14. The **Assign to WBS** dialog box appears. 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.**

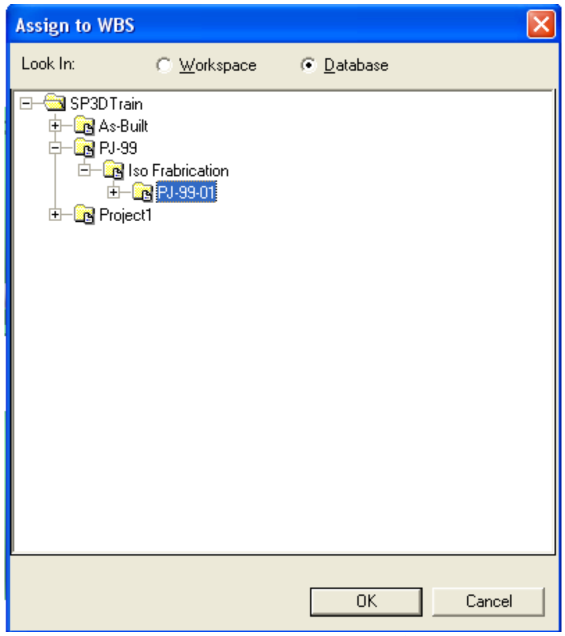


Figure 11: Assign To WBS Dialog Box

15. Select the **WBS** tab in the **Workspace Explorer**. Expand **PJ-99**. Right-click **PJ-99-01** and click the **Select Nested** command 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:

16. 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 SP3D window, as shown in Figure 12. 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.**

17. In the **Management Console**, expand the drawing hierarchy to **Unit 01>Isometrics>Iso WBS Isometrics**, as shown in Figure 12.

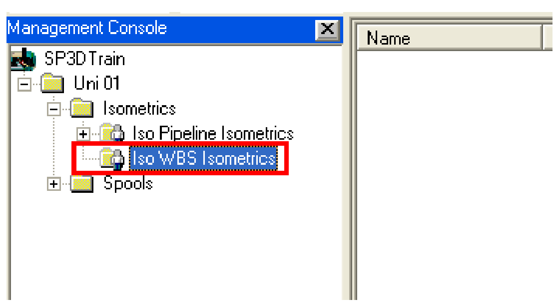


Figure 12: Management Console

18. Right-click the **Iso WBS Isometrics** component in the **Management Console** and select the **Run Query** option, as shown in Figure 13. SP3D 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.

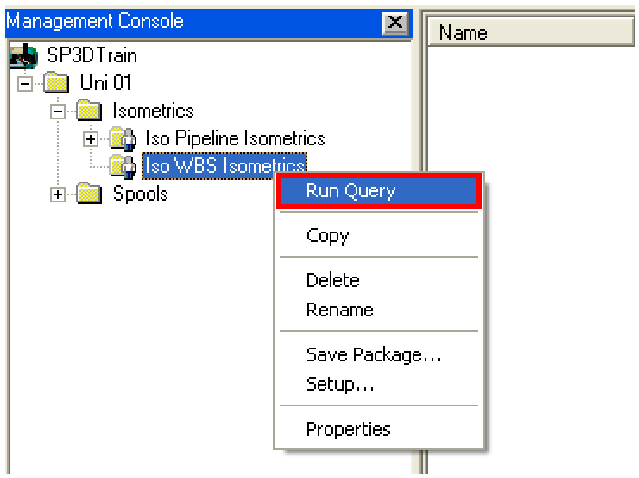


Figure 13: Run Query Option

19. The active project **PJ-99** and WBS item **Iso Fabrication** appears in the hierarchy of the **Management Console**, as shown in Figure 14.

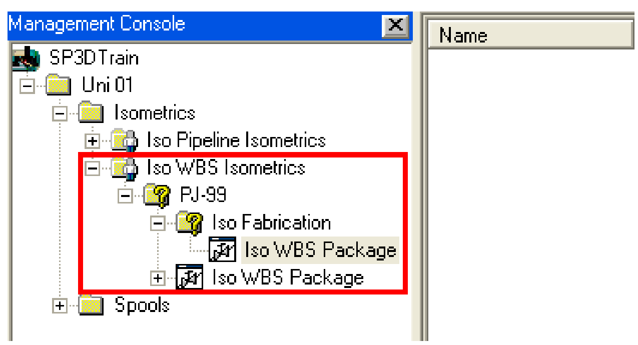


Figure 14: Drawing Management Console

20. Now right-click the **Iso WBS Package** under **Iso WBS Isometrics > PJ-99 > Iso Fabrication** and click the **Create Drawings** option, as shown in Figure 15, to create the isometric drawing documents.

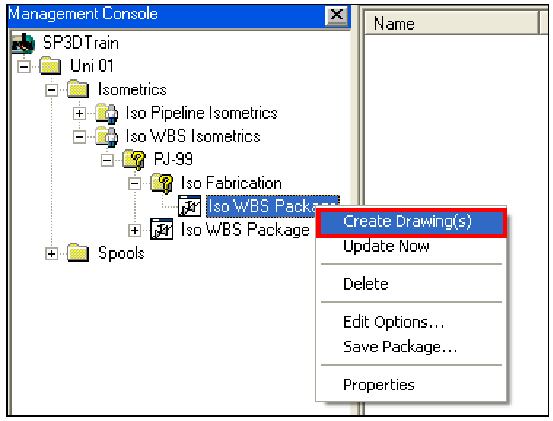


Figure 15: Create Drawing(s) Option

SP3D generates isometric drawing documents for all the WBS items available in the **Iso Fabrication** WBS parent, as shown in Figure 16.

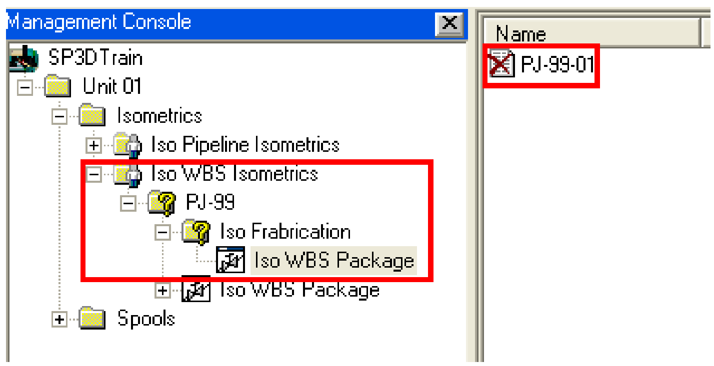


Figure 16: Isometric Drawings for the created WBS items in the Management Console

21. Right-click the **Iso WBS Package** isometric drawing type and click the **Update Now** option, as shown in Figure 17, to update the drawings for the WBS group.

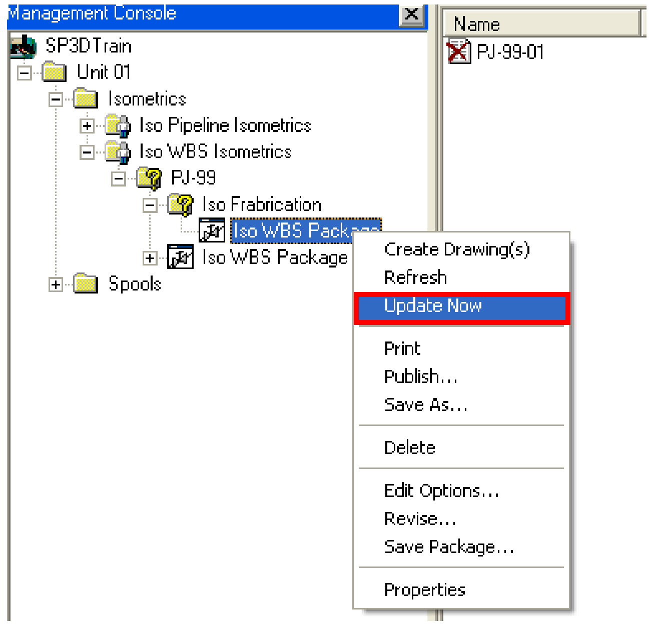


Figure 17: Updating the Drawing Documents Created from WBS Items

SP3D 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**, as shown in Figure 18, which shows that the isometric drawing is generated**.**

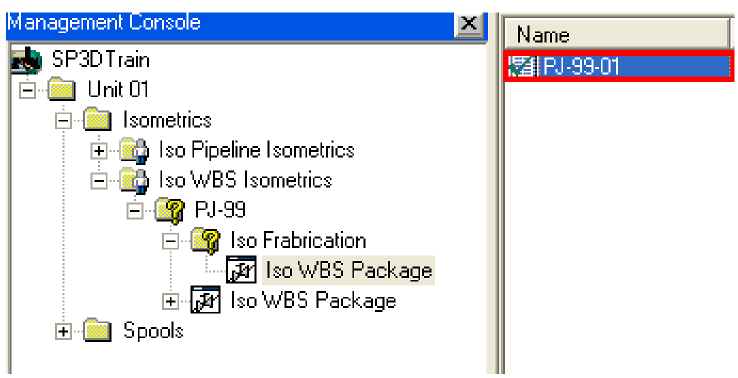


Figure 18: Successfully Created Drawing PJ-99-01

After the updation process is complete, double-click the isometric drawings created for the WBS item. A pictorial representation of the isometric drawings will be displayed, as shown in Figure 19.

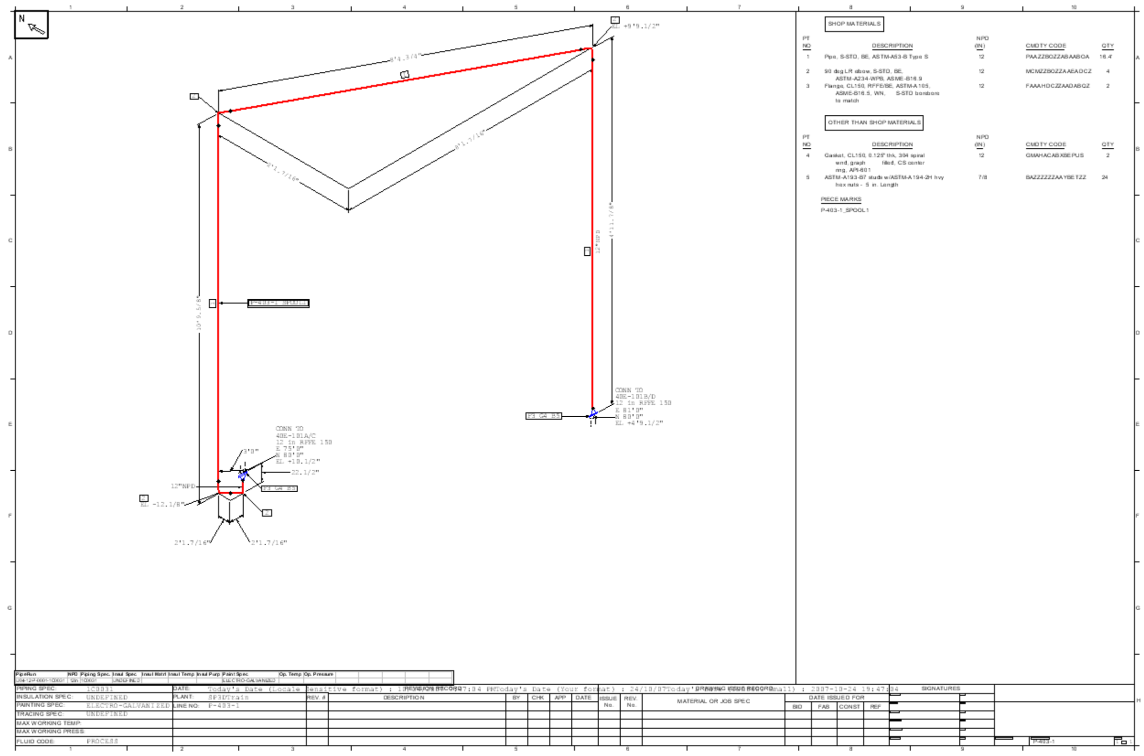


Figure 19: Isometric Drawing for PJ-99-01 WBS Item

Steps for Creating Isometric Drawings by Automatically Assigning Objects to WBS Items:

Exercise 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 Figures 20 and 21.

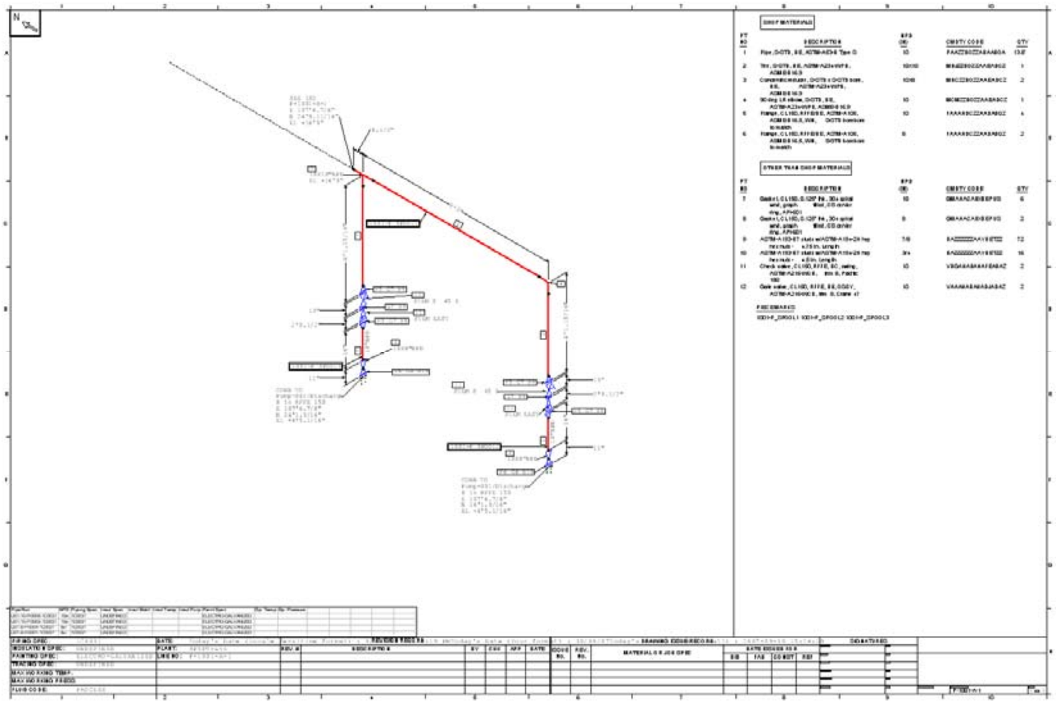


Figure 20: Isometric Drawing for the First WBS Group

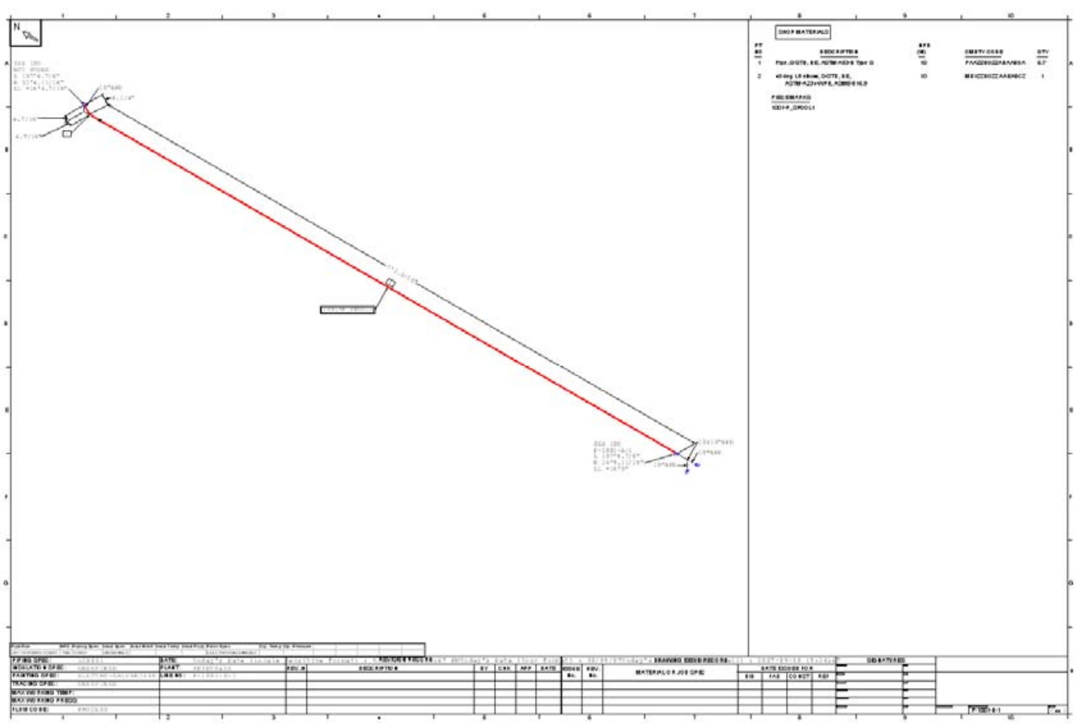


Figure 21: Isometric Drawing for the Second WBS Group

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.

Manually Assigning Piping Objects to Active Project:

Assign all the piping objects of pipeline **1001-P** to an active project **PJ-99** in Unit **U01** of your workspace.

23. Select the **Piping Parts** option in the **Locate Filter** drop-down list on the **Common** toolbar to select only the piping parts in the graphic view.

24. Use the **Inside fence** option on the **Common** toolbar to select all piping parts belonging to the pipeline **1001-P** and select the piping parts, as shown in Figure 22.

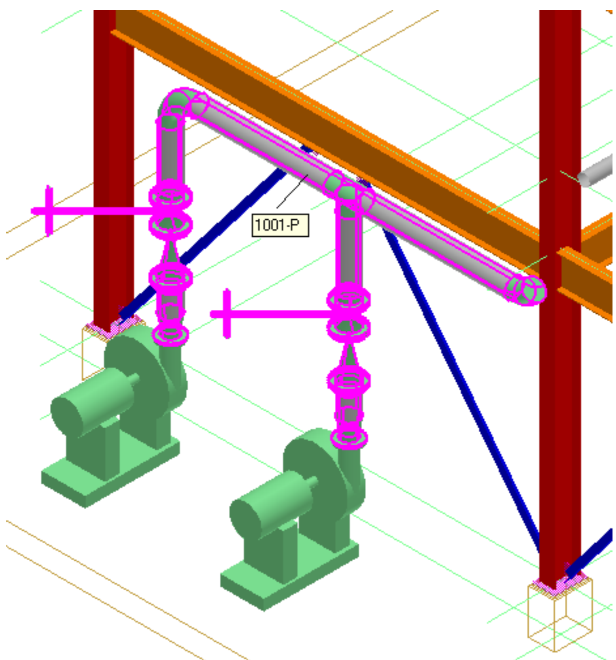


Figure 22: Select Pipe Parts in Pipeline 1001-P

25. To specify the **Active Project**, select the **More…** option in the **Active Project** drop-down list on the main toolbar.

26. Select the **Database** option to see all the WBS projects in the displayed dialog box. And then select

PJ-99, as shown in Figure 23, to specify **PJ-99** as an active project.

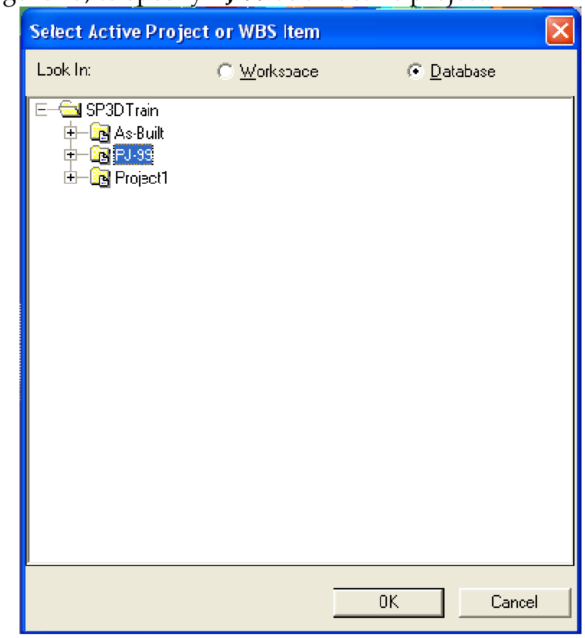


Figure 23: Select Active Project or WBS Item Dialog Box

27. Click **OK** in the dialog box. The selected active project should appear, as shown in Figure 24, on the main toolbar.

Figure 24: Active Project: PJ-99

28. Select the **Project > Claim** command, as shown in Figure 25, to associate the selected piping objects with the active project **PJ-99**.

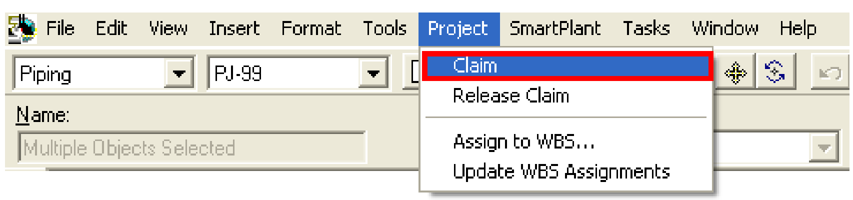


Figure 25: Claim Command

29. After the claim process is complete, SP3D displays the **Claim** dialog box, as shown in Figure 26. Click **Close** to close the dialog box.

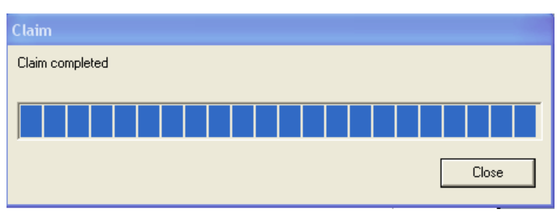


Figure 26: Claim Dialog Box

Automatically Assigning Piping Objects to the Created WBS Items:

The **Group Pipe Parts** button 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.
Figure 27 shows the label used in this session for the query upon which the grouping of objects will be based.

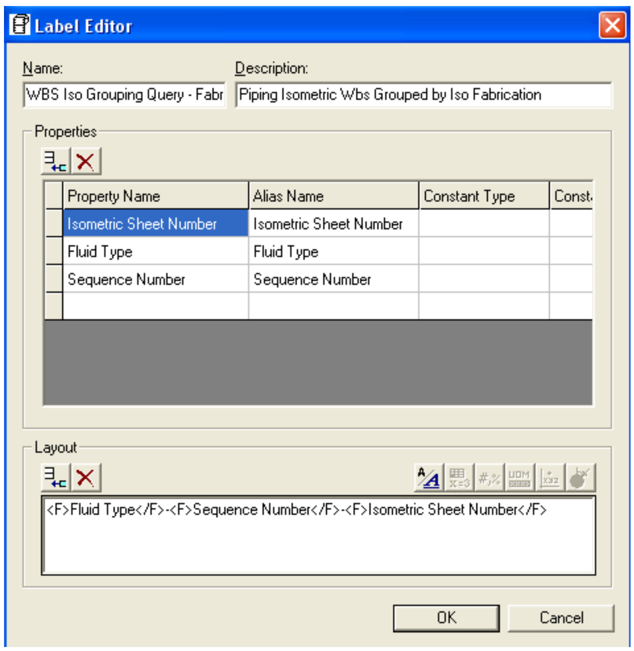


Figure 27: Label Definition

Fluid Type:	**Pipeline attribute**
Sequence Number:	**Pipeline attribute**
Isometric Sheet Number:	**Pipe Part attribute**

30. Select the piping parts of the pipeline **1001-P**, as shown in Figure 28, to assign them to isometric sheet number **A**.

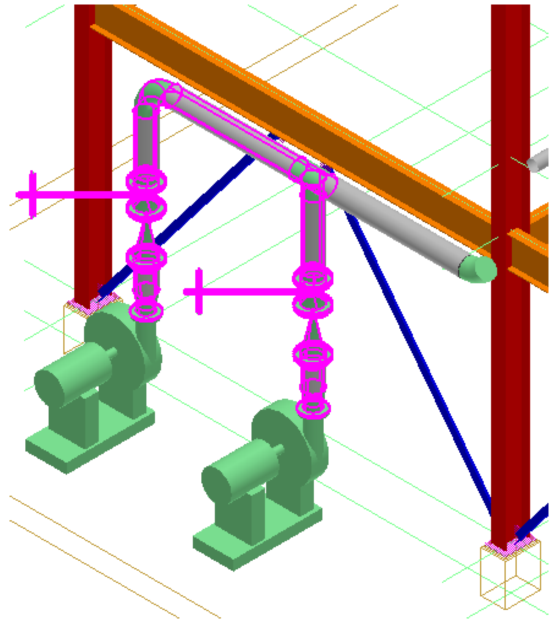


Figure 28: Selected Pipe Parts

31. 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. Key in **A** in the **Isometric Sheet Number** box in the **Selection Properties** dialog box, as shown in Figure 29, and click **OK**.

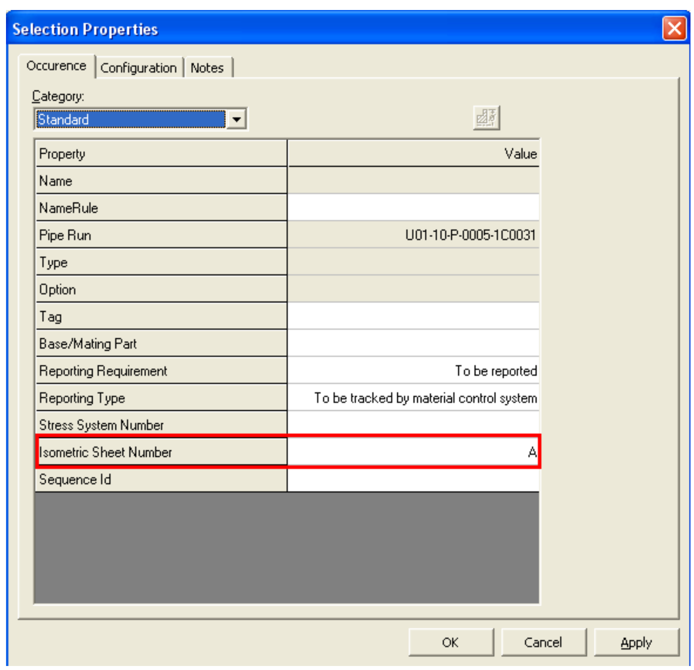


Figure 29: Selected Pipe Parts Properties

32. Now select the remaining pipe parts belonging to **1001-P**, as shown in Figure 30.

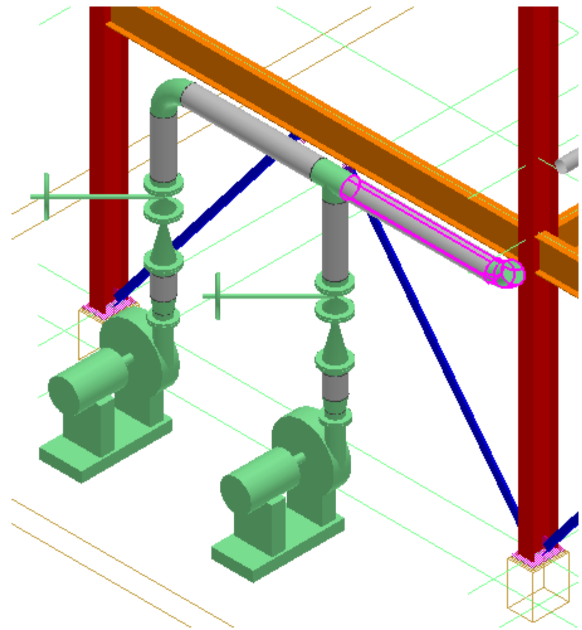


Figure 30: Selected Pipe Parts

33. 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. Key in **B** in the **Isometric Sheet Number** box in the **Selection Properties** dialog box, as shown in Figure 31, and click **OK**.

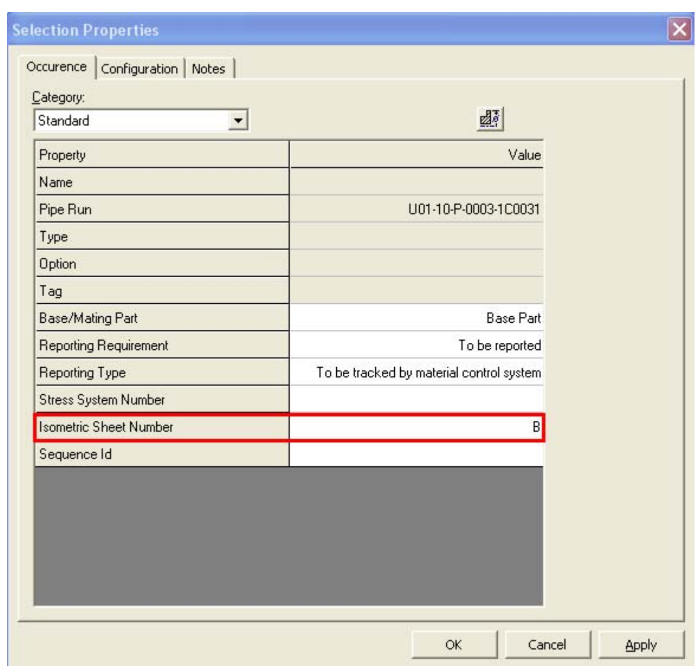


Figure 31: Selection Properties Dialog Box

34. Now to assign these pipe parts to the appropriate WBS items, select the **Group Pipe Parts** button on the vertical toolbar, as shown in Figure 32.

Figure 32: Group Pipe Parts Button on the Vertical Toolbar

35. The **Automated WBS Creation** dialog box appears. Set the following specifications in this dialog box, as shown in Figure 33, 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: U01
WBS Item's Parent: Iso Fabrication
Maximum Number Of Objects: 100
Pull In Associated Items: Connection and Reportable
Maintain Existing WBS Items?: True

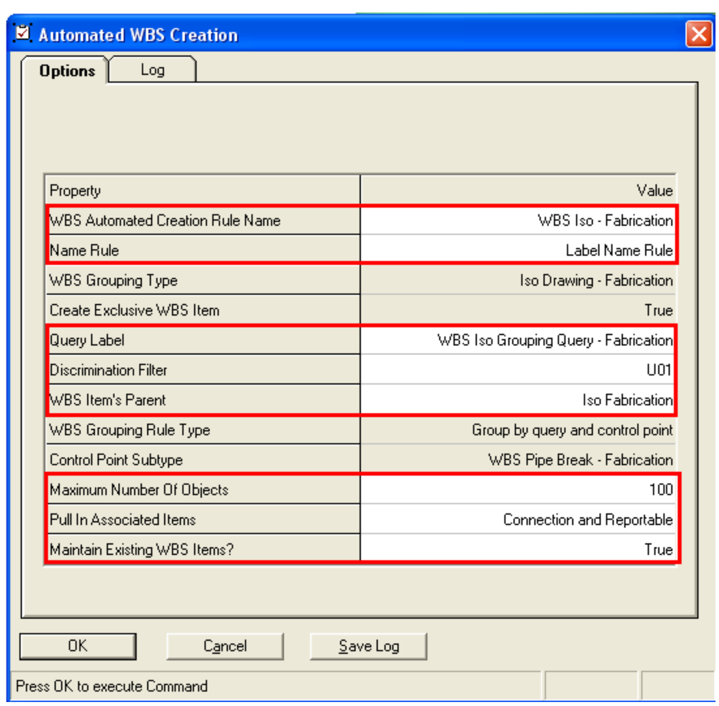


Figure 33: Automated WBS Creation Dialog Box

36. SP3D prompts you to view the log file once the process is complete. Click **No** to close the log dialog box and click **Cancel** on the **Automated WBS Creation** dialog box.

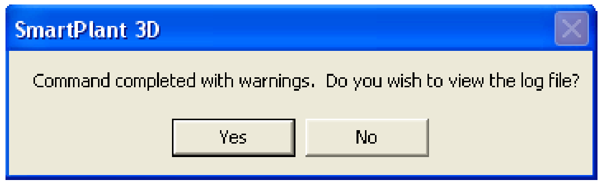


Figure 34: Automated WBS Creation log file dialog

Two WBS items are created in the WBS hierarchy. Select the **WBS** tab in the **Workspace Explorer** to view the WBS item **Iso Fabrication,** as shown in Figure 35.

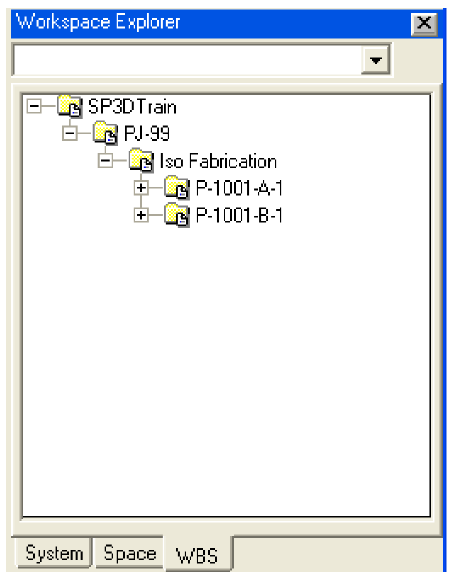


Figure 35: WBS Hierarchy on Workspace Explorer

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.

37. 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.

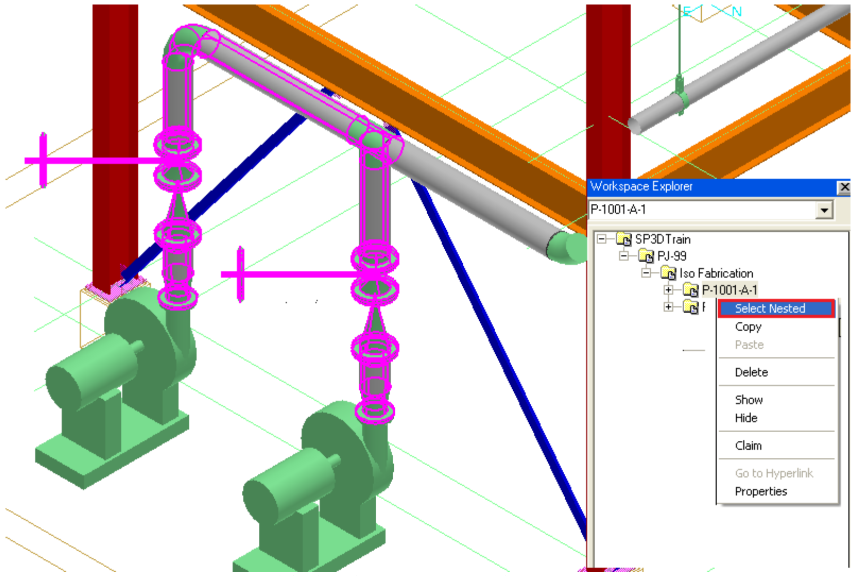


Figure 36: Highlighted Pipe Parts Belonging to P-1001-A-1

Creating Isometric Drawing from WBS items:

38. Switch to the **Drawings and Reports** from **Tasks>Drawings and Reports** 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.

39. In the **Management Console** and expand the drawing hierarchy to **Unit 01>Isometrics>Iso WBS Isometrics**, as shown in Figure 37. SP3D displays a message box. Click **OK** to continue.

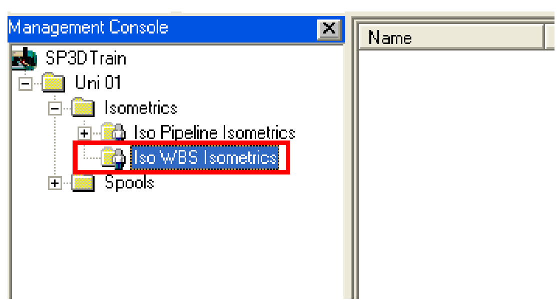


Figure 37: Management Console

40. Right-click the **Iso WBS Isometrics** component in the **Management Console** and select the **Run Query** option, as shown in Figure 38. SP3D 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.

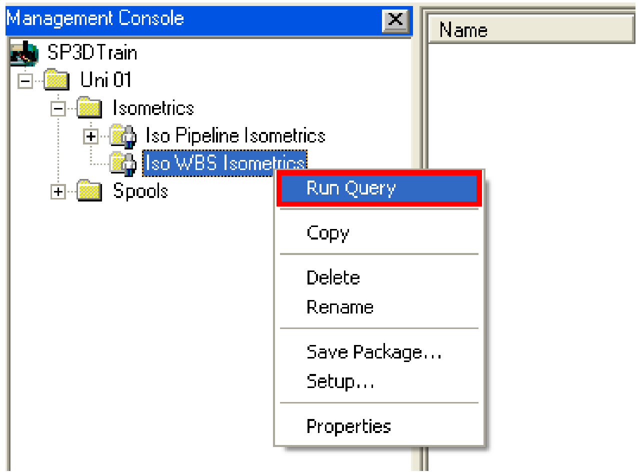


Figure 38: Run Query Option

The active project **PJ-99** and the WBS item **Iso Fabrication** appears in the hierarchy of the **Management Console**, as shown in Figure 39.

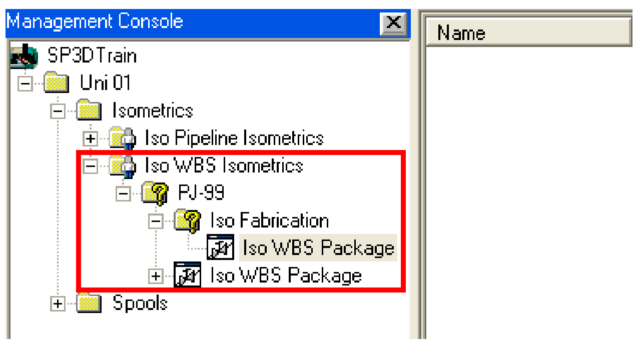


Figure 39: Drawing Management Console

41. Now right-click the **Iso WBS Package** option in the hierarchy and click the **Create Drawings** option, as shown in Figure 40, to create the isometric drawing documents.

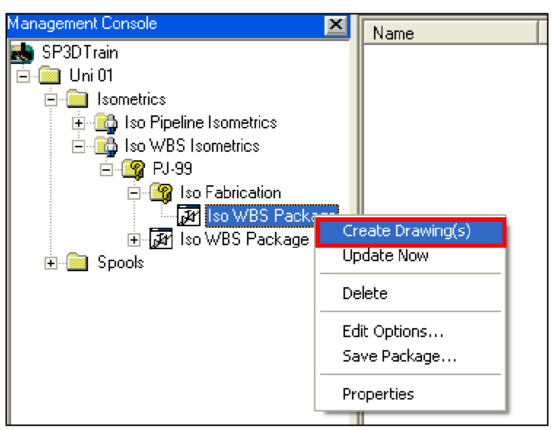


Figure 40: Create Drawing(s) Option

SP3D generates isometric drawing documents for all the WBS items available in the **Iso Fabrication** WBS parent, as shown in Figure 41.

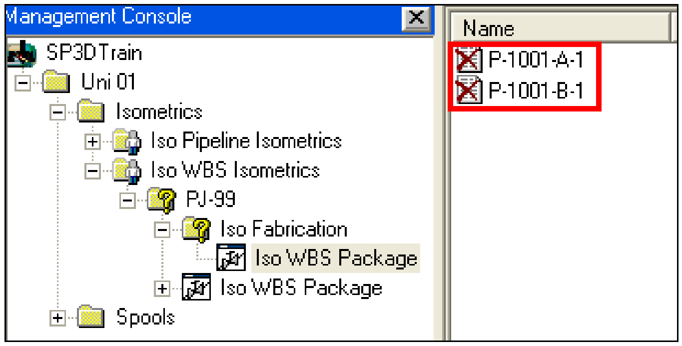


Figure 41: Isometric Drawings for the created WBS Items in the Management Console

42. Right-click the **Iso WBS Package** isometric drawing type and click the **Update Now** option, as shown in Figure 42, to update the drawings for both the WBS groups.

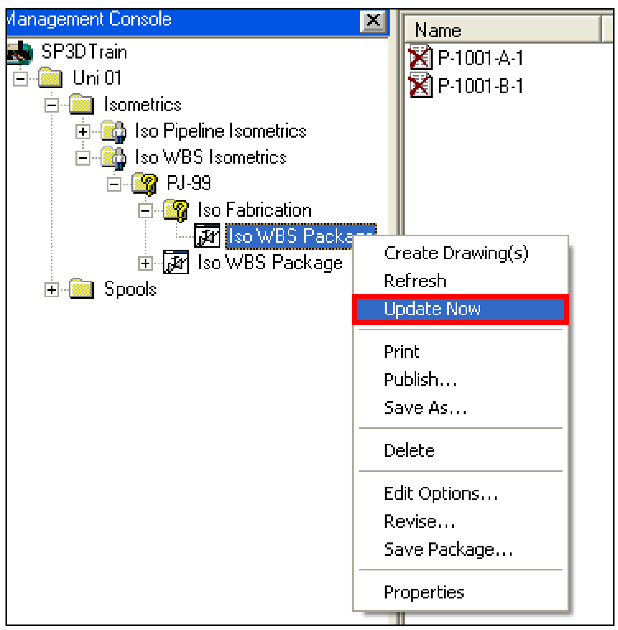


Figure 42: Updating the Drawing Documents Created from WBS Items

SP3D finds all the piping parts belonging to the created WBS items and generates the isometrics drawings for both the WBS groups **P-1001-A-1** and **P-1001-B-1**. The green check mark then appears on both the groups in the **Management Console**, as shown in Figure 43, which shows that the isometric drawing is generated.

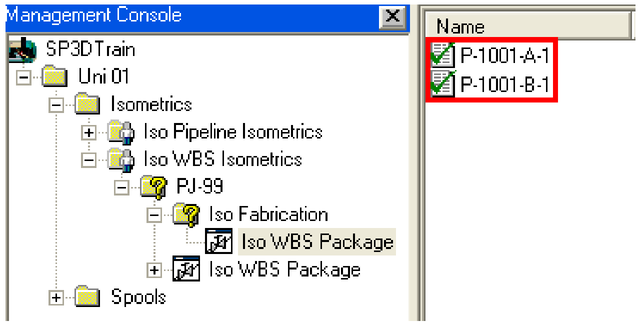


Figure 43: Successfully Created Drawings P-1001-A-1 and P-1001-B-1

43. 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 will be displayed, as shown in Figures 44 and 45.

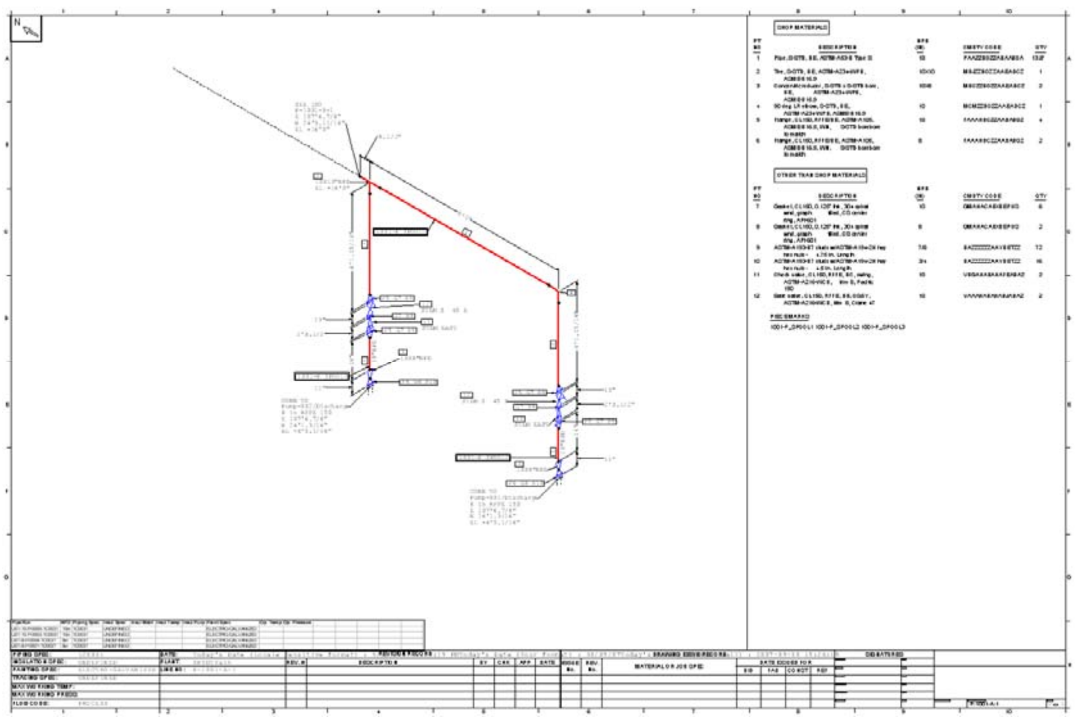


Figure 44: Isometric Drawing for P-1001-A-1

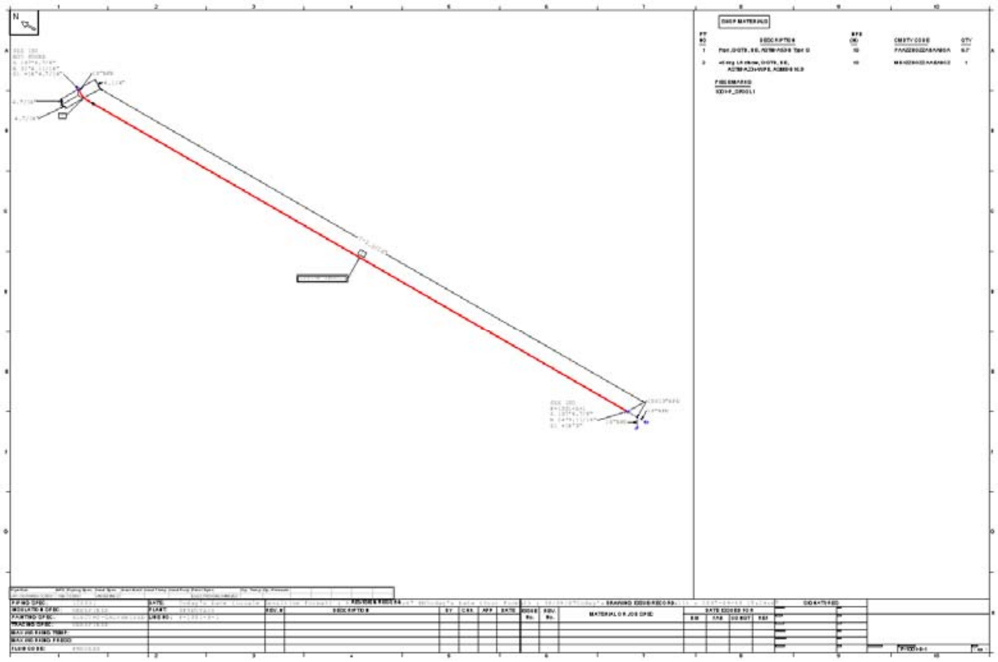


Figure 45: Isometric Drawing for P-1001-B-1

Steps for Creating Isometric Drawings from Pipeline Objects:

Exercise 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 Figure 46.

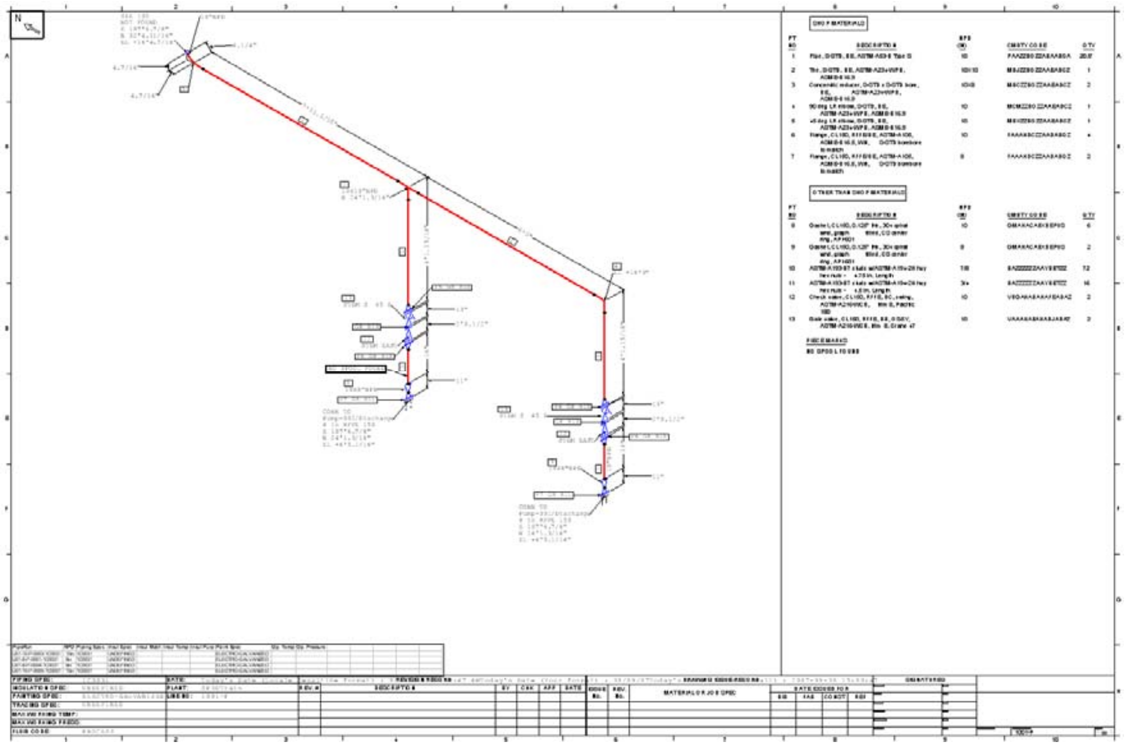


Figure 46: Isometric Drawing for Pipeline 1001-P

Before beginning the 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 Figure 47. 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.

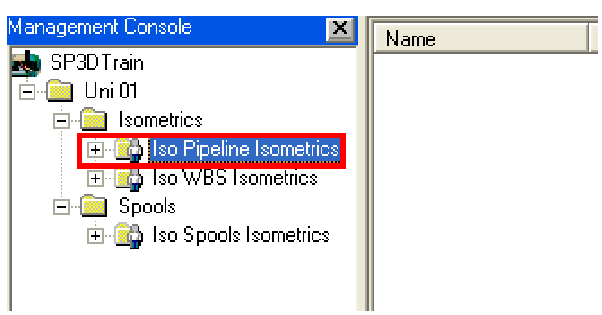


Figure 47: Management Console

1. In the **Management Console** expand the drawing hierarchy to **Unit 01>Isometrics>Iso Pipeline Isometrics**.

2. Right-click the **Iso Pipeline Isometrics** component in the **Management Console** and click the **Run Query** command, as shown in Figure 48. SP3D 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.

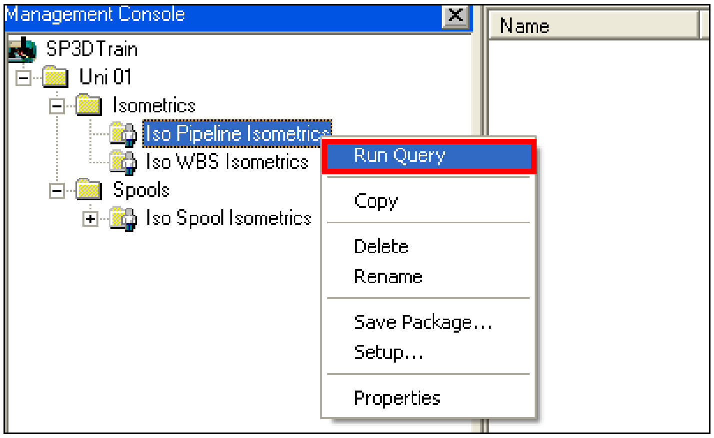


Figure 48: Run Query Option

3. 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, as shown in Figure 49.

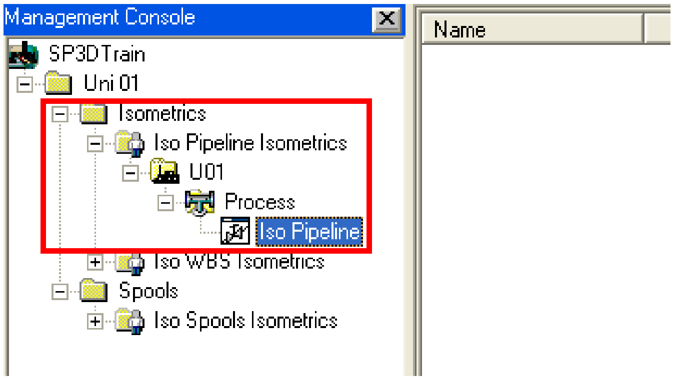


Figure 49: Piping Hierarchy in Management Console

Right-click the **Iso Pipeline** component in the hierarchy and click the **Create Drawings** command, as shown in Figure 50, to create the isometric drawing documents.

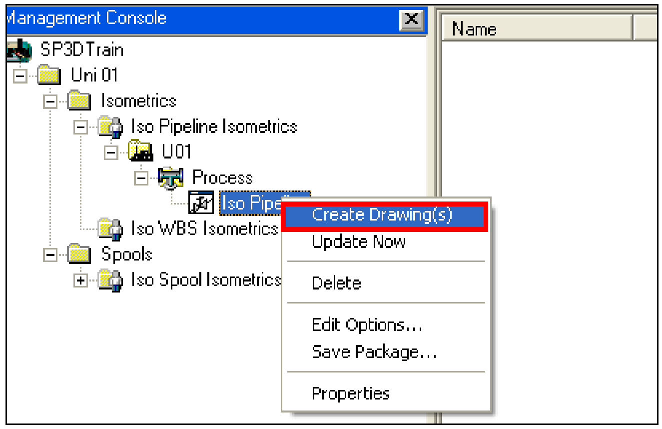


Figure 50: Create Drawing(s) Option

4. SP3D generates isometric drawing documents for all the pipelines available in Unit **U01**, as shown in Figure 51.

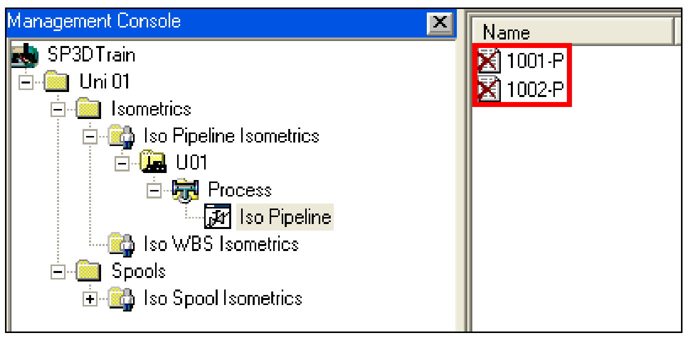


Figure 51: Isometric Drawings for Pipelines in U01 in the Management Console

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.

Right-click the isometric drawing document of **1001-P** and click the **Update Now** command, as shown in Figure 52, to update the drawing.

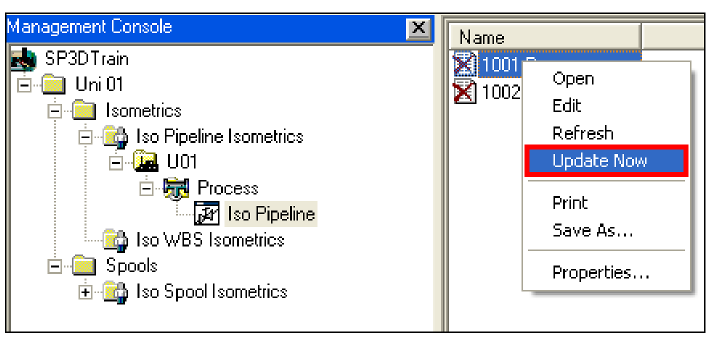


Figure 52: Updating the 1001-P Drawing Document

5. SP3D gets the piping parts and generates the isometric drawing of the pipeline **1001-P**. A green

check mark appears on **1001-P**, as shown in Figure 53, which shows that the isometric drawing for **1001-P** has been generated**.**

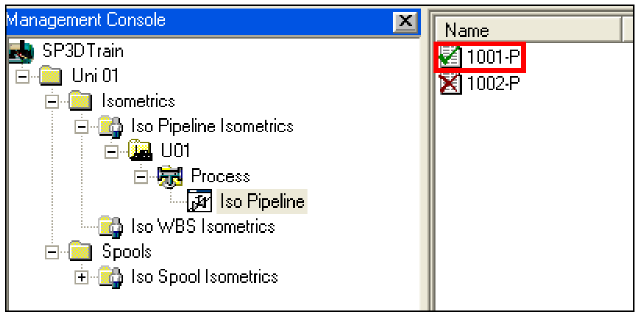


Figure 53: Created Successfully Drawing 1001-P

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 pictorial representation of the isometric drawings is shown in Figure 54.

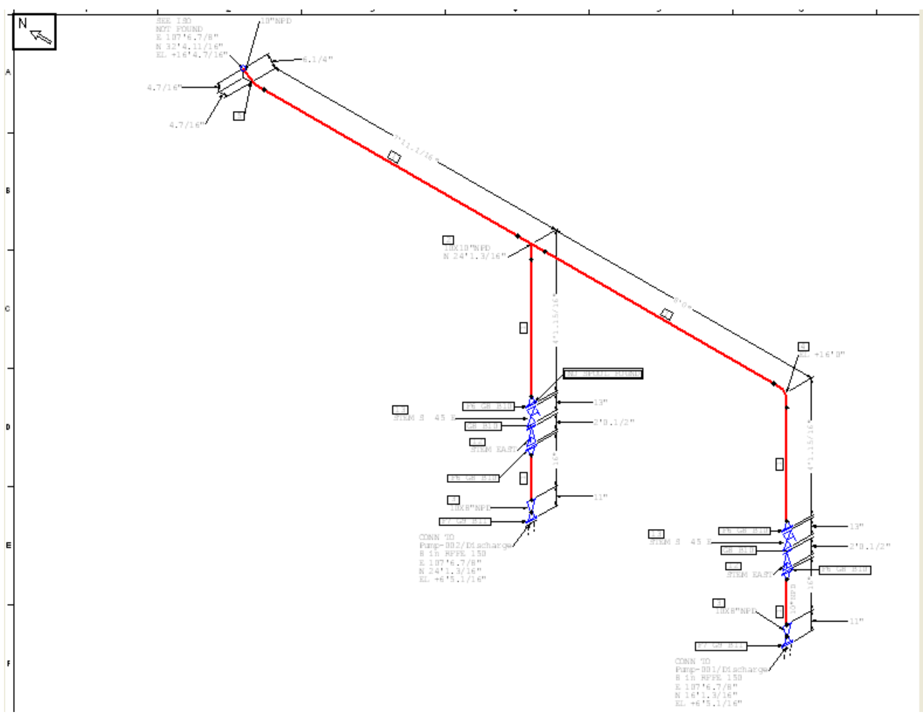


Figure 54: Isometric Drawing for Pipeline 1001-P

The drawing viewer also displays the material take-off sheet for **1001-P**, as shown in Figure 55**.** This sheet displays the details of the material required for **1001-P**.

8 9 10

SHOP MATERIALS

PT NO	DESCRIPTION	NPD (IN)	CMDTY CODE	QTY
1	Pipe, S-STD, BE, ASTM-A53-B Type S	10	PAAZZBOZZABAABOA	20.5'
2	Tee, S-STD, BE, ASTM-A234-WPB, ASME-B16.9	10X10	MDJZZBOZZAAEADCZ	1
3	Concentric reducer, S-STD x S-STD bore, BE, ASTM-A234-WPB, ASME-B16.9	10X8	MBCZZBOZZAAEADCZ	2
4	90 deg LR elbow, S-STD, BE, ASTM-A234-WPB, ASME-B16.9	10	MCMZZBOZZAAEADCZ	1
5	45 deg LR elbow, S-STD, BE, ASTM-A234-WPB, ASME-B16.9	10	MBXZZBOZZAAEADCZ	1
6	Flange, CL150, RFFE/BE, ASTM-A105, ASME-B16.5, WN, S-STD borebore to match	10	FAAAHDCZZAADABQZ	4
7	Flange, CL150, RFFE/BE, ASTM-A105, ASME-B16.5, WN, S-STD borebore to match	8	FAAAHDCZZAADABQZ	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 rng, API-601	10	GMAHACABXBEPUS	6
9	Gasket, CL150, 0.125" thk, 304 spiral wnd, graph filled, CS center rng, API-601	8	GMAHACABXBEPUS	2
10	ASTM-A193-B7 studs w/ASTM-A194-2H hvy hex nuts - 4.75 in. Length	7/8	BAZZZZZZAAYBETZZ	72
11	ASTM-A193-B7 studs w/ASTM-A194-2H hvy hex nuts - 4.5 in. Length	3/4	BAZZZZZZAAYBETZZ	16
12	Check valve, CL150, RFFE, BC, swing, ASTM-A216-WCB, trim 8, Pacific 180	10	VBGAHABAHAFEADAZ	2
13	Gate valve, CL150, RFFE, BB, OS&Y, ASTM-A216-WCB, trim 8, Crane 47	10	VAAAHABAHADJADAZ	2

PIECE MARKS

NO SPOOL FOUND

A B C D

Figure 55: Material Take-Off for the Pipeline 1001-P

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.

Steps for Updating Isometric Drawings:

Exercise 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 will resemble Figure 56 and 57.

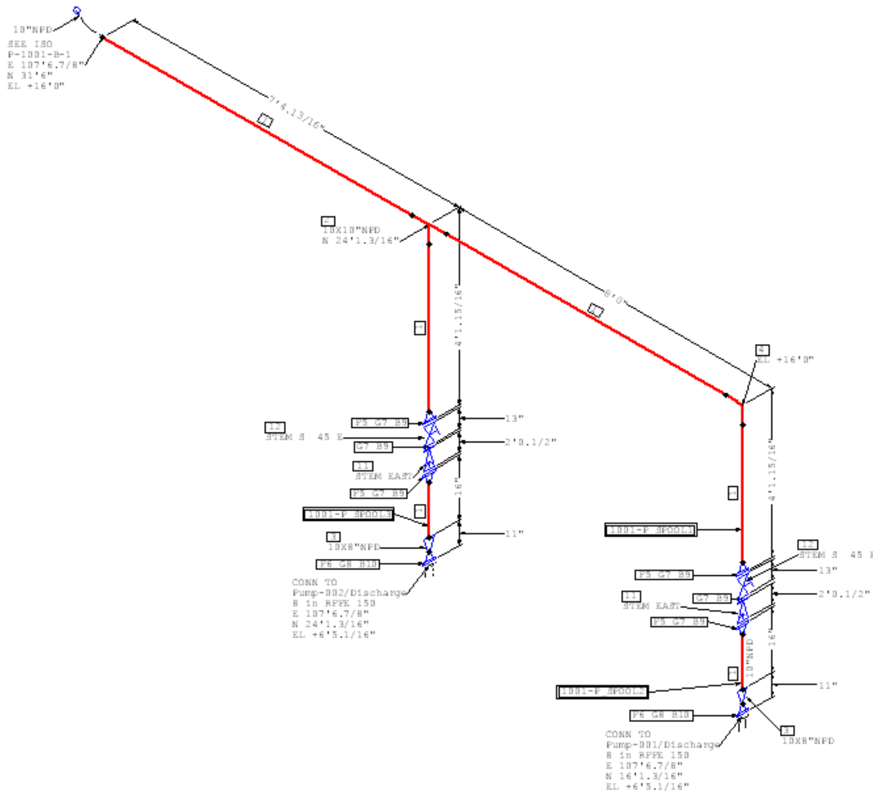


Figure 56: Isometric Drawing from WBS P-1001-A-1

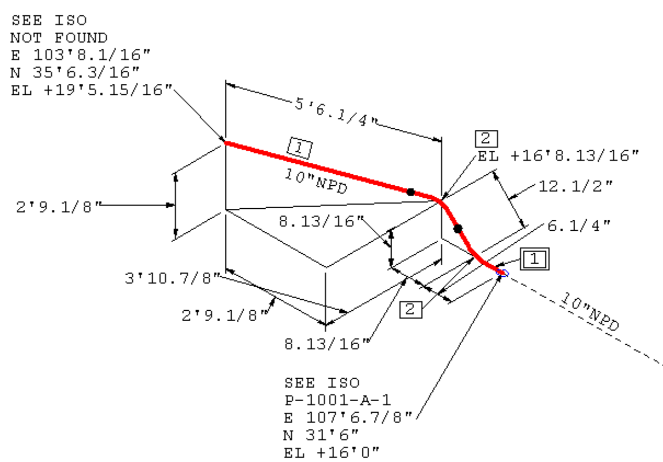


Figure 57: Isometric Drawing from WBS P-1001-B-1

1. Switch to the **Piping** task from the **Tools>Piping** command to be able to work in the piping environment.

2. 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. 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** command. After inserting the elbow and routing the pipe the view of the model should resemble Figure 58.

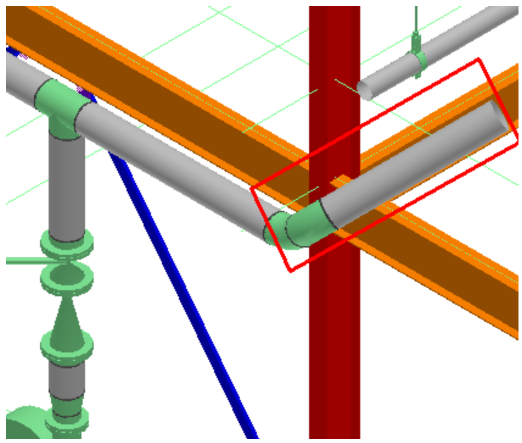


Figure 58: Placed Elbow and Routed Pipe

3. Make sure the **Locate Filter** is set to **Piping Parts**. Now select the 45 deg elbow and pipe you have placed, as shown in Figure 59.

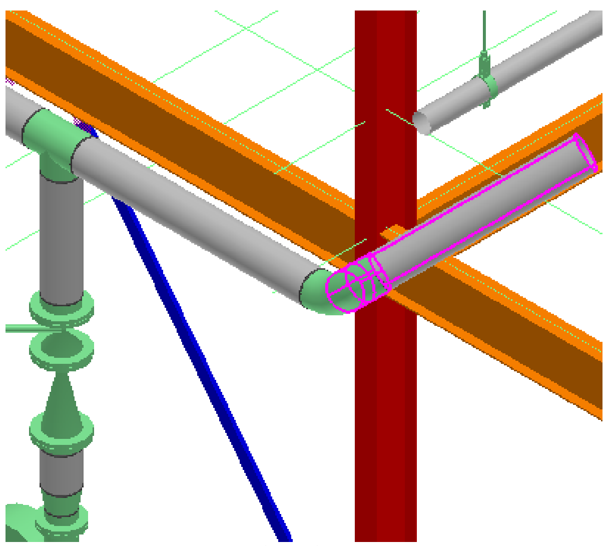


Figure 59: Selected Pipe Parts

4. 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, as shown in Figure 60, and click **OK**.

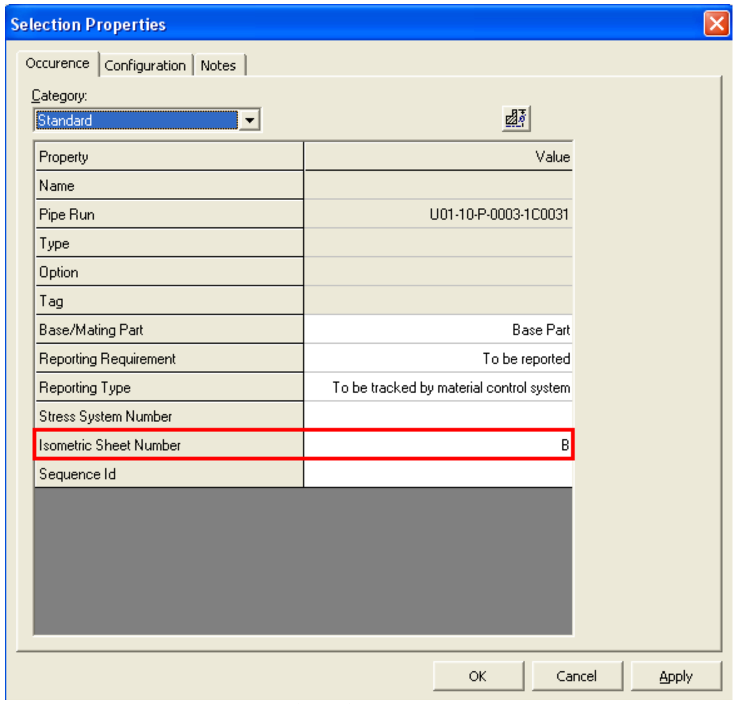


Figure 60: Selected Pipe Parts Properties

5. Select the pipe, as shown in Figure 61.

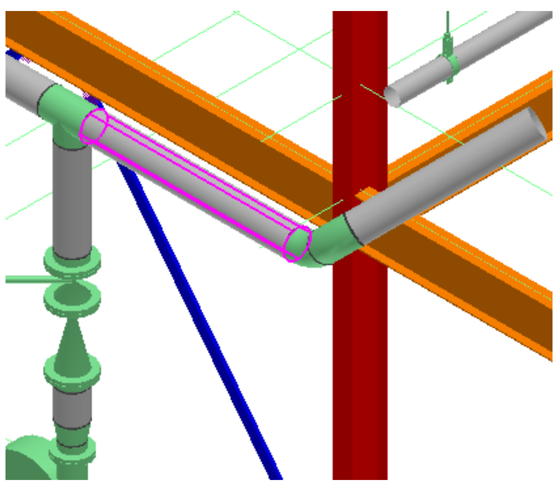


Figure 61: Selected Pipe.

6. 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, as shown in Figure 62, and click **OK**.

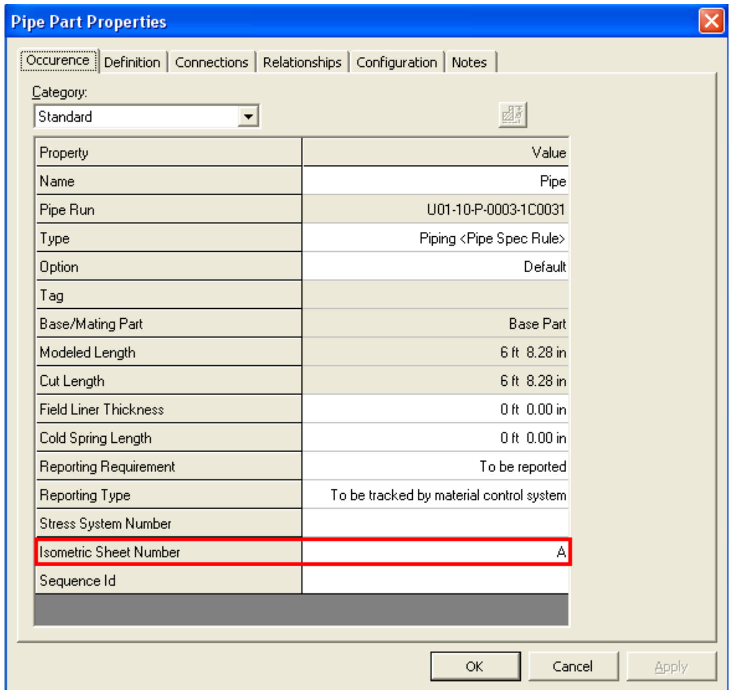


Figure 62: Pipe Part Properties

7. Now to assign these pipe parts to the appropriate WBS items, click the **Group Pipe Parts** button on the vertical toolbar.

8. The **Automated WBS Creation** dialog box appears. 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: U01
WBS Item's Parent: Iso Fabrication
Maximum Number Of Objects: 100
Pull In Associated Items: Connection and Reportable
Maintain Existing WBS Items?: False

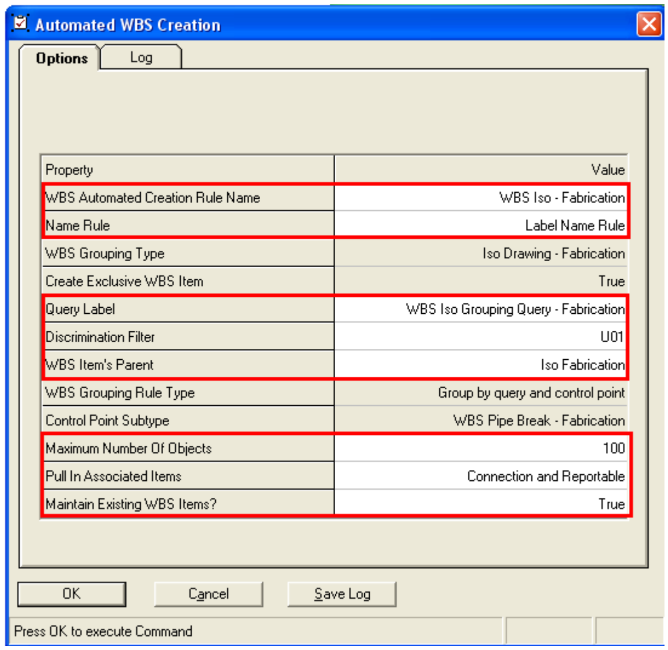


Figure 63: Automated WBS Creation Dialog Box

9. SP3D prompts you to view the log file once the process is complete. Click **No** to close the log dialog box and click **Cancel** on the **Automated WBS Creation** dialog box.

10. Switch to the **Drawings and Reports** task to update the isometric drawings for the created WBS items.

11. In the **Management Console**, expand the drawing hierarchy to **Unit 01>Isometrics>Iso WBS Isometrics > PJ-99> Iso Fabrication > Iso WBS Package**, as shown in Figure 64.

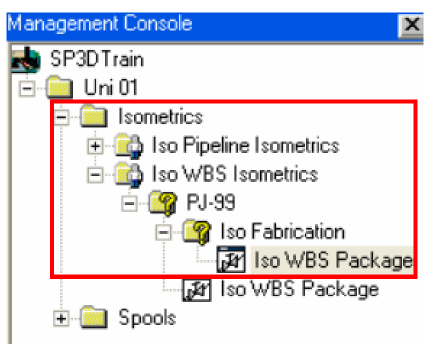


Figure 64: Iso WBS Package in Management Console

12. Right-click the **Iso WBS Package** isometric drawing type and click the **Update Now** option, as shown in Figure 65, to update the isometric drawings.

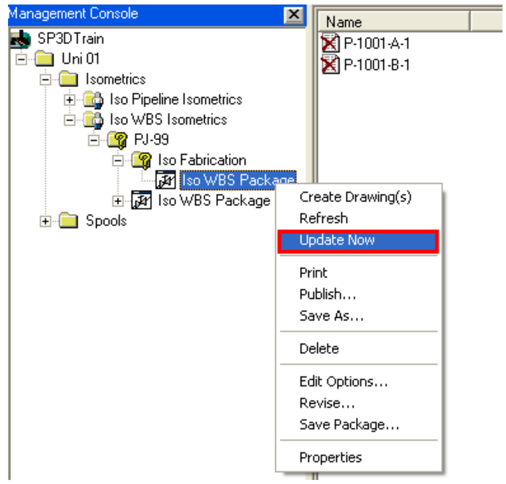


Figure 65: Updating Drawing Documents

13. After the updation is complete, double-click the isometric drawings created for the WBS items one by one. A pictorial representation of both isometrics will be displayed, as shown in Figures 66 and 67.

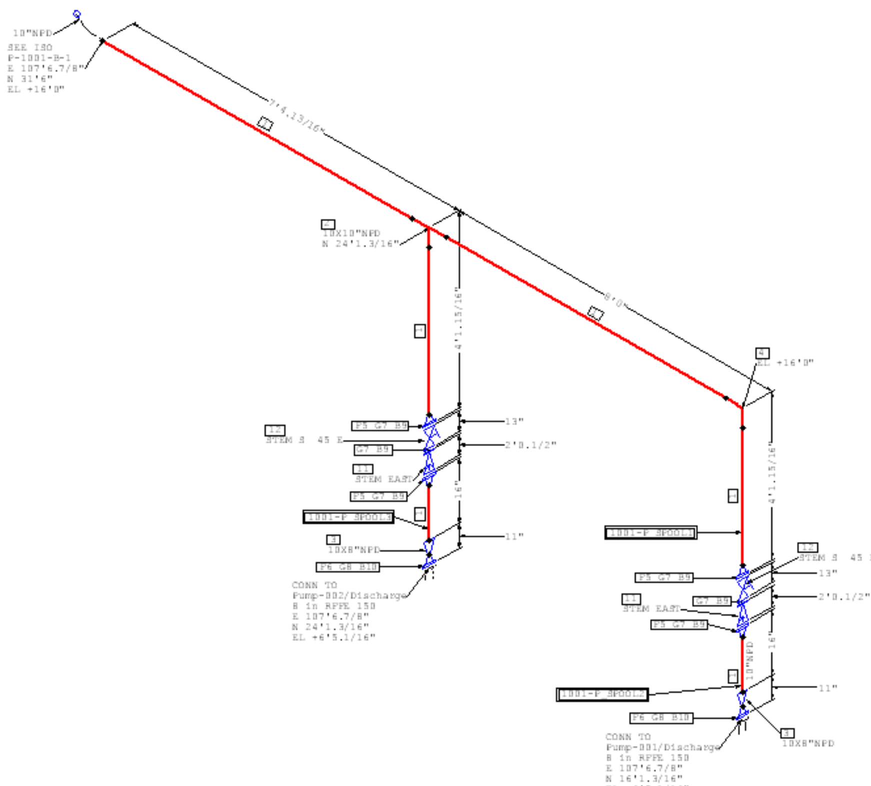


Figure 66: Isometric Drawing from WBS P-1001-A-1

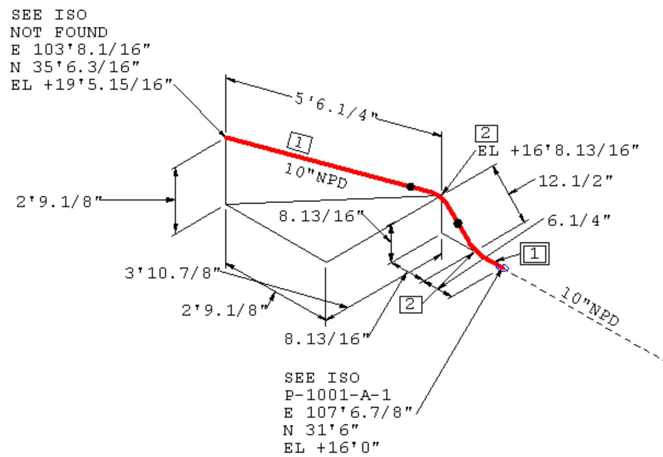


Figure 67: Isometric Drawing from WBS P-1001-B-1

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.

Steps for Changing the Status of WBS Items:

Exercise Objective: In this exercise you will be changing the status of the WBS item **PJ-99-01**to **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 beginning the 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 from the **Tools>Piping** command.

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, as shown in Figure 68, to change the properties of the WBS item **PJ-99-01.**

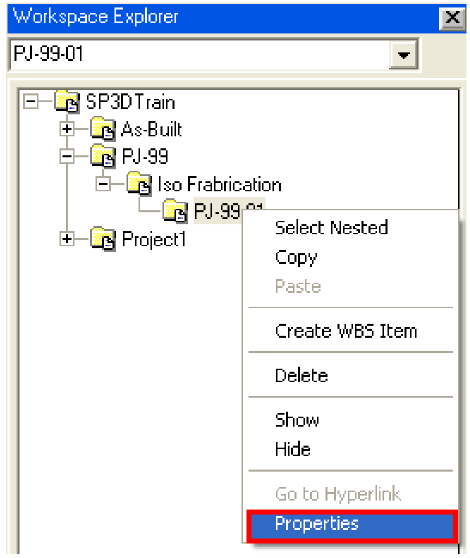


Figure 68: Properties Option for the **PJ-99-01** *WBS Item*

4. The **WBS Item Properties** dialog box appears. On the **Configuration** tab, select the **Approved** option in the **Status** drop-down list and click **OK**.

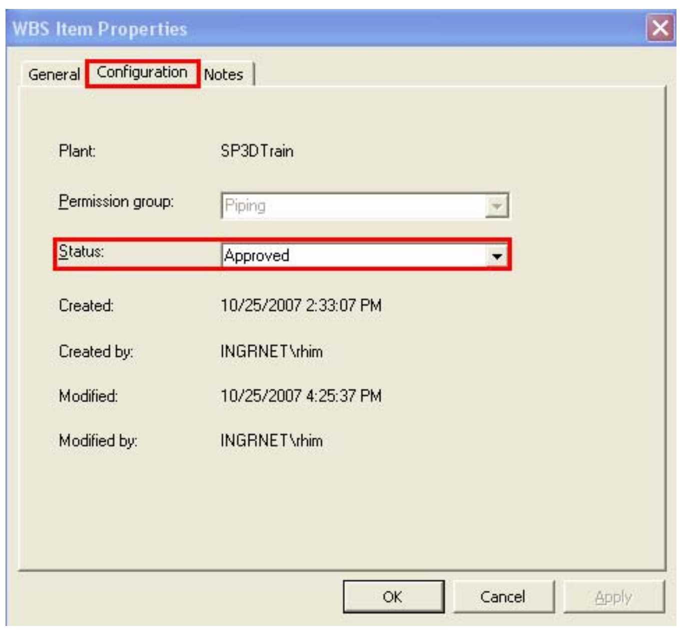


Figure 69: WBS Item Properties Page

5. Now place a temperature indicator **TI-1504** on the pipeline **403-P**, as shown in figure 70.

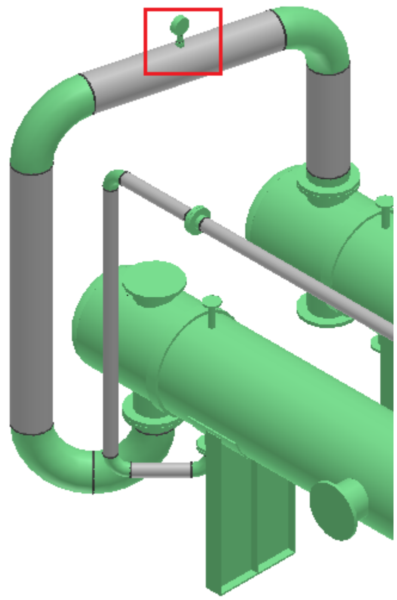


Figure 70: Temperature Indicator on 403-P Pipeline

6. In the **Workspace Explorer** window select the **System** tab. Expand **A2>U04>Process** and select the pipeline system **403-P**. The pipeline **403-P** will be highlighted in the graphic view.

7. Now select the **Project** > **Assign to WBS…** command. The **Assign to WBS** dialog box appears.

8. In the **Assign to WBS** dialog box expand **PJ-99** folder and select **PJ-99-01**, as shown in Figure 71. Then click **OK.** A message is displayed stating that the object you are trying to assign is in non-working status. This message indicates that you cannot assign piping parts to a WBS item in non-working status.

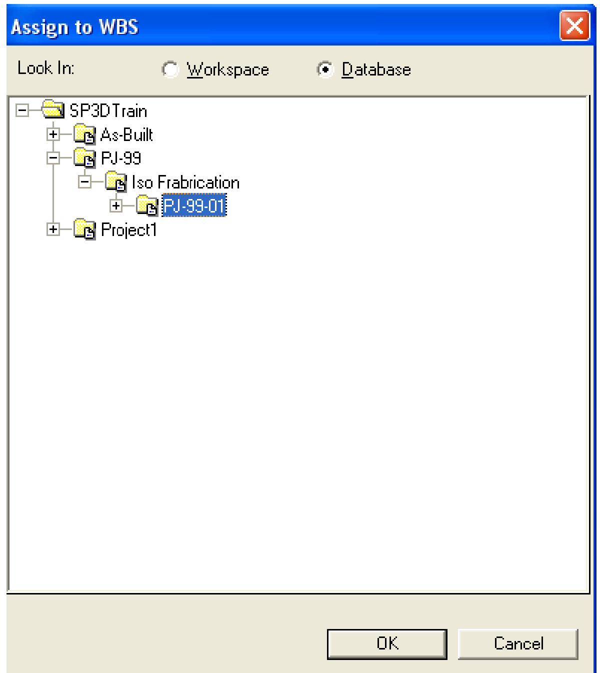


Figure 71: Assign To WBS Dialog Box

9. Select the **Piping Parts** option in the **Locate Filter** drop-down list and select the temperature indicator **TI-1504** in the graphic view.

10. Right-click the temperature indicator **TI-1504** and open the **Pipe Instrument Properties** dialog box.

11. The **WBS Item** record is not shown on the **Relationships** tab on the **Pipe Instrument Properties** dialog box.

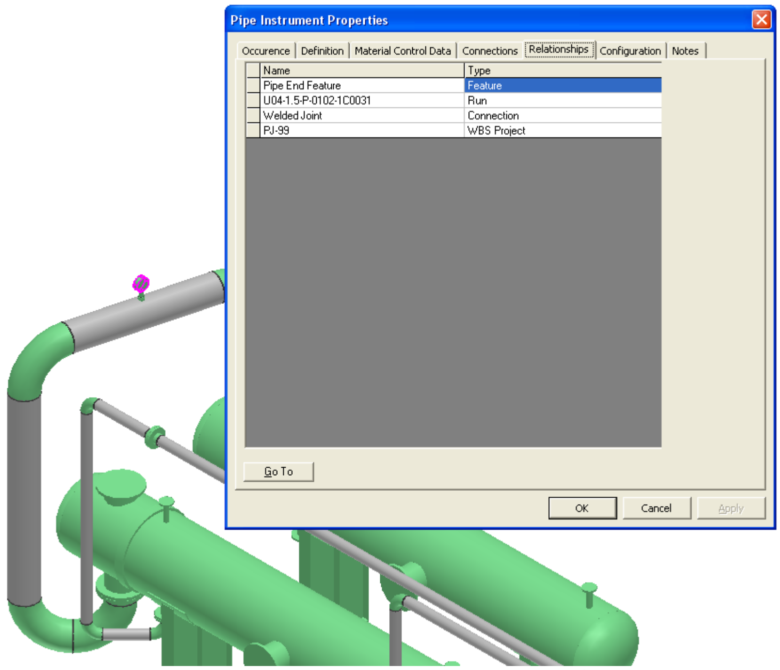


Figure 72: Pipe Instrument Properties Dialog Box

Resolving Isometric Extraction Problem:

Since 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, as shown in Figure 73.

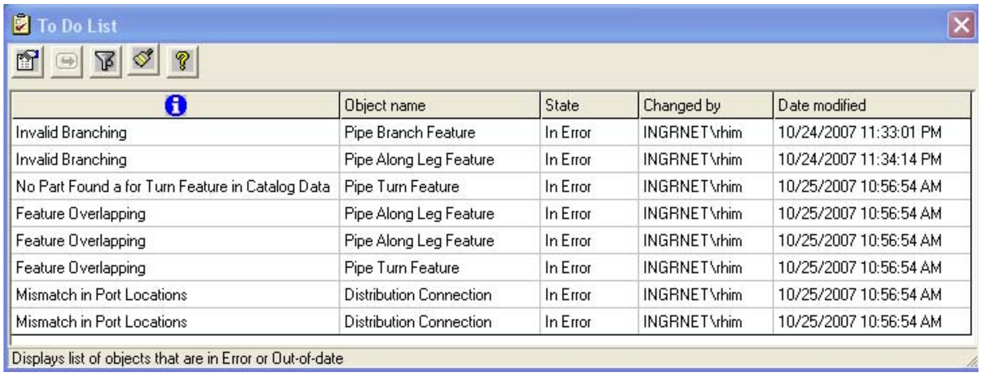


Figure 73: To Do List Dialog Box

Refer to *Session 11: To Do List* of *SP3D 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.

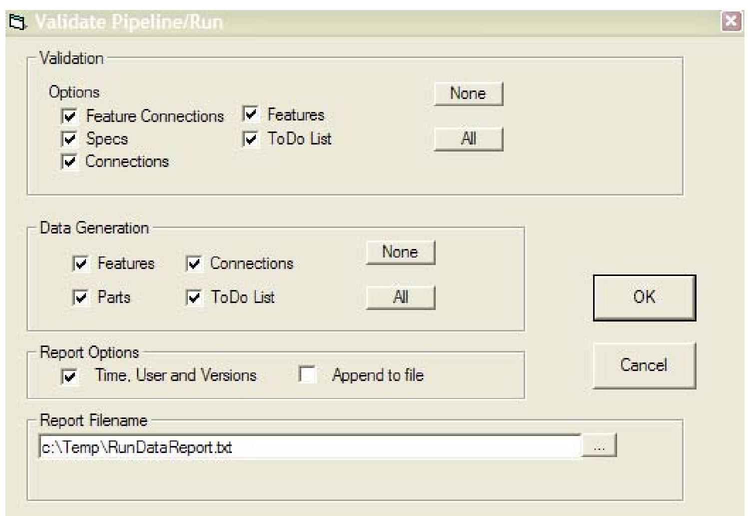


Figure 74: Validate Pipeline/Run Dialog Box

- **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.

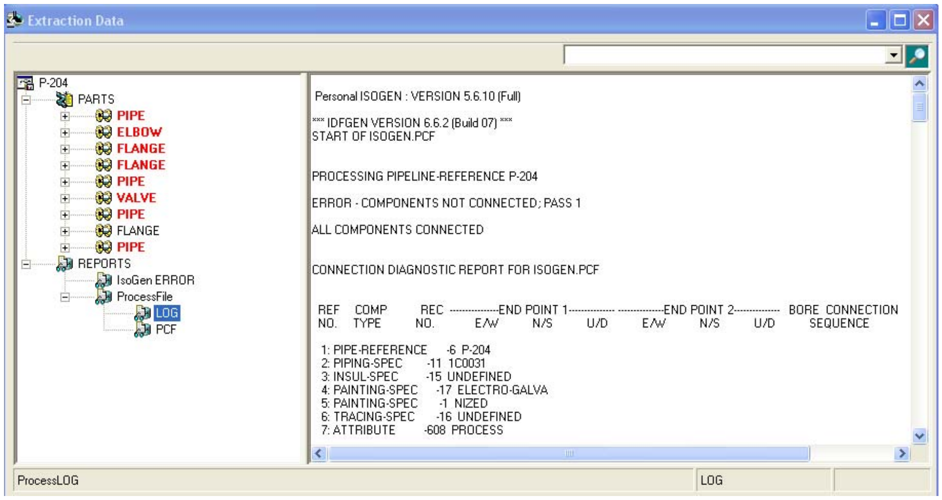


Figure 75: View Extraction Data Dialog Box

- **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.

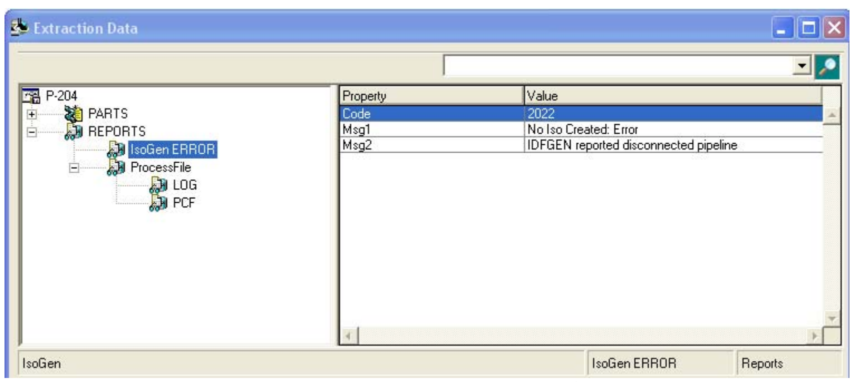


Figure 76: IsoGen Error Log File

- **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.

Figure 77: Personal ISOGEN Returns an Error Number

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.*