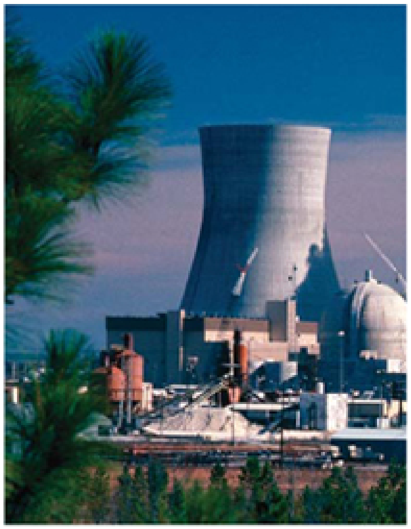
Piping

User's Guide

Process, Power & Marine



Version 6.0, Service Pack 1 September 2005 DSP3D-PE-200025E

Copyright

Restricted Rights Legend

Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c) of the *Contractor Rights in Technical Data* clause at DFARS 252.227-7013, subparagraph (b) of the *Rights in Computer Software or Computer Software Documentation* clause at DFARS 252.227-7014, subparagraphs (b)(1) and (2) of the *License* clause at DFARS 252.227-7015, or subparagraphs (c) (1) and (2) of *Commercial Computer Software---Restricted Rights* at 48 CFR 52.227-19, as applicable.

Unpublished---rights reserved under the copyright laws of the United States.

Intergraph Corporation
Huntsville, Alabama 35894-0001

Warranties and Liabilities

All warranties given by Intergraph Corporation about equipment or software are set forth in your purchase contract, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties. Intergraph believes the information in this publication is accurate as of its publication date.

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

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license.

No responsibility is assumed by Intergraph for the use or reliability of software on equipment that is not supplied by Intergraph or its affiliated companies. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Trademarks

Intergraph, the Intergraph logo, SmartSketch, FrameWorks, SmartPlant, IntelliShip, INtools, PDS, and MARIAN are registered trademarks of Intergraph Corporation. Microsoft and Windows are registered trademarks of Microsoft Corporation. ISOGEN is a registered trademark of Alias Limited. ACIS is a registered trademark of SPATIAL TECHNOLOGY,INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Portions of 2D DCM, 3D DCM, and HLM from D-Cubed Limited are incorporated. All rights reserved. Other brands and product names are trademarks of their respective owners.

Table of Contents

Preface

This document is a user's guide for the SmartPlant® 3D Piping task and provides command reference information and procedural instructions.

SmartPlant 3D Documentation Set

The SmartPlant® 3D documentation set is available as Adobe® PDF files. The content of the PDF files is the same content as online Help. To access these PDF documents in the software, click **Help > Printable Guides**.

The documentation set is divided into four categories:

- Administrative guides contain information about installing, configuring, customizing, and troubleshooting SmartPlant 3D.
- User's guides provide command reference and how-to information for working in each SmartPlant 3D task.
- Reference data guides define the reference data workbooks. Not all tasks have reference data.
- Third-party guides from other vendors for software that works with SmartPlant 3D.

Administrative Guides

SmartPlant 3D Installation Guide - Provides instructions on installing and configuring the software on both the client and server computers.

SmartPlant 3D/IntelliShip Programmer's Guide - Provides information about custom commands, naming rules, and symbol programming.

SmartPlant 3D Reference Data Guide - Provides instructions about the Bulkload utility, codelists, and the reference data common to several disciplines.

SmartPlant 3D Symbols Reference Data Guide - Provides information about the Visual Basic Part Definition Wizard and the three-dimensional symbols used in all tasks.

SmartPlant 3D The Engineering Framework Reference Guide - Provides information about installing, configuring, and using The Engineering Framework with SmartPlant 3D.

SmartPlant 3D Troubleshooting Guide- Provides information on how to resolve errors that you may encounter in the software by documenting troubleshooting tips, error messages, and to do list messages.

User's Guides

Catalog User's Guide - Provides information about viewing, editing, and creating reference data and select lists (codelists).

Common User's Guide - Provides information about defining workspaces, navigating in the model, precision input, filtering, manipulating views, and running reports.

Drawings and Reports User's Guide - Provides information about creating drawing and report deliverables.

Electrical User's Guide - Provides information about routing electrical cable, cableway, cable tray, and conduit.

Equipment and Furnishings User's Guide - Provides information about placing equipment.

Grids User's Guide - Provides instructions for creating coordinate systems, elevation grid planes, vertical grid planes, radial cylinders, radial planes, grid arcs, and grid lines.

Hangers and Supports User's Guide - Provides instructions on placing piping, duct, cableway, and conduit supports in the model.

HVAC User's Guide - Provides instructions for routing HVAC duct.

Piping User's Guide - Provides instructions for routing pipe and placing valves, taps, and pipe joints.

Project Management User's Guide - Provides instructions for setting up the databases, creating permission groups, backing up and restoring project data, assigning access permissions to the model, managing interference detection, defining and managing locations for Global Workshare, controlling duplication and consolidation of plants, tools for synchronization, regeneration of report databases, and version upgrade.

Space Management User's Guide - Provides instructions for placing volumes (such as drawing volumes, obstruction zones) in the model.

Structural Analysis User's Guide - Provides instructions for defining loads, load cases, load combinations, and the importing and exporting of analytical data.

Structure User's Guide - Provides instructions for placing structural members such as: beams, columns, braces, slabs, openings, stairs, ladders, equipment foundations, and handrails.

Systems and Specifications User's Guide - Provides instructions for creating systems and their hierarchies and selecting which specifications are available for each system type.

SmartPlant 2D Symbols User's Guide - Provides instructions for creating cross section symbols.

Reference Data Guides

Drawings and Reports Reference Data Guide - Provides information about reports reference data.

Electrical Reference Data Guide - Provides information about electrical cable, cableway, cable tray, and conduit reference data.

Equipment and Furnishings Reference Data Guide - Provides information about equipment reference data and name rules.

Hangers and Supports Reference Data Guide - Provides information about hangers and supports reference data.

HVAC Reference Data Guide - Provides information about HVAC reference data.

Piping Reference Data Guide - Provides information about piping reference data including piping specifications, piping specification rules, piping parts, piping symbols, and name rules.

Space Management Reference Data Guide - Provides information about space management reference data.

Structure Reference Data Guide - Provides information about structural reference data and name rules.

Third-Party Guides

AText Reference Guide - Provides information about alternative text for isometric drawings. This guide is from Alias, the makers of ISOGEN®.

Option Switches Reference Guide - Provides information about the ISOGEN option switches for isometric drawings. This guide is from Alias, the makers of ISOGEN.

Symbol Keys Reference Guide - Provides information about the symbol keys for isometric drawings. This guide is from Alias, the makers of ISOGEN.

Documentation Comments

Send documentation comments or suggestions to PPMdoc@intergraph.com.

What's New in Piping

The following changes have been made to the Piping task.

Version 6.0

- Weld and spool names can now be sequenced by flow direction or topology. For more information, see Sequence Objects Command.
- Spooling options always default to those defined in the Catalog with interactive setting changes persisting for the current session only. For more information, see *Spooling: An Overview*, page 143.
- Non-radial branching is supported using either the **Insert Component** command or the **Route Pipe** command. You can create tangential branches with reinforcing welds, reinforcement pads, or non-radial weldolets. For more information, see *Insert Component Command*, page 122 or the *Route Pipe Command*, page 30.
- You can now stop heat-tracing, insulation, or a surface coating at an arbitrary location along the pipe using a feature break instead of at a weld or other break in the line. For more information, see *Inserting Splits: An Overview*, page 109.
- The software handles permission groups and routing differently. For more information, see *Permission Groups and Routing: An Overview*, page 16.
- You can now insert a tap on a pipe. For more information, see *Insert Tap Command*, page 138.
- You can now route pipes by cardinal points instead of the pipe centerline. Cardinal points support routing by invert elevation for use in modeling underground piping. For more information, see *Set Offset Reference Dialog Box*, page 38.
- The software will now allow you to place multiple adjacent reducers either manually or automatically by reference data rules.
- You can now display NPD units in the properties dialog boxes and **New Pipe Run** dialog box using session file units and the NPD conversion table or the native pipe run NPD units. The option is on the **Tools > Options > Units of Measure** tab.
- Mating parts can be changed to base parts using either the ribbon or the property page.
- The software can check pipe bends as they are modeled to ensure that they have adequate lengths for fabrication on an allocated bending machine.
- You can now break spools at user defined control points.
- The new Project menu is available in the Piping task.

- The Connections/Welds locate filter has been separated to two filters, one for connections and one for welds.
- Name rules are now implemented.

Piping: An Overview

The Piping task is used to model distributed pipelines in your model using a point-by-point design method. Using the Piping task, you can create a fully rendered three-dimensional model of the various pipelines in your model. You also can use this task to insert piping components, instruments, and splits during design and then spool the pipe. You can start the Piping task by clicking **Tasks** > **Piping**.

Modeling of pipelines is aided by the piping specification, which limits and sometimes automatically selects piping parts. Within a particular design context or pipeline service, the specification author makes decisions in advance relating to both allowed parts and requirements for the parts that may be used in that service. Limiting the selection of parts through the use of a piping specification helps the designer by eliminating the need to make decisions related to applicability, cost, procurement, and safety of particular parts within particular pipeline services.

Part selection is further aided through the provision of rules regarding the usage of particular types of parts in particular design situations. The piping specification contains a grouping of piping materials classes that define the requirements, characteristics, and behavior of the piping commodities for a specific service. For more information on the piping reference data and piping specifications, see the *Piping Reference Data Guide* or the *Catalog Help*.

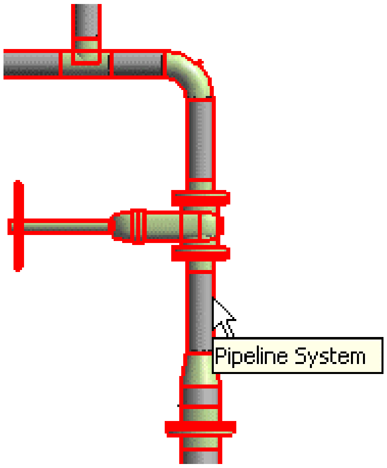
Before you start modeling, there are some relationships and concepts that you need to know.

Piping Systems

Piping systems are a way of organizing pipelines in your model. You can create piping systems based on the area where the pipelines are located, by what the pipelines carry, by the piping designer who models the pipelines, or by any other method that you choose. Piping systems are just a way to group objects. You create piping systems in the Systems and Specifications task.

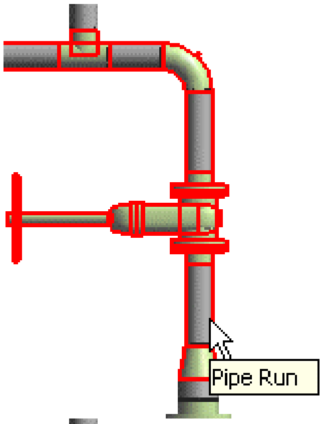
Pipelines

Pipelines are a way of organizing pipe runs in your model and are created in the Systems and Specifications task. You are not restricted to piping systems when creating pipelines. You can create pipelines under any previously created system. When you create a pipeline, you are required to define a fluid requirement and a fluid type.



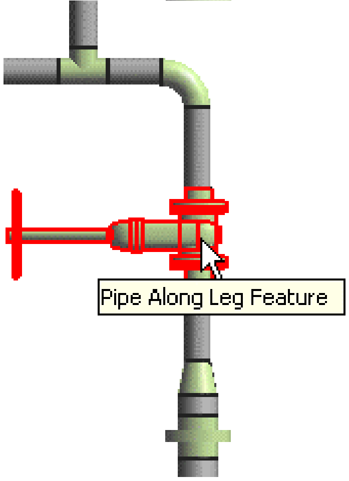
Pipe Runs

A pipe run is a connected series of pipe features that normally have the same nominal piping diameter (NPD) and flow direction, and are normally governed by the same pipe specification. All pipe features belong to a pipe run. One or more pipe runs make up a pipeline.



Pipe Features

When you route a pipe run, you place features defining high-level design information as you go. The software automatically selects the specific parts based on the pipe specification of the pipe run. You may want to think of features as logical collections of parts driven by the pipe specification. There are several basic features: straight, turn, branch, end, run change, split, and along leg component.

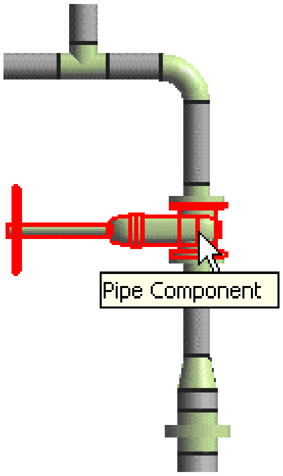


Note

- Pipe specifications are defined in the piping reference data. You can create and customize the pipe specifications to suit your needs. See the *Piping Reference Data Guide* for more information on defining pipe specifications.

Pipe Parts

Parts are the physical components that compose a feature and are generally selected by the software. For example, the flanges, gaskets, and the gate valve itself are examples of the parts composing the gate valve feature. In special situations where the pipe specification does not select the part that you want, you can override the part selection by explicitly selecting the part from the catalog. While this part does not follow the pipe specification defined for the pipe run, the part is still in the pipe run.



Pipe Ports

Ports are the connection points on the pipe parts.

Connections

Connections define how the pipe ports connect to one another.

The Piping task has the following commands:

	Select - Used to select objects in the model. For more information, see *Editing Properties: An Overview*, page 69.
	Route Pipe - Places pipe runs in the model. For more information, see *Route Pipe Command*, page 30.
	Insert Split - Places a welded split or a takedown joint in the selected pipe. For more information, see *Insert Split Command*, page 110.
	Insert Component - Places valves, tees, and other inline components in the selected pipe. For more information, see *Insert Component Command*, page 122.
	Insert Tap - Places a tap in the selected component. For more information, see *Insert Tap Command*, page 138.
	Generate Spools - Generates piping spools for the piping in the model. For more information, see *Generate Spools Command*, page 146.
	Create Penetration Spools - Generates penetration spools for pipes that have penetration plates place in the model. For more information, see *Create Penetration Spools Command*, page 151.
	Sequence Objects - Renames the components of a pipeline or spool so that they are in a sequential order. For more information, see *Sequence Objects Command*, page 155.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Inserting Components: An Overview*, page 121
- *Inserting Splits: An Overview*, page 109
- *Moving Pipe Features: An Overview*, page 63
- *Routing a Pipe Run: An Overview*, page 29
- *Spooling: An Overview*, page 143

Permission Groups and Routing: An Overview

Several different users using different permission groups can work together when routing if you know how the software handles the different situations. Starting in version 6.0, Piping fully supports different users, with different sets of privileges, working on different runs such as when working in a Global Workshare Configuration. Before version 6.0, pipe route legs could be shared between pipe runs. In version 6.0 and later, the software creates an Intermediate End Feature (IEF) at the end of a pipe run connected to another pipe run and creates a logical connection between the two IEFs/runs. The legs stop at the IEF and are no longer shared between pipe runs. You do not need to create a separate permission group for the pipe run and for the features of the pipe run. Everything can now be in the same permission group.

Assignment of Permission Groups

The first thing to know is how permission groups are assigned.

- Objects that you create directly are assigned to the active permission group.
- Objects created automatically by the software are assigned a permission group determined by an internal set of rules. The permission group assigned is not necessarily the active permission group. Examples of automatically placed objects include connections and a pipe automatically inserted when two touching valves are separated.
- Parts generated by features are assigned the permission group of the parent feature. Remember however, that runs can be in a different permission group than its collective features and parts.
- End features use the permission group of the run to which they belong.
- Connections use the permission group of the parts to which they are connected. If the connection is between parts with different permission groups, the permission group to which the user has write access is used. If the connection is between an equipment nozzle and a route part, the route part permission group is used for the connection.
- Connection objects (welds, bolt sets, gaskets, clamps) use the permission group of the connection that generated it.

Systems and Permission Groups

A system is a logical grouping of sub-systems. When you add or remove a sub-system, you are modifying the definition of the parent system. Therefore, you must have write access to the parent system. You do not need write access to the grandparent system. For example, to create a pipe run, you need write access to the parent pipeline. However, you do not need write access to the system to which the pipeline belongs.

When participating in a Global Workshare Configuration, you should manage all permission groups at the host site. The requirement that the sub-system have write access to the parent system is not enforceable if the sub-system is located at the satellite site and the parent system is at the host site. In this case, the sub-system has read-only access to the parent system. However, the software automatically compensates for the lack of write access during the replication process.

Example Configuration A

In this example, two users, John and Peter, are working on the same run with exclusive access. John is responsible for part of the run, and Peter is responsible for the other part of the run. Neither John nor Peter should be able to modify the work of the other person. The administrator should configure the permission groups as follows:

- Create three different permission groups: PG-Run, PG-John, and PG-Peter. Both John and Peter should have full control access to PG-Run. John should have full control access to PG-John while Peter should have read only access to PG-John. Similarly, Peter should have full control access to PG-Peter while John should have read only access to PG-Peter.

The run should be created using the PG-Run permission group. When John works on his parts of the run, he should use PG-John as the active permission group. When Peter works on his parts of the run, he should use PG-Peter as the active permission group. The two halves of the run should connect at a component such as a valve. For example, John routes his part of the run, places a flange, and then places a gate valve. Peter then places flange manually connecting to the open port of the gate valve, and then continues his part of the run.

Example Configuration B

In this example, two users, John and Peter, are working on different but in-line connected runs with exclusive access. For example, John places an elbow, a straight piece of pipe, and a gate valve then stops. Peter connects to the open port of the valve, and then continues routing. The administrator should configure the permission groups as follows:

- Create two different permission groups: PG-John and PG-Peter. John should have full control access to PG-John while Peter should have read only access to PG-John. Similarly, Peter should have full control access to PG-Peter while John should have read only access to PG-Peter.

John should create the run using the PG-John permission group and route his part of the run. When Peter works on his part of the run, he should use PG-Peter as the active permission group. The Intermediate End Features will handle the connection between the two parts of the pipe run.

Example Configuration C

In this example, two users, John and Peter, are working on different runs connected by a branching component (such as a tee, olet, tap, or Y). The administrator should configure the permission groups as follows:

- Create two permission groups: PG-John and PG-Peter. John should have full control access to PG-John while Peter should have read only access to PG-John. Similarly, Peter should have full control access to PG-Peter while John should have read only access to PG-Peter.

John creates an initial header pipe run using PG-John as the active permission group and routes it as needed. Peter now wants to branch from John's pipe run. Peter sets PG-Peter as the active permission group and selects the header in John's pipe run from which to branch. Instead of creating the header component (such as a tee), the software generates a To Do List item for John. When John updates the out-of-date To Do List item, the software modifies the header to add the tee, and then generates a To Do List item for Peter. When Peter updates his out-of-date To Do List item, the software "fixes" the branch leg (the olet is created or the end of the branch leg is adjusted to the tee port). This is called double hand-shaking mechanism.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Inserting Components: An Overview*, page 121
- *Inserting Splits: An Overview*, page 109
- *Moving Pipe Features: An Overview*, page 63
- *Routing a Pipe Run: An Overview*, page 29
- *Spooling: An Overview*, page 143

Understanding the Piping Workflow: An Overview

All piping elements are placed in the model using information defined in the piping reference data. Using the Catalog task or the reference data workbooks, you can create custom pipe specifications, edit pipe specification rules, and define pipe parts and symbols. Your first step should be to review, edit, and otherwise customize the delivered piping reference data. Refer to the *Piping Reference Data Guide* or the *Catalog Help* for more information.

After the reference data is customized to suit your needs, you need to define piping systems and pipelines in the Systems and Specifications task. You cannot place pipe runs in the model until the pipelines are defined.

After the piping reference data and the needed systems are defined, you can begin placing pipe in your model.

After pipe is in your model, the Piping task enables you to spool pipe into sections ready for manufacturing. To create penetration spools, at least one penetration plate must exist in the model. You can place penetration plates in the Hangers and Supports task.

Related Topics

- *Permission Groups and Routing: An Overview*, page 16
- *Piping and The Engineering Framework*, page 23
- *Piping Common Tasks*, page 20
- *Piping: An Overview*, page 13

Piping Common Tasks

The following tasks are used frequently in the piping task.

Customize Reference Data

- Create custom pipe specifications.
- Define parts and symbols. For more information on creating symbols, see *SmartPlant 3D Symbols Reference Data Guide*.

Create Needed Systems

- Create new pipe systems. For more information, see *Create a Piping System*, page 22.
- Create new pipelines. For more information, see *Create a Pipeline System*, page 21.

Route Pipe Runs

- Route new pipe runs in the pipelines that you defined. For more information, see *Create a New Pipe Run*, page 50.

Insert Splits, Components, and Taps

- Insert pipe splits in the pipe runs to create the needed spools lengths. You can insert splits while routing the pipe runs or after the runs are in the model. For more information, see *Place Splits while Routing Pipes*, page 54, *Insert a Takedown Joint*, page 116, and *Insert a Welded Split*, page 116.
- Insert valves, reducers, tees, and other components. You can insert components while routing the pipe runs or after the runs are in the model. For more information, see *Place Components while Routing Pipes*, page 53 and *Insert a Component*, page 131.
- Insert ports on components for venting, drainage, and for the connection of other components such as instruments. For more information, see *Insert a Tap*, page 140.

Create Piping Spools

- Split the pipe into spools by using the **Generate Spools** or **Create Penetration Spools** commands. For more information, see *Create Spools*, page 149 and *Create Penetration Spools*, page 153.

Create a Pipeline System

In the Systems and Specifications task:

1. Click **Task > Systems and Specifications**.
2. In the tree, select the system in which you want to create the pipeline system.
3. From the ribbon, select **New Pipeline System** .
4. Type a description for the pipeline system.
5. Specify a fluid requirement and fluid type for the pipeline.
6. Click **OK**.
7. Select the new pipeline system in the tree, and then right-click and select **Properties**.
8. Change any properties of the system as needed.

In the Piping task:

1. Click **Select** on the vertical toolbar.
2. Select **All** in the **Locate Filter**.
3. In the Workspace Explorer, right-click on the parent to the pipeline system that you are creating.
4. Click **New System > New Pipeline** on the pop-up menu.
5. Type a description for the pipeline system.
6. Specify a fluid requirement and fluid type for the pipeline.
7. Click **OK**.
8. Right-click on the new pipeline system in the Workspace Explorer, and then select **Properties**.
9. Change any additional properties as needed.

Related Topics

- *Create a New Pipe Run*, page 50
- *Create a Pipeline System*, page 21
- *Create a Piping System*, page 22
- *Edit Pipe Run Properties*, page 71
- *Place a Tee from a P&ID*, page 133
- *Place Components while Routing Pipes*, page 53
- *Route Pipe across P&ID Off-Page Connectors*, page 52
- *Select Pipe Run Settings*, page 61

Create a Piping System

In the Systems and Specifications task:

1. Click **Task > Systems and Specifications**.
2. In the tree, select the system in which you want to create the piping system.
3. From the ribbon, select **New Piping System** .
4. Select the new system in the tree, and then right-click and select **Properties**.
5. Change any properties of the system as needed.

In the Piping task:

1. Click **Select** on the vertical toolbar.
2. Select **All** in the **Locate Filter**.
3. In the Workspace Explorer, right-click on the parent to the piping system that you are creating.
4. Click **New System > New Piping System** on the pop-up menu.
5. Right-click on the new piping system in the Workspace Explorer, and then select **Properties**.
6. Change any additional properties as needed.

Related Topics

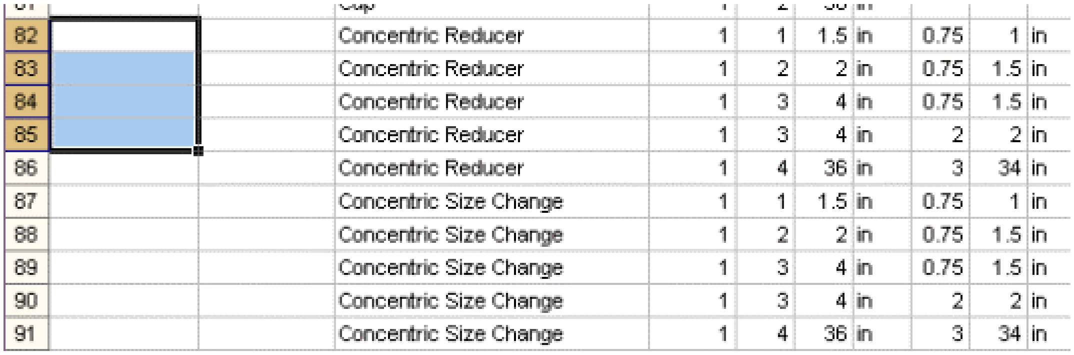
- *Choose a Working Plane*, page 60
- *Create a New Pipe Run from a P&ID Run*, page 51
- *Create a New Pipe Run*, page 50
- *Create a Pipeline System*, page 21
- *Edit Pipe Run Properties*, page 71
- *Place a Tee from a P&ID*, page 133
- *Place Components while Routing Pipes*, page 53
- *Place Splits while Routing Pipes*, page 54
- *Route a Pipe Run a Specified Distance and Direction*, page 58
- *Route a Pipe Run a Specified Distance*, page 57
- *Route a Pipe Run to a Specific Coordinate Location*, page 56
- *Route a Pipe Run with an Offset*, page 59
- *Route a Sloped Pipe Run*, page 55
- *Route Pipe across P&ID Off-Page Connectors*, page 52
- *Select Pipe Run Settings*, page 61

Piping and The Engineering Framework

This section describes how to use the Piping task with The Engineering Framework. For more information about using The Engineering Framework, refer to the *SmartPlant 3D The Engineering Framework Reference Guide* available from the **Help > Printable Guides** command.

Piping Catalog Data

SmartPlant P&ID and SmartPlant 3D must use the same naming convention for piping components and equipment for proper correlation with The Engineering Framework (TEF). For example, piping reducers Catalog data must be changed. Specifically, all pipe specification reducer components must have the name "Concentric Reducer" instead of the default name "Concentric Size Change". You can either edit the existing catalog spreadsheet entries or copy the existing entries and add new entries as shown below. Then Bulkload this spreadsheet into the Catalog.



82			Concentric Reducer	1	1	1.5	in	0.75	1	in
83			Concentric Reducer	1	2	2	in	0.75	1.5	in
84			Concentric Reducer	1	3	4	in	0.75	1.5	in
85			Concentric Reducer	1	3	4	in	2	2	in
86			Concentric Reducer	1	4	36	in	3	34	in
87			Concentric Size Change	1	1	1.5	in	0.75	1	in
88			Concentric Size Change	1	2	2	in	0.75	1.5	in
89			Concentric Size Change	1	3	4	in	0.75	1.5	in
90			Concentric Size Change	1	3	4	in	2	2	in
91			Concentric Size Change	1	4	36	in	3	34	in

Correlating Pipelines

Existing pipelines in the model that you want to correlate to P&ID data must be reassigned to the piping system that you want to correlate to. You cannot correlate the pipeline until it is moved. After it is in the correct piping system, you can correlate existing pipelines by:

1. Select **Framework > Correlate with Design Basis**.
2. Select the pipeline in the model.
3. Select the P&ID that contains the pipeline, then click **Open**.
4. Select the pipeline in the P&ID.
5. Review any data mismatch between the P&ID pipeline and the model pipeline. Fix the data as appropriate.
6. Click **Update**.

Previously correlated pipe runs (displayed as green in the P&ID) that change color to red or blue in the P&ID on subsequent retrieval of a new revision of the P&ID data need to be updated as follows:

1. Set the **Locate Filter** to **Piping Runs**.
2. Select the run in either the model or the P&ID.
3. Select **Framework > Compare to Design Basis**.
4. Any data item that has changed is highlighted in red.
5. Click **Update**.

Note

- When you click **Update**, any in-line component is also updated if it is still a valid piping component and if the component symbol still exists on the pipe run. If the symbol has been deleted, you will need to delete this component.
- In-line components placed with the use of dimensional data from SmartPlant Instrumentation must be updated in a separate process.

Correlating Instruments

You can correlate existing instruments in the model by:

1. Select **Framework > Correlate with Design Basis**.
2. Select the modeled instrument.
3. Select the P&ID that contains the instrument, then click **Open**.
4. Select the instrument in the P&ID.
5. Review any data mismatches between the P&ID instrument and the modeled instrument. Fix the data as appropriate.
6. Click **Update**.

You can place new correlated instruments using existing SmartPlant 3D Catalog instrument components or using the automatic creation of the instrument from the dimensional data supplied by SmartPlant Instrumentation.

To place standard Catalog instruments, you must manually match the instrument to place with the type of instrument called-out in the P&ID:

1. Route the pipe run from the P&ID. For detailed steps, see *Create a New Pipe Run from a P&ID Run*, page 51.
2. Select **Framework > View P&ID**.
3. Select the P&ID that contains the instrument to place, then click **Open**.
4. Click **Insert Component** on the vertical toolbar.
5. Select the area of the pipe run to place the instrument.
6. Select the component in the P&ID view.
7. In the **Type** option on the ribbon, select **<Specify Component Tag>**.
8. Select **Browse instruments**.

9. Select the proper instrument as indicated on the P&ID.
10. Position the instrument, and click **Finish**.
11. Open the properties for the instrument and size it appropriately for the pipe run.

To place instruments built on the fly from the dimensional data sheet data published from SmartPlant Instrumentation, Smart Plant 3D reads the dimensional data sheet, applies those values to the instrument in the background, and then places the instrument on the pipe feature:

1. Route the pipe run from the P&ID. For detailed steps, see *Create a New Pipe Run from a P&ID Run*, page 51.
2. Select **Framework > View P&ID**.
3. Select the P&ID that contains the instrument to place, then click **Open**.
4. Click **Insert Component** on the vertical toolbar.
5. Select the instrument in the P&ID.
6. Position the instrument on the pipe run, and then click **Finish**.

Only those instruments that were built from dimensional data sheets or have had a name change require updating.

To update instruments built/placed from dimensional data sheets:

1. Retrieve the new DDP from The Engineering Framework.
2. Select the instrument in the model.
3. Select **Framework > Compare to Design Basis**.
4. Any data item that has changed is highlighted in red.
5. Click **Update**.

In-line instruments placed from the catalog are automatically updated/revalidated when the parent pipe run is updated. To update instruments placed from the catalog:

1. Set the **Locate Filter** to **Piping Runs**.
2. Select the run in the model.
3. Select **Framework > Compare to Design Basis**.
4. Any data item that has changed is highlighted in red.
5. Click **Update**.

Off-Page Connectors

Off-page connectors (OPC) connect multi-page P&ID drawings. Unlike other P&ID elements, the OPC is correlated when the two pipelines are joined. The actual P&ID off-page connector 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. To correlate a pipeline that is located on multiple drawings, you should:

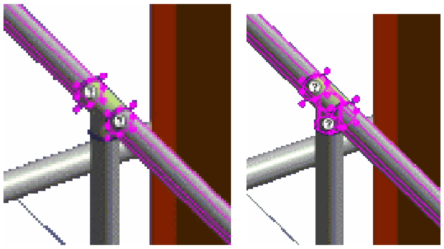
1. Route normally off the first, find a logical stopping point (such as an elbow).
2. Route normally from the second P&ID.
3. Then connect the two lines by extending or routing. A weld is placed between the two pipe segments (at the elbow). This automatically correlates the OPC.

For more detailed steps, see *Route Pipe across P&ID Off-Page Connectors*, page 52.

Topology Checker

The topology checker simply starts at one end of the pipeline and moves the end of the line. There are some rules that you should be aware to make sure the topology checker is running properly.

- Tees and Reducers have the unique ability to below to two different runs.
- A pipe run must be continuous for the topology to be properly checked. This means that no component (tees included) can belong to a different pipeline along that line. For example, if the third component along a pipe run was a tee that belonged to the branch run, the topology checker would not give proper results since the tee breaks the run. There are piping components that are still members of that first run, but the run is not continuous (left image: tee is not highlighted). After this tee is made a member of the original run, the topology checker will give proper results (right image: tee is highlighted).



An easy check to make sure you do not have this issue, is to:

1. Set the locate filter to **Piping Runs**.
2. Select each run. If the complete run highlights everything is ok. If one or more components do not highlight, the topology checker is probably not returning true results.

To change a piping component to another run:

1. Set the locate filter to **Piping Features**.
2. Select the component.
3. Change the system to which that component is a member.

Tees placed during routing should NOT be rotated and place by the branch port (making the existing route a branch instead of a header). This will cause a topology-checking problem. All tees, on headers, should be taken at the default orientation.

Related Topics

- *Create a New Pipe Run from a P&ID Run*, page 51
- *Route Pipe across P&ID Off-Page Connectors*, page 52

Routing a Pipe Run: An Overview

The **Route Pipe** command defines the geometry and properties of a pipe network. **Route Pipe** creates pipe runs, models the newly created run, or extends an existing pipe run. **Route Pipe** also creates features and components as driven by constraints, pipe specifications, and catalogs.

The **Route Pipe** command also works with the **Insert Split** and **Insert Component** commands to add features while routing. After placing the feature, such as a gate valve, the route command automatically restarts from the open port of the inserted feature. **Route Pipe** also uses the concept of routing on a plane to enable quick, precision creation of pipe runs.

The **Route Pipe** command supports the following:

- Routing using an existing pipe run
- Routing using a newly created pipe run
- Routing using a pipe run defined in the P&ID design basis (if P&ID design basis data is available)
- Routing to and from an end feature
- Routing to and from an equipment nozzle
- Routing to and from a straight feature (creating a branch)
- Inserting a component while routing
- Inserting a split while routing

When routing any pipe run, the major steps include selecting a starting point, defining the characteristics of the run (nominal pipe diameter, piping specification, options), routing the run point-by-point, and selecting the end point of the run.

Routing To or From a Straight Feature, End Feature, or Nozzle

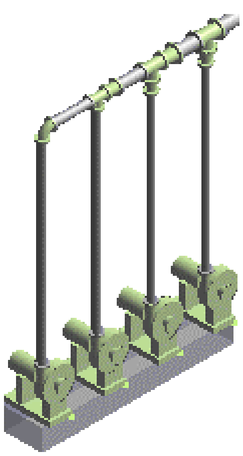
In addition to the basic functions of creating new runs or extending previous runs, the **Route Pipe** command supports routing to or from straight features, end features, and nozzles.

When routing a pipe run, you can define the run starting point either before or after clicking the **Route Pipe** command. In either case, if a feature was selected, the command determines the default working plane based on that feature. Additionally, the command determines whether to continue the pipe run associated with that feature or to create a new run. The software automatically generates any parts required for the connection, including branch components. This functionality allows you to easily extend or branch out of a previously modeled pipe run.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Inserting Components: An Overview*, page 121
- *Inserting Splits: An Overview*, page 109
- *Route Pipe Command*, page 30

Route Pipe Command



Creates, models, and extends pipe runs quickly and precisely. All pipe run features and components are driven by constraints, piping specifications, and catalogs.

By default, the software uses fast mode to display dynamically pipe run as you define the route. When in fast mode, the software does not render elbows or turns during the dynamic display. After you define the pipe run location, the software displays the elbows and turns. You cannot route a non-radial branch while in fast mode. Press SHIFT+F to turn off fast mode.

Related Topics
- *Choose a Working Plane*, page 60
- *Create a New Pipe Run from a P&ID Run*, page 51
- *Create a New Pipe Run*, page 50
- *Create a Pipeline System*, page 21
- *Create a Piping System*, page 22
- *Deleting Features: An Overview*, page 103
- *Edit Pipe Run Properties*, page 71
- *Insert Component Command*, page 122
- *Insert Split Command*, page 110
- *Place a Tee from a P&ID*, page 133
- *Place Components while Routing Pipes*, page 53
- *Place Splits while Routing Pipes*, page 54
- *Route a Pipe Run a Specified Distance and Direction*, page 58
- *Route a Pipe Run a Specified Distance*, page 57
- *Route a Pipe Run to a Specific Coordinate Location*, page 56
- *Route a Pipe Run with an Offset*, page 59
- *Route a Sloped Pipe Run*, page 55
- *Route Pipe across P&ID Off-Page Connectors*, page 52
- *Route Pipe Ribbon*, page 30
- *Routing a Pipe Run: An Overview*, page 29
- *Select Pipe Run Settings*, page 61

Route Pipe Ribbon

Sets options for routing a pipe.

Settings - Displays the **Route Pipe Settings** dialog box.

Start Point - Click to specify the starting location for the pipe run. The starting point can be an equipment nozzle, an open port of a feature such as a tee or valve, the end feature of another pipe run, or a point in space.

End Point - Click to specify the end point of the current pipe run leg. The end point can be an equipment nozzle, an open port of a feature such as a tee or valve, the end feature of another pipe run, or a point in space. If the end point is a point in space, then that point serves as the starting point for the next leg of the pipe run.

Plane - Activates options for selecting a working plane for the route path. Six options are available:

- Plan Plane - Defines the work surface as the XY plane at the depth of the active end. You also can press CTRL+1 to select this option.
- Elevation Plane: East-West - Defines the work surface as the XZ plane. You also can press CTRL+2 to select this option.
- Elevation Plane: North-South - Defines the work surface as the YZ plane. You also can press CTRL+3 to select this option.
- Plane by Turn/Branch - Defines the work surface as the plane defined by an existing turn or branch. You select the turn or branch to set the plane. You also can press CTRL+4 to select this option.
- Plane by Three Points - Defines the work surface using three points that you define. You also can press CTRL+5 to select this option.
- No Plane - Clears any work surfaces. The software does not project points that you place to any plane. You also can press CTRL+6 to select this option.

Run - Displays existing pipe runs along with the **<New Pipe Run>**, **<New/Continue Run>**, **<Select from P&ID>**, and **More** options. Select the pipe run to route or select **<New Pipe Run>** to create a new pipe run to route. If a new pipe run is created, all runs associated with the parent pipeline selected on the **New Pipe Run** dialog box appear in the list.

Select **<New/Continue Pipe Run>** to automatically extend an existing pipe run by selecting the end feature of that pipe run. If a run is continued, all runs associated with the pipeline parent of the continued run appear in the list. If you do not select an end feature, the **New Pipe Run** dialog box automatically activates to create a pipe run. For more information, see *New Pipe Run Dialog Box*, page 33.

Select **<Select from P&ID>** to graphically select a run in a P&ID to route. When you select the run in the P&ID, the software checks to see if the run already exists in the SmartPlant 3D model. If the run does exist, the current run is set to that run. If the run does not exist, the **New Pipe Run** dialog box displays using values from the run that you selected in the P&ID as the defaults. When you click **OK**, the run is created, and then correlated to the run in the P&ID.

Select **More** to select a pipe run that is associated with a different pipeline parent. For more information, see *Select Pipe Run Dialog Box*, page 40.

Lock Angle - Locks or unlocks the **Angle** box.

Angle - Enter or select an angle for the current route segment. You can enter any value needed, provided it does not conflict with the defined specifications. If the **Angle Lock** is unlocked, this box displays a dynamic readout of the current bend angle.

Turn Option - Select the option code associated with the turn component that you want to use while routing. Only those option codes defined in the controlling piping specification and shared by *all* of the short codes that can be placed automatically while routing appear.

Lock Length - Locks or unlocks the **Length** box.

Length - If **Lock Length** is locked, enter or select a length for the current route segment. If **Lock Length** is unlocked, this box displays the length of the current pipe run leg being placed.

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

Offset - Controls the SmartSketch offset constraint. Type the distance that you want to offset the pipe that you are routing. Select **<Disabled>** if you do not want to use the offset constraint to help route the pipe. Select **Set Offset Reference** to define the reference point. For more information on setting the reference point, see *Set Offset Reference Dialog Box*, page 38.

Lock Slope - Locks or unlocks the **Slope** box.

Slope - Displays the defined minimum slope for the run. If a value greater than or equal to the minimum slope of the run is entered, it is applied only to the leg currently being routed. After the leg is modeled, the value of the **Slope** box reverts to the minimum slope defined for the pipe run. You cannot enter a slope value that is less than the minimum slope specified for the pipe run. Select **Use Default Slope** to automatically change the slope value to match the **Minimum Slope** property of the pipe run. You can change the units of measure for the slope using **Tools > Options > Units of Measure**.

Related Topics

- *New Pipe Run Dialog Box*, page 33
- *Route Pipe Command*, page 30
- *Route Pipe Ribbon*, page 30
- *Set Offset Reference Dialog Box*, page 38

New Pipe Run Dialog Box

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

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

Standard

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

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

Name Rule - Specify the naming rule that you want to use to name this pipe run. You can select one of the listed rules or select **User Defined** to specify the pipe run name yourself in the **Name** box.

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

Nominal Diameter - Select the nominal pipe diameter (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 display on a session file basis using the **Tools > Options** command on the **Units of Measure** tab.

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

Minimum Slope - Specifies the slope for the pipe run. You can specify the slope as a ratio, a percentage, or in degrees.

- As a ratio, if a run drops 1/4" for every foot of horizontal distance, specify the slope as **1/4"/1'-0"**.

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

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

Testing

Testing Requirements - Specifies whether non-destructive testing of welds is required.

Testing Type - Select the type of non-destructive weld testing for the welds. If you want to add, edit, or remove values that are available for selection, edit the **Testing Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

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

Temperature and Pressure

Design Maximum Temperature - Specify the maximum design temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Design Minimum Temperature - Specify the minimum design temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Design Maximum Pressure - Specify the maximum design pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Operating Maximum Temperature - Specify the maximum operating temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Operating Minimum Temperature - Specify the minimum operating temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Operating Maximum Pressure - Specify the maximum operating pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Testing Maximum Temperature - Specify the maximum testing temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Testing Minimum Temperature - Specify the minimum testing temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Testing Maximum Pressure - Specify the maximum testing pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Design Minimum Pressure - Specify the minimum design pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Operating Minimum Pressure - Specify the minimum operating pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Testing Minimum Pressure - Specify the minimum testing pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Interior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Exterior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Insulation and Tracing

Insulation Specification - Select the insulation specification that you want to use for the pipe run. Select **More** to choose an insulation specification that is allowed for the system. You assign which insulation specifications are available in which systems in the System and Specifications task.

Insulation Purpose - Displays the purpose of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Purpose** box. However, if **Insulation Specification** is manually defined, you can select the purpose of the insulation in this box. Available purposes are read from the reference data.

Insulation Material - Displays the material of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Material** box. However, if **Insulation Specification** is manually defined, you can select the insulation material that you want to use.

Insulation Thickness - Displays the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data.

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the pipe run is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the run. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Responsibility

Cleaning Responsibility - Select the party responsible for cleaning the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Design Responsibility - Select the party responsible for designing the object. If you want to add, edit, or remove values that are available for selection, edit the **Design Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Fabrication Responsibility - Select the party responsible for fabricating the object. If you want to add, edit, or remove values that are available for selection, edit the **Fabrication Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Installation Responsibility - Select the party responsible for installing the object. If you want to add, edit, or remove values that are available for selection, edit the **Installation Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Painting Responsibility - Select the party responsible for painting the object. If you want to add, edit, or remove values that are available for selection, edit the **Painting Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Requisition Responsibility - Select the party responsible for ordering the object. If you want to add, edit, or remove values that are available for selection, edit the **Requisition Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Supply Responsibility - Select the party responsible for delivering the object. If you want to add, edit, or remove values that are available for selection, edit the **Supply Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Testing Responsibility - Select the party responsible for testing the weld on the object. If you want to add, edit, or remove values that are available for selection, edit the **Testing Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Related Topics
- *Pipe Run Properties Dialog Box*, page 81
- *Route Pipe Command*, page 30
- *Route Pipe Ribbon*, page 30

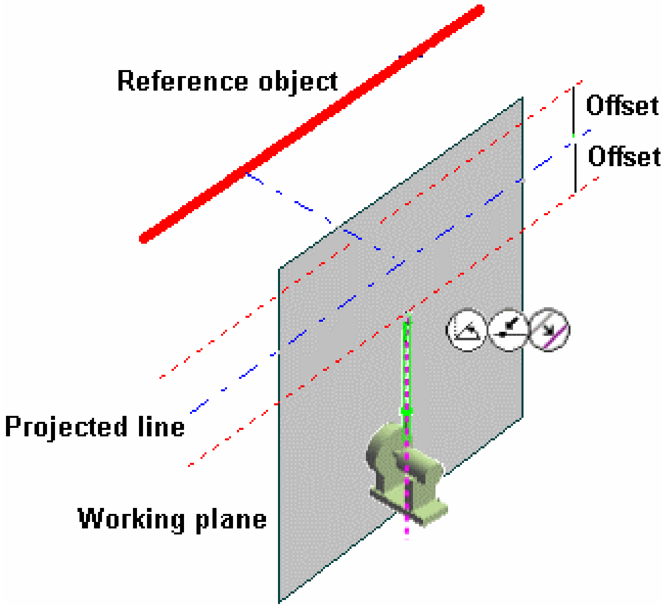
Set Offset Reference Dialog Box

Sets options for reference offsets.

Offset Reference Type - Select the offset type you want to use.

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

If the referenced object is planar surface (such as a slab) or a linear element (such as a structural beam), then the offset distance is measured from the surface or line to the selected reference plane (for example, the east/west plane) on which the pipe is being routed.



Offset - Type the distance that you want to offset. You can turn off the offset option by selecting **<Disabled>**.

Measured From - Select the object from which you want to measure the offset. You can select from:

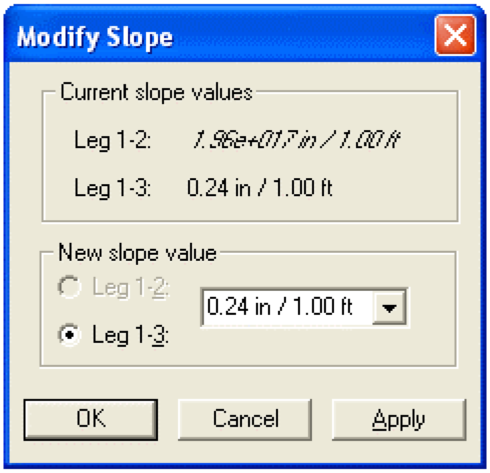
- **Centerline** - Select this option if you want to measure the offset from the centerline of a pipe run.
- **Pipe Surface** - Select this option if you want to measure the offset from the outside surface of a pipe run.
- **Insulation Surface** - Select this option if you want to measure from the outside insulation surface.
- **Largest Surface** - Select this option if you want to measure from the object that has the greatest diameter on the pipe run. The object could be the pipe surface, insulation surface, a flange, a valve, or any other pipe part.
- **Staggered** - Select this option to measure from the largest object on the pipe that you are routing to the largest surface on the pipe that you are referencing. Use this option when you want to get pipe centerlines as close as possible by staggering the location of split flanges.

Related Topics

- *Route a Pipe Run with an Offset*, page 59
- *Route Pipe Ribbon*, page 30

Modify Slope Dialog Box

Modifies the slope of a pipe run leg that is connected to a turn. You can modify the slope of only one leg at a time. You must select a turn feature, and then select the **Modify Slope** option on the ribbon to activate this dialog box.



Leg 1-2: *1.96e+017 in / 1.00 ft*

Leg 1-3: 0.24 in / 1.00 ft

○ Leg 1-2:

● Leg 1-3: 0.24 in / 1.00 ft

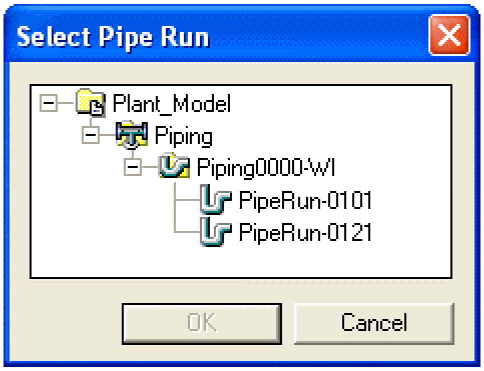
Current Slope Values - Displays the current slope for each leg connected to the turn. Legs with an invalid slope display in *italics*.

New Slope Value - Select the leg for which you want to modify the slope, and then select a new slope. The leg that you are editing highlights in the model.

Related Topics
- *Edit Turn Feature Ribbon*, page 47
- *Route Pipe Ribbon*, page 30

Select Pipe Run Dialog Box

This dialog box displays all of the pipe runs that have been placed in the defined Workspace. Select a pipe run from the hierarchy, and then click **OK**. The dialog box displays when you select **More...** from the **Run** box in the Route Pipe ribbon.



Related Topics
- *Pipe Run Properties Dialog Box*, page 81
- *Route Pipe Command*, page 30
- *Route Pipe Ribbon*, page 30

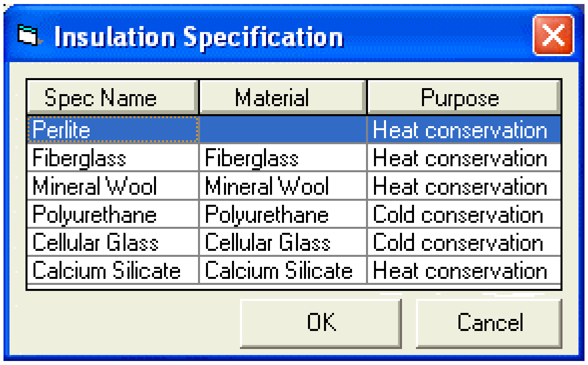
Select System Dialog Box

This dialog box displays all of the defined pipeline systems so that you can select the pipeline system that you want.

Look in - Specify where you want to look for the pipeline system. Select **Workspace** to look for the pipeline system in your defined workspace only. Select **Database** to look for the pipeline system in the entire model database.

Insulation Specification Dialog Box

Displays the available insulation specifications for the active piping system. Select the insulation specification that you want to use, and then click **OK**.



Spec Name	Material	Purpose
Perlite		Heat conservation
Fiberglass	Fiberglass	Heat conservation
Mineral Wool	Mineral Wool	Heat conservation
Polyurethane	Polyurethane	Cold conservation
Cellular Glass	Cellular Glass	Cold conservation
Calcium Silicate	Calcium Silicate	Heat conservation

Note

- You can modify the list of available insulation specification by switching to the Systems and Specifications task, selecting the piping system or pipeline system that you want to modify, and then selecting the **Allowed Specifications** command on the ribbon bar.

Edit Pipe Straight Feature Ribbon

Displays options for editing a straight feature.

Properties - Displays the **Pipe Straight Feature Properties** dialog box. For more information, see *Pipe Straight Feature Properties Dialog Box,* page 91.

Move From - Click to specify the starting location of the move vector. If you do not define a starting point, the software assumes that the current location of the object is the starting point.

Move To - Click to specify the ending location of the move vector.

Plane - Activates options for selecting a working plane for the route path. Six options are available:

- Plan Plane - Defines the work surface as the XY plane at the depth of the active end. You also can press CTRL+1 to select this option.
- Elevation Plane: East-West - Defines the work surface as the XZ plane. You also can press CTRL+2 to select this option.
- Elevation Plane: North-South - Defines the work surface as the YZ plane. You also can press CTRL+3 to select this option.

- Plane by Turn/Branch - Defines the work surface as the plane defined by an existing turn or branch. You select the turn or branch to set the plane. You also can press CTRL+4 to select this option.
- Plane by Three Points - Defines the work surface using three points that you define. You also can press CTRL+5 to select this option.
- No Plane - Clears any work surfaces. The software does not project points that you place to any plane. You also can press CTRL+6 to select this option.

Run - Displays the name of the pipe run with which the selected pipe is associated. All other pipe runs associated with the same pipeline parent also appear in the list. Another pipe run can be selected if needed. Select **More** to select a pipe run associated with a different pipeline.

Type - Displays the short code associated with the selected pipe. The **Type** list also contains the short codes associated with any other pipes defined in the piping specification for the current nominal diameter. Selecting an entry from the **Type** list replaces the selected pipe with one of a different type.

Option - Displays the option of the selected pipe. The **Option** list also contains any options defined in the piping specification for the short code selected in the **Type** box. For example, if two different wall thickness values are available in the piping specification for the short code selected in the **Type** box, both options appear in this list. Selecting another entry from the **Option** list updates the object accordingly.

Lock Length - Defines whether or not the length of the selected pipe should remain constant while moving.

When locked, the software automatically modifies the turn points, along with the length and angle of adjacent straight features, to remain connected to the moved leg. The length of the moved leg does not change.

When not locked, the software extends or shortens the associated legs to connect with the new position of the moved pipe. The length of the moved leg can change. Any component, such as a valve, on the moved pipe maintains its relative position from the pipe ends.

Offset - Controls the SmartSketch offset constraint. Type the distance that you want to offset the pipe that you are editing. Select **<Disabled>** if you do not want to use the offset constraint. Select **Set Offset Reference** to define the reference point. For more information on setting the reference point, see *Set Offset Reference Dialog Box*, page 38.

Slope - Displays the slope of the selected straight feature.

Related Topics

- *Move a Straight Pipe*, page 67

Edit Pipe End Feature Ribbon

Displays options for editing an end component, such as a cap, blind flange, or a pipe end.

Properties - Displays the **Pipe End Feature Properties** dialog box. For more information, see *Pipe End Feature Properties Dialog Box*, page 87.

Move From - Click to specify the starting location of the move vector. If you do not define a starting point, the software assumes that the current location of the object is the starting point.

Move To - Click to specify the ending location of the move vector.

Plane - Activates options for selecting a working plane for the route path. Six options are available:

- Plan Plane - Defines the work surface as the XY plane at the depth of the active end. You also can press CTRL+1 to select this option.
- Elevation Plane: East-West - Defines the work surface as the XZ plane. You also can press CTRL+2 to select this option.
- Elevation Plane: North-South - Defines the work surface as the YZ plane. You also can press CTRL+3 to select this option.
- Plane by Turn/Branch - Defines the work surface as the plane defined by an existing turn or branch. You select the turn or branch to set the plane. You also can press CTRL+4 to select this option.
- Plane by Three Points - Defines the work surface using three points that you define. You also can press CTRL+5 to select this option.
- No Plane - Clears any work surfaces. The software does not project points that you place to any plane. You also can press CTRL+6 to select this option.

Run - Displays the name of the pipe run with which the selected end feature is associated.

Type - Displays the short code associated with the selected end. The **Type** list also contains the short codes associated with any other end components defined in the piping specification for the current nominal diameter. Selecting an entry from the **Type** list replaces the selected end component with one of a different type.

Option - Displays the option of the selected end. The **Option** list also contains any options defined in the piping specification for the short code selected in the **Type** box. Selecting another entry from the **Option** list updates the object accordingly.

Lock Angle - Locks or unlocks the **Angle** box.

Angle - Displays the angle of any turn or branch component that is located at the other end of the leg with which the selected end feature is associated. If no turn or branch exists at the other end of the leg, then this box is disabled. Changing the angle value adjusts the position of the selected end feature to achieve the specified angle and then locks that angle value. When this box is unlocked, the value updates dynamically as the selected end component is moved.

Lock Length - Locks or unlocks the **Length** box.

Length - Displays the length of the run leg connected to the end being edited. Entering a value in this box adjusts the position of the selected end feature to achieve the specified length and then locks that length value. When this box is unlocked, the value updates dynamically as the selected end component is moved.

Offset - Controls the SmartSketch® offset constraint. Type the distance that you want to offset the pipe that you are routing. Select **<Disabled>** if you do not want to use the offset constraint to help route the pipe. Select **Set Offset Reference** to define the reference point. For more information on setting the reference point, see *Set Offset Reference Dialog Box*, page 38.

Lock Slope - Locks or unlocks the **Slope** box.

Slope - Displays the defined minimum slope for the end feature. Select **Use Default Slope** to automatically change the slope value to match the **Minimum Slope** property of the pipe run. You can change the units of measure for the slope using **Tools > Options > Units of Measure**.

Related Topics
- *Move a Pipe End*, page 66

Edit Branch Feature Ribbon

Displays options for branches placed in the model.

Properties - Activates the properties dialog box for the selected branch. For more information, see *Pipe Component Feature Properties Dialog Box*, page 127.

Move From - Click to identify the origin of the move vector. If you do not define a starting point, the software assumes that the current location of the branch is the starting point.

Move To - Click to identify the termination point of the move vector.

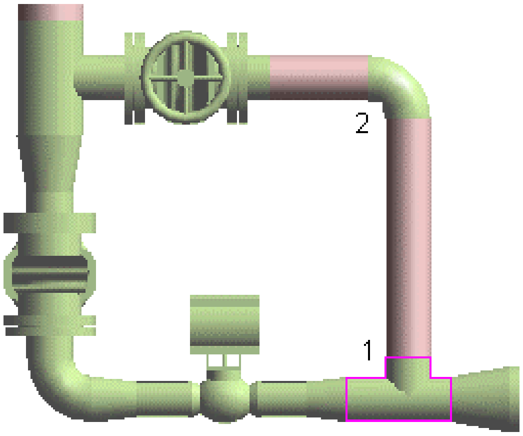
Type - Displays the short code associated with the selected branch. The **Type** list also contains the short codes associated with any other branches defined in the piping specification for the current nominal diameter. Selecting an entry from the **Type** list replaces the selected branch with one of a different type.

Option - Displays the option of the selected branch. The **Option** list also contains any options defined in the piping specification for the short code selected in the **Type** box. Selecting another entry from the **Option** list updates the object accordingly.

Run - Displays the name of the pipe run to which the branch is associated. All other pipe runs associated with the same pipeline parent also appear in the list. Another pipe run can be selected if needed. Select **More** to select a pipe run associated with a different pipeline.

Angle 1 - Specifies the angle of the branch.

Angle 2 - Specifies the angle of the turn or branch feature located at the other end of the branch leg. If this box is blank, there is no turn or branch feature located at the end of the branch leg.



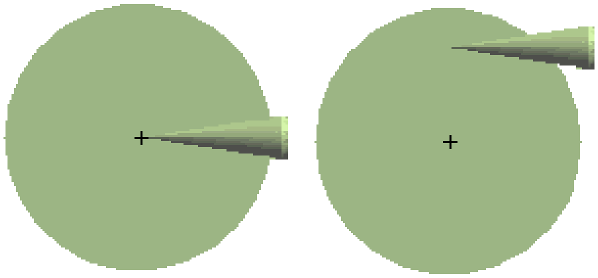
Locations of Angle 1 and Angle 2

Flip - Orients the branch so that the selected port is located at the insertion point. If a lateral, tee, or other tee-type branch is located along the length of a pipe or between two components, only the inline ports of the component are available when flipping. However, if the same component is located at the end of a pipe run, all component ports are available.

Rotate - Select to rotate the branch about the pipe interactively. This option is available only when nothing has been connected to the branch port.

Angle - Type an angle at which you want the branch rotated about the pipe. This option is available only when nothing has been connected to the branch port.

Branch Position - Select the position of the branch centerline relative to the header centerline. Select **Radial** to place the branch such that its centerline intersects the header centerline. Select **Non-Radial** to place the branch such that its center line does not intersect the header centerline. If you select **Non-Radial**, you can specify the distance the branch centerline is offset from the header centerline up to being tangent to the outside diameter of the header.



Radial and Non-Radial Positions

Related Topics

- *Delete a Branch*, page 104
- *Edit Branch Properties*, page 71
- *Move a Branch*, page 65

Edit Turn Feature Ribbon

Displays options for turns or bends placed in the model.

Properties - Activates the properties dialog box for the selected turn. For more information, see *Pipe Turn Feature Properties Dialog Box,* page 95.

Move From - Click to identify the origin of the move vector. If you do not define a starting point, the software assumes that the current location of the turn is the starting point.

Move To - Click to identify the termination point of the move vector.

Plane - Activates options for selecting a working plane for the turn. Six options are available:

- Plan Plane - Defines the work surface as the XY plane at the depth of the active end. You also can press CTRL+1 to select this option.
- Elevation Plane: East-West - Defines the work surface as the XZ plane. You also can press CTRL+2 to select this option.
- Elevation Plane: North-South - Defines the work surface as the YZ plane. You also can press CTRL+3 to select this option.
- Plane by Turn/Branch - Defines the work surface as the plane defined by an existing turn or branch. You select the turn or branch to set the plane. You also can press CTRL+4 to select this option.
- Plane by Three Points - Defines the work surface using three points that you define. You also can press CTRL+5 to select this option.
- No Plane - Clears any work surfaces. The software does not project points that you place to any plane. You also can press CTRL+6 to select this option.

Run - Displays the name of the pipe run with which the selected turn is associated. All other pipe runs associated with the same pipeline parent also appear in the list. Another pipe run can be selected if needed. Select **More** to select a pipe run associated with a different pipeline.

Type - Displays the short code associated with the selected turn. The **Type** list also contains the short codes associated with any other turns defined in the piping specification for the current nominal diameter. Selecting an entry from the **Type** list replaces the selected turn with one of a different type.

Option - Displays the option of the selected turn. The **Option** list also contains any options defined in the piping specification for the short code selected in the **Type** box. For example, if two different wall thickness values are available in the piping specification for the short code selected in the **Type** box, both options appear in this list. Selecting another entry from the **Option** list updates the object accordingly.

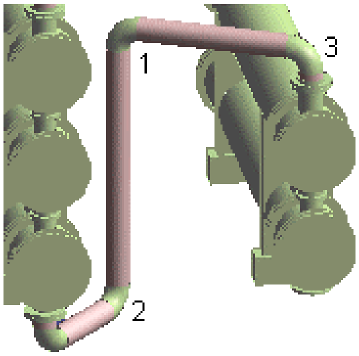
Flip - Orients the turn so that the other turn port is connected to the end of the run. This option is available only when a turn is connected to the end of a run and no other part has been connected to the other turn port.

Lock Angle - Locks or unlocks the **Angle 2** and **Angle 3** boxes. Locking the corresponding angle value creates a constraint along which the selected turn angle can be moved.

Angle 2 - Specifies the angle of the turn feature located at the end of one associated leg. If this box is blank, there is no turn feature located at the end of the leg. Although this value can be changed, the location of the corresponding turn cannot. Modification of this value repositions the selected turn until the specified angle is achieved.

Angle 1 - Specifies the angle of the turn that you are editing. Modification of this value repositions the selected turn until the specified angle is achieved.

Angle 3 - Specifies the angle of the turn feature located at the end of one associated leg. If this box is blank, there is no turn feature located at the end of the leg. Although this value can be changed, the location of the corresponding turn cannot. Modification of this value repositions the selected turn until the specified angle is achieved.



Location of Angles 1, 2, and 3

Offset - Controls the SmartSketch offset constraint. Type the distance that you want to offset the pipe that you are routing. Select **<Disabled>** if you do not want to use the offset constraint to help route the pipe. Select **Set Offset Reference** to define the reference point. For more information on setting the reference point, see *Set Offset Reference Dialog Box*, page 38.

Modify Slope - Activates the **Modify Slope** dialog box, with which you can modify the slope of the legs on either side of the selected turn. For more information, see *Modify Slope Dialog Box*, page 39.

Related Topics

- *Delete a Turn*, page 106
- *Edit Turn Feature Properties*, page 72
- *Move a Turn*, page 68

Edit Pipe Part Ribbon

Displays options for editing the selected object. This ribbon displays when you set the **Locate Filter** to **Piping Parts**, and select a part in the model.

Properties - Displays the properties dialog box for the selected object.

Run - Displays the name of the pipe run with which the selected object is associated. You cannot change the run.

Type - Displays the short code associated with the selected object. The **Type** list also contains the short codes associated with any other objects defined in the piping specification for the current nominal diameter. Selecting an entry from the **Type** list replaces the selected object with one of a different type.

Option - Displays the option of the selected object. The **Option** list also contains any options defined in the piping specification for the short code selected in the **Type** box. For example, if two different wall thickness values are available in the piping specification for the short code selected in the **Type** box, both options appear in this list. Selecting another entry from the **Option** list updates the object accordingly.

Name - Specifies the object name. You can enter a new name if needed.

Tag - Specifies the engineering tag for the object.

Base/Mating Part - Displays whether the selected object is a base part or a mating part. If the object is a mating part, you can use this option to change the object to a base part. 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.

Related Topics
- *Move a Straight Pipe*, page 67

Create a New Pipe Run

1. Click **Route Pipe** on the vertical toolbar.
2. Select the run starting point.

   Tip

   - If you select a feature located at the end of an existing run, the software continues the run of the selected feature. Skip to step 5.
   - If you select an equipment nozzle, a point in space, a tap, a branch port, or a point along a straight feature, the software prompts you to create a new pipe run.
   - If you select an equipment nozzle that is correlated with P&ID design basis data, the software automatically determines which run should be connected.
3. On the **New Pipe Run** dialog box, type a name for the **Pipe Run**. If you do not enter a name, the software automatically generates a name.

   Select Pipe Run Settings, page 61
4. Click **OK** to close the **New Pipe Run** dialog box.
5. Select points to define the route of your pipe run.

   Tip

   - You can use **Pin Point**, **Point Along**, and the SmartSketch® relationship indicators when defining your pipe run.
6. Select the ending point of the pipe leg. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- You can continue routing an existing run by selecting a run end feature or by selecting the run name from the list of available runs.
- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. Do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics
- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Create a New Pipe Run from a P&ID Run

1. Click **Framework > Retrieve** to retrieve the P&ID drawing that contains the run that you want to route. You can skip this step if the P&ID has already been retrieved.
2. Click **Framework > View P&ID** to view the P&ID drawing that contains the run that you want to route.
3. Select **All** or **Pipe Runs** in the **Locate Filter**.
4. Click **Route Pipe** on the vertical toolbar.
5. In the **Run** pulldown list on the ribbon bar, select **<Select from P&ID>**.
6. Select the run to route from the P&ID drawing.
7. The **New Pipe Run** dialog box appears with properties populated from the design basis. Click **OK** to accept the properties given, which saves the correlated run information and closes the **New Pipe Run** dialog box.
8. Select points to define the route of your pipe run.

   Tips

   - If process data was not defined on the P&ID, then you must manually enter a temperature and pressure.
   - You can use **Pin Point**, **Point Along**, and the SmartSketch® relationship indicators when defining your pipe run.
   - If you select an equipment nozzle that is correlated with P&ID design basis data, the software automatically determines which run should be connected.
9. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- Your plant must be registered with the Engineering Framework to use this procedure.

Related Topics
- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Piping and The Engineering Framework*, page 23
- *Routing a Pipe Run: An Overview*, page 29

Route Pipe across P&ID Off-Page Connectors

1. Click **Framework > Retrieve** to retrieve the P&IDs that contains the runs to route. You can skip this step if the P&IDs have already been retrieved.
2. Click **Framework > View P&ID** to view the P&ID that contains the first part of the run to route.
3. Select **All** or **Pipe Runs** in the **Locate Filter**.
4. Click **Route Pipe** on the vertical toolbar.
5. In the **Run** pulldown list on the ribbon bar, select **<Select from P&ID>**.
6. From the P&ID drawing, select the run to route.
7. The **New Pipe Run** dialog box appears with properties populated from the design basis. Click **OK** to accept the properties given, which saves the correlated run information and closes the **New Pipe Run** dialog box.
8. Select points to define the route of your pipe run.

   Tip

   - You can use **Pin Point**, **Point Along**, and the SmartSketch® relationship indicators when defining your pipe run.
9. Right-click to end a pipe run in space (not terminated on a nozzle or branch).

   Note

   - After routing, notice that the correlation color in the P&ID Viewer does not show a data match. This mismatch is because the second part of the run noted through the off-page connector has not yet been routed and correlated. The off-page connector displays the corresponding P&ID to find the matching connector.
10. Click **Framework > View P&ID** to view the corresponding P&ID.
11. Click **Route Pipe** on the vertical toolbar.
12. In the **Run** pulldown list on the ribbon bar, select **<Select from P&ID>**.
13. Select the other side of the run to route from the corresponding P&ID.
14. The **New Pipe Run** dialog box appears with properties populated from the design basis. Click **OK** to accept the properties given, which saves the correlated run information and closes the **New Pipe Run** dialog box.
15. Route the run to the end of the first pipe run.

Related Topics

- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Piping and The Engineering Framework*, page 23
- *Routing a Pipe Run: An Overview*, page 29

Place Components while Routing Pipes

1. Click **Route Pipe** on the vertical toolbar.
2. Select the run starting point.

   Tip

   - If you select a feature located at the end of an existing run, the software continues the run of the selected feature. Skip to step 5.
   - If you select an equipment nozzle, a point in space, or a point along a straight feature, the software prompts you to create a new pipe run.
3. On the **New Pipe Run** dialog box, type a name for the **Pipe Run**. If you do not enter a name, the software automatically generates a name.

   Select Pipe Run Settings, page 61
4. Click **OK** to close the **New Pipe Run** dialog box.
5. Select points to begin routing your pipe run.
6. Click **Insert Component** when you want to insert a component.

   Insert a Component, page 131
7. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- You can continue routing an existing run by selecting a run end feature or by selecting the run name from the list of available runs.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Routing a Pipe Run: An Overview*, page 29

Place Splits while Routing Pipes

1. Click **Route Pipe** on the vertical toolbar.
2. Select the run starting point.

   Tip

   - If you select a feature located at the end of an existing run, the software continues the run of the selected feature. Skip to step 5.
   - If you select an equipment nozzle, a point in space, or a point along a straight feature, the software prompts you to create a new pipe run.
3. On the **New Pipe Run** dialog box, type a name for the **Pipe Run**. If you do not enter a name, the software automatically generates a name.

   Select Pipe Run Settings, page 61
4. Click **OK** to close the **New Pipe Run** dialog.
5. Select points to begin routing your pipe run.
6. Click **Insert Split** when you want to insert a split.

   Insert a Welded Split, page 116

   Insert a Takedown Joint, page 116
7. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- You can continue routing an existing run by selecting a run end feature or by selecting the run name from the list of available runs.

Related Topics
- *Deleting Features: An Overview*, page 103
- *Routing a Pipe Run: An Overview*, page 29

Route a Sloped Pipe Run

1. Click **Route Pipe** on the vertical toolbar.
2. Select the run starting point in an elevation view.
3. On the **New Pipe Run** dialog box, select the parent **Pipeline**.
4. In the **Name** box, type a name for the pipe run. If you do not enter a name, the software automatically generates a name.
5. Select the **Minimum Slope** box, and then enter the slope for the pipe run. For example, **2 deg** or **0.5in / 1.0ft**.
6. Click **OK** to close the **New Pipe Run** dialog box.
7. Specify if you are routing from the high point or the low point of the pipe run.
8. Select points to define the route of your pipe run.

   Tip

   - You can use **Pin Point**, **Point Along**, and the SmartSketch® relationship indicators when defining your pipe run.
9. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Notes

- The starting point for the pipe run must be selected in an elevation view. You cannot select the starting point in an isometric or plan view. However, after the starting point is defined, you can route the pipe run by defining points in a view with any orientation.
- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. Do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics
- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Route a Pipe Run to a Specific Coordinate Location

1. Click **Tools** > **PinPoint**.
2. Select the coordinate system that contains the location to which you want to route.
3. Select the **Rectangular Coordinates** option on the **PinPoint** ribbon.
4. Click **Route Pipe** on the vertical toolbar.
5. Select the run starting point.
6. Define the pipe run properties on the **New Pipe Run** dialog box, and then click **OK**.
7. Select the **Set Target to Origin** option on the PinPoint ribbon to move the **PinPoint** target to the specified coordinate system's origin.
8. In the **PinPoint** ribbon, type the easting, northing, or elevation coordinates to which you want to route.
9. Click in the view to route the pipe run to the specified point.

Note

- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. Do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics

- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Route a Pipe Run a Specified Distance

1. Click **Tools** > **PinPoint**.
2. Select the **Relative Tracking** option on the **PinPoint** ribbon.
3. Select the **Rectangular Coordinates** option on the **PinPoint** ribbon.
4. Click **Route Pipe** on the vertical toolbar.
5. Select the run starting point.
6. Define the pipe run properties on the **New Pipe Run** dialog box, and then click **OK**.
7. In the **PinPoint** ribbon, type the easting, northing, or elevation distance (from the **PinPoint** target) to route the pipe run.
8. Click in the view to route the pipe run to the specified point. The software moves the **PinPoint** target to the new location.
9. In the **PinPoint** ribbon, type the easting, northing, or elevation distance to route the pipe run.
10. Continue defining points as needed.
11. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. You do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics

- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Route a Pipe Run a Specified Distance and Direction

1. Click **Tools** > **PinPoint**.
2. Select the **Relative Tracking** option on the **PinPoint** ribbon.
3. Select the **Spherical Coordinates** option on the **PinPoint** ribbon.
4. Click **Route Pipe** on the vertical toolbar.
5. Select the run starting point.
6. Define the pipe run properties on the **New Pipe Run** dialog box, and then click **OK**.
7. In the **PinPoint** ribbon, type the absolute distance and direction to route.
8. Click in the view to route the pipe run to the specified point. The software moves the **PinPoint** target to the new location.
9. Continue defining points as needed.
10. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. Do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics

- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Route a Pipe Run with an Offset

1. Click **Route Pipe** on the vertical toolbar.
2. Select the run starting point.
3. Define the pipe run properties on the **New Pipe Run** dialog box, and then click **OK**.
4. From the **Offset** option, select **Set Offset Reference**.
5. Select the option to use from the **Measured from** field.
6. Type the offset distance to use in the **Offset** box.
7. Click **OK** on the **Set Offset Reference** dialog box.
8. Select the appropriate route plane.
9. Move the cursor over the object in the model from which you want to measure the offset until the object is added to the SmartSketch locate list. This glyph appears.
10. Move the cursor until the offset line and glyph appear, and then click to route the pipe run.
11. Select the ending point of the pipe run. Right-click to end a pipe run that does not terminate in a nozzle or branch.

Note

- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. You do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Routing a Pipe Run: An Overview*, page 29
- *Set Offset Reference Dialog Box*, page 38

Copy a Pipe Run

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Runs** in the **Locate Filter**.
3. Select the pipe run to copy.
4. Click **Edit > Copy**.
5. Click **Edit > Paste**.
6. Select **Move** from the main ribbon bar.
7. Identify a location in the model for the pipe run.

Note

- If you position an end of the copied pipe run so that it touches a pipe run end or nozzle, the system automatically connects the two.

Related Topics
- *Change the Flow Direction of a Pipe Run*, page 61
- *Copy a Pipe Run*, page 59
- *Delete a Pipe Run*, page 107
- *Extend an Existing Pipe Run*, page 60
- *Move a Pipe Run*, page 65

Extend an Existing Pipe Run

1. Click **Route Pipe** on the vertical toolbar.
2. Select the end feature from which to extend the pipe run.
3. Route the pipe run as needed.

Related Topics
- *Change the Flow Direction of a Pipe Run*, page 61
- *Copy a Pipe Run*, page 59
- *Delete a Pipe Run*, page 107
- *Deleting Features: An Overview*, page 103
- *Extend an Existing Pipe Run*, page 60
- *Routing a Pipe Run: An Overview*, page 29

Choose a Working Plane

1. Click **Plane** on the ribbon.
2. Select the routing plane for your pipe run.

Note

- When you select a routing plane, the software highlights the plane in the model.
- You also can press CTRL-1, CTRL-2, CTRL-3, CTRL-4, CTRL-5, and CTRL-6 to select a working plane.

Related Topics
- *Deleting Features: An Overview*, page 103
- *Routing a Pipe Run: An Overview*, page 29

Select Pipe Run Settings

1. On the **New Pipe Run** dialog box, choose the pipeline for the new run in the **Pipeline** box.

   Tip

   - The **Pipeline** box lists up to the last ten selected pipelines. Choosing the **More** option at the bottom of the list allows you to browse all pipelines in the model.
2. In the **Name** box, type the name of the new pipe run that you are creating.
3. Set the specification of the new pipe run in the **Specification** box.
4. In the **Nominal Diameter** box, select the NPD for the pipe run.
5. Click **OK** to return to the **Route Pipe** command.

Notes

- Use the **Category** option to view more properties that you can set for the pipe run.
- If the piping specification that you selected has service limits defined, you must define at least one temperature value and one pressure value for the pipe run. Do this by selecting **Temperature and Pressure** from the **Category** option.

Related Topics

- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Change the Flow Direction of a Pipe Run

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Runs** in the **Locate Filter**.
3. Select the pipe run for which you want to change the flow direction.
4. Click one of the flow direction indicators that appear on the pipe run.
5. Select a new flow direction on the **Flow** dialog box.

Notes

- The flow directional indicators are listed below:

  Bi-directional

  Upstream

  Downstream

  No flow

ⓘ Undefined

- You also can click **Properties** on the ribbon to edit the flow direction property of a run.

Related Topics

- *Change the Flow Direction of a Pipe Run*, page 61
- *Copy a Pipe Run*, page 59
- *Delete a Pipe Run*, page 107
- *Extend an Existing Pipe Run*, page 60
- *Pipe Run Properties Dialog Box*, page 81

Moving Pipe Features: An Overview

After you complete a pipe run, you can move the features of that run. By dragging the features, you can alter the route of the pipe, connection points, or component placement. Using this ability to move features of your pipe runs, you can precisely locate each feature in the layout of your piping systems.

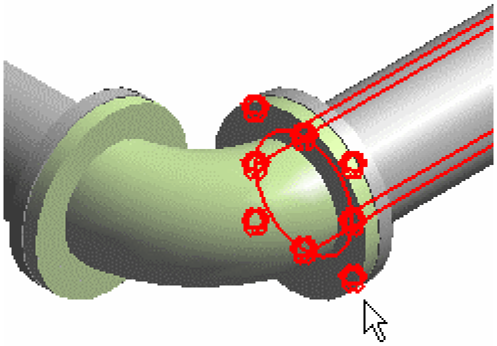
The various types of features that you can move in this method include the following:

Straight Features

The straight feature and other connected route features move subject to the following rules:

- Moving a straight feature moves the entire leg to which the feature is connected.
- The move direction is always perpendicular to the axis of the straight feature.
- A branch feature connected to the moved leg maintains its original angle.
- Movement stops when parts on the associated leg overlap, or when they overlap with adjacent parts on connected legs.
- When equipment moves, the leg moves with the equipment nozzle to remain collinear with the nozzle as much as possible.

When moving straight features, you can select one of the orientation handles that appear at the ends of the straight feature as your "move from" point. These orientation handles allow you to move the pipe in relation to the edge of the pipe instead of the pipe centerline. For example, you can use the orientation handles to align the bottom of pipe with the top of a beam.



Turn Features

The turn feature and other connected route features move subject to the following rules:

- If you move a turn to a position where the two adjacent legs are collinear (that is, the turn reaches 180 degrees), the software deletes the turn.
- The turn stops moving when it starts to overlap with an adjacent along leg feature.
- If neither angle 2 or 3 is locked, the move is along the bisector. If either angle 2 or 3 is locked, the move is along the other leg.

End Features

The end feature and other connected route features move subject to the following rules:

- Moving the end feature edits the length of the straight feature.
- Moving the end feature edits the topology of the turn feature or the branch feature if connected to another leg. The default motion plane is the connected Turn Feature or Along Leg Feature.
- You cannot move an end feature connected to a nozzle.
- You cannot move the end feature past the point where the straight feature length becomes zero, such as the point where an end feature touches a nozzle or free port of another feature.

Branch Features

- If the branch and header are in the motion plane, the branch feature moves along the branch leg.
- If the branch and header are not in the motion plane, the end of the branch leg and branch feature move to keep the branch angle constant.
- You can move the branch feature along the associated straight feature. As you move the feature, the angle between the branch feature and the branch leg appears in a dynamic readout.

Along Leg Features

- You can move along leg features along the pipe on which you placed them. The software prevents you from moving the along leg feature past the end of the pipe or to a position that would cause parts to overlap.
- Along Leg Features that are not branches (valves and flanges are examples) maintain a position at a constant distance from the fixed end of the leg.

You can move features only after you finish routing a pipe run and placing components. Edit commands are not available until you complete routing or placement. You can complete a routing by right-clicking or by pressing **Esc**.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Inserting Components: An Overview*, page 121
- *Inserting Splits: An Overview*, page 109
- *Move a Branch*, page 65
- *Move a Feature Break*, page 118
- *Move a Pipe End*, page 66
- *Move a Split*, page 118
- *Move a Straight Pipe*, page 67
- *Move a Turn*, page 68
- *Routing a Pipe Run: An Overview*, page 29

Move a Pipe Run

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Run** in the **Locate Filter**.
3. Select the pipe run to move.
4. Select **Move** on the main ribbon.
5. Select a location either on the pipe run or in the model to use as the move-from location.

   Tip

   - Use the end of the pipe run as the move-from location if you are trying to connect the pipe run to a nozzle or another pipe run.
6. Identify the move-to location in the model.

Related Topics

- *Moving Pipe Features: An Overview*, page 63

Move a Branch

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the branch that you want to move and drag it to the new location.

Notes

- You can use the **Move From** and **Move To** options to specify a move vector. If you do not define a starting point by using **Move From**, the software assumes that the current location of the object is the starting point.
- You can use the **Angle** box to type a new angle for the branch. The software moves the branch along the pipe to the specified angle.

Related Topics

- *Delete a Branch*, page 104
- *Edit Branch Feature Ribbon*, page 44
- *Edit Branch Properties*, page 71
- *Move a Branch*, page 65
- *Moving Pipe Features: An Overview*, page 63

Move a Pipe End

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the pipe end that you want to move and drag it to the new location.

   Tip

   - Make sure that you select the pipe end and not the pipe itself by looking for the SmartSketch® **End Point** glyph.

Notes

- You can use the **Move From** and **Move To** options to specify a move vector. If you do not define a starting point by using **Move From**, the software assumes that the current location of the object is the starting point.
- You can use the **Length** box to type a new length for the pipe. The software moves the pipe end to the specified length.

Related Topics

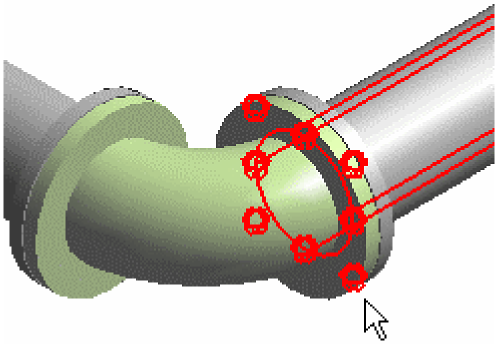
- *Edit Pipe End Feature Ribbon*, page 43
- *Moving Pipe Features: An Overview*, page 63

Move a Straight Pipe

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the pipe feature that you want to move.
4. Set the **Length Lock** as needed.
5. Select and drag the pipe to the new location.

Notes

- You can use the **Move From** and **Move To** options to specify a move vector. If you do not define a starting point by using **Move From**, the software assumes that the current location of the object is the starting point.
- When you move a straight feature, such as a pipe, the default working plane is the plane of one of the adjacent turns. The software selects the turn first by the plane that includes branches, and then by the plane of larger nominal diameter.
- You can select one of the orientation handles that appear at the ends of the straight feature as your move from point. These orientation handles allow you to move the pipe in relation to the edge of the pipe instead of the pipe centerline. For example, you can use the orientation handles to align the bottom of pipe with the top of a beam.



Related Topics

- *Delete a Straight Pipe*, page 105
- *Edit Pipe Straight Feature Ribbon*, page 41
- *Edit Straight Pipe Properties*, page 72
- *Move a Straight Pipe*, page 67
- *Moving Pipe Features: An Overview*, page 63

Move a Turn

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the turn that you want to move, and drag the turn to the new location.

Notes

- You can use the **Move From** and **Move To** options to specify a move vector. If you do not define a starting point by using **Move From**, the software assumes that the current location of the object is the starting point.
- You can use the **Angle** box to type a new angle for the turn. The software moves the turn along the pipe to the specified angle.

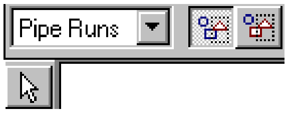
Related Topics

- *Delete a Turn*, page 106
- *Edit Turn Feature Properties*, page 72
- *Edit Turn Feature Ribbon*, page 47
- *Move a Turn*, page 68
- *Moving Pipe Features: An Overview*, page 63

Editing Properties: An Overview

All objects that compose a pipeline have properties that you can edit. Using the **Select** command on the vertical toolbar, you select the pipeline object that you want to edit.

An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Pipe Runs**, you can select only pipe runs in a graphic view or in the **Workspace Explorer**.



The Piping task includes these filters:

Pipelines - Allows you to select only pipelines in a graphic view or in the **Workspace Explorer**.

Piping Runs - Allows you to select only *pipe runs* in a graphic view or in the **Workspace Explorer**.

Piping Features - Limits the selection of objects to only pipe *features*. Some examples of features are straight features, such as pipe, and in-line component features, such as valves.

Piping Parts - Allows you to select in a graphic view or in the **Workspace Explorer**, only the individual pipe parts that are associated with a feature. For example, a valve feature might have several parts: the valve body, the gaskets, and the flanges.

Taps - Allows you to select only taps in a graphic view or in the **Workspace Explorer**.

Welds - Allows you to select only welds in the graphic view or in the **Workspace Explorer**.

Clamps - Allows you to select only clamps in the graphic view or in the **Workspace Explorer**.

Connections - Allows you to select only connections in the graphic view or in the **Workspace Explorer**. The software creates a connection whenever a piping component is connected to another component, an equipment nozzle, or a pipe.

Spools - Allows you to select only spools in a graphic view or on the **Assembly** tab in the **Workspace Explorer**. You can break pipelines into spools using the **Generate Spools** command.

Penetration Spools - Allows you to select only penetration spools. Before you can create penetration spools, you must place a penetration plate in the Hangers and Supports task.

Equipment - Allows you to select only equipment in a graphic view or in the **Workspace Explorer**.

Piping Nozzle - Allows you to select only pipe nozzles in a graphic view or in the **Workspace Explorer**.

All - Allows you to select any object, even objects created in another task.

Use the **Inside** fence command to select all objects entirely inside the fence.

Use the **Inside/Overlapping** fence command to select all objects entirely inside the fence and those objects outside but touching the fence at some point.

Related Topics

- *Pipe Run Properties Dialog Box*, page 81
- *Pipe Straight Feature Properties Dialog Box*, page 91
- *Pipe Turn Feature Properties Dialog Box*, page 95
- *Pipe Weld Properties Dialog Box*, page 101
- *Pipeline System Properties Dialog Box*, page 79

Edit Pipeline Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipeline** in the **Locate Filter** box.
3. Select the pipeline to edit.
4. Click **Edit > Properties**.

Related Topics

- *Pipeline System Properties Dialog Box*, page 79

Edit Pipe Run Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Runs** in the **Locate Filter** box.
3. Select the pipe run to edit.
4. Click **Edit > Properties**.

Notes

- You can edit the flow direction of the pipe run by clicking the flow direction indicator on the pipe run and then selecting the new direction.
- You also can click **Properties** on the ribbon to edit properties.

Related Topics
- *Pipe Run Properties Dialog Box*, page 81

Edit Branch Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter** box.
3. Select the branch, such as a tee, to edit.
4. Click **Edit > Properties**.
5. Edit the branch properties.

Notes

- You also can click **Properties** on the ribbon to edit branch properties.

Related Topics
- *Delete a Branch*, page 104
- *Edit Branch Feature Ribbon*, page 44
- *Edit Branch Properties*, page 71
- *Move a Branch*, page 65

Edit Straight Pipe Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter** box.
3. Select the straight pipe to edit.
4. Click **Edit > Properties**.

Note

- You also can click **Properties** on the ribbon to edit properties.

Related Topics
- *Delete a Straight Pipe*, page 105
- *Edit Pipe Straight Feature Ribbon*, page 41
- *Edit Straight Pipe Properties*, page 72
- *Move a Straight Pipe*, page 67
- *Pipe Straight Feature Properties Dialog Box*, page 91

Edit Turn Feature Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter** box.
3. Select the turn to edit.
4. Click **Edit > Properties**.

Note

- You also can click **Properties** on the ribbon to edit properties.

Related Topics
- *Delete a Turn*, page 106
- *Edit Turn Feature Properties*, page 72
- *Edit Turn Feature Ribbon*, page 47
- *Move a Turn*, page 68
- *Pipe Turn Feature Properties Dialog Box*, page 95

Edit Part Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Parts** in the **Locate Filter**.
3. Select the part, such as a gasket or flange, to edit.
4. Click **Edit > Properties**.

Related Topics

- *Edit Part Properties*, page 73
- *Edit Pipe Run Properties*, page 71
- *Pipe Part Properties Dialog Box*, page 99

Common Property Tabs: An Overview

The software displays some common property tabs on the properties dialog boxes for all piping objects. Instead of repeatedly listing the common tabs with each piping object property dialog box, they are documented here for easy reference.

Configuration Tab

Displays the creation, modification, and status information about an object.

Plant - Displays the name of the plant. You cannot change this value.

Permission Group - Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in the Project Management task.

Status - Specifies the current status of the selected object or filter. Depending on your access level, you may not be able to change the status of the object.

Created - Displays the date and time that the object was created.

Created by - Displays the user name of the person who created the object.

Modified - Displays the date and time when the object was modified.

Modified by - Displays the user name of the person who modified the object.

Connections Tab

Displays the connection information for the object, the properties and their values, as defined in the reference data. For more information about the information defined in the reference data, see the *Piping Reference Data Guide*.

Connector - Select the connector for which you want to view properties.

Property - Displays the name of the property as defined in the reference data.

Value - Displays the value of the corresponding property.

Definition Tab

The **Definition** tab displays the object properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one object and then selected the properties command, only the common properties between the selected objects appears.

The properties that appear depend on what you defined in the reference data. Refer to the *Piping Reference Data Guide* for more information on the properties.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

Note

- Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, only one of the notes will show on the drawing. It is important to know about and consider this situation when defining notes on an object in the modeling phase.

  For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point - Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name - Lists all notes for the selected key point on the object.

Date - Displays the date the note was created. The system automatically supplies the date.

Time - Displays the time the note was created. The system automatically supplies the time.

Purpose of note - Specifies the purpose of the note.

Author - Displays the logon name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text - Defines the note text. The software does not limit the length of the note text.

New Note - Creates a new note on the object.

Standard Note - Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note - Highlights the note in the graphic view so you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note - Deletes the currently displayed note.

Occurrence Tab

Displays instance-specific information about the object.

Category - Select the properties that you want to view for the object. Object properties are divided into several different categories: Standard, Weight and CG, Fabrication and Construction, and Bend Data.

Standard

Name - Displays the name of the object.

Property/Value table - Displays all the occurrence properties for the object as defined in the reference data. If the table is blank, the object for which you are viewing properties does not have any occurrence properties defined for it in the reference data. For more information about occurrences defined in the reference data, see the *Piping Reference Data Guide*.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed in global system coordinates along the X-, Y-, and Z-axes.

Dry Weight - Displays the dry weight of the object.

Wet Weight - Displays the wet weight of the object.

Dry CG X - Displays the X-axis location of the dry center-of-gravity.

Dry CG Y - Displays the Y-axis location of the dry center-of-gravity.

Dry CG Z - Displays the Z-axis location of the dry center-of-gravity.

Wet CG X - Displays the X-axis location of the wet center-of-gravity.

Wet CG Y - Displays the Y-axis location of the wet center-of-gravity.

Wet CG Z - Displays the Z-axis location of the wet center-of-gravity.

Fabrication and Construction

Fabrication Requirement - Select the fabrication requirement for the selected object. If you want to add, edit, or remove values that are available for selection, edit the **Fabrication Type** sheet in the **AllCodeLists.xls** workbook.

Fabrication Type - Select the fabrication type for the selected object. If you want to add, edit, or remove values that are available for selection, edit the **Fabrication Type** sheet in the **AllCodeLists.xls** workbook.

Construction Requirement - Select the construction requirement for the selected object. If you want to add, edit, or remove values that are available for selection, edit the **Construction Type** sheet in the **AllCodeLists.xls** workbook.

Construction Type - Select the construction type for the selected object. If you want to add, edit, or remove values that are available for selection, edit the **Construction Type** sheet in the **AllCodeLists.xls** workbook.

Bend Data

Note

- The Bend Data category is only available if the administrator has bulk loaded the Pipe Bending Manufacturability Rules.xls workbook, and you are viewing properties of a pipe part with bends defined.

Minimum Length Check - Set this option to **True** if you want the software to validate pipe bending manufacturability rules for the selected pipe part.

Bend Type - Defines the pipe bend type.

Preferred Bending Machine - Displays the name of the preferred bending machine as defined in the reference data.

Minimum Tangent Data Set - Select the set of minimum tangent lengths to use with this pipe bend configuration.

End Trimming - Specify whether or not ends are trimmed.

Grip Length - Defines the grip length.

Pull Length - Defines the pull length.

Number of Bends in Piece - Displays the number of bends in the pipe part.

Number of Tangents in Piece - Displays the number of tangents in the pipe part.

Related Topics

- *Common Property Tabs: An Overview*, page 74

Relationship Tab

Displays all objects related to the object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Name - Displays the name of the related object.

Type - Displays the type of related object.

Go To - Displays the properties of the selected object.

Pipeline System Properties Dialog Box

Displays pipeline system properties for review and editing.

Related Topics

- *Configuration Tab*, page 74
- *Edit Pipeline Properties*, page 70
- *General Tab (Pipeline System Properties Dialog Box)*, page 79
- *Notes Tab*, page 75
- *Specifications Tab (Pipeline System Properties Dialog Box)*, page 80

General Tab (Pipeline System Properties Dialog Box)

Displays and defines the general properties of the selected pipeline.

Name - Specifies the name of the pipeline. This name is usually generated by the active name rule. You can type a different name for the pipeline if you want. The name must be unique throughout the life cycle of the plant.

Name rule - Specifies the name rule used to generate the name. Set this option to **User Defined** if you want to specify the name yourself.

Type - Displays the type of object. On the **Pipeline Properties** dialog box, the type is **Pipeline System**.

Parent System - Displays the system to which the pipeline belongs. You can change the parent system of the pipeline if needed.

Description - Type a general description of the pipeline.

Sequence Number - Type an identification number for the pipeline. Generally, this is defined in the P&ID data.

Fluid Requirement - Select the fluid requirement for the pipeline. You must select a fluid requirement.

Fluid Type - Select the fluid type for the pipeline. You must select a fluid type.

Correlation Status - Displays whether the pipeline has been correlated to a pipeline in a P&ID.

Correlation Basis - Specifies if the pipeline is correlated to a P&ID pipeline. Select **Correlate object** if the pipeline has a correlating pipeline in a P&ID. Select **No correlation is required** if the pipeline does not have a correlating pipeline in a P&ID.

- *Pipeline System Properties Dialog Box*, page 79

Specifications Tab (Pipeline System Properties Dialog Box)

Displays all the allowed specifications for the selected pipeline. You define piping specifications in the reference data, refer to the *Piping Reference Data Guide* for more information. You can change which specifications are available for the pipeline by using the **Allowed Specifications** command in the System and Specifications task.

Related Topics

- *Pipeline System Properties Dialog Box,* page 79

Pipe Run Properties Dialog Box

Displays pipe run properties for review and editing.

Related Topics

- *Configuration Tab*, page 74
- *Edit Pipe Run Properties*, page 71
- *General Tab (Pipe Run Properties Dialog Box)*, page 81
- *Notes Tab*, page 75
- *Relationship Tab*, page 78

General Tab (Pipe Run Properties Dialog Box)

Displays and defines the general properties of the selected pipe run.

Note

- Because pipe run properties are customizable in the piping reference data, only the properties that are required by the software are documented.

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

Standard

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

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

Name Rule - Specify the naming rule that you want to use to name this pipe run. You can select one of the listed rules or select **User Defined** to specify the pipe run name yourself in the **Name** box.

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

Nominal Diameter - Select the nominal pipe diameter (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 display on a session file basis using the **Tools > Options** command on the **Units of Measure** tab.

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

Minimum Slope - Specifies the slope for the pipe run. You can specify the slope as a ratio, a percentage, or in degrees.

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

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

Testing

Testing Requirements - Specifies whether non-destructive testing of welds is required.

Testing Type - Select the type of non-destructive weld testing for the welds. If you want to add, edit, or remove values that are available for selection, edit the **Testing Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

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

Temperature and Pressure

Design Maximum Temperature - Specify the maximum design temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Design Minimum Temperature - Specify the minimum design temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Design Maximum Pressure - Specify the maximum design pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Operating Maximum Temperature - Specify the maximum operating temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Operating Minimum Temperature - Specify the minimum operating temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Operating Maximum Pressure - Specify the maximum operating pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Testing Maximum Temperature - Specify the maximum testing temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Testing Minimum Temperature - Specify the minimum testing temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Testing Maximum Pressure - Specify the maximum testing pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Design Minimum Pressure - Specify the minimum design pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Operating Minimum Pressure - Specify the minimum operating pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Testing Minimum Pressure - Specify the minimum testing pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Interior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Exterior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Insulation and Tracing

Insulation Specification - Select the insulation specification that you want to use for the pipe run. Select **More** to choose an insulation specification that is allowed for the system. You assign which insulation specifications are available in which systems in the System and Specifications task.

Insulation Purpose - Displays the purpose of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Purpose** box. However, if **Insulation Specification** is manually defined, you can select the purpose of the insulation in this box. Available purposes are read from the reference data.

Insulation Material - Displays the material of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Material** box. However, if **Insulation Specification** is manually defined, you can select the insulation material that you want to use.

Insulation Thickness - Displays the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data.

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the pipe run is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the run. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Responsibility

Cleaning Responsibility - Select the party responsible for cleaning the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Design Responsibility - Select the party responsible for designing the object. If you want to add, edit, or remove values that are available for selection, edit the **Design Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Fabrication Responsibility - Select the party responsible for fabricating the object. If you want to add, edit, or remove values that are available for selection, edit the **Fabrication Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Installation Responsibility - Select the party responsible for installing the object. If you want to add, edit, or remove values that are available for selection, edit the **Installation Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Painting Responsibility - Select the party responsible for painting the object. If you want to add, edit, or remove values that are available for selection, edit the **Painting Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Requisition Responsibility - Select the party responsible for ordering the object. If you want to add, edit, or remove values that are available for selection, edit the **Requisition Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Supply Responsibility - Select the party responsible for delivering the object. If you want to add, edit, or remove values that are available for selection, edit the **Supply Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Testing Responsibility - Select the party responsible for testing the weld on the object. If you want to add, edit, or remove values that are available for selection, edit the **Testing Responsibility** sheet in the **AllCodeLists.xls** workbook in the reference data.

Related Topics

- *Pipe Run Properties Dialog Box*, page 81

Pipe End Feature Properties Dialog Box

Displays end feature properties for review and editing.

Related Topics
- *Configuration Tab*, page 74
- *General Tab (Pipe End Feature Properties Dialog Box)*, page 87
- *Relationship Tab*, page 78

General Tab (Pipe End Feature Properties Dialog Box)

Displays and defines the general properties of the selected end feature.

Pipe end feature properties are divided into different categories: **Standard**, **Insulation and Tracing**, and Surface Treatment and Coating. You select which category that you want to define values for by using the **Category** option.

Standard

Pipeline - Displays the pipeline system to which the end feature belongs. You cannot change the pipeline of the end feature.

Pipe Run - Displays the pipe run to which the end feature belongs. You cannot change the pipe run of the end feature.

Specification - Specify the active pipe specification for the end feature. All pipe specifications that are allowed for the pipeline display for selection. If appears in the box, the pipe specification is inherited from the pipe run to which the end feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run Specification**.

Nominal Diameter - Specify the nominal diameter (NPD) of the end feature. If appears in the box, the NPD for the end feature is inherited from the pipe run to which the end feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run NPD**. If you have manually selected the part, the NPD of the selected part appears and cannot be changed.

Type - Displays the types of end features allowed by the selected **Specification** and **Nominal Diameter**. End features are listed by short codes. You can change the end feature type if needed. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Option - Displays a list of available end features. The software generates this list based on the pipe specification, the NPD, and the short code you selected in the **Type** box. For example, if you selected a **90E Default** in the **Type** box and more than one elbow works in this pipe run, the elbows appear here for selection. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Base Part Name - Displays the part name for the end feature. The part name is selected by the software using the pipe specification guidelines. Select **Select Manually** to override the pipe specification, and select the part number from the catalog. If you select a part manually, the **Nominal Diameter**, **Type** and **Option** properties are disabled.

Note

- The **Select Manually** option may not be available. It is enabled and disabled in the reference data.

Correlation Status - Displays whether or not the pipe end feature has been correlated to a pipe end feature in a P&ID.

Correlation Basis - Specifies if the pipe end feature is correlated to a P&ID pipe end feature. Select **Correlate object** if the pipe end feature has a correlating pipe end feature in a P&ID. Select **No correlation is required** if the pipe end feature does not have a correlating pipe run in a P&ID.

Insulation and Tracing

Insulation Specification - Displays the insulation specification for the object. If appears in the box, the insulation specification for the object is inherited from the pipe run to which the object belongs.

You define which insulation specifications are available for which systems in the System and Specifications task. You define insulation specifications in the reference data. If you select **Not Insulated**, the software disables the remaining insulation properties.

Select **User Defined** to specify the remaining insulation properties using custom settings.

Select **More** to select a insulation specification from the reference data. The software automatically enters information from the insulation specification. For more information, see *Insulation Specification Dialog Box*, page 41.

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

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

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

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the object is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the object. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Interior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Exterior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Related Topics

- *Pipe End Feature Properties Dialog Box*, page 87

Pipe Straight Feature Properties Dialog Box

Displays straight feature properties for review and editing.

Related Topics
- *Configuration Tab*, page 74
- *Edit Straight Pipe Properties*, page 72
- *General Tab (Pipe Straight Feature Properties Dialog Box)*, page 91
- *Relationship Tab*, page 78

General Tab (Pipe Straight Feature Properties Dialog Box)

Displays and defines the general properties of the straight feature.

Note

- Straight features inherit some properties from the parent pipe run. If you change the pipe run property, the corresponding straight properties also change automatically.

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

Standard

Pipeline - Displays the system to which the straight feature belongs. You cannot change the system of the straight feature.

Pipe Run - Specifies the pipe run to which the straight feature belongs. Other pipe runs in the same pipeline as the original pipe run appear for selection.

Specification - Specify the active pipe specification for the straight feature. All pipe specifications that are allowed for the System parent of the pipe run appear for selection. If appears in the box, the pipe specification is inherited from the pipe run to which the straight feature belongs. If the icon does not appear and you want to use the pipe specification from the pipe run, select **Use Run Specification**.

Nominal Diameter - Specify the nominal diameter (NPD) of the straight feature. If appears in the box, the NPD for the straight feature is inherited from the pipe run to which the straight feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run NPD**. If you have manually selected the part, the NPD of the selected part appears and cannot be changed.

Type - Displays the types of straight features allowed by the selected **Specification** and **Nominal Diameter**. Straight features are listed by short codes. You can change the straight feature type if needed. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Option - Displays a list of available straight features. The software generates this list based on the pipe specification, the NPD, and the short code that you selected in the **Type** box. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Base Part Name - Displays the part name for the straight feature. The software selects the part number using the pipe specification guidelines. Choose the **Select Manually** option to override the pipe specification, and select the part number from the catalog. If you select a part manually, the **Nominal Diameter**, **Type** and **Option** properties are disabled.

Length - Displays the length of the straight feature. You cannot change the length of the straight feature.

Insulation and Tracing

Insulation Specification - Displays the insulation specification for the object. If appears in the box, the insulation specification for the object is inherited from the pipe run to which the object belongs.

You define which insulation specifications are available for which systems in the System and Specifications task. You define insulation specifications in the reference data. If you select **Not Insulated**, the software disables the remaining insulation properties.

Select **User Defined** to specify the remaining insulation properties using custom settings.

Select **More** to select a insulation specification from the reference data. The software automatically enters information from the insulation specification. For more information, see *Insulation Specification Dialog Box*, page 41.

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

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

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

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the object is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the object. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Interior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Exterior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Related Topics

- *Pipe Straight Feature Properties Dialog Box*, page 91

Pipe Turn Feature Properties Dialog Box

Displays elbows, pipe bends, and other turn properties for review and editing.

Related Topics
- *Configuration Tab*, page 74
- *Delete a Turn*, page 106
- *Edit Turn Feature Properties*, page 72
- *General Tab (Pipe Turn Feature Properties Dialog Box)*, page 95
- *Move a Turn*, page 68
- *Relationship Tab*, page 78

General Tab (Pipe Turn Feature Properties Dialog Box)

Displays and defines the general properties of the selected elbow or pipe bend.

Note

- Turn features inherit some properties from the parent pipe run. If you change the pipe run property, the corresponding turn properties also automatically change.

Pipe turn feature properties are divided into several different categories: **Standard**, **Insulation and Tracing**, and Surface Treatment and Coating. You select which category that you want to define values for by using the **Category** option.

Standard

Pipeline - Displays the name of the parent pipeline. This property is read-only.

Pipe Run - Specify the pipe run to which the turn belongs. Other pipe runs in the same pipeline as the original pipe run appear for selection.

Specification - Specify the active pipe specification for the turn. All pipe specifications that are allowed for the System parent of the pipe run display for selection. If appears in the box, the pipe specification is inherited from the pipe run to which the turn belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run Specification**.

Nominal Diameter - Specify the nominal diameter (NPD) of the turn. If appears in the box, the NPD for the turn is inherited from the pipe run to which the turn belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run NPD**. If you have manually selected the part, the NPD of the selected part appears and cannot be changed.

Type - Displays the types of turns allowed by the selected **Specification** and **Nominal Diameter**. Turns are listed by short codes. You can change the turn type if needed. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Option - Displays a list of available turns. The software generates this list based on the pipe specification, the NPD, and the short code you selected in the **Type** box. For example, if you selected a **90E Default** in the **Type** box and more than one elbow works in this pipe run, the elbows appear here for selection. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Base Part Name - Displays the part name for the turn. The part number is selected by the software using the pipe specification guidelines. Select the **Select Manually** option to override the pipe specification, and select the part number from the catalog. If you select a part manually, the **Nominal Diameter**, **Type** and **Option** properties are disabled.

Angle - Displays the angle of the turn feature. The angle is measured from the imaginary extension of the run.

Correlation Status - Displays whether the object has been correlated to an object in a P&ID.

Correlation Basis - Specifies if the object is correlated to a P&ID object. Select **Correlate object** if the object has a correlating object in a P&ID. Select **No correlation is required** if the object does not have a correlating object in a P&ID.

Insulation and Tracing

Insulation Specification - Displays the insulation specification for the object. If appears in the box, the insulation specification for the object is inherited from the pipe run to which the object belongs.

You define which insulation specifications are available for which systems in the System and Specifications task. You define insulation specifications in the reference data. If you select **Not Insulated**, the software disables the remaining insulation properties.

Select **User Defined** to specify the remaining insulation properties using custom settings.

Select **More** to select a insulation specification from the reference data. The software automatically enters information from the insulation specification. For more information, see *Insulation Specification Dialog Box*, page 41.

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

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

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

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the object is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the object. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Interior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the

Exterior Surface Treatment sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Related Topics

- *Pipe Turn Feature Properties Dialog Box*, page 95

Pipe Part Properties Dialog Box

Displays properties for pipe parts for review and editing.

Related Topics
- *Configuration Tab*, page 74
- *Edit Part Properties*, page 73
- *Notes Tab*, page 75
- *Relationship Tab*, page 78

Connection Properties Dialog Box

Displays connection properties for review and editing. Use the **Relationships** tab to view the properties of the bolts, gaskets, washers, nuts, welds, and other pieces that are in the connection.

Related Topics
- *Configuration Tab*, page 74
- *General Tab (Connection Properties Dialog Box)*, page 99
- *Relationship Tab*, page 78

General Tab (Connection Properties Dialog Box)

Displays and defines the general properties of the selected connection.

Category - Select the properties that you want to see. You can view system properties or user-defined properties.

Pipeline - Displays the pipeline to which the connection belongs. You cannot change this value.

Connected Type - Displays the type of connection. You cannot change this value.

Related Topics
- *Connection Properties Dialog Box*, page 99

Pipe Bolt Set Properties Dialog Box

Displays bolt set properties for review and editing.

Related Topics
- *Configuration Tab*, page 74
- *Definition Tab*, page 75
- *Occurrence Tab*, page 76
- *Relationship Tab*, page 78

Pipe Gasket Properties Dialog Box

Displays gasket properties for review and editing.

Related Topics
- *Occurrence Tab*, page 76
- *Relationship Tab*, page 78

Pipe Weld Properties Dialog Box

Displays weld properties for review and editing.

Related Topics

- *Configuration Tab*, page 74
- *Edit Weld Properties*, page 118
- *General Tab (Pipe Weld Properties Dialog Box)*, page 101
- *Relationship Tab*, page 78

General Tab (Pipe Weld Properties Dialog Box)

Displays and defines the general properties of the selected weld. Properties are defined in the reference data.

Weld properties are divided into different categories: **Standard** and **Testing**. You select which category that you want to define values for by using the **Category** option.

Standard

Name - Displays the name of the weld. The weld name is based on the **Name Rule** selection. If you want to type a new name for the weld, then in the **Name Rule** option, select **User Defined**, and then type a name for the weld in the **Name** box.

Name Rule - Specify the naming rule that you want to use to name this weld. You can select one of the listed rules or select **User Defined** to specify the pipe run name yourself in the **Name** box.

Class - Displays the weld class. Valid classes are listed in the **AllCodeLists.xls** workbook on the **Weld Type** sheet.

Type - Specifies the weld type. Valid types are listed in the **AllCodeLists.xls** workbook on the **Weld Type** sheet. If appears in the box, the weld type is inherited from the pipe run to which the weld belongs. If the icon does not appear, and you want to use the weld type from the pipe run, select **Use Run Specification**.

Welding Procedure Specification - Specifies the welding procedure specification.

Materials Category - Displays the materials category for the weld.

Materials Grade - Displays the materials grade for the weld.

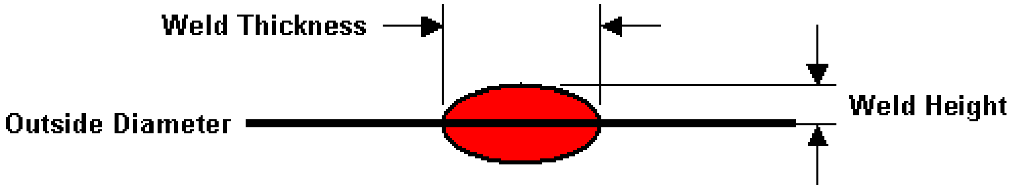
Gap - Displays the gap between the welded ends of the welded joint.

Pass Length - Displays the weld length.

Thickness - Displays the weld thickness.

Height - Displays the weld height.

Sequence Id - Displays the sequence ID for the weld. The sequence ID is generated when you run the **Sequence Objects** command.



Cross Section View of Weld

Testing

Testing Requirements - Specifies whether non-destructive testing of welds is required.

Testing Type - Select the type of non-destructive weld testing for the welds. If you want to add, edit, or remove values that are available for selection, edit the **Testing Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

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

Related Topics
- *Pipe Weld Properties Dialog Box*, page 101

Deleting Features: An Overview

Just as you place features to place parts in your pipe run, you delete features to remove unwanted parts. You cannot delete parts directly. The software attempts to maintain the design integrity of the model by adjusting all previously connected features. The behavior of various objects during deletion is detailed below:

Pipelines

- Deleting a pipeline deletes all pipe runs, features, and parts associated with that pipeline.
- The Piping task cuts the pipeline legs at all boundaries and creates gaps. The software neither extends nor inserts a straight feature.

Pipe Runs

- Deleting the run deletes all features (and thereby all parts) belonging to the run.
- The Piping task cuts the pipe run legs at all boundaries and creates gaps. The software neither extends nor inserts a straight feature.

Straight Features

- Deleting a straight feature removes connected turn features. The straight feature connected by the associated turn feature is extended to the turn point.
- Deleting a straight feature also deletes any adjacent branch features if the straight feature is connected to the branch port of the feature. For tee-type branches, the system replaces the header portion of the branch with a straight feature. Branching valves, instruments and piping specialties will not be deleted in either of these cases mentioned above.
- Deletion of a straight feature adjacent to a split feature removes the split feature if no other part is connected to the split feature.
- Deletion of a straight feature adjacent to a run change feature removes the run change feature.
- If the straight feature connects to a component, the software does not delete the component when the straight feature is deleted. However, any mating parts generated to facilitate connection to the component are deleted.

Turn Features

- Straight features connected to the deleted turn extend to the turn point.
- Deleting a turn feature removes all adjacent run change features and mating parts.

- When a branch feature is adjacent to the turn feature, the software inserts a straight feature to avoid deleting a branch.

Inline Components or Split Features

- If an inline component or split feature is deleted, the software deletes all mating parts. The two connected straight features merge, forming one straight feature, provided they share the same parent pipe run and the same nominal piping diameter (NPD).

Related Topics

- *Delete a Branch*, page 104
- *Delete a Component*, page 105
- *Delete a Pipe Run*, page 107
- *Delete a Pipeline*, page 107
- *Delete a Split*, page 106
- *Delete a Straight Pipe*, page 105
- *Delete a Tap*, page 142
- *Delete a Turn*, page 106

Delete a Branch

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the branch to delete.
4. Click **Delete**.

Related Topics

- *Delete a Branch*, page 104
- *Deleting Features: An Overview*, page 103
- *Edit Branch Properties*, page 71
- *Move a Branch*, page 65

Delete a Straight Pipe

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the pipe feature to delete.
4. Click **Delete** ✕.

Related Topics

- *Delete a Straight Pipe*, page 105
- *Deleting Features: An Overview*, page 103
- *Edit Straight Pipe Properties*, page 72
- *Move a Straight Pipe*, page 67

Delete a Component

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the component, such as a valve, to delete.
4. Click **Delete** ✕.

Notes

- The software automatically deletes all mating components (flanges and gaskets, for example) that were placed with the component that you deleted.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Edit Component Ribbon*, page 124
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63

Delete a Split

1. Click **Select**.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the split or takedown joint to delete.
4. Click **Delete**.

Related Topics

- *Delete a Split*, page 106
- *Deleting Features: An Overview*, page 103

Delete a Turn

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the turn to delete.
4. Click **Delete**.

Related Topics

- *Delete a Turn*, page 106
- *Deleting Features: An Overview*, page 103
- *Edit Turn Feature Properties*, page 72
- *Move a Turn*, page 68

Delete a Pipe Run

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Runs** in the **Locate Filter**.
3. Select the pipe run to delete.
4. Click **Delete** .

Related Topics

- *Change the Flow Direction of a Pipe Run*, page 61
- *Copy a Pipe Run*, page 59
- *Delete a Pipe Run*, page 107
- *Deleting Features: An Overview*, page 103
- *Extend an Existing Pipe Run*, page 60

Delete a Pipeline

1. Click **Select** on the vertical toolbar.
2. Select **Pipelines** in the **Locate Filter**.
3. Select the pipeline to delete.
4. Click **Delete** .

Related Topics

- *Deleting Features: An Overview*, page 103

Inserting Splits: An Overview

The **Insert Split** command can be used either during pipe routing or by itself. While routing, when you select the **Insert Split** command the software places the default split type at the location of the last point you selected.

If you select the **Insert Split** command and then select an existing pipe, you have two options.

- You can have the software cut the pipe feature into two features that represent two pipe parts joined by a weld joint or a takedown joint.
- You can have the software cut the pipe feature into two features that represent a single pipe part using a feature break. When splitting using a feature break, no weld or connection is placed so the two-features combined represent 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 break in the line.

When you insert a takedown joint, the software automatically selects and positions the necessary mating parts based on the split feature information in the pipe specification defined for the pipe run. If you select a sleeved weld joint, the software automatically selects and positions the sleeve. The software uses the reference data that defines specific parts for standard splits. The allowed parts are generally a very restricted subset of the parts (flanges and sleeves) that are allowed for general placement by the specification.

You can select an existing split and edit any part of the split feature definition by clicking the appropriate ribbon control.

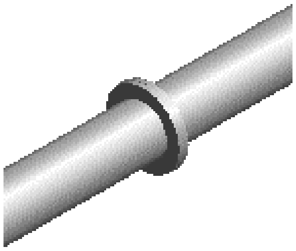
Note

- Special takedown joints not covered by the pipe specification split rules can be manually created by inserting flange components using the **Insert Component** command. When you place a flange manually as a component, the software automatically cuts the pipe and inserts the appropriate mating part for the flange.

Related Topics

- *Edit Split Properties*, page 117
- *Insert a Feature Break*, page 117
- *Insert a Takedown Joint*, page 116
- *Insert a Welded Split*, page 116

Insert Split Command



Divides the pipe into sections. You can use any split feature defined in the reference data, such as a flange or a butt weld, to connect the two resulting pipes.

You can select this command while routing pipe, or select it after routing pipe to insert a split in an existing pipe. If you select the **Insert Split** command and then select an existing pipe, you have two options.

- You can have the software cut the pipe feature into two features that represent two pipe parts joined by a weld joint or a takedown joint.
- You can have the software cut the pipe feature into two features that represent a single pipe part using a feature break. When splitting using a feature break, no weld or connection is placed so the two-features combined represent 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 break in the line.

Related Topics
- *Deleting Features: An Overview*, page 103
- *Edit Split Properties*, page 117
- *Insert a Feature Break*, page 117
- *Insert a Takedown Joint*, page 116
- *Insert a Welded Split*, page 116
- *Inserting Splits: An Overview*, page 109
- *Moving Pipe Features: An Overview*, page 63

Insert Split Ribbon

Sets options for adding a break into a pipe run.

Pipe Split Feature Properties - Opens the **Pipe Split Feature Properties** dialog box in which use you define addition properties that are not available on the ribbon. For more information, see *Pipe Split Feature Properties Dialog Box*, page 112.

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

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

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

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

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

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

Type - Select the type of split component to place. The software generates this list based on the components allowed by the pipe specification for the selected pipe. You can also specify the component that you want to use by selecting **Specify Component Tag** and then entering the component tag.

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.

Related Topics

- *Edit Split Properties*, page 117
- *Insert a Feature Break*, page 117
- *Insert a Takedown Joint*, page 116
- *Insert a Welded Split*, page 116
- *Insert Split Command*, page 110
- *Specify Component Tag Dialog Box*, page 126

Edit Split Feature Ribbon

Displays options for modifying a split in a pipe.

Properties - Activates the properties dialog box. For more information, see *Pipe Split Feature Properties Dialog Box*, page 112.

Move From - Click to identify the origin of the move vector. If you do not define a starting point, the software assumes the current location of the split is the starting point.

Move To - Click to identify the termination point of the move vector.

Run - Displays the name of the pipe run or runs that are adjacent to the selected split feature. If two different runs are listed, you can specify either run as the owner of the split.

Weld Joint - Select this option to change the split to a welded joint.

Takedown Joint - Select this option to change the split to a takedown joint.

Type - Select the type of split. The software generates this list based on those components allowed by the pipe specification for the selected pipe.

Option - Displays the option of the selected pipe. Additionally, the **Option** list contains any options defined in the piping specification for the short code selected in the **Type** list. For example, if two different rating are available in the piping specification for a flange short code selected in the **Type** list, both options appear in this list. Selecting another value from the **Option** list updates the object accordingly.

Related Topics

- *Move a Feature Break*, page 118
- *Move a Split*, page 118
- *Moving Pipe Features: An Overview*, page 63

Pipe Split Feature Properties Dialog Box

Displays split feature properties for review and editing.

Related Topics

- *Configuration Tab*, page 74
- *Edit Split Properties*, page 117
- *General Tab (Pipe Split Feature Properties Dialog Box)*, page 112
- *Relationship Tab*, page 78

General Tab (Pipe Split Feature Properties Dialog Box)

Displays and defines the general properties of the split feature.

Note

- Split features inherit some properties from the parent pipe run. If you change the pipe run property, the corresponding split properties also automatically change.

Category - Pipe split properties are divided into different categories: **Standard**, **Insulation and Tracing**, and **Surface Treatment and Coating**. You can select the category to define values for by using the **Category** option.

Standard

Pipeline - Displays the system to which the split feature belongs. You cannot change the system of the split feature.

Pipe Run - Specifies the pipe run to which the split feature belongs. Other pipe runs in the same system as the original pipe run appear for selection.

Specification - Specifies the active pipe specification for the split feature. All pipe specifications that are allowed for the system parent of the pipe run appear for selection. If appears in the box, the pipe specification is inherited from the pipe run to which the split feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run Specification**.

Nominal Diameter - Specifies the nominal diameter (NPD) of the split feature. If ↘ appears in the box, the NPD for the split feature is inherited from the pipe run to which the split feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run NPD**. If you have manually selected the part, the NPD of the selected part appears and cannot be changed.

Weld/Takedown Category - Select the category for the split.

Type - Displays the types of split features allowed by the selected **Specification** and **Nominal Diameter**. Split features are listed by short codes. You can change the split feature type if needed. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Option - Displays a list of available split features. The software generates this list based on the pipe specification, the NPD, and the short code that you selected in the **Type** box. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Base Part Name - Displays the part name for the split feature. The software selects the part name using the pipe specification guidelines. Select the **Select Manually** option to override the pipe specification, and select the part number from the catalog. If you select a part manually, the **Nominal Diameter**, **Type**, and **Option** properties are disabled.

Correlation Status - Displays whether the split has been correlated to a split in a P&ID.

Correlation Basis - Specifies if the split is correlated to a P&ID split. Select **Correlate object** if the split has a correlating split in a P&ID. Select **No correlation is required** if the split does not have a correlating split in a P&ID.

Insulation and Tracing

Insulation Specification - Displays the insulation specification for the object. If ↘ appears in the box, the insulation specification for the object is inherited from the pipe run to which the object belongs.

You define which insulation specifications are available for which systems in the System and Specifications task. You define insulation specifications in the reference data. If you select **Not Insulated**, the software disables the remaining insulation properties.

Select **User Defined** to specify the remaining insulation properties using custom settings.

Select **More** to select a insulation specification from the reference data. The software automatically enters information from the insulation specification. For more information, see *Insulation Specification Dialog Box*, page 41.

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

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

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

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the object is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the object. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the

Interior Surface Treatment sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Exterior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Related Topics

- *Pipe Split Feature Properties Dialog Box*, page 112

Insert a Welded Split

1. Click **Insert Split** on the vertical toolbar.
2. Select the pipe to split.
3. Click **Welded Joint** on the ribbon.
4. Select the type and option for the split in the **Type** and **Option** boxes.
5. Click to define the position of the welded split.

Note

- You can use the **Tools > Pinpoint** and **Tools > Point Along** commands to locate more precisely the position of the split.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Edit Split Properties*, page 117
- *Insert a Feature Break*, page 117
- *Insert a Takedown Joint*, page 116
- *Insert Split Command*, page 110
- *Insert Split Ribbon*, page 110
- *Inserting Splits: An Overview*, page 109
- *Moving Pipe Features: An Overview*, page 63

Insert a Takedown Joint

1. Click **Insert Split** on the vertical toolbar.
2. Select the pipe to split.
3. Click **Takedown** on the ribbon.
4. Select the type and option for the split in the **Type** and **Option** boxes.
5. Click to define the position of the split.

Note

- You can use the **Tools > Pinpoint** and **Tools > Point Along** commands to locate more precisely the position of the split.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Edit Split Properties*, page 117
- *Insert a Feature Break*, page 117
- *Insert a Welded Split*, page 116
- *Insert Split Command*, page 110
- *Insert Split Ribbon*, page 110
- *Inserting Splits: An Overview*, page 109
- *Moving Pipe Features: An Overview*, page 63

Insert a Feature Break

1. Click **Insert Split** on the vertical toolbar.
2. Click **Feature Break** on the ribbon.
3. Select the pipe feature to break.
4. Click to define the position of the feature break.

Note

- You can use the **Tools > Pinpoint** and **Tools > Point Along** commands to locate more precisely the position of the feature break.
- When splitting using a feature break, no weld or connection is placed so the two-features combined represent 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 break in the line.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Edit Split Properties*, page 117
- *Insert a Feature Break*, page 117
- *Insert a Takedown Joint*, page 116
- *Insert a Welded Split*, page 116
- *Insert Split Command*, page 110
- *Insert Split Ribbon*, page 110
- *Inserting Splits: An Overview*, page 109
- *Moving Pipe Features: An Overview*, page 63

Edit Split Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter** box.
3. Select the split to edit.
4. Click **Edit > Properties**.

Notes

- You also can click **Properties** on the ribbon to edit properties.

Related Topics

- *Edit Split Feature Ribbon*, page 111

Edit Weld Properties

1. Click **Select** on the vertical toolbar.
2. Select **Welds** in the **Locate Filter** box.
3. Select the weld to edit from the **Workspace Explorer** or in the model.
4. Click **Edit > Properties**.

Related Topics

- *Pipe Weld Properties Dialog Box*, page 101

Move a Split

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the split or takedown joint to move.
4. Click **Move From** to identify the origin of the move vector. If you do not define a starting point, the software assumes the current location of the split is the starting point.
5. Click **Move To** .
6. Move the split to a new location.

Notes

- You can use the **Tools > Point Along** and **Tools > Pinpoint** commands to help locate the split more precisely.

Related Topics

- *Edit Split Feature Ribbon*, page 111
- *Moving Pipe Features: An Overview*, page 63

Move a Feature Break

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the feature break to move.
4. Click **Move** on the main toolbar.
5. Click **Move From** to identify the origin of the move vector. If you do not define a starting point, the software assumes the current location of the feature break is the starting point.
6. Click **Move To** .
7. Move the feature break to a new location.

Notes

- You can use the **Tools > Point Along** and **Tools > Pinpoint** commands to help locate the feature break more precisely.

Related Topics

- *Edit Split Feature Ribbon*, page 111
- *Moving Pipe Features: An Overview*, page 63

Inserting Components: An Overview

The **Insert Component** command adds valves, strainers, laterals, and other components to a pipe run. You can add components either during the routing of a pipe run or after the pipe has been routed.

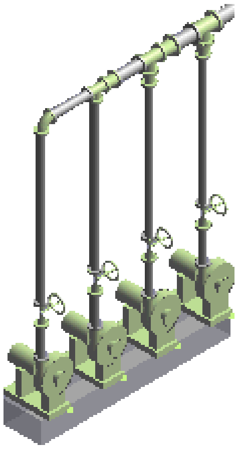
The system uses the pipe specification, nominal diameter of the selected pipe run, and the geometry of the insertion point to filter the available components. For example, if the insertion point is not at the end of a pipe run or at an equipment nozzle, turn components are not included in the list of available components. When you insert a component, the software generates any mating and connection parts required to connect the inserted part to the adjacent objects.

When inserting components, you can use the **Tools > Pinpoint** and **Tools > Point Along** commands to position components precisely in a pipe run.

Related Topics

- *Edit Component Properties*, page 134
- *Insert a Component while Routing*, page 131
- *Insert a Component*, page 131
- *Insert a Piping Specialty Item*, page 132
- *Insert an Instrument*, page 132
- *Move a Component*, page 134
- *Rotate a Component*, page 135
- *Select Component Settings*, page 133

Insert Component Command



Places components such as valves, steam traps, strainers, tees, turns, fire protection items, and other piping specialty items. When inserting components, the software automatically splits the pipe and adds the necessary mating components.

You can select a component in a P&ID, and then select this command to correlate the component in the P&ID to the component in your model.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Edit Component Properties*, page 134
- *Insert a Component while Routing*, page 131
- *Insert a Component*, page 131
- *Insert a Piping Specialty Item*, page 132
- *Insert an Instrument*, page 132
- *Inserting Components: An Overview*, page 121
- *Move a Component*, page 134
- *Moving Pipe Features: An Overview*, page 63
- *Rotate a Component*, page 135
- *Select Component Settings*, page 133

Insert Component Ribbon

Sets options for adding a valve or other component into a pipe run.

Properties - Activates the **Pipe Component Feature Properties** dialog box. You can use this dialog box to specify how the specifications of the component are defined. You can specify component specifications in one of three ways:

- Select the pipe run into which the component is being placed.
- Select a specification directly.
- Select the part specification in the catalog.

Select Straight or End Feature - Select the pipe, pipe end, or nozzle on which you want to place a component.

Type - Displays a list of available component short codes based on the piping specification and nominal diameter of the object selected along with the geometry at the insertion location. Select the type of component that you want to insert. You also can select the **Specify Component Tag** option to specify the component that you want to use by typing in the component tag. Also select **Specify Component Tag** to place instrument specialty or piping specialty items.

Option - Displays the option of the selected component. In addition, the **Option** list contains any options defined in the piping specification for the short code selected in the **Type** list. Selecting a value from the **Option** list updates the objects.

Enter Insertion Point - Positions the component that you are placing along the pipe that you selected. You can use the **Tools > Pinpoint** and **Tools > Point Along** commands to define the precise location of the component on the pipe run.

Finish - Places the component in the pipe run.

Run - Specifies to which pipe run the component is to belong. If the component that you are placing is a branching component, the software displays a list of run from which you can select. If you need a new run, select **<New Pipe Run>**. For more information on creating a pipe run, see *New Pipe Run Dialog Box*, page 33.

Name - Displays the name of the component being inserted.

Flip - Select this option to change the component port that is connected to the selected port. If you are inserting a lateral, tee, or other tee-type branch into a length of pipe or between two other components, only the inline ports of the component are available. However, if you insert the same component at the end of a pipe or at a nozzle, all component ports are available.

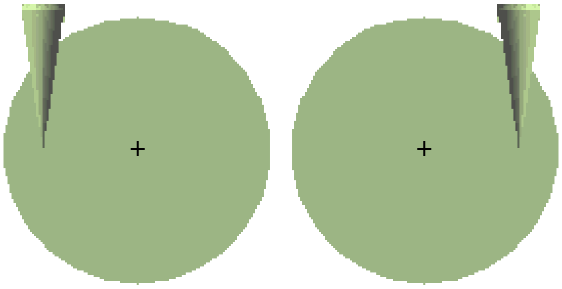
Reference position - Slides the component that you are inserting so that its origin or selected port is positioned at the insertion point. If the selected reference position does not lie along the axis of the leg, then the software projects the position so that it intersects the axis and the component slides so that the projected point is located at the insertion point.

Rotate - Select to rotate the component about the pipe interactively.

Angle - Type an angle at which you want the component rotated about the pipe.

Operator Angle - Type an angle at which you want the valve operator rotated about valve stem. This angle does not affect the valve body.

Branch Position - Select which side of the pipe to place the non-radial tangential or non-tangential component.



Related Topics

- *Insert a Component*, page 131
- *Insert Component Command*, page 122
- *Specify Component Tag Dialog Box*, page 126

Edit Component Ribbon

Changes options for components placed in the model.

Properties - Activates the properties dialog box for the selected component. See *Pipe Component Feature Properties Dialog Box*, page 127 for more information.

Move From - Click to identify the origin of the move vector. If you do not define a starting point, the software assumes the current location of the component is the starting point.

Move To - Click to identify the termination point of the move vector.

Run - Specifies the name of the pipe run in which the component is placed.

Type - Displays the short code associated with the selected component. In addition, the **Type** list contains the short codes associated with any other components with similar geometry defined in the piping specification for the current nominal diameter. Selecting an entry from the **Type** list replaces the selected component with one of a different type.

Option - Displays the option of the selected component, if applicable. In addition, the **Option** list contains any other options defined in the piping specification for the short code selected in the **Type** list. For example, if two different rating values were available in the piping specification for the short code selected in the **Type** list, both options would appear in this list. Selecting a value from the **Option** list updates the object.

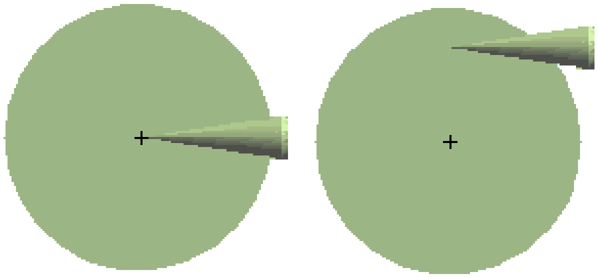
Name - Displays the name of the component that you are modifying. Type another name if needed.

Flip - Select this option to change the component port that is connected to the selected port. If you are inserting a lateral, tee, or other tee-type branch into a length of pipe or between two other components, only the inline ports of the component are available. However, if you insert the same component at the end of a pipe or at a nozzle, all component ports are available.

Rotate - Select to rotate the component about the pipe interactively.

Angle - Type an angle at which you want the component rotated about the pipe.

Branch Position - Select the position of the branch centerline relative to the header centerline. Select **Radial** to place the branch such that its centerline intersects the header centerline. Select **Non-Radial** to place the branch such that its center line does not intersect the header centerline. If you select **Non-Radial**, you can specify the distance the branch centerline is offset from the header centerline up to being tangent to the outside diameter of the header.



Radial and Non-Radial Positions

Related Topics

- *Insert Component Command*, page 122
- *Move a Component*, page 134
- *Rotate a Component*, page 135

Specify Component Tag Dialog Box

Allows you to specify the component that you want to use by the component tag. You activate this dialog box by selecting the **Insert Component** command, and then selecting **<Specify Component Tag>** from the **Type** option.

Component tag - Type the tag of the component that you want to use. Select **Browse Instruments** to place an instrument. Select **Browse Piping Specialties** to place a piping specialty item.

Related Topics

- *Insert Component Ribbon*, page 123
- *Insert Split Ribbon*, page 110

Pipe Component Feature Properties Dialog Box

Displays properties for component features, such as valves, for review and editing.

Related Topics

- *Configuration Tab*, page 74
- *Edit Split Properties*, page 117
- *General Tab (Pipe Component Feature Properties Dialog Box)*, page 127
- *Relationship Tab*, page 78

General Tab (Pipe Component Feature Properties Dialog Box)

Displays and defines the general properties of the component feature.

Note

- Component features inherit some properties from the parent pipe run. If you change the pipe run property, the corresponding component properties also automatically change.

Category - Component properties are divided into different categories: **Standard**, **Insulation and Tracing**, and **Surface Treatment and Coating**. You can select the category to define values for by using the **Category** option.

Standard

Pipeline - Displays the system to which the component feature belongs. You cannot change the system of the component feature.

Pipe Run - Specifies the pipe run to which the component feature belongs. Other pipe runs in the same system as the original pipe run appear for selection.

Specification - Specifies the active pipe specification for the component feature. All pipe specifications that are allowed for the system parent of the pipe run appear for selection. If appears in the box, the pipe specification is inherited from the pipe run to which the component feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run Specification**.

Nominal Diameter - Specifies the nominal diameter (NPD) of the component feature. If appears in the box, the NPD for the component feature is inherited from the pipe run to which the component feature belongs. If the icon does not appear, and you want to use the pipe specification from the pipe run, select **Use Run NPD**. If you have manually selected the part, the NPD of the selected part appears as a read-only value.

Type - Displays the types of component features allowed by the selected **Specification** and **Nominal Diameter**. component features are listed by short codes. You can change the component feature type if needed. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Option - Displays a list of available component features. The software generates this list based on the pipe specification, the NPD, and the short code that you selected in the **Type** box. If you have manually selected the part, this box displays **Part selected manually** and cannot be changed.

Base Part Name - Displays the part name for the component feature. The software selects the part name using the pipe specification guidelines. Select the **Select Manually** option to override the pipe specification, and select the part number from the catalog. If you select a part manually, the **Nominal Diameter**, **Type** and **Option** properties are disabled.

Correlation Status - Displays whether the component has been correlated to a component in a P&ID.

Correlation Basis - Specifies if the component is correlated to a P&ID component. Select **Correlate object** if the component has a correlating component in a P&ID. Select **No correlation is required** if the component does not have a correlating component in a P&ID.

Insulation and Tracing

Insulation Specification - Displays the insulation specification for the object. If appears in the box, the insulation specification for the object is inherited from the pipe run to which the object belongs.

You define which insulation specifications are available for which systems in the System and Specifications task. You define insulation specifications in the reference data. If you select **Not Insulated**, the software disables the remaining insulation properties.

Select **User Defined** to specify the remaining insulation properties using custom settings.

Select **More** to select a insulation specification from the reference data. The software automatically enters information from the insulation specification. For more information, see *Insulation Specification Dialog Box*, page 41.

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

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

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

Insulation Temperature - Type the temperature. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Heat Tracing Requirement - Select whether or not the object is heat-traced. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Type - Select the type of heat-tracing. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium - Select the heat-tracing medium to apply to the object. If you want to add, edit, or remove values that are available for selection, edit the **Heat Tracing Medium** sheet in the **AllCodeLists.xls** workbook in the reference data.

Heat Tracing Medium Temperature - Specify the temperature of the heat-tracing medium. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Surface Treatment and Coating

Interior Surface Treatment - Select the treatment for the interior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Interior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Exterior Surface Treatment - Select the treatment for the exterior of the object. If you want to add, edit, or remove values that are available for selection, edit the **Exterior Surface Treatment** sheet in the **AllCodeLists.xls** workbook in the reference data.

Cleaning Requirement - Select the cleaning requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Cleaning Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Requirement - Select the steam out requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Steamout Requirement** sheet in the **AllCodeLists.xls** workbook in the reference data.

Steamout Pressure - Specify the steamout pressure for the object. Include the pressure unit of measure, psi for pounds per square inch for example, when specifying this value.

Steamout Temperature - Specify the steamout temperature for the object. Include the temperature unit of measure, K for Kelvin, F for Fahrenheit, or C for Celsius for example, when specifying this value.

Coating Requirement - Select the coating requirement for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Type - Select the type of coating for the object. If you want to add, edit, or remove values that are available for selection, edit the **Coating Type** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Color - Select the color of the object coating. If you want to add, edit, or remove values that are available for selection, edit the **Coating Color** sheet in the **AllCodeLists.xls** workbook in the reference data.

Coating Area - Enter total area that the coating covers.

Insert a Component

1. Click **Insert Component** on the vertical toolbar.
2. Select the pipe, pipe end, component port, or nozzle on which to insert the component.
3. Select the component type and option in the **Type** and **Option** boxes.
4. Click to define the position of the component if you are placing it in a straight feature.
5. If needed, change the position of the component using **Flip**, **Reference Position**, and **Angle** options on the ribbon.
6. Click **Finish**.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Insert Component Ribbon*, page 123
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63
- *Select Component Settings*, page 133

Insert a Component while Routing

1. Click **Insert Component** on the vertical toolbar.
2. Select the component type and option in the **Type** and **Option** boxes.
3. If needed, change the position of the component using **Flip**, **Reference Position**, and **Angle** options on the ribbon.
4. Click **Finish**.
5. Continue routing the pipe run.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Insert Component Ribbon*, page 123
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63
- *Select Component Settings*, page 133

Insert an Instrument

1. Click **Insert Component** on the vertical toolbar.
2. Select the pipe, pipe end, or nozzle in which to insert the instrument specialty object.
3. In the **Type** option, select **Specify Component Tag**.
4. In the **Component Tag** box, type the instrument tag or select **Browse Instruments**.
5. Select the instrument object to place.
6. Click to define the position of the component if you are placing it in a straight feature.
7. Click **Finish**.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Insert Component Ribbon*, page 123
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63
- *Select Component Settings*, page 133

Insert a Piping Specialty Item

1. Click **Insert Component** on the vertical toolbar.
2. Select the pipe, pipe end, or nozzle in which to insert the piping specialty item.
3. In the **Type** option, select **Specify Component Tag**.
4. In the **Component Tag** box, type the object tag or select **Browse piping specialty**.
5. Select the piping specialty item to place.
6. Click to define the position of the component if you are placing it in a straight feature.
7. Click **Finish**.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Insert Component Ribbon*, page 123
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63
- *Select Component Settings*, page 133

Place a Tee from a P&ID

1. Click **Framework > Retrieve** to retrieve the P&ID drawing that contains the pipe runs that branch. You can skip this step if the P&ID has already been retrieved.
2. Click **Framework > View P&ID** to view the P&ID drawing that contains pipe runs.
3. Review the P&ID to determine where you need to insert the tee or branch in the model. Specifically, you need to identify the header and branch pipe runs.
4. Click **Insert Component** on the vertical toolbar.
5. Select the header run in which to insert the tee or other branch component.
6. Select the component type and option in the **Type** and **Option** boxes.
7. Position the component on the header pipe run.
8. Click **Finish**.
9. Using the **Route Pipe** command, route the branch pipe run into the branch component that you just placed on the header pipe run.

Related Topics

- *Deleting Features: An Overview*, page 103
- *New Pipe Run Dialog Box*, page 33
- *Routing a Pipe Run: An Overview*, page 29

Select Component Settings

1. Click **Properties** on the ribbon.
2. If needed, select the type and option for the component in the **Type** and **Option** boxes. Other properties can be modified as needed.
3. Place the component.

Notes

- The **Configuration** tab and **Relationship** tab of the properties dialog box are available only when you edit an existing object.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Insert a Component*, page 131
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63

Edit Component Properties

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter** box.
3. Select the component, such as a valve, to edit.
4. Click **Edit > Properties**.
5. Edit the component properties.

Notes

- Use the **Rotate** option on the ribbon bar to dynamically rotate the component. Use the **Angle** option to type the rotation angle.
- You also can click **Properties** on the ribbon to edit component properties.

Move a Component

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the component, such as a valve, to move, and then drag it to the new location.

Notes

- You can also use the **Move From** and **Move To** ribbon commands to move the component. If you do not select a move from location, the current location of the object is used as the from location.
- You can use the **Tools > Point Along** and **Tools > Pinpoint** commands to locate the component precisely.

Related Topics

- *Deleting Features: An Overview*, page 103
- *Edit Component Ribbon*, page 124
- *Inserting Components: An Overview*, page 121
- *Moving Pipe Features: An Overview*, page 63

Rotate a Component

1. Click **Select** on the vertical toolbar.
2. Select **Pipe Features** in the **Locate Filter**.
3. Select the component, such as a valve, to rotate.
4. Click **Rotate** and move the mouse to rotate the component.
5. Click to release the component at the new rotation angle.

Note

- You also can enter a rotation angle in the **Angle** box.

Related Topics

- *Edit Component Ribbon*, page 124
- *Move a Component*, page 134

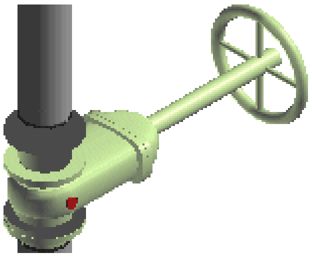
Inserting Taps: An Overview

The **Insert Tap** command inserts a port on a component after the component is placed in the model. Taps are added to components for venting, drainage, and for the connection of other component, such as instruments.

Related Topics

- *Insert a Tap*, page 140
- *Move a Tap*, page 141
- *Rotate a Tap*, page 141

Insert Tap Command



Places a tap on components in the model. Use this command when you need a tap for a drain, vent, or instrument connection. You can place taps on all piping components; elbows, tees, caps, valves, pipes, and so forth; except for mating parts.

Related Topics

- *Insert a Tap*, page 140
- *Inserting Taps: An Overview*, page 137
- *Move a Tap*, page 141
- *Rotate a Tap*, page 141

Insert Tap Ribbon

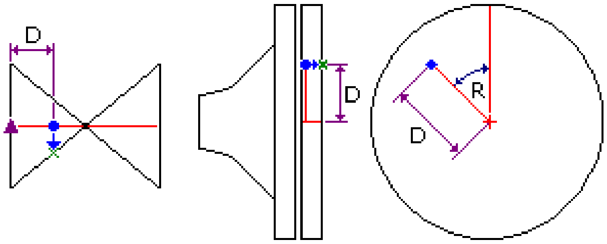
Sets options for adding a tap to a valve or other component.

Select Part - Select the component in which you want to place the tap. You can select any piping component; elbow, tee, cap, valve, pipe, and so forth; except a mating part.

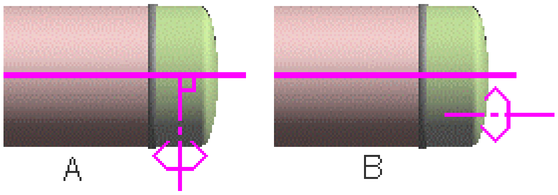
Option - Displays a list of available tap IDs that you can place in the selected component. The software generates this list based on the pipe specification of the component into which you are inserting the tap.

Finish - Places the tap in the component.

Distance - Specify the distance along the component between the tap and the specified reference position on the component. If **Orientation** is set to perpendicular, the distance is measured along the length of the component path as shown on the valve below. If **Orientation** is set to parallel, the distance is measured from the specified reference point and at the specified angle as shown on the blind flange below.



Orientation - Defines whether the tap is perpendicular (A) or parallel (B) to the selected component. This option is available only when you select an end feature (cap, blind flange, and so forth) or a turn feature (such as an elbow) as the component.



Reference position - Specifies the port on the component from which to measure the distance to the tap.

Rotate - Select to rotate the tap about the component interactively.

Angle - Type an angle at which you want the tap rotated about the component.

Related Topics

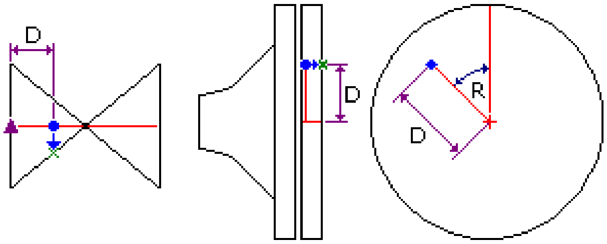
- *Insert a Tap*, page 140
- *Insert Tap Command*, page 138

Edit Tap Ribbon

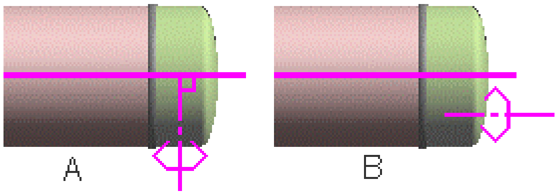
Changes options for taps placed in the model.

Option - Displays a list of available tap IDs that you can place in the selected component. The software generates this list based on the pipe specification of the component on which you inserted the tap.

Distance - Specify the distance along the component between the tap and the specified reference position on the component. If **Orientation** is set to perpendicular, the distance is measured along the length of the component path as shown on the valve below. If **Orientation** is set to parallel, the distance is measured from the specified reference point and at the specified angle as shown on the blind flange below.



Orientation - Defines whether the tap is perpendicular (A) or parallel (B) to the path passing through the port. This option is available only when you select an end feature (cap, blind flange, and so forth) or a turn feature (such as an elbow) as the component.



Reference position - Specifies the port on the component from which to measure the distance to the tap.

Rotate - Select to rotate the tap about the component interactively.

Angle - Type an angle at which you want the tap rotated about the component.

Related Topics

- *Insert Tap Command*, page 138
- *Move a Tap*, page 141
- *Rotate a Tap*, page 141

Pipe Tap Feature Properties Dialog Box

Displays tap feature properties for review and editing.

Related Topics

- *Configuration Tab*, page 74
- *Edit Tap Properties*, page 141

Insert a Tap

1. Click **Insert Tap** on the vertical toolbar.
2. Select the cap, elbow, tee, pipe, or other component in which to insert the tap. You cannot select a mating part.
3. Select the tap ID in the **Option** box.
4. If necessary, select a reference position other than the default. As the cursor moves over each entry in the list, the corresponding position highlights in the 3D model view.
5. Optionally, enter a distance relative to the reference position to place the tap.
6. Rotate the tap, if necessary.

7. Click **Finish**.

Related Topics
- *Insert Tap Ribbon*, page 138
- *Inserting Taps: An Overview*, page 137

Move a Tap

1. Click **Select** on the vertical toolbar.
2. Select **Taps** in the **Locate Filter**.
3. Select the tap to move.
4. Enter a new distance, orientation, or rotation value.

Related Topics
- *Edit Tap Ribbon*, page 139
- *Inserting Taps: An Overview*, page 137

Rotate a Tap

1. Click **Select** on the vertical toolbar.
2. Select **Taps** in the **Locate Filter**.
3. Select the tap to rotate.
4. Click **Rotate** and move the mouse to rotate the tap.

Note

- You also can enter a rotation angle in the **Angle** box.

Related Topics
- *Edit Tap Ribbon*, page 139
- *Move a Tap*, page 141

Edit Tap Properties

1. Click **Select** on the vertical toolbar.
2. Select **Taps** in the **Locate Filter** box.
3. Select the tap to edit.
4. Click **Edit > Properties**.

Related Topics
- *Pipe Tap Feature Properties Dialog Box*, page 140

Delete a Tap

1. Click **Select** on the vertical toolbar.
2. Select **Taps** in the **Locate Filter**.
3. Select the tap to delete.
4. Click **Delete** on the toolbar.

Related Topics

- *Edit Tap Ribbon*, page 139
- *Inserting Taps: An Overview*, page 137

Spooling: An Overview

Spools are collections of piping parts and the welds that hold them together. The software creates these spool assemblies by applying a set of rules that breaks the assemblies into pieces of a pipeline system. Each piece ends with a spool-breaking component or connection type. After a pipeline is spooled, the pipeline becomes the parent object of the spools. This process allows spools to be named after the pipeline and helps keeps track of spools that may be out of date based on the modification date of the parent object.

There are two types of spools. Standard spools are based solely upon the pipeline that contains the spools and a set of spooling rules. Penetration spools can include parts from one or more pipeline systems. Each of the penetration spools is welded to a common penetration plate.

The **Generate Spools** and **Create Penetration Spools** commands provide a dialog box on which you can select objects for spooling. Each spool must consist of connected objects for the spooling to occur without errors. After generating the spools, the software stores them in the Plant database and displays the spools on the **Assembly** tab in the **Workspace Explorer**.

After a spool is generated, the spool can be used to create an isometric drawing for fabrication. These drawings, typically referred to a spool sheets, are tied to the spool and are marked as being out-of-date if the spool is subsequently modified. The spool information can also be included on erection (construction) isometrics. To display the latest spool information on isometric drawings, set the option for showing spool piece identifiers for an isometric drawing style. Then, extract some test drawings using the updated isometric drawing style.

What Spools Include

The software includes in spools any piping component or piece of pipe that is assigned a fabrication type of *by fabricator*. Implied objects like bolts, nuts, and gaskets that are associated with these parts are not included in spools. You can specify options to include pipe support parts in the same spools as the components to which they are welded.

Additionally, the software provides the ability to create penetration spools, which are used in marine and offshore projects. In penetration spools, multiple piping spools are included in the assembly along with the common penetration plate to which they are attached.

Where Spools Break

When the software processes a pipeline system, spools are created by identifying components or end-preparations that represent intrinsic breaks in the line. The software always breaks spools at field welds, bolted connections, and compression fittings. You can specify whether spools break or not at unions.

In addition, you can place control points using the **Insert > Control Point** command to break spools at arbitrary locations that you define along the pipeline. You have the option to break only at control points, break at control points and intrinsic breaks, or to ignore control points.

If the software finds a gap when processing a pipeline system, the software breaks the spool and proceeds with processing the remainder of the pipeline.

How Spools Are Named

The software uses a default naming rule to give each spool a unique name. The default spool name contains a prefix based on the spool parent and a mark number. The mark number is use to ensure that the spool name is unique and to provide a sequencing of spools within a pipeline. You can modify the naming rules to match your company needs. For more information about naming rules, see the *SmartPlant 3D Reference Data Guide* available from the **Help > Printable Guides** command.

You also can interactively change the name of a spool by selecting it, displaying the **Properties** dialog box, and typing a different name.

Specifying Oversized Spools

The software checks at creation time the size of each spool against the maximum length, height, and width spool dimensions that you can specify. The default values in the reference data are 40 feet by 8 feet by 8 feet, and you can temporarily change these values if required. To change the dimension values permanently, edit options on the **PipeMfgSpoolRule** sheet in the **BulkLoadIsoKeys.xls** workbook and then bulk load the workbook into the Catalog.

The software marks spools that exceed the maximum size as oversized so that the designer can insert additional breaks as needed. This check helps to ensure the spools that require dipping in a tank for galvanization, de-scaling and pickling, and so forth will actually fit in the immersion tank. Also, spools can be marked oversized that are too large to fit in fabrication equipment such as bending machines or annealing ovens, or are too large for shipping. You can check for oversized spools in the following ways:

- View the **Assembly** tab of the **Workspace Explorer**. The icon for an oversized spool contains a black X: ✖.
- Click **File > Define Workspace** and view the **Assembly** tab on the **Filter Properties** dialog box. Again, the icon for an oversized spool contains a black X: ✖.
- View the **Properties** dialog box for the spool. The **SpoolOversizedFlag** property on the **Occurrence** tab is set to **True** if the spool is oversized.

Updating Spools

The software uses a detailed change management routine that prevents existing spools from being arbitrarily renumbered. When a pipeline is re-spooled, only the portion of the pipeline that has changed is impacted by the changes to its spools. New spools are only created as needed and existing spools do not lose their numbers (even if the result is that the numbers are now out of sequence within the pipeline). Existing spools are only deleted as required. This change management ensures that spool are not needlessly modified and that spool sheets are not marked as being out-of-date unless that is absolutely the case.

Related Topics

- *Create Penetration Spools*, page 153
- *Create Spools*, page 149

Generate Spools Command

Breaks pipelines into spools. You can view the spools on the **Assembly** tab in the **Workspace Explorer**.

Related Topics
- *Create Spools*, page 149
- *Spooling: An Overview*, page 143

Spool Generation Dialog Box

Allows you to specify the pipeline system or systems that you want to spool. You use this dialog box when generating standard spools. If you want to generate penetration spools, click the **Generate Penetration Spools** command.

OK - Processes the pipelines into spools using the options that you specify.

Cancel - Exits the dialog box.

Save Log - Saves the information on the **Log** tab to a text file.

Refresh Options from Catalog - Reloads the spool options that are defined in the Catalog to the **Options Tab**.

Related Topics
- *Create Spools*, page 149
- *Generate Spools Command*, page 146
- *Log Tab (Spool Generation Dialog Box)*, page 148
- *Options Tab (Penetration Spool Generation Dialog Box)*, page 152
- *Options Tab (Spool Generation Dialog Box)*, page 147
- *Spooling: An Overview*, page 143

Spooling Tab

Allows you to select the objects that you want to spool. For the **Generate Spools** command, you can choose pipeline systems. For the **Generate Penetration Spools** command, you can choose penetration plates.

Related Topics
- *Create Spools*, page 149
- *Generate Spools Command*, page 146
- *Spooling: An Overview*, page 143

Options Tab (Spool Generation Dialog Box)

Displays the spooling options to use. Each time you open the session file all spooling options default to the spool settings saved in the Catalog. You can change the spooling options if needed for a special situation, but you cannot save your changes to the Catalog or to the session file. The next time you open your session file, the spool options default to the Catalog settings.

If you need to change the spooling options in the Catalog, edit options on the **PipeMfgSpoolRule** sheet in the **BulkLoadIsoKeys.xls** workbook and then bulk load the workbook into the Catalog.

Name Rule - Specifies the naming rule for spools.

Break Spools at Union - Specifies that the software breaks the spools at unions.

Include Stub Ins with Header Spool - Specifies that a spool can include the stub-in pipe and all the parts of this stub-in branch until the first spool breaking component is encountered.

Include Items Welded to Pipe in Spools - Includes welded objects, such as pipe hanger or support parts, in the same spool as the components to which they are welded.

Delete Existing Spools - Changes the spool numbering only at modified or added piping. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers.

This setting guarantees that the software applies the same piece marks (spool names) to spools, even if spools are regenerated.

Ignore Boundaries of Object Being Spooled - When set to **False**, the software will not cross the boundary of the pipeline or block. When set to **True**, the software will cross the boundary of the pipeline or block for spool generation until an intrinsic spool break is found. This feature is intended for use when spooling by block.

Maximum Spool Length - Provides the maximum length of a spool for oversizing calculation purposes. You should enter **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Width - Provides the maximum width of a spool for oversizing calculation purposes. You should enter **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Height - Provides the maximum height of a spool for oversizing calculation purposes. You should enter **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Spool Break By Control Point - Specifies if spools should break at control points. You can place control points using the **Insert > Control Point** command. You must set the control point **Subtype** to **Spool Break** in order to use the control point with this option.

Select **Ignore Control Points** to ignore the control points during spooling. Select **Break at Control Points** to break spools at the normal intrinsic line breaks and at control points. Select **Break Only at Control Points** to break spools only at control points.

Related Topics
- *Create Spools*, page 149
- *Generate Spools Command*, page 146
- *Spooling: An Overview*, page 143

Log Tab (Spool Generation Dialog Box)

Displays a log of the last pipe spooling process. You can save the log to a text file by clicking **Save Log**. The log is divided into four parts:

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

To Do List - Lists spools that have objects in the **To Do List**. The spools cannot be created until you fix the objects on the **To Do List**. Click **View > To Do List** (or press CTRL+T) to view the **To Do List**.

Modified - Lists all the spools that where modified during the last spooling process.

New - Lists all the spools that were created during the last spooling process.

Tip

- You can select a spool, pipe, or pipe component in the list to highlight it in the model.

Related Topics
- *Create Spools*, page 149
- *Generate Spools Command*, page 146
- *Spooling: An Overview*, page 143

Spool Properties Dialog Box

Displays information about a piping spool. You can create spools using the **Generate Spools** command.

Occurrence Tab, page 76
Relationship Tab, page 78
Configuration Tab, page 74

Related Topics
- *Create Spools*, page 149

Create Spools

1. On the vertical toolbar, click **Generate Spools**.
2. Select the **Spooling** tab.
3. Select the pipelines to spool.
4. Select the **Options** tab.
5. Review the spooling options, edit if needed.
6. Click **OK**.
7. Select the **Log** tab.
8. Review the spooling log. Click **Save Log** to save the log to a text file if needed.

Notes

- After spooling, you can view the spools on the **Assembly** tab in the **Workspace Explorer**.
- To show spool identification numbers on an isometric drawing, set **Drawing > Content > Show Spool ID** to **True** in the **Isometric Style Options Browser**.

Related Topics
- *Generate Spools Command*, page 146
- *Spooling: An Overview*, page 143

Edit Spool Properties

1. On the **Assembly** tab in the **Workspace Explorer**, select a spool, either a standard spool or a penetration spool.

   Tip

   - To create standard piping spools, click **Generate Spools**. To create penetration spools, click **Create Penetration Spools**.
2. Click **Edit > Properties**.
3. Edit the properties as necessary.

Notes

- The **SpoolOversizedFlag** property on the **Occurrence** tab is set to **Yes** if the spool is oversized.

Related Topics

- *Spool Properties Dialog Box*, page 149

Create Penetration Spools Command

Creates penetration spools. This special type of spool consists of one or more spools penetrating a plate part. There is a tight connection between the spools and the plate part using a penetration plate created in the Hangers and Supports task. You can view the penetration spools on the **Assembly** tab in the **Workspace Explorer**.

Important

- Before you can generate penetration spools, you must have at least one penetration plate in the model. Switch to the Hangers and Supports task and use the **Place Support by Structure** command. For more information about placing penetration plates in the model, see the *Hangers and Supports User's Guide* available from the **Help > Printable Guides** command in the software.

Related Topics
- *Create Penetration Spools*, page 153
- *Spooling: An Overview*, page 143

Generate Penetration Spools Dialog Box

Allows you to specify the penetration plate that you want to spool. You use this dialog box when generating penetration spools.

Spooling Tab, page 146
Options Tab (Penetration Spool Generation Dialog Box), page 152

Related Topics
- *Create Penetration Spools Command*, page 151
- *Create Penetration Spools*, page 153
- *Spooling: An Overview*, page 143

Spooling Tab

Allows you to select the objects that you want to spool. For the **Generate Spools** command, you can choose pipeline systems. For the **Generate Penetration Spools** command, you can choose penetration plates.

Related Topics
- *Create Spools*, page 149
- *Generate Spools Command*, page 146
- *Spooling: An Overview*, page 143

Options Tab (Penetration Spool Generation Dialog Box)

Displays the spooling options to use. Each time you open the session file all spooling options default to the spool settings saved in the Catalog. You can change the spooling options if needed for a special situation, but you cannot save your changes to the Catalog or to the session file. The next time you open your session file, the spool options default to the Catalog settings.

If you need to change the spooling options in the Catalog, edit options on the **PipeMfgSpoolRule** sheet in the **BulkLoadIsoKeys.xls** workbook and then bulk load the workbook into the Catalog.

Name Rule - Specifies the naming rule for spools.

Break Spools at Union - Specifies that the software breaks the spools at unions.

Include Stub Ins with Header Spool - Specifies that a spool can include the stub-in pipe and all the parts of this stub-in branch until the first spool breaking component is encountered.

Include Items Welded to Pipe in Spools - Includes welded objects, such as pipe hanger or support parts, in the same spool as the components to which they are welded.

Do Not Recreate Existing Spools - Changes the spool numbering only at modified or added piping. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers.

This setting guarantees that the software applies the same piece marks (spool names) to spools, even if spools are regenerated.

Ignore Boundaries of Object Being Spooled - When set to **False**, the software will not cross the boundary of the pipeline or block. When set to **True**, the software will cross the boundary of the pipeline or block for spool generation until an intrinsic spool break is found. This feature is intended for use when spooling by block.

Maximum Spool Length - Provides the maximum length of a spool for oversizing calculation purposes. You should enter **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Width - Provides the maximum width of a spool for oversizing calculation purposes. You should enter **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Height - Provides the maximum height of a spool for oversizing calculation purposes. You should enter **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Spool Break By Control Point - Specifies if spools should break at control points. You can place control points using the **Insert > Control Point** command. You must set the control point **Subtype** to **Spool Break** in order to use the control point with this option.

Select **Ignore Control Points** to ignore the control points during spooling. Select **Break at Control Points** to break spools at the normal intrinsic line breaks and at control points. Select **Break Only at Control Points** to break spools only at control points.

Related Topics

- *Create Spools*, page 149
- *Generate Spools Command*, page 146
- *Spooling: An Overview*, page 143

Penetration Spool Properties Dialog Box

Displays information about a penetration spool. You can create penetration spools using the **Create Penetration Spools** command.

Occurrence Tab, page 76
Relationship Tab, page 78
Configuration Tab, page 74

Related Topics

- *Create Penetration Spools*, page 153

Create Penetration Spools

Important

- Before you can generate penetration spools, you must have at least one penetration plate in the model. Switch to the Hangers and Supports task and use the **Place Support by Structure** command. For more information about placing penetration plates in the model, see the *Hangers and Supports User's Guide* available from the **Help > Printable Guides** command in the software.

1. On the vertical toolbar, click **Create Penetration Spools** .
2. Select the **Spooling** tab.
3. Select a penetration plate for spooling.
4. Click **OK**.

Notes

- You must have one penetration plate only per pipeline system before penetration spooling can take place. If you have more than one plate connected to the pipe, the software displays an error message.
- After spooling, you can view the spools on the **Assembly** tab in the **Workspace Explorer**.
- To show spool identification numbers on an isometric drawing, set **Drawing > Content > Show Spool ID** to **True** in the **Isometric Style Options Browser**.

Related Topics

- *Create Penetration Spools Command*, page 151
- *Spooling: An Overview*, page 143

Sequence Objects Command

Renames objects in the selected pipeline, pipe run, or spool such that the names are in order. You can select to sequence the objects based on flow direction or topology.

Related Topics

- *Sequence Objects Ribbon*, page 155
- *Sequence Weld Names in a Pipe Run*, page 158
- *Sequence Weld Names in a Pipeline*, page 158
- *Sequence Weld Names in a Spool*, page 158

Sequence Objects Ribbon

Sets options for sequencing names in a spool, pipeline, or pipe run.

Grouping/Sequencing Object Type - Select by what you want to sequence. You can select pipeline, pipe run, or spool. This option defines the collection of target objects to be sequenced, and defines the boundaries of the sequencing.

Cancel - Reset the ribbon options.

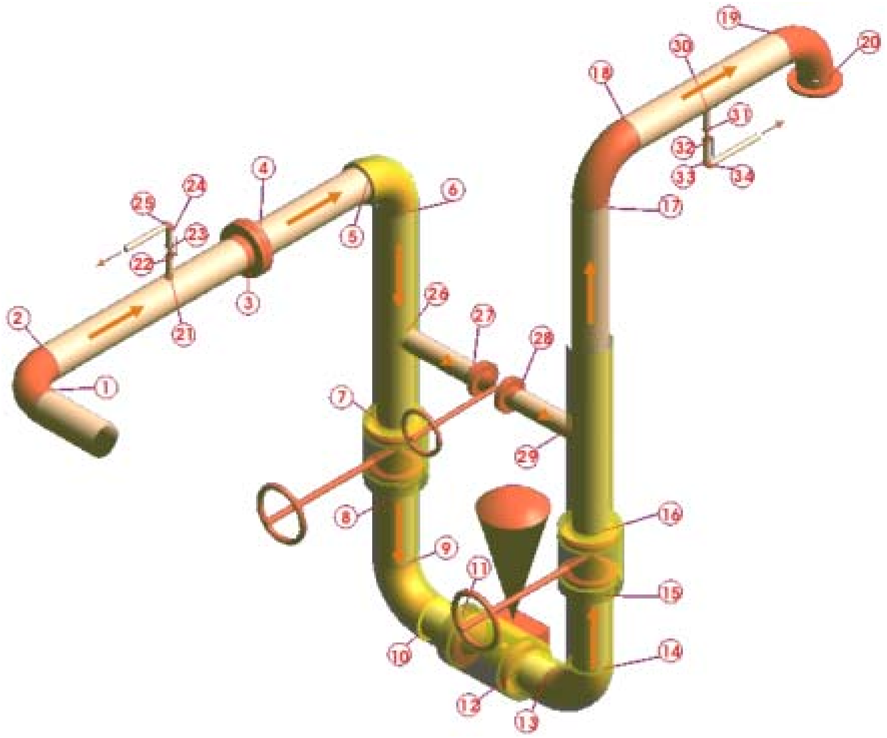
Finish - Orders the object names using the options that you have set.

Target Object Type - Select what you want to re-sequence (rename) in the selected grouping.

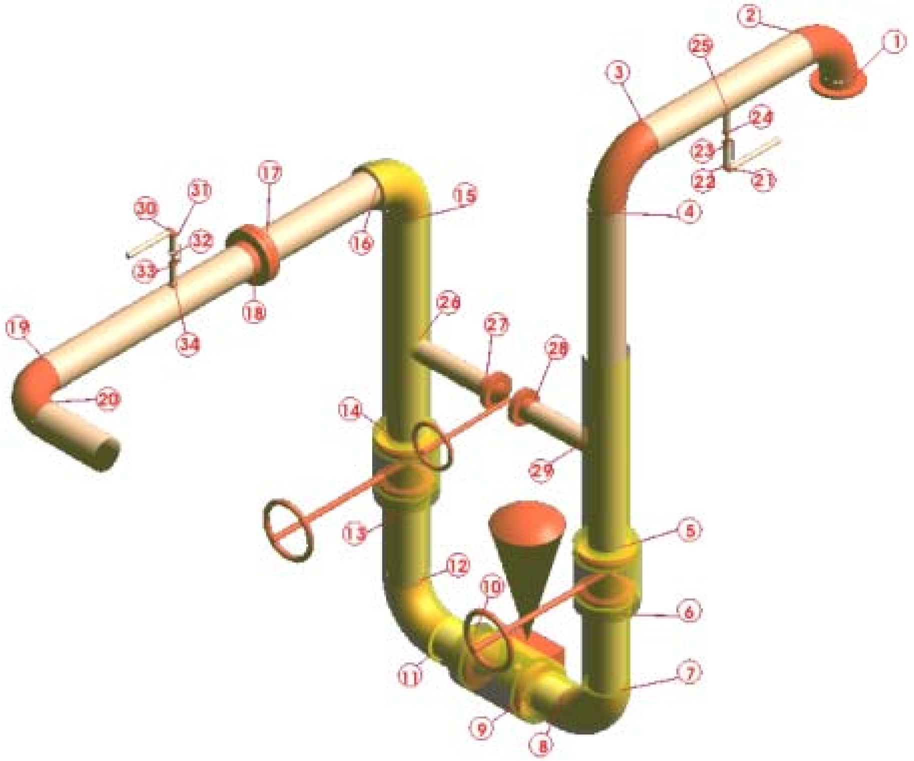
Name Rule - Select the naming rule to use to rename the target objects.

Sequencing Type - Defines the logical order to use to define the sequence number part of the name.

- Flow Direction - Begin sequencing at the furthermost upstream point on the line and proceed along the line with the flow direction. Subsequent braches are then processed according to their flow direction. The following example shows weld sequencing when using flow direction.



- Topology - Start from the point on the primary header of the spool, pipeline, or pipe run with the highest coordinate value and work back along the target objects until all have been processed. Subsequent branches are then processed from the furthermost point into the branch connection. The following example shows weld sequencing when using topology.



Revision Control - Specifies what to do with existing sequence number when you run the command again. Select **Retain existing numbers** to keep the sequence number on any objects that have one, but create a new sequence number for objects that do not have a sequence number. Select **Generate new numbers** to discard all sequence numbers for the selected object and generate new numbers for everything.

Related Topics

- *Sequence Objects Command*, page 155
- *Sequence Weld Names in a Pipe Run*, page 158
- *Sequence Weld Names in a Pipeline*, page 158
- *Sequence Weld Names in a Spool*, page 158

Sequence Weld Names in a Pipeline

1. Click **Sequence Objects** on the vertical toolbar.
2. In the **Grouping/Sequencing Object Type** box, select **Pipeline**.
3. Select the pipeline in the Workspace Explorer or in a graphics view.
4. In the **Target Object Type** box, select **Pipe Weld**.
5. In the **Sequencing Type** box, select how to sequence the welds.
6. Click **Finish**.

Related Topics
- *Sequence Objects Command*, page 155

Sequence Weld Names in a Pipe Run

1. Click **Sequence Objects** on the vertical toolbar.
2. In the **Grouping/Sequencing Object Type** box, select **Pipe Run**.
3. Select the pipe run in the Workspace Explorer or in a graphics view.
4. In the **Target Object Type** box, select **Pipe Weld**.
5. In the **Sequencing Type** box, select how to sequence the welds.
6. Click **Finish**.

Related Topics
- *Sequence Objects Command*, page 155

Sequence Weld Names in a Spool

1. Click **Sequence Objects** on the vertical toolbar.
2. In the **Grouping/Sequencing Object Type** box, select **Spool**.
3. Select the spool in the Workspace Explorer.
4. In the **Target Object Type** box, select **Pipe Weld**.
5. In the **Sequencing Type** box, select how to sequence the welds.
6. Click **Finish**.

Related Topics
- *Sequence Objects Command*, page 155

Piping Glossary

A

actuator
A device used to operate a valve using electric, pneumatic, or hydraulic pressure.

angle
The circular measurement taken from the intersection of two pipes at a turn or branch.

attribute
A single type of non-graphics information that is stored about an object such as diameter or end preparation.

B

bill of materials
An indented and exploded list of parts in a feature.

C

commodity code
A user-defined code that provides an index to parts in a catalog.

commodity item
A standard component found in a manufacturer's catalog (an off-the-shelf component).

component
A piece of a pipe feature or pipe run.

critical flow
The state of flow for a given discharge at which the specific energy is minimum.

D

d/D
The maximum ratio of flow depth over pipe diameter.

E

exit elevation
The lowest downstream elevation point on the internal diameter of a pipe.

F

face-to-face
The overall length of a component from the inlet face to the outlet face.

feature
A logical collections of parts driven by the piping specification. There are four basic features: straight, turn, branch, and inline component.

flat face
A flange surface on which the gasket sealing area is the entire surface from the inside face diameter to the outside face diameter.

flow rate
The quantity of fluid flowing per unit of time.

flow time
The time is takes for the flow, from the head of the piped system, to reach a downstream point.

full penetration weld
A type of weld where the weld material extends through the complete thickness of the components being joined.

G

gasket
A component used to seal a joint between two other components.

H

handwheel
A wheel-shaped operator intended to be turned by hand to operate the valve stem or operator shaft to which it is attached.

header
The portion of a pipeline topology associated with the primary fluid flow.

head loss
The loss of pressure due to friction or shape of a structure.

hydrostatic test
A pressure test in which water is used to detect leaks in a component.

I

implied piping component
Piping components which are created in the database as part of a feature, but which are not represented graphically.

inline
A term used to refer to those piping components that can be inserted in a pipe feature.

invert elevation
The lowest point on the internal diameter of the pipe.

isometric
Relating to or being a drafting system characterized by three equal axes at right angles; a view in which the horizontal lines of an element are drawn at an angle to the horizontal and all verticals are projected at an angle from the base.

isometric drawing
A line drawing, always shown in an isometric perspective, that is used for fabricating and erecting piping systems. An iso usually shows a complete line from one piece of equipment to another and provides all information necessary for fabrication and erection of piping.

isometric drawing style
A set of options that control the drawing output, including format and content. Each style has a unique set of options stored in reference data. You can use the Isometric Style Configuration Manager to edit the options.

L

lever
A handle type operator.

M

material list
An option category that controls the format and content of the bill of materials.

N

NPD (Nominal Piping Diameter)
The diameter of a pipe.

normal vector
In drawing extraction commands, the normal vector is the view orientation of a drawing view set.

nozzle
A piping connection point to a piece of equipment.

nozzle standout
The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

O

object
Information that can be linked or embedded into an OLE-compliant product. A group of elements that represent a real-world object. An object can also be made up of a single element.

occurrence [of part or equipment]
Instantiation of a part or equipment in the plant that refers to the part library; an

instance of a specific object. The design can be built several times and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property
A characteristic that applies to an individual object in the model. Occurrence properties are designated with oa: in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the Properties dialog boxes. Depending on the object, some occurrence properties are read-only.

P

parts
The physical components that comprise a feature and are generally selected by the software. For example, the flanges, gaskets, and the gate valve itself are examples of the parts comprising the gate valve feature.

part number
Unique identifier of a part.

part override
An option used to place a component not defined in the pipe specification.

peak flow
The maximum flow rate of water through a specific size pipe.

PinPoint
Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

pipe
Piping part that is hollow and approximately cylindrical, may have a constant cross-section along its length, and conveys a working media. (AP217)

pipeline
A set of graphically connected pipe runs including all branches. In Drawings and Reports, the Pipeline List displays pipeline systems when you select the Final isometric drawing style.

pipe run
A pipe run is a connected series of pipe components that have the same nominal piping diameter (NPD) and flow direction, and are governed by the same pipe specification.

piping
Pipe, tubing and support structure used to move liquids into the plant.

piping system
Type of distribution system which allocates and controls the dispersion of a working media (fluid or gas) among functional devices. The piping system may be comprised of other piping systems, parts, devices, pipelines, and/or pipe runs.

port
A connection point to a pipe or a component such as a valve.

R

raised face
The raised area of a flange face which is the gasket sealing surface between two mating flanges.

revision cloud
A set or arcs used to enclose changes that have occurred since the last revision.

S

schedule
A system for indicating the wall thickness of a pipe. The higher the schedule number, the thicker the wall for a certain pipe size.

seal weld
A weld to seal or prevent leakage from a joint but does not contribute to the strength of the joint.

specific energy
Depth of flow plus velocity head.

split
A feature that represents a break and a joint in the piping, for example, butt-weld, coupling, flange set, union, and so forth.

stem
A rod that transmits the motion from the operator to the closure element of the valve.

stud
A bolt, threaded on both ends, used to connect components.

T

trim
A common term that refers to a valve's working parts and to their materials.

turn
A feature that represents an angular change in direction of a pipe run, for example, an elbow or an angle valve.

V

valve
A component used to control the flow of fluid contained in a pipeline.

W

working plane
The available plane of movement for endpoint selection. The working area is defined as a 2-D plane at the height of the active end feature above the baseline of the model.

Index