

**Pro/ENGINEER<sup>®</sup>**

**Wildfire<sup>™</sup> 2.0**

**Pro/PIPING<sup>™</sup> (Non Spec-Driven)  
Help Topic Collection**

**Parametric Technology Corporation**

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**Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA**

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## **Pro/PIPING (Non Specification-Driven)**

### **Using Pro/PIPING (Non Specification-Driven)**

#### **About Pro/PIPING and Piping Systems (Non Spec-Driven)**

Pro/PIPING is an optional Pro/ENGINEER module. Accessed through Assembly mode, Pro/PIPING enables you to generate 3-D piping systems in Pro/ENGINEER assemblies. You can create piping systems in either the specification-driven or non specification-driven piping design modes. Creating a piping system involves establishing pipeline parameters, routing lines, and inserting fittings.

A piping system consists of pipelines, fittings, and equipment.

- A pipeline is composed of pipe segments and fittings. A segment of line may consist of one or more consecutive sections, that is, straight pieces of pipe with bends or miter cuts or a section of flexible tube, between breaks. The system creates a break whenever you create a fitted corner or insert an internal fitting in the pipeline.
- A fitting is a part or assembly that you use to connect lines to other lines, fittings, and equipment, or to perform specific functions in the piping system (for example, flow valves).
- Equipment in the working assembly (for example, a coil or a tank) is represented by assembly components with predefined entry ports.

#### **Tailoring the Pro/PIPING Working Environment**

You can tailor your Pro/PIPING work environment to your specific needs using various tools:

- Using simplified representations, you can create a special version of a piping assembly that includes only those components that you need to route a pipeline.

Using simplified representations can improve regeneration, retrieval, and display time, which allows you to work more efficiently. Simplified representations simplify a view by controlling which members of an assembly the system retrieves into a session and shows.

The By Rule functionality enables you to exclude piping solids and fittings from a simplified representation by assigning a parameter and then excluding all components that have a particular value for that parameter.

Although you cannot exclude assembly features, you can easily place all piping features on a layer by type.

- Using top-down design tools, you can set up a well-structured design using advanced component creation tools containing skeleton models and copied geometric and datum references.

- Using reference control and investigation tools, you can view and manage the complex web of dependencies (references) that evolve as you make features in a design.
- Using viewing and environment control settings, you can set different visualization (display) modes for components in an assembly. You can assign wireframe, hidden line, no hidden line, shaded, or blanked display modes to components.
- The Piping System Tree provides an intuitive pipeline-level visual representation of a Pro/PIPING piping project. You can use show/blank and centerline/solid display modes, and also highlight pipelines. These display modes are accessed from this fully customizable piping interface.

### **About Pro/PIPING Piping Design Modes (Non Spec-Driven)**

Pro/PIPING allows you to create complex piping systems based on the piping design mode you choose. You set the piping design mode by setting the `piping_design_method` configuration option. The following design modes are available:

- **Non Specification-Driven**—Creates piping systems using manual tasks. All line stocks must be manually created and associated to a routed pipeline. All fittings must also be manually inserted. The Non-Specification-Driven piping design mode does not use project-specific data during pipe modeling. Furthermore, this design mode does not support design compatibility checks such as size mismatch detection, end type mismatch, and so on.
- **Specification-Driven**—Creates piping systems based on the piping design method practiced by the Plant, Shipbuilding, and Aerospace design industries. Pro/PIPING creates specification-driven piping systems using piping specifications and automated modeling tasks. These tasks use archived piping data and project-specific data from the Specification Database. Pro/PIPING creates all line stocks on-the-fly and automatically associates them to the appropriate pipelines. All fittings are selected automatically during fitting insertion. The Specification-Driven piping design mode supports intelligent design rule checking during modeling.
- **User-Driven**—Enables you to switch between Non Specification-Driven and Specification-Driven piping modes and convert your piping assemblies at any time in the design process. Select or clear the **Spec Driven** check box on the **PIPING** menu to control the project piping design mode and convert piping assemblies to the selected mode.

### **To Access the Non Specification-Driven Design Mode**

1. Configure Pro/PIPING for the Non Specification-Driven or User-Driven piping design modes.
2. Create or open a piping assembly.

3. Click **Applications > Piping**. The **PIPING** menu appears. If you are using the User-Driven mode, clear the **Spec Driven** check box on the **PIPING** menu.

You use the **PIPING** menu to design your non specification-driven piping system.

### About the Pro/PIPING User Interface

The Pro/PIPING user interface has features that increase usability, decrease mouse picks, and eliminate the need to use Pro/TABLE.

Pro/PIPING includes many dialog boxes and a Model Tree. For example, the **Report Pipeline** dialog box controls several important Pro/PIPING functions. In many of the dialog boxes, there is a **select the active assembly** option that you use first to orient yourself correctly before you initiate an action on the assembly. You can also choose an assembly with the Pro/ENGINEER menus or the Model Tree.

The Model Tree lets you bypass traditional menu structures by right-clicking and accessing the shortcut menu that contains multiple Pro/PIPING functions.

The **COMPONENT** menu is accessible while in Piping mode.

The **View** tab in the Model Tree allows you to change views.

### Using the PIPING Menu (Non Spec-Driven)

The **PIPING** menu contains the following Pro/PIPING commands:

- **Active Asm**—Opens the **SELECT** dialog box that allows you to change the current active assembly. Select the active assembly from either the Model Tree, Piping System Tree, or the working window. The name of the currently active assembly appears in the graphics window.
- **Component**—Opens the **COMPONENT** menu that allows you to manipulate assembly components.

For more information on components, refer to the topics about basic assemblies in the Using Foundation Modules portion of Pro/ENGINEER help.

- **Pipeline**—Opens the **OPER PIPE LINE** menu on which you can create and immediately route a new pipeline or delete, rename, suppress, or resume existing pipelines.
- **Route**—Opens the **SELECT** menu on which you can add to or edit the path of an existing pipeline assembly.
- **Fitting**—Opens the **FITTING** menu on which you can insert pipe fittings into a current active assembly.
- **Modify Pipe**—Opens the **MODIFY PIPELINE** dialog box in which you can modify several piping options in the active assembly.
- **Modify Dim**—Opens the **MODIFY** menu on which you can modify the active assembly component dimensions.
- **Regenerate**—Opens the **PRT TO REGEN** menu on which you can update modified objects and assembly dimensions.

- **Fabrication**—Opens the **FABRICATION** menu on which you can cut pipes, number pipe segments, and create pipe solids and insulation.
- **Drawing**—Opens the **Generate Isometric** dialog box in which you can use the Pro/PIPING drawing tool.
- **Set Up**—Opens the **PIPE SETUP** menu on which you can set up how Pro/PIPING creates piping systems.
- **Info**—Opens the **Report Pipeline** dialog box in which you can retrieve piping system information and save your analyses for future retrieval.

## Piping System Tree

### About the Piping System Tree (Non Spec-Driven)

The Pro/PIPING Piping System Tree allows you to control your pipeline display. This tree-based interface organizes all pipelines in the active assembly and its subassemblies by piping system (categories) and orders pipelines according to their names. The ability to include all subassemblies streamlines the pipeline display process. This is especially important if you create pipelines based on the "one pipeline per assembly" modeling method.

The Piping System Tree does the following:

- Displays all pipelines in the top level assembly and the subassemblies below it.
- Organizes pipelines in a collapsible and expandable hierarchy according to the value assigned to the `piping_system_tree_format` configuration option. The pipelines are grouped according to the `MNEMONIC` configuration option by default. This organization provides a clear and intuitive visual representation of the piping project that allows immediate piping system recognition and pipeline selection. For example, you can group and display all water pipes or all specification 12A pipelines.
- Allows for efficient pipeline selection. You can quickly select pipelines and perform the required tasks. Note that unlike the Pro/ENGINEER Model Tree, the pipelines and assembly components cannot be sequentially reordered using the Piping System Tree.

### Viewing the Piping System Tree



on the top menu bar allows selection between the standard Model Tree and the Piping System Tree with pipeline view.

### Pipeline Selection and Display Features

The Piping System Tree provides the following dynamic pipeline selection and display options:

- **Pipeline**—Displays or hides a pipeline.

- **Label**—Creates, deletes, displays, or hides a pipeline label.
- **Solid**—Creates, deletes, displays, or hides a pipe solid.
- **Insulation**—Creates, deletes, displays, or hides pipe insulation.
- **Flow**—Displays and allows the reversal of the flow direction of the pipeline or pipe segment nodes.

### Customize the Pro/PIPING Piping System Tree

The Piping System Tree is fully customizable. You can reorganize pipelines in multiple ways by setting the `piping_system_tree_format` configuration option.

### To Access the Piping System Tree (Non Spec-Driven)

1. Open or create a Pro/PIPING assembly.
2. Click **Applications** > **Piping**. The Piping System Tree opens. From the Piping System Tree, you can select pipelines and display modes.

If the Piping System Tree does not open, do the following:

- Click . The Piping System Tree opens.
- Click **Show** > **Expand All** to expand the Piping System Tree and display all its components.

#### Note:

If you expand a pipeline with no components, a warning message is displayed.

The sequence of components is based on the direction of flow of the network.

### To Select Pipelines (Non Spec-Driven)

1. Click . The Piping System Tree opens.
2. Select one of the following:
  - **PIPING SYSTEM TREE**—Selects all pipelines in the active assembly including those in the subassemblies.
  - **Piping System (Category)**—Selects all pipelines in the piping system. For example, if a piping system is configured for `MNEMONIC` and you select `WATER`, all water pipelines in the active assembly and also from the top assembly are selected.
  - **Pipeline**—Selects one pipeline.

You can also select individual pipelines in the Pro/ENGINEER graphics window.

### To Display the Pipeline Component View

1. Click . The Piping System Tree opens.
2. Click the plus sign (+) adjacent to the branch node or double-click the branch node. The branch node expands to show the pipeline components.
3. Click a pipeline to select it in the expanded Piping System Tree. The pipeline is highlighted in the Pro/ENGINEER graphics window.

### To Display or Blank Pipelines (Non Spec-Driven)

1. Click . The Piping System Tree opens.
2. Select one or more pipelines from the Piping System Tree and right-click. A shortcut menu appears.
3. Click **Pipeline > Show** from the menu to display the pipeline in the graphics window or click **Pipeline > Hide** to hide the pipeline in the graphics window.

### To Create or Delete Pipeline Labels from the Piping System Tree

1. Click . The Piping System Tree opens.
2. Select one or more pipelines from the Piping System Tree and right-click. A shortcut menu appears.
3. Click **Label > Create** to create a label for the pipeline or click **Label > Delete** to delete the labels of the selected pipelines.

**Note:** **Delete** is available only if a label is created for a pipeline.

### To Display or Hide Pipeline Labels

1. Click . The Piping System Tree opens.
2. Select one or more pipelines from the Piping System Tree and right-click. A shortcut menu appears.
3. Click **Label > Show** to display all the labels of the selected pipeline or click **Label > Hide** to hide the labels of the selected pipelines.

**Note:** **Show** and **Hide** are available for a pipeline only if a label is created for the pipeline.

### To Create or Delete Pipe Solids from the Piping System Tree

1. Click . The Piping System Tree opens.

2. Select one or more pipelines from the Piping System Tree and right-click. A shortcut menu appears.
3. Click **Solid > Create** on the shortcut menu to create solids for the entire pipeline or pipe segments or click **Solid > Delete** to delete the solids for the entire pipeline or pipe segments.

**Note: Delete** is available only if the selected pipe segment nodes have solids.

### To Display or Hide Pipe Solids

1. Click . The Piping System Tree opens.
2. Select one or more pipelines from the Piping System Tree and right-click. A shortcut menu appears.
3. Click **Solid > Show** on the shortcut menu to show solids for the entire pipeline or pipe segments or click **Solid > Hide** to hide solids for the entire pipeline or pipe segments.

**Note: Hide** is available only if the selected pipe segment nodes have unsuppressed solids.

### To Create or Delete Pipe Insulation Quilts

1. Click . The Piping System Tree opens.
2. Select one or more pipelines from the Piping System Tree and right-click. A shortcut menu appears.
3. Click **Insulation > Create** to create pipe insulation quilts for the entire pipeline.

To delete the pipe insulation quilts for the entire pipeline, click **Insulation > Delete**.

**Note:** The **Insulation** option is available in the Piping System Tree and in the graphics window only if a pipe solid has been created.

### To Display or Hide Pipe Insulation Quilts

1. Click . The Piping System Tree opens.
2. Select one or more pipelines from the Piping System Tree and right click. A shortcut menu appears.
3. Click **Insulation > Show** in the shortcut menu to show pipe insulation quilts for the entire pipeline.

To hide pipe insulation quilts for the entire pipeline, click **Insulation > Hide**.

**Note:** The **Insulation** option is available only if the selected pipe segment nodes have unsuppressed pipe insulation quilts.

## To Display Centerlines or Solids (Non Spec-Driven)

1. Click . The Piping System Tree opens.
2. Select one or more pipelines and right-click. A shortcut menu appears.
3. Click **Solid > Show** to display pipe centerlines for the selections. The selected pipelines appear.

## Piping System Tree Centerline and Solid Display Modes

The Piping System Tree centerline and solid display modes differ from the centerline and solid display capabilities in **Set Display** and the mode set by the `pipe_solid_centerline` configuration option in the following ways:

- The Piping System Tree centerline and solid display modes allow you to select one or more pipelines in the active assembly, including subassemblies. You can display the selected pipeline with a pipe solid or centerline. If you are creating pipelines based on the "one pipeline per assembly" modeling method, it is recommended that you use this centerline and solid display mode.
- The **Set Display** centerline and solid display modes (**PIPING > Set Up > Set Display**) enables you to select only pipeline in the active assembly, but not in the subassemblies. You can display the selected pipeline with a pipe solid or centerline.
- The mode set by the `pipe_solid_centerline` configuration option indicates the center of the pipe solid geometry with a line. This is a global setting for all pipelines.

## To Highlight Pipelines (Non Spec-Driven)

1. Click **Show > Highlight Model**.
2. Select pipelines from Piping System Tree. The selected pipelines are highlighted in the Piping System Tree and in the Pro/ENGINEER graphics window.
3. Repeat this process to turn off the **Highlight Model** display mode.

## About Customizing the Piping System Tree (Non Spec-Driven)

Customizing the Piping System Tree gives you the flexibility to display and organize your piping projects in the following ways:

- **Reorganize Pipelines**—Reorganize pipelines in multiple ways by setting the `piping_system_tree_format` configuration option to modify categories.
- **Modify the Column Display**—Display informational columns and select items to display in these Piping System Tree columns.

## To Reorganize Pipelines (Non Spec-Driven)

1. Click **Tools > Options**. The **Options** dialog box opens.

2. In the **Option** box, type the `pipng_system_tree_format` configuration option.
3. In the **Value** box, type one or any combination of the following values:
  - SIZE
  - SPECIFICATION
  - MNEMONIC
  - NUMBER
  - INSULATION
  - Any ASCII characters (default is `MNEMONIC`).

Delimiter characters must be enclosed by commas (,), omitting any white space. For example, `MNEMONIC, -, SPECIFICATION` groups all pipelines according to fluid carried and piping specification.

### To Modify the Column Display (Non Spec-Driven)

1. Click . The Piping System Tree opens.
2. Click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.
3. In the **Model Tree Columns** dialog box, under **Not Displayed**, select the items you want to display and click . The selected items now appear under **Displayed**.
4. Click **OK**. The selected items and their respective values are displayed in the columns of the Piping System Tree.

### Configuring Pro/PIPING (Non Specification-Driven)

#### About Configuring Pro/PIPING (Non Spec-Driven)

You can customize the way the Non Specification-Driven piping design mode operates by entering `config.pro` configuration file options and their values in the **Options** dialog box (**Tools > Options**). For example, you can modify piping design modes, pipeline formats, file names and directory locations, and configure the Pro/PIPING ISOGEN interface.

Pro/PIPING provides a list of configuration options arranged in alphabetical order. Each option contains the following information:

- Configuration option name.
- Default and available variables or values. All default values are in italics.
- Brief description and notes describing the configuration option.

## ISOGEN Configuration Options

Pro/PIPING allows you to create isometric pipeline drawings by providing an isometric drawing generation (ISOGEN) interface. Using the Generate Isometric dialog box (**PIPING > Drawing**), you can create ISOGEN .pcf files for use with an ISOGEN drawing tool. The following configuration options must be set in the Pro/PIPING config.pro file to use the ISOGEN interface:

- isogen\_mandatory\_attr\_file
- isogen\_symbol\_map\_file
- isogen\_output\_files\_dir
- isogen\_nominal\_size\_map\_file
- isogen\_pcf\_filename\_format
- isogen\_endtype\_map\_file

### Note:

- After you set the configuration options, all settings take effect immediately in the current Pro/ENGINEER session.
- The Symbol Map, Endtype Map, and Nominal Size ISOGEN Map files are required to map Pro/PIPING component information to ISOGEN component information.

## To Set Pro/PIPING Configuration Options (Non Spec-Driven)

1. Locate the configuration option you want to set using the configuration topics.
2. Click **Tools > Options**. The **Options** dialog box opens.
3. Select the configuration option from the list or in the **Option** field, type the configuration option name.
4. In the **Value** field, enter the value. You can use the **Value** list to see which values are available. If the value requires an integer, type it.
5. Click **Add / Change**. The configuration option and its value appear in the list. A green status icon confirms the change.
6. When you finish configuring Pro/PIPING, click **Apply** or **OK**.

**Note:** It is recommended that you set the Pro/PIPING configuration options before starting a new piping project.

## isogen\_endtype\_map\_file (Non Spec-Driven)

<proe\_load\_directory>/text/piping\_data/isodata/isogen\_endtype\_map.ptd,  
<absolute directory path and valid file name>

Specify a path and name for the file that maps Pro/PIPING endtypes with ISOGEN endtypes.

**isogen\_mandatory\_attr\_file (Non Spec-Driven)***isogen.flis*, <valid file name>

Specify a name for the file that contains ISOGEN mandatory attribute files.

**isogen\_nominal\_size\_map\_file (Non Spec-Driven)**<*proe\_load\_directory*>/text/piping\_data/isodata/isogen\_nominal\_size\_map.ptd, <absolute directory path and valid file name>

Specify a path and name for the file that maps Pro/PIPING sizes with ISOGEN nominal sizes in millimeters and inches.

**isogen\_output\_files\_dir (Non Spec-Driven)**<*proe\_run\_directory*>, <absolute directory path>

Specify a directory path for archiving the ISOGEN output files.

**isogen\_pcf\_filename\_format (Non Spec-Driven)***MNEMONIC-SPECIFICATION-NUMBER*, <a combination of the following keywords: SIZE, SPECIFICATION, MNEMONIC, NUMBER, INSULATION, and any valid file name characters>. Delimiter characters must be enclosed by commas (,) omitting any white space. For example *MNEMONIC,-,SIZE*.

Specify a name format for ISOGEN (.pcf) files. Pro/PIPING creates ISOGEN file names from the values assigned to the keywords. See the following example:

<b>ISOGEN File Name Format</b>	<b>Assigned Values</b>	<b>ISOGEN (.pcf) File Name</b>
MNEMONIC-NUMBER-INSULATION	WATER,1001,IS	WATER-1001-IS.PCF
SHIP1-MNEMONIC-NUMBER-INSULATION	WATER,1001,IS	SHIP1-WATER-1001-IS.PCF

**isogen\_symbol\_map\_file (Non Spec-Driven)**<*proe\_load\_directory*>/text/piping\_data/isodata/isogen\_symbol\_map.ptd, <absolute directory path and valid file name>

Specify a path and name for the file that maps Pro/PIPING fitting library part names with ISOGEN component names and symbol keys (SKEY).

**pipe\_3D\_bend\_theor\_int\_pts (Non Spec-Driven)***yes*, *no*

Set the theoretical intersections and bends in three-dimensional piping models.

- *yes*—Pro/PIPING displays theoretical intersections and bends in three-dimensional models.
- *no*—Pro/PIPING does not display theoretical intersections and bends in three-dimensional models.

### **pipe\_extend\_dim\_scheme (Non Spec-Driven)**

*cartesian, cylindrical, spherical*

Specify a dimension scheme for piping extend segment.

### **pipe\_pre\_22\_bendinfo\_conv**

*<user-defined directory>, <absolute directory path>*

Convert bend report information generated prior to Pro/ENGINEER 200i-2 to current saved analysis format.

### **pipe\_solid\_centerline (Non Spec-Driven)**

*yes, no*

Set the pipe solid centerline display mode. Pro/PIPING displays a pipe solid centerline in the graphics window to indicate the center of the pipe solid. After you set this option, the display changes immediately in the current session of Pro/ENGINEER

- *yes*—Pro/PIPING displays pipe solid centerlines.
- *no*—Pro/PIPING does not display pipe solid centerlines.

**Note:** The `pipe_solid_centerline` mode differs from the centerline or solid display capabilities in both the **Set Display** and the Piping System Tree in the following ways:

- The mode set by the `pipe_solid_centerline` configuration option indicates the center of the pipe solid geometry with a line. This is a global setting for all pipelines.
- The **Set Display** centerline or solid display mode (**PIPING > Set Up > Set Display**) enables you to select pipelines in both the active assembly as well as in the subassemblies. You can display the selected pipeline with a pipe solid or centerline.
- The Piping System Tree centerline or solid display mode allows you to select one or more pipelines in the active assembly, including subassemblies. You can display the selected pipeline with a pipe solid or centerline. Note that if you are creating pipelines based on the "one pipeline per assembly" modeling method, it is recommended that you use this centerline/solid display mode.

**pipe\_update\_pre\_20\_int\_pts (Non Spec-Driven)***yes, no*

Pro/PIPING can update pipelines created prior to Pro/PIPING Release 20.0. This enables users to get intersection point entities. Pro/PIPING creates these entities in session (**Info > Regen Info**) or when you retrieve the piping model.

- *yes*—Pro/PIPING updates pipelines created prior to Pro/PIPING Release 20.0.
- *no*—Pro/PIPING does not update pipelines created prior to Pro/PIPING Release 20.0.

**piping\_design\_method (Non Spec-Driven)***non\_spec\_driven, spec\_driven, user\_driven*

Specify the piping design method for the piping project.

- *non\_spec\_driven*—Activates the Non Specification-Driven piping design mode.
- *spec\_driven*—Activates the Specification-Driven piping design mode.
- *user\_driven*—Activates the User-Driven piping design mode. This mode allows you to switch between Specification-Driven and Non Specification-Driven piping design modes and convert existing assemblies at any time in the design process.

**piping\_fluid\_parameter (Non Spec-Driven)***yes, no*

Specify pipeline fluid parameter availability.

- *yes*—Pro/PIPING sets the **Parameters** tab in the **Create Pipeline** dialog box as available.
- *no*—Pro/PIPING sets the **Parameters** tab in the **Create Pipeline** dialog box as unavailable.

**piping\_system\_tree\_format (Non Spec-Driven)**

*MNEMONIC*, <one or a combination of the following keywords: SIZE, SPECIFICATION, MNEMONIC, NUMBER, INSULATION, and any ASCII characters>. Delimiter characters must be enclosed by commas (,) omitting any white space. For example *MNEMONIC,-,SIZE*.

Specify the piping system tree format. Pro/PIPING creates piping systems or categories from the values assigned to the keywords. These categories organize all pipelines in the Piping System Tree. The following table describes the format:

<b>Piping System Tree Format</b>	<b>Pipeline Values</b>	<b>Piping System (Category) Name</b>
MNEMONIC-SPECIFICATION	WATER, 11C	WATER-11C

MNEMONIC-NUMBER-INSULATION	WATER, 1001, IS	WATER-1001-IS
SHIP1-MNEMONIC-NUMBER-INSULATION	WATER, 1001, IS	SHIP1-WATER-1001-IS

### **pipng\_wildfire\_convert (Non Spec-Driven)**

*automatic, never, manual*

Specify a value for this configuration option when working with piping assemblies that are created using versions of Pro/PIPING earlier than Wildfire.

- *automatic*—Automatically converts all piping assemblies that are under the active top assembly when you invoke the Pro/PIPING application.
- *never*—Does not convert piping assemblies.
- *manual*—Displays the **Piping Assemblies Conversion** dialog box when you invoke the Pro/PIPING application. This is the default value.

**Note:** A piping assembly gets modified if it is updated to the Wildfire format.

### **pro\_insulation\_dir**

*<user-defined directory>, <absolute directory path>*

Specify the directory to search for insulation files used in Pro/PIPING.

### **pro\_pip\_fitt\_dir (Non Spec-Driven)**

*<user-defined directory>, <absolute directory path>*

Specify the fitting library directory path. Pro/PIPING uses this library during fitting insertion.

### **pro\_pip\_instk\_dir (Non Spec-Driven)**

*<user-defined directory>, <absolute directory path>*

Specify the line stock library directory path. Pro/PIPING uses this library during pipeline creation.

## **Non Specification-Driven Pipelines**

### **About Pipelines**

Pro/PIPING defines a pipeline as a set of pipe segments that can be straight or flexible. A piping system typically has several pipelines composed of non-branching chains of pipe segments. You may declare necessary pipelines in advance, and then select one for routing.

Pro/PIPING stores pipelines in an assembly as assembly features. You can route and modify them in subassemblies, referencing components from any level of the current simplified representation. Using the **Component** command in the **PIPING** menu, you can create and delete subassemblies using the same functionality available in Assembly mode.

To define a piping system, you assign characteristics such as line stocks, pipeline parameters, allowable line shapes, and corner types.

- When you assign a line stock to pipe segments, you define material, outside diameter, line shape and corner types, and other pipe parameters. The system can write line stock parameters to a file for storage in libraries.
- Each pipeline has a set of parameters that define the pipe name, the type of fluid or gas to be carried, and user-defined parameters such as flow direction and pressure.

The following restriction applies to the use of parameters in Pro/PIPING:

In report relations, you cannot use piping report parameters that are longer than 64 characters.

- The line shape of pipe segments varies. It can be straight (that is, rigid), flexible, or composed of alternating straight and flexible segments. You would typically use alternating segments only when you are routing a hose and need to control tangency for a short section, as when leaving a fitting or going through a bulkhead.
- Specify a corner fitting. Corners can be fitted, mitered, or bent.

### To Create a Piping System

1. Retrieve an assembly that contains all required components with coordinate systems and entry ports. You can also create an assembly that contains only piping.
2. You can copy surfaces in advance and route by referencing them in a subassembly.

You can either route up to previously assembled fittings or route pipelines and insert fittings later.

3. Click **Applications > Piping** to access the Pro/PIPING module.
4. From the **PIPING** menu, create or read in line stock files.
5. Create pipelines by choosing **Pipeline > Create/Route** and typing a name.
6. Choose **LINE STOCK NAMES > Create**. Enter a new line stock name. The **LineStock** dialog box opens.
7. Assign a line stock.

8. Route pipelines and add branches between pipelines. While routing, pipeline segments do not have to be continuous; you can create individual segments and later connect them using intermediate segments, or leave them unconnected.
9. Insert fittings.
10. Make a solid pipe.
11. Extract a model, if needed, to retrieve a pipeline with equipment and/or assembly components.

### To Delete a Pipeline (Non Spec-Driven)

1. Click **PIPING > Pipeline > Delete**.
2. Select a pipeline to delete. The **CONFIRMATION** menu appears.
3. Click **Confirm**. The **CHILD OPTS** menu appears.
4. Select one of the following menu commands:
  - **Delete All**—Deletes all highlighted children. The **CONFIRMATION** menu appears. Click **Confirm**.
  - **Suspend All**—Suspends all highlighted children. The children remain displayed in the graphics window.
  - **Child Info**—Displays all information about the children to be deleted or suspended in the INFORMATION WINDOW.

### To Rename a Pipeline (Non Spec-Driven)

1. Click **PIPING > Pipeline > Rename**.
2. Select a pipeline to rename.
3. Enter a new name for the pipeline and click . Pro/PIPING confirms the status of this task in the message area.

### To Suppress a Pipeline (Non Spec-Driven)

1. Click **PIPING > Pipeline > Suppress**.
2. Select the pipeline to suppress. The **CONFIRMATION** menu appears.
3. Click **Confirm**. Pro/PIPING suppresses the selected pipeline and all of its highlighted children.

### To Resume a Suppressed Pipeline (Non Spec-Driven)

1. Click **PIPING > Pipeline > Resume**. The **SEL MENU** appears and lists all suppressed pipelines by name.

2. Select a suppressed pipeline. Pro/PIPING resumes the pipeline. You can view the resumed pipeline in either the Model Tree, the Piping System Tree, or the graphics window.

## Non Specification-Driven Piping Assemblies

### About Piping Assemblies (Non Spec-Driven)

Pro/PIPING organizes all pipelines, fittings, and equipment for a piping system in assemblies and subassemblies. Pro/PIPING creates piping assemblies after you create pipelines. All subsequent modeling occurs within the currently active assembly. Pro/PIPING allows you to do the following:

- Change active assemblies at any time in the piping design process by selecting a pipeline in the graphics window.
- Convert existing piping assemblies created in one piping design mode to the other mode on-the-fly using the User-Driven piping design mode.

### To Make a Piping Assembly Active (Non Spec-Driven)

You can change the currently active assembly at any time in the piping design process.

1. Create or open a piping assembly.
2. Click **Applications > PIPING**. The **PIPING** menu appears.
3. Click **PIPING > Active Asm**.
4. Select a pipeline from the graphics window. The assembly associated with the selected pipeline becomes the active assembly. Pro/PIPING displays the active assembly name in the graphics window.

You can also select an assembly from the Model Tree or the Piping System Tree.

### About Converting Piping Assemblies (Non Spec-Driven)

The User-Driven piping design mode allows you to convert piping assemblies at any time in the piping design process using the **Spec Driven** check box on the **PIPING** menu. You can convert existing Non Specification-Driven or Specification-Driven piping assemblies to the other design mode on-the-fly and continue designing your piping system.

After setting the `piping_design_method` configuration option to the User-Driven mode, the following actions occur:

- If a piping assembly is not open, Pro/PIPING defaults to the Non Specification-Driven design mode. Select the **Spec Driven** check box to switch to the Specification-Driven design mode.
- If an existing piping assembly is open or after you open an existing piping assembly, Pro/PIPING switches to the piping design mode in which the assembly

was created. The **Spec Driven** check box indicates the design mode. You can always convert the assembly to switch design modes.

### Assembly Conversion Processes

Pro/PIPING executes two different piping assembly conversion processes based on the piping assembly type.

Assembly Type	Conversion Process
Specification-Driven to Non Specification-Driven	<p>Clear the <b>Spec Driven</b> check box on the <b>PIPING</b> menu. After this process, Pro/PIPING considers all pipelines as Non Specification-Driven.</p> <p>All existing fittings will remain Specification-Driven.</p>
Non Specification-Driven to Specification-Driven	<p>Select the <b>Spec Driven</b> check box on the <b>PIPING</b> menu. You must assign specification-specific data in the <b>Convert Pipeline</b> dialog box for each pipeline in the active assembly. After entering this data, the Pro/ENGINEER message area displays each pipeline and its conversion status. All pipelines must be converted to Specification-Driven.</p> <p>All existing fittings will remain Non Specification-Driven.</p>

**Note:**

- The piping assembly conversion processes occur on a pipeline basis only. Fittings are not converted.
- Mixed models are not allowed. Non Specification-Driven and Specification-Driven pipelines cannot be included in the same piping assembly.
- Mixed assemblies are allowed. Non Specification-Driven and Specification-Driven assemblies and subassemblies can be grouped.
- All existing line stocks are automatically deleted. The Non Specification-Driven mode permits different line stock assignments to each pipe segment.
- Pipe solids failure can occur due to pipeline size changes.

### To Convert Non Specification-Driven Piping Assemblies (Non Spec-Driven)

You can convert Non Specification-Driven Piping assemblies to Specification-Driven Piping assemblies. This allows you to use existing pipe systems in the Specification-Driven Piping design mode.

1. Configure Pro/PIPING for the User-Driven piping design mode.
2. Create or open a Non Specification-Driven piping assembly to convert.
3. Click **Active Asm** on the **PIPING** menu and select a piping assembly from either the Piping System Tree or the graphics window.
4. Select the **Spec Driven** check box on the **PIPING** menu. The **Convert Pipeline** dialog box opens. Pro/PIPING displays the active assembly under **Active Assembly**.
5. If you have set the `piping_schematic_driven` configuration option to `yes` the **Schematic Driven** option will be available under **Pipeline**. To convert a pipeline using the schematic information, perform the following steps:

- a. Check the **XML File** option under **Schematic Driven** to make  active.
- b. Click  and select an XML file from the directory path specified by the `piping_schematic_xml_dir` configuration option or your current working directory. You can browse to any required directory to select the XML file.
- c. The **Schematic Label** option under **Schematic Driven** lists all the eligible pipelines for conversion, based on the information in the specified XML file. Select a pipeline from the eligible list.
- d. Click the **Label** tab. The **Specification, Size, Schedule, Mnemonic, Number, and Insulation** boxes get updated.
- e. Click the **Parameters** tab. The **Oper. Pressure, Oper. Temperature, Design Pressure, and Design Temperature** boxes get updated if the schematic information contains the respective operating and design pipeline parameters.
- f. Click the **Defined** tab. The **User Defined Parameters** get updated.

**Note:** To convert a pipeline without the schematic information, clear the **XML File** option and perform the following steps.

6. Select a pipeline to convert from the **Select Pipeline** box. If only one pipeline is available, Pro/PIPING displays its name beside **Pipeline**.
7. Click the **Label** tabbed page. Under **Pipeline Label**, do the following:
  - o Select a specification, size, and schedule for the selected pipeline.
  - o If applicable, select **Mnemonic, Number, and Insulation**. Note that a pipeline number uniquely identifies a pipeline and it is needed for the Piping System Tree to organize pipelines (by their number) in a piping system.
8. If applicable, enter pipeline parameter values. Select the **Parameters** tabbed page. Under **Pipeline Parameters**, enter the parameter values. By default, Pro/PIPING makes the pipeline parameters unavailable. The `piping_fluid_parameter` configuration option sets the pipeline parameter availability (default is `no`).

9. If applicable, enter user-defined pipeline parameters. Select the **Defined** tabbed page. Under **User Defined Parameters**, do one of the following:
  - Type a parameter name in the **Name** box (must begin with a letter) and a corresponding value in the **Value** box. Click  to add the parameter to the pipeline.
  - Retrieve existing user-defined pipeline parameters from a file. Click . The **Open** dialog box opens. Select the file to retrieve and double-click.

Pro/PIPING displays the user-defined parameters.

10. Click  to save the parameters to a file.

11. Click  to accept all pipeline data and begin the conversion process.

The Pro/ENGINEER message area displays the pipeline conversion status. Pro/PIPING generates new line stocks based on the assigned specification.

**Note:**

- You must perform this conversion process each time you switch from the Non Specification-Driven to the Specification-Driven design mode.
- You must select each assembly to convert. Mixed assemblies are allowed.
- Every pipeline in the active assembly must be converted.
- All existing fittings remain Non Specification-Driven.
- All existing line stocks are automatically deleted.
- Pipe solids failure can occur due to pipeline size changes.

**To Convert Specification-Driven Piping Assemblies (Non Spec-Driven)**

You can convert Specification-Driven Piping assemblies to Non Specification-Driven Piping assemblies. This allows you to use the existing pipe systems in the Non Specification-Driven piping design mode.

1. Configure Pro/PIPING for the User-Driven piping design mode.
2. Create or open a Specification-Driven piping assembly to convert.
3. On the **PIPING** menu, clear the **Spec Driven** check box. Pro/PIPING considers all pipelines as Non Specification-Driven.

**Note:**

- Every pipeline in the active assembly must be converted because the Non Specification-Driven mode permits different line stock assignments for each pipe segment.

- All existing fittings remain Specification-Driven. So, if you convert the pipeline to Specification-Driven, Pro/PIPING selects and inserts all Specification-Driven fittings according to the pipeline specification.
- Pipe solids failure can occur due to pipeline size changes.

## Converting Piping Assemblies to Pro/ENGINEER Wildfire Format

When you work with piping assemblies that are created in versions of Pro/PIPING earlier than Pro/ENGINEER Wildfire, the piping data stored in the assembly needs to be updated or converted to the Wildfire format. Otherwise, you cannot check design rules, display flow direction, or perform automatic reorder of the piping model. If parameters such as STOCKNUMBER and MATERIALDATA are stored in the fitting component features that are created in versions of Pro/PIPING earlier than Pro/ENGINEER Wildfire, then these parameters must be converted to appropriate parameters in the Wildfire format. For example, STOCKNUMBER must be converted to STOCKNO and MATERIALDATA must be converted to MATERIALCODE and MATERIALDESC in the Wildfire format.

When you specify `automatic` as a value for the `piping_wildfire_convert` configuration option, piping assemblies are automatically converted when you invoke the Pro/PIPING application. If you set the `piping_wildfire_convert` configuration option to `manual`, you can use the **Piping Assemblies Conversion** dialog box and select one of the following conversion types to convert pipeline assemblies. The **Piping Assemblies Conversion** dialog box opens when you click **Applications > Piping**.

- `Automatic`—Automatically converts all the piping assemblies in the current Pro/PIPING session.
- `Never`—Piping assemblies are not converted in the current Pro/PIPING session.
- `Manual`—Selects the piping assemblies to convert from the Select Pipeline Assembly to Convert list. All the assembly names are selected by default. You can select any assembly that you want to convert from the list.

**Note:** When you convert piping assemblies, the assembly revision number is updated automatically.

## Non Specification-Driven Parameters

### About Non Specification-Driven Parameters

You set up non specification-driven piping by adding and modifying various parameters. Pro/PIPING provides the following parameters:

- **Line Stock Parameters**—Enables you to modify the line stock data.
- **Pipeline Parameters**—Allows you to modify pipelines.
- **Bend Parameters**—Enables you to change bend tables.
- **Design Rule Parameters**—Allows you to change the design rules.

## Line Stock Parameters

### About Line Stock Parameter Files

A line stock is a set of parameters that define material, grade, outside diameter, and other parameters of pipeline segments. Pro/PIPING stores the parameters in the line stock feature. Before you create a pipeline, you must define a line stock.

You can assign different line stocks to different line segments of the same pipeline because they may have different outside diameters, materials, and grades. Keep in mind that the system determines a pipeline's physical (solid) shape based on the line stock from which it actually derived the pipe.

You can edit line stock parameters using the **LineStock** dialog box, and you can add user-defined attributes to the list. You can also read and write a line stock to a `.stk` file.

You can set up a library of line stocks that you can retrieve to route a particular pipeline. These files have the extension `.stk`. You can also create line stocks on the fly whenever you assign a new line stock to a line segment. Add the `pro_pip_lnstk_dir` line stock configuration file option to your configuration file. This directory is the path to your library of line stocks.

When a line stock parameter has multiple values, the first value is the default, and all specified options are available when you work with a pipeline.

The system uses the parameters identified as `check pipe` to compare the characteristics of a pipeline when you check for line stock violations.

### To Add Line Stock Parameter Files

1. Click **PIPING > Set Up > Line Stock > Create**.
2. Enter the new line stock name. The **LineStock** dialog box opens with the default line stock parameters.
3. Edit the line stock parameters as desired and click  to apply the line stock changes or  to save the line stock into file.
4. A dot and message appear at the bottom of the **LineStock** dialog box, informing you of the destination of the saved parameters.

### Using the LineStock Dialog Box

The **LineStock** dialog box has three tabs: **LineStock**, **DesignRule**, and **Defined**. The tabs separate and identify the parameters and allow you to add and modify line stock and bend parameters. The variable type (integer, double or string) is automatically assigned based on the syntax of the entry for the user defined parameters only.

## LineStock

The **LineStock** tabbed page consists of several areas in which you can change line stock parameters.

- **General Parameters**
  - **Stock No**—Pipe is the default. You can change this value to any continuous string, for example, plastic\_tubing.
  - **Material**—Steel is not a required value. You can change this value to any continuous string, for example, solid\_brass.
  - **Grade**—A123 is not a required value. You can change this value to any continuous string.
- **X Section**—**Rectangular** or **Circular**. When you select one of these, a square or round pipe example is displayed in the **Section Parameters** box.
- **Section Type**—**Hollow** or **Solid**.
- **Section Parameters**—**Height**, **Width**, **Angle**, and **Thickness** if the **X Section** option is set to **Rectangular**. **Pipe OD** and **Thickness** if the **X Section** option is set to **Circular**.
- **Shape Type**—**Straight** or **Flexible** is indicated by an option button that specifies the default setting.
  - When the **Shape Type** is **Straight**, the **Corner Type** is active, and you can select one of the three **Corner Types**, **Bend**, **Fitting**, or **Miter Cut**.
  - When the **Shape Type** is **Flexible**, the **Corner Type** and **Bend** information are not active.

You can also edit the following values:

- **Weight/Length (WT\_LEN)**—The weight per unit length parameter that defines the density of any pipe solid part that is created on the pipe line.
- **Bend Radius**, **Bend Angle**, **Miter Cut Length**, and **Miter Cut Number** for the **Corner Type** you have chosen.
- **Corner Type**—**Bend**, **Fitting**, or **Miter Cut**.

The indicator dot assigns the default corner type.

## DesignRule

The **DesignRule** tabbed page contains the default Design Rule parameters. You can change active parameters. If you click , all the Line Stock parameters are saved to a file. If you click , the changes to the Design Rule parameters are applied to the routed pipe.

When the Line Stock parameter is set to **Shape Type/Straight**, all of the Design Rule parameters are active. If it is set to **Shape Type/Flexible**, some of the Design Rule parameters are not available.

### Defined

The **Defined** tabbed page contains user defined parameters. You can add user-defined Line Stock or Bend parameters by typing a name and value, and then clicking .

- If you click , the Line Stock is saved to a file.
- If you click , the Line Stock is created with the assembly.

While you are editing, the parameter is not applied to the routed pipe parametrically. The parameter name should begin with a letter.

### Design Rule Parameters

Use the **DesignRule** tab in the **LineStock** dialog box to verify that parameters of a routed pipe are within limitations.

Design Rule Parameter	Value Checked
MAX_OVERALL_LENGTH	Length of a pipe segment.
MAX_BENDS_NUMBER	Number of bends in a pipe segment.
MIN_BEND_SEPARATION	Minimum distance between bends in one continuous pipe segment.
MIN_STRAIGHT_LENGTH	Minimum distance (in assembly units) allowable from the pipe end to the next bend or endpoint.
MIN_BEND_RADIUS	Minimum radius allowable in a bend.
MAX_BEND_RADIUS	Maximum radius allowable in a bend.
MIN_BEND_ANGLE	Minimum angle allowable in a bend.
MAX_BEND_ANGLE	Maximum angle allowable in a bend.
BEND_ARC_THRESHOLD	Angle at which Pro/PIPING defines the dimensioning scheme for large angle bends, such as 180 degrees.
MIN_BRANCH_SEPARATION	Minimum distance between branches in one continuous pipe segment.

MIN_BRANCH_ANGLE	Minimum angle allowable in a branch.
BEND_TABLE_NAME	Name of the bend table.

### Additional Features of the LineStock Dialog Box

Other features of the **LineStock** dialog box are:

- A right click in the Model Tree that lets you edit parameters in the **LineStock** dialog box.
- An indicator dot in **Shape Type** and **Corner Type** options to denote the default type.
- A new configuration file option for the line stock directory: `pro_pip_lnstk_dir <full path to user directory>`.
- Standard values for Diagonal Miter Cut corners. Line stock variables like `WT_LEN` can now be used in Pro/REPORT relations as numeric values.

### Example: Typical Line Stock Parameter File

```

TYPE PIPE
MATERIAL Steel
GRADE A123
SECTION_TYPE HOLLOW
OD 0.25
THICKNESS 0.032
SHAPE_TYPE STRAIGHT FLEXIBLE
CORNER_TYPE BEND FITTING MITER_CUT
WT/LEN 10
BEND_RADIUS 1.5 2 3
BEND_ANGLE 90 45 60
MITER CUT LENGTH 0.05 0.1 0.5
MITER CUT NUMBER 1 2 3

```

### Writing Relations

You can access piping line stock parameters, pipeline parameters, and pipeline bend parameters through writing relations. The syntax for the relation is the same as that for part modeling.

The feature name of the ID must refer to the line stock, pipeline feature, or pipe bend for the pipe, not to the pipe routing feature. You cannot set their values through the relations you create, but you can control dimension values.

The following file is an example of how to write a relation to control the width of a channel in a pipeline. The following relations control the width of the channel, `d7:0`,

to keep it equal to the outer diameter of the pipe, as specified in the pipeline feature's line stock parameters (assuming that the only two fluids in use are oil and H2O and that each gets assigned a single line stock):

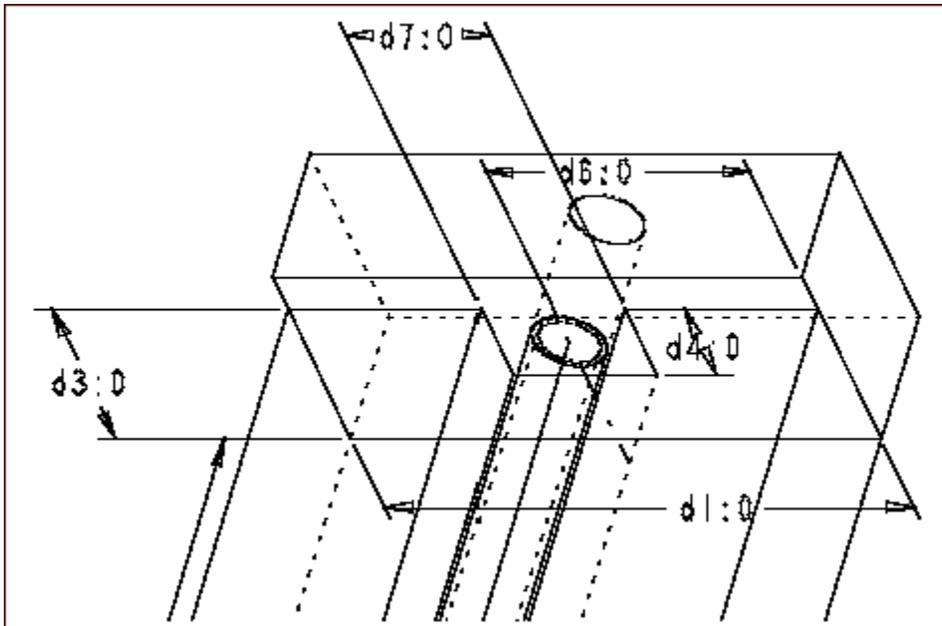
```
IF fluid:fid_LINE1 = 'oil'  
d7:0 = OD:fid_OIL_STK  
ELSE  
d7:0 = OD:fid_H2O_STK  
ENDIF
```

If this pipeline (LINE1) has `oil` as the value for the `fluid` parameter, make the channel width equal to the outer diameter in the line stock `OIL_STK`. Otherwise, the channel width should be equal to the outer diameter of the line stock `H2O_STK`.

Because the outer diameter of the pipeline, `OD`, is a parameter of a line stock (in this case `OIL_STK` or `H2O_STK`), the system uses the feature ID of the appropriate line stock in the relation.

Also, because this is an assembly relation, not a part relation referencing an assembly, a component ID is not required in the relation. Keep in mind that the system determines a pipeline's physical (solid) shape based on the line stock from which it actually derives the pipe.

### Example: Using Relations in Pro/PIPING



You can write a relation to control the width of the channel, `d7:0`, to keep it equal to the outer diameter of the pipe, which is specified in the line stock parameters of the pipe feature.

## Setting the WT\_LEN Parameter

WT\_LEN is a line stock parameter from which the piping solid generates the density value for mass property calculations. Pro/ENGINEER sets the density value for a piping solid using the following equation:

$$\text{Density} = \text{WT\_LEN} / A; A = \text{PI}/4 * (\text{OD}^2 - (\text{OD} - 2*\text{WT})^2)$$

where

^ is power

OD is outline diameter (from line stock)

WT is wall thickness (from line stock)

When you regenerate the piping solid in Assembly mode, the solid checks the current line stock for the segment and looks at the OD and WT\_LEN values to verify that the current density is correct.

However, if you regenerate the piping solid in Part mode, the density value remains frozen because the line stock parameters are not available.

## To Assign a Default Line Stock for New Line Segments

1. Click **PIPING > Route**.
2. Select the pipeline to route. The **ROUTE PIPE** menu appears.
3. Click **Pipe Envrmt**. The **PIPE ENV** menu appears.
4. Click **Line Stock**. In the message window, the system indicates the name of the current line stock, and the **LINE STOCK NAMES** menu appears.
5. Choose a line stock to assign to the next line segment or use **Create** to create a new line stock.

## To Assign the Default Corner Type for New Line Segments

1. Click **PIPING > Route**.
2. Select the pipeline to route. The **ROUTE PIPE** menu appears.
3. Click **Pipe Envrmt**. The **PIPE ENV** menu appears.
4. Click **Corner Type**. The **CORNER TYPE** menu appears.
5. From the **CORNER TYPE** menu, click one of these commands:
  - o **Fitting**—Creates sharp corners so that you may later add corner fittings. Fitted corners create breaks in a pipeline.

- **Bend**—Creates each corner by bending a tube. Type a bend radius by selecting any of the predefined values, or click **New Value** and type a desired value. If you are not changing the corner type, but only want to reassign default bend corner values, click **PIPE ENV > Corner Vals**, select a value from the **SEL VALUE** menu, and type the replacement bend radius value.
- **Miter**—Creates a corner by adding a miter cut. Select a value from the **SEL VALUE** menu to specify the number of miter cuts. To select values other than 1, 2, or 3, select **New Value** and enter the number of miter cuts required.

## To Assign a Line Shape as a Default for New Line Segments

1. In the **ROUTE PIPE** menu, click **Pipe Envrnmt > Line Shape**.
2. Using the **LINE SHAPE** menu, do one of the following:
  - Click **Flexible** and **Free Length** to have the system determine the length of the flexible pipe segment.
  - Click **Flexible** and **Set Length** to specify a length for the flexible pipe after creating each segment.
  - Click **Straight**.

**Free Length** and **Set Length** are only available when you have set **Flexible** in the **LINE SHAPE** menu.

3. Click **Done**.

## Assigning a Flexible Line Shape

As with all piping environment commands, these settings take effect beginning with the next routed pipeline segment until you change them. The default length for a new segment is the length the segment would be if you used **Free Length**.

- Once you click type a value for a pipe segment length, the system constrains the flexible segment length at that value. Use the **Modify Pipeline** dialog box to reroute a fixed length segment as a free length.
- A dimension controls the fixed length. You can modify it as you would any other dimension. Because the pipeline controls the corresponding solid pipe feature, you must select the pipeline to modify a fixed length, not the solid (if there is one).
- You cannot set fixed lengths for straight segments even if the straight segments are part of a segment that is routed mostly as flexible pipe.

## Pipeline Parameters

### About Pipeline Parameters

Every pipeline has a set of associated parameters, in addition to the parameters in the referenced line stock, that apply to that individual pipeline. You may write the pipeline parameters to a file and may include system parameters (such as the name of the line parameter file and fluid or gas type), as well as user-defined parameters.

The values for the pipeline parameters that you write to the file automatically define the parameter types such as string, integer, or double. This is valid for the pipeline, line stock, and insulation parameters.

### To Create User Defined Pipeline Parameters

1. Click **Piping > Modify Pipe**. The **Modify Pipeline** dialog box opens.
2. Click **Parameters** and then under **Pipe Line Parameters** click **Pipe Line**. The **Edit Pipeline** dialog box opens.
3. Select a pipeline from the drop-down list box. Under **User Defined Parameters**, type a new name and value.
4. Click  and . The new parameter is applied to the pipeline.

#### Note:

- You can also **Delete** and **Read** in parameters from a file in this dialog box.
- The parameter's name should begin with a letter.

### Using the Modify Pipeline Dialog Box

The **Modify Pipeline** dialog box lets you modify several options in the active assembly, the corners, parameters, line stocks, and flex shapes. In the **Active Assembly** area of this dialog box, click  to activate the assembly in the Pro/ENGINEER window.

In the **Modify Options** area, the following choices appear.

- **Corner**—Select types and values of single or multiple corners.
- **Line Stock**—Set the values of selected single or multiple segments.
- **Parameters**—Modify Pipeline and Bend parameters.
- **Flex Shape**—Modify the line shape of a flexible hose between Fixed and Free length.
- **Flow Direction**—Select the flow unit and change the direction of the flow.

When you click an option under **Modify Options**, the last area in the dialog box changes. For example, if you click **Corner**, a **Corner** group box appears. You need to

select a pipe segment corner to change it to a Bend, Fitting, or Miter Cut in the **New Type** box and to modify the value of the bend or fitting in the **New Value** box.

If you click the **Parameters** option, the **Parameters** group box opens.

In the **Parameters** area, you can select **Pipe Line Parameters > Pipe Line** or **Bend Parameters > Bend** to modify these parameters. If you click **Pipe Line**, the **Edit Pipeline** dialog box opens. If you click **Bend**, the **Bend Parameters** Model Tree opens.

If you click the **Line Stock** option, the **Line Stock** group box opens.

Modify the line stock listed in the **Modify Linestock** drop-down box with the **Edit** option or create a new line stock with the **Create new** option. If you click **Create new**, enter the line stock name at the command prompt.

If you click the **Flex Shape** option, the **Flex Shape** group box opens.

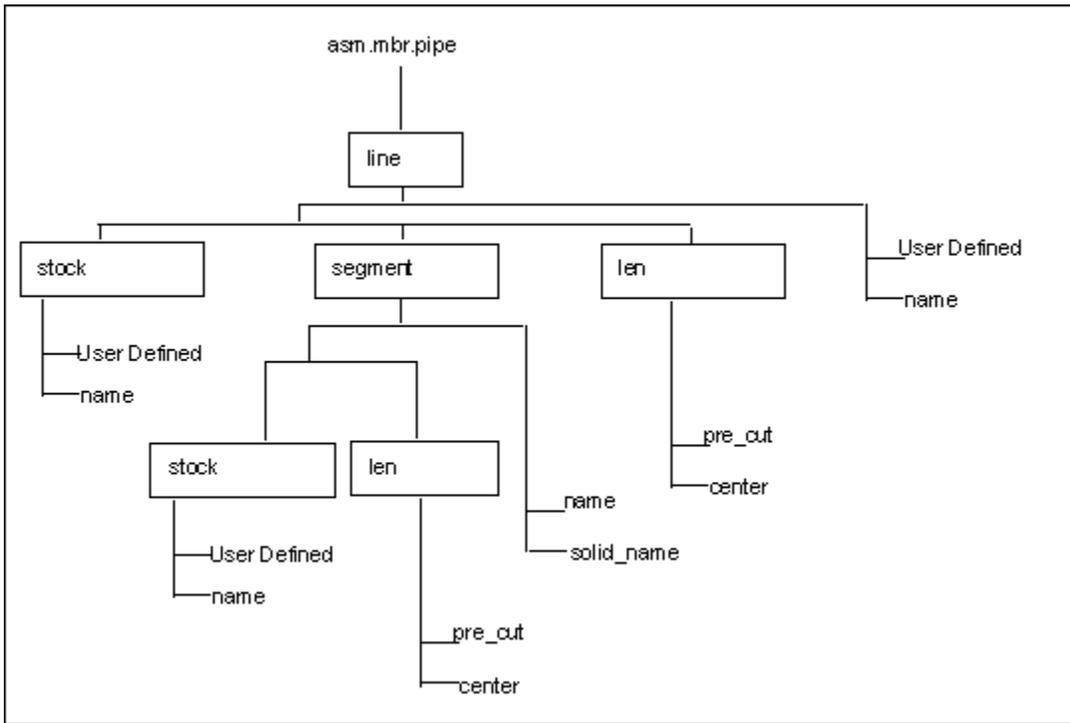
Modify the **Flex Shape** by selecting a flexible pipe segment. The **Modify Shape** group box contains two shapes that you can change, **Fixed Length** and **Free Length**. If you click **Fixed Length**, the **Length** box activates with a number you can modify.

### Using the Edit Pipeline Dialog Box

The **Edit Pipeline** dialog box opens after you click **Parameters** in the **Modify Pipeline** dialog box. The **Edit Pipeline** dialog box lets you perform the following tasks:

- Select a pipeline parameter from the Pro/ENGINEER window.
- Give the parameter a new name.
- Add the parameter to the pipeline.
- Save the parameter to a file.
- Read in the parameter as a .pp1 file.
- Delete the parameter from the pipeline.
- Cancel changes made to the pipeline.
- Modify an existing pipeline label.

**Pro/REPORT Pipeline Parameters**



**Pro/REPORT Parameters Applicable to Pro/PIPING**

**Linestock Parameters**

Parameter Name	Definition
&asm.mbr.pipe.stock.name	Lists name(s) of line stock(s).
&asm.mbr.pipe.stock.User Defined	Lists specified parameters for the line stock.
&asm.mbr.pipe.stock.line.name	Lists names of all pipelines using each line stock.
&asm.mbr.pipe.stock.line.User Defined	Lists parameters for the pipeline using the specified line stock.
&asm.mbr.pipe.stock.material	Lists material for all line stocks.
&asm.mbr.pipe.stock.OD	Lists outer diameter for line stocks.

&asm.mbr.pipe.stock.wallthick	Lists wall thickness for line stocks.
&asm.mbr.pipe.stock.len.pre_cut	Lists total precut length of each line stock.
&asm.mbr.pipe.stock.len.center	Lists total centerline length of each line stock.

### Pipeline Parameters

Parameter Name	Definition
&asm.mbr.pipe.line.segment.stock.name	Lists names of line stocks for each segment in the pipeline.
&asm.mbr.pipe.line.segment.stock.User Defined	Lists parameters for line stocks of which the pipeline is made.
&asm.mbr.pipe.line.name	Lists names of all pipelines.
&asm.mbr.pipe.line.User Defined	Lists parameters for each pipeline.
&asm.mbr.pipe.line.stock.name	Lists names of line stocks used in each pipeline.
&asm.mbr.pipe.line.stock.User Defined	Lists parameters for line stocks used in pipeline.
&asm.mbr.pipe.line.len.pre_cut	Lists precut length of each pipeline.
&asm.mbr.pipe.line.len.center	Lists centerline length of each pipeline.
&asm.mbr.pipe.line.segment.name	Lists names of all segments in pipeline.
&asm.mbr.pipe.line.segment.solid_name	Lists names of all solids (if defined) in pipeline.
&asm.mbr.pipe.line.segment.len.pre_cut	Lists precut length of segments in pipeline.
&asm.mbr.pipe.line.segment.len.center	Lists centerline length of

	segments in pipeline.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.ang</code>	Lists clocking angle value from evaluate feature.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.cs1</code>	Lists name of initial coordinate system for each clocking angle.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.cs2</code>	Lists name of final coordinate system for each clocking angle.

### Pipe Segment Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.name</code>	Lists system default name of the pipeline segments.
<code>&amp;asm.mbr.pipe.segment.solid_name</code>	Lists solid part name for each segment (if defined).
<code>&amp;asm.mbr.pipe.segment.stock.name</code>	Lists line stock used in each segment.
<code>&amp;asm.mbr.pipe.segment.stock.User Defined</code>	Lists parameters in line stocks assigned to each segment.
<code>&amp;asm.mbr.pipe.segment.line.name</code>	Lists name of pipeline to which each segment belongs.
<code>&amp;asm.mbr.pipe.segment.line.User Defined</code>	Lists pipeline parameters for each segment.
<code>&amp;asm.mbr.pipe.segment.len.pre_cut</code>	Lists precut length of each segment.
<code>&amp;asm.mbr.pipe.segment.len.center</code>	Lists centerline length of each segment.
<code>&amp;asm.mbr.pipe.segment.segfrom</code>	Lists location from which each pipeline segment in the assembly is routed. This is the component to which the <i>from</i> entry port

	belongs.
<code>&amp;asm.mbr.pipe.segment.segto</code>	Lists the location to which each pipeline segment in the assembly is routed. This is the component to which the <i>to</i> entry port belongs.
<code>&amp;asm.mbr.pipe.segment.clk_ang.ang</code>	Lists clocking angle value of evaluate feature.
<code>&amp;asm.mbr.pipe.segment.clk_ang.cs1</code>	Lists name of initial coordinate system for each clocking angle.
<code>&amp;asm.mbr.pipe.segment.clk_ang.cs2</code>	Lists name of final coordinate system for each clocking angle.

### Pipeline Segment Bend Information Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_angle</code>	Lists bend angle.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_num</code>	Lists bend location bend number.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_radius</code>	Lists bend radius.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_x</code>	Lists x-offset bend location information.

<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_y</code>	Lists y-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_z</code>	Lists z-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_bend</code>	Lists bend machine bend angle.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_num</code>	Lists bend machine bend number.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_offset</code>	Lists bend machine offset.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_radius</code>	Lists bend machine bend radius.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_twist</code>	Lists bend machine twist angle.
<code>&amp;asm.mbr.pipe.segment.csys.name</code>	Lists coordinate system name for segment bend information.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_bend</code>	Lists bend machine bend angle.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_num</code>	Lists bend machine number.

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<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_offset</code>	Lists bend machine offset.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_radius</code>	Lists bend machine radius.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_twist</code>	Lists bend machine twist angle.
<code>&amp;asm.mbr.pipe.segment.mach_csys.name</code>	Lists name of coordinate system with bend machine information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_angle</code>	Lists bend location bend angle.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_num</code>	Shows bend location bend number.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_radius</code>	Lists bend location bend radius.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_x</code>	Lists x-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_y</code>	Lists y-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_z</code>	Lists z-offset bend location information.

&asm.mbr.pipe.segment.loc_csys.name	Lists coordinate system name for segment bend location information.
-------------------------------------	---------------------------------------------------------------------

### Holes Report Parameters

Parameter Name	Definition
&asm.mbr.pipe.segment.holes.holes_params.hole_num	List holes report number.
&asm.mbr.pipe.segment.holes.holes_params.hole_offset	List holes report offset.
&asm.mbr.pipe.segment.holes.holes_params.hole_angle	List holes report angle.
&asm.mbr.pipe.segment.holes.holes_param..hole_diam	List holes report diameter.

### Bend Parameters

#### About Bend Parameters

Bend parameters are parameters that determine the size of pipe bends in an assembly.

#### To Create Bend Parameters in a Pipeline Segment

1. Click **PIPING > Modify Pipe**. The **MODIFY PIPELINE** dialog box opens.
2. Under **Modify Options** click **Parameters** and then under **Parameters** click **Bend Parameters > Bend**. The **Bend Parameters** window opens.
3. Expand the view of the **Bend Parameters** window by choosing **View > Expand All** or any other view you want.
4. Click  to select one or more bends from the active assembly on the screen, or choose one or more bends directly from the list in the Model Tree.
5. Click **Edit > Columns > Add/Remove**. Only selected bends are available for editing. The **Add/Remove Columns** dialog box opens.

6. Make the following changes, as necessary:
  - Add parameters to the bend by specifying a value in the column for the appropriate parameter.
  - Add columns to the table to include new parameters.
  - Delete parameters from the bend by removing the parameter value, and from all bends by selecting all bends and then removing the entire column.
  - Rename parameters by editing the column heading, and rename the parameter values by editing the table cells.
7. To update the bend parameters, click **Apply**. To update and exit, click **OK**.

### User-Defined Bend Parameters in Pro/REPORT Tables

Using the **LineStock** dialog box, you can view and modify parameters for all pipe bends in an assembly. You can access these parameters in report tables using the following parameter: `&asm.mbr.pipe.segment.csys.bend.User Defined`.

You can create default user-defined bend parameters by specifying them in the line stock file. When you specify them in the line stock file, the system applies them only when creating the bend during routing or when you change a corner to a bend when the line stock is active.

These parameters have no effect on an existing bend if you assign a new line stock to the segment, change the value of the default bend parameter in the current line stock, or add a bend parameter to or delete one from the current line stock.

Any new bends that you create after changing or modifying the line stock of the pipeline have the new default parameters and values of the current line stock.

Bends that you create by modifying a corner to type **Bend** acquire their default bend parameters from the current line stock for the segment.

As an alternative to creating default bend parameters in every bend, you can choose to assign parameters only to bends that vary from the standard, and then use report relations to assign a default value for bends without the parameter for the report table.

### Pro/REPORT Parameters Applicable to Pro/PIPING

#### Linestock Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.stock.name</code>	Lists name(s) of line stock(s).
<code>&amp;asm.mbr.pipe.stock.User Defined</code>	Lists specified parameters for the line stock.

&asm.mbr.pipe.stock.line.name	Lists names of all pipelines using each line stock.
&asm.mbr.pipe.stock.line.User Defined	Lists parameters for the pipeline using the specified line stock.
&asm.mbr.pipe.stock.material	Lists material for all line stocks.
&asm.mbr.pipe.stock.OD	Lists outer diameter for line stocks.
&asm.mbr.pipe.stock.wallthick	Lists wall thickness for line stocks.
&asm.mbr.pipe.stock.len.pre_cut	Lists total precut length of each line stock.
&asm.mbr.pipe.stock.len.center	Lists total centerline length of each line stock.

### Pipeline Parameters

Parameter Name	Definition
&asm.mbr.pipe.line.segment.stock.name	Lists names of line stocks for each segment in the pipeline.
&asm.mbr.pipe.line.segment.stock.User Defined	Lists parameters for line stocks of which the pipeline is made.
&asm.mbr.pipe.line.name	Lists names of all pipelines.
&asm.mbr.pipe.line.User Defined	Lists parameters for each pipeline.
&asm.mbr.pipe.line.stock.name	Lists names of line stocks used in each pipeline.
&asm.mbr.pipe.line.stock.User Defined	Lists parameters for line stocks used in pipeline.
&asm.mbr.pipe.line.len.pre_cut	Lists precut length of each pipeline.

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<code>&amp;asm.mbr.pipe.line.len.center</code>	Lists centerline length of each pipeline.
<code>&amp;asm.mbr.pipe.line.segment.name</code>	Lists names of all segments in pipeline.
<code>&amp;asm.mbr.pipe.line.segment.solid_name</code>	Lists names of all solids (if defined) in pipeline.
<code>&amp;asm.mbr.pipe.line.segment.len.pre_cut</code>	Lists precut length of segments in pipeline.
<code>&amp;asm.mbr.pipe.line.segment.len.center</code>	Lists centerline length of segments in pipeline.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.ang</code>	Lists clocking angle value from evaluate feature.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.cs1</code>	Lists name of initial coordinate system for each clocking angle.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.cs2</code>	Lists name of final coordinate system for each clocking angle.

### Pipe Segment Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.name</code>	Lists system default name of the pipeline segments.
<code>&amp;asm.mbr.pipe.segment.solid_name</code>	Lists solid part name for each segment (if defined).
<code>&amp;asm.mbr.pipe.segment.stock.name</code>	Lists line stock used in each segment.
<code>&amp;asm.mbr.pipe.segment.stock.User Defined</code>	Lists parameters in line stocks assigned to each segment.
<code>&amp;asm.mbr.pipe.segment.line.name</code>	Lists name of pipeline to which each segment belongs.
<code>&amp;asm.mbr.pipe.segment.line.User Defined</code>	Lists pipeline parameters for each segment.

<code>&amp;asm.mbr.pipe.segment.len.pre_cut</code>	Lists precut length of each segment.
<code>&amp;asm.mbr.pipe.segment.len.center</code>	Lists centerline length of each segment.
<code>&amp;asm.mbr.pipe.segment.segfrom</code>	Lists location from which each pipeline segment in the assembly is routed. This is the component to which the <i>from</i> entry port belongs.
<code>&amp;asm.mbr.pipe.segment.segto</code>	Lists the location to which each pipeline segment in the assembly is routed. This is the component to which the <i>to</i> entry port belongs.
<code>&amp;asm.mbr.pipe.segment.clk_ang.ang</code>	Lists clocking angle value of evaluate feature.
<code>&amp;asm.mbr.pipe.segment.clk_ang.cs1</code>	Lists name of initial coordinate system for each clocking angle.
<code>&amp;asm.mbr.pipe.segment.clk_ang.cs2</code>	Lists name of final coordinate system for each clocking angle.

### Pipeline Segment Bend Information Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_angle</code>	Lists bend angle.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_num</code>	Lists bend location bend

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	number.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_radius</code>	Lists bend radius.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_x</code>	Lists x-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_y</code>	Lists y-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_z</code>	Lists z-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_bend</code>	Lists bend machine bend angle.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_num</code>	Lists bend machine bend number.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_offset</code>	Lists bend machine offset.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_radius</code>	Lists bend machine bend radius.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_twist</code>	Lists bend machine twist angle.
<code>&amp;asm.mbr.pipe.segment.csys.name</code>	Lists coordinate system name for segment bend information.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.bend_name</code>	Lists the bend name created

	by the system.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_bend</code>	Lists bend machine bend angle.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_num</code>	Lists bend machine number.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_offset</code>	Lists bend machine offset.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_radius</code>	Lists bend machine radius.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_twist</code>	Lists bend machine twist angle.
<code>&amp;asm.mbr.pipe.segment.mach_csys.name</code>	Lists name of coordinate system with bend machine information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_angle</code>	Lists bend location bend angle.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_num</code>	Shows bend location bend number.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_radius</code>	Lists bend location bend radius.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_x</code>	Lists x-offset bend location

	information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_y</code>	Lists y-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_z</code>	Lists z-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.name</code>	Lists coordinate system name for segment bend location information.

### Holes Report Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.holes.holes_params.hole_num</code>	List holes report number.
<code>&amp;asm.mbr.pipe.segment.holes.holes_params.hole_offset</code>	List holes report offset.
<code>&amp;asm.mbr.pipe.segment.holes.holes_params.hole_angle</code>	List holes report angle.
<code>&amp;asm.mbr.pipe.segment.holes.holes_param..hole_diam</code>	List holes report diameter.

### Using the Modify Pipeline Dialog Box

The **Modify Pipeline** dialog box lets you modify several options in the active assembly, the corners, parameters, line stocks, and flex shapes. In the **Active Assembly** area of this dialog box, click  to activate the assembly in the Pro/ENGINEER window.

In the **Modify Options** area, the following choices appear.

- **Corner**—Select types and values of single or multiple corners.
- **Line Stock**—Set the values of selected single or multiple segments.

- **Parameters**—Modify Pipeline and Bend parameters.
- **Flex Shape**—Modify the line shape of a flexible hose between Fixed and Free length.
- **Flow Direction**—Select the flow unit and change the direction of the flow.

When you click an option under **Modify Options**, the last area in the dialog box changes. For example, if you click **Corner**, a **Corner** group box appears. You need to select a pipe segment corner to change it to a Bend, Fitting, or Miter Cut in the **New Type** box and to modify the value of the bend or fitting in the **New Value** box.

If you click the **Parameters** option, the **Parameters** group box opens.

In the **Parameters** area, you can select **Pipe Line Parameters > Pipe Line or Bend Parameters > Bend** to modify these parameters. If you click **Pipe Line**, the **Edit Pipeline** dialog box opens. If you click **Bend**, the **Bend Parameters Model Tree** opens.

If you click the **Line Stock** option, the **Line Stock** group box opens.

Modify the line stock listed in the **Modify Linestock** drop-down box with the **Edit** option or create a new line stock with the **Create new** option. If you click **Create new**, enter the line stock name at the command prompt.

If you click the **Flex Shape** option, the **Flex Shape** group box opens.

Modify the **Flex Shape** by selecting a flexible pipe segment. The **Modify Shape** group box contains two shapes that you can change, **Fixed Length** and **Free Length**. If you click **Fixed Length**, the **Length** box activates with a number you can modify.

### Using the Edit Pipeline Dialog Box

The **Edit Pipeline** dialog box opens after you click **Parameters** in the **Modify Pipeline** dialog box. The **Edit Pipeline** dialog box lets you perform the following tasks:

- Select a pipeline parameter from the Pro/ENGINEER window.
- Give the parameter a new name.
- Add the parameter to the pipeline.
- Save the parameter to a file.
- Read in the parameter as a .pp1 file.
- Delete the parameter from the pipeline.
- Cancel changes made to the pipeline.
- Modify an existing pipeline label.

## About Bend Notes

Bend notes are similar to other notes. You can resize them, orient them horizontally, vertically, or angularly, change the font, move them, and so on. By default, the system places bend notes horizontally with a leader attachment. When you modify a pipe bend, the system edits the text in the note, not the values themselves.

In Drawing mode, you can automatically display information for routed pipeline segments with bend type corners (for example, bend user-defined parameters, spool parameters, and the solid name of the segment to which the bend belongs).

You can automatically display this information by specifying a default note to attach to each pipe bend type corner and to each end of the segment.

## To Create Piping Bend Notes

1. In Drawing mode, when you attach a note to a pipe bend, add the extension `:att_pipe_bend` to the parameter name. Doing this indicates that the parameter value should refer to the bend to which the note is attached (for example, `"&bend_radius:att_pipe_bend, &material:att_pipe_bend`).
2. When you enter a parameter using this format, the system initially searches the pipe bend parameters for that bend, then line stock parameters for that segment, pipeline parameters for that pipeline, and pipe solid parameters (if there is an existing solid), in that order. If the system cannot find a value, it maintains the text.

## Labeling Piping Bends Automatically

In Drawing mode, the system determines the default pipe bend note and determines whether to place pipe bend notes at all, according to the setting of the drawing setup file option `default_pipe_bend_note`.

- If you set the option to no (the default), the system does not show bend notes in the drawing.
- If you set the value as text within quotation marks, the system uses that value when creating the bend notes. For single line notes, the text can include bend parameters `&bendname:att_pipe_bend`, `"&bend_radius:att_pipe_bend"` and `"&bend_num:att_pipe_bend."` The text can also include all user-defined bend parameters, all default linestock parameters and all user-defined linestock parameters.
- If you specify a directory path as the value, the system references a previously created note saved as a file to create single or multiple line bend notes, for example, in UNIX:  
`". /mybendnote.txt."`

As there are no defined bends in a flexible pipe, this functionality applies only to straight pipes with bent corners and to the ends of such pipes.

### Tip: Generating Bend Information

You can generate bend location and bend machine information by using **Segment** under **Definition** in the **Report Pipeline** dialog box or setting up report information in the assembly in which the pipe segment was created.

## Bend Tables

### About Bend Tables (Non Spec-Driven)

Pro/PIPING uses bend tables to accurately calculate the length of straight pipe (developed length) required to make a bend of a specific bend radius and angle.

The developed pipe length for the same bend can differ due to different materials and pipe outside diameters (OD).

Bend tables are normalized for 90 degree bends. For other than 90 degree bends, Pro/ENGINEER takes those values and multiplies them by  $A/90$ , where A represents the specific bend angle in degrees.

### Bend Table Menu

The Bend Table menu allows you to access all Pro/PIPING bend tables for creation and modification.

From the **BEND TAB** menu (**PIPING > Set Up > Bend Allow**), you can perform the following tasks:

- **Define**—Create Pro/PIPING bend tables.
- **Delete**—Delete Pro/PIPING bend tables.
- **Edit**—Modify Pro/PIPING bend tables. You can update a bend table at any time.
- **Show**—View Pro/PIPING bend tables. You can confirm bend table data.
- **Read**—Read a bend table from a file to a part.
- **Write**—Write a bend table from a part to a file.

### Defining Bend Tables (Non Spec-Driven)

Pro/PIPING allows you to define new bend tables. You define bend tables for 90 degree bends. For other than 90 degree bends, Pro/ENGINEER takes those values and multiplies them by  $A/90$  where A represents the specific bend angle in degrees.

You do not have to insert a bend allowance value (A) in every cell in a bend allowance table. If you leave a particular cell blank, the system calculates the length value using interpolation.

The following table describes the Pro/PIPING bend table format:

Column 1	Column 2	Column 3	Column 4	...
FORMULA				
equation				
...				
ENDFORMULA				
!				
CONVERSION				
equation				
...				
START MATERIALS				
MATERIAL				
...				
END MATERIALS				
!				
TABLE				
radius	radius	radius	radius	radius
outside diameter	bend allow	bend allow	bend allow	...
outside diameter	...	...	...	...

**Note:**

- Enter the words FORMULA, END FORMULA, CONVERSION, END CONVERSION, START MATERIALS, END MATERIALS and TABLE exactly as shown.
- The following definitions apply for the above bend table format:
  - **FORMULA**—An equation for the developed length.
  - **TABLE**—A combination of pipe outer diameter (OD) and bend radius table data that will never be encountered in your pipe assembly. For example, 0.25 bend radius with an OD of 0.5 inches.
- Comment lines can appear anywhere in a bend table. Each comment line in the table must start with an exclamation point (!).

**To Define a Bend Table**

1. From the **PIPING** menu, click **Set Up > Bend Allow > Define**.
2. Enter a bend table name and click . A Pro/TABLE window opens containing a table outline.

3. Enter the data in the table or select **File > Read** to use an existing bend table as a baseline. The **Read File** dialog box opens.
4. Enter the name of the existing bend table file and click **Ok**. The system reads the file data and writes it to the current bend table file. Modify the bend table data, if needed.
5. Click **File > Save** and **Exit** to save and close the bend table. You must now assign the new bend table to a pipe segment or line stock to use it.

### To Modify a Bend Table

Pro/PIPING allows you to modify bend tables. You can edit only those bend tables created in the current assembly or applied to the current assembly. To modify a bend table within Pro/PIPING, do the following:

1. From the **PIPING** menu, click **Set Up > Bend Allow > Edit**. The **TBL NAMES** menu appears.
2. Select the bend table to modify from the **TBL NAMES** menu. This menu lists all bend tables that were applied to the assembly or were created when the assembly was currently in session.
3. A Pro/TABLE window opens and displays the selected table.
4. Modify the bend table.
5. Click **File > Save** or **Exit**. Pro/PIPING saves the bend table in the current directory.

**Note:** You must assign a bend table to a pipe segment or line stock to use it.

### To Delete a Bend Table

1. From the **PIPING** menu, click **Set Up > Bend Allow > Delete**.
2. Select the bend table to delete from the **TBL NAMES** menu. This menu lists all bend tables that were applied to the assembly or were created when the assembly was currently in session. Pro/PIPING deletes the selected bend table.

### To Read a Bend Table to a Model

You can read a bend table from a bend table (.bnd) file into a model. Pro/PIPING creates a bend table from the file data and applies it to the current model.

1. From the **PIPING** menu, click **Set Up > Bend Allow > Read**. The **DATA FILES** menu appears.
2. Select a bend table name. Pro/PIPING lists all saved bend tables. If you select **Names**, the **Open** dialog box opens in which you select a bend table (.bnd) file to read. The Pro/ENGINEER message area confirms the action.

## To Assign a Bend Table

A Bend Table is assigned to each segment through the Line Stock.

1. Click **PIPING > Set Up > Line Stock > Edit**. The **Line Stock Names** menu appears.
2. Click a line stock name. The **LineStock** dialog box appears.
3. Click the **Design Rule** tab. The **Design Rule Parameters** dialog box appears.
4. Select a bend table from the **Bend Table Name** box. The default value is `NO_TABLE`. This box lists all bend tables defined for the pipe segment or line stock.
5. Click  to apply the bend table. The **Design Rule Parameters** dialog box closes.
6. Click  to close the **Design Rules** dialog box.

## Formula Equations (Non Spec-Driven)

Formula is an equation used to calculate the developed length for any radius or outside diameter (OD) that falls outside the range of the table data. Pro/PIPING provides the following variables for use in the table:

- `L`—Developed straight length of the bend.
- `R`—Bend radius.
- `T`—Pipe outer diameter (OD).
- `ANGLE`—Bend Angle (in degrees).

## Formula Equation Rules

Follow these rules when you enter the formula equation in a bend table:

- The formula must be the first equation in the table.
- The formula must begin with the descriptor `FORMULA`, in the first column.
  - If an equation is simple, write it in the second column on the same line as the descriptor, as in the following examples:  
$$\text{FORMULA } L = (0.55 * T) + (PI * R) / 2.0$$
$$\text{FORMULA } L = (ANGLE * PI / 180) * (R + T / 2)$$
  - If the formula is more complex and/or contains some logic statements, then the `FORMULA` descriptor must precede the actual formula on its own line. The formula must also be concluded by the descriptor `END FORMULA`, on its own line and in the first column. The following example illustrates this process:

```

FORMULA
IF (R/T) < 1.10 | (R/T) > 1.83
L = (PI*R*T)/2.0
ELSE
L = (PI/2)*(R + (T*0.35))
END FORMULA

```

In a multi-statement formula like this, you can enter the statements in columns other than column one.

### Conversion Equations (Non Spec-Driven)

The conversion equation tells the system how table data ( $A$ ) is related to the developed length ( $L$ ). If no conversion equation is defined, the system assumes that the table data equals the required length ( $L = A$ ). For example, if the table data represents the bend deduction value, the equation might look like the following:

$$L = 2*(T + R) - A$$

**Note:**  $L$  must never be negative.

### Conversion Equation Variables

You can use the following variables in a conversion equation:

- $L$ —Developed straight length of stock.
- $R$ —Bend radius.
- $T$ —Material thickness.
- $ANGLE$ —Bend Angle (in degrees).
- $A$ —Bend allowance value contained in the table data section.

### Conversion Equation Rules

Follow these rules when you enter the conversion equation in a bend table:

- Only use a conversion equation when the developed length  $L$  is not equal to your table values  $A$ .
- The conversion must begin with the descriptor `CONVERSION`, in the first column.
  - If the equation is a simple one, then it can be written in the second column on the same line as the descriptor, as in this example:

```
CONVERSION L = 2*(T +R) - A
```

- If the formula is more complex and/or contains some logic statements, then the `CONVERSION` descriptor must precede the actual formula on its own line

and the formula must be concluded by the descriptor `END CONVERSION`, again on its own line and in the first column.

## Design Rule Parameters

### About Design Rule Parameters

In Pro/PIPING, Design Rule parameters are line stock parameters that check selected pipelines to ensure that the pipes stay within their limitations. If there are any violations, this is considered a Line Stock violation. The system compares the characteristics of a selected pipeline with the values of the Design Rule parameters. If there are any violations, the pipeline highlights, which gives you the opportunity to correct the pipeline parameters.

### To Define Design Rule Parameters (Non Spec-Driven)

You use design rule parameters to verify whether the parameters of a routed pipe are within limitations. Design Rule Parameters are available in the **LineStock** dialog box.

1. Open the **LineStock** dialog box (**Piping > Set Up > Line Stock > Create**) and go to the **DesignRule** tab.
2. The following list of the Design Rule parameter values appears.
  - **Max Overall Length**—Maximum allowable overall length of a continuous pipe segment.
  - **Min Segment Length**—Minimum allowable length of a continuous pipe segment.
  - **Max Segment Length**—Maximum allowable length of a continuous pipe segment.
  - **Max Bends Number**—Maximum allowable number of bends in a continuous pipe segment.
  - **Min Bend Separation**—Minimum allowable distance, in assembly units, between bends in one continuous pipe segment.
  - **Min Straight Length**—Minimum allowable length of a straight pipe segment.
  - **Min Bend Radius**—Minimum allowable bend radius, in assembly units, for a pipe segment.
  - **Max Bend Radius**—Maximum allowable bend radius, in assembly units, for a pipe segment.
  - **Min Bend Angle**—Minimum angle of the bend, in assembly units, for a continuous pipe segment.
  - **Max Bend Angle**—Maximum angle of the bend, in assembly units, for a continuous pipe segment.

- **Bend Arc Threshold**— Angle at which Pro/PIPING defines the dimensioning scheme for large angle bends, such as 180 degrees (Default value in degrees: 165. You can enter a value between 0 and 360.)
  - **Min Branch Separation**—Minimum separation distance between two branches in a continuous pipe segment.
  - **Min Branch Angle**—Minimum allowable angle between a branch and the main.
  - **Bend Table Name**—Assign a bend table to the line stock. You can create a bend table at any time (Default value: NO\_TABLE).
3. Click in any of the boxes and change the values, if required .
  4. Click  to apply the design rule parameters and close the **LineStock** dialog box.
  5. Click  to close the **LineStock** dialog box.

**Note:** All bend radius and bend angle boxes are unavailable. Pro/PIPING retrieves this bend data from the Specification Database Bend Master Catalog (MCAT) file.

### To Check for Line Stock Violations

1. Click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Click **Check Pipe** in the **Info Type** box.
3. Click  to select a pipe line.
4. The dialog box displays violations in Line Stock parameter.

The system highlights the pipeline if the violation applies to the entire pipeline. If it only applies to certain geometry of the pipe (such as a bend), it highlights that geometry in a different colour.

You can also click **Info**, and an INFORMATION WINDOW opens. This window displays the Line Stock name, the parameter name, the value of the parameter, and the value of the violation (if there is one). You can save, edit, and search in this information window.

### About the Bend Arc Threshold

The **Bend Arc Threshold** is a **Design Rule** parameter that appears in the **LineStock** dialog box. It is the angle at which Pro/PIPING defines the dimensioning scheme for large angle bends such as 180 degrees. The dimensioning scheme for the bend refers to how the bend is located along the pipe. Pro/PIPING can locate the bend by either the Theoretical Intersection Point or the Bend Arc Center Point.

- Bend angles that are larger than or equal to 180 degrees get dimensioned based on the Bend Arc Center Point.

- Bend angles that are smaller than 180 degrees get dimensioned based on the Theoretical Intersection Point of the bend.

For a 180 degree bend, one bend value is reported for the location of the center of the arc of the 180 degree bend.

### To Set the Bend Arc Threshold

1. Open the **LineStock** dialog box and go to the **DesignRule** tab.
2. The default value of **Bend Arc Threshold** is 165 degrees. Modify this parameter as desired between 0 and 360.

### Example: Bend Arc Threshold

IF

BEND\_ARC\_THRESHOLD = 165 (default)

THEN

Route a 180-degree bend—Gets reported as one 180-degree bend from the arc center of the bend.

IF

BEND\_ARC\_THRESHOLD = 180

THEN

Route a 180-degree bend—Gets reported as one 180-degree bend from the arc center of the bend.

IF

BEND\_ARC\_THRESHOLD = 181

THEN

Route a 180-degree bend—Gets reported as two 90-degree bends from the theoretical intersection points.

### Design Rules Checking

#### About Design Rules Checking

You can enable or disable the design rule check during the pipe modeling process. When you enable the design rule check, the location of violations can be displayed and you can rectify them at an early stage. The modeling process then proceeds with greater accuracy.

Use the design rule checking functionality to:

- Select the design rules to check whenever violations can occur due to a specific pipe modeling operation. You can select all design rules or only a few design rules.
- Indicate the location of a design rule violation.

**Note:** To enable the checking of design rules for piping assemblies that are created using versions of Pro/ENGINEER earlier than Wildfire, set the `piping_wildfire_covert` configuration option to `automatic` or `manual`.

## To Check for Design Rule Violations (Non Spec-Driven)

1. Click . The **Design Rules Check** dialog box opens.

**Note:** The **Design Rules Check** dialog box opens only when:

- An active assembly contains a pipeline for which you can check design rules.
  - The `pipe_allow_design_check` configuration option is set to `on` if you are working with a pipeline assembly that has been created using versions of Pro/ENGINEER earlier than Wildfire.
2. Under **Setup**, click one or more of the following design rules to check:
    - **Missing Fitting**—Checks for missing fittings in the assembly. Under **Design Rules Tree**, click a missing fitting violation type to display the violation information under **Violation Details**.
    - **Segment Violations**—Checks for violations in segment length. Under **Design Rules Tree**, click a segment violation type to display the violation information under **Violation Details**.
    - **Bend Violations**—Displays the bend violation results for a pipeline. Under **Design Rules Tree**, click a bend violation type to display the violation information under **Violation Details**.
    - **Branch Violations**—Displays the branch violations in a pipeline. Under **Design Rules Tree**, click a branch violation type to display the information under **Violation Details**.
    - **Flow Violations**—Checks for conflicts in the flow direction. Under **Design Rules Tree**, click **Flow Violations** to display the name of the datum point feature at the location of flow conflict under **Violation Details**.

The number of violations are displayed in parentheses next to the appropriate tree node under **Design Rules Tree**.

**Note:** If you do not want to check for any of the previous design rule violations, clear the appropriate design rule from the **Setup** list.

3. If required, click **Tree** and select one of the following:
  - **Expand All**—Expands all the tree nodes.

- **Expand One**—Expands only the selected tree node and not its component child nodes.
- **Expand Branch**—Expands the selected tree node and all its component child nodes.
- **Collapse All**—Collapses all the tree nodes.
- **Collapse One**—Collapses only the selected tree node or a child node of a tree node.
- **Collapse Branch**—Collapses the selected tree node and all its component child nodes.

**Note:** A tree node and its child nodes represent design rules in all the above cases.

4. Click  or  to select a record under **Violation Details**.
5. Click  to display the design rule violation information in the INFORMATION WINDOW.

### Displaying Missing Fitting Violations

Missed fitting types are displayed under **Design Rules Tree** in the **Design Rules Check** dialog box as **Corner Fitting**. Violation results for missing corner fittings are displayed under **Violation Details**.

### Displaying Segment Violations

Segment violation types are displayed under **Design Rules Tree** in the **Design Rules Check** dialog box as follows:

- **Min Segment Length**—Displays violation results if you create a pipe segment with a length shorter than the minimum segment length.
- **Max Segment Length**—Displays violation results if you create a pipe segment with a length greater than the maximum segment length.

**Note:** The violation results are displayed in the following columns under **Violation Details** when you click **Min Segment Length** or **Max Segment Length**:

- **Line Stock**—The name of the line stock.
- **Current**—The current segment length.
- **Required**—The required minimum or maximum segment length.
- **Total Segment Length**—Displays violation results if you create pipe segments of a pipe in the active assembly where the pipe segments have length greater than the total pipe length. The violation results are displayed in the following columns under **Violation Details**:

- **Line Stock**—The name of the line stock.
- **Current**—The current segment length.
- **Required**—The required total segment length.

### Displaying Bend Violations

Bend violation types are displayed under **Design Rules Tree** in the **Design Rules Check** dialog box as follows:

- **Max Bend Numbers**—Displays violation results if you create bends on a pipe segment greater than the specified maximum bend number value.
- **Start Clamp Length**—Displays violation results if you create a pipe segment with its start clamp length less than the specified value.
- **Middle Clamp Length**—Displays violation results if you create a pipe segment with its middle clamp length less than the specified value.
- **End Clamp Length**—Displays violation results if you create a pipe segment with its end clamp length less than the specified value.
- **Min Bend Radius**—Displays violation results if you create a pipe bend with its radius less than the specified value.
- **Max Bend Radius**—Displays violation results if you create a pipe bend with its radius greater than the specified value.
- **Min Bend Angle**—Displays violation results if you create a pipe bend with its angle less than the specified value.
- **Max Bend Angle**—Displays violation results if you create a pipe bend with its angle greater than the specified value.

In all the previous cases, the violation results are displayed in the following columns under **Violation Details**:

- **Line Stock**—The name of the line stock.
- **Current**—The current bend number, start clamp length, middle clamp length, end clamp length, bend radius, or the bend angle.
- **Required**—The required bend number, start clamp length, middle clamp length, end clamp length, minimum bend radius, maximum bend radius, minimum bend angle, or the maximum bend angle.

### Displaying Branch Violations

Branch violations are displayed under **Design Rules Tree** in the **Design Rules Check** dialog box as follows:

- **Min Branch Separation**—Displays violation results if you create branches on a pipe segment with the distance between the branches less than the specified minimum branch separation.

- **Min Branch Angle**—Displays violation results if you create a branch with the angle between the branch and the main pipeline less than the specified minimum branch angle value.

In the previous cases, the violation results are displayed in the following columns under **Violation Details**.

- **Line Stock**—The name of the line stock.
- **Current**—The current branch separation or the branch angle.
- **Required**—The required minimum branch separation or the minimum branch angle.

### Checking Design Rules

Pro/PIPING determines whether any related design-rule check has been enabled at the end of each of the pipe modeling operations as discussed in the following table. Use the **Define Design Rules** dialog box to change the default values specified in the table below.

<b>Design Rule</b>	<b>Where to Define in the SpecDB</b>	<b>Where to Store in the Model</b>	<b>Check After</b>
Missing Fitting			
Corner fitting			Route Pipe, Modify Corner Type, Delete Fitting
Segment Violations			
Minimum segment length	MIN_LEN column of the Pipe Manufacture Directory file	MIN_SEGMENT_LENGTH line stock parameter	Route Pipe, Modify Dimension, Modify Pipe, Modify Corner Type, Insert Fitting, Replace Fitting, Redefine Fitting, Cut Pipe
Maximum segment length	MAX_LEN column of the Pipe Manufacture Directory file	MAX_SEGMENT_LENGTH line stock parameter	Route Pipe, Modify Dimension, Modify Pipe, Modify Corner Type, Insert Fitting, Delete Fitting, Replace Fitting, Redefine Fitting, Delete Cut

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Total segment length	MAX_OVERALL_LENGTH column of the Pipe Manufacture Directory file	MAX_OVERALL_LENGTH line stock parameter	Route Pipe, Modify Dimension, Modify Pipe, Modify Corner Type, Insert Fitting, Delete Fitting, Replace Fitting, Redefine Fitting, Delete Cut
Bend violations			
Maximum bend numbers	MAX_BENDS column of the Pipe Manufacture Directory file	MAX_NUMBER_BENDS line stock parameter	Route Pipe, Modify Corner Type, Delete Fitting, Delete Cut
Start clamp length	START_CLAMP_LENGTH column of the Bending Machine file	The first element of the MIN_STRAIGHT_LENGTH_END line stock parameter	Route Pipe, Modify Dimension, Modify Pipe, Modify Corner Type, Insert Fitting, Replace Fitting, Redefine Fitting, Cut Pipe
Middle clamp length	MID_CLAMP_LENGTH column of the Bending Machine file	MIN_BEND_SEPARATION line stock parameter	Route Pipe, Modify Dimension, Modify Pipe, Modify Corner Type
End clamp length	END_CLAMP_LENGTH column of the Bending Machine file	The second element of the MIN_STRAIGHT_LENGTH_END line stock parameter	Route Pipe, Modify Dimension, Modify Pipe, Modify Corner Type, Insert Fitting, Replace Fitting, Redefine Fitting, Cut Pipe
Minimum bend radius	MIN_RADIUS column of the Bend file	MIN_BEND_RADIUS line stock parameter	Modify Corner Type, Update Bend Parameters
Maximum bend radius	MAX_RADIUS column of the Bend file	MAX_BEND_RADIUS line stock parameter	Modify Corner Type, Update Bend Parameters
Minimum bend angle	MIN_ANGLE column of the Bend file	MIN_BEND_ANGLE line stock parameter	Route Pipe, Modify Dimension, Modify Corner Type, Update Bend Parameters

Maximum bend angle	MAX_ANGLE column of the Bend file	MAX_BEND_ANGLE line stock parameter	Route Pipe, Modify Dimension, Modify Corner Type, Update Bend Parameters
Branch violations			
Minimum branch separation		MIN_BRANCH_SEPARATION line stock parameter  The default value is 50 mm or its equivalent in the pipeline assembly length unit.	Route Pipe Branch To, Route Pipe Branch From, Modify Dimension
Minimum branch angle		MIN_BRANCH_ANGLE line stock parameter  The default value is 30 degrees.	Route Pipe Branch To, Route Pipe Branch From, Modify Dimension
Flow conflict			Route Pipe, Modify Pipeline, Extension or Series Flow Direction

## Non Specification-Driven Pipe Fabrication

### About Fabricating Pipes

The Non Specification-Driven method of modeling includes the following Pipe Fabrication functionality:

- Pipe piece cutting functionality that allows you to insert or delete a cut.
- Create a solid representation for piping insulation and store the information in the model.
- Create a pipe solid for a pipe segment or the entire pipeline in the current active assembly.
- Extract or delete a model.

## Cutting Pipe Pieces

### About Cutting Pipe Pieces

Use **Cut Pipe** to split a pipeline into pieces that can be manufactured. **Cut Pipe** allows you to alter the length of pipelines by splitting a straight segment into shorter raw pipe blanks, or cutting the pipe at a specific location to generate the desired spool.

The piece cutting functionality allows you to perform the following tasks:

- Cut a pipeline at a specified location.
- Cut a straight pipeline segment at multiple locations by specifying the stock length or using a user-specified length of pipe.
- Specify a cut location by referencing other objects such as an adjacent pipe segment, or a data reference plane.
- Specify a cut location by an offset distance.
- Cut all parallel pipe segments at locations identical to the reference pipe segment.
- Assign shop or field weld attributes.

### To Cut Pipe at a Single Location Using Placement

Use **Cut Pipe** to cut a pipeline at a specified location using distance or ratio placement.

1. Click **Applications > Piping > Fabrication > Cut Pipe**. The **Cut Pipe** dialog box opens.



2. Under **Piece Cutting Options**, select .
3. Select the pipe segment in which you want to insert a cut. By default, the selected location is the placement location of the cut. The pipeline label is displayed under **Select Pipe Segment** as read only information. The distance of the selected location on the pipe segment is displayed in the thumb wheel input panel.
4. Under **Cut Placement Options**, click one of the following:



- —Distance Placement. Use the thumb wheel to reposition the cut location along the pipe segment or specify the distance in the input panel.

**Note:** By default, Pro/PIPING considers the reference location from the start end of the pipe segment. Click  under **Location Parameter** to flip the location with reference to the end of the pipe segment. Pro/PIPING changes  to  to indicate the change in the reference location.

- —Ratio Placement. Specify a distance ratio from the upstream end of the specified pipe segment.

**Note:** By default, Pro/PIPING considers the reference location from the start end of the pipe segment. Click  under **Location Parameter** to flip the location with reference to the end of the pipe segment. Pro/PIPING changes  to  to indicate the change in the reference location.

5. To assign a weld attribute to this cut, under **Weld Type**, select **Field** or **Shop**.

6. Do one of the following:

- Click  to insert a cut at the defined location, generate the specified weld attribute, and close the **Cut Pipe** dialog box.
- Click  to insert a cut at the defined location, generate the specified weld attribute, and continue inserting cuts with the **Cut Pipe** dialog box.
- Click  to cancel the action and close the dialog box.

**Note:** You can control the display format of the linear dimension and the incremental step value for the thumb wheel movement using the `pipng_thumb_wheel_increment` configuration option.

### To Cut Pipe at a Single Location Using a Plane Reference

Use the following procedure to insert a cut at a defined distance from a specified plane and generate a weld attribute.

1. Click **Applications > Piping > Fabrication > Cut Pipe**. The **Cut Pipe** dialog box opens.

2. Under **Piece Cutting Options**,  is selected by default. If it is not, select it.
3. Select an approximate location on the pipe segment in which you want to insert a cut. The pipeline label is displayed under **Select Pipe Segment** as read-only information.

4. Under **Cut Placement Options**, click .
5. Select a datum plane that is perpendicular to the pipe segment. The name of the reference is displayed under **Location Parameter**.
6. Use the thumb wheel to position the cut location along the pipe segment or specify the distance in the input panel.
7. Under **Weld Type**, the **Field** option is selected by default. If it is not, select it to assign a field weld attribute to this cut.

8. To insert the cut, do one of the following:

- Click  to insert a cut at the defined location, generate the specified weld attribute, and close the **Cut Pipe** dialog box.
- Click  to insert a cut at the defined location, generate the specified weld attribute, and continue inserting cuts with the **Cut Pipe** dialog box.

### To Cut Pipe at a Single Location Using a Pipe Reference

In the following procedure, you place a cut at the specified location on a segment and then create a reference from an adjacent upstream pipe segment that is perpendicular to the segment where the cut is defined.

1. Click **Applications > Piping > Fabrication > Cut Pipe**. The **Cut Pipe** dialog box opens.



2. Under **Piece Cutting Options**,  is selected by default. If it is not, select it.
3. Select an approximate location on the pipe segment in which you want to insert a cut. The pipeline label is displayed under **Select Pipe Segment** as read-only information.



4. Under **Cut Placement Options**, select .
5. Use the thumb wheel to position the cut location along the pipe segment or specify the distance in the input panel.
6. Under **Weld Type**, the **Weld** option is selected by default. If it is not, select it to assign a field weld attribute to this cut.
7. To insert the cut, do one of the following:

- Click  to insert a cut at the defined location, generate the specified weld attribute, and close the **Cut Pipe** dialog box.
- Click  to insert a cut at the defined location, generate the specified weld attribute, and continue inserting cuts with the **Cut Pipe** dialog box.

### To Cut Pipe at Multiple Locations Based on Pipe Stock or User Specified Length

Use this procedure to cut a straight pipeline segment automatically at multiple locations by specifying the stock length or based on a user-specified length of pipe. When you cut at multiple locations, the specified length is applied at intervals along the straight pipe segment.

After you create multiple cuts in this manner, they are treated as individual cuts. You can manipulate them and delete them individually.

1. Click **Applications > Piping > Fabrication > Cut Pipe**. The **Cut Pipe** dialog box opens.



2. Under **Piece Cutting Options** select .
3. Select a straight pipe segment in which you want to insert multiple cuts. The pipeline label is displayed under **Select Pipe Segment** as read-only information.
4. Under **Interval Length**, specify the desired **Pipe Stock** or **Length** to cut the pipe segment at multiple locations.
5. Under **Weld Type**, the **Field** option is selected by default. If it is not, select it to assign a field weld attribute to this cut. Field Welds with unique identification numbers will be created at each cut location.
6. To insert the cut, do one of the following:

- Click  to insert cuts at the defined location, generate the specified weld attribute, and close the **Cut Pipe** dialog box.
- Click  to insert cuts at the defined location, generate the specified weld attribute, and continue inserting cuts with the **Cut Pipe** dialog box.

### To Cut Parallel Pipes at Locations Identical to the Reference Pipe

Use the following procedure to cut a group of parallel pipe segments in a single operation. All cuts on the selected parallel pipelines are created at locations that are identical to the locations of cuts on the reference pipeline.

1. Click **Applications > Piping > Fabrication > Cut Pipe**. The **Cut Pipe** dialog box opens.



2. Click  to cut parallel pipes at locations identical to locations on the reference pipe.
3. Select a pipe segment that is part of a pipe run and has one or many pipe cuts. The pipeline label is displayed under **Select Reference Pipe Run** as read-only information.

**Note:** Pro/PIPING does not allow selection of pipe segment of a pipe run that has no pipe cuts on it.

4. Select the pipe segments parallel to the reference pipe run. The pipeline label of the selected pipe segments is displayed under **Select Target Pipe Run** as read-only information.

**Note:** Pro/PIPING does not allow selection of pipe segments that are not parallel to the reference pipe run.

5. To insert cuts on parallel pipe runs at locations identical to locations on the reference pipe run, do one of the following:

- Click  to insert cuts on the parallel pipe runs and close the **Cut Pipe** dialog box.
- Click  to insert cuts on the parallel pipe runs and then continue inserting cuts with the **Cut Pipe** dialog box.

### Tip: Selecting a Reference Pipe for Cutting Parallel Pipes

Cuts on the target pipe runs are created using the placement type information associated with the corresponding cut on the reference pipe run.

To create cuts on the target pipe runs at identical locations identical to locations on the reference pipe run:

- Select only those pipe runs that are parallel to the reference pipe run as the target pipe runs.
- Select the pipe run with multiple cuts as the reference pipe run. All these cuts on the reference pipe run must be created using the Length, Ratio Placement, or Distance Placement options. If the cuts on the reference pipe run are created using any other placement type option, Pro/PIPING creates cuts on the target pipe runs using the default Length placement type option.

### Assigning a Weld Type

You can assign a shop weld attribute or field weld attribute at pipe piece cut locations.

The Piping Isometric Drawing functionality uses both the shop weld and field weld information to mark the shop or field welds on isometric drawings.

The shop or field welds are graphically represented and have a unique identification number which is automatically generated. For example, shop weld numbers are SW1, SW2, and so on.

At any time during modeling, you can use the standard datum point creation

tool, , on the Pro/ENGINEER toolbar to display or hide the shop or field welds.

When cutting a set of parallel pipe runs at a location identical to the location on the reference pipe run, the cut on the target pipe runs is created with the same weld type as that of the corresponding cut on the reference pipe run.

### To Delete a Pipe Piece Cut

Use this procedure to delete a single pipe cut piece or to delete all cuts on a single segment, overall pipe run, entire pipeline, or in a block.

1. Click **PIPING** > **Fabrication** > **Delete Cut**. The **Delete Cut** dialog box opens.
2. Based on the location of the cuts to be deleted, do one of the following:
  - Select **Selected Cut** to delete only the selected cut. The name of the selected cut location is displayed under **Select Pipe Cut**.

- Select **All Cuts in Pipe Segment** to delete the cuts on a pipe segment. The pipeline label of the selected pipe segment and the pipe segments' ID are displayed under **Select Pipe Segment**.
  - Select **All Cuts in Pipe Run** to delete all the cuts existing on an pipe run. The pipeline label of the selected pipe segment and the pipe segments' ID are displayed under **Select Pipe Segment**.
  - The pipe run includes all pipe segments between two consecutive free ends along a straight line, or a free end and a corner, or a corner and a free end, or a corner and another corner of a pipeline.
  - Select **All Cuts at Block Boundary** to delete all the pipe cuts on the boundaries of the selected block. Under **Select Block Name**, select a block name from the list.
  - Select **All Cuts on Pipeline** to delete the cuts in a pipeline. Select any pipe segment on the pipeline whose cuts are to be deleted. The pipeline label of the selected pipe segment is displayed under **Select Pipeline**.
3. To delete the selected cuts, do one of the following:
- Click  to delete all the selected cuts and close the **Delete Cut** dialog box.
  - Click  to delete the selected cuts. The pipeline automatically repaints with the cut deleted and the **Delete Cut** dialog box remains open for further cuts, if any.

## Pipeline Insulation

### About Pipeline Insulation (Non Spec-Driven)

With Pro/PIPING you can assign insulation to a pipeline, part of a pipeline, and fittings. After you assign the insulation, Pro/PIPING creates insulation features that can be accessed in the Pro/ENGINEER Model Tree.

You can perform the following insulation tasks at any time in the piping design process:

- Assign insulation to an entire pipeline during pipeline creation or during piping design.
- Assign insulation to any part of a selected pipeline. Target and insulate specific pipeline areas.
- Modify any insulation assignment in your piping system.
- Delete any insulation assignment in your piping system.

- Create solid representations of insulation for immediate graphical identification and confirmation.
- Generate detailed insulation reports for further insulation analysis.

### To Modify Insulation Assigned to a Pipeline (Non Spec-Driven)

1. Click **PIPING > Modify Pipe**. The **Modify Pipeline** dialog box opens.
2. Under **Active Assembly**, Pro/PIPING displays the current active assembly. If the pipeline to insulate is in a different assembly, click  and select the assembly that contains the pipeline. You can select the assembly from either the Model Tree or the Piping System Tree, or by selecting the pipeline in the graphics window.
3. Under **Modify Options**, click **Parameters** and then under **Pipe Line Parameters** click **Pipe Line**. The **Edit Pipeline** dialog box opens.
4. If active assembly contains multiple pipelines, select the pipeline to insulate from the **Select Pipeline** box.
5. Click the **Label tab**.
6. Under **Pipeline Label**, modify the insulation code for the selected pipeline. Two conditions apply for insulation modification:
  - If an insulation code was assigned to the pipeline, the **Insulation** box displays the code. You can select another insulation code from the list to change the insulation.
  - If an insulation code was not assigned to the pipeline, select the **Insulation** check box and select an insulation code from the list to insulate the pipeline.
7. Click  to save the selections. Pro/PIPING modifies the insulation data in the insulation feature in the active assembly.
8. Retrieve insulation information (**PIPING > Info**) to view the assigned insulation data.
9. Click **PIPING > Regenerate** to regenerate the assembly to update the model display.

**Note:** Pro/PIPING creates insulation quilts automatically after you create pipe solids.

### To Delete Insulation (Non Spec-Driven)

**Note:** The Model Tree view must be set to display features in order to delete insulation features from within it. Click **Settings > Tree Filters**. The **Model Tree Items** dialog box opens. Under **Display**, check the **Features** box and click **OK**.

1. Open the assembly containing the pipeline and the insulation to be deleted.

2. Click . The Model Tree opens.
3. In the Model Tree, locate the assembly containing the insulated pipeline and corresponding the insulation feature. You can delete the following types of insulation features:
  - **Insulation Features**—Stores the insulation data (codes, names, thickness, sizes, and so on) for a pipeline. Pro/PIPING identifies the insulation features by insulation code.
  - **Insulation Segment Features**—Stores the start and end points for insulation on part of a pipe segment. Note that the insulation segment feature is a child of the related insulation feature. If you delete an insulation feature, Pro/PIPING deletes all related children, including the insulation segment feature, if applicable. If you delete the insulation feature from an insulated pipeline that also contains a partially insulated segment, all insulation data will be deleted.
4. To locate the insulation segment feature to delete:
  - a. Select the insulation feature and right-click.
  - b. From the shortcut menu, select **Info > Parent/Child**. The **Reference Information Window** opens. Under **Children of Current Feature**, Pro/PIPING lists all children of the selected insulation feature. Insulation segment features are labeled `Insulation Seg id <number>`.
5. Select the insulation feature to delete from the Model Tree. The feature is highlighted on the pipeline in the graphics window. Right-click, and a shortcut menu appears.
6. Click **Delete**. A warning appears to confirm the action.
7. Click . Pro/PIPING deletes the selected insulation feature from the Model Tree and the graphics window.

### To Report Insulation Information (Non Spec-Driven)

1. Assign insulation to a pipeline or part of a pipeline.
2. Click **PIPING > Info**. The **Report Pipeline** dialog box opens.
3. Under **Info Type**, select **General**.
4. Under **Definition**, select **Insulation**.
5. Click  and select an insulated pipeline or insulated pipe segment from which to retrieve information. You can select from either the Model Tree, the Piping System Tree, or the graphics window.

Pro/PIPING displays the selected pipe data in the box under **Definition** and displays the insulation information under **Results**.

6. Scroll to view the entire report or click  and view the report in the separate INFORMATION WINDOW.

### About Assigning Insulation to Part of a Pipeline (Non Spec-Driven)

Pro/PIPING allows you to assign insulation to part of a pipeline (insulation segment) using the **Create Insulation** dialog box (**PIPING > Fabrication > Insulation**). During piping design, the following special cases can arise and necessitate an insulation segment:

- **Pipe Insulation in a Special Area**—Pipelines and fittings not normally insulated require insulation inside special areas such as purifier rooms.
- **Pipe Insulation near Equipment**—Pipelines can produce condensation and extreme temperatures. Insulate pipelines near equipment to prevent damage.

### To Assign Insulation to Part of a Pipeline (Non Spec-Driven)

1. Click **PIPING > Active Asm** and select the active assembly that contains the pipeline to partially insulate.
2. Click **PIPING > Fabrication > Insulation**. The **Create Insulation** dialog box opens. **Start** under **Insulation Segment Points** is selected by default.
3. Follow these steps to select insulation start and end points. Start and end points must be on the same pipeline:
  - a. Under **Select Start Point**, click  and select a pipe segment to specify the insulation start point. Crosshairs indicate the start point on the pipe segment, and the pipe segment ID populates the **Select Start Point** box.
  - b. Click **End** under **Insulation Segment Points**.
  - c. Under **Select End Point**, click  and select the insulation end point. Crosshairs indicate the end point on the pipe segment, and the pipe segment ID is displayed in the **Select End Point** box.
4. Under **Location Type**, select one of the following options to specify both start and end point placement.

**Note:** You must select the **Start** and **End** options under **Insulation Segment Points** to place the appropriate points.



—Specifies the start or end point distance ratio from the pipe segment's upstream end.



—Specifies the distance ratio of the start or end point from the pipe segment's upstream end.

**Note:** By default, Pro/PIPING considers the reference location from the start end of the pipe segment. Click  under **Location Parameter** to flip the location with reference to the end of the pipe segment. Pro/PIPING changes  to  to indicate the change in the reference location.



—Places a start or end point on the pipe segment perpendicular to that pipe segment.

**Note:** The pipe segment's start point must be perpendicular to the pipe segment's end point to make this option available.



—Places a start or end point from a specified datum plane. The datum plane must be perpendicular to the pipe segment's start or end points. Pro/PIPING creates a reference to the datum plane.



—Places a start or end point from a specified fitting. The selected fitting must exist in the same pipeline and in the same straight pipe piece. Click  under **Location Parameter** to flip the reference port.

5. Under **Location Parameter**, specify the point location parameters based on the selected placement option. Use the thumb wheel to move the point placement or type a value in the thumb wheel box for exact placement.



If you clicked , select a datum plane from the graphics window. Pro/PIPING displays the datum place name in the **Location Parameter** box.

6. Under **Insulation Parameters**, select or enter the insulation name of the insulation assigned between the two specified points.
7. Under **Specify Insulation Parameters**, the following parameters are displayed:
  - **Material**—Displays name of the insulation material
  - **Stock No**—Displays the Line Stock number
  - **Thickness**—Displays the thickness of the insulation
  - **Color Name**—Displays the insulation colour name

You can change the parameters as per your requirements.



8. Click  to save the selections. Pro/PIPING saves the insulation data in an insulation feature in the active assembly.
9. Retrieve insulation information (**PIPING > Info**) to view the assigned insulation data.

**Note:** Pro/PIPING creates insulation quilts automatically after you create pipe solids.

### To Redefine Insulation Assigned to Part of a Pipeline (Non Spec-Driven)

Pro/PIPING creates insulation segment features after you assign insulation to part of a pipeline. You can redefine insulation segment features at any time in the piping design process.

1. Open the assembly associated with the pipeline containing the insulation segment feature to be redefined.
2. Click . The Model Tree opens.
3. In the Model Tree, locate the insulation segment feature to redefine. The Model Tree lists the following insulation features:
  - **Insulation Features**—Stores the insulation data (codes, names, thickness, sizes, and so on) for a pipeline. Pro/PIPING identifies the insulation features by insulation code. Select the feature to highlight it in the graphics window. Note that you cannot redefine insulation features from the Model Tree (you must modify the database insulation files).
  - **Insulation Segment Features**—Stores the start and end points for insulation on part of a pipe segment.
4. Select the insulation segment feature to redefine from the Model Tree. The feature is highlighted on the pipeline.
5. Right-click and a shortcut menu appears.
6. Click **Edit Definition**. The **Redefine Insulation** dialog box opens with the insulation segment selected.
7. Select **Start** under **Insulation Segment Points** and perform any of the following tasks:
  - Change the insulation start point. Under **Select Start Point**, click  and select a pipe segment to specify the insulation start point. Crosshairs indicate the start point on the pipe segment and the pipe segment ID is displayed in the **Select Start Point** box.
  - Change the insulation end point. Select **End** under **Insulation Segment Points**. Under **Select End Point**, click  and select the insulation end point. Crosshairs indicate the end point on the pipe segment and the pipe segment ID is displayed in the **Select End Point** box.
  - Change the start and end point placement. Under **Location Type**, select one of the following options to specify both start and end point placement. Note that you must select **Start** and **End** under **Insulation Segment Points** to place the appropriate points:



—Specifies the start or end point distance from the pipe segment's upstream end.



—Specifies the start or end point distance ratio from the pipe segment's upstream end.



—Places a start or end point on the pipe segment perpendicular to that pipe segment. Note that the pipe segment's start point must be perpendicular to the pipe segment's end point to make this option available.



—Places a start or end point from a specified datum plane. The datum plane must be perpendicular to the start or end point pipe segment. Pro/PIPING creates a reference to the datum plane.



—Places a start or end point from a specified fitting. The selected fitting must exist in the same pipeline and in the same straight pipe piece.

Under **Location Parameter**, specify the point location parameters based on the selected placement option. Use the thumb wheel to move the point placement or enter a value in the thumb wheel box for exact placement.



If you selected , select a datum plane from the graphics window. Pro/PIPING displays the datum place name in the **Location Parameter** box.

- Change the insulation. Under **Insulation Parameters**, select a new insulation code to assign new insulation between the specified placement points.
8. Click  to assign new insulation to the pipe segment. Pro/PIPING creates a new insulation feature and reassigns the insulation segment feature to it.

**Note:**

- The Model Tree must display features to redefine insulation segment features from the Model Tree. Click **View > Model Tree Setup > Item Display**. Under **Display**, check the **Features** box and click **OK**.
- Pro/PIPING creates insulation quilts automatically after you create pipe solids.

## Routing

### About Routing

A physical pipeline consists of pipes and fittings. Before routing a new line, you must create a pipeline feature by assigning its name, line stock, and pipeline parameters.

When creating new routing features, the system takes the default settings for shape and corner type from the current line stock parameter file for the pipeline being routed.

Parameters that are set in a line stock determine routing commands. While routing, you can assign another line stock for segments that start after a fitting or from a new start point.

You can also change line shape and corner type for subsequent segments.

While routing lines or after routing is completed, you may insert fittings or modify pipeline properties according to your design needs. You can route pipelines only within the assembly or subassembly to which they belong.

When routing, keep in mind the following:

- You should create meaningful references to equipment or component geometry to allow pipeline segments to adjust their location whenever a piece of equipment or an assembly component moves.
- You can use any coordinate system as a piping port (which eliminates the prior requirement of names needing to begin with PORT) in the working assembly and define them before assembling the equipment and fittings. The system determines the From/To report information for a pipe segment based on the entry ports to which it is attached at each end:
  - If the coordinate system belongs to the top-level assembly, it lists the top-level assembly.
  - If the coordinate system belongs to the part, it lists the part; if it is not routed to an entry port, it does not show an entry.

### About Routing Pipelines

To route pipelines parametrically, you can reference existing assembly components. Pro/PIPING offers flexibility by enabling you to route lines continuously or to join segments as the design intent develops.

To route a pipeline, you locate the centerline by referencing assembly geometry. You can also use specific fittings and equipment with entry ports as locating points. Entry ports enable you to specify the orientation of contact points in fittings, equipment, and connecting lines.

There is a **Select Any** command for routing reference. You do not have to specify port, pipe end, and so on, before you make a selection. The system figures it out for you and allows you to use **Query Select** if necessary.

Depending on the line shape (straight or flexible), you may employ different routing strategies.

### To Route a Typical Pipeline

1. Set a start point.
2. Create segments using the selected routing action. You may create individual unconnected segments. Indicate to the system a new start point where necessary.

When routing segments continuously, the endpoint that was just created automatically becomes the start point for the next routing operation.

3. Connect individual segments of the same pipeline.
4. Create other pipelines to add branches to the main pipeline; then create branches.

### Using the Route Pipe Menu

The following commands are available on the **ROUTE PIPE** menu to route and manipulate line segments:

- **Pipe Envrnmt**—Set the default shape and corner type.
- **Set Start**—Set a start point or create a branch start point.
- **Extend**—Create a new pipe segment by extending in the specified direction.
- **To Pnt/Port**—Create a pipe piece from the current start point to the selected point, port, or pipe end.
- **Follow**—Create a pipeline by following an axis, curve, existing pipeline, or sketched path.
- **Connect**—Connect entry ports and/or individual unconnected segments of the same pipeline.
- **Branch**—Create a section that taps into a pipeline at the point you have specified.
- **Insert Point**—Insert point in the previously routed pipe segment.
- **Delete Last**—Delete the last routing operation.
- **Redefine**—Redefine references of the routing operation.
- **Suppress**—Suppress the pipe segments.
- **Resume**—Resume the suppressed pipe segment.
- **Modify Dim**—Change dimensions of any piping features or assembly components.
- **Replace**—Replace selected pipeline segment with a routing operation.

These commands are available in both the Non Specification-Driven and Specification-Driven modes.

### **About External References**

When you route pipes, the system automatically creates external references if necessary.

An external reference (or an external dependency) is a relationship between a Pro/ENGINEER object (part, subassembly) and some information from another object that is not inherently available to the referencing objects all of the time.

Pro/ENGINEER always creates an external reference in the context of an assembly, where some information is referenced that is external to the model you are changing.

Whenever you retrieve the subassembly, the system automatically retrieves all of the components that belong to it. Therefore, information needed to place features and components is available.

However, if the subassembly existed among some other components in a higher level assembly, and some of those other components that are members of the higher level assembly—but not the subassembly where the feature is being added—were referenced, the subassembly would have an external reference.

The subassembly would never know exactly where the feature should be placed unless the subassembly was currently in session in the context of the top-level assembly.

For example, if you create a Pipe To Point feature using a datum point in a part that is in a subassembly other than the one with the piping feature, the pipe routing feature now has an external reference.

This feature belongs to the assembly where it was created and every time you regenerate the assembly, Pro/ENGINEER looks for this feature's references in the part in the other subassembly.

All of the information from the assembly is available during this regeneration. However, this dependency of the pipe routing is accurately and completely updated when it is in the context of the assembly containing both.

### **To Manage External References**

The Reference Viewer and Reference Control tools (available with the Design Manager functionality) enable you to manage these external references by doing the following:

1. Define the scope in which to create external references to other models in the context of an assembly. You can restrict any component to reference only the skeleton model of the subassembly to which this component belongs, or only components within that subassembly.
2. Define how the system will behave when you attempt to create an external reference that violates the defined scope.

3. Identify the features in a model that have external references as well as the chain of dependencies from the feature to the referenced entity.
4. Obtain information about those models that have external references to a specified model.
5. To control the settings of these reference control and investigation tools, set the configuration file options `allow_ref_scope_change`, `default_ext_ref_scope`, and `scope_invalid_refs`.

### About Setting a Start Point

Before you can route a pipe segment from one point to another, extend a pipe segment, or branch a segment into a pipeline at a specified point, you must set a start point.

You can set a start point using an existing point, entry port, or pipe end; or you can create a new point. The system displays a start point as a highlighted circle.

### To Set a Start Point Using a Point, Entry Port, or Pipe End

1. Click **Piping > Route**. The system prompts you to select a pipeline.
2. Select a pipeline. The **ROUTE PIPE** menu appears.
3. Click **Set Start**. The **PIPE SELECT** menu appears.
4. Do one of the following:
  - Click **Select Any** to select any appropriate point, port or pipe end.
  - Click **Point/s** and select a datum point.
  - Click **Entry Port** and select an entry port at a fitting or equipment. You can also set start at a created entry port.
  - Click **Branch Point** and create a datum point using **Make Datum**.
  - Click **Pipe End** and select a pipe end.
5. The system informs you that it has defined the start point and highlights it. Proceed to route a line using other routing commands.

### Routing Techniques

With Pro/PIPING, you can route a new line or an existing line by using one of the following techniques:

- Create a pipe segment by routing it from the current start point to a selected point or entry port. You can select multiple points through which to route. If necessary, set a start point first.
- Create a new pipe segment by extending in the specified direction or up to specified geometry. If necessary, set a start point first.

- Connect entry ports or individual unconnected segments of the same pipeline.
- Create a segment that taps into a pipeline at a specified point.
- Create a pipeline by following an axis, cylinder, existing pipeline, or sketched geometry (this functionality is not available for flexible pipes).

### About Branching to Another Pipeline

Using the **Branch** command in the **ROUTE PIPE** menu, you can create a branch in the piping system from one of the following:

- Datum point or entry port to a pipeline
- Pipe end of another pipeline
- Branch point on a pipe centerline to another pipeline

When a branch is attached to a pipeline, the system automatically creates cuts in the solids that meet at the branch point. The point of connection is located on a pipe centerline.

You can create a branch starting from the end of an unconnected segment or from a point or entry port. However, you can not branch to a pipeline in another assembly or to another segment of the same pipeline. (This functionality is not available at all for flexible pipes.)

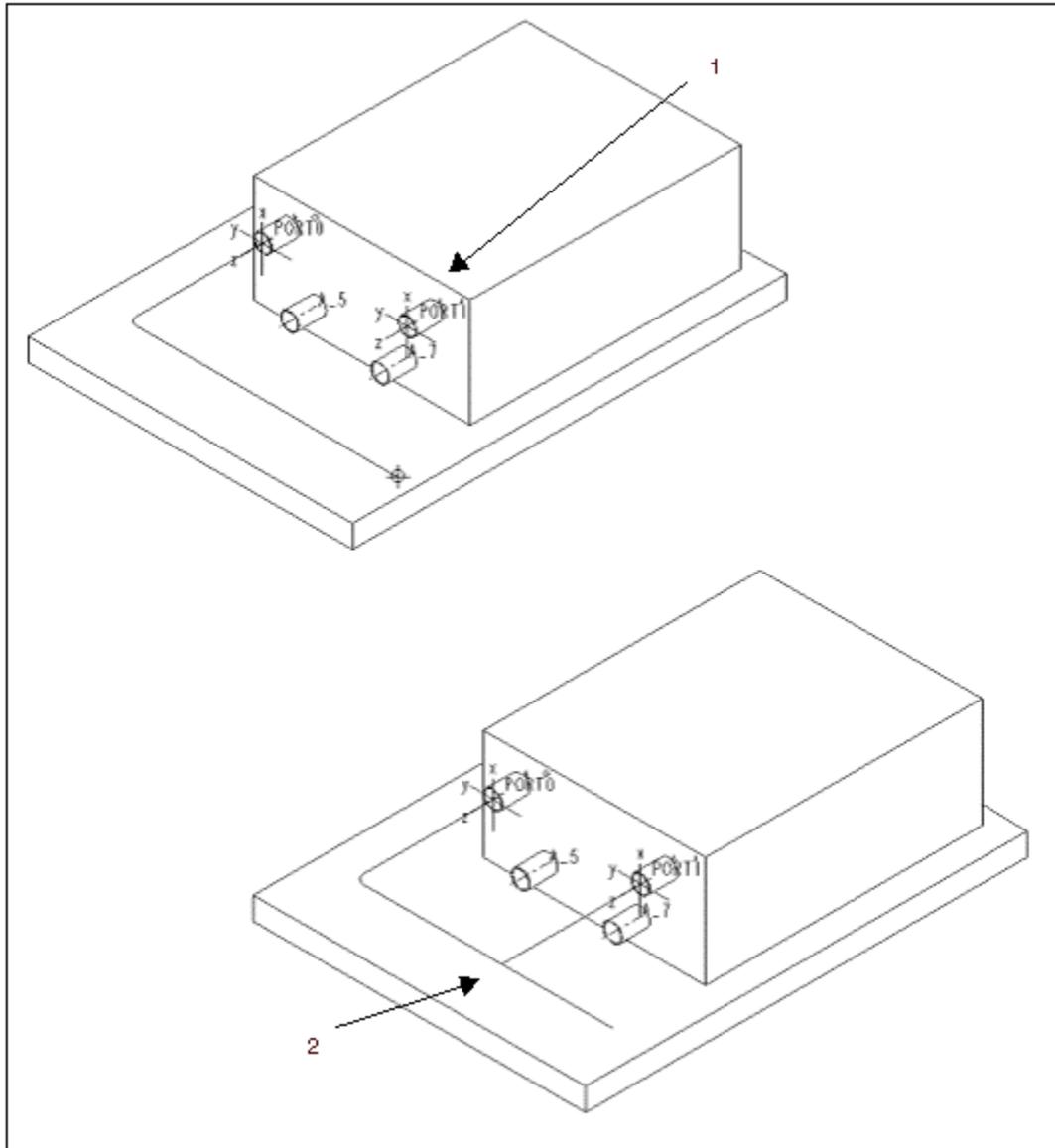
### To Create a Branch

1. Click **Route** > **Set Start** to set a start point; then click **Route** > **Branch**.
2. Using the **BRANCH TO** menu, do one of the following:
  - Click **To Point**. Select a connect point or create one using these commands in the **PNT DIM MODE** menu:
    - Offset**—Measures the distance from a selected reference plane.
    - Length Ratio**—Expresses the distance from the point to the end of the segment nearest the selected point as a ratio of the segment's total length. Represents the resulting fraction in decimal form. The point retains the relative position along the length of the segment corresponding to any changes that you make in its length.
    - Actual Len**—Measures the distance from the nearest end of the segment. The point stays at that fixed distance from the end regardless of any changes you make to the length of the segment.
  - Click **Perpendicular** to create a branch perpendicular to a pipeline.
3. Select a straight portion of a segment in the same assembly and part of a different pipeline.

Fittings cannot be placed at branch points. A straight break tee fitting can be used if desired.

### Example: Creating a Branch

You cannot branch to a pipeline in a different subassembly or to a different segment of the same pipeline.



1 Create a branch using this entry point as a start point

2 Branch has been added perpendicular to a pipeline

### About Connecting Entry Ports or Unconnected Segments

You can connect disconnected segments of the same pipeline or entry ports using the **Connect** command in the **ROUTE PIPE** menu. The system does not place a restriction on the orientation of the segments; it connects them even if they are not coplanar or parallel.

You can create four possible types of straight segments:

- **Collinear Segments**—System extends the selected segments until they meet, forming a single segment. You can not modify this type of connection and the system does not create a dimension.
- **Coplanar Parallel Segments**—System extends the two segments and adds an intermediate segment. You can modify the extension endpoints and the angles of the corners. If the current corner type is **Bend**, the system automatically adds bends. You can control the extension distances at both ends, or the extension distance and angle at one end.
- **Nonplanar Segments**—System extends two segments and adds an intermediate segment. You can modify the extension endpoints and the angles of the corners as for coplanar parallel segments.
- **Coplanar Nonparallel Segments**—System extends each segment until they intersect, creating a single corner. You can move the new endpoints of the connected segments to create an intermediate segment.

### To Connect Pipeline Segments

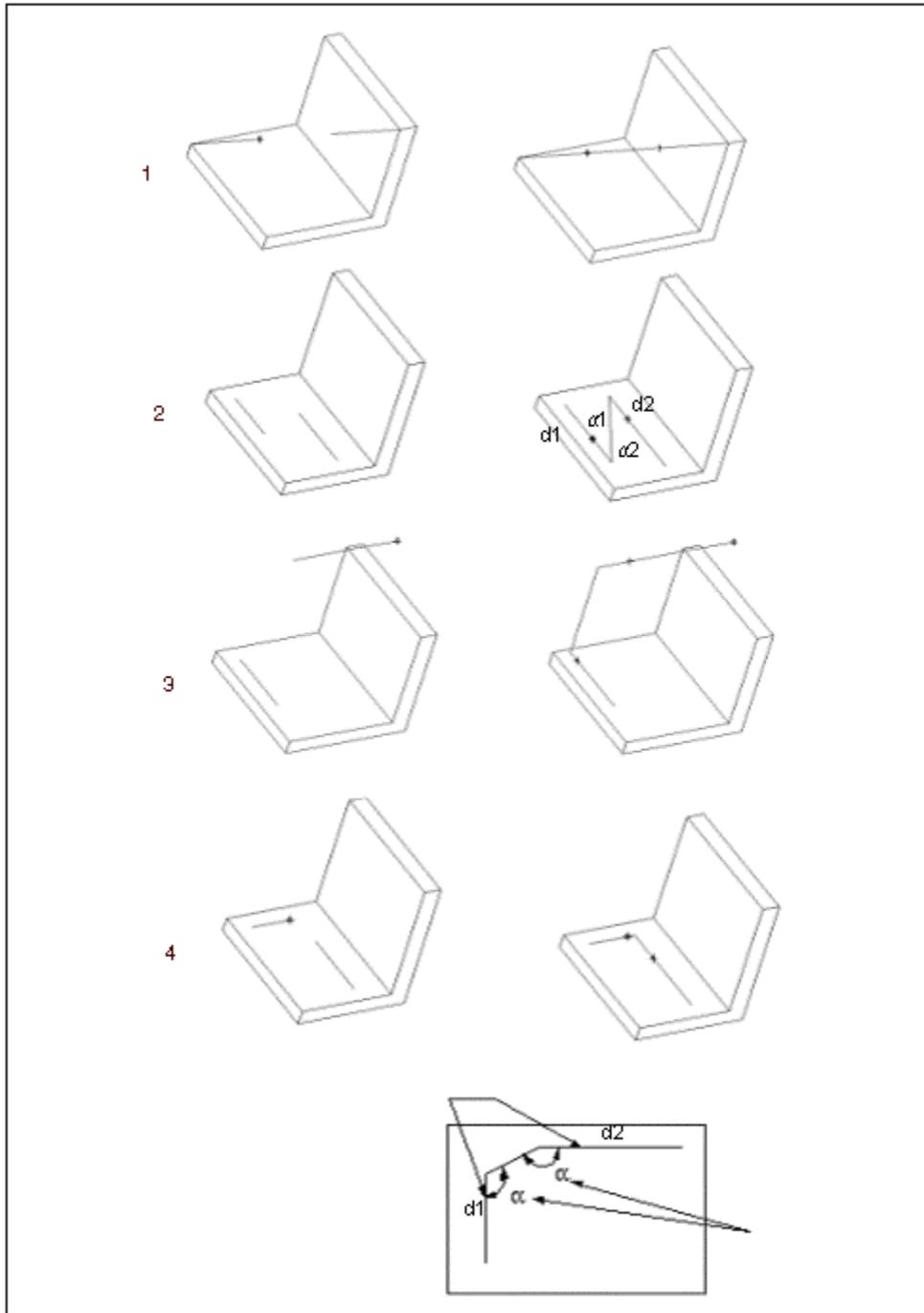
1. Click **ROUTE PIPE > Connect**.
2. For the first end, click **Entry Port** or **Pipe End** from the **PIPE SELECT** menu, and select the connect point.
3. Do the same for the other end. The system connects the two selected ends with line segments as above.
4. Click **CONNECT > Change**.
5. Using the **CHG CONNECT** menu, change the conditions of the connect feature by modifying the settings for the operation:
  - Use **Sel Connect** to reselect connect end points.
  - Use **Set Angle** to set the angle at a corner.
  - Use **Set Ends** to extend connect ends.

Driven dimensions is the default for newly created connect features—the dimensions of connect ends are driven by connected geometry, and the extension value updates when you relocate connected components. However, you can specify a fixed dimension value instead—the extension value does not change when you relocate connected components—and you can toggle between driven and fixed dimensions.

When you modify the extension dimension of a connect feature using **Mod Dim** from the **PIPE MODIFY** menu, the **EXT LENGTH** menu appears and provides two options:

- **Enter Value**—Defines a connect end dimension as fixed and allows you to specify the extension value (as is the case when you use **Set Ends**)
- **Rtrn to Driven**—Defines a connect end dimension as driven by connected geometry and returns the selected dimension to its default driven value

### Example: Straight Segments



1 Collinear

2 Coplanar

3 Noncoplanar

4 Coplanar Nonparallel

You can control  $a_1$ ,  $d_1$ ,  $a_2$ ,  $d_2$ , or  $d_1$ ,  $d_2$ .

## About Routing Flexible Pipes

When routing flexible segments of pipelines, you can alternate line shape between straight and flexible segments on the same pipeline to control the resulting shape.

For example, to make sure a hose exits from a fitting, goes through a hole without interfering, and so on, the physical hose would be all flexible, constrained by the fitting or the hole.

Only the **To Pnt/Port** or **Extend** routing commands are available for flexible pipes; therefore, for routing that models the physical constraint, you may need to make a section straight to use commands such as **Axis** or **Extend**.

When routing flexible pipes, keep in mind the following:

- To route a flexible pipe through a hole, add a straight portion by changing the shape in the **PIPE ENV** menu, or route it by choosing **Axis** from the **PIPE FOLLOW** menu, and switch the line shape of subsequent segments back to flexible.
- You can route flexible pipes from an entry port of a fitting. However, you can not insert fittings on flexible pipes. To ensure that a hose leaves a fitting without interfering, you can use a straight segment with an extend where it leaves the fitting.

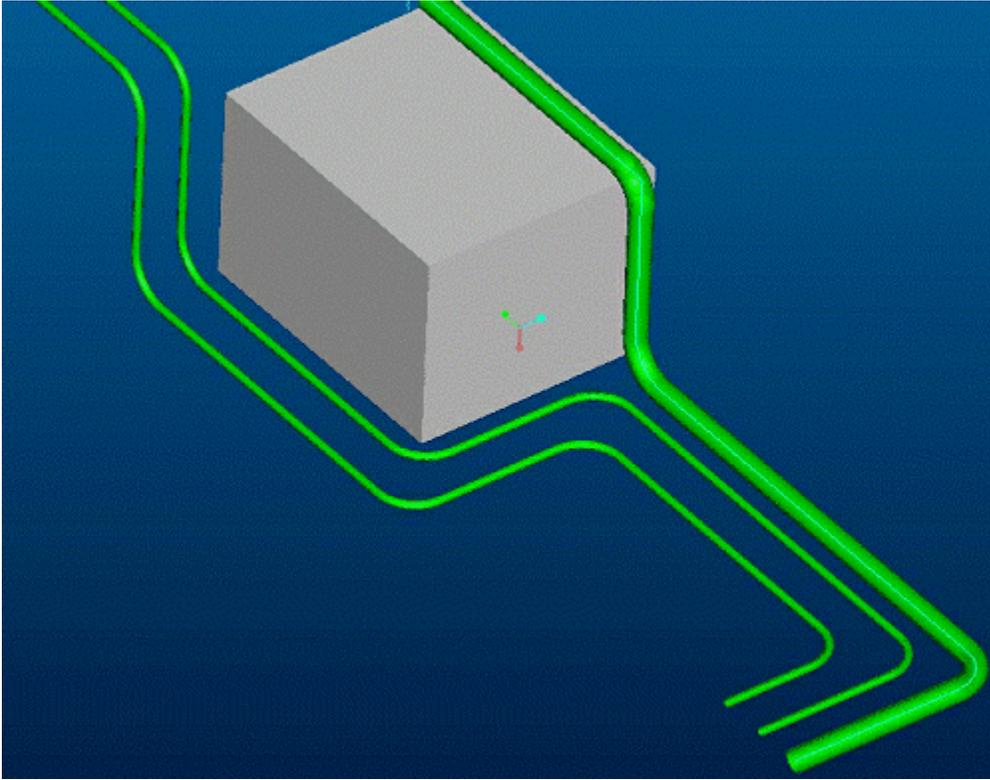
## To Create Pipe Routing Using To Pnt/Port

1. Using the **To Pnt/Port** command, route a pipeline from the currently active pipe end, entry port, or **Set Start** feature to a datum point, entry port, or a pipe end.
2. If there is no currently active start point, you must create one.

## Tip: Routing Pipelines

- You typically route pipelines between pieces of equipment and then insert fittings, but you can just as easily assemble the fittings first and route between them.
- Pipeline routing in the assembly is highlighted.
- You can now have straight routing follow a datum curve.

## Example: Routing



## Routing to Datum Points

### About Routing to Datum Points

When you create a datum point array in a piping assembly it is important to note that the resultant pipe segment will be routed through the datum points in the order the points were created.

### To Route a Pipeline to a Datum Point

#### Note:

While routing a pipeline, you can reassign line shape for pipe segments using the **Pipe Envrnmt** command in the **ROUTE PIPE** menu.

On-the-fly datum points can be created in all routing options that require datum selection.

1. Click **Route > To Pnt/Port**. The **PIPE SELECT** menu appears.
2. Click **Select Any** to select any appropriate point, port or pipe end.  
Click **Point/s** to select a point.  
Click **Entry Port** to select an entry port.

Click **Pipe End** to select a pipe end.

- If you choose a point, the system asks if you want to route through the whole datum point array. If you click **Yes**,

For straight segments, the **BEND RAD** menu appears.

- Click **Single Rad** to route through the datum point feature using the default bend radius from the active line stock and click **Quit**.
- Click **Multiple Rad** to specify the desired bend radius for each bend while routing, using the **SEL VALUE** menu and click **Quit**.

For flexible segments with set length, the **FIX LENGTH** menu appears.

- Click **Single Val** to route through the datum points, using the default length.
- Click **Multiple Val** to specify the desired length for each segment while routing.

The PIPE TO POINT feature is created.

### Creating a Datum

Routing Command	Create Datum Feature
To Pnt/Port	Point, Csys
Extend - Direction	Axis, Csys
Extend - Up To	Plane, Point
Connect	Csys
Set Start	Point, Csys

### Routing with Follow

#### About Routing with Follow

When you create pipe segments by following an axis/cylinder or sketched geometry, you do not need to set a start point.

Using the **PIPING > Route > ROUTE PIPE > Follow** command, you can use one of the routing methods listed in the **PIPE FOLLOW** menu:

- Sketch**—Create a line segment based on sketched geometry.
- Curve**—Create a line segment based on a curve.
- Pipe Line**—Create a line segment based on an existing pipe line.
- Axis**—Create a line segment coincident with an axis.

## Creating Pipeline Routing with Follow

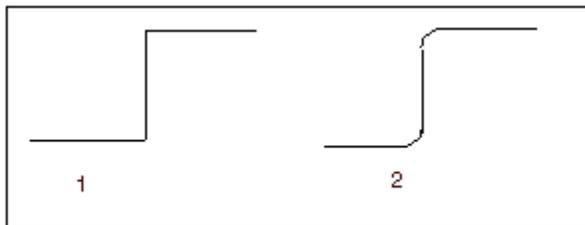
Using the **Follow** command in the **ROUTE PIPE** menu, you can create line segments based on sketched geometry, a curve, an existing pipeline, or coincident with an axis without having to set a start point.

### Following Sketched Geometry

You can create pipe line segments according to your sketch with the **Sketch** command. This routing action can be useful when routing a pipe line with respect to equipment or some structural elements. The new segments must lie in the same plane. The following rules apply:

- Following sketched geometry does not create a connection between the existing pipe end and new sketched segments, even if both end points are coincident. To join two endpoints, even if they are immediately on top of each other, use the **To Pnt/Port** or **Connect** commands.
- Bend radii or miter corners are added automatically to sketched geometry.

### Example: Generating Sketched Geometry



1 Sketched geometry

2 After the line was generated

### To Route a Pipe by Following an Axis/Cylinder

1. Click **Piping** > **Route** > **Follow** > **Axis** > **Done**.
2. Select an axis or cylinder to follow.
3. Adjust the segment ends on-the-fly by choosing **Change** > **Ends** > **Done** > **Mod End**. A cross indicates the selected end of the segment.
4. From the **CHOOSE** menu, select **Accept** or **Next**.
5. Using the **END COND** menu, do one of the following:
  - Click **On End** to constrain the pipe segment by the length of a cylinder or axis selected to follow; then click **Done**.or
  - Click **Drag** to extend the selected end of the pipe segment by dragging. Extend the line; then press the left mouse button to end the line. Adjust the length of the extension segment.

6. Adjust the other end of the segment by repeating Steps 1 through 5.
7. Click **Done > Done Follow**.

### Following an Axis or Cylinder

To create a pipeline segment by following an axis or a cylinder, use the **Axis** command in the **PIPE FOLLOW** menu. When you select a cylinder, the system creates an internal axis for the segment to follow.

By default, the ends of the routed segment correspond to the endpoints of the axis it followed. However, you may extend the ends beyond the axis that it follows.

### To Route a Pipeline to Follow an Existing Pipeline

1. Click **Route > Follow > Pipe Line > Done**.
2. Select an existing pipe segment to follow.
3. From the **START/END** menu, click **Specify** or **Entire Line** to specify if the routed pipeline should follow only a portion of the pipeline segment or the entire pipeline.
4. From the **CONSTRAINT** menu, click a command to constrain the pipeline to route
  - **Thru Point**—Selects or creates a point through which the routed pipeline must pass. This point should lie relatively near the starting point of the pipe length selected to follow.
  - **Start Point**—Selects or creates a point at which the routed pipeline must start.
  - **DistInPlane**—Selects a reference plane parallel to the starting segment of the reference pipeline, and defines the distance between the reference pipeline and the pipeline to route on this reference plane. The value of this distance must be at least the sum of the radii of the pipe being routed and the pipe followed to prevent the pipes from intersecting.
5. Do one of the following:
  - If you used **Thru Point** or **Start Point** to constrain the routed pipe, click **Create** or **Select** from the **PIPE POINT** to specify a point to serve as the through point or start point for the routed pipeline. The newly routed pipeline appears.
  - If you used **DistInPlane** to constrain the routed pipe, use the **SETUP PLANE** menu to select or make a datum plane through the pipe you are following on which the new routing will appear. Specify the direction from the reference pipeline the system should offset the routed pipeline, and type the distance. The newly routed pipeline appears.
6. Click **INT FOLLOW > Change** to modify the conditions for the follow operations, if desired, and click **Done Follow**.

## Following an Existing Pipeline

To route a pipeline segment by following an existing pipeline, use the **Pipe Line** command in the **PIPE FOLLOW** menu. You would typically use this method when routing pipelines that are very close together with similar geometry, as with a heat exchanger or pipes grouped to pass through a bulkhead.

## Non Specification-Driven Piping Flow Direction

### About Non Specification-Driven Piping Flow Direction

A pipeline has specific flow direction requirements based on the process flow requirements of the fluid inside the pipeline, such as gas, water, or fuel.

Pro/PIPING automatically assigns a default flow direction to each pipeline you create during modeling. A set of Flow Direction Rules determines the flow of direction. You can enable or disable the display of flow direction during the modeling of the pipeline in the active assembly.

Using Flow Direction, you can perform the following tasks:

- Display the currently assigned flow direction of the following:
  - Specified pipeline series
  - All series in a pipeline
  - Specified extension
- Reverse the assigned flow direction of all series in the specified pipelines.
- Reverse the assigned flow direction of specified pipeline extensions.

### Default Flow Direction Determination Rules

When you create a pipeline in any sequence, the flow direction function assigns default flow directions automatically based on the following rules:

- **Rule 1**—The flow direction of a pipeline series is automatically determined by the direction of the first pipe segment in the series.

#### Note:

- You can create a pipeline series by concatenating several independent segments that were created with the **Extend**, **Follow**, and **Connect** commands. Each of these commands can have opposing flow directions. When you connect the segments to form a pipeline series, the flow direction of the series is assigned based on the direction of the pipe segment that was created first.
- If you create the first segment in the correct flow direction, this ensures the correct flow direction for the entire series. Otherwise, you must reverse the resulting series.

- **Rule 2**—The flow direction of a new series, which is split and created as a result of inserting a branch fitting or stub-in, is the same as the original series from which it is created.

### To Display or Hide the Flow Direction of a Pipeline

1. Click  to switch between displaying and hiding the flow direction of the pipelines in the active assembly.
2. Click  to display the Piping System Tree. Select a pipe segment in the graphics window or a node from the Piping System Tree and right-click. A shortcut menu appears. Click **Flow** > **Show** to display the flow direction of the selected pipe series or pipeline.

**Note:** If you are working with pipeline assemblies that have been created using the earlier versions of Pro/ENGINEER, the toolbar option  appears only if you set the `pipe_allow_design_check` configuration option to on.

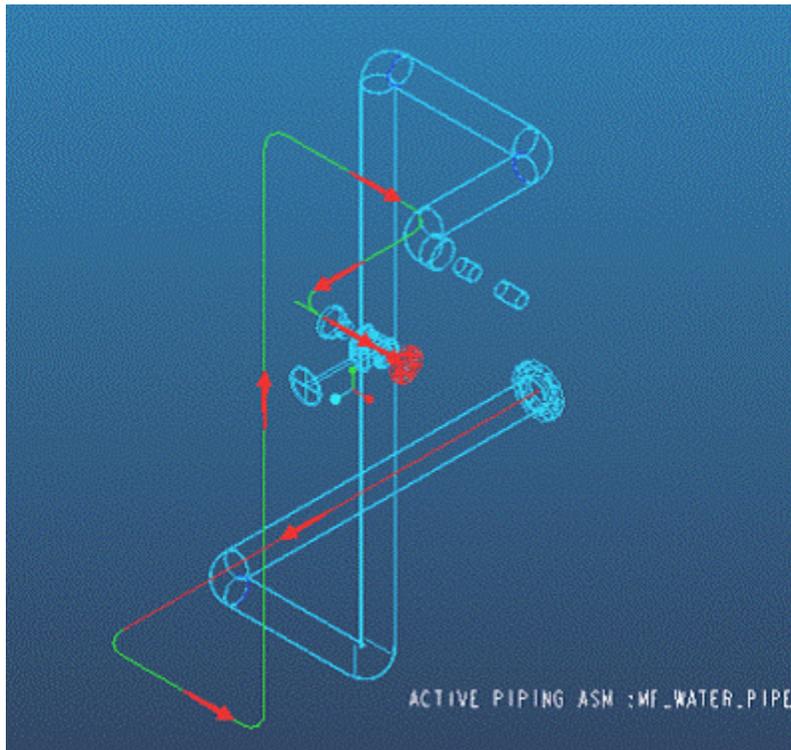
### To Display the Flow Direction of a Pipeline

1. Click **PIPING** > **Modify Pipe**. The **Modify Pipeline** dialog box opens.
2. Under **Active Assembly** select the active assembly.
3. Under **Modify Options**, click **Flow Direction**.
4. Under **Flow Direction**, select the flow unit you want to modify by selecting from the list. The options are:
  - **Pipeline**
  - **Extension**
  - **Series**

Select the unit. In the graphics window the flow direction is indicated by arrows along the pipeline.

5. Under **Flow Reversal**, click  to reverse the flow direction.
6. Click  to close the dialog box.

## Example: Displaying the Flow Direction of a Specified Pipeline Series



### Flow Direction Violations

You can view warning messages about violations in the **Design Rules Check** dialog box. The location of flow violation is marked on the screen with a dot.

You can encounter following flow violations:

- Converging flow at a junction.
- Diverging flow at a junction.
- Converging pipelines (two different pipelines that share a physical connection point but are opposed in direction).
- Diverging pipelines (two different pipelines that share a physical connection point but are opposed in direction).
- Pipelines with dead loops (loops that feed back into the pipeline and have no outlet).

### Suppressing and Resuming Pipe Segments

#### About Suppressing and Resuming Pipe Segments

When you route pipelines, you can suppress or resume one or more pipe segments to modify the pipeline trajectory.

**Note:**

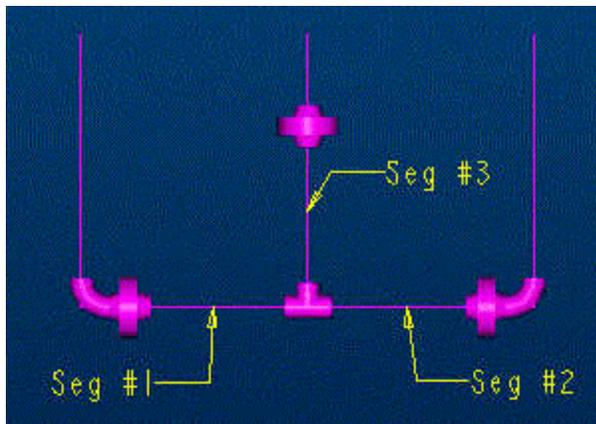
- The pipe segments that you want to suppress must belong to the same pipeline that you are currently routing.
- You can suppress pipe solids.
- Fittings of type break, cut, or elbow, that are placed on the ends of a suppressed pipe segment, are automatically changed to type "on end."
- All the suppressed components are resumed during the resume operation.
- You cannot resume a suppressed pipe segment if its end is used by a new segment due to routing.
- You cannot suppress pipe segments that have insulation, branch points, pipeline labels, or spools.

**To Suppress and Resume Pipe Segments**

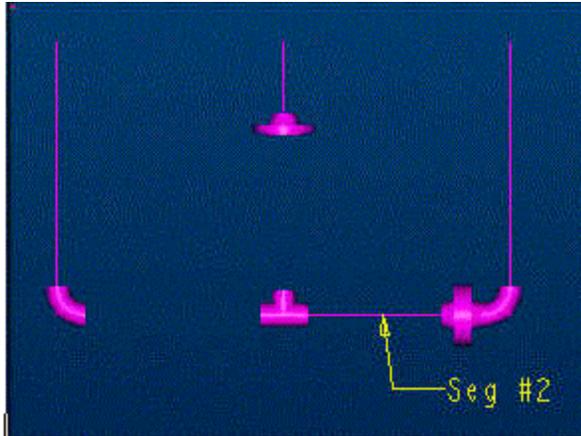
1. Click **Piping > Route > Suppress**.
2. Select the pipe segment to suppress. Pro/PIPING suppresses the selected pipelines and all the highlighted children.
3. Click **Piping > Route > Resume**.
4. Select the suppressed pipe segments that you want to resume. Pro/PIPING resumes all the suppressed segments and their associated components. If the selected pipe segment cannot be resumed, it is highlighted in red.

**Example: Suppressing and Resuming Pipe Segments**

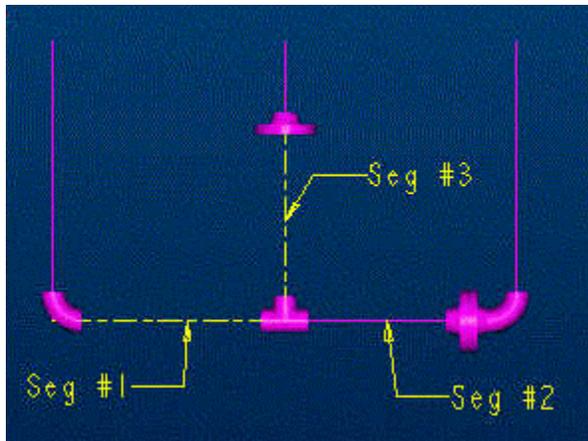
The original pipeline trajectory is shown in the next figure.



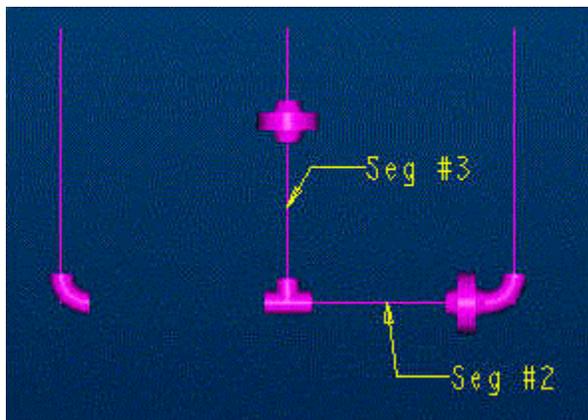
If you suppress segments #1 and #3, Pro/PIPING suppresses the associated flanges and gaskets that are mated, as shown next:



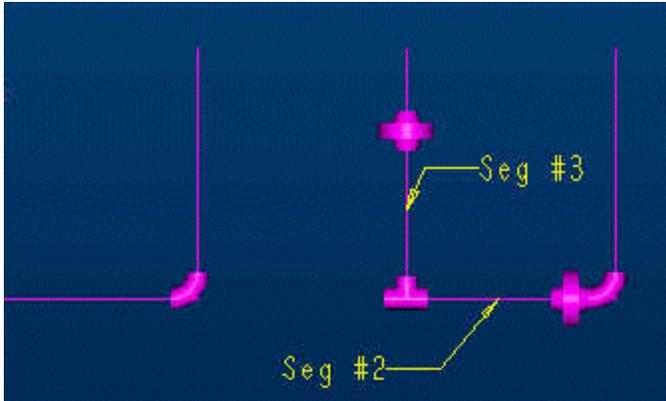
If you click **Resume**, all the segments that were previously suppressed are highlighted in yellow.



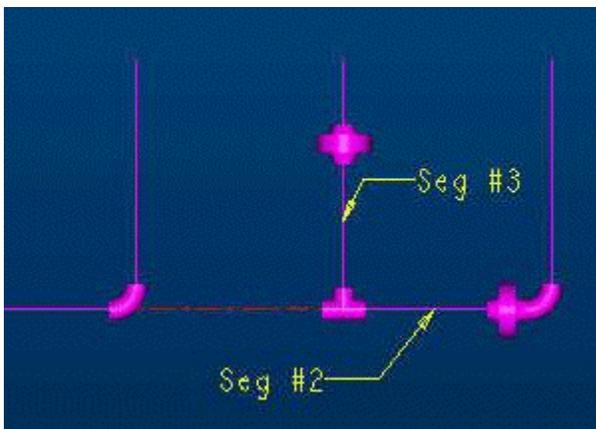
If you select segment # 3, Pro/PIPING resumes segment #3, the flange, and the gasket.



Pro/PIPING uses the free segment end for the new routing.



If you click **Resume** again, segment #1 is not resumed, because its end is occupied by an elbow of type "on end."



## Pipe Extends

### About Non Specification-Driven Routing with Extend

You can route a new pipe segment by extending the segment in a selected direction. To extend a pipe segment, the reference coordinate system can be one of the following:

- Any coordinate system in the active assembly or any other assembly
- The global coordinate system in the active assembly
- The routing coordinate system

By default, Pro/PIPING uses the reference coordinate system that was used for the previous extend operation in the active assembly. If an extend operation was not performed earlier in the active assembly, the global coordinate system is used by default.

## To Extend a Pipeline by Referencing a Plane or a Point (Non Spec-Driven)

1. Click **PIPING** > **Route** > **Set Start**, and specify the start point for the pipeline extension.
2. Click **PIPING** > **Route** > **Extend**. The **Extend** dialog box opens.
3. Under **Select Reference**, do the following:

- Under **Selection**, click  to select the reference coordinate system for the pipe segment to be routed or select one of the following from the list adjacent to  to specify the reference coordinate system:

**Global Csys**—Selects the global coordinate system as the reference coordinate system.

**Route Csys**—Selects the routing coordinate system as the reference coordinate system.

Other coordinate systems that you have used for the previous extend operations in the same extend session.

- Under **Type**, select the extend type as **Plane/Point** to specify that the pipe segment will be extended up to the selected plane or a point.
  - Select a plane or a point as the reference.
4. Under **Upto Plane/Point**, select one of the following from the **Options** list:
    - **Length**—Extends the pipe segment to the given length up to a reference plane or a point. Specify a value for length in the **Value** box.
    - **Offset**—Extends the pipe segment up to the reference plane or point until the end of the pipe segment is at a given offset distance from the reference plane or point. By default, Pro/PIPING displays the distance between the start of the pipe segment and the reference plane or point in the **Value** box. Change this value if required.
    - **Offset Fm Ref**—Extends the pipe segment up to the reference plane or point until the end of the pipe segment is at a given offset distance from another selected reference plane or coordinate system. Do each of the following:

Under **Select Plane/Csys**, select the reference plane or coordinate system.

Specify a value for the offset distance in the **Value** box or click  and drag the pipe segment to the required length. Pro/PIPING displays a preview of the pipe segment that is extended using the drag operation, in the graphics window.

If a coordinate system is selected, use the **Projection** list to specify an axis as the projection axis. If you project the end of the pipe segment to the

selected projection axis, the corresponding coordinate value will be equal to the given offset distance.

**Note:** The **Projection** box is not available if you select a plane under **Select Plane/Csys**.

- **Offset Edge**—Extends the pipe segment up to the reference plane or point until the end of the pipe segment is at a calculated offset distance from the reference plane or point. Specify a value for the offset distance in the **Value** box.
 

**Note:** Calculated offset distance = offset distance (+) or (-) 1/2 pipe outer diameter, depending on the offset direction.
  - **Offset Edge Fm Ref**—Extends the pipe segment up to the reference plane or point until the end of the pipe segment is at a calculated offset distance from another selected reference plane or pipe edge.
  - **Stop At**—Extends the pipe segment up to the reference plane or point.
5. Click  to extend the pipe segment up to the reference plane or point with the required length.

### To Extend a Pipeline by Referencing an Edge, Axis, or a Segment (Non Spec-Driven)

1. Click **PIPING** > **Route** > **Set Start**, and specify the start point for the pipeline extension.
2. Click **PIPING** > **Route** > **Extend**. The **Extend** dialog box opens.
3. Under **Select Reference**, do the following:
  - Under **Selection**, click  to select the reference coordinate system for the pipe segment to be routed or select one of the following from the list adjacent to  to specify the reference coordinate system:
 

**Global Csys**—Selects the global coordinate system as the reference coordinate system.

**Route Csys**—Selects the routing coordinate system as the reference coordinate system.

Other coordinate systems that you have used for the previous extend operations in the same extend session.
  - Under **Type**, select the extend type as **Axis/Edge/Segment** to specify that the pipe segment will be extended parallel to the selected axis, edge, or pipe segment.
  - Select an axis, edge, or a pipe segment as the reference.

4. Under **Parallel to Axis/Edge/Pipe Segment**, select one of the following from the **Options** list:

- **Length**—Extends the pipe segment to the given length along the reference axis, edge, or a pipe segment. Specify a value for the length in the **Value** box.
- **Offset Fm Ref**—Extends the pipe segment along the reference axis, edge, or a pipe segment till the end of the pipe segment is at a given offset distance from another selected reference plane or csys. Do each of the following:

Under **Select Plane/Csys**, select the reference plane or coordinate system.

Specify a value for the offset distance in the **Value** box or click  and drag the pipe segment to the required length. Pro/PIPING displays a preview of the pipe segment that is extended using the drag operation, in the graphics window.

If a coordinate system is selected, use the **Projection** list to specify an axis as the projection axis. If you project the end of the pipe segment to the selected projection axis, the corresponding coordinate value will be equal to the given offset distance.

**Note:** The **Projection** box is not available if you select a plane under **Select Plane/Csys**.

**Note:** The options **Offset**, **Offset Edge**, **Offset Edge Fm Ref**, and **Stop At** are not available if you use an axis, edge, or a pipe segment as a reference to extend the pipeline.

5. Click  to extend the pipe segment along the reference axis, edge, or a pipe segment with a required length.

### To Extend a Pipeline Using a Csys Axis Reference (Non Spec-Driven)

1. Click **PIPING** > **Route** > **Set Start**, and specify the start point for the pipeline extension.
2. Click **PIPING** > **Route** > **Extend**. The **Extend** dialog box opens.
3. Under **Select Reference**, do the following:
  - Under **Selection**, click  to select the reference coordinate system for the pipe segment to be routed or select one of the following from the list adjacent to  to specify the reference coordinate system:

**Global Csys**—Selects the global coordinate system as the reference coordinate system.

**Route Csys**—Selects the routing coordinate system as the reference coordinate system.

Other coordinate systems that you have used for the previous extend operations in the same extend session.

- Under **Type**, select the extend type as **Csys Axis** to specify that the pipe segment will be extended along the selected axis of the reference coordinate system.
4. Under **Along Csys Axis**, do the following:
- a. Under **Select Axis**, select one of the reference coordinate system axes as the reference axis.
  - b. Under **Options**, select one of the following to specify the length of the pipe segment:
 

**Length**—Extends the pipe segment to the given length along the reference axis. Specify a value for the length in the **Value** box.

**Offset Fm Ref**—Extends the pipe segment along the reference axis until its end is at a given offset distance from another selected reference plane or coordinate system. Do each of the following:
5. Under **Select Plane/Csys**, select the reference plane or coordinate system.
- Specify a value for the offset distance in the **Value** box or click  and drag the pipe segment to the required length. Pro/PIPING displays a preview of the pipe segment that is extended using the drag operation, in the graphics window.
  - If a coordinate system is selected, use the **Projection** list to specify an axis as the projection axis. If you project the end of the pipe segment to the selected projection axis, the corresponding coordinate value will be equal to the given offset distance.
- Note:** The **Projection** box is not available if you select a plane under **Select Plane/Csys**.
- Note:** The options **Offset**, **Offset Edge**, **Offset Edge Fm Ref**, and **Stop At** are not available if you use the **Csys Axis** extend type.
6. Click  to extend the pipe segment along the reference axis with the required length.

### To Extend a Pipeline Using a Csys Reference (Non Spec-Driven)

1. Click **PIPING** > **Route** > **Set Start**, and specify the start point for the pipeline extension.
2. Click **PIPING** > **Route** > **Extend**. The **Extend** dialog box opens.
3. Under **Select Reference**, do the following:

- Under **Selection**, click  to select the reference coordinate system for the pipe segment to be routed or select one of the following from the list adjacent to  to specify the reference coordinate system:
    - Global Csys**—Selects the global coordinate system as the reference coordinate system.
    - Route Csys**—Selects the routing coordinate system as the reference coordinate system.

Other coordinate systems that you have used for the previous extend operations in the same extend session.
  - Under **Type**, select the extend type as **Csys** to specify that the pipe segment will be extended to the given coordinates of the reference coordinate system.
4. Under **To Coordinates**, select the following dimension options:
- Select the system as **Spherical**, **Cylindrical**, or **Cartesian**, and use the **Angle** box to specify whether you want the angles in the selected system to be represented in degrees, as a gradient ratio, or as a gradient percentage.

**Note:**

If you select the system as **Cartesian**, you cannot edit the angle.

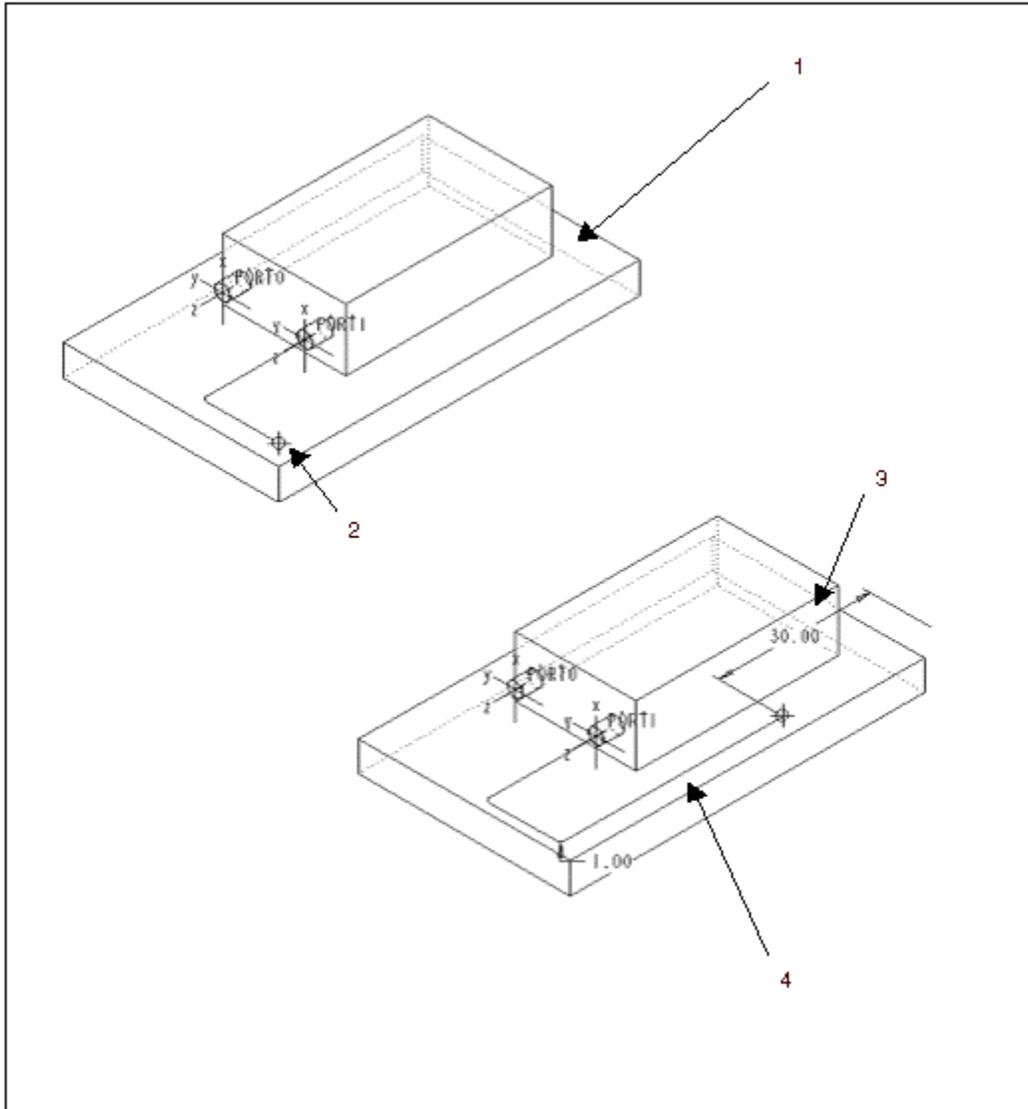
The value that you specify for the angle as a ratio or as a percentage is not a limited value.
  - Select one of the following dimension types:
    - Relative**—The extension of the end of the segment to be extended is referenced to the start of the segment.
    - Absolute**—The extension of the end of the segment to be extended is referenced to the selected reference coordinate system.
  - Under **Locking Axes**, select the extend directions along the x-, y-, or z-axes, or along all the three axes.
  - Under **Coordinates**, type the coordinate values along the respective axes in the available boxes or use the thumb wheel increments to specify the values. You can also click , and drag the pipe segment to the required length. Pro/PIPING displays a preview of the pipe segment that is extended using the drag operation, in the graphics window. Click  to lock the required coordinate boxes. The display changes to  to indicate that the boxes are locked.

**Note:** The coordinates displayed under **Coordinates** vary depending on the selection that you make in the **System** box. If you select the system as **Spherical**, the coordinates displayed are r, theta, and phi. If you select the

system as **Cylindrical**, the coordinates displayed are r, theta, and z. If you select the system as **Cartesian**, the coordinates displayed are x, y, and z.

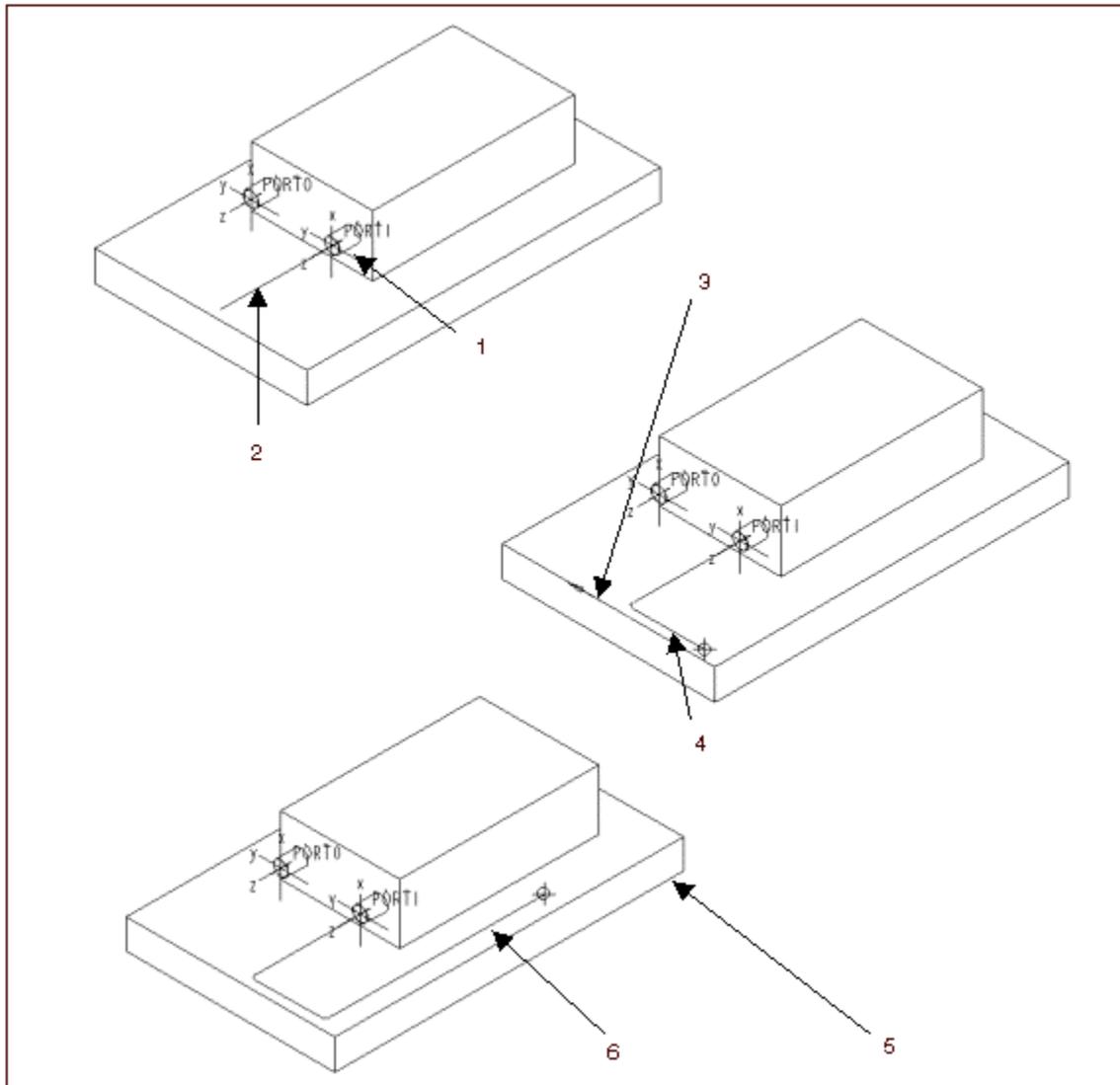
5. Click  to extend the pipe segment to the specified coordinates.

### Example: Extend up to a Surface or Point



- 1 Select this surface to extend up to
- 2 Start from this point
- 3 Enter the offset distance from the selected surface
- 4 A new segment is created

### Example: Extend a Pipeline in a Particular Direction



- 1 Select this entry point as a start point
- 2 A segment is created in the direction of Z-axis with the specified length
- 3 Select this edge to indicate the extension direction. A red arrow indicates positive direction
- 4 To create this segment, type a negative length value
- 5 The edge in whose direction the segment has to be created
- 6 Create this segment in the direction of the specified edge

## Routing Flexible Hoses with Minimum Bend Radius

### About Routing Flexible Hoses with Minimum Bend Radius

You can route a pipeline with a minimum bend radius, defined for different gauges to ensure accurate representation of a flexible hose.

Using **Min Bend Radius** in the **LineStock** dialog box, you can:

- Define the minimum bend radius for routing flexible hose lines
- Replace the linestock of existing hoses and automatically update the minimum bend radius values for those hoses
- Verify that no bend radius violations exist using the **Design Rules Check** dialog box.

### To Route Flexible Hoses with Minimum Bend Radius

1. Click **Applications > Piping > Set Up > Line Stock > Create**.
2. Type the new line stock name and click . The **LineStock** dialog box opens.
3. Click the **DesignRule** tab in the **LineStock** dialog box.
4. Type a value for the **Minimum Bend Radius** under **Design Rule Parameters**. The parameter value is used as a guideline to maintain an accurate representation of the flexible hose.
5. Route the flexible hose.
6. Check for bend radius violations using the **Design Rules Check** dialog box or click **Info > Geometry Checks**.

**Note:** Depending on the hose configuration, the specified minimum bend radius value can be violated during routing. A message appears when a violation has occurred. All violations of the bend radius are reported by both the **Design Rules Check** dialog box and the standard Pro/ENGINEER Geometry Check tool.

## Points on a Segment

### About Adding Points to Existing Routing

Using the **Insert Point** command in the **ROUTE PIPE** menu, you can add a new point to existing straight routing and drag it to the new position.

**Note:**

- Straight routing can follow a datum curve.
- You cannot use the **Insert Point** command on a pipe segment that is created using the **Follow** or **Connect** commands.

### To Add Static Points to Existing Routing

1. Click **ROUTE PIPE > Insert Point**.
2. Select a straight pipeline segment to modify.
3. To modify a free-length flexible pipe, go to Step 6. A theoretical bend intersection point appears on the screen as 'x.'
4. Using the **CHOOSE POINT** menu, do one of the following:
  - o Click **Select** and select an existing datum point. The pipe segment is reshaped to pass through the selected datum point.
  - o Click **Create** and **Static**, and click **Done**. The **DATUM POINT** dialog box opens. Create a datum point.

**Note:** To create a datum point using any other datum point creation tool, do the following:

- a. Open a datum point creation dialog box by clicking a datum point creation tool. For example, open the **Offset CSys Datum Point** dialog box without closing the **DATUM POINT** dialog box that you have opened earlier.
  - b. Create a datum point using the **Offset CSys Datum Point** dialog box.
  - c. Select the newly created datum point so that it appears as a reference in the **DATUM POINT** dialog box and click **Ok**.
5. Use the **Offset Surf** or the **Offset Csys** commands in the **OFFSET PNT** menu to define the dimensioning references of the datum point.

The selected pipe segment is replaced with new points and Pipe to Point features.

6. Select a free-length flexible pipeline. You can add, delete, and move interpolation points as needed or move the control polygon points using the **Modify Curve** dialog box.

### To Add Dynamic Points to Existing Routing

1. Click **ROUTE PIPE > Insert Point**.
2. Select a straight pipe segment to insert the points. The ends of the pipe segment are highlighted.
3. Click **Create** and **Dynamic** in the **CHOOSE POINT** menu and click **Done**. An additional point is highlighted between the ends of the pipe segment.
4. Select a point or segment or a section on the pipe to move. Two points are highlighted on the pipe segment based on your selection.
5. Click **No** or **Yes** in the message area depending on whether you want to create a single break point or two break points.
6. Define the motion plane using the **SETUP PLANE** menu.

7. Use the sliders in the **MOVE DRAG SEGMENT** thermo tool to move the break point as well as the selected piece of the pipe segment. You can move the selected section of the pipe segment normal to the plane as well as normal to the pipe segment. The new shape of the pipeline is displayed dynamically according to the slider movement.

**Note:**

Set the sensitivity slider to 100 before you use the other sliders to move the point or the pipe segment.

A section of the pipe segment is marked between the highlighted points as discussed in step 4.

After setting the slider values, left-click in the area above the first slider in the **MOVE DRAG SEGMENT** thermo tool control area to accept the new shape, middle-click to abort the operation, or right-click to reset the **MOVE DRAG SEGMENT** thermo tool.

8. Use the **Offset Surf** or the **Offset Csys** options in the **OFFSET PNT** menu to define the dimensioning references of the datum point.

The original pipe segment is replaced with Pipe to Point features. The position of this point determines the position of the segment split when you create a dynamic point on a straight segment.

## Creating a Dynamic Point

While modifying a pipeline segment, you can now insert two points on the segment. Working directly in the Pro/ENGINEER window, you can click and drag the straight segment until you define the shape that you want.

## Fittings

### About Fittings

A fitting is a part or assembly that you add to a piping system to perform specific functions. You can store fittings in libraries so that you can easily retrieve them when you need to insert them in the pipeline. Generally, you can create four types of fittings:

- Corner
- End
- Straight Break
- Straight Continues

Pro/PIPING treats a fitting as an assembly component. To be used as a fitting, a model must have the following features:

- Entry ports created at the connect points. An entry port is a coordinate system that you create in an assembly, in a part, or at the topmost level of a fitting (that is, at the highest level in a fitting assembly).

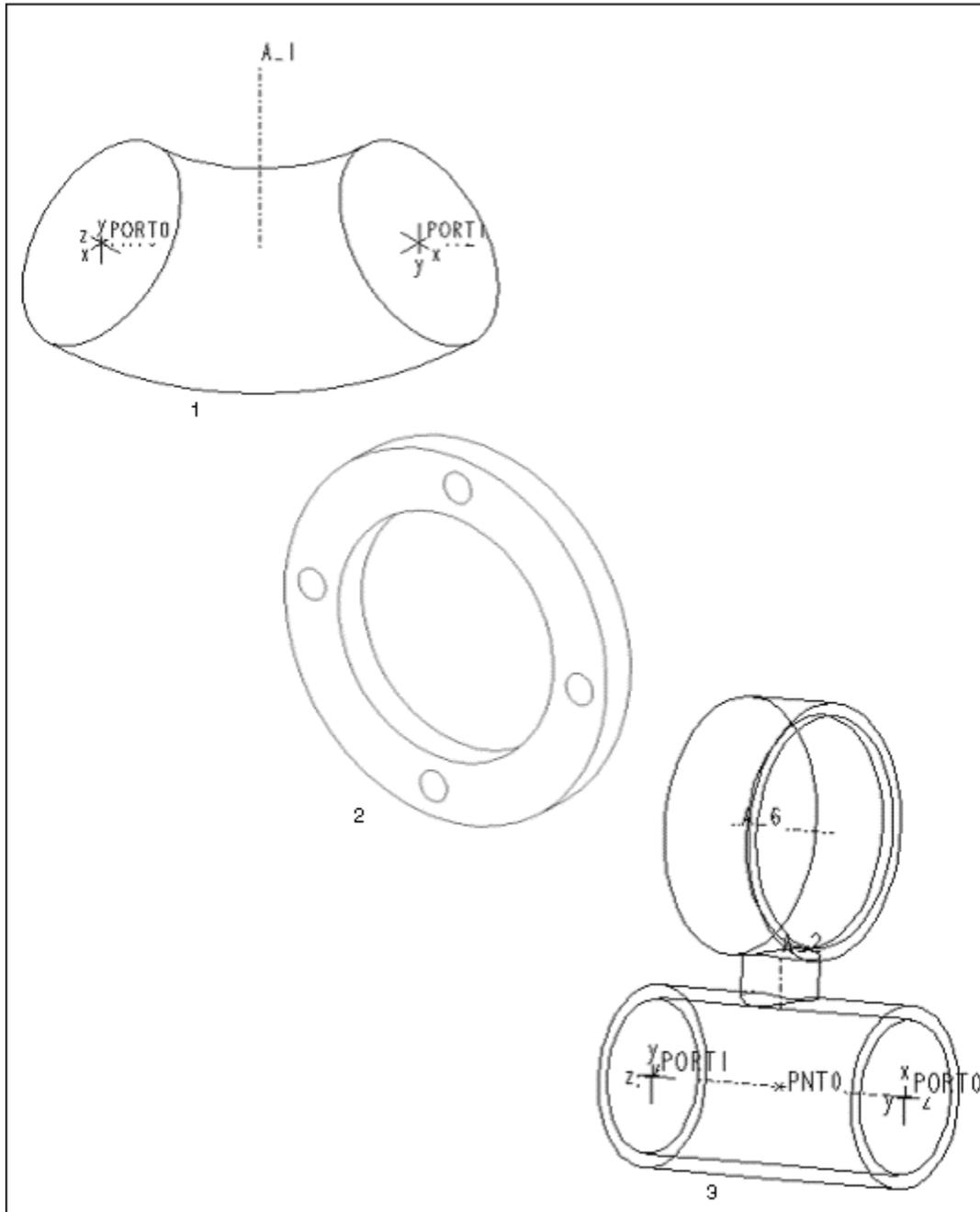
Entry ports are specially named coordinate systems that can be referenced by certain piping routing features. By choosing **Entry Port** from the **PIPE SELECT** menu, you can select such a coordinate system. It allows you to define contact points, a direction for line routing, and the orientation of fittings.

When creating an entry port, keep in mind the following:

- The direction of the z-axis determines the direction for routing. In fittings, the z-axis always points out of the fitting and the pipeline enters or exits the fitting parallel to the z-axis.
  - In a fitting, entry ports are located along the centerline; the desired penetration distance of the pipe solid determines the offset of the entry port from the face of the fitting. The system trims pipe segments that go into the fitting at the entry ports, although the pipe centerline may continue through the pipe fitting.
- A defined datum point for the system to use to align the fitting with the centerline of connecting pipe segments or at the end of a pipe segment. The alignment point is typically located along the z-axis at the center of symmetry of the fitting.

Using commands in the **FITTING** menu, you can insert a fitting, delete it, redefine it, and replace it.

You can also perform these operations using the **Info**, **Edit Definition**, **Reroute**, and **Delete** options from the shortcut menu that appears in the graphics window when you right-click.

**Example: Fittings**

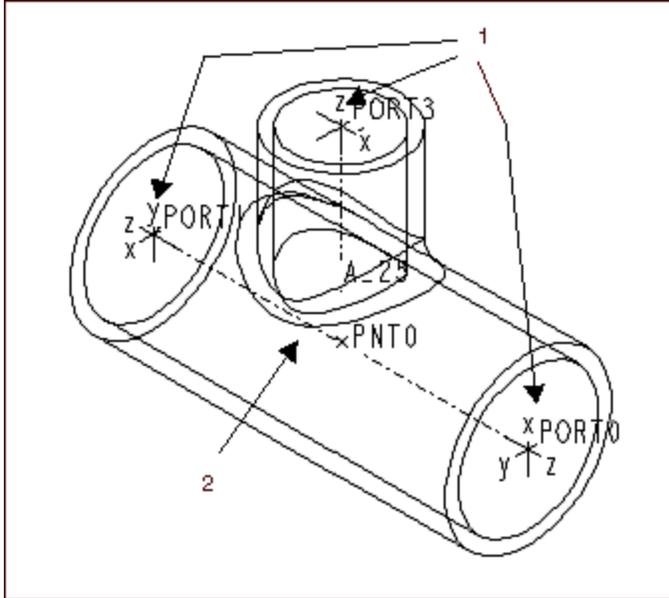
1 Elbow

2 Flange

3 Miter

**Example: Tee Fitting with Required References**

For fittings with three defined entry ports, the alignment point should lie at the intersection of the z-axes. The z-axis of an entry port must point out of the fitting.



1 Three entry points are defined in this fitting

2 Datum point is created along the centerline

### About Inserting a Fitting

You can insert a fitting at a pipe end, at a fitting type corner of a pipeline, and along a straight segment of a pipeline. When inserting a fitting, keep in mind the following:

- You must place fittings using special techniques. You cannot click **Assemble** from the **COMPONENT** menu to insert a fitting. You can assemble fittings as components and route up to them as for other equipment. However, fitting-specific functionality will not apply.
- Once you have inserted a fitting in a pipeline, it becomes associated with this pipeline. You can use only two of a fitting's entry ports to insert the fitting in a pipeline. If there are more than two entry ports (as in a tee fitting, for example), you must twist the fitting to the desired orientation and then route a pipe segment to the third entry port.

### To Insert a Fitting

1. Click **Fitting > Insert**.
2. Using the **INSERT TYPE** menu, do one of the following:
  - To insert a fitting at a pipe end, click **End**. Select a pipe end at which to attach the fitting, an entry port on the fitting from which you want the pipe to come, and a point on the fitting to align with the pipe end.
  - To insert a fitting at a corner, click **Corner**. Select a fitting-type pipe corner, and go to Step 5.

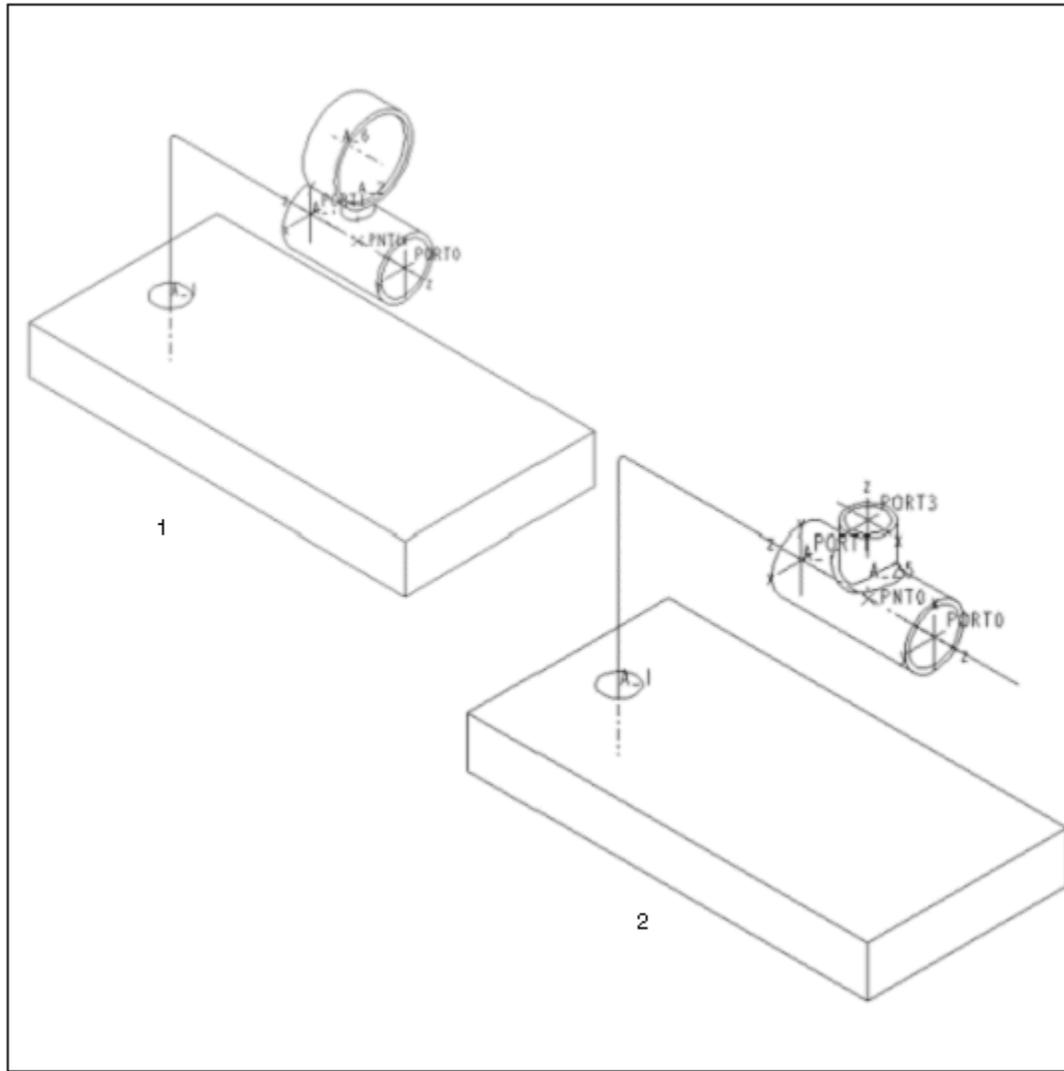
- To insert a fitting that breaks the pipeline at a point on a straight segment of pipe, such as a valve, click **Straight Brk**. Select or create a point on a pipe segment.

This changes the number of solids required for the pipeline.

- To insert a fitting that does not break the pipe at a point on a straight segment of pipe, such as a support, click **Straight Cont**. Select or create a point on a pipe segment.
3. Select a fitting from the Open dialog.
  4. Select a point on the fitting to align with the pipe centerline.
  5. The system places the fitting automatically. Adjust the orientation using commands in the **ORIENT FIT** menu (**Fitting** > **Redefine**).
  6. When you have placed the fitting properly, click **ORIENT FIT** > **Done**.

**Note:** You can also insert a missing fitting node using the standard Model Tree Options.

### Example: Pipe End Fitting and Straight Break



1 Sample of Pipe End Fitting

2 Sample of Straight Break

### About Deleting a Fitting

Using the **Delete** command in the **FITTING** menu, you can remove a fitting and all subsequent segments. However, when removing a fitting, keep in mind the following:

- If you delete a straight break fitting, the **Straight Brk** feature remains, unless you specifically delete it. This feature causes a break in the piping solid for the pipeline at that point. Thus, a straight segment of pipe has two solids, one on either side of the break.

When the system asks you if you want to remove the pipe break as well, if you type [yes], it deletes the datum point and pipe joint feature. However, you also

have to specify whether you want to delete the solids as well, if they exist, since the centerline segments they referenced will no longer exist. There is now only one segment.

- The **Delete** command works only on inserted fittings and does not remove other components. When you remove a straight-continuous fitting, you may also want to delete the datum point used for placement if you created it while placing the fitting.
- If you delete a pipe joint on a segment that has other fitting children, the system asks you if you want to automatically reroute the fitting child to keep the current location along the pipe segment.

If you type [yes], it automatically reroutes the point used for placement to the new pipeline segment that it created when you deleted the pipe joint so that it maintains the same position. If you type [no], use the **Fix Model** environment to fix the fitting child.

### To Redefine the Orientation of a Fitting

1. Click **Fitting > Redefine**.
2. Select a fitting to redefine.
3. Click **REDEF FITT > Orientation**. To flip the fitting, go to Step 4. To twist it, go to Step 5.
4. Click **ORIENT FIT > Flip**. The system flips the position of the entry ports.
5. Click **Twist** and do one or all of the following, as necessary:
  - Using **Set Zero Ref**, set an edge or axis as the zero angular reference for the x-axis orientation of the highlighted port.

You cannot twist internally inserted corner fittings.

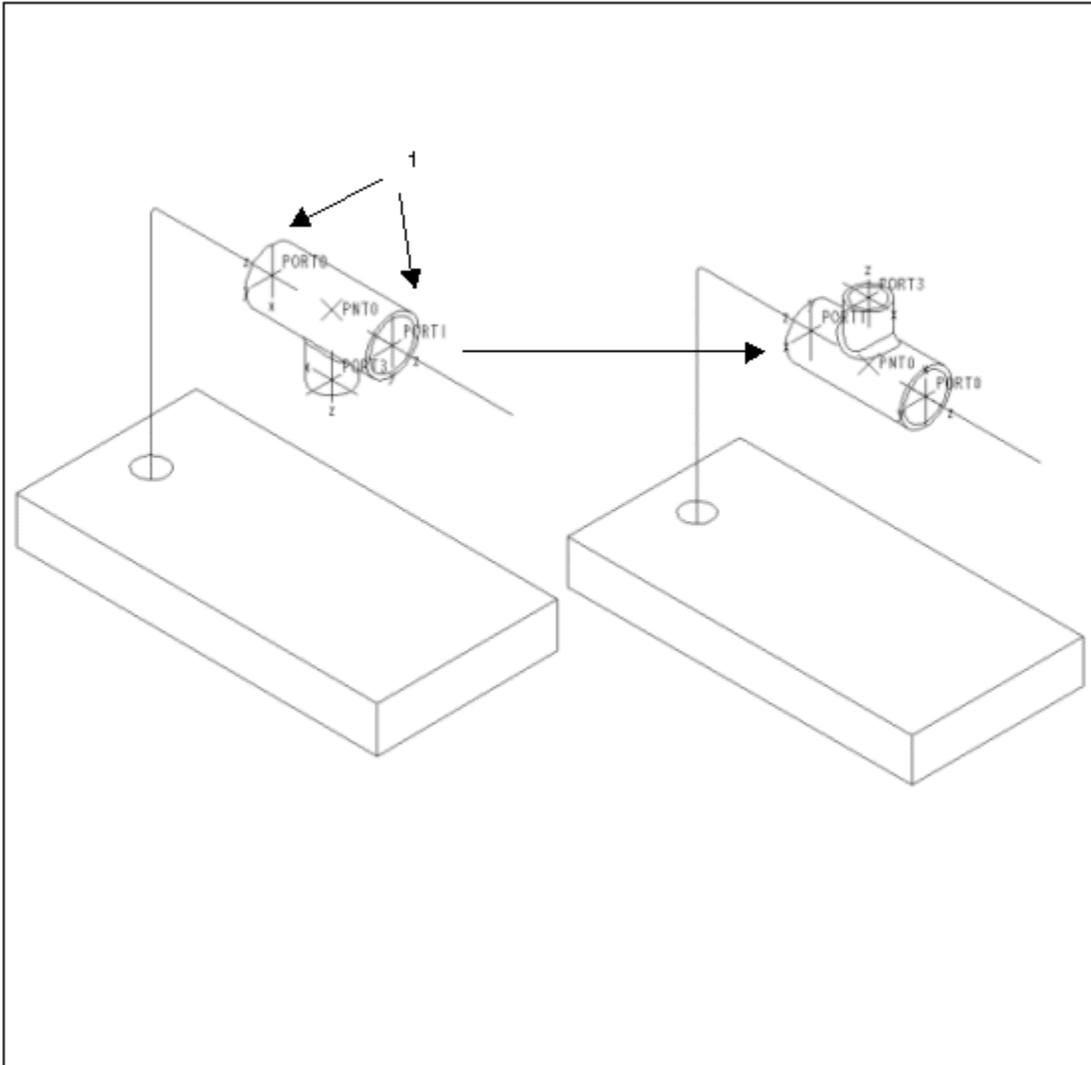
- Using **Clear Zero Ref**, remove a previously set zero reference by changing the fitting twist to be relative to its current position. The fitting maintains its position, and any twist dimension disappears.

To modify twist dimensions, click **Assembly > Modify > Value, Piping > Modify**, or **Piping > Route > Select > Query Sel**. A Query Bin window opens. Click **Accept**. Then click **Route Pipe > Mod Dim**.

- Using **Enter Value**, specify the rotation of the highlighted port in degrees about the reference-port z-axis. If there is no zero reference, this angle is relative to the current position. If there is a zero reference, the angle is between the zero reference and the x-axis on the highlighted entry port.
- Using **Align X**, align the x-axis of the highlighted entry port to the geometry reference.
- Using **Align Y**, align the y-axis.

The system updates the dimension and changes the position of the fitting accordingly.

### Example: Redefine the Orientation of a Fitting



1 Reorient the fitting by flipping these entry ports

### About Redefining a Fitting

Using the **Redefine** command in the **FITTING** menu, you can redefine the point of the fitting and place it, and you can change the orientation of the fitting while inserting it by flipping or twisting it.

### To Redefine the Point of a Fitting

1. Click **Fitting** > **Redefine**.
2. Select a fitting. The **REDEF FITT** menu appears.

3. Click **Point**. The **FITT PNT** menu appears.
4. Click **Place Point**. The **Select** menu appears.
5. Select a point and click **Done**.
6. If required, click **FITT PNT > Align Point** to change the alignment of the point.

### About Replacing a Fitting

You can manually replace a fitting with another one or replace a fitting using a functional interchange member, family table, or layout.

### To Replace a Fitting

1. Click **Fitting > Replace**.
2. Select a fitting on the screen or Model Tree, or use **Sel By Menu** to select one from the namelist menu.
3. From the **REPLACE WITH** menu, click a command to replace the fitting.
4. Adjust the fitting orientation, using commands in the **ORIENT FIT** menu.

Replacing a fitting can occasionally create conflicts with existing pipeline segments. For example, when you replace an internal fitting with one that has fewer entry ports, regeneration may fail.

Or, when you replace a fitting after routing a line segment from an entry port of this fitting, this segment is a child of the original fitting from which it was routed.

When you create such a conflict, the system suspends the replacement process and Pro/PIPING enters a special resolve environment. You can usually resolve the conflict by reselecting the contact points (entry ports).

### To Modify Penetration Distance

Pipe solids associated with a fitting terminate at the entry port of the fitting's coordinate system.

1. To control the penetration distance in a fitting, modify the location of the entry port coordinate system in the fitting model.
2. Regenerate the fitting model, and the system updates the solid pipe.

## Modifying

### Modifying in Routing

#### About Modifying in Routing

Using commands in the **ROUTE PIPE** menu, you can delete the last routing operation, redefine the references of a routing operation, modify dimensions, and replace one or more contiguous pipeline segments with new routing.

Alternatively, use the **Modify Dim** option from the **Piping** menu to modify dimensions.

#### To Redefine the References of a Routing Operation

1. Click **ROUTE PIPE > Redefine**.

You can also redefine the references of a routing feature in the standard assembly environment using **Redefine** or **Reroute** in the **ASSY FEAT** menu, or from the Model Tree Window.

2. Select a pipe segment to replace. The selected routing feature and all subsequent features temporarily disappear.
3. Reroute the segment using the same type of routing action as the original. After you enter routing specifications, the pipeline regenerates and the subsequent features reappear referencing the newly created routing feature.

#### To Modify Dimensions of Routing Features of a Pipeline

1. Click **Piping > ROUTE** and select a pipeline segment to route. The **ROUTE PIPE** menu appears.
2. Click **Modify Dim**. Select a pipeline.

Solid pipe dimensions are read-only. You must select the pipeline of the segment that has the dimensions you want to modify. When you modify these dimensions and regenerate the assembly, the system updates the solid pipe feature to reflect any changes in the underlying routing. Thus, you cannot modify the dimensions of pipe solids, although you can view reference dimensions and show them in drawings of the solid.

3. Select a dimension to modify and type a new value and press ENTER.
4. Continue to modify other dimensions or click **SELECT > Done Sel**. The system automatically regenerates the assembly to update the pipeline with the new dimensions.

#### To Replace One or More Contiguous Pipeline Segments

1. Click **PIPING > Route** and select a pipeline segment to route. The **ROUTE PIPE** menu appears.

2. Click **Replace** > **Enter Replace**.
3. Select routing features to replace, then click **Done Sel**. The selected features and all child routing features temporarily disappear.
4. Route the new pipe using any combination of routing commands.
5. When you have finished replacing the segments, click **ROUTE PIPE** > **Replace**.
6. Click either **Done Replace** to confirm your changes, or **Quit Replace** to cancel them. The temporarily rolled-back pipeline reappears attached to the new end of the routing that replaced the old one.

### Changing Line Stock and Line Shape Parameters While Routing Pipelines

At the outset of routing a pipeline, the system uses default parameters from the current line stock of the pipeline being routed. These environment settings are modal; that is, they remain set for all further routing until you change them.

While routing a pipeline, you can reassign line stock, corner type, and line shape for subsequent segments using the **Pipe Envrnmt** command in the **ROUTE PIPE** menu. You can reassign a line stock for segments that start from a new start point or after a fitting.

### Rerouting a Routing Feature Attached to a Set Start Feature

If you want to start a pipeline from a different **Set Start** feature, instead of deleting a routing feature attached to the **Set Start** feature and creating new routing, you can do one of the following:

- Redefine or reroute the initial **Set Start** feature to the desired reference by selecting it by ID or from the Model Tree.
- Replace the routing that is attached to the **Set Start** feature with another **Set Start** feature in the desired position plus another segment routed to connect it to the rest of the pipeline.

### Modifying Pipe Options

#### About Modifying Pipe Options

You can modify pipes in an active assembly with the **Modify Pipeline** dialog box.

To modify existing segments:

- Change dimensions of a feature created by a single routing action. Only dimensions created when routing the pipeline will be available for modification.
- Assign a different line stock to a pipeline segment.
- Change pipeline parameters.
- Change bend parameters.

- Change the corner type.

### To Modify Pipeline Parameters

1. Click **PIPING > Modify Pipe**. The **Modify Pipeline** dialog box opens.
2. Click the **Parameters** button. The **Parameters** portion of the dialog box opens.
3. Click **Pipe Line**. The **Edit Pipeline** dialog box opens.
4. Using the **Delete** button, you can delete pipeline parameters. When removing pipeline parameters, if you select a pipeline that has children (any routing features, inserted fittings, pipe solids, branches, and so on), you must confirm if you want the system to delete them.
5. Using the **Add** button, you can add user-defined pipeline parameters. Type in a name and value.

### To Modify Line Stock Parameters

You can display line stock parameters and modify them in the Model Tree.

1. Select and right-click on the parameter in the Model Tree. The shortcut menu appears.
2. Choose the **Modify** command from the menu. The **LineStock** dialog box opens.

### To Modify Bend Parameters

To modify bend parameters, use the Model Tree-like user interface. To access the Model Tree, follow this procedure.

1. In the **Modify Pipeline** dialog box, click **Modify Options > Parameters**. The dialog box changes to show **Pipe Line Parameters** and **Bend Parameters**.
2. Click **Bend**. The **Bend Parameters** Model Tree window opens.
3. Select the bend parameter you want to change and press the right mouse button to activate the menu. Click the **Modify** command.

### Using the Modify Pipeline Dialog Box

The **Modify Pipeline** dialog box lets you modify several options in the active assembly, the corners, parameters, line stocks, and flex shapes. In the **Active**

**Assembly** area of this dialog box, click  to activate the assembly in the Pro/ENGINEER window.

In the **Modify Options** area, the following choices appear.

- **Corner**—Select types and values of single or multiple corners.
- **Line Stock**—Set the values of selected single or multiple segments.
- **Parameters**—Modify Pipeline and Bend parameters.

- **Flex Shape**—Modify the line shape of a flexible hose between Fixed and Free length.
- **Flow Direction**—Select the flow unit and change the direction of the flow.

When you click an option under **Modify Options**, the last area in the dialog box changes. For example, if you click **Corner**, a **Corner** group box appears. You need to select a pipe segment corner to change it to a Bend, Fitting, or Miter Cut in the **New Type** box and to modify the value of the bend or fitting in the **New Value** box.

If you click the **Parameters** option, the **Parameters** group box opens.

In the **Parameters** area, you can select **Pipe Line Parameters > Pipe Line or Bend Parameters > Bend** to modify these parameters. If you click **Pipe Line**, the **Edit Pipeline** dialog box opens. If you click **Bend**, the **Bend Parameters Model Tree** opens.

If you click the **Line Stock** option, the **Line Stock** group box opens.

Modify the line stock listed in the **Modify Linestock** drop-down box with the **Edit** option or create a new line stock with the **Create new** option. If you click **Create new**, enter the line stock name at the command prompt.

If you click the **Flex Shape** option, the **Flex Shape** group box opens.

Modify the **Flex Shape** by selecting a flexible pipe segment. The **Modify Shape** group box contains two shapes that you can change, **Fixed Length** and **Free Length**. If you click **Fixed Length**, the **Length** box activates with a number you can modify.

### Using the Edit Pipeline Dialog Box

The **Edit Pipeline** dialog box opens after you click **Parameters** in the **Modify Pipeline** dialog box. The **Edit Pipeline** dialog box lets you perform the following tasks:

- Select a pipeline parameter from the Pro/ENGINEER window.
- Give the parameter a new name.
- Add the parameter to the pipeline.
- Save the parameter to a file.
- Read in the parameter as a .pp1 file.
- Delete the parameter from the pipeline.
- Cancel changes made to the pipeline.
- Modify an existing pipeline label.

### To Modify a Corner Type

1. Click **Piping > Modify Pipe**. The **Modify Pipeline** dialog box opens.
2. Under **Active Assembly** select the active assembly.

3. Under **Modify Options**, select **Corner Type**.
4. Under **Corner**, select the corners you want to modify.
5. In the **New Type** box, select a new corner type from the list.

**Note:** The **New Type** box displays only those corner types that are specified in the line stock of the selected pipe.

- If you select a corner of type Bend, the **Bend Radius** box displays the default values for the bend radius defined in the line stock. Select the required bend radius value from the **Bend Radius** list.

The values in Bend Radius reflect the values present in the line stock. You can also enter a new value by typing.

**Note:** The **Bend Radius** box is available only when you select the new type as bend.

- If you select a corner of type Miter Cut, the **Cut Length** box displays a default value for the miter cut length. The **Cut Number** box displays the default value for the cut number. If more than one value is specified for the cut number in the line stock, select the required cut number value from the **Cut Number** list.
- The **Cut Length** and the **Cut Number** fields are available only when you select the new type as miter cut.

6. Click  to apply the changes and close the dialog box.

### To Modify the Line Shape of a Flexible Hose

1. Using the **Modify Pipeline** dialog box, select an active assembly with the arrow.
2. Under **Modify Options**, click **Flex Shape**.
3. Under **Flex Shape**, select a pipeline segment with the arrow.
4. Under **Modify Shape**, click **Fixed Length** or **Free Length** and type a value in the **Length** option.
5. Click **Apply**, **OK**, and **Close**.

### Changing Pipelines

Using the **PIPING > Pipeline > OPER PIPE LINE** menu, you can perform the following operations on pipelines:

- Create/Route
- Delete
- Rename
- Suppress

- Resume

## Resolving a Failed Regeneration

Sometimes changes you make to pipelines conflict with other geometry or feature parameters. As a result, your attempt to regenerate the assembly may fail.

When a regeneration fails, you must resolve the problem before continuing with normal model processing. To facilitate this, Pro/ENGINEER provides a special error resolution environment, that is, the Resolve environment to help you correct the problem.

All Pro/PIPING processing resides in Assembly mode, therefore, follow the Assembly mode recovery procedure. All assembly functionality is available to repair the model.

For example, if the failure lies in the creation or modification of a solid pipe, you can repair the underlying pipe routing feature using Pro/PIPING functionality.

## Modifying Line Segments

### About Creating Line Segments Based on Sketched Geometry

To create pipeline segments by referencing a sketch, use the **Sketch** command in the **PIPE FOLLOW** menu. You may find this routing action useful when routing a pipeline with respect to equipment or some structural elements, provided the new segments lie in the same plane.

When using this routing technique, remember these points:

- Following sketched geometry does not create a connection between the existing pipe end and new sketched segments even if both endpoints are aligned. To join two endpoints (even if they are immediately on top of each other), use the **To Pnt/Port** or **Connect** commands after finishing the sketched sections.
- You do not have to add bend radii or miter corners to sketched geometry. The system does this automatically.

### To Create Pipeline Segments by Sketching

1. Click **Follow > Sketch > Done**.
2. Select a sketching plane.
3. Specify a horizontal or vertical reference for sketching.
4. Sketch geometry, following a pipe centerline. Use the current sketching techniques. Remember to locate sketched geometry with respect to the assembly geometry to which the pipe placement references.

If the current corner type of the line stock is set to **Bend**, you can sketch arcs in addition to straight lines. However, you should use them to route pipelines around objects rather than to create bends in a pipeline. The system automatically creates bends in a pipeline at corners after you finish sketching.

5. After the sketch regenerates successfully, click **SKETCHER > Done**.
6. Click **INT FOLLOW > Change** and modify the conditions for the follow operations if desired.
7. Finalize the feature by choosing **Done Follow**.
8. The system prompts you to connect the sketched segment to another pipe segment at each free end. If you do not want to connect the segment, click **CONNECT > Quit Connect**.

## Solids

### About Creating a Solid Pipe

Initially, you route pipes by routing centerlines. Once you have completed the routing, you can use the **Pipe Solid** command in the **PIPING** menu to make a solid pipe based on the green pipe routing centerline.

A solid pipe appears as a solid cylinder of the correct diameter and wall thickness. When the system makes a pipe solid, it cuts holes into it at the location of each incoming or outgoing branch and trims the incoming and/or outgoing branches back to accommodate inserted fittings.

You must make a pipeline solid for the following tasks:

- Showing pipe solid parts in the Bill of Materials
- Retrieving a solid pipe in Part, Assembly, or Drawing mode without the original reference assembly
- Performing engineering calculations such as mass property calculations and interference calculations
- Creating brackets and other parts that reference the pipe's solid geometry

You should create piping solids near the end of the process because each time the number of solids changes (by deleting a straight break fitting, for example), you must define or delete solids.

### To Create a Solid Pipe

1. Click **PIPING > Fabrication > Pipe Solid**. The **Pipe Solid** dialog box opens.
2. Select a piping assembly under **Active Assembly**.
3. Under **Segment Tree** and **SEGMENT**, click a pipe segment in the piping assembly that you want to make solid (all segments appear with a default name). The **Make** and **Erase** buttons get activated.
4. Click **Make**. The pipe becomes solid. In the Segment Tree, a pipe solid icon replaces the segment.

5. If you want to remove the solid pipe effect and go back to the pipeline, click the segment and then click **Erase** (**Erase** also deletes the solid part from the session). The screen updates immediately.

## Working with Solid Pipes

When working with solid pipes, keep in mind the following issues:

- **Regenerating after modifying a pipe centerline**—A solid pipe updates to match a modification to the pipe centerline when the system regenerates it. To update the solid pipe, perform an automatic regeneration or specifically select the solid pipe to regenerate.
- **Assigning names to new pipe solids**—Each pipe solid that you create using **Pipe Solid** must have a different name even if it is exactly the same as another solid.

You can assemble multiple instances of the same solid if the shape remains the same. However, if you need fifteen 10 1/4-inch diameter hoses with different shapes, for example, you will need fifteen solid names. If all similar shapes must have the same name in the BOM for ordering parts, you can use parameters and report relations.

- **Deleting a solid and recreating it**—If you want to use the same name when you delete and recreate a solid, you must delete the pipe solid from the assembly, from memory, and from the working directory. However, when using Pro/PDM, if you delete the solid or recreate it and it uses the same name, you still cannot submit it to Pro/PDM.

If you create a new piping solid, you must use a new name in order to submit it. If you delete the part and submit a new one with the same name, the system recognizes that it is not the same part because all of the revision/status information is different from the part with that name in the database.

- **Viewing pipe solid dimensions**—Making a pipe solid creates read-only dimensions that define the pipe's straight pipeline lengths between bend locations, diagonal miter lengths, and corner radii. You can show these dimensions in Part and Drawing modes.

The system provides the solid dimensions for information purposes only. To modify the pipe solid dimensions, you must modify the dimensions of the underlying green routing centerline.

## Using the Pipe Solid Dialog Box

The **Pipe Solid** dialog box performs several functions:

- Creates a pipe solid from a segment in an assembly
- Uses a Start Part in the **Copy From** box to create a new pipe solid
- Creates a pipe solid with a new name in the **SOLID NAME** column

The **Pipe Solid** dialog box has the following options:

- **Active Assembly**—Contains the name of the active assembly you select under **Select Assembly**.
- **Segment Tree**—Displays the tree structure and the names of the pipe segments in the assembly you have selected.
  - **SEGMENT**—Segment names in the piping assembly
  - **SOLID NAME**—New name that you have changed from the original Piping solid part name.
  - **START PART**—Name of the original part that is in the template directory.The Segment Tree also displays an icon of the new pipe solid.
- **Make**—Creates a new pipe solid that appears in the Pro/ENGINEER window.
- **Erase**—Reverts the pipe solid to a pipeline segment and erases all parts from the session.
- **Copy From**—Goes to a template directory of Piping template parts from which you choose a part.
- —Sets the chosen **Start Part** to all selected pipe segments.

### About Naming a Pipe Solid Segment

All of the pipelines in the active assembly filter out existing pipe solids and allow the part name and Start Part to be modified.

The system automatically assigns a default name for pipe solids based on the segment name.

### To Name a Pipe Solid Segment

1. Choose **PIPING > Fabrication > Pipe Solid**. The **Pipe Solid** dialog box opens.
2. Under **Active Assembly**, select the active assembly with the arrow. In the **Segment** column, all of the segments in the piping assembly appear.
3. Click on one of the segments, for example, PL2001. The **Make** and **Erase** buttons get activated.
4. Type a new name in the **SOLID NAME** box and click **OK**. The pipe segment becomes a pipe solid and updates automatically in the Pro/ENGINEER window. The dialog box closes.

### To Turn Off the Display of Piping Solids

1. Click **PIPING > Set Up > Set Display**.
2. Select the pipeline.

3. From the **PIPE DISPLAY** menu, click **Center Line**. This will turn off the display of solids.

To prevent the display of piping solid centerlines, set the configuration file option `pipe_solid_centerline = NO` or place the solid on a layer at the part level and blank it. The piping solid remains, but the system does not display the dashed center axis.

If you prefer to see the solid centerline, set this option to `YES` and control its display using **Wireframe**, **Hidden Line**, and **No Hidden** in the top menu bar.

### About Setting the Display of Pipeline Solids

On the **PIPE DISPLAY** menu, you can simplify the display by turning off piping solids. In some cases, depending on the assembly, this can dramatically improve repaint times.

### To Make Solid Square Pipes

1. Open the **LineStock** dialog box with the **PIPING > Pipeline > Create/Route** command.
2. Under **General Parameters**, select the type, material, and the grade of the line stock.
3. Under **X Section**, select **Rectangular** and under **Section Type**, select **Solid**. An example of a solid, square pipe appears under **Section Parameters**.
4. You can change the height, weight, and angle of the pipe in the boxes.
5. Click **Save** and **OK**.

### Making Solid Square Pipes

You can create solid or hollow pipes with a rectangular or square shape. This functionality allows the use of piping for square or rectangular tubing for furniture and some HVAC ductwork. You can also create a solid or hollow circular shape.

Use the **LineStock** dialog box to create solid or hollow pipes with a rectangular or square shape.

### About Extraction of a Model

You can retrieve a collection of contiguous pipe solids, fittings, and/or assembly members as a part in Part mode. However, you must first extract a model from the current assembly.

When extracting a model, the system creates a part out of designated components, merging them by reference to preserve the associativity between the new part and the piping assembly. Therefore, whenever the piping system changes in the working assembly, the extracted model also changes.

After you have created the extracted part, you can store it to disk as an individual model. The system also stores it as a component within the piping assembly. You

can use it to create views in drawings without retrieving an entire piping assembly. However, you cannot show dimensions in a drawing or obtain piping report information.

By extracting a model, you can retrieve a merged part that may contain piping solids, fittings, and equipment as well as any other assembly components that you choose to include.

### To Extract a Model

1. Click **PIPING > Fabrication > Extract Mdl > Extract**.
2. Assign a part name by entering a name.
3. Select contiguous line segments and equipment components.
4. When you have finished, click **Done Sel**. The system informs you that it has created a merged part containing all selected components.

The merged part overlays the original assembly component and may affect shaded display; therefore, you may want to place it on a layer and blank it.

### Piping Template Parts

#### About Piping Template Parts

With Pro/PIPING you can now specify a template part that will be used for the creation of pipe solids. The piping template part may contain datum features, parameters, saved views, and colors, but cannot contain any geometry features. This functionality also helps you to orient piping features while in Drawing mode.

Set the `start_model_dir` configuration file option to specify a complete path to the directory where template parts and assemblies can be stored. This directory will be the default directory. When you select a Piping template part from which to copy, the **File Open** dialog box goes to this directory by default.

This functionality also helps you to orient piping features in Drawing mode.

The **Pipe Solid** dialog box supports this functionality. In this dialog box, you can perform the following functions:

- Assign a standard template part with the **Copy From** command.
- Pipe solid parts have default names based on the pipeline name that can be Renamed in the **Solid Name** dialog box.
- Create the pipe solid with the **Make** or **OK** command.
- Remove the pipe solid with the **Erase** command.

The Pro/ENGINEER window updates immediately when you use the **Make** and **Erase** commands. The dialog box also shows a sample of a solid pipe after you click **Make**.

## To Use a Piping Template Part

1. In the **Pipe Solid** dialog box (**Piping > Fabrication > Pipe Solid**), under **Active Assembly**, click **Select Assembly** to select a pipeline segment in the assembly. The pipeline segment is highlighted.
2. Under **Segment Tree** the tree structure and the names of the pipe segments in the assembly you have selected appear.
  - **SEGMENT**—Segment names in the piping assembly
  - **SOLID NAME**—New name that you have changed from the original Piping solid part name.
  - **START PART**—Name of the original part that is in the template directory.
  - **Copy From**—Goes to a template directory of Piping template parts from which you choose a part.
3. Click **Make** to create a new pipe solid that appears in the Pro/ENGINEER window.
  - **SOLID NAME**—Allows you to change the name of the Piping solid part to a new pipe solid name before you make the pipe solid.
  - —Sets the chosen Piping template part to all selected pipe segments.

## Using the Pipe Solid Dialog Box

The **Pipe Solid** dialog box performs several functions:

- Creates a pipe solid from a segment in an assembly
- Uses a Start Part in the **Copy From** box to create a new pipe solid
- Creates a pipe solid with a new name in the **SOLID NAME** column

The **Pipe Solid** dialog box has the following options:

- **Active Assembly**—Contains the name of the active assembly you select under **Select Assembly**.
- **Segment Tree**—Displays the tree structure and the names of the pipe segments in the assembly you have selected.
  - **SEGMENT**—Segment names in the piping assembly
  - **SOLID NAME**—New name that you have changed from the original Piping solid part name.
  - **START PART**—Name of the original part that is in the template directory.

The Segment Tree also displays an icon of the new pipe solid.

- **Make**—Creates a new pipe solid that appears in the Pro/ENGINEER window.
- **Erase**—Reverts the pipe solid to a pipeline segment and erases all parts from the session.

- **Copy From**—Goes to a template directory of Piping template parts from which you choose a part.
- —Sets the chosen **Start Part** to all selected pipe segments.

### Copying from Piping Template Parts

A Piping template part is a standard part which you can copy to create new parts or assemblies. Piping template parts can contain datums, color, relations, layers, saved views, and parameters subject to the following conditions:

- A Piping template part must not have any external dependencies.
- A Piping template part cannot have any solid geometry.
- A Piping template assembly must contain only assembly features.

When copying from a Piping template part, Pro/ENGINEER allows you to browse your directory structure and select a part or assembly to copy from, as appropriate.

## Drawings

### About Drawings

In Drawing mode, you can display piping items and manipulate the display of centerlines and dimensions, and you can use Pro/REPORT functionality to obtain information related to your piping assembly. This section explains how to manipulate the display of piping items in drawings, generate bend information tables, and use Pro/REPORT with Pro/PIPING.

### To Show and Manipulate Pipe Bend Notes in a Drawing

1. In Drawing mode, add an assembly to your session.
2. Click **View > Show and Erase**, and then click the Note icon in the **Show / Erase** dialog box.
3. Specify the pipe segments within which to show the bend notes by clicking **Feature, Part, View, Feature and View**, or **Part and View**. The system attaches the notes to the theoretical bend intersection if it is showing, or to the green pipeline routing centerline bend arc if it is not.

If a default text style has been set before creating the notes, the system applies it to these automatic notes.

4. Manipulate the notes by doing the following, as necessary:
  - Update their placement (after modifying the theoretical bend intersections) by clicking **View > Update > Drawing View**.
  - Blank or unblank the note leader by choosing **MOD OPTIONS > Del Ref**.

## Using the Report Pipeline Dialog Box

The **Report Pipeline** dialog box combines the traditional piping information tools with **Results** and allows you to define and use **Saved Analyses** for future use. The **Report Pipeline** dialog box displays **Info Type**, **Definition**, **Results**, and **Saved Analyses**.

The **Info Type** box displays the following information:

- General
  - Pipeline, pipe segment, and insulation under **Definition**
- Length
  - Pipeline, line stock, and segment under **Definition**
  - **Length Type** consists of pre/cut and centerline
- Bill of Materials Generated for the Active Assembly
- Bend Location
- Bend Machine
- Holes Report
- Clocking Angle
- Output File Interchange Format
- Check Pipe
  - Pipeline under **Definition**.
- Show Report Setup
- Insulation Location

The **Definition** box displays a pipe segment, pipeline, or insulation.

The **Results** box displays the name of the pipe segment, pipeline, or assembly that you selected with .



 displays the INFORMATION WINDOW with the extended information from the **Results** box.

The **Saved Analyses** box displays the names of the saved analysis files, which are based on the information in the **Results** box.

## Displaying Dimensions and Bend Notes in Drawing Mode

To display dimensions and bend notes in Drawing mode after making the pipe solid, Click **Show and Erase** from the **View** menu. You must click **View**, **Feature**, or **Feature and View** to view the solid model's dimensions or the routing feature dimensions.

You can manipulate the display of pipe solid centerlines, pipe solid dimensions, and bend notes in drawings using the following methods:

- To control the display of pipe solid centerlines in drawings,
  - Blank the centerlines by setting the configuration file option `pipe_solid_centerline` to `no`.
  - Include pipe solid centerlines in the hidden line removal process by setting the drawing setup file option `hlr_for_pipe_solid_cl` to `yes`.
- To display drawing dimensions for isometric drawings of piping assemblies, set the drawing setup file option `allow_3d_dimensions` to `yes`.

In a 3-D view, the system does not show any dimensions that are not in the view plane of the drawing unless you have set this drawing setup file option.
- To display bend notes in your piping assemblies, set the drawing setup file option `default_pipe_bend_note`.

### **About Showing Piping Parameters in Drawing Notes**

You can attach a note to a pipe centerline, a pipe solid, a pipe bend, or a pipe solid centerline.

### **To Show Piping Parameters in Drawing Notes**

1. Define a new assembly parameter.
2. Accept the default value.
3. Add a relation setting the new parameter to the desired line stock parameter using the following syntax:

```
<param_name>=<linestock_param>:fid_<LINESTOCK_ID>:  
<SESSION_ID>
```

or

```
<param_name>=<linestock_param>:fid_<LINESTOCK_NAME>:  
<SESSION_ID>
```

An example of how to set a new parameter is this:

```
param1 = OD:fid_54:7
```

where

- `param1` is a new assembly parameter
  - `54` is line stock ID
  - `7` is session ID of the assembly where the line stock resides
  - `OD` is line stock parameter
4. Regenerate the assembly to update the new parameter value.

5. Create the drawing note referencing the new parameter, for example, `&param1` to include the OD from the example above in a note.

## Showing Theoretical Bend Intersections on a Drawing

Using drawing setup file options, you can control the display of theoretical intersections and bends for bent tubes in a drawing:

- `show_pipe_theor_cl_pts` controls the centerlines and intersection points:
  - To show centerlines with bends only, specify the value as `bend_cl` (the default).
  - To show only centerlines with theoretical bend intersection points, specify the value as `theor_cl`.
  - To show both bends and theoretical intersection points, specify the value as `both`.
- `pipe_pt_shape` controls the display of the intersection points. To specify the display, set it to `cross` (the default), `dot`, `circle`, `triangle`, or `square`.
- `pipe_pt_size` controls the size of the intersection points. To change the size, modify the numerical value.

When you change the setting of any of these options in the drawing setup file, Pro/ENGINEER updates all centerline and point intersection displays in the drawing, but you must repaint or regenerate the view to update the views.

If you create a dimension to an intersection point and then change the setup options to turn off their display, the dimension remains and is associative.

To control the hidden line display of the pipe centerline in drawings, set the drawing setup file option `hlr_for_pipe_solid_cl`.

To update pipelines created in previous releases, set configuration file option `pipe_update_pre_20_int_pts = yes` to get intersection points entities. These entities are created in session by **Info > Regen Info** or when the model is retrieved.

## Installation Drawing Generation

### About Installation Drawing Generation

Piping Installation drawing shows different views of a piping model with dimensions and various annotations. All the hidden lines are removed in the Installation drawing. Installation drawing is used to guide the piping installation work.

Pro/PIPING allows you to automatically display all pipe segments in the Installation drawing using a single line or double line representation. You can insert a pipe slope symbol on the pipe segments to display the slope of pipe segments in the Installation drawing.

## To Select Display Style for Pipe Segments

Use this procedure to specify the display style for pipe segments in the Installation drawing.

1. Click **File** > **Open** and select the drawing to be activated.
2. Choose **Applications** > **Piping**.
3. Click **View** > **Pipeline Display** > **Style** from the top menu bar. The **Display Style** dialog box opens.
4. Under **Select View**, click one of the following options to select views from the drawing:
  - Click **View** to interactively select a view or multiple views from the drawing. Click  and select a view or multiple views from the drawing.
  - Click **Sheet** to automatically select all the views from the current drawing sheet.
  - Click **All Sheets** to automatically select all the views from all the drawing sheets.

The names of the selected views are displayed in the selection list.

5. Under **Select Style**, click one of the following options to specify the style for displaying pipe segments in the drawing:
  - Click **Single Line** to display pipe segments in single line representation.
  - Click **Double Line** to display pipe segments in double line representation.
  - Click **Double Line Above** to display pipelines of different sizes in different display styles. Type a value for the pipe size in the adjacent field.

**Note:** To display a pipe in double line style, its solid representation must be present in the pipeline assembly.

Pro/PIPING displays all pipelines with a size greater than the specified size in double line representation and all pipelines with a size smaller than or equal to the specified size in single line representation.

6. Click **OK** to display the pipelines in the selected views in the specified style, and close the dialog box.

Click **Cancel** to cancel the action and close the dialog box.

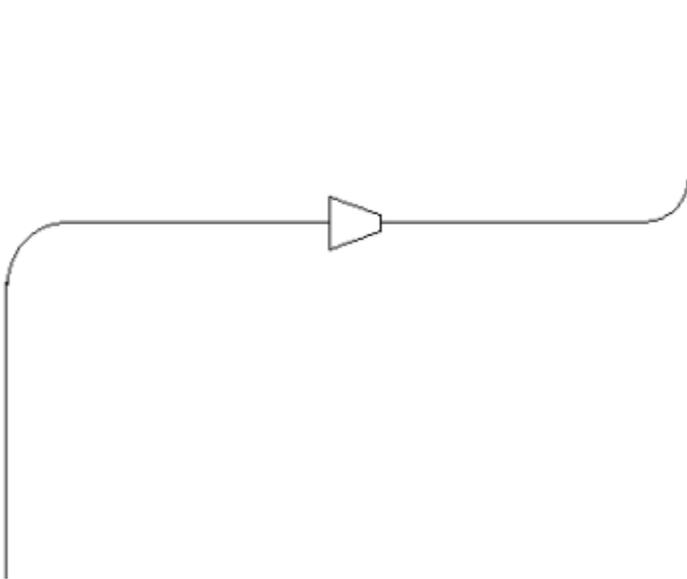
Click **Apply** to display the pipelines in the selected views in the specified style, and keep the dialog box open for further selections.

## Pipe Display Style

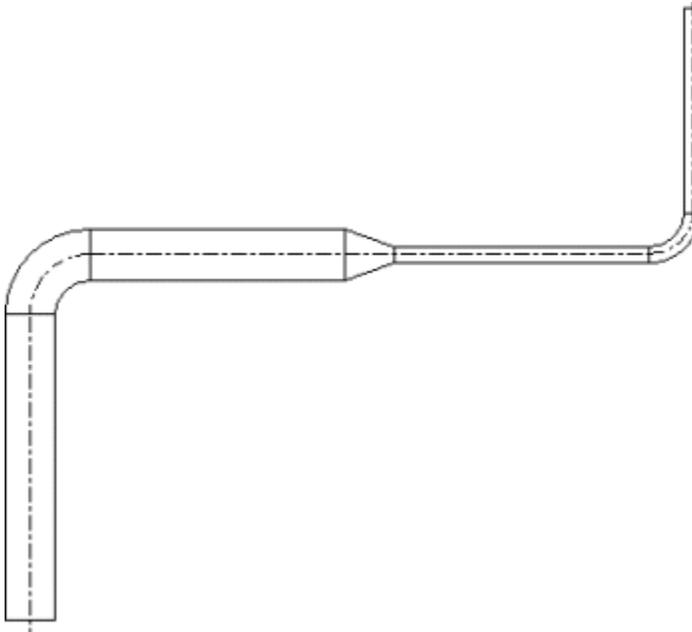
You can display pipelines in the selected views in single line representation or double line representation. You can also choose to display all pipelines with a size greater

than the specified size in double line representation and all pipelines with a size smaller than or equal to the specified size in single line representation.

The following figure shows all the pipe segments of a pipeline, irrespective of their sizes, displayed in single line representation.



The following figure shows all the pipe segments of a pipeline, irrespective of their sizes, displayed in double line representation.



### To Select Slope Symbol for Pipe Segments

Use this procedure to select a symbol for displaying the slope of pipe segments in the Installation drawing.

1. Click **File** > **Open** and select a drawing.

2. Click **Insert > Piping Symbol > Slope** from the top menu bar. The **Slope Mark** dialog box opens.
3. Under **Select View**, click one of the following options to select views from the drawing:
  - Click **View** to interactively select a view or multiple views from the drawing. Click  and select a view or multiple views from the drawing.
  - Click **Sheet** to automatically select all the views from the current sheet.
  - Click **All Sheets** to automatically select all the views from all the sheets.

The names of the selected views are displayed in the selection list.

4. Under **Select Symbol**, use one of the following methods to select a symbol for displaying the slope of pipe segments in the drawing:
  - Select a symbol from the adjacent selection list. All symbols in the current drawing are displayed in the list.

Alternatively, enter a symbol name in the adjacent field. Before loading the specified symbol, Pro/PIPING searches the directories in the following order:

The current working directory

The path specified using the `search_path` configuration option

The `symbols` directory under the project data directory specified using the `piping_project_data_dir` configuration option

- Click  to browse the directory specified by the `pro_symbol_dir` configuration option and select a symbol.

**Note:** The default symbol for pipe slope is  `slope.sym` and is located under the `$PTCSRC/text/piping_data/sample_project/symbols` directory.

5. Under **Select Symbol Parameter**, specify the parameters for the selected symbol.
  - Click **Symbol Height** to specify the height of the pipe slope symbol on the drawing. Enter a value in the adjacent field. The unit for this value is the active drawing unit.

**Note:** If you do not specify the height for the slope symbol, Pro/PIPING considers the height to be equal to the outer diameter of the pipe.
  - Click **Interval Between Symbols** to specify the distance between two consecutive pipe slope symbols. Enter a value in the adjacent field. The unit for this value is the active drawing unit.

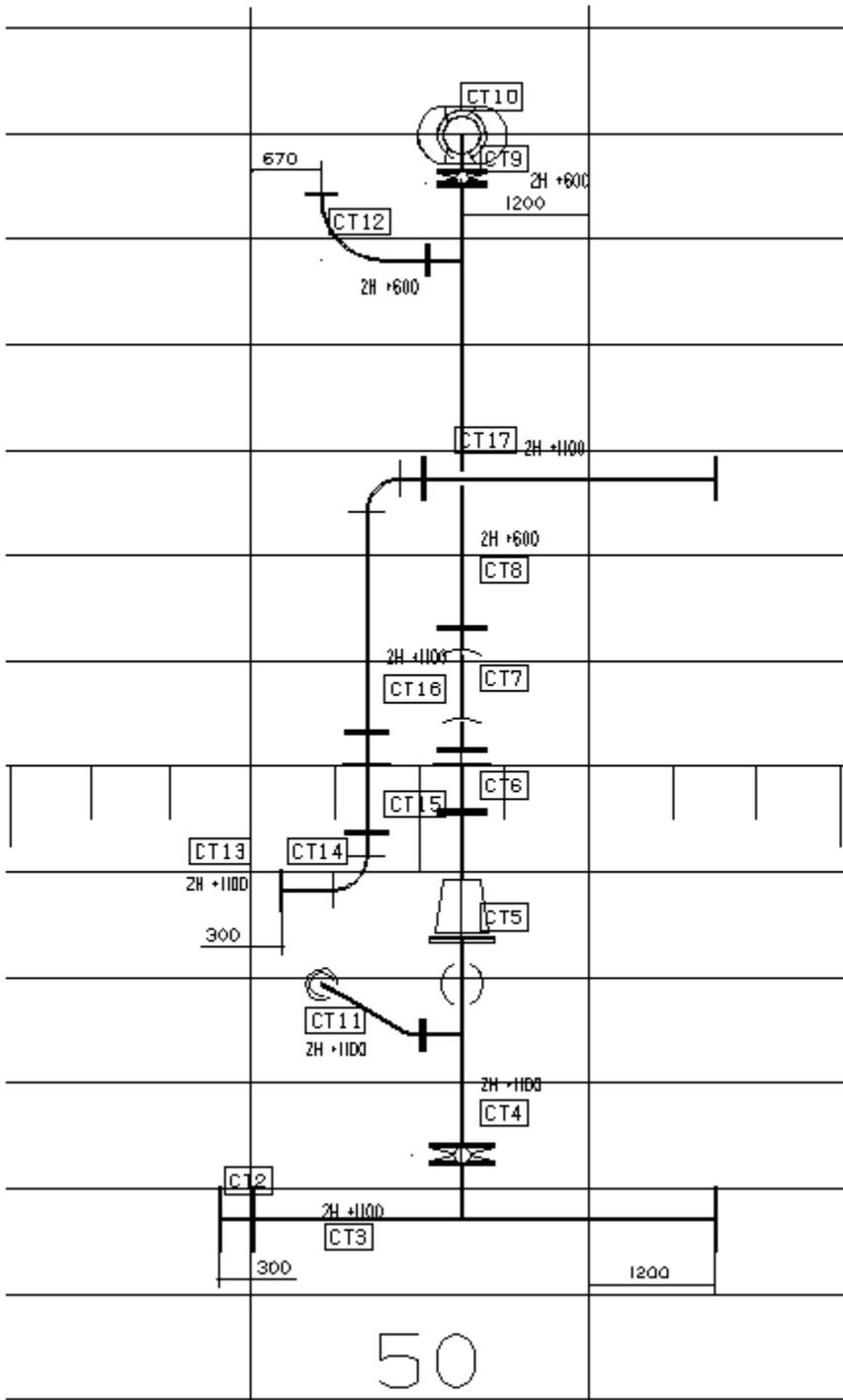
**Note:** The default value is 5 mm or its equivalent in the active drawing units.

6. Click **OK** to display the pipelines in the selected views in the specified style and close the dialog box.

Click **Cancel** to cancel the action and close the dialog box.

Click **Apply** to display the pipelines in the selected views in the specified style, and keep the dialog box open for further selections.

### Example: Piping Installation Drawing

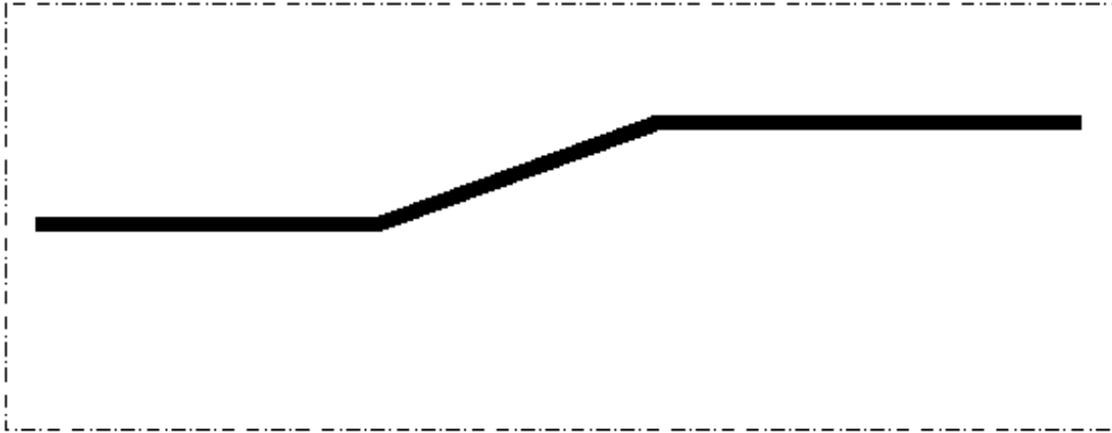


### Pipe Slope Symbol

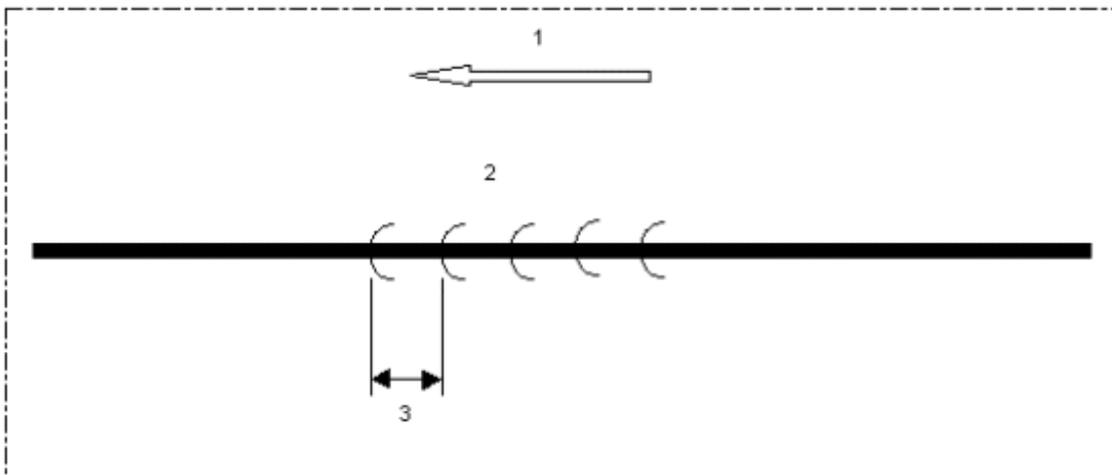
Pro/PIPING allows you to display the slope of pipelines in each view of the piping Installation drawing using a slope symbol mark.

If a pipe segment is not parallel to the selected view, the slope of the pipeline can not be shown in the selected view of the drawing. To display the slope of a pipeline, a set of slope symbols are placed along the pipe segment. The direction of the symbols indicate the slope direction.

Front View of the Pipeline



Top View of the Pipeline



1 Slope Direction

2 Slope Symbol

3 Pipe Slope Symbol Interval

The pipe slope symbol is created as a Pro/DETAIL drawing symbol. When the location of the pipe segment on the drawing is changed, the pipe slope symbol also changes its location.

When the outer diameter of the pipe segment in the model is changed, use **Regenerate > Draft** to update the diameter of the pipe slope symbol.

## Isometric Drawing Generation (ISOGEN)

### About Isometric Drawing Generation (Non Spec-Driven)

Pro/PIPING provides an Isometric Drawing Generation (ISOGEN) interface allowing you to create isometric drawings of pipelines. The ISOGEN Interface exports material and geometrical data from a Pro/PIPING model to a readable ISOGEN .pcf (Piping Component File) format. This format serves as an input to ISOGEN for creating pipeline, spool and system isometric drawings.

The **Generate Isometric** dialog box (**PIPING > Drawing**) allows you to create isometric drawings from Pro/PIPING models. Using this dialog box, you can select pipelines, set up file names and directories, and export the data to an intermediate .pcf file. ISOGEN reads this .pcf file data and generates the isometric drawings in various standard data formats.

**Note:** You must first set the ISOGEN configuration options to use the ISOGEN interface generation tool.

### To Generate an Isometric Drawing (Non Spec-Driven)

**Note:** You must first set the ISOGEN configuration options to use the ISOGEN interface generation tool.

1. Click **PIPING > Drawing**. The **Generate Isometric** dialog box opens and prompts you to select a pipeline. The **Selection** tab is selected by default.
2. Under **Select Isometric Drawing Type**, specify the isometric drawing type:
  - **Pipeline**—Generates a pipeline isometric for each selected pipeline. Pro/PIPING writes data related to each selected pipeline in a separate PCF file and generates as many PCF files as the number of selected pipelines.
  - **System**—Generates a system isometric for each selected pipeline system, which consists of physically connected pipelines. For each selected pipeline, the data for connected pipelines is exported to a PCF file. If two or more of the selected pipelines belong to one network, Pro/PIPING ignores any pipeline data that has already been exported.
3. Under **Reference Csys**, click  and select the reference coordinate system for the coordinate output in the PCF files. All the coordinates in the PCF file are based on the selected coordinate system. By default, the default coordinate system in the top assembly will be used.
4. Under **Select pipeline(s)**, click  and select one or more multiple pipelines to generate the isometric. You can select pipeline features or segment features from the Model Tree, the Piping System Tree, or the graphics window. Pro/PIPING displays all selected pipelines under **Select pipeline(s)**.
5. Under **Options**, use the following boxes to override any of the ISOGEN configuration options.

**Note:** Any changes to these four options will remain in effect throughout the Pro/ENGINEER session unless you change them during the session.

- **Output Directory**—The value of the `isogen_output_files_dir` configuration option appears in the input panel. If you want to store the PCF file in another directory, click . The **Select Directory to place output PCFs** dialog box opens. Browse for the directory and click . If there is no value for `isogen_output_files_dir`, this input panel will be blank and all files generated by the interface will be written to the working directory.
  - **Isogen File**—The value of the `isogen_mandatory_attr_file` configuration option appears in the input panel. To select a different ISOGEN file click . The **Select ISOGEN File** dialog box opens. Browse for the file and double-click.
  - **Symbol Map File**—The value of the `isogen_symbol_map_file` configuration option appears in the input panel. To change to a different symbol map file click . The **Select Symbol Map File** dialog box opens. Browse for the file and double-click.
  - **Endtype Map File**—The value of the `isogen_endtype_map_file` configuration option appears in the input panel. To change the endtype map file click . The **Select Endtype Map File** dialog box opens. Browse for the file and double-click it.
6. Click the **General Attr** tab. These attributes are supported by ISOGEN and can be used to determine various isometric drawing options.
  7. Under **Specify General Attributes**, click the **System Attr** tab. Optional ISOGEN system attributes are displayed:
    - **Date**—Specify the date. Pro/PIPING displays the current system date by default. You can change the value using the format `--/--/--`. The value entered here is written to the PCF file for the DATE-DMY ISOGEN attribute.
    - **Project Identifier**—Specify the project number or name.
    - **Revision Number**—Specify the revision number.
    - **Area** or **Batch**—Specify the sections of a project.
    - **Replot**—Specify the number of isometric drawings to be replotted.
    - **Spool Prefix**—Specify the spool prefix.
    - **Weld Prefix Erection**—Specify the erection weld prefix.
    - **Weld Prefix Fabrication**—Specify the fabrication weld prefix.
    - **Weld Prefix General**—Specify the weld prefix.

- **Weld Prefix Offshore**—Specify the offshore weld prefix.
8. Click the **User Attr** tab for ISOGEN attributes. Select from the option list to specify the user attributes. The list contains all the available user attributes supported by ISOGEN. They are MISC-SPEC1 to MISC-SPEC5 and ATTRIBUTE1 to ATTRIBUTE99. Select an attribute, specify a corresponding value, and ENTER. Pro/PIPING displays the attributes.
  9. Click the **Pipeline Attr** tab. The attributes under **Specify Pipeline Attributes** are usually applicable to one pipeline, but you can also use them for multiple pipelines. ISOGEN supports all of the following attributes. However, not every attribute is available for every Pro/PIPING model.
    0. **Pipeline Type**—Select the pipeline construction type. All the valid types are listed. Note that you should be informed about the other ISOGEN pipeline construction types before changing this value to anything other than General.
    1. **Bend Radius**—Specify the standard pulled bend radius.
    2. **Painting Spec**—Specify the painting specification.
    3. **Tracing Spec**—Specify the tracing specification.
    4. **Start Coords**—Specify the pipeline replot start coordinates.
    5. **Specific Gravity**—Specify the gravity of the pipeline contents.
    6. **Output File Name**—Specify the name of the drawing output file. The value entered is used only when a single pipeline is selected. If pipeline isometric or multiple pipelines are selected, the value is ignored.
    7. **Highest Part Number**—Specify the highest material list part number generated on the previous run of the pipeline.
    8. **Highest Weld Number**—Specify the highest weld number generated on the previous run of the pipeline.

**Note:** If you select multiple pipelines and data exists on this tabbed page (other than the default **General** in the **Pipeline Type** option), the following message appears:

Multiple Pipelines Selected. Use data in Pipeline Attr tab for all pipelines?

10. Click  to export the data or  to ignore the data. If you click , the **Output File Name** attribute is not exported.
11. Click  to apply the specified options on the selected pipelines, create the necessary PCF files, and close the dialog box.

### Component Material Description (Non Spec-Driven)

The material description of each extracted Pro/PIPING component appears in the bill of materials section of the isometric drawing. The material description in the PCF file format requires an item code and description as shown below:

MATERIALS

ITEM-CODE data

DESCRIPTION data (Single line Description)

ITEM-CODE data

DESCRIPTION data (Multiple line Description)

DESCRIPTION data

**Note:** Pro/PIPING uses the STOCKNO (stock number) as the ITEM-CODE. The STOCKNO uniquely identifies each Pro/PIPING component.

### Creating Pipe Component Information (Non Spec-Driven)

The ISOGEN drawing generation tool allows you to create pipe component information. The coordinate information of pipe segments and other components is converted to millimeters if the corresponding assembly uses metric units. If the corresponding assembly uses imperial units, the coordinate information of pipe segments and other components is converted to inches. Use the ISOGEN switches to display the units on the isometric drawing during the isometric drawing generation. The nominal size information is extracted from the Pro/PIPING component.

The ISOGEN component identifier and the information extracted is shown in the next table.

PCF Identifier	Description
BRANCH1-POINT BRANCH2-POINT	<p>Identifies branch points on various components. For example, a typical record using millimeter (mm) coordinates and an inch bore is as follows:</p> <pre>BRANCH1-POINT 125.0 25.0 0.0 6 BW</pre> <p><b>Note:</b> The data string after the identifier BRANCH1-POINT includes coordinates, a nominal size, and an endtype as expected by PCF file format.</p>
CENTRE-POINT	<p>Identifies the center point on various components. For example, a typical record using millimeter (mm) coordinates and an inch bore is as follows:</p> <pre>CENTRE-POINT 125.0 0.0 0.0</pre> <p><b>Note:</b> The data string after the identifier CENTRE-POINT includes coordinates.</p>

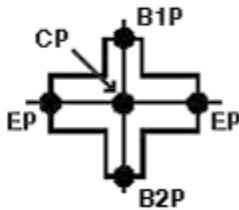
END-POINT	<p>Identifies end-points on various components. For example, a record using millimeter (mm) coordinates and an inch bore is as follows:</p> <pre>END-POINT 150.0 25.0 0.0 6BW</pre> <p><b>Note:</b> The data string after the identifier END-POINT includes coordinates, a nominal size, and an endtype as expected by the PCF file format.</p>
SKEY	<p>Represents a ISOGEN SKEY (Symbol Key). For example, a record identifying SKEY is as follows:</p> <pre>SKEY VGBW</pre>
ITEM-CODE	<p>Represents a unique identifier for each component. Pro/PIPING component STOCKNO (stock number) is used as ITEM-CODE. The ITEM-CODE and the description are used to populate the bill of materials. For example, a record identifying ITEM-CODE is as follows:</p> <pre>ITEM-CODE GATVAA050</pre>
ANGLE	<p>Identifies the angle value if the angle is other than 90 or 180 degrees for ELBOW, BEND, OLET, TEE, CROSS, and ELBOW-TEED type components. Angle values are in hundredths of degrees. For example, a record identifying an ANGLE of 60 degrees is as follows:</p> <pre>ANGLE 6000</pre>
SPINDLE-DIRECTION	<p>Direction of Spindle (Operator) on VALVE. Defines primary direction only. Valid directions values are NORTH, SOUTH, EAST, WEST, UP and DOWN. For example, a record identifying SPINDLE-DIRECTION is as follows:</p> <pre>SPINDLE-DIRECTION NORTH</pre>
FLAT-DIRECTION	<p>Direction of Flat side of Eccentric Reducers (ERED). Defines primary direction only. Valid directions values are NORTH, SOUTH, EAST, WEST, UP and DOWN. For example, a record identifying FLAT-DIRECTION is as follows:</p> <pre>FLAT-DIRECTION NORTH</pre>
DIRECTION	<p>Used to indicate VALVE Spindle direction and Eccentric Reducer Flat direction that is in primary plane. Value in a form Up 45 deg. North will be shown as a graphical output in primary direction.</p>

	<p>For example, a record identifying DIRECTION is as follows:</p> <p>Up 45 deg. North</p>
--	-------------------------------------------------------------------------------------------

**Note:** ISOGEN identifiers BRANCH1-POINT, BRANCH2-POINT, and CENTRE-POINT are specific to components and are not applicable for pipe segments.

### Example: ISOGEN Identifiers on a Component (Non Spec-Driven)

ISOGEN identifiers on a cross component are displayed in the following example.



B1P: BRANCH1-POINT

B2P: BRANCH2-POINT

CP: CENTRE-POINT

EP: END-POINT

### ISOGEN Symbol Map File (Non Spec-Driven)

The ISOGEN Symbol Map file is a Pro/TABLE file that maps a Pro/PIPING fitting generic name against a corresponding symbol key (SKEY) and the ISOGEN component name. Pro/PIPING uses symbol keys for all fitting generic part sizes. You can modify the ISOGEN Symbol Map file by using a text editor.

The ISOGEN Symbol Map file format is described in the following table.

Field Name	Field Type	Description
FITTING_GENERIC_NAME	Character	Pro/PIPING fitting generic name
ISOGEN_COMPONENT_NAME	Character	ISOGEN component name
ISOGEN_SKEY	Character	ISOGEN component symbol key (SKEY)
ISOGEN_ITEM	Character	ISOGEN component material item type. It must be one of the following or empty:x

		ERECTION-ITEM x FABRICATION-ITEM x OFFSHORE-ITEM
--	--	--------------------------------------------------------

**Note:** If no material item type is found for a fitting, Pro/PIPING checks all the end types of the fitting ports. If any of the end type is welded (an end type that contains 'W' or 'w' character) or slip on (an end type that contains the 'SO' or 'so' string), the FABRICATION-ITEM type is assigned to that fitting. Otherwise the ERECTION-ITEM type is assigned.

### Symbol Key Selection

Pro/PIPING extracts the SKEY from the ISOGEN Symbol Map file by matching the generic name of the fitting with the name in the FITTING\_GENERIC\_NAME column. Pro/PIPING extracts the symbol key name in the ISOGEN\_SKEY column and then the endtype string from the ISOGEN Endtype Map file.

SKEY selection occurs according to the following conditions:

- If the SKEY is only two characters long, Pro/PIPING appends the endtype string to the symbol key to form the final SKEY used in the PCF file.
- If the SKEY has the double asterisks (\*\*) as a substring and an endtype string is found, the endtype string replaces the double asterisks in the SKEY to form the final SKEY.
- If either of these cases fail, the default SKEY is used.

### Example: ISOGEN Symbol Map File (Non Spec-Driven)

#### Default Path:

<ProE load directory>/text/piping\_data/isodata/isogen\_symbol\_map.ptd

Fitting_Generic_Name	Isogen_Component_Name	Isogen_Skey	ISOGEN_ITEM
vl_gate_flrf	VALVE	VGFL	ERECTION-ITEM
vl_swing_check_flrf	VALVE	VSFL	ERECTION-ITEM
vl_angle_flff	VALVE-ANGLE	AVFL	FABRICATION-ITEM
flange_neck_rf	FLANGE	FLFL	FABRICATION-ITEM
elbow_90long_bw_steel	ELBOW	ELBW	FABRICATION-ITEM
elbow_45long_bw_steel	ELBOW	ELBW	FABRICATION-

			ITEM
tee_reducing_bw_steel	TEE	TEBW	FABRICATION-ITEM
tee_straight_bw_steel	TEE	TEBW	FABRICATION-ITEM
red_concentric_bw_steel	REDUCER-CONCENTRIC	RCBW	FABRICATION-ITEM
red_eccentric_bw	REDUCER-ECCENTRIC	REBW	FABRICATION-ITEM

### ISOGEN Endtype Map File (Non Spec-Driven)

The ISOGEN Endtype Map file is a Pro/TABLE file that maps Pro/PIPING end types to corresponding ISOGEN end types. You can modify the ISOGEN Endtype Map file by using a text editor.

The ISOGEN Endtype Map file format is described in the following table.

Field Name	Field Type	Description
ENDTYPE	Character	Pro/PIPING end type
ISOGEN_ENDTYPE	Character	ISOGEN end type

### Default End Type Mapping

If you do not provide an ISOGEN Endtype Map file or if you do not list the entries for the end types used by the Pro/PIPING fitting components in the Endtype Map file, Pro/PIPING maps the end types by default.

The string matching rules for default end type mapping are listed in the next table.

Endtype	Isogen Endtype
endtype string starting with B	BW
endtype string starting with C	CP
endtype string starting with F	FL
endtype string starting with P	PL
endtype string starting with SC	SC
endtype string starting with SW	SW
No match found	BW

**Note:** The default mapping is internal to the interface.

### Example: ISOGEN Endtype Map File (Non Spec-Driven)

**Default Path:**

<ProE load directory>/text/piping\_data/isodata/isogen\_endtype\_map.ptd

Endtype	Isogen_Endtype
PE	PL
FLFF	FL
GKFF	FL
FLRF	FL
SWLD	SW
BWLD	BW

### ISOGEN Nominal Size Map File (Non Spec-Driven)

The ISOGEN Nominal Size Map file is a Pro/TABLE file that maps the Pro/PIPING sizes used for modeling to the industry standard nominal size or nominal bore (NB). Nominal bore can be measured in millimeters (mm) or inches.

Pro/PIPING extracts the component size and its units from the model using the following conditions:

- If the units are metric, UNITS-BORE is selected as MM.
- If the units are in inches, UNITS-BORE is selected as INCH.

Pro/PIPING matches the extracted size against the nominal size in either the NB\_MM or NB\_INCH column. The nominal size must correspond to the model size in the SIZE column.

You can modify the ISOGEN Nominal Size Map file by using a text editor. The ISOGEN Nominal Size Map file format is described in the following table.

Field Name	Field Type	Description
SIZE	Character	Pro/PIPING Size String
NB_MM	Character	Nominal Bore (NB) Value
NB_INCH	Character	Nominal Bore (NB) Value

**Note:**

- Pro/PIPING allows you to specify any character as a nominal size. However, the ISOGEN PCF file format expects a number as the pipe nominal size.
- For NB values, use only numbers from 0 to 9, the period (.), or the slash (/).
- For NB values in millimeters, only integers can be entered.
- For NB values in inches, you can use fractions (for example, 1-1/2 for a one and one-half inch bore).
- NB values cannot include white spaces.
- For NB values, do not use the inch symbol (") for inches.

**Default Nominal Size Mapping**

If you do not list a size in the SIZE column or if you do not define the nominal size in the selected NB column, Pro/PIPING outputs the model size to the PCF file by default where NB is needed. If a nominal size map file does not exist, the same result occurs.

**Note:** An invalid nominal size may result from a wrong isometric size annotation or from an ISOGEN PCF file export malfunction.

**Example: ISOGEN Nominal Size Map File (Non Spec-Driven)**

Default Path:

```
<ProE load
directory>/text/piping_data/isodata/isogen_nominal_size_map.ptd
```

Size	Nb_inch	Nb_mm
10A	¼	10
15A	½	15
20A	¾	20
25A	1	25
32A	1-1/4	32
40A	1-1/2	40
50A	2	50
¼"	¼	10
½"	½	15
¾"	¾	20

1"	1	25
1-1/4"	1-1/4	32
1-1/2"	1-1/2	40
2"	2	50

For the complete ISOGEN nominal size map file, see the default path.

### ISOGEN Mandatory Attribute File (Non Spec-Driven)

The ISOGEN Mandatory Attribute file contains attributes that correspond to the ISOGEN mandatory attributes listed in ISOGEN-FILES. The configuration option `isogen_mandatory_attr_file` sets the ISOGEN Mandatory Attribute File name. The default is `isogen.flc`. You can use the default file provided by the ISOGEN package or create a file.

### Example: ISOGEN Mandatory Attribute File (Non Spec-Driven)

#### Default Path:

<ProE load directory>/text/piping\_data/isodata/isogen.flc

Attributes	Directory Path
PLOT FILE-PREFIX	C:\PISOGEN\PROJECTS\abc\SPOOLS\OUTPUT\PLOT
OPTION-SWITCHES-LONG	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.OPL
MESSAGE	C:\PISOGEN\PROJECTS\abc\SPOOLS\OUTPUT\SPOOLS.MES
DRAWING-FRAME	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.DXF
DRAWING-DEFINITION	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.DDF
ALTERNATIVE-TEXT	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.ALT
POSITIONED-TEXT	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.POS

WELDING- DEFINITION	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.WDF
MATERIAL- LIST- DEFINITION	C:\PISOGEN\PROJECTS\abc\SPOOLS\SPOOLS.MLD

### ISOGEN Error Log File (Non Spec-Driven)

If Pro/PIPING encounters errors while processing pipelines during isometric drawing generation, Pro/PIPING creates a log file named `isointf.err`. The error log file contains a collection of error logs from multiple sessions. The `isogen_output_files_dir` configuration option sets the directory in which Pro/PIPING stores this log file.

Pro/PIPING creates the following two types of error log entries:

- **Errors**—Identifies problem pipeline segments, components, and fittings that caused the errors. Pro/PIPING highlights these items in red and displays them in the Pro/ENGINEER graphics window. To restore these components to their original color, click .
- **Warnings**—Identifies issues that occur due to processing errors.

### Example: ISOGEN Error Log File (Non Spec-Driven)

Date : 25-4-2000

ERROR LOG file for **\*\*ISOMETRIC DRAWING GENERATION INTERFACE\*\***

---

ERROR : Processing Fitting TEE\_RED\_BW-STEEL-65X50 in Pipeline 65A-SPEC-STEAM-1.

WARNING : Material Description not found for 3 item(s) in Pipeline 65A-SPEC-STEAM-1.

---

ERRORS : ISOGEN may fail to process intermediate output PCF files.

REPAINT to restore Highlighted components in case of error.

WARNINGS : ISOGEN may not fail to process but some of the desirable data may be missing.

This file may be appended with errors occurring in future sessions.

---

## Piping Information

### About Gathering Piping Information

Piping information is available from the main Pro/ENGINEER menu. When you choose **Info > Piping**, the **Report Pipeline** dialog box opens. If you are in the Piping

module, choose **Piping > Info**, and the same dialog box opens. In this dialog box you can perform the following actions:

- Show general information for a pipe segment or a pipe line.
- Display the length of pipelines, pipe segments, or line stocks.
- Generate a bill of materials (BOM) for the active assembly.
- Generate bend location information for a selected segment.
- Generate bend machine information for a selected segment.
- Generate holes report information for a selected segment.
- Calculate the clocking angle between two csys on a flexible pipe segment.
- Generate a File Interchange Format file (FIF) for a selected segment.
- Check a pipeline for line stock parameter violations.
- View and remove designated report setup information.
- Use the **Saved Analyses** tool to name, retrieve, and designate piping objects for report setup.

### To Obtain Piping Information

Click **PIPING > Info**. The **Report Pipeline** dialog box opens.

From the main Pro/ENGINEER menu, you can also click **Info > Piping** to obtain the same results.

### Obtaining Engineering Information

Through Pro/PIPING, you can obtain various forms of engineering information concerning piping systems. Specifically, you can do the following:

- Determine the length of pipelines and segments, including the length of each stock type.
- Generate manufacturing information for a selected pipe, as well as generate information about  $xyz$  bend locations.
- Generate a Bill of Materials listing each pipe solid and fitting, including a breakdown by pipeline.
- Access information about a pipeline such as the name of the working assembly, the pipeline, and the current line stock.
- Access information about a pipeline segment such as the name of the working assembly, pipeline, and current line stock, the material, the diameter, the length, and the wall thickness.
- Check for line stock violations.

## Using the Report Pipeline Dialog Box

The **Report Pipeline** dialog box combines the traditional piping information tools with **Results** and allows you to define and use **Saved Analyses** for future use. The **Report Pipeline** dialog box displays **Info Type**, **Definition**, **Results**, and **Saved Analyses**.

The **Info Type** box displays the following information:

- General
  - Pipeline, pipe segment, and insulation under **Definition**
- Length
  - Pipeline, line stock, and segment under **Definition**
  - **Length Type** consists of pre/cut and centerline
- Bill of Materials Generated for the Active Assembly
- Bend Location
- Bend Machine
- Holes Report
- Clocking Angle
- Output File Interchange Format
- Check Pipe
  - Pipeline under **Definition**.
- Show Report Setup
- Insulation Location

The **Definition** box displays a pipe segment, pipeline, or insulation.

The **Results** box displays the name of the pipe segment, pipeline, or assembly that you selected with .



 displays the INFORMATION WINDOW with the extended information from the **Results** box.

The **Saved Analyses** box displays the names of the saved analysis files, which are based on the information in the **Results** box.

## General Information

### About Obtaining Pipeline Information

Using the **Info Type** option in the **Report Pipeline** dialog box, you can select the pipeline you want. All the pertinent information about this pipeline appears in the **Results** box.

### To Obtain Pipeline Information

1. In the **Report Pipeline** dialog box, click **Info Type > General**
2. Under **Definition**, click **Pipe Line**.
3. Click  in the **Pipe Line** option.
4. Select a pipeline in the Pro/ENGINEER window.

The **Results** box for the selected pipeline provides the name of the working assembly, the pipeline, and the current line stock.

5. Click . The INFORMATION WINDOW opens and displays the pipeline information.

### About Obtaining Pipeline Segment Information

The **Segment** option in the **Report Pipeline** dialog box provides information about a selected pipeline segment.

If you click **Info Type > General**, you can obtain the name of the working assembly, pipeline, and current line stock, the material, the diameter, the length, and the wall thickness. Height and width are also shown for square pipes.

You can also obtain the clocking angle between two fittings on a flexible pipe segment, and create bend machine or locations tables. The **Clocking Angle** and **Bend Information** table defines the kind of information you can obtain for a particular type of segment.

### To Obtain Pipeline Segment Information

1. From the **PIPING** menu, click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Under **Info Type**, click **General** or **Length** and under **Definition**, click **Segment**.
3. Click  to select a segment. Pipeline segment information appears in the **Results** window.

4. Click . The INFORMATION WINDOW opens and displays the pipe segment information.

### Clocking Angle and Bend Information

Type of Segment	Bend Information	Clocking Angle
All flexible	NO	YES
Flexible with bends or a miter cut	NO	NO
Flexible with straight sections	NO	YES
All bends	YES	NO
All miter cuts	NO	NO
Single straight segment	NO	NO
Bend/miter cut mix	YES	NO

### Pipe Length Information

#### About Obtaining Pipe Length Information

Using the **Length** option in the **Report Pipeline** dialog box, you can determine the length of pipelines and segments, including the length of each stock type.

#### To Obtain Pipe Length Information

- In the **Report Pipeline** dialog box, under **Info Type**, click **Length**.
- Under **Definition**, click **Pipe Line**, **Line Stock**, or **Segment**.
  - If you click **Pipe Line**, select a name from the **LINE NAMES** menu, and click **Done Sel**. Go to Step 3.
  - If you click **Segment**, click  and select a segment from the screen. Go to Step 3.
  - If you click **Line Stock**, select a name from the **STOCK NAMES** menu, and click **Done Sel**. The system displays the stock length information in the **Results** box.
- Using commands in the **Length Type** box, do one of the following:
  - Click **Pre-Cut** to calculate the true pipeline length, the green centerline length plus the extra needed for miter cuts and branches. The system calculates the farthest extent of the solid along the direction of the pipeline

to take into account miter cuts or branch cuts. Therefore, for precut length, it provides the minimum length of stock that should be purchased to make the piping, including complex cut geometry.

- Click **Center Line** to calculate the pipeline length along its centerline, the length of the green centerline of the selection (segment or entire line) minus any cut-back operations for inserted fittings or branches. The length is always equal to the length of the green centerline of the branch to the reference centerline.



4. Click . The INFORMATION WINDOW opens and displays the requested information.
5. To save the information from the **Results** box and create a data file, click **Saved Analyses**, type a name for the file, and click **Save**. You can retrieve the saved analysis with the **Retrieve** button or delete it after you have closed the dialog box. The file is saved with an extension `.dat` in the working directory.

## Piping Bill of Materials

### About Generating a Piping Bill of Materials

Using the **Info Type > Bill of Materials** option in the **Report Pipeline** dialog box, you can generate a Piping Bill of Materials for the active assembly that lists pipelines, solids, lengths, stocks, parts, and assembly components. You may have to scroll down in the window to see all of the information.

### To Generate a Piping Bill of Materials

1. Click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Click **Info Type > Bill of Materials**.
3. The **Results** window shows the piping BOM information.
4. Click . The INFORMATION WINDOW opens. This window contains the piping BOM information table that displays the assembly name, quantity, pipeline or segment name, line stock, length, and part name.

## Piping Bend Information

### About Bend Machine Information

When you collect data for pipe segment Bend Machine information, Pro/ENGINEER automatically selects the coordinate system at the beginning of the segment where the measurements are taken.

Bend Machine information generates a table of information about the bend number, offset, bend angle, bend radius, and twist angle. This information is based on the structure of the pipe segment and depends on the direction and start point.

### To Generate a Bend Machine Table

1. Click **Piping > Info**. The **Report Pipeline** dialog box opens.
2. Under **Info Type**, click **Bend Machine**.
3. Under **Definition**, click  and select a pipe segment.
4. Click the **Designate for Report Setup** box to make the information available in Pro/REPORT. The bend machine information is displayed under **Results**.



5. Click . The INFORMATION WINDOW opens. This window contains the bend machine information table which displays the bend number, offset, twist angle, bend angle, and bend radius.
6. Under **Saved Analyses**, specify a name for the saved information and click **Save**. The name appears in the **Saved Analyses** box.

**Note:** You can regulate the number of decimal places displayed under **Results** by specifying the value for the `default_dec_places` configuration option. The default value is 2.

### To Generate a Bend Location Table

1. Click **Piping > Info**. The **Report Pipeline** dialog box opens.
2. Under **Info Type**, click **Bend Location**.
3. Under **Definition**, click  to select a pipe segment and a coordinate system. The bend machine information is displayed in the **Results** box .
4. Click the **Designate for Report Setup** check box to make the information available in Pro/REPORT.



5. Click . The INFORMATION WINDOW opens. This window contains the bend location information table that displays the bend number, bend angle, bend radius, and the XYZ coordinates.
6. Under **Saved Analyses**, type a name for the saved information and click **Save**. The name appears in the **Saved Analyses** box.

**Note:** You can regulate the number of decimal places displayed under **Results** by specifying the value for the `default_dec_places` configuration option. The default value of this configuration option is 2.

## About Setting Up Bend Information for Use in Pro/REPORT

To generate bend tables, set up report information in the assembly in which the pipe segment was created. The information in the table references this predefined information from that assembly.

To avoid generating blank bend information in a table, first use **Show Report Setup** in the piping assembly to define which information should be available in the drawing tables. Because information appears for each assembly level, you must filter the table by level to display information about a subassembly only.

When you use **Show Report Setup** to create or select a coordinate system, keep in mind the following restrictions:

- The selected coordinate system must be contained in the same assembly within which the pipe segment was created. You may need to copy a coordinate system from a different subassembly in order to reference the correct position. You can copy one from another coordinate system and have the report information reference the copy.
- The z-axis of the coordinate system must lie along the starting line segment of the pipe. The direction of the z-axis must be the same as that of the starting line segment of the pipe.

## To Set Up Piping Bend Information for Use in Pro/REPORT

1. To link the coordinate system to the correct pipe segment, click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Click **Info Type > Show Report Setup**.
3. Select a segment with .
4. Select a coordinate system in the active assembly or create a new one. The **Results** box displays the piping segment bend information for Pro/REPORT and the bend location information.

You can select multiple coordinate systems for each type of bend information. To set up bend information for another segment or for additional coordinate systems, repeat Steps 2 and 3.

The information marked by clicking the **Designate for Report Setup** check box while generating the bend machine or location table and the holes report is available in the **Names Designated for Report Setup** box.

5. Click . The INFORMATION WINDOW with the piping segment bend information for Pro/REPORT and the bend location information opens. You can now access this information in a Pro/REPORT table in Drawing mode.
6. Create a drawing with a repeat region table, adding the appropriate pipe segment parameters.
7. Set the table to the appropriate active assembly, set the attributes to **Recursive**, and update the table.

## Holes Report Information

### About Generating Holes Report Information

Using the **Holes Report** option in the **Report Pipeline** dialog box, you can generate holes report information for a selected segment. Holes report information is generated only for the holes that are placed on the straight pipe section. The holes that intersect the bend pipe section are not reported. The holes to be reported must have the following:

- Circular cross section.
- The axis of the hole should be normal to the pipe centerline.

The holes report is generated for the holes created by:

- Branch sections of the considered pipe segment
- Hole and Cut features affecting pipe solid geometry

### To Generate Holes Report Information

1. Click **Piping > Info**. The **Piping Info** dialog box opens.
2. Under **Info Type**, click **Holes Report**.
3. Under **Definition**, select a pipe segment.

Check the **Designate for Report Setup** box to make the information available in Pro/REPORT.

4. The holes report information is displayed in the **Results box**. Click . The INFORMATION WINDOW opens and the holes report information is displayed.
5. Under **Saved Analyses**, enter a name for the saved information and click **Save**. The name appears in the **Saved Analyses** box.

**Note:** You can regulate the number of decimal places displayed under **Results** by specifying the value for the `default_dec_places` configuration option. The default value of this configuration option is 2.

### Setting Up Holes Report Information for Use in Pro/REPORT

To generate holes report, set up report information in the assembly in which the pipe segment was created. The information in the table references this predefined information from that assembly.

To avoid generating blank holes report, first use **Show Report Setup** in the piping assembly to define which information should be available.

When creating or selecting a coordinate system when using **Show Report Setup**, the selected coordinate system must lie within the same assembly within which the pipe segment was created or in any of its subassemblies. You may need to copy a coordinate system from a different subassembly in order to reference the correct

position. You can copy one from another coordinate system and have the report information reference the copy.

### To Set Up Holes Report Information for Use in Pro/REPORT

1. To link the coordinate system to the correct pipe segment, click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Click **Info Type > Show Report Setup**.
3. Select a segment with .
4. Select a coordinate system in the active assembly or create a new one.

The information marked by checking the **Designate for Report Setup** box while generating the holes report is available in the **Names Designated for Report Setup** box.

5. The **Results** box displays the holes report information.
6. Click . The INFORMATION WINDOW with the piping segment bend information for Pro/REPORT and the holes report information opens. You can now access this information in a Pro/REPORT table in Drawing mode.
7. Create a drawing with a repeat region table, adding the appropriate pipe segment parameters.
8. Set the table to the appropriate active assembly, set the attributes to **Recursive**, and update the table.

### Clocking Angle Information

#### About Clocking Angle Information

The system determines the clocking, or twist, angle between fittings on the ends of a flattened hose. The selected segment may not have any bends or miter cuts, but may have straight sections included that control the tangency of the shape. This information is necessary to manufacture hoses.

You can access clocking angle information through a Pro/REPORT parameter, an evaluate feature, or by the **Report Pipeline** dialog box.

#### To Set Up Clocking Angle Information for Pro/REPORT Tables

1. In a 3-D assembly, click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Under **INFO TYPE**, click **Clocking Angle**.
3. Under **Definition**, click **Segment** and select a pipe segment with flexible portions.

You cannot select a segment with a bend or miter cut.

4. Click **1st Coordinate System** to select the first coordinate system. The coordinate system origin must be coincident with the highlighted end of the selected pipe segment, and the z-axis of the coordinate system must be aligned pointing inward along the pipe segment from that end.
5. Click **2nd Coordinate System** to select the final coordinate system. It must lie at the highlighted end of the specified segment. If an evaluate feature does not exist for the clocking angle for the segment, the system creates one.

An evaluate feature is described in the Part Modeling online Help.

6. Check the **Designate for Report Setup** box. The **Saved Analyses** box opens.

Specify the name you want and click .

7. The system displays an INFORMATION WINDOW and writes it to a file. In a drawing table, use parameters under report parameter using `clk_ang`.

### Accessing Clocking Angle as a Pro/REPORT Parameter

Using pipeline and pipe segment Pro/REPORT parameters, you can access clocking angle information specific to particular segments.

### Accessing Clocking Angle as a System Parameter

When you set up clocking angle information for use in Pro/REPORT, the system automatically creates an evaluate feature in the active assembly containing the pipe segment. The measurement name is `clk_ang_id##`, where `id##` is the ID number of the evaluate feature.

The session ID is of the parent assembly in which the clock angle evaluate feature is created. You can access the clocking angle as a system parameter by making notes, relations, and other parameters evaluate on the following:

```
clk_ang:fid_<EVALUATE_ID>:<SESSION_ID>
```

or

```
clk_ang:fid_<EVALUATE_NAME>:<SESSION_ID>
```

### Clocking Angle and Bend Information

Type of Segment	Bend Information	Clocking Angle
All flexible	NO	YES
Flexible with bends or a miter cut	NO	NO
Flexible with straight sections	NO	YES
All bends	YES	NO

All miter cuts	NO	NO
Single straight segment	NO	NO
Bend/miter cut mix	YES	NO

### Example: Clocking Angle

This is an example of how to show this value in a 3-D note:

1. Create a clock angle feature in piping mode.
2. Create an assembly parameter, clockangle, that is driven by the following relation:

```
clockangle = clk_ang:fid_30:57
```

where

30 is the evaluate\_ID

57 is the session\_ID

3. Create an assembly 3-D bend note: The clock angle is &clockangle.

A `seg_name` feature parameter is created in the evaluate feature to identify its parent segment. Upon regeneration, the system updates an evaluate feature for a particular segment of pipe to a new value if the segment changes.

You can redefine the feature by choosing either:

**Feature > Redefine > References > Segment**

or

Under **Info**, click **Clocking Angle** and under **Definition**, click **Segment** in the **Report Pipeline** dialog box.

### To Access the Clocking Angle as General Information

1. Click **Applications > Piping > Info**. The **Report Pipeline** dialog box opens.
2. Under **Info Type**, click **Clocking Angle** and under **Definition**, click  and select a pipe segment in the Pro/ENGINEER window.
3. Select a pipe segment with flexible portions.

You cannot select a pipe segment with a bend or miter cut.

4. Select or create a coordinate system at the highlighted end of the segment.
  - o The coordinate system origin must be coincident with the highlighted end of the selected pipe segment.
  - o The z-axis of the coordinate system must be aligned pointing inward along the pipe segment from that end.

- The coordinate system should belong to the active assembly.
5. Select the final coordinate system at the highlighted end of the selected segment, using the same constraints as before.

The system displays an INFORMATION WINDOW and writes the information to a file.

### Calculating Clocking Angle Between Fittings

Within Pro/ENGINEER, you can obtain information about the clocking angle between fittings in a flexible hose. When two end fittings of a flexible pipe (or hose) are assembled to a hose and the hose is taken from its routed state to a flattened state, the fittings must be assembled to the hose correctly twisted relative to each other. This is done so that they will fit in the assembly properly without requiring the hose itself to twist. Many fittings are not axially symmetric and require a specific orientation when they are assembled.

The system calculates the clocking angle by sliding the initial coordinate system along the flexible pipe segment in the orientation of least energy until its origin is coincident with the final coordinate system at the other end of the segment. It then determines the angle between the two x-axes of the coordinate systems.

Using Pro/ENGINEER, you can access this information as general information or as a system parameter. You can also show clocking angle information using Pro/REPORT parameters.

### File Interchange Format Information

#### About Output FIF

You can generate a file interchange format (FIF) file for a selected pipeline segment that contains Bend information. This FIF file is formatted with the necessary commands to drive an Eagle Eaton Leonard bending machine.

#### To Generate an FIF File

1. In the **Report Pipeline** dialog box under **Info Type**, click **Output FIF**.
2. Click the **Segment** arrow. The **SELECT** menu appears.
3. Select a pipe segment in the Pro/ENGINEER window.
4. The **Results** box lists the file interchange format (FIF) bend information for that segment.

5. To view the results in a larger window, click .

## Check Pipe Information

### About Checking Pipelines for Line Stock Violations

Use the **Check Pipe** option in the **Report Pipeline** dialog box to determine if any violations of the corresponding line stock exist in selected pipelines. By isolating potentially problematic areas, you can take the necessary measures to prevent failures. The system compares the characteristics of the selected pipeline with the values of the parameters set in the line stock file, and then highlights all locations on the pipe curve where violations occur.

### To Check a Pipe for Line Stock Violations

1. Click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Under **Info Type**, click **Check Pipe**.
3. Under **Definition**, select a pipe line to check.
4. The **Results** box shows any **Line Stock** parameter violations it has detected.

The system highlights the pipeline in the following cases:

- If the violation applies to the entire pipeline.
- If the violation applies to certain geometry of the pipe, for example, a bend.



You can also click  to open the INFORMATION WINDOW. This window displays the Line Stock name, the parameter name, the value of the parameter, and the value of the violation (if there is one). You can save, edit, and search in this information window.

### Using the Report Pipeline Dialog Box

The **Report Pipeline** dialog box combines the traditional piping information tools with **Results** and allows you to define and use **Saved Analyses** for future use. The **Report Pipeline** dialog box displays **Info Type**, **Definition**, **Results**, and **Saved Analyses**.

The **Info Type** box displays the following information:

- General
  - Pipeline, pipe segment, and insulation under **Definition**
- Length
  - Pipeline, line stock, and segment under **Definition**
  - **Length Type** consists of pre/cut and centerline
- Bill of Materials Generated for the Active Assembly
- Bend Location

- Bend Machine
- Holes Report
- Clocking Angle
- Output File Interchange Format
- Check Pipe
  - Pipeline under **Definition**.
- Show Report Setup
- Insulation Location

The **Definition** box displays a pipe segment, pipeline, or insulation.

The **Results** box displays the name of the pipe segment, pipeline, or assembly that you selected with .



 displays the INFORMATION WINDOW with the extended information from the **Results** box.

The **Saved Analyses** box displays the names of the saved analysis files, which are based on the information in the **Results** box.

## Report Setup

### About Using Show Report Setup

You can use Pro/REPORT functionality with Pro/PIPING to view and remove report setups.

When you use **Show Report Setup** in a piping assembly, remember the following:

- The system creates pipe segment bend information and stores it in the piping assembly. You do not generate the bend information directly in the drawing. However, you can access it automatically by using the Pro/REPORT functionality in Drawing mode after you use **Show Report Setup** in Assembly/Piping mode. You can format the Pro/REPORT table in any way that you click.
- To create a piping report, be sure to set the repeat region attribute to **Recursive**.
- To create a report that contains only piping information in the top-level assembly, set the **Recursive** attribute in the repeat region and add the filter  `[&rpt.level==1]`.
- You cannot use report parameters that contain more than 32 characters in report relations, for example, as filters for what is shown in a table.

## To Show Report Setup

1. Click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Under **Info Type**, click **Show Report Setup**.
3. Select a pipe segment in the Pro/ENGINEER window.
4. To view reports which have been defined for bend machine, bend location, or clocking angle information, see the **Names Designated for Report Setup** box.



If you click , the INFORMATION WINDOW opens.

## Saved Analyses

### About Saved Analyses

The **Saved Analyses** function allows you to gather piping information with selected object references (segment, csys, and so on). You can easily name and retrieve them for subsequent analysis, and designate them for report setup.

### To Save an Analysis

1. In the **Report Pipeline** dialog box, save the information from the **Results** box.
2. Click **Saved Analyses** and type a name for the file with an extension ".dat" in the working directory.
3. Click **Save**. After you have closed the dialog box, you can retrieve the saved analysis.
4. In the **Saved Analyses** box, select the name and click **Retrieve**.

You can also save the file again or delete it.

### To Retrieve Object Selections

You can retrieve a saved analysis with the **Retrieve** button in the **Report Pipeline** dialog box. You can also delete the saved analysis in the dialog box. The file is saved with an extension .dat in the working directory.

## Pro/REPORT with Pro/PIPING

### About Pro/REPORT Functionality in Pro/PIPING

You can use Pro/REPORT functionality with Pro/PIPING to create a customized table or Bill of Materials (BOM) for a piping assembly. A piping assembly drawing is the same as a drawing of any other Pro/ENGINEER object, except that it uses special parameters. The customized table updates automatically to reflect changes in the piping assembly. You can customize drawing tables to reflect the standard practices of your company.

When you use Pro/REPORT functionality in a piping assembly, keep in mind the following:

- The system creates pipe segment bend information and stores it in the piping assembly. You do not generate the bend information directly in the drawing. However, you can access it automatically by using the Pro/REPORT functionality in Drawing mode after you use **Show Report Setup** in Assembly/Piping mode. You can format the Pro/REPORT table in any way that you choose.
- To create a piping report, be sure to set the repeat region attribute to **Recursive**.
- To create a report that contains only piping information in the top-level assembly, set the Recursive attribute in the repeat region and add the filter `[&rpt.level==1]`
- You cannot use report parameters that contain more than 32 characters in report relations, for example, as filters for what is shown in a table.

### To Set Up Piping Bend Information for Use in Pro/REPORT

1. To link the coordinate system to the correct pipe segment, click **PIPING > Info**. The **Report Pipeline** dialog box opens.
2. Click **Info Type > Show Report Setup**.
3. Select a segment with .
4. Select a coordinate system in the active assembly or create a new one. The **Results** box displays the piping segment bend information for Pro/REPORT and the bend location information.

You can select multiple coordinate systems for each type of bend information. To set up bend information for another segment or for additional coordinate systems, repeat Steps 2 and 3.

The information marked by clicking the **Designate for Report Setup** check box while generating the bend machine or location table and the holes report is available in the **Names Designated for Report Setup** box.

5. Click . The INFORMATION WINDOW with the piping segment bend information for Pro/REPORT and the bend location information opens. You can now access this information in a Pro/REPORT table in Drawing mode.
6. Create a drawing with a repeat region table, adding the appropriate pipe segment parameters.
7. Set the table to the appropriate active assembly, set the attributes to **Recursive**, and update the table.

## About Setting Up Bend Information for Use in Pro/REPORT

To generate bend tables, set up report information in the assembly in which the pipe segment was created. The information in the table references this predefined information from that assembly.

To avoid generating blank bend information in a table, first use **Show Report Setup** in the piping assembly to define which information should be available in the drawing tables. Because information appears for each assembly level, you must filter the table by level to display information about a subassembly only.

When you use **Show Report Setup** to create or select a coordinate system, keep in mind the following restrictions:

- The selected coordinate system must be contained in the same assembly within which the pipe segment was created. You may need to copy a coordinate system from a different subassembly in order to reference the correct position. You can copy one from another coordinate system and have the report information reference the copy.
- The z-axis of the coordinate system must lie along the starting line segment of the pipe. The direction of the z-axis must be the same as that of the starting line segment of the pipe.

## Pro/REPORT Parameters Applicable to Pro/PIPING

### Linestock Parameters

Parameter Name	Definition
&asm.mbr.pipe.stock.name	Lists name(s) of line stock(s).
&asm.mbr.pipe.stock.User Defined	Lists specified parameters for the line stock.
&asm.mbr.pipe.stock.line.name	Lists names of all pipelines using each line stock.
&asm.mbr.pipe.stock.line.User Defined	Lists parameters for the pipeline using the specified line stock.
&asm.mbr.pipe.stock.material	Lists material for all line stocks.
&asm.mbr.pipe.stock.OD	Lists outer diameter for line stocks.
&asm.mbr.pipe.stock.wallthick	Lists wall thickness for line stocks.

&asm.mbr.pipe.stock.len.pre_cut	Lists total precut length of each line stock.
&asm.mbr.pipe.stock.len.center	Lists total centerline length of each line stock.

### Pipeline Parameters

Parameter Name	Definition
&asm.mbr.pipe.line.segment.stock.name	Lists names of line stocks for each segment in the pipeline.
&asm.mbr.pipe.line.segment.stock.User Defined	Lists parameters for line stocks of which the pipeline is made.
&asm.mbr.pipe.line.name	Lists names of all pipelines.
&asm.mbr.pipe.line.User Defined	Lists parameters for each pipeline.
&asm.mbr.pipe.line.stock.name	Lists names of line stocks used in each pipeline.
&asm.mbr.pipe.line.stock.User Defined	Lists parameters for line stocks used in pipeline.
&asm.mbr.pipe.line.len.pre_cut	Lists precut length of each pipeline.
&asm.mbr.pipe.line.len.center	Lists centerline length of each pipeline.
&asm.mbr.pipe.line.segment.name	Lists names of all segments in pipeline.
&asm.mbr.pipe.line.segment.solid_name	Lists names of all solids (if defined) in pipeline.
&asm.mbr.pipe.line.segment.len.pre_cut	Lists precut length of segments in pipeline.
&asm.mbr.pipe.line.segment.len.center	Lists centerline length of segments in pipeline.

<code>&amp;asm.mbr.pipe.line.segment.clk_ang.ang</code>	Lists clocking angle value from evaluate feature.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.cs1</code>	Lists name of initial coordinate system for each clocking angle.
<code>&amp;asm.mbr.pipe.line.segment.clk_ang.cs2</code>	Lists name of final coordinate system for each clocking angle.

### Pipe Segment Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.name</code>	Lists system default name of the pipeline segments.
<code>&amp;asm.mbr.pipe.segment.solid_name</code>	Lists solid part name for each segment (if defined).
<code>&amp;asm.mbr.pipe.segment.stock.name</code>	Lists line stock used in each segment.
<code>&amp;asm.mbr.pipe.segment.stock.User Defined</code>	Lists parameters in line stocks assigned to each segment.
<code>&amp;asm.mbr.pipe.segment.line.name</code>	Lists name of pipeline to which each segment belongs.
<code>&amp;asm.mbr.pipe.segment.line.User Defined</code>	Lists pipeline parameters for each segment.
<code>&amp;asm.mbr.pipe.segment.len.pre_cut</code>	Lists precut length of each segment.
<code>&amp;asm.mbr.pipe.segment.len.center</code>	Lists centerline length of each segment.
<code>&amp;asm.mbr.pipe.segment.segfrom</code>	Lists location from which each pipeline segment in the assembly is routed. This is the component to which the <i>from</i> entry port belongs.

<code>&amp;asm.mbr.pipe.segment.segto</code>	Lists the location to which each pipeline segment in the assembly is routed. This is the component to which the <i>to</i> entry port belongs.
<code>&amp;asm.mbr.pipe.segment.clk_ang.ang</code>	Lists clocking angle value of evaluate feature.
<code>&amp;asm.mbr.pipe.segment.clk_ang.cs1</code>	Lists name of initial coordinate system for each clocking angle.
<code>&amp;asm.mbr.pipe.segment.clk_ang.cs2</code>	Lists name of final coordinate system for each clocking angle.

### Pipeline Segment Bend Information Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_angle</code>	Lists bend angle.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_num</code>	Lists bend location bend number.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_radius</code>	Lists bend radius.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_x</code>	Lists x-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_y</code>	Lists y-offset bend location information.

Pro/PIPING (Non Spec-Driven) - Help Topic Collection

<code>&amp;asm.mbr.pipe.segment.csys.bend.loc_z</code>	Lists z-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_bend</code>	Lists bend machine bend angle.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_num</code>	Lists bend machine bend number.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_offset</code>	Lists bend machine offset.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_radius</code>	Lists bend machine bend radius.
<code>&amp;asm.mbr.pipe.segment.csys.bend.mach_twist</code>	Lists bend machine twist angle.
<code>&amp;asm.mbr.pipe.segment.csys.name</code>	Lists coordinate system name for segment bend information.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_bend</code>	Lists bend machine bend angle.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_num</code>	Lists bend machine number.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_offset</code>	Lists bend machine offset.

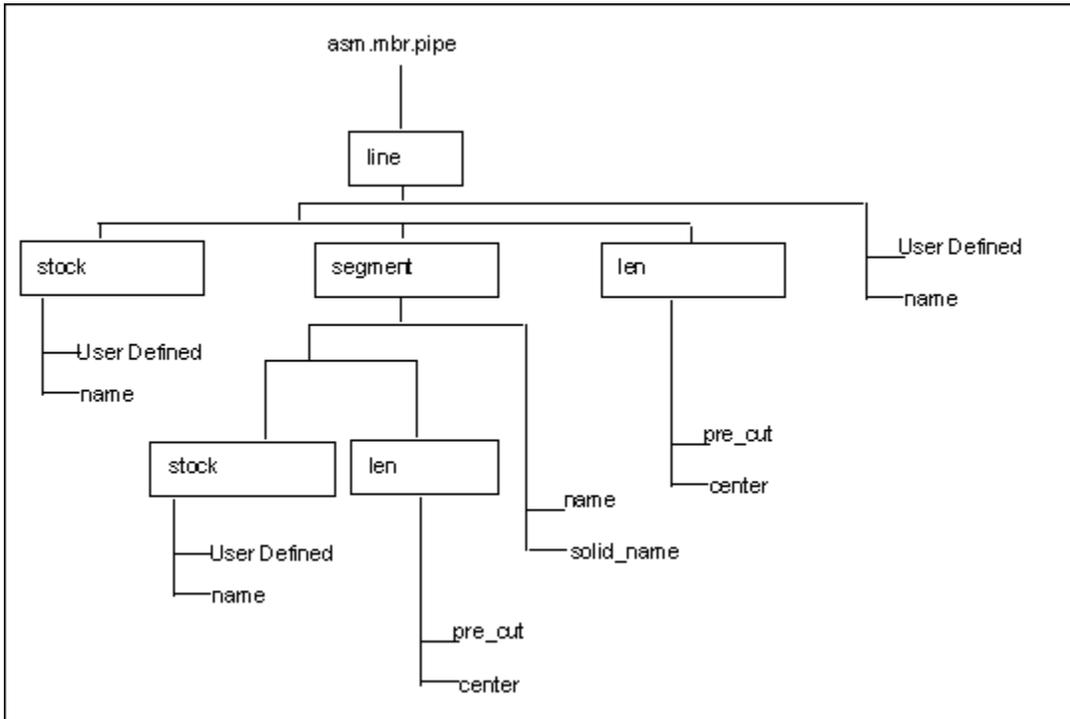
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_radius</code>	Lists bend machine radius.
<code>&amp;asm.mbr.pipe.segment.mach_csys.bend.mach_twist</code>	Lists bend machine twist angle.
<code>&amp;asm.mbr.pipe.segment.mach_csys.name</code>	Lists name of coordinate system with bend machine information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.User Defined</code>	Lists a user-defined bend parameter.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.bend_name</code>	Lists the bend name created by the system.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_angle</code>	Lists bend location bend angle.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_num</code>	Shows bend location bend number.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_radius</code>	Lists bend location bend radius.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_x</code>	Lists x-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_y</code>	Lists y-offset bend location information.
<code>&amp;asm.mbr.pipe.segment.loc_csys.bend.loc_z</code>	Lists z-offset bend location information.

<code>&amp;asm.mbr.pipe.segment.loc_csys.name</code>	Lists coordinate system name for segment bend location information.
------------------------------------------------------	---------------------------------------------------------------------

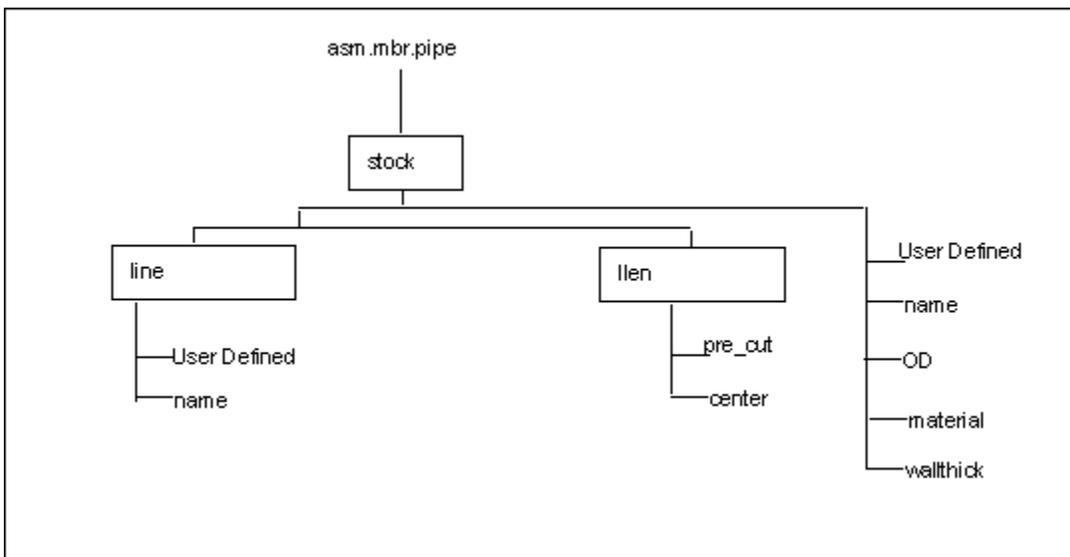
### Holes Report Parameters

Parameter Name	Definition
<code>&amp;asm.mbr.pipe.segment.holes.holes_params.hole_num</code>	List holes report number.
<code>&amp;asm.mbr.pipe.segment.holes.holes_params.hole_offset</code>	List holes report offset.
<code>&amp;asm.mbr.pipe.segment.holes.holes_params.hole_angle</code>	List holes report angle.
<code>&amp;asm.mbr.pipe.segment.holes.holes_param..hole_diam</code>	List holes report diameter.

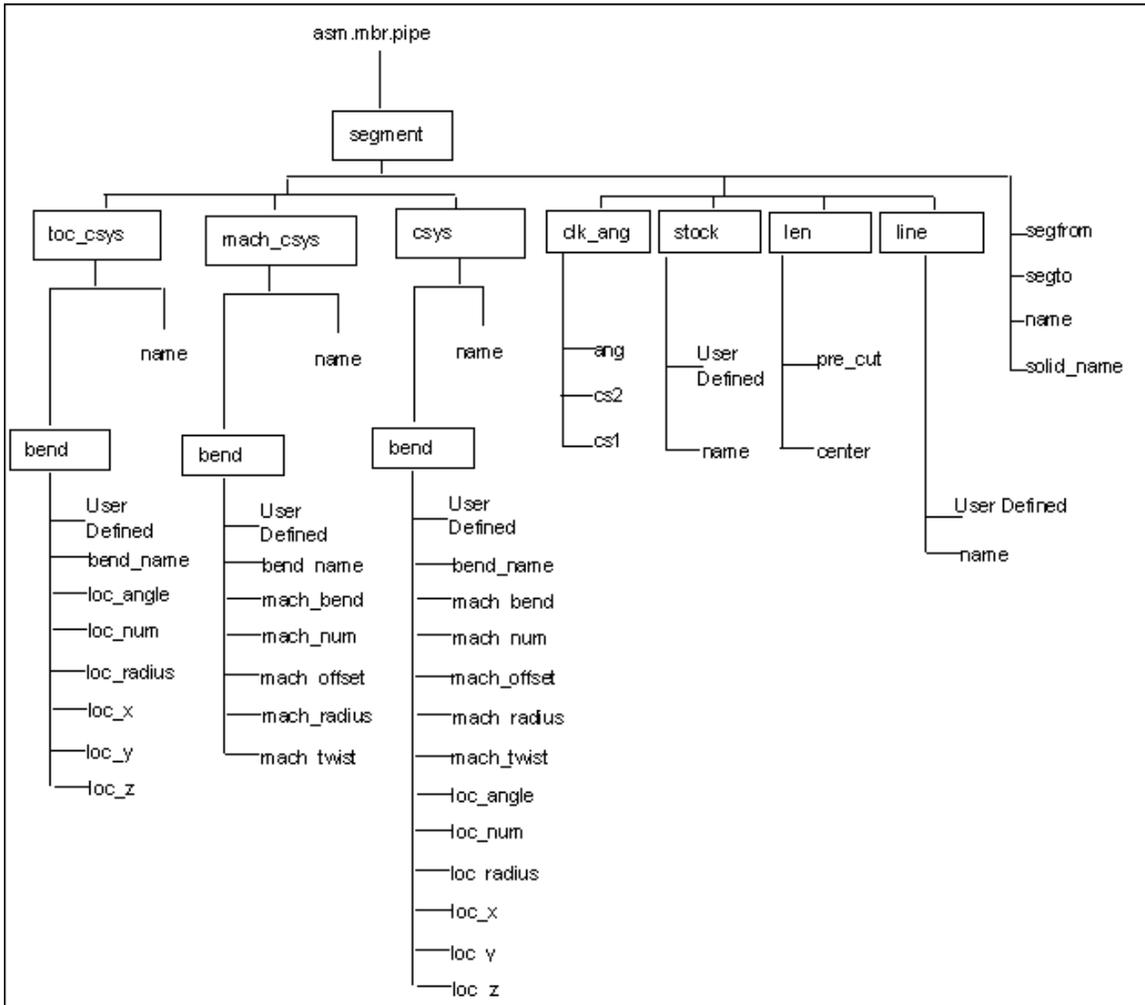
**Pro/REPORT Pipeline Parameters**



**Pro/REPORT Pipe Stock Parameters**



### Pro/REPORT Pipe Segment Parameters



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