
PROCAD SPOOLCAD User Guide

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PROCAD SPOOLCAD

Features

PROCAD SPOOLCAD provides a complete selection of piping components. This includes pipe, flanges, fittings, valves, and supports. The drawings are done in schematic format and are not to be scaled. All material and dimensional data is stored in material information blocks. You can dimension the isometric, generate a Bill of Material, and transfer data to the AutoDATA program for further processing.

PROCAD SPOOLCAD's commands are structured to flow logically and assist you by displaying useful messages.

Modular Programming

PROCAD SPOOLCAD uses module functions for maximum memory usage efficiency. The application loads general program files at start-up. The system loads all other functions when you invoke them.

English and Metric Units Operation

You can apply *working* and *display* unit settings to your standard. Use ***Standard Manger*** to set these units.

Working units: refers to length and coordinates information used to place components.

Display units: refers to units that appear when selecting component sizes.

Available **Working Units** include:

- **English:** length and coordinates information displayed in imperial units
- **Metric:** length and coordinates information displayed in metric units

Available **Displayed Units** include:

- **English - US:** component sizes displayed in US English units (example, 4")
- **English - NPS:** component sizes displayed in English units using standard nominal pipe size designations (example, NPS 4)
- **Metric - OD:** component sizes displayed in metric units using standard pipe OD size designations (example, 114.3mm)
- **Metric - NPS:** component sizes displayed in metric units using standard nominal pipe size designation (example, 100mm)

Specification Driven

Pipe specifications constitute the foundation of PROCAD SPOOLCAD. When you place a piping component in a drawing, PROCAD SPOOLCAD attaches the specification information to that component.

PROCAD SPOOLCAD also uses specifications to ensure appropriate placement of components. For example, if you insert a butt weld fitting when the current pipe size is in the screwed/socket weld range, an error message appears, informing you of the discrepancy.

You can deactivate or override specs when necessary.

Customization

You can customize many of the application's default settings to conform to specific standards, including layers, component sizing, and text heights.

Online Messages

PROCAD SPOOLCAD displays messages in the command line and/or dialog boxes.

Repeating Commands

As in AutoCAD, you can repeat many commands by pressing the **Enter** key or the space bar.

Error Detection

Every function contains error detection to prevent you from entering incorrect data. When you enter erroneous input, the system alerts you and prompts you for different input.

Correct Fitting Type Placement

The spec file determines the 'small fittings' boundary between butt weld fitting and screwed (or socket weld) fittings. This allows PROCAD SPOOLCAD to automatically select the appropriate fitting type.

This feature also affects valves.

Automated Layering

PROCAD SPOOLCAD places components in designated layers. For example, it places valves in the VALVES layer and instruments in the INSTR layer. You can modify the layering system to conform to their layering standards. Use the Standard Manager to set the layers.

Dimensional Data Library

The [Network]\Datafiles\Libraries directory contains all the dimensional data files used by PROCAD SPOOLCAD. You can modify or expand these files for non-standard components or sizes.

Program Toggles

You can change or check application settings using System Toggles. The toggles let you control many of the application's settings.

Program Overrides

Application overrides allow you to change the settings during the drafting session. These overrides apply to the current session only. The system resets them when you load the next drawing.

Software Updates and Technical Support









When properly installed, your PROCAD SPOOLCAD or PROCAD SPOOLCAD + folder contains a **Support Web Page** icon. Click this icon and follow the *Product Updates* link to receive updates to your software program files. You can only access this feature through the Internet.

Tutorials

When properly installed, your PROCAD SPOOLCAD or PROCAD SPOOLCAD+ folder contains a **Tutorials Web Page** icon. Click this icon and select the application that you would like to view the tutorial for.




Opening the Drawing Manager

In general, the Drawing Manager will open automatically upon opening the PROCAD SPOOLCAD application. To open the Drawing Manager at any time click the **Drawing Manager** button at the top of the tabbed toolbox.

In this field...	Do this...
 (Open Standard)	Click to load the settings for the highlighted Standard. A drawing will need to be opened using AutoCAD's Open command from the File menu. Use this option when a Drawing path is not defined in the Standard.
	Click to toggle the Drawing Manager display from showing the drawings to not showing the drawings.
 (Open Drawing)	Click to load the highlighted drawing. The settings for the associated Standard will be loaded at the same time.
	Click to create a new drawing.
	Click to create a copy of the highlighted drawing.
	Click to rename the highlighted drawing.
	Click to delete the highlighted drawing.
	Click to open Windows Explorer to the folder currently selected. If a Standard is highlighted, Windows Explorer will open to the drawing location folder defined.
Help	Click to access Help for this dialog box.

Viewing the Drawing Status

To the left of the drawing names in the Drawing Manager are icons indicating the status of each individual drawing. More information about the drawing becomes available when the drawing name is highlighted.

Status Icon	Definition
	The drawing is available and ready for editing.
	The drawing is read-only. It may be opened for viewing but cannot be saved.
	The drawing is currently in use and has been locked. To find out who is currently editing the drawing, highlight the drawing name in the manager.

Starting a New Drawing

To start a new drawing...

1. Open the **Drawing Manager**.
2. Select the standard whose settings you want the new drawing to use.
3. If there are subfolders in the drawing folder location defined in the Standard, use the “+” and “-“ to navigate to the desired folder.
4. Click the **New** button.
5. Enter a name for the drawing.
6. Click **Open**. A blank drawing file appears.
7. Set the following drawing parameters: pipe spec, main pipe size, and line number.
8. Insert a drawing border. See *Inserting a Border*.

➤ **Note:** A new blank drawing will be generated if only the Standard is opened

Opening an Existing Drawing

There are two methods of opening an existing drawing. The first will load the drawing and the Drawing Standard settings in one step. The second is AutoCAD’s **Open** command from the **File** menu. This method requires that the Standard has been previously loaded.

To open an existing drawing and Standard settings...

1. Open the **Drawing Manager**.
2. Select the standard whose settings you want the new drawing to use.
3. If there are subfolders in the drawing folder location defined in the Standard, use the “+” and “-“ to navigate to the desired folder.
4. Highlight the drawing that you wish to open.
5. Double-click on the drawing name

-or-

Click the **Open Drawing** button.



-or-

Right-click and select **Open** or **Open Read Only** from the context menu.

Opening a Standard

If a drawing location has not been defined within a Drawing Standard, then the Standard must be opened before a drawing may be opened.

To open an existing drawing and Standard settings...

1. Open the **Drawing Manager**.
2. Select the standard whose settings you want to open.
3. Double-click on the standard name

-or-

Click the **Open Standard** button. 

-or-


Right-click and select **Open** from the context menu.

- **Note:** Opening a Standard on its own will load the settings and also load the standard's template in preparation for a new drawing.

Copying a Drawing

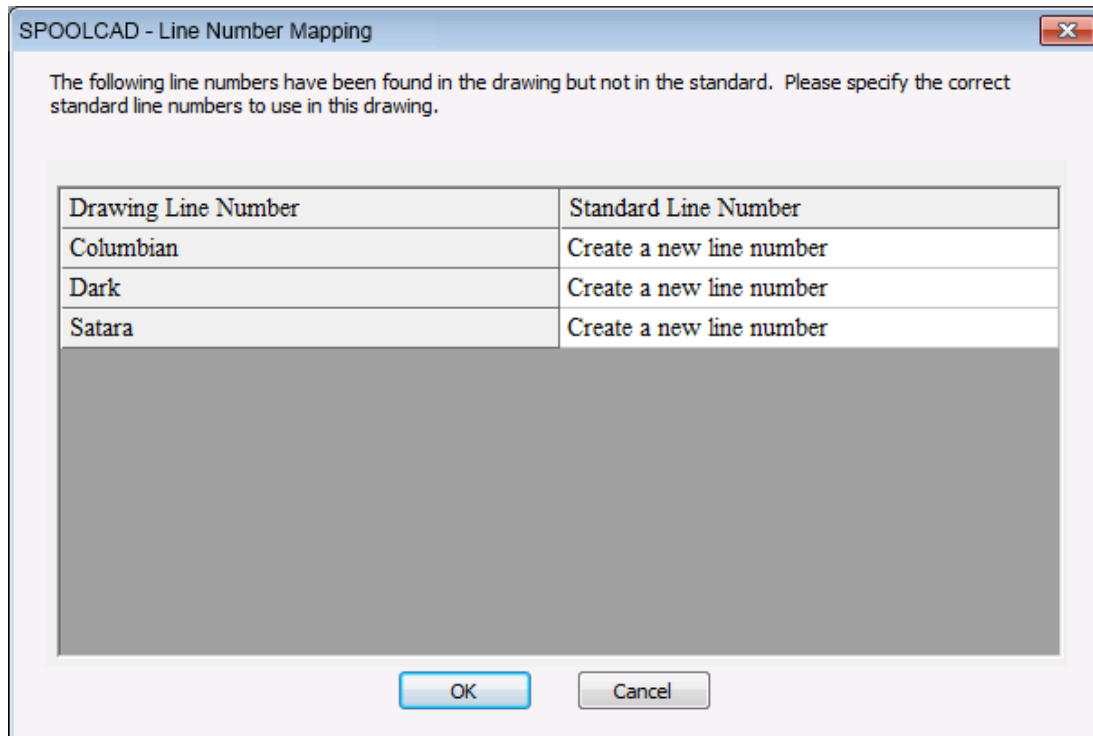
Copying a similar drawing as a base for a new drawing can save drafting time.

To copy a drawing...

1. Open the **Drawing Manager**.
 2. Select the standard where the drawing you wish to copy resides.
 3. If there are subfolders in the drawing folder location defined in the Standard, use the “+” and “-“ to navigate to the desired folder.
 4. Highlight the drawing that you wish to copy.
 5. Click the **Copy Drawing** button. 
- or-
- Right-click and select **Copy** from the context menu.
6. The new drawing will appear in the Drawing Manager. Use the Rename command to give the drawing a new name.

If you need to copy a drawing from one standard to another, open the drawing you wish to copy and use AutoCAD’s **Save As** command from the **File** menu. Drawings from previous standards contain drawing and line numbers specific to that standard.


Line numbers that exist in the drawing, but not in the standard, will be automatically added to the standard. A prompt will appear the first time that you open the drawing.



- **Note:** The user is solely responsible for ensuring that the drawing is copied and used by a Standard that is using the same working units as the original drawing. Problems with scaling will be readily apparent if this is not accounted for.

Deleting a Drawing

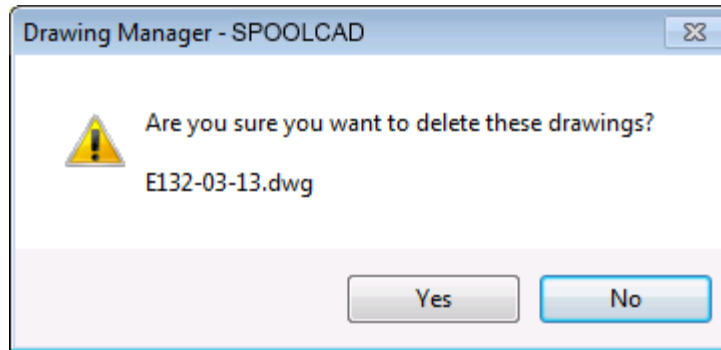
To delete a drawing...

1. Open the **Drawing Manager**.
2. Select the standard where the drawing you wish to copy resides.
3. If there are subfolders in the drawing folder location defined in the Standard, use the “+” and “-” to navigate to the desired folder.
4. Highlight the drawing you wish to copy.
5. Click the **Delete** button. 

-or-

Right-click and select **Delete** from the context menu.

A warning message will appear.



6. Click the **Yes** button.


➤ **Note:** This will permanently delete the drawing.

Restoring a DrawingTo restore a drawing...

1. Open the **Drawing Manager**.
2. Select the standard where the drawing you wish to recover resided.
3. If there are subfolders in the drawing folder location defined in the Standard, use the “+” and “-“ to navigate to the desired folder.
4. Right-click and select **Show Backups** from the context menu.
Any backup files with the extension *.bak will now appear in the drawing list.
5. Use the *rename a drawing* procedure to rename to a *.dwg extension.

Renaming a Drawing

To rename a drawing...

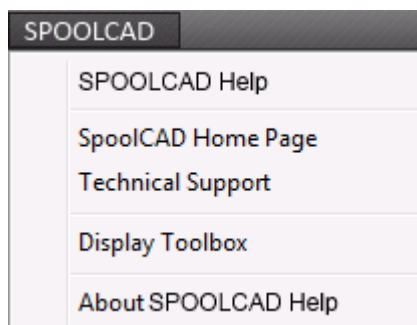
1. Open the **Drawing Manager**.
 2. Select the standard where the drawing you wish to rename resides.
 3. If there are subfolders in the drawing folder location defined in the Standard, use the “+” and “-“ to navigate to the desired folder.
 4. Highlight the drawing you wish to rename.
 5. Click the **Rename** button. 
- or-
- Right-click and select **Rename** from the context menu.

6. Type the new name for the drawing and then press **Enter**.

Navigating PROCAD SPOOLCAD

PROCAD SPOOLCAD Menu

Use the *PROCAD SPOOLCAD* menu to access these selections.



Use this menu item...	To do this...
SPOOLCAD Help	View the available online help file Note: Many dialog boxes contain a Help button; click this button to view information about that specific dialog box
PROCAD Home Page	Access PROCAD's website
Technical Support	Access PROCAD's technical support website
Display Toolbar	Display the Toolbar
Display Toolbox	Display the Toolbox The Toolbox and Toolbar contain the same tools, however, the Toolbox uses a tabbed interface that keeps the tool buttons visible at all times; this user guide refers to the Toolbox in all

	procedures
About SPOOLCAD	View software version information

Using the Tabbed Toolbox



The tabbed Toolbox provides easy access to PROCAD SPOOLCAD's tools, grouping them under tabs that keep the tool buttons visible at all times.


Accessing Tools

To insert a component or perform an operation, click the desired tab, and then click the required tool button.

Some buttons act as toggle switches. If a toggle is active, the button appears highlighted; if inactive, the button appears normal.

Display or Hide the Toolbox

To display the Toolbox, select **Display Toolbox** from the *PROCAD SPOOLCAD* menu.

To hide the Toolbox, click  in the top right-hand corner of the Toolbox.

Toolbox Status Display

Current settings for the **Spec**, **Main Size**, **Branch Size**, and **Line Number** appear in drop-down lists at the bottom of the Toolbox.

Customizing the Toolbox

You can dock the Toolbox on the left or right side of the screen, or float it over the drawing area.

To reposition the Toolbox, grab the title bar and drag it to a new location. The Toolbox automatically docks when placed close to the left or right side of the screen.

Double-click the title bar to toggle the Toolbox between docked and floating mode.

Some tabs contain more tools than what appears on the screen. To view all commands, scroll down.

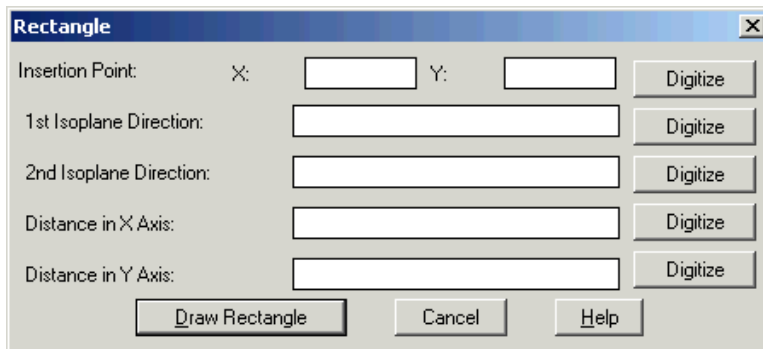
Toolbox Buttons

For a complete list of all Toolbox buttons, see *Appendix F: Button References*.

Using the Digitize Buttons

In some dialog boxes, a **Digitize** button appears next to an input box.

For example, the **Rectangle** dialog box contains both input boxes and **Digitize** buttons.



If you click **Digitize**, the dialog box temporarily disappears, and command line messages prompt you to select points within the drawing.

After you click the required point(s), the dialog box returns, and the points you selected appear in the input boxes.

When all input boxes contain values, click the **Draw Rectangle** button. The rectangle appears in the drawing.

Settings and Overrides

Component Grouping

PROCAD applications make extensive use of AutoCAD groups. Groups are very useful for managing items belonging to a component.

When you insert a drawing item, the system combines all of its components into a single selectable group. For example, when you insert a valve with a stem, the system collects these items (including the INFO block) into a single selectable group.

When you move or erase that valve, the system treats the valve, the valve stem and the INFO block as a single item.

The advantage of grouping, as opposed to creating single block, is that you can edit any item in the group without exploding that group, as is the case with blocks.

Object Grouping

To select one line or element from a group, disable Object Grouping.

To enable or disable Object Grouping...

Use the following keystroke combinations to toggle Object Grouping on or off:

Shift + Ctrl + A

Setting Specifications

When you start a new drawing, PROCAD SPOOLCAD does the following:

- Loads the **A** spec
- Sets the main pipe size
- Sets the branch pipe size

➤ Note: PROCAD SPOOLCAD does not set a line number.

The A spec settings include rating, flange face, small fittings rating, gasket thickness, WOL, and EOL weights. See *Specifications for Generic Spec A* for details. Spec A is a generic spec supplied with PROCAD SPOOLCAD.

To edit existing specs or create new specs, use *Spec Generator*. Refer to the *Spec Generator User Guide*.

Selecting a Pipe Spec

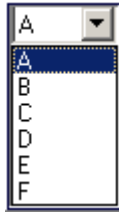
You can select a pipe spec using either of the following:

- Toolbox drop-down list
- **Select Spec** dialog box

To select a pipe spec using the Toolbox drop-down list...

1. On the Toolbox, click the arrow on the **Spec** drop-down list box.

A list of pipe specs appears.



2. In the list, click the required spec.

To select the pipe spec using a dialog box...

1. From the **Settings** tab, select **Pipe Specs**.



The **Select Spec** dialog box appears.

The current spec selection appears at the top.

2. In the list, click the spec you want to use.
3. Choose one of the following:
 - Click **OK** to set the spec and return to the drawing
 - Click **Cancel** to exit without changing the pipe spec

Selecting a Main Pipe Size

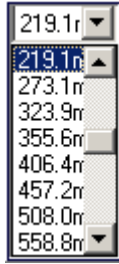
You can select a main pipe size using either of the following:

- Toolbox drop-down list
- **Select Pipe Size** dialog box

To select a pipe size using the drop-down list...

1. On the Toolbox, click the arrow on the **Main Size** drop-down list box.

A list of pipe sizes appears.



2. In the scroll list, click the required pipe size.

To select a pipe size using a dialog box...

1. From the **Settings** tab, select **Main Size**.



The **Select Pipe Size** dialog box appears.

The current main pipe size appears at the top.

2. In the scroll list, click the required pipe size.
3. Choose one of the following:
 - Click **OK** to set the pipe size and return to the drawing
 - Click **Cancel** to exit without changing the pipe size

Selecting a Branch Pipe Size

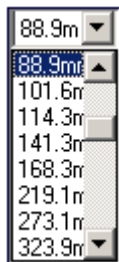
You can select a main pipe size using either of the following:

- Toolbox drop-down list
- **Select Branch Size** dialog box

To select a branch pipe size using the drop-down list...

1. On the Toolbox, click the arrow on the **Branch Size** drop-down list box.

A list of pipe sizes appears.



2. In the scroll list, click the required branch pipe size.

To select a branch pipe size using a dialog box...

1. From the **Settings** tab, select **Branch Size**.



The **Select Branch Pipe Size** dialog box appears.

The current branch pipe size appears at the top.

2. In the scroll list, select the required branch pipe size.
3. Choose one of the following:
 - Click **OK** to set the branch size and return to the drawing
 - Click **Cancel** to exit without changing the branch size

Setting a Line Number

To set a line number...

1. On the toolbox, click the arrow on the **Line Number** drop-down list box.
2. In the scroll list, click the required line number.

➤ **Note:** To create a new line number, use the Standard Manager.

To set a line number using Select Block...

1. On the **Line Number** dialog box, click **Select Block**.

The cursor changes to a square.
2. Click the info block of a component in the drawing that contains the required line number.
3. Press **Enter**.

The current line number changes to match the selected component.

Setting Spec Overrides

You can set spec overrides at any time during a drafting session.

During normal operation, PROCAD SPOOLCAD draws components that conform to the currently selected specification. Spec overrides allow you to change certain settings during the drafting session. When you open another drawing, PROCAD SPOOLCAD resets all overrides to the default value.

When you start a drafting session, all overrides are inactive except **Valve Stem**. To activate an override, click the appropriate override tool button on the **Settings** tab.

When you activate a spec override, dialog boxes appear every time you invoke the affected function. These boxes prompt you to accept the override or reset to the original spec.

Piping Spec Overrides

Use this procedure to set the following overrides:

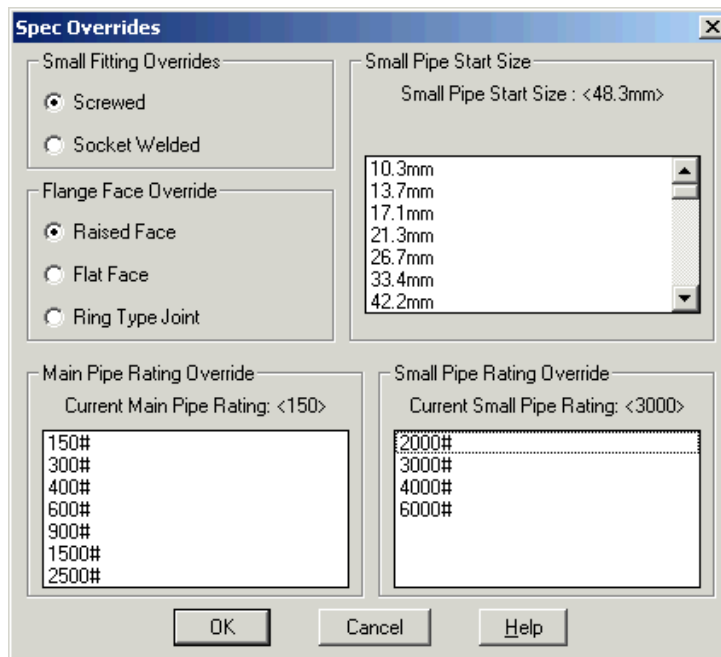
- Small fitting: screwed or socket welded
- Flange face: raised face, flat face or ring type joint
- Main pipe rating
- Small pipe start size
- Small pipe rating

To set the piping spec overrides...

1. From the **Settings** tab, select **Spec Override**.



The **Spec Overrides** dialog box appears. Current settings appear above the list boxes.



2. In this dialog box, do the following:

In this...	Do this...
Small Fitting Overrides options	Set the required small fittings override
Flange Face Override options	Set the required flange face override
Main Pipe Rating	Select the main pipe rating override value

Override list	
Small Pipe Start Size list	Select the small pipe start size override; this is the size at which PROCAD SPOOLCAD inserts fittings as screwed or socket weld instead of butt weld
Small Pipe Rating Override list	Select the small pipe rating override value

3. Choose from the following options:
 - Click **OK** to set the selected overrides
 - Click **Cancel** to exit without changing the override settings

Welded Valves Override

Use this override to insert welded valves instead of flanged valves for fittings in the butt weld size range. This override does not affect screwed and socket weld valves.

To set the welded valve override...

From the *Settings* tab, select **Welded Valves**



Valve Face to Face Dimension Override

Use this override to change the valve face-to-face dimension.

When you insert a valve, a dialog box prompts you to accept or change the valve face-to-face dimension.

PROCAD SPOOLCAD activates this override automatically if valve face-to-face dimension data does not appear in the default dimensional data file (see *Dimensional and Description Data Files*).

To set the valve face-to-face dimension override...

From the *Settings* tab, select **Valve F/F Dimension**.



Gasket Thickness Override

Use this override to change gasket thickness. When you insert a gasket, a dialog box prompts you to accept the default or type a new value.

To set the gasket thickness override...

From the *Settings* tab, select **Gasket Thickness**.



Bolt Length Override

Use this override to change the bolt length. When you insert bolts, a dialog box prompts you to accept the default value or type a new bolt length.

To set the bolt length override...

From the **Settings** tab, select **Bolt Length**.



Nipple Length Override

Use this override to change the pipe nipple length. When you insert pipe nipples, a dialog box prompts you to accept the default value or type a new value.

To set the nipple length override...

From the **Settings** tab, select **Nipple Length**.



Shop or Field Override

Use this override to change the shipping destination. When you insert a component that requires destination information, a dialog box prompts you to accept the default value or select a new destination.

To set the shop or field destination override...

From the **Settings** tab, select **Shop/Field**.



Using System Toggles

System toggles let you control how closely PROCAD SPOOLCAD conforms to the currently selected specification.

You activate or deactivate toggles using tool buttons on the **Settings** tab. If a toggle is active, the tool button appears highlighted; if inactive, the button appears normal.

Screwed by Socket Weld Valves Toggle

Use this toggle to insert a screwed by socket weld valve instead of screwed, or socket weld valve.

To use this toggle you must select **Socket Welded** in the **Small Fitting Overrides** options of the **Spec Override** dialog box. See *Inserting Screwed by Socket Weld Valves*.

After inserting a screwed by socket weld valve, this toggle deactivates automatically. You must select this toggle every time you want to insert this type of valve.

The default setting is 'off'.

To activate or deactivate the screwed by socket weld toggle...

From the **Settings** tab, select **Screwed/SW Valve**.



Spec Check Toggle

Use this toggle to override certain spec settings. When you insert affected components, dialog boxes prompt you to select from a variety of spec options associated with that component.

For example, when you insert flanged or welded ball valves, check valves and plug valves, a dialog box prompts you to select a valve type. When you insert screwed/socket weld valves, a dialog box prompts you to select a valve rating.

The default setting is 'off'.

To activate or deactivate the spec check toggle...

From the **Settings** tab, select **Spec Check**.



or

From the **Spec Check** options on the **General Drawing Parameters** dialog box, select **On** or **Off**.

Branch Check Toggle

Use this toggle to verify the branch fitting type specified in the currently selected specification.

When you insert a branch fitting, a dialog box prompts you to override the spec setting or cancel the insertion.

The default setting is 'on'.

To activate or deactivate the branch check toggle...

From the **Settings** tab, select **Branch Check**.



or

From the **Branch Check** options on the **General Drawing Parameters** dialog

Valve Stem Toggle

Use this toggle to insert valve stems at the same time you insert valves. When you insert valves, a dialog box prompts you to add a valve stem.

The default setting is 'on'.

To activate or deactivate the valve stem toggle...

From the **Settings** tab, select **Valve Stem**.



or

From the **Valve Stem** options on the **General Drawing Parameters** dialog box, select **On** or **Off**.

Material ON/OFF Toggle

Use this toggle to insert material data when you insert piping components. When you insert components, PROCAD SPOOLCAD writes all available material data into the component's INFO block.

Caution: The system uses material data to generate the Bill of Material. If you deactivate this toggle, you cannot generate a Bill of Material.

The default setting is 'on'.

To activate or deactivate the Material ON/OFF toggle...

From the **Settings** tab, select **Material ON/OFF**.



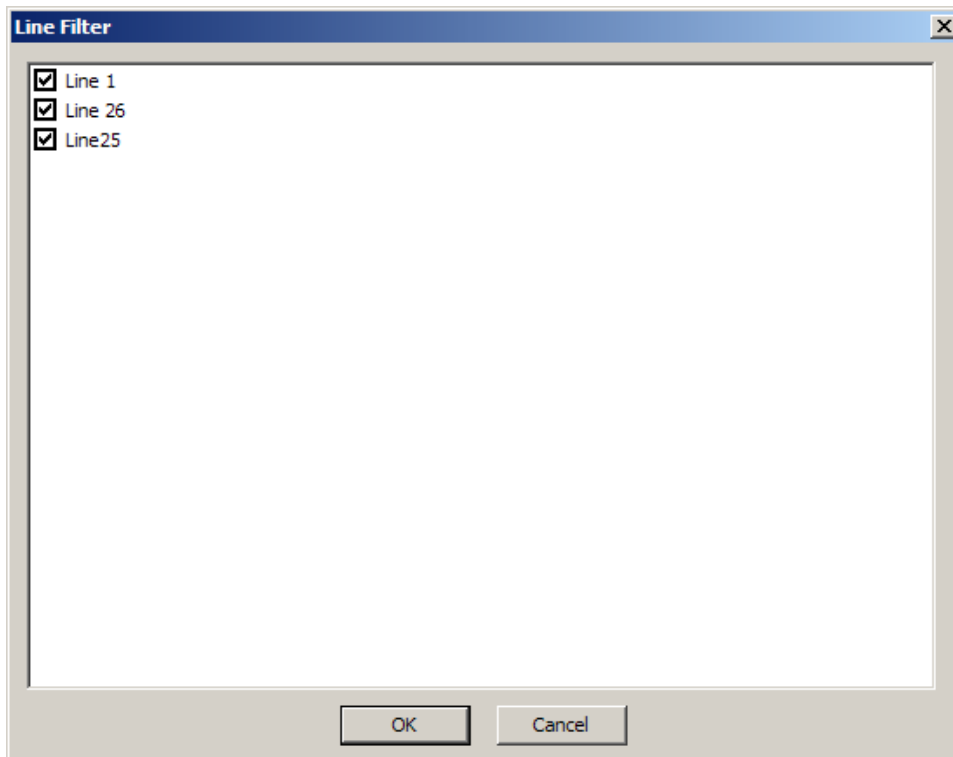
Line Filter

Use this option to filter the line numbers that are available in the **Line Number** drop-down box.

From the **Settings** tab, select **Line Filter**.



The **Line Filter** dialog box will appear displaying all Line Numbers that are currently active for the drawing's Standard. Only the checked values will appear in the **Line Number** drop-down box.



Setting the Drawing Isoplane

Isometric Plane Settings

PROCAD SPOOLCAD has seven preset isometric plane settings. Set the correct isoplane setting before inserting a component.

If you try to insert a component in a direction does not conform to current isoplane, the **Select Isoplane Setting** dialog box appears, prompting you to select a different isoplane.

PROCAD SPOOLCAD does not verify the isoplane when inserting fittings such as pipe.

The isoplane setting affects the following:

- Snap and Grid
- Crosshairs orientation
- Side orientation angles used in symbols drafting

Changing the Current Isoplane Setting

You can change the current isoplane setting using either of the following:

- Toolbox shortcut
- Tool button

- **Important Note:** Do not use the F5 key to change the isoplane orientation.

To change the isoplane setting using the Toolbox shortcut...

1. On the bottom of the Toolbox, click the isoplane button.



2. Continue clicking this button until the required isoplane setting image appears.

To change the isoplane setting using a tool button...

1. From the **Settings** tab, select **Isoplanes**.



The **Select Isoplane Setting** dialog box appears.

2. From this dialog box select one of the following options:
 - Click **Accept Current** to use the current isoplane setting
 - Click a different isoplane setting
 - Click **Cancel** to exit without changing the isoplane setting

Changing Drawing Settings

The **Change Current Settings** dialog box lets you modify the following drawing settings from one location:

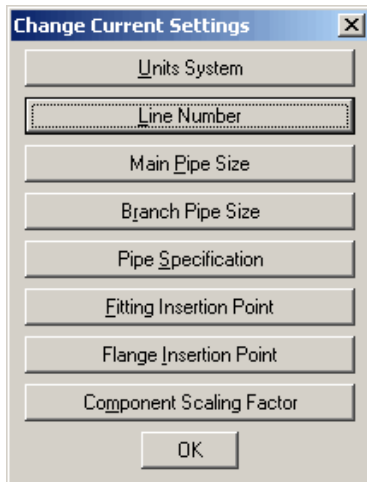
- Line Number
- Main Pipe Size
- Branch Pipe Size
- Pipe Specification
- Fitting Insertion Point
- Flange Insertion Point
- Component Scaling Factor

To change drawing settings...

1. From the **Settings** tab, select **Change Settings**.



The **Change Current Settings** dialog box appears.



2. Click the setting you want to change.
A dialog box appears listing the values or options available for the selected setting.
3. From the list of values or options, select the required setting.
4. Select one of the following options:
 - Click **OK** to change the setting and return to the **Change Current Settings** dialog box
 - Click **Cancel** to exit without changing the setting
5. On the **Change Current Settings** dialog box, click **OK** to return to the drawing.
 - **Note:** You must change the **Units System** settings using *Standard Manager*. Refer to the *Standard Manager User Guide*.

Viewing Drawing Settings

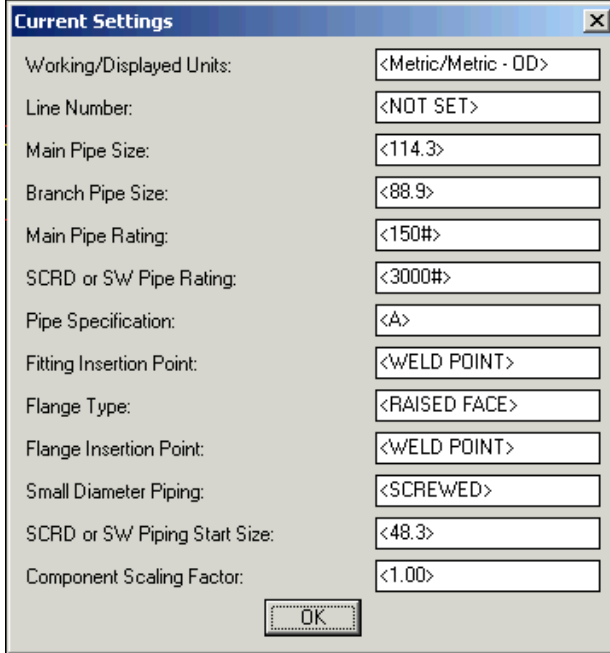
Use this procedure at any time during the drafting session to view the current settings

To view the current drawing settings...

1. From the **Settings** tab, select **List Settings**.



The **Current Settings** dialog box appears.



The 'Current Settings' dialog box contains the following fields and values:

Field	Value
Working/Displayed Units:	<Metric/Metric - OD>
Line Number:	<NOT SET>
Main Pipe Size:	<114.3>
Branch Pipe Size:	<88.9>
Main Pipe Rating:	<150#>
SCRD or SW Pipe Rating:	<3000#>
Pipe Specification:	<A>
Fitting Insertion Point:	<WELD POINT>
Flange Type:	<RAISED FACE>
Flange Insertion Point:	<WELD POINT>
Small Diameter Piping:	<SCREWED>
SCRD or SW Piping Start Size:	<48.3>
Component Scaling Factor:	<1.00>

An 'OK' button is located at the bottom right of the dialog box.

2. Click **OK** to return to the drawing.

Viewing Pipe Spec Settings

Use this procedure to view the current pipe spec settings. To edit pipe spec settings, use *Spec Generator*. Refer to the *Spec Generator User Guide*.

To view piping spec settings...

1. From the **Settings** tab, select **Pipe Specs**.



The **Select Spec** dialog box appears. The current spec selection appears at the top.

2. Click **View Spec**.

The **View Spec** dialog box appears.

3. Click **General Spec Parameters** to view service and material information.

The **View General Spec Parameters** dialog box appears. You cannot edit information in this dialog box.

4. Click **OK** to return to the **View Spec** dialog box.
5. Click **OK** again to return to **Select Spec** dialog box.
6. Click **OK** a third time to return to the drawing.

Border Drawing Blocks

Overview

Border Drawings

PROCAD SPOOLCAD supplies a set of generic border drawing files that contain the following components:

- Border line
- Title block

You can modify these files or replace them with your own border drawings.

You can also use *Standard Manager* to map to your own in-house border files. This allows you to link each standard to borders customized for that standard. Refer to the *Standard Manager User Guide* for details.

When you insert a border, PROCAD SPOOLCAD sets the drawing limits, grid, and snap settings. What needs to be done is making it so that the border is not only

Inserting Borders Blocks

Inserting a Border

When you start a new drawing, insert a drawing border before you insert piping components.

PROCAD SPOOLCAD sets the default border to size D (A1 for metric).

To insert a border drawing...

1. From the *Utilities* tab, select **Border**.



The **Drawing Size** dialog box appears.

2. From the size options, select the appropriate border drawing size.
3. Click **OK**.

The border appears in the drawing.

Adding Information to Border Blocks

To add information to generic borders blocks, use AutoCAD's text tools.

Modifying a Generic Border Drawing

You can modify any element in a generic border drawing. Keep each element on its original layer.

You can also add attributes to a generic border (refer to AutoCAD's manual on how to add attributes).

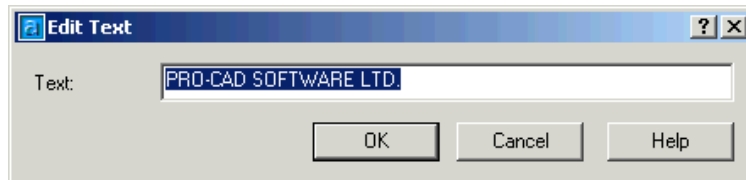
If you do not want to alter the generic border drawings, use *Standard Manager* to map to your own in-house border drawings.

Changing the Company Name

To change the company name in a generic border drawing...

1. Open a border drawing either from the Datafiles\TemplateProject\Borders or from <Project>\Borders.
2. Double-click the **PROCAD SOFTWARE** text in the title block.

The **Edit Text** dialog box appears. You can edit the existing text or delete it and add it to



3. Type your company name.
4. Click **OK**.
5. Save and close the drawing.

The next time you insert this border, your company name will appear in the title block.

Replacing Generic Border Drawings

You can replace generic border drawings with your own border files. Use the *Standard Manager* to map to your own in-house border drawings.

Dimensional and Description Data Files

Dimension Data Files

The system uses dimensional data files to determine the availability and dimension of a component. PROCAD compiles its dimensional data from catalogues supplied by major manufacturers. Dimensional data for non-standard components is captured from charts supplied by certified engineering authorities.

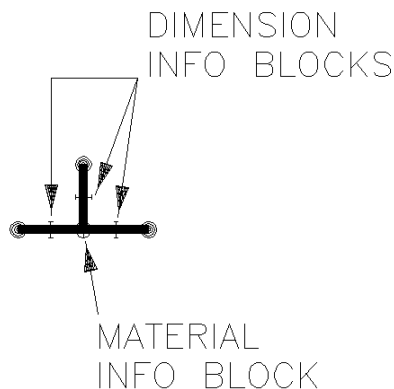
In some instances, different manufacturers supply different dimensional data for the same component. Use *Spec Generator* to review and/or edit specification information for non-standard items. Refer to the *Spec Generator User Guide* for more information.

- **Note:** 90 degree elbows, reducing elbows, and 180 degree returns do not have a dimensional data file. The system calculates the dimensions of these fittings.
- All dimensions in data files are in inches. If you using the Metric system, PROCAD SPOOLCAD converts the dimension to Metric by multiplying it by 25.4.

Dimensional INFO Blocks

The system inserts DIMENSION blocks with components that have different dimensions for different sides. In the case of fittings like a reducing tee, the center-to-end dimension for the main side is different from the center-to-end dimension of the branch.

The following illustration demonstrates how the system inserts these blocks.



PROCAD SPOOLCAD groups the dimension blocks with the component it represents. If you move a component, these blocks move with it.

These blocks contain dimensional data only. PROCAD SPOOLCAD does not use these blocks when generating a Bill of Material.

Description Data Files

Description data files are stored in the [Network]\Datafiles\Specs directory. They contain each component's description data including material, wall thickness, manufacturer, and other information. Use *Spec Generator* to modify these files. Refer to the *Spec Generator User Guide*.

Component Description INFO Blocks

PROCAD SPOOLCAD tracks component information using INFO blocks. These blocks contain the material and dimensional data of components. PROCAD SPOOLCAD uses this information for isometric dimensioning and to generate the Bill of Material.

The system inserts these blocks in the DEFPOINTS layer, which is not plotted by AutoCAD, and the blocks appear in the middle of the component. These blocks also serve as the termination point of leader lines for Bill of Material identification balloons.

PROCAD SPOOLCAD groups INFO blocks with the component, so when you delete that component you also delete the component's INFO block.

The following illustration shows the different types of INFO blocks:

◇ PIPE	Y GASKETS
⊕ PIPE NIPPLES	⊙ BOLTS
⊕ BUTT WELD FITTINGS	┘ MISCELLANEOUS
⊗ SCREWED FITTINGS	△ SUPPORTS
I FLANGES	≡ DIMENSION INFO
+ VALVES	⊙ NOZZLE INFO

Viewing Component Information

To view component information...

1. From the *Settings* tab, select **Item Info**.



The cursor changes to a square.

2. In the drawing, click the INFO block of a component. Use your zoom tool to find it if necessary.
3. Press **Enter**.

The **Material Data List** dialog box appears, displaying the component's information.

Editing INFO Blocks

You can edit INFO blocks using AutoCAD's Dynamic Attribute Edit (DDATTE) command; however, PROCAD recommends you use *Spec Generator* to change component data. Refer to the *Spec Generator User Guide*.

Modifying and/or Adding Data

Use *Spec Generator* to modify or add specification information. Refer to the *Spec Generator User Guide*.

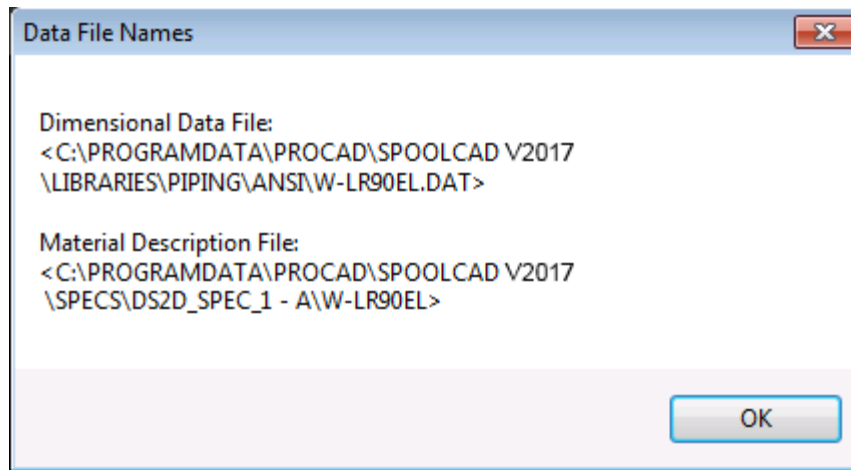
Finding a Data File

To find the name of a data file for a specific component...

1. Insert a component in the drawing.
2. From the *Utilities* tab, select **Data File Names**.



The **Date File Names** dialog box appears and displays the data file locations.



3. Click **OK**.

Equipment Drafting

The system checks for invalid isoplanes before placing vessel components and prompts you to change isoplane orientation if necessary.

Horizontal Vessels

PROCAD SPOOLCAD draws horizontal vessels in both sectional and full views.

Select the correct isoplane setting before inserting vessels.

To represent a section of the vessel...

1. From the *Equipment* tab, select **Horizontal Vessel – Flat View**.



2. In the drawing, click to select the location for the vessel's center point.
3. Click again to set the radius of the vessel.

The vessel appears in the drawing.

To represent the entire vessel...

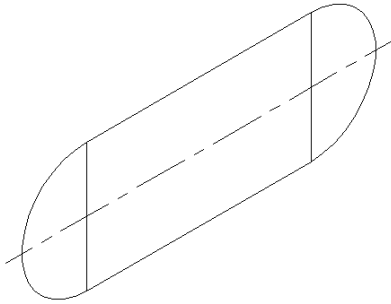
1. From the *Equipment* tab, select **Horizontal Vessel – Full View**.



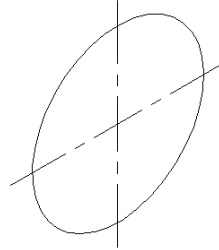
2. In the drawing, click to select the location for vessel's first seam.
3. Click again to select the location for the second seam.
4. Click to set the radius of vessel.

The vessel appears in the drawing.

Horizontal Vessel Drawing Samples



HORIZONTAL VESSEL—FULL VIEW



HORIZONTAL VESSEL—FLAT VIEW

Vertical Vessels

PROCAD SPOOLCAD draws vertical vessels in both sectional and full views.

Select the correct isoplane setting before inserting vessels.

To represent a section of the vessel...

1. From the *Equipment* tab, select **Vertical Vessel – Flat View**.



2. In the drawing, click to select the location of the vessel's center point.
3. Click to select the radius of vessel.

The vessel appears in the drawing.

To represent the entire vessel...

1. From the *Equipment* tab, select **Vertical Vessel – Full View**.



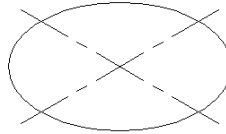
2. In the drawing, click to select the location of the vessel's first seam.
3. Click again to select the location for the second seam.
4. Click to set the radius of vessel.

The vessel appears in the drawing.

Vertical Vessel Drawing Samples



VERTICAL VESSEL—FULL VIEW



VERTICAL VESSEL—FLAT VIEW

Rectangles

PROCAD SPOOLCAD provides several generic rectangle tools. The system inserts rectangles as polylines.

Rectangle dialog boxes use **Digitize** buttons. See *Using the Digitize Buttons*.

Corner Base Rectangle

This tool inserts a standard rectangle from its lower left corner.

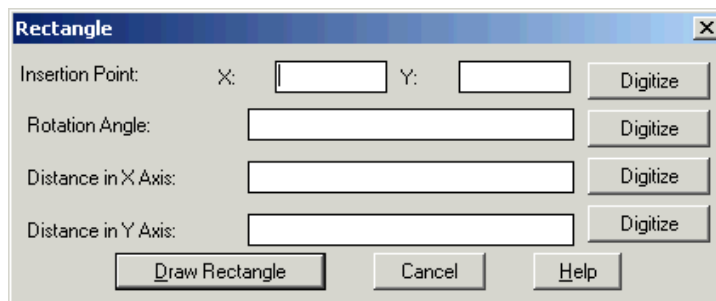
The isoplane setting does not affect corner base rectangles.

To insert a corner base rectangle...

1. From the **Equipment** tab, select **Corner Rectangle**.



The **Rectangle** dialog box appears.



2. In this dialog box, do the following:

In this...	Do this...
Insertion Point X and Y input boxes	Type the coordinates where you want to insert the rectangle, or click Digitize and click in the drawing to select the insertion point
Rotation Angle input box	Type the rotation angle, or click Digitize and click in the drawing to select the angle
Distance in X and Y Axis input boxes	Type the distance values for each axis, or click Digitize and click in the drawing to select distance values

- Click **Draw Rectangle**.

The rectangle appears in the drawing.

Center Point Rectangle

This tool inserts a standard rectangle from its center point.

Center point rectangles are not affected by the isoplane setting.

To insert a center point rectangle ...

- From the *Equipment* tab, select **Center Rectangle**.



The **Rectangle** dialog box appears.

- In this dialog box, do the following:

In this...	Do this...
Insertion Point X and Y input boxes	Type the coordinates where you want to insert the rectangle, or click Digitize and click in the drawing to select the insertion point
Rotation Angle input box	Type the rotation angle, or click Digitize and click in the drawing to select the angle
Distance in X and Y Axis input boxes	Type the distance values for each axis, or click Digitize and click in the drawing to select distance values

- Click **Draw Rectangle**.

The rectangle appears in the drawing.

ISO Rectangles

Corner Point ISO Rectangle

This tool inserts an ISO rectangle from its corner point.

Select the correct isoplane setting before inserting a corner point ISO rectangle.

To insert a corner point ISO rectangle...

1. From the *Equipment* tab, select **Iso rectangle – Base**.



The **Rectangle** dialog box appears.

2. In this dialog box, do the following:

In this...	Do this...
Insertion Point X and Y input boxes	Type the coordinates where you want to insert the rectangle, or click Digitize and click in the drawing to select the insertion point
1st and 2nd Isoplane Direction input boxes	Type the angle value (in degrees) for the isoplane directions, or click Digitize and click in the drawing to set the direction
Distance in X and Y Axis input boxes	Type the distance values for each axis, or click Digitize and click in the drawing to select distance values

3. Click **Draw Rectangle**.

The rectangle appears in the drawing.

Center Point ISO Rectangle

This tool inserts an ISO rectangle from its center point.

Select the correct isoplane setting before inserting a center point ISO rectangle.

To insert a center point ISO rectangle...

1. From the *Equipment* tab, select **Iso Rectangle – Center**.



The **Rectangle** dialog box appears.

2. In this dialog box, do the following:

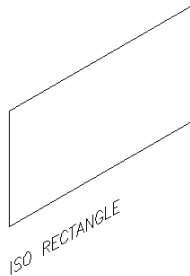
In this...	Do this...
Insertion Point X and Y input boxes	Type the coordinates where you want to insert the rectangle, or click Digitize and click in the drawing to select the insertion point
1st and 2nd Isoplane Direction input boxes	Type the angle value (in degrees) for the isoplane directions, or click Digitize and click in the drawing to set the directions
Distance in X and Y Axis input boxes	Type the distance values for each axis, or click Digitize and click in the drawing to select distance values

3. Click **Draw Rectangle**.

The rectangle appears in the drawing.



STANDARD RECTANGLE



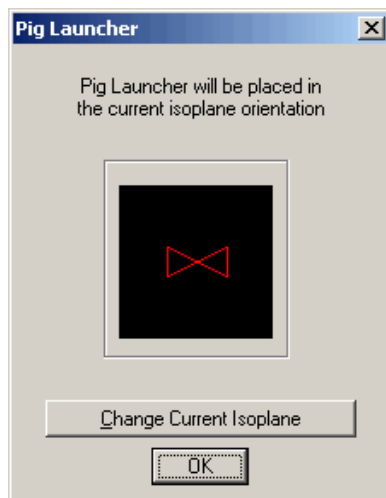
Pig Launcher

To insert a Pig Launcher...

- From the *Equipment* tab, select **Pig Launcher**.



The **Pig Launcher** isoplane dialog box appears.



2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the insertion point.
4. Click to select the direction.

The pig launcher appears in the drawing.

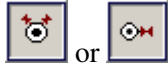
Orifice Taps

The orifice tap symbols show the orientation of orifice taps for meter run installation. There are two views available: offset and parallel.

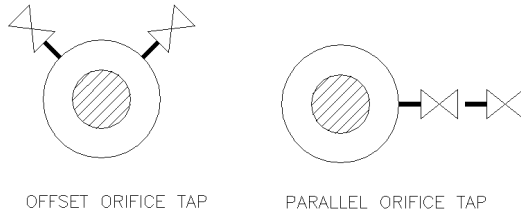
Orifice taps do not contain material INFO blocks. After inserting the orifice tap symbol in the drawing, create a special item INFO block using the *Inserting a Special Item* procedure, and then add the orifice taps to the Bill of Material procedure.

To insert orifice taps...

1. From the **Equipment** tab, select **Orifice Taps**.



2. In the drawing, click to select the insertion point.
3. Click to select the direction. The tap appears in the drawing.



4. Insert an INFO block into the orifice tap image. See *Inserting a Special Item*.
5. If you have generated a Bill of Material, add the orifice tap to the material listing.

AutoRoute

AutoRoute allows you to insert piping components sequentially. When activated, AutoRoute uses the end point of the last component you placed as the starting point for the next component, following the same direction. If you change direction, a dialog box prompts to select a new isoplane setting.

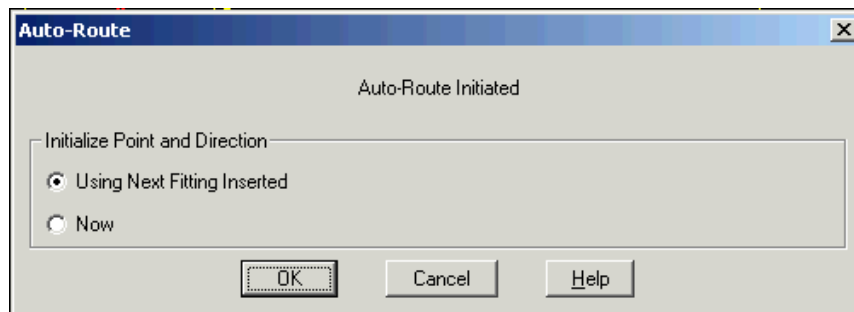
Activating AutoRoute

To activate AutoRoute...

1. From the **AutoRoute** tab, select **Enable AutoRoute**.



The **Auto-Route** dialog box appears.



2. Select an initialization option:
 - **Using Next Fitting Inserted** activates AutoRoute after you insert the next component
 - **Now** activates AutoRoute and prompts you for the start point and direction of the first component
3. Click **OK**.
4. Select and insert the required piping components into the drawing.
5. To disable AutoRoute, click **Enable AutoRoute** again.

Adding a Branching Component in AutoRoute

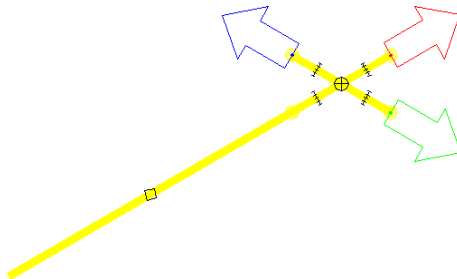
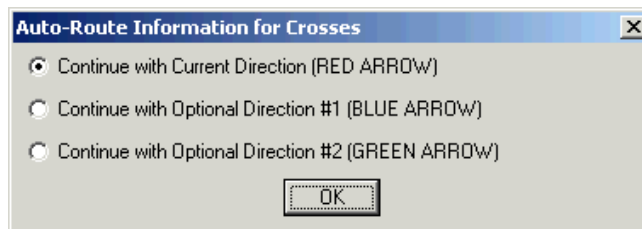
Use this procedure when adding branching components while using AutoRoute.

When inserting branching components such as straight or reducing tees and crosses, a dialog box displays the available directions with different colored arrows and prompts you to select one.

To add a branching component while using AutoRoute...

1. Select and insert the branching component.

The **Auto-Route Information for Tees or Crosses** dialog box appears.



2. From the options, select the direction in which you want to continue adding components.
3. Click **OK** to continue inserting components.
 - **Note:** If you select to continue adding components on a *branch* side for reducing fittings, PROCAD SPOOLCAD resets the pipe size to that of the branch.
4. When you finish adding fittings to one branch, disable AutoRoute, enable it again, and add fittings to another branch.
 - **Note:** Ensure your pipe size is correct if adding components to another branch of a reducing fitting.

Removing an AutoRoute Component

Use this tool to delete the last component inserted while using AutoRoute.

If you delete a component manually, AutoRoute continues placing components from the end point of the deleted item, creating a gap in the drawing.

To remove the last component you inserted...

From the *AutoRoute* tab, select **Undo Last Item**.



Showing Current AutoRoute Position

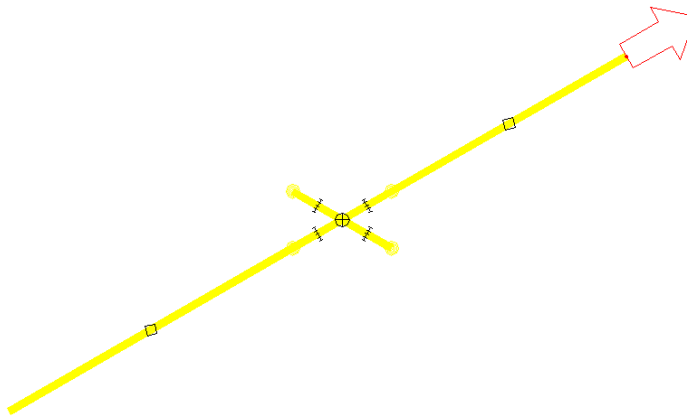
Use this tool to view the current insertion point and direction of the AutoRoute session.

To show your current position...

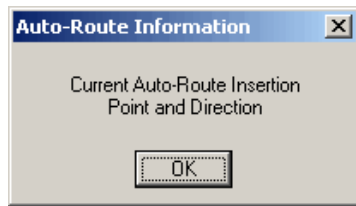
1. From the *AutoRoute* tab, select **Show Current Position**.



An arrow appears showing the insertion point and direction of the AutoRoute session.



The **AutoRoute Information** notification box appears.



2. Click **OK**.

Pipe and Pipe Nipples

Pipe

When you insert pipe in a drawing, PROCAD SPOOLCAD inserts an INFO block in the center of the pipe segment. This INFO block contains the pipe's material information, including wall thickness.

If you dimension the drawing, PROCAD SPOOLCAD inserts the pipe's cut length into the INFO block.

If you do not dimension the drawing, insert the pipe's cut length using the **Calculate Pipe Length** tool from the *Bill of Material* tab.

The current isoplane setting does not affect pipe placement.

Inserting Pipe

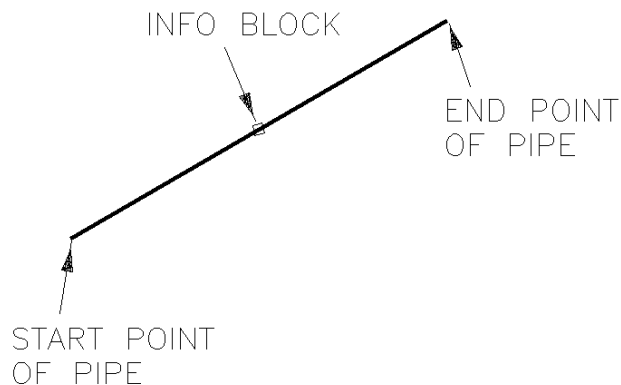
To insert a pipe segment...

1. From the *Fittings* or *Pipe* tab, select **Pipe**.



2. In the drawing, click to select the pipe's start point.
3. Click to select the pipe's end point.

The pipe segment appears in the drawing.



➤ **Notes:** If AutoRoute is active, select the pipe end point only.

Auto Pipe Placement

You can simplify pipe and elbow placement by using a routing polyline and the AutoELBOW and PIPE tool. See *AutoELBOW* and PIPE for details.

Pipe Nipples

The current isoplane setting does not affect pipe nipple placement. If AutoRoute is active, select only the end point.

Inserting a Pipe Nipple

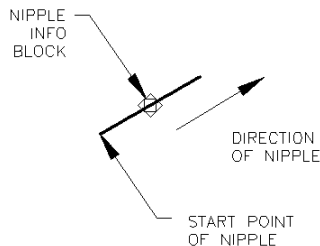
To insert a pipe nipple...

1. From the *Pipe* tab, select **Pipe Nipple**.



2. In the drawing, click to select the start point.
3. Click to select the direction.

The pipe nipple appears in the drawing.



Overriding the Default Nipple Length

See *Nipple Length Override*.

When you activate this override, a dialog box prompts you for a new nipple length value when you insert a pipe nipple in the drawing.

Inserting Threaded End Nipples (for Screwed Fittings)

If you set the small pipe fitting type in the current spec to screwed, a dialog box prompts you to select **Threaded Both Ends** or **Plain One End** when you insert the pipe nipple. This affects the nipple's dimension value.

- **Threaded Both Ends** deducts two thread engagements from the nipple's dimensional length.
- **Plain One End** deducts only one thread engagement.

PROCAD SPOOLCAD adds nipple end information (TBE or POE/TOE) to the nipple's INFO block.

To set the small pipe fitting setting to screwed, see *Piping Spec Overrides*.

Inserting Plain End Nipples (for Socket Weld Fittings)

If you set the small pipe fitting type in the current spec to socket weld, PROCAD SPOOLCAD automatically adds a **Plain Both Ends** nipple.

The system adds nipple length and ends (PBE - Plain Both Ends) to the nipple INFO block.

To set the small pipe fitting setting to socket weld, see *Piping Spec Overrides*.

Stub-In

Use this tool to insert a Stub-in or Stub-on

Inserting a Stub-in or Stub-on

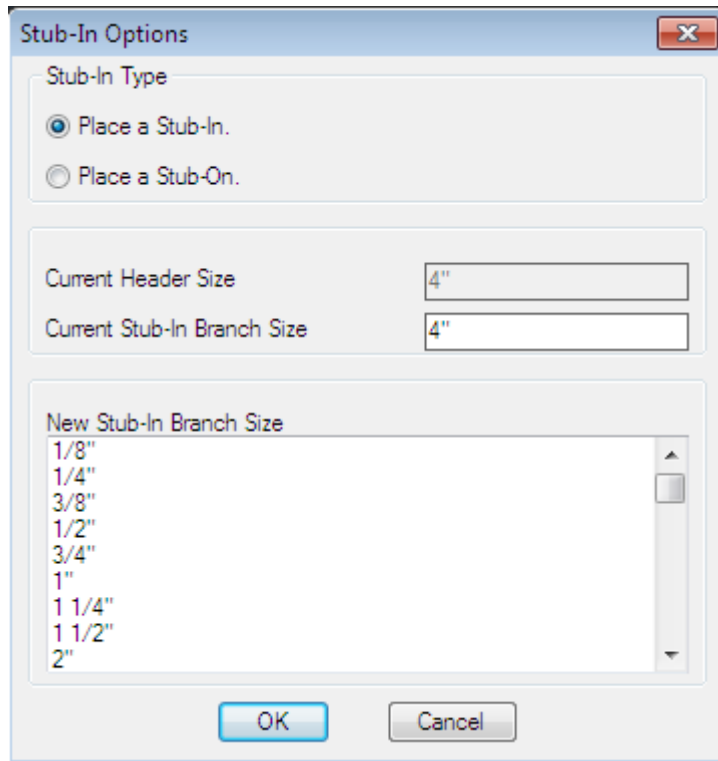
To insert a stub-in or stub-on...

1. From the *Pipe* tab, select **Stub-in**.



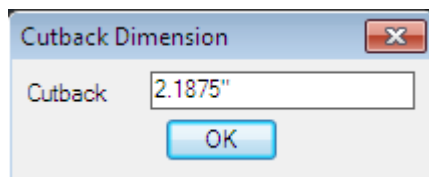
2. In the drawing, click on a location along a pipe for the starting point.

The **Stub-in Options** dialog box appears



3. Under **Stub-in Type**, Choose one of the following options:
 - Place a Stub-in
 - Place a Stub-on
4. Choose the stub-in branch size.
5. Click the **OK** button.

The **Cutback Dimension** dialog box appears.



6. Change the cutback dimension if needed and click the **OK** Button.
7. Choose the direction of the stub-in.

Inserting a Pipe Break

Use this tool to insert a pipe break.

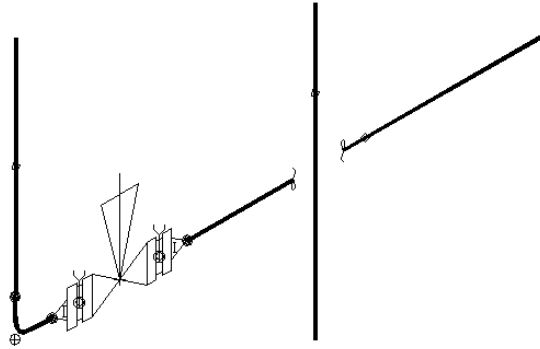
To insert a pipe break...

1. From the **Pipe** tab, select **Pipe Ends**.



2. In the drawing, click to select the first break point.
3. Click to select the second break point on the same pipe.

PROCAD SPOOLCAD breaks the pipe and adds pipe end symbols.



Inserting a Pipe End Symbol

Use this tool to insert a single pipe end symbol to the end of a pipe segment.

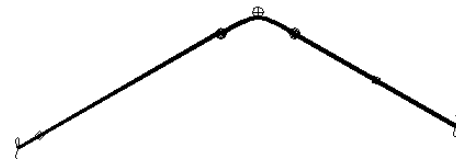
To insert a pipe end symbol...

1. From the **Pipe** tab, select **Pipe End**.



2. Click the pipe segment near its end point.

The pipe end symbol appears at the end of the pipe segment.



Pipe Through Wall or Ground

Use this tool to indicate that a pipe penetrates a ground or wall barrier.

Select the correct isoplane setting before inserting this symbol.

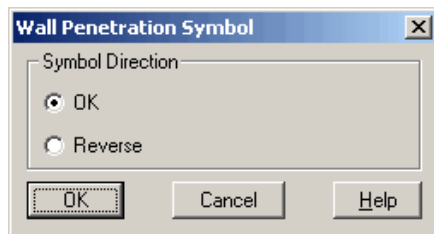
To insert this symbol...

1. From the **Pipe** tab, select **Ground/Wall Penetration Symbol**.



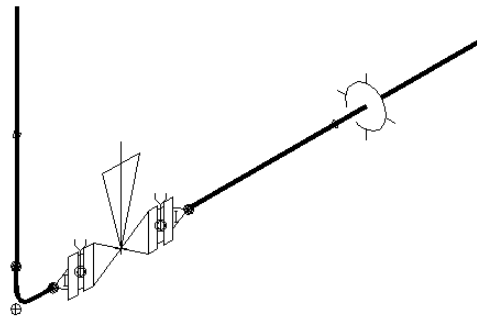
2. In the drawing, click to select the pipe where you want to add the symbol.

The **Wall Penetration Symbol** dialog box appears.



3. Under **Symbol Direction**, choose one of the following options:
 - Select **OK** to accept the current arrow direction
 - Select **Reverse** to change the direction
4. Click the **OK** button.

The ground/wall penetration symbol appears in the drawing.



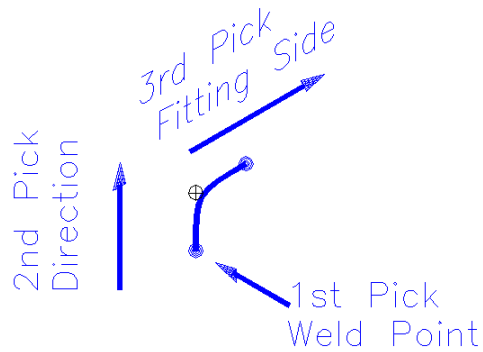
Fittings

Fitting Insertion Points

PROCAD SPOOLCAD inserts fittings from the weld point, center point or face (flanges). For most fittings PROCAD SPOOLCAD defaults to the weld point.

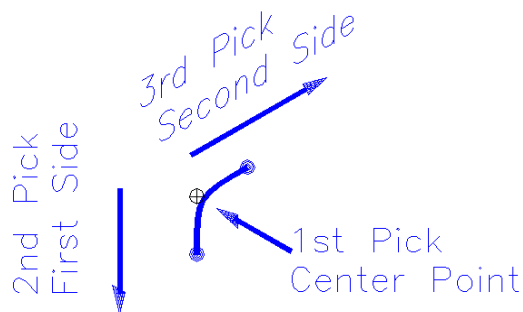
Inserting Fittings From the Weld Point

When you insert a fitting from its weld point, follow the selection sequence shown below:



Inserting Fittings from the Center Point

When you insert a fitting from its center point, follow the selection sequence shown below:



AutoRoute and Fitting Insertion

When AutoRoute is active, PROCAD SPOOLCAD automatically inserts fittings by the weld point.

Many fittings do not require any input for start point and direction when the AutoRoute is active. AutoRoute deactivates itself after placing line end components, such as weld caps. See *AutoRoute* for more information.

Out-of-Spec Fittings

You can add an out-of-spec fitting such as socket weld in a spec that specifies screwed fittings. A dialog box alerts you and prompts you to either accept the override or reset settings to conform to the current pipe spec.

Fitting Orientation

Select the correct isoplane setting before inserting any fitting. PROCAD SPOOLCAD always sets the side orientation angles of valves and some fittings. This eliminates the need to supply the side orientation of a fitting or a valve.

If you select an invalid direction, the isoplane dialog box appears, prompting you to select a different setting.

- **Note:** Some fittings (such as 90 degree elbows) are not affected by this feature.

Fitting Types

PROCAD SPOOLCAD inserts fittings as butt weld, screwed (threaded) or socket weld.

The piping specs and drawing settings control which fittings appear in the drawing. Use the following keystroke combinations to toggle Object Grouping on or off.

Shift + Ctrl + A

See Setting Specifications and Setting Spec Overrides.

General Notes on Fitting Insertion

To insert butt weld fittings when the current pipe size is in the small fittings range, change the start size of small fittings to less than that of the current pipe size. See *Piping Spec Overrides*.

To insert screwed (or socket weld) fittings when the current pipe size is in the butt weld fittings range, change the start size of small fittings to be equal to or larger than that of the current pipe size. See *Piping Spec Overrides*.

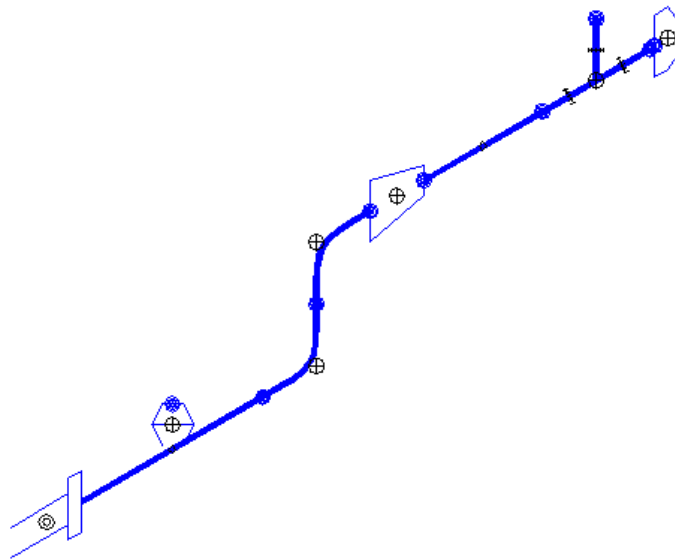
To insert a socket weld fitting in a spec that specifies screwed fittings and vice versa, set the small fitting overrides. See *Piping Spec Overrides*.

Butt Weld Fittings

When pipe size is in the butt weld range, PROCAD SPOOLCAD inserts butt weld fittings and flanged valves. To insert welded valves instead, set the welded valves override. See *Welded Valves Override*.

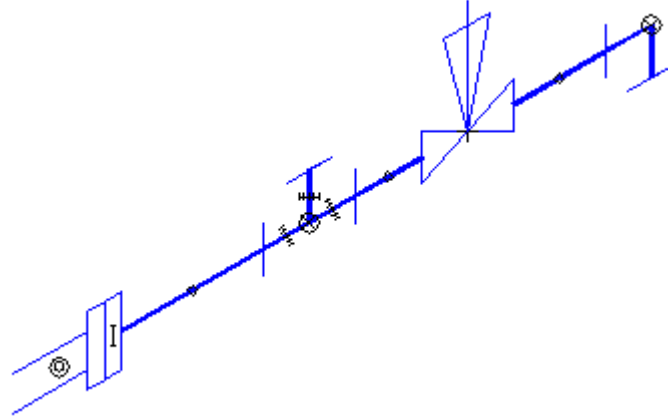
The system inserts weld dots at all fitting weld points.

You can set the butt weld elbows to appear as curved or square.



Screwed (Threaded) Fittings

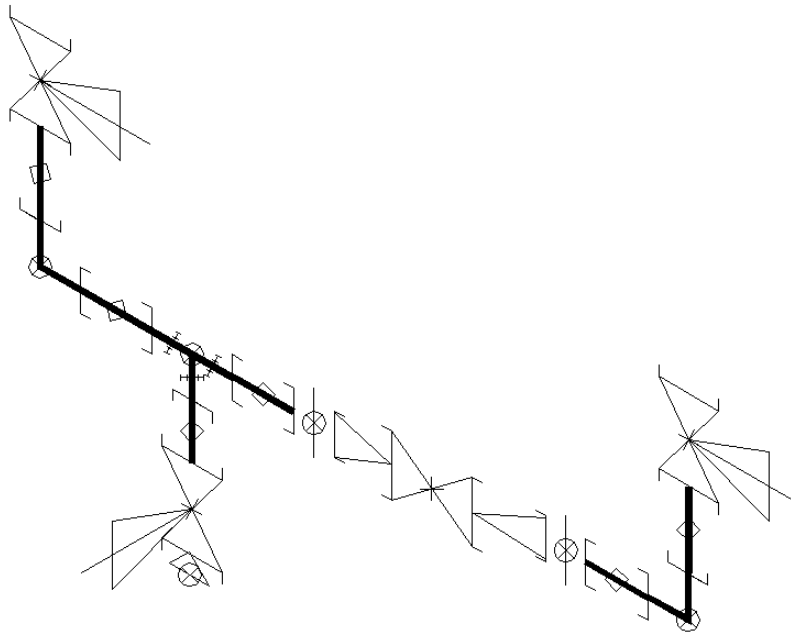
When the pipe size is in the small fitting range and the current spec specifies screwed fittings, PROCAD SPOOLCAD inserts screwed fittings.



Socket Weld Fittings

When the pipe size is in the small fitting range and the current spec specifies socket weld fittings, the system inserts socket weld fittings.

The system draws socket weld fittings symbols with socket extensions on both sides of the fitting.



Common Fitting Modifications

Modifying Fitting Material Information

To modify material information for fittings, use *Spec Generator*. See the *Spec Generator User Guide*.

AutoELBOW and PIPE

Use this procedure to generate automatic elbow and pipe fittings. Insert a routing polyline before using the AutoELBOW and PIPE tool.

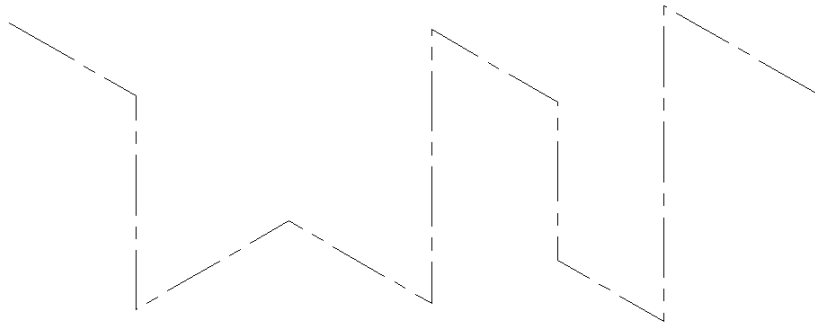
To automatically insert 90-degree long radius elbows and pipe...

1. From the **Line Types** tab, select **Pipe Routing Line**.



2. Draw the pipe route from start to end point. A polyline appears in the drawing.

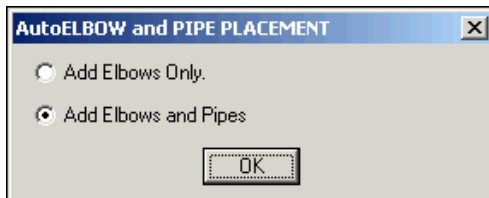
➤ **Note:** You must construct pipe routing line as a single polyline.



3. From the **Pipe** tab *or* the **Fittings** tab, select **AutoELBOW and Pipe**.

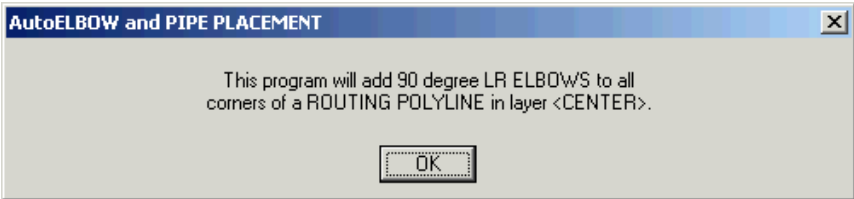


The **AutoELBOW and PIPE PLACEMENT** dialog box appears.



4. Choose one of the following options:
 - Click **Add Elbows Only** to insert only elbows
 - Click **Add Elbows and Pipes** to insert pipes and elbows
5. Click **OK**.

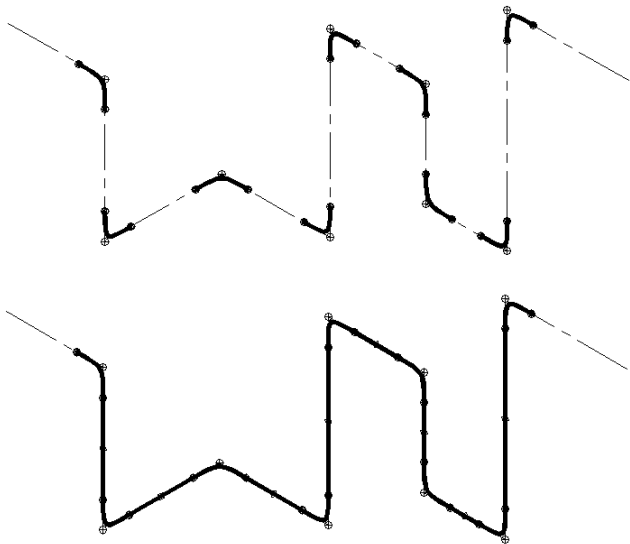
The **AutoELBOW and PIPE Placement** notification box appears.



6. Click **OK**.


The cursor changes to a square.
7. In the drawing, click on the polyline.


The elbows (and pipes) appear in the drawing.



Inserting Fittings

PROCAD SPOOLCAD's Toolbox has two fittings tabs:

Tab	Name	Contents
	Fittings	Pipe, pipe bends, elbows, reducers, swages, laterals, crosses, tees, 'Olet fittings, AutoELBOW and PIPE

	Miscellaneous Fittings	Flanged spool, flange isolation kit, quick connector, transition piece, re-pad, bleed ring, expansion joint, spectacle blind, plug, cap, union, couplings, strainers
---	------------------------	--

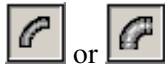
Generally, you must select the appropriate isoplane before inserting fittings. However, some fittings are not limited by the isoplane and can be inserted in any orientation.

90 degree Short and Long Elbows

90° elbows are not restricted by the current isoplane and you can insert them in any orientation.

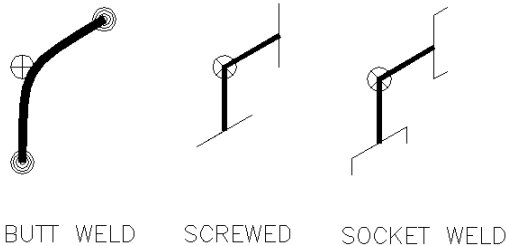
To insert a 90 degree elbow...

1. From the **Fittings** tab, select **90° LR Elbow** or **90° SR Elbow**.



2. In the drawing, click to select the weld (or center) point.
3. Click to select the direction of the elbow.
4. Click to select the side of the elbow.

The elbow appears in the drawing.



45 degree Elbows

Select the correct isoplane setting before inserting 45° elbows.

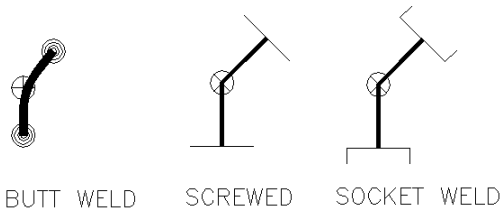
To insert a 45 degree elbow...

1. From the **Fittings** tab, select **45° Elbow**.



2. In the drawing, click to select the weld (or center) point.
3. Click to select the direction of the elbow.
4. Click to select the side of the elbow.

The elbow appears in the drawing.

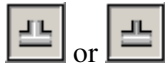


Straight and Reducing Tees

Select the correct isoplane settings before inserting straight or reducing tees.

To insert a straight or reducing tee...

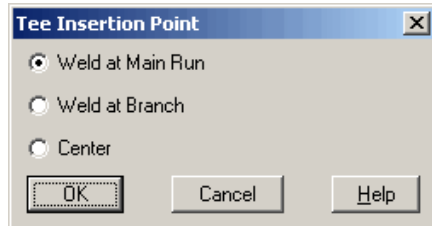
1. From the **Fittings** tab, select **Straight Tee** or **Reducing Tee**.



If you select **Reducing Tee**, the **Confirm Fitting Size** dialog box appears.

2. For reducing tees, select new main and branch sizes from the scroll lists if required.
3. Click **OK**.

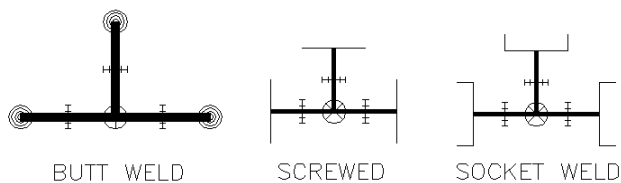
For both straight and reducing tees, the **Tee Insertion Point** dialog box appears.



➤ **Note:** The **Weld at Branch** option is available only for straight tees.

4. From the options, select the insertion point of the fitting.
5. Click **OK**.
6. In the drawing, click to select the run direction.
7. Click to select the branch direction.

The tee appears in the drawing.



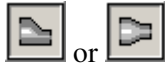
Reducers

Select the correct isoplane settings before inserting reducers.

After you insert a reducer, PROCAD SPOOLCAD switches the pipe size to the branch pipe size.

To insert a reducer...

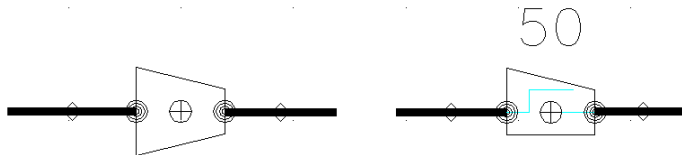
1. From the **Fittings** tab, select **Eccentric** or **Concentric Reducer**.



The **Confirm Fitting Size** dialog box appears.

2. From the scroll lists, select new inlet and outlet pipe sizes if required.
3. Click **OK**.
4. In the drawing, click to select the first end weld point.
5. Click to select the direction.
6. If placing an eccentric reducer, click to select the flat side.

The reducer appears in the drawing.



- **Notes:** The system prints the offset value for eccentric reducers above the fitting.

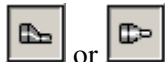
Swages

Select the correct isoplane settings before inserting swages.

After you insert a swage, PROCAD SPOOLCAD switches the pipe size to the branch pipe size.

To insert a swage...

1. From the **Fittings** tab, select **Eccentric** or **Concentric Swage**.

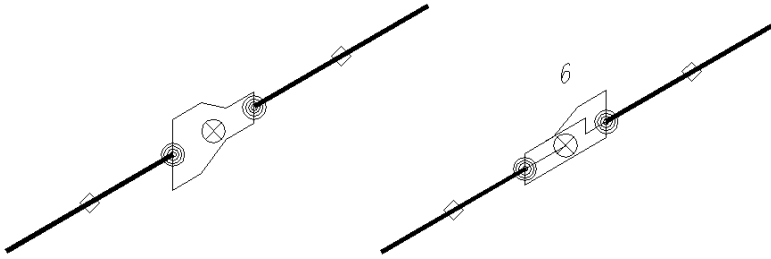


The **Confirm Fitting Size** dialog box appears.

2. From the scroll lists, select new inlet and outlet pipe sizes if required.
3. Click **OK**.

4. In the drawing, click to select the first end point.
5. Click to select the direction.
6. If you are placing eccentric swages, click to select the flat side of the swage.
The **Concentric Swage** or **Eccentric Swage End Type** dialog box appears.
7. From the options, select the appropriate **End Type**.
8. Click **OK**.

The swage appears in the drawing.



- **Notes:** The system displays the offset value for eccentric swages above the fitting.

180° Returns

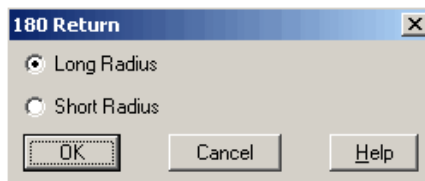
180° returns are not restricted by the current isoplane and you can insert them in any orientation.

To insert a 180° return...

1. From the **Fittings** tab, select **180° Return**.

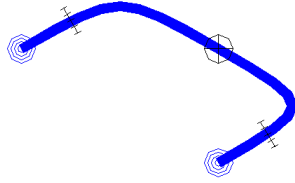


The **180 Return** dialog box appears.



2. From the options, select the required radius.
3. Click **OK**.
4. In the drawing, click to select the weld (or center) point of the return.
5. Click to select the direction of the return.
6. Click to select the side of the return.

The return appears in the drawing.



Reducing Elbows

Reducing elbows are not restricted by the current isoplane and you can insert them in any orientation.

To insert a reducing elbow.....

1. From the **Fittings** tab, select **Reducing Elbow**.



The **Confirm Fitting Size** dialog box appears.

2. From the scroll lists, select new inlet and outlet sizes if required.
3. Click **OK**.
4. In the drawing, click to select the large side of the elbow.
5. Click to select the direction toward the center of the elbow.
6. Click to select the direction of the small side of the elbow.

The elbow appears in the drawing.



Trimmed Elbows

Trimmed elbows are not restricted by the current isoplane and you can insert them in any orientation.

To insert a trimmed elbow...

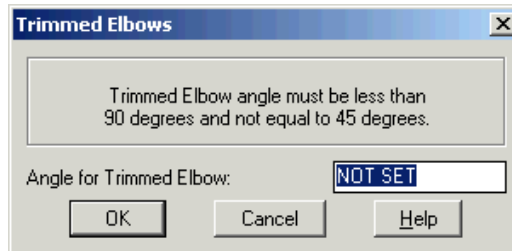
1. From the **Fittings** tab, select **Trimmed Elbow**.



2. In the drawing, click to select the weld end of the elbow.

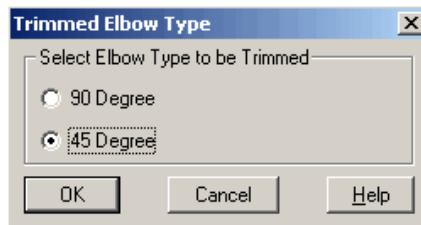
3. Click to select the direction of the elbow.
4. Click to select the side of the elbow.

The **Trimmed Elbows** dialog box appears.



5. In the **Angle for Trimmed Elbow** box, type the angle value.

If you type a value less than 45 degrees, the **Trimmed Elbow Type** dialog box appears.



6. From the options, select the required elbow type.
7. Click **OK**.

The elbow appears in the drawing.

Pipe Bends

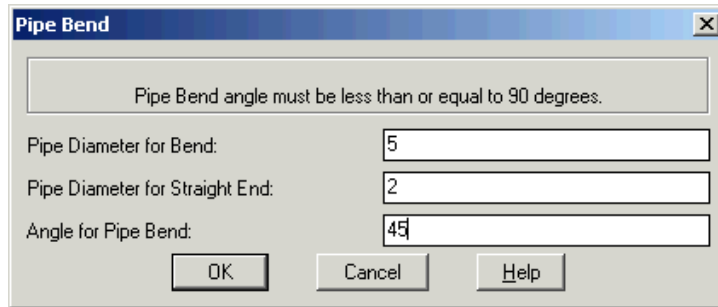
Pipe bends are not restricted by the current isoplane and you can insert them in any orientation.

To insert a pipe bend...

1. From the **Fittings** tab, select **Pipe Bend**.

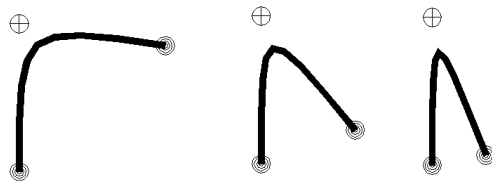


The **Pipe Bend** dialog box appears.



2. In the input boxes, type pipe bend and straight end values, and pipe bend angle.
3. Click **OK**.
4. In the drawing, click to select the weld (or center) point of bend.
5. Click to select the direction of the pipe bend.
6. Click to select the side of the pipe bend.

The pipe bend appears in the drawing.

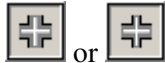


Straight and Reducing Crosses

Select the correct isoplane setting before inserting crosses.

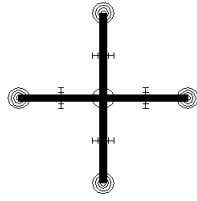
To insert a straight or reducing cross...

1. From the **Fittings** tab, select **Straight Cross** or **Reducing Cross**.

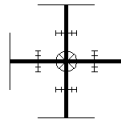


If you insert a reducing cross, the **Confirm Fitting Size** dialog box appears.

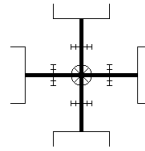
2. For reducing crosses, select new main and branch sizes from the scroll lists if required.
3. Click **OK**.
4. In the drawing, click to select the weld (or center) point.
5. Click to select the main direction.
6. Click to select the branch direction. The cross appears in the drawing.



BUTT WELD



SCREWED



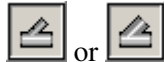
SOCKET WELD

Straight and Reducing Laterals

Select the correct isoplane setting before inserting laterals. The *angle* of the branch is not restricted by the isoplane and can be added in any orientation.

To insert a straight or reducing lateral...

1. From the **Fittings** tab, select **Reducing Lateral** or **Straight Lateral**.

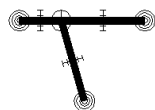


or

If you insert a reducing lateral, the **Confirm Fitting Size** dialog box appears.

2. For reducing laterals, select new main and branch sizes from the scroll lists if required.
3. Click **OK**.
4. In the drawing, click to select the weld (or center) point of the fitting.
5. Click to select the run direction.
6. Click to select the branch direction.

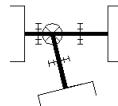
The lateral appears in the drawing.



BUTT WELD



SCREWED



SOCKET WELD

Caps

Select the correct isoplane setting before inserting caps.

To insert a cap...

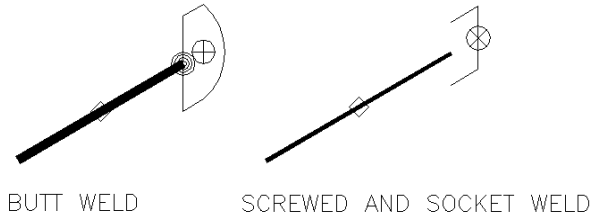
1. From the **Miscellaneous Fittings** tab, select **Cap**.



2. In the drawing, click to select the weld point.

- Click to select the direction.

The cap appears in the drawing.



Re-Pads

Select the correct isoplane setting when inserting re-pads.

To insert a re-pad...

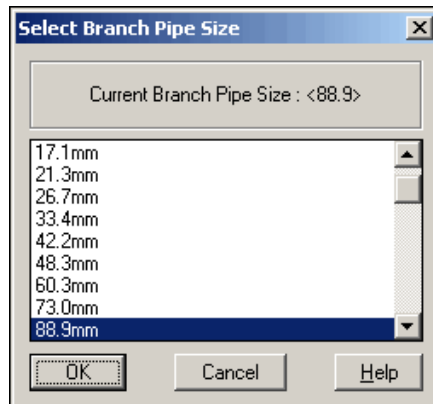
- From the *Miscellaneous Fittings* tab, select **Re-Pad**.



A message box appears, displaying the current pipe size.

- Click **OK**.

The **Select Branch Pipe Size** dialog box appears.



- From the scroll list, select the appropriate branch size.
- Click **OK**.
- In the drawing, click to select the main/branch intersection point.
- Click to select the direction of the main pipe.
- Click to select the direction of the branch pipe.

The **Reinforcing Pad** dialog box appears.



8. In the **Diameter** box, type the diameter value.
9. Click **OK**.

The re-pad appears in the drawing.



Weldolets®, Thredolets® and Sockolet®

'Olet fittings are not restricted by the current isoplane and you can insert them in any orientation.

To insert a Weldolet®, Thredolet®, or Sockolet®...

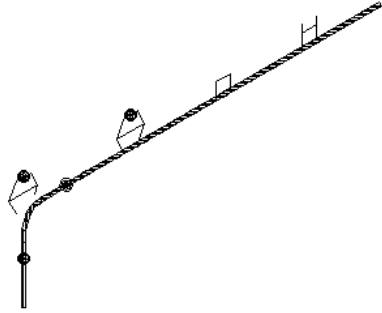
1. From the **Fittings** tab, select **Weldolet®**, **Thredolet®**, or **Sockolet®**



The **Confirm Fitting Size** dialog box appears.

2. From the scroll lists, select new inlet and outlet sizes if required.
3. Click **OK**.
4. In the drawing, click to select the pipe or routing centerline for the fitting base.
5. Click to select the branch direction.

The 'olet appears in the drawing.



Elbolets

Select the correct isoplane setting before inserting an Elbolet®.

To insert an Elbolet®...

1. From the **Fittings** tab, select **Elbolet®**.



The **Confirm Fitting Size** dialog box appears.

2. From the scroll lists, select new inlet and outlet sizes if required.
3. Click **OK**.
4. In the drawing, click to select the elbow center point.
5. Click to select the pipe opposite the Elbolet® direction.
6. Click to select the other side of the elbow. It must be different than the first direction.

The Elbolet® appears in the drawing.

Couplings

Select the correct isoplane setting before inserting couplings.

To insert a coupling...

1. From the **Miscellaneous Fittings** tab, select **Half Coupling**, **Reducing Coupling**, or **Full Coupling**.

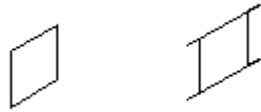


If you insert a reducing coupling, the **Confirm Fitting Size** dialog box appears.

2. For reducing couplings, select new inlet and outlet sizes from the scroll lists if required.

3. Click **OK**.
4. In the drawing, click to select the start point.
5. Click to select the direction.


The coupling appears in the drawing.



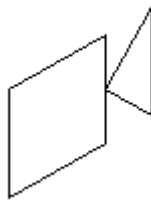
Plugs

Select the correct isoplane setting before inserting plugs.

To insert a plug...

1. From the *Miscellaneous Fittings* tab, select **Plug**.

2. In the drawing, click to select the insertion point.
3. Click to select the direction.


The plug appears in the drawing.



Unions

Select the correct isoplane setting before inserting unions.

To insert a union...

1. From the *Miscellaneous Fittings* tab, select **Unions**.

2. In the drawing, click to select the insertion point.
3. Click to select the direction of the union.

The union appears in the drawing



Mitered Elbows

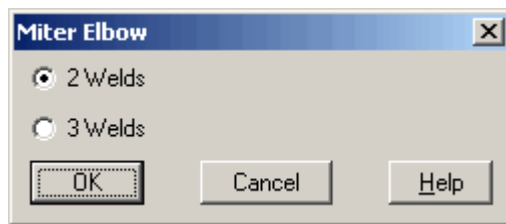
Mitered elbows are not restricted by the current isoplane and you can insert them in any orientation.

To insert a mitered elbow...

1. From the **Fittings** tab, select **Mitered Elbow**.

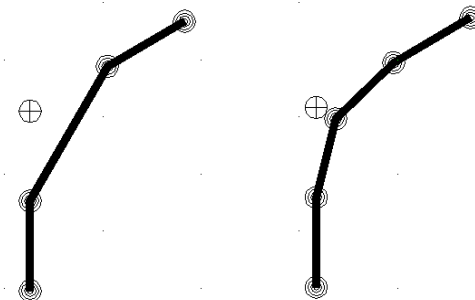


The **Miter Elbow** dialog box appears.



2. From the options, select the number of welds required.
3. Click **OK**.
4. In the drawing, click to select the weld (or center) insertion point.
5. Click to select the main direction.
6. Click to select the side direction.

The elbow appears in the drawing.



Transition Pieces

Select the correct isoplane setting before inserting transition pieces.

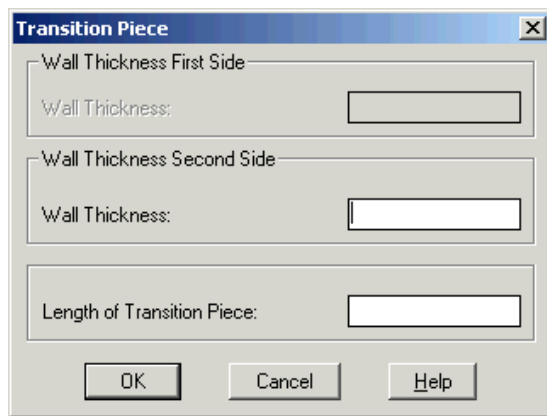
To insert a transition piece...

1. From the *Miscellaneous Fittings* tab, select **Transition Piece**.



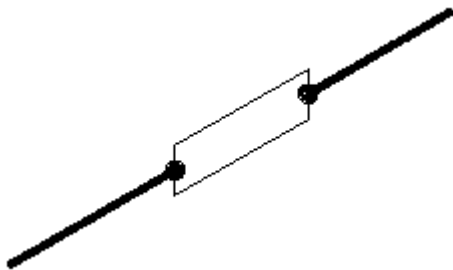
2. In the drawing, click to select the weld point.
3. Click to select the direction.

The **Transition Piece** dialog box appears.



4. In the input boxes, type values for the wall thickness and length of the transition piece.
5. Click **OK**.

The transition piece appears in the drawing.



Bleed Rings

Select the correct isoplane setting before inserting bleed rings.

To insert a bleed ring...

1. From the *Miscellaneous Fittings* tab, select **Bleed Ring**.

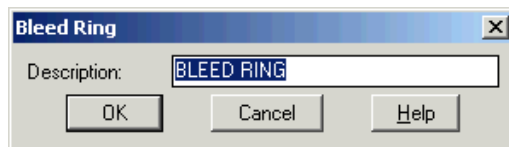


The **Bleed Ring** Width dialog box appears.



2. In the **Width** box, type the bleed ring thickness.
3. Click **OK**.
4. In the drawing, click to select the start point.
5. Click to select the direction.

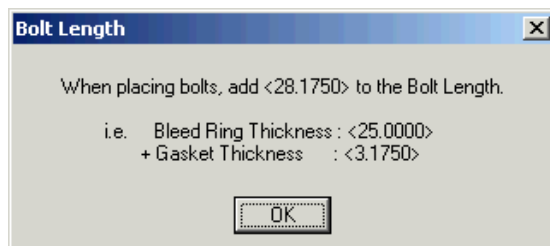
The **Bleed Ring** Description dialog box appears



6. In the **Description** input box, do one of the following:
 - Accept the default description
 - Type a new description
7. Click **OK**.

The bleed ring appears in the drawing.

The **Bolt Length** notification box appears.

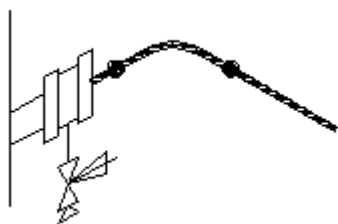


8. Click **OK**.

The **Bolt Length** override is now active.

When inserting bolts in a component grouping that contains a bleed ring, add the bleed ring's thickness and one extra gasket thickness to the bolt length.

- **Note:** If you do not know the required bolt length when you insert the bleed ring, use the **Dynamic Attribute Edit** tool later to edit bolt length information in the bolt's INFO block. See *Dynamic Attribute Edit*.



Spectacle Blinds

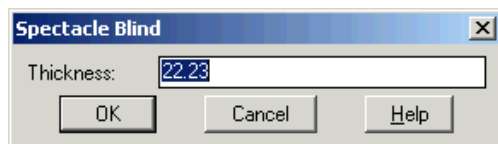
Select the correct isoplane setting before inserting spectacle blinds.

To insert a spectacle blind...

1. From the *Miscellaneous Fittings* tab, select **Spectacle Blind**.

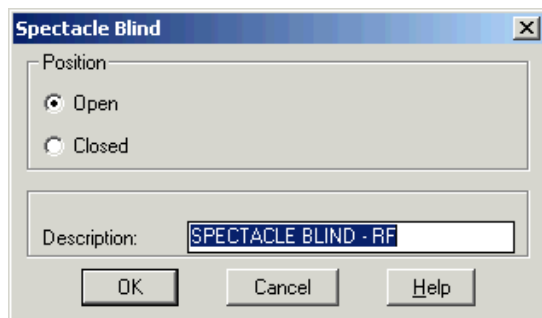


The **Spectacle Blind Thickness** dialog box appears.



2. In the **Thickness** box, do one of the following:
 - Accept the default value
 - Type a new value
3. Click **OK**.
4. In the drawing, click to select the start point.
5. Click to select the direction of the pipe.

The **Spectacle Blind Position/Description** dialog box appears.



6. Under **Position**, set the position to open or closed.
7. In the **Description** box, do one of the following:
 - Accept the default description
 - Type a new description
8. Click **OK**.

The spectacle blind appears in the drawing.

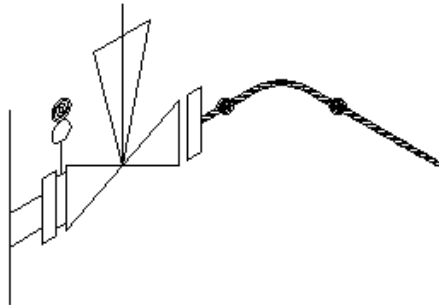
The **Bolt Length** notification box appears.

9. Click **OK**.

The **Bolt Length** override is now active.

When inserting bolts in a component grouping that contains a spectacle blind, add the spectacle blind's thickness and one extra gasket thickness to the bolt length.

- **Note:** If you do not know the required bolt length when you insert the spectacle blind, use the **Dynamic Attribute Edit** tool later to edit bolt length information in the bolt's INFO block. See *Dynamic Attribute Edit*.



Expansion Joints

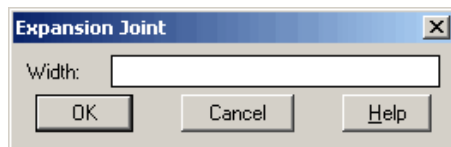
Select the correct isoplane setting before inserting expansion joints.

To insert an expansion joint...

1. From the *Miscellaneous Fittings* tab, select **Expansion Joint**.



The **Expansion Joint** Length dialog box appears.



2. In the **Width** box, type the width of the expansion joint.

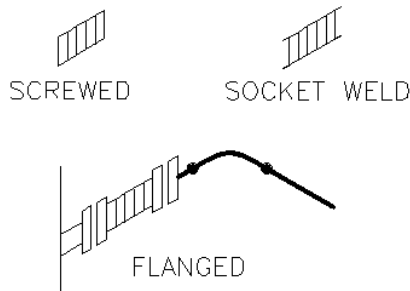
3. Click **OK**.
4. In the drawing, click to select the start point.
5. Click to select the pipe direction.

The **Expansion Joint** Description dialog box appears.



6. In the **Description** box, do one of the following:
 - Accept the default description
 - Type a new description
7. Click **OK**.

The expansion joint appears in the drawing.



Spool Piece

Select the correct isoplane setting before inserting spool pieces.

Spool pieces do not appear in the Bill of Material. If you want spool pieces to appear in the BOM, see *Inserting a Special Item*.

To insert a flanged spool piece between two flanges...

1. From the *Miscellaneous Fittings* tab, select **Flanged Spool**.



If you have not set a line number, the **Line Number** dialog box appears. Type a line number and click **OK**.

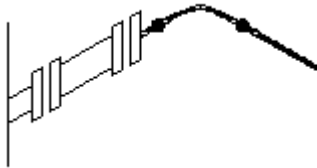
2. In the drawing, click to select the face of the flange.
3. Click to select the direction.
4. Click to select the end point.

The **Spool Piece Length** dialog box appears.



5. In the **Spool Piece Length** box, type the length value for the spool piece.
6. Click **OK**.

The spool piece appears in the drawing.



Flange Isolation Kit

1. From the *Miscellaneous Fittings* tab, select **Flange Isolation Kit**.



2. In the drawing, click where you want to insert the start point.

The **Isolation Kit** Description dialog box appears.

3. In the **Description** input box, do one of the following:
 - Accept the default description
 - Type a new description
4. Click **OK**.

The flange isolation kit's INFO block appears in the drawing.

Quick Connector

1. From the *Miscellaneous Fittings* tab, select **Quick Connector**.



2. In the drawing, click where you want to insert the start point.
3. Click to select the direction.

The **Quick Connector** Description dialog box appears.

4. In the **Description** input box, do one of the following:
 - Accept the default description.
 - Type a new description

5. Click **OK**.

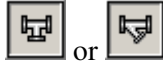
The quick connector appears in the drawing.

T & Y Strainers

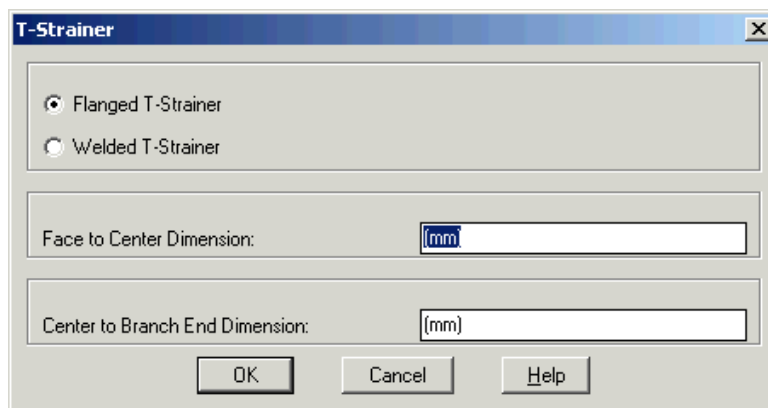
Select the correct isoplane setting before inserting strainers.

To insert a T & Y Strainer...

1. From the *Miscellaneous Fittings* tab, select **T-Strainer** or **Y-Strainer**.

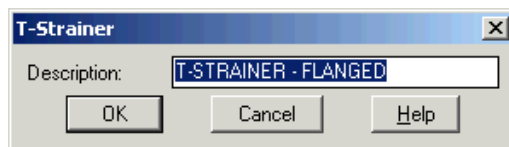


The **T-Strainer** or **Y-Strainer** dialog box appears.



2. From the options, select the flanged or welded strainer.
3. In the input boxes, type face-to-center and center-to-branch dimension values.
4. Click **OK**.
5. In the drawing, click to select the start point.
6. Click to select the main direction.
7. Click to select the branch direction.

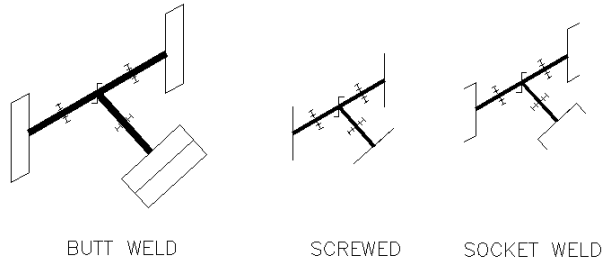
The **T-Strainer** or **Y-Strainer** Description dialog box appears.



8. In the input box, do one of the following:
 - Accept the default description
 - Type a new description

9. Click **OK**.

The strainer appears in the drawing.



Conical Strainers

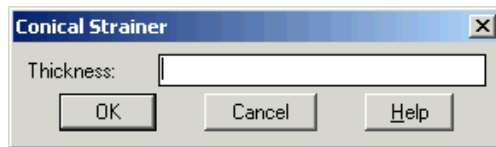
Select the correct isoplane setting before inserting strainers.

To insert a conical strainer...

1. From the **Miscellaneous Fittings** tab, select **Conical Strainer**.



The **Conical Strainer** Thickness dialog box appears.



2. In the **Thickness** box, type the thickness for the strainer.
3. Click **OK**.
4. In the drawing, click to select the start point.
5. Click to select the direction.

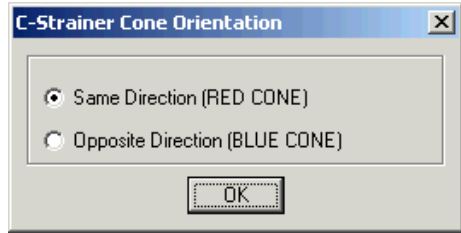
The **Conical Strainer** Description dialog box appears.



6. In the input box, do one of the following:
 - Accept the default description
 - Type a new description
7. Click **OK**.

The strainer appears in the drawing with different colored cones in each direction

The **C-Strainer Cone Orientation** dialog box appears.



8. From the options, select the required cone direction.
9. Click **OK**.
10. The strainer appears in the selected orientation.

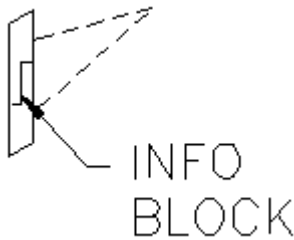
The **Bolt Length** notification box appears.

11. Click **OK**.

The **Bolt Length** override is now active.

When inserting bolts in a component grouping that contains a conical strainer, add the strainer's thickness and one extra gasket thickness to the bolt length.

- **Note:** If you do not know the required bolt length when you insert the conical strainer, use the **Dynamic Attribute Edit** tool later to edit bolt length information in the bolt's INFO block. See *Dynamic Attribute Edit*.



End Cover

Select the correct isoplane setting before inserting an end cover.

To insert an end cover...

1. From the *Miscellaneous Fittings* tab, select **End Cover**.



2. In the drawing, click where you want to insert the start point.
3. Click the direction you want the cover to appear.
4. The quick connector appears in the drawing.

Flanges, Gaskets, and Bolts

About Flanges

PROCAD SPOOLCAD includes a variety of flange types:

- Weld neck
- Long weld neck
- Screwed
- Socket weld
- Lap joint
- Slip-on
- Blind
- Nozzles
- Orifice
- MSS

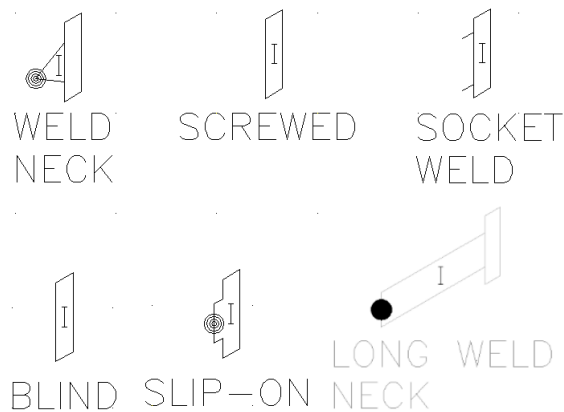
Flange face types include raised face, flat face, and ring type joint.

You can add flanges in conjunction with gaskets, bolts, mating flanges, or with valves.

You can insert flanges from weld or face-of-flange points.

Automated Selection of Flange Types

PROCAD SPOOLCAD automatically selects the flange type based on current spec and drawing settings. If the pipe size is in the butt weld fittings range, the system inserts a weld neck flange. Otherwise, the system inserts a screwed or socket weld flange.



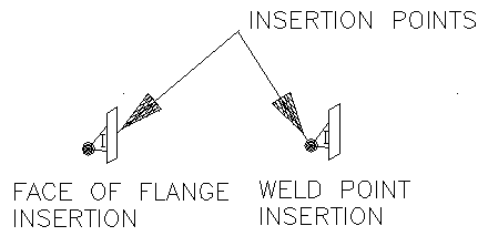
Out-of-Spec Flanges

You can add out-of-spec flanges, such as ring type joint flanges, in a spec that specifies raised face flanges. Set this override in the **Flange Face Override** options on the **Spec Overrides** dialog box. See *Piping Spec Overrides*.

When you insert the out-of-spec flange, a dialog box prompts you to accept the override or reset the settings to conform to the spec file.

Inserting Flanges

Insertion Point of Flanges



Inserting a Flange

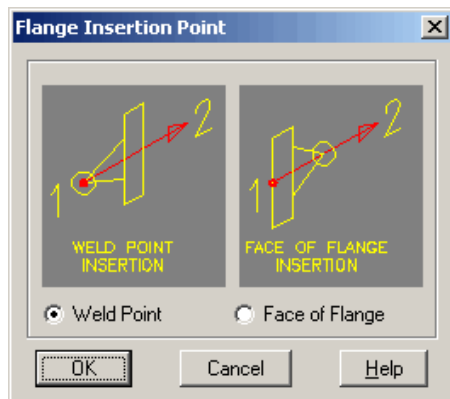
Select the correct isoplane setting before inserting flanges.

To insert a flange...

1. From the **Flanges** tab, select **Flange**.



The **Flange Insertion Point** dialog box appears.



2. From the options, select **Weld Point** or **Face of Flange**.

3. Click **OK**.
4. In the drawing, click to select the flange insertion point.
5. Click to select the direction.

The flange appears in the drawing.

Inserting a Long Weld Neck Flange

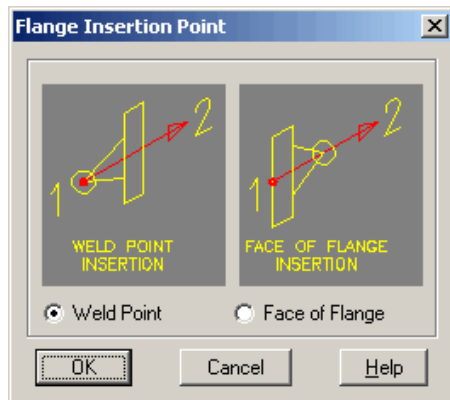
Select the correct isoplane setting before inserting flanges.

To insert a long weld neck flange...

1. From the **Flanges** tab, select **Long Weld Neck Flange**.



The **Flange Insertion Point** dialog box appears.



2. From the options, select **Weld Point** or **Face of Flange**.
3. Click **OK**.
4. In the drawing, click to select the flange insertion point.
5. Click to select the direction.

The flange appears in the drawing.

Inserting Flange Sets

Select the correct isoplane setting before inserting a flange set.

If the **Gasket Thickness Override** is active when you insert a flange set, a dialog box prompts you to confirm the current gasket thickness or type a new value. See *Gasket Thickness Override* for details.

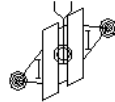
To insert a flange set complete with gasket and bolts...

1. From the **Flanges** tab, select **Flange Set**.



2. In the drawing, click to select the weld point of the flange.
3. Click to select direction of the flange.

The flange set appears in the drawing.



FLANGE SET WITH
GASKET AND BOLTS

Inserting Flange, Gasket, and Bolts

To insert a flange, gasket, and bolts...

1. From the **Flanges** tab, select **Flange, Gasket and Bolts**.

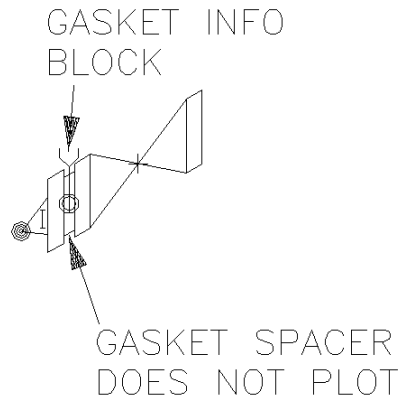


The **Flange Insertion Point** dialog box appears.

2. From the options, select the required insertion point.
3. Click **OK**.
4. In the drawing, click to select the insertion point.
5. Click to select the direction.

The flange, gasket and bolts appear in the drawing.

- **Notes:** When PROCAD SPOOLCAD inserts the gasket, it creates a spacer between flanges to indicate the presence of a gasket.
- PROCAD SPOOLCAD inserts the gasket spacer the DEFPOINTS layer, which AutoCAD does not plot.



Blind Flanges

To insert a blind flange...

1. From the **Flanges** tab, select **Blind Flange**.



2. In the drawing, click to select the insertion point.
3. Click to select the direction.

The flange appears in the drawing.

Lap Joint Flanges

To insert a lap joint flange...

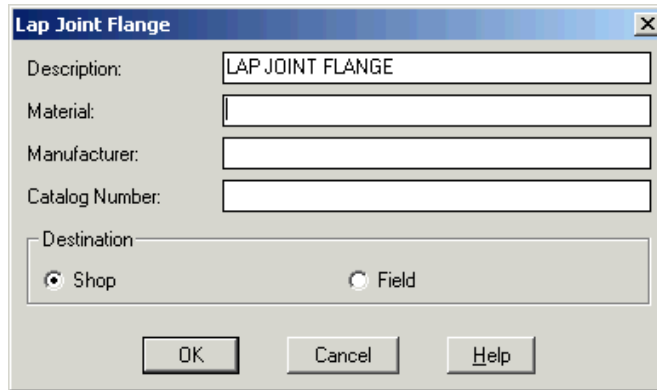
1. From the **Flanges** tab, select **Lap Joint Flange**.



The **Lap Joint Flange Insertion Point** dialog box appears.

2. From the options, select the required insertion point.
3. Click **OK**.
4. In the drawing, click to select the insertion point.
5. Click to select the direction.

The **Lap Joint Flange** dialog box appears.



6. In the input boxes, type the material, manufacturer and catalog number information.
7. Under **Destination**, select the **Shop** or **Field** option.
8. Click **OK**.

The flange appears in the drawing.

Slip-On Flanges

To insert a slip-on flange...

1. From the **Flanges** tab, select **Slip-On Flange**.



The **Flange Insertion Point** dialog box appears.

2. From the options, select the required insertion point.
3. Click **OK**.
4. In the drawing, click to select the weld (or face-of-flange) point.
5. Click to select the direction.

The flange appears in the drawing.

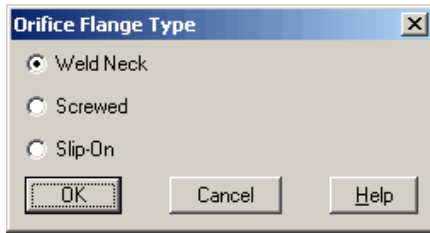
Orifice Flange Sets

To insert an orifice flange set...

1. From the **Flanges** tab, select **Orifice Flange Set**.

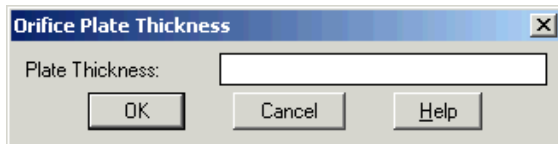


The **Orifice Flange Type** dialog box appears.



2. From the options, select the flange type
3. Click **OK**.
4. In the drawing, click to select the insertion point.
5. Click to select the direction.

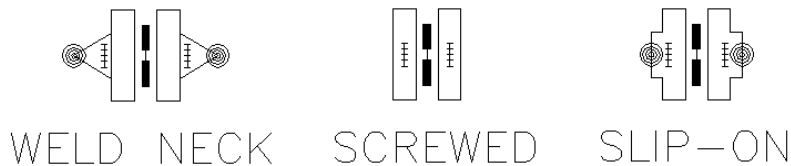
The **Orifice Plate Thickness** dialog box appears.



6. In the **Plate Thickness** box, type the orifice plate thickness.
7. Click **OK**.

The orifice flange appears in the drawing.

- **Note:** PROCAD SPOOLCAD adds two gasket thickness values to the dimension of the plate.



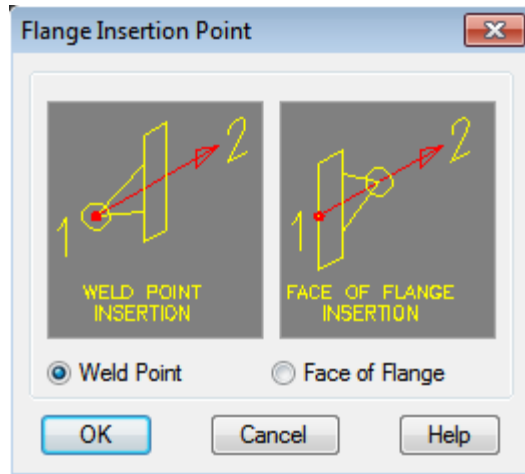
Single Orifice Flange

To insert a single orifice flange...

1. From the **Flanges** tab, select **Single Orifice Flange**.

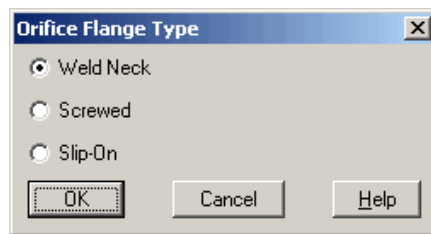


The **Flange Insertion Point** dialog box appears.



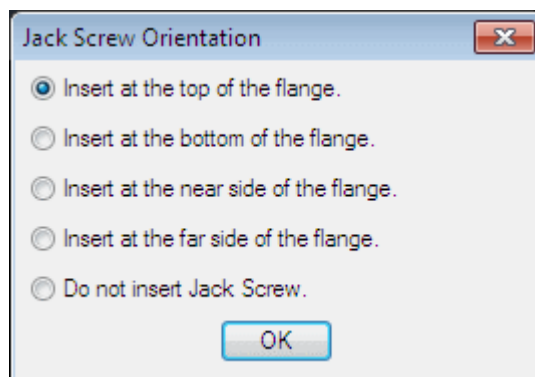
2. From the options, select the flange direction
3. Click **OK**.

The **Orifice Flange Type** dialog box appears.



4. From the options, select the flange type
5. In the drawing, click to select the insertion point.
6. Click to select the direction.

The **Jack Screw Orientation** dialog box appears.



7. From the options, select the jack screw orientation.
8. Click **OK**

Inserting a MSS Flange

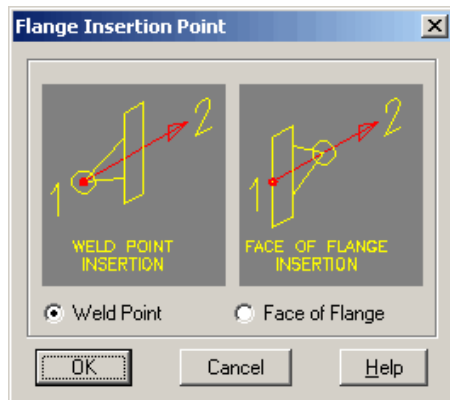
Select the correct isoplane setting before inserting flanges.

To insert a flange...

1. From the **Flanges** tab, select **MSS Flange**.



The **Flange Insertion Point** dialog box appears.



2. From the options, select **Weld Point** or **Face of Flange**.
3. Click **OK**.
4. In the drawing, click to select the flange insertion point.
5. Click to select the direction.

The flange appears in the drawing.

Inserting a MSS Blind Flange

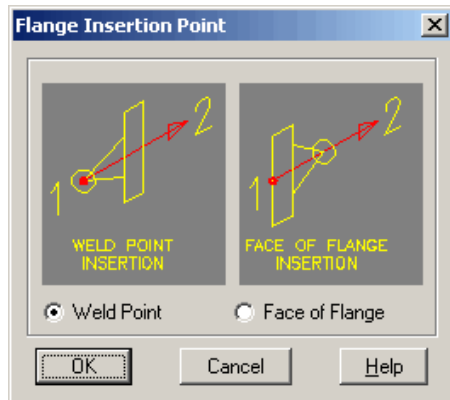
Select the correct isoplane setting before inserting flanges.

To insert a flange...

1. From the **Flanges** tab, select **MSS Blind Flange**.



The **Flange Insertion Point** dialog box appears.



2. From the options, select **Weld Point** or **Face of Flange**.
3. Click **OK**.
4. In the drawing, click to select the flange insertion point.
5. Click to select the direction.

The flange appears in the drawing.

API Flange

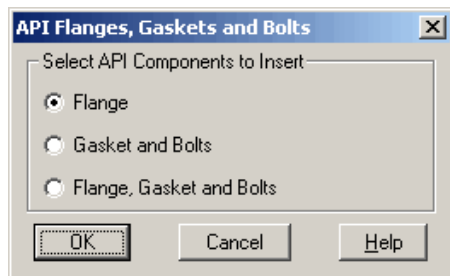
PROCAD SPOOLCAD uses a limited number of sizes for API flanges, gaskets and bolts. See *Appendix A: Pipe Sizes* for details.

To insert an API flange...

1. From the **Flanges** tab, select **API Flange**.

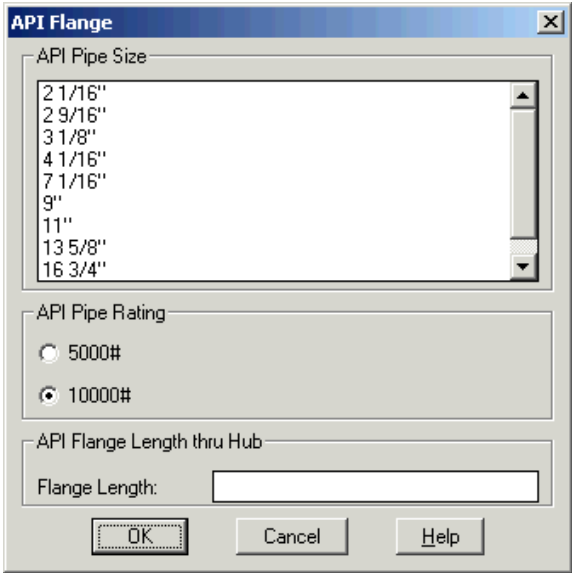


The **API Flanges, Gaskets and Bolts** dialog box appears.



2. From the options, select the required components.
3. Click **OK**.

The **API Flange** dialog box appears.

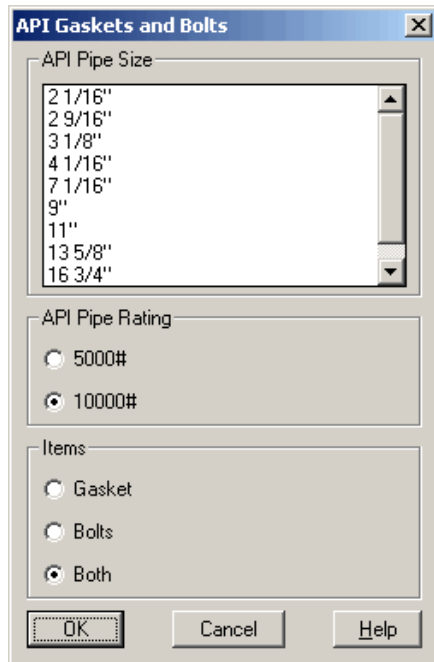


4. In this dialog box, do the following:

In this...	Do this...
API Pipe Size list	Select the appropriate pipe size
API Pipe Rating options	Select the required rating value
Flange Length input box	Type the required flange length

5. Click **OK**.
6. In the drawing, click to select the insertion point.
7. Click to select the direction.

If you selected **Gaskets and Bolts** in Step 1, the **API Gaskets and Bolts** dialog box appears.



8. In this dialog box, do the following:

In this...	Do this...
API Pipe Size list	Select the appropriate pipe size
API Pipe Rating options	Select the required rating value
Items options	Select the items you want to insert

9. Click **OK**.

The API flange appears in the drawing.

- **Note:** If material information is not available for the selected pipe size and you choose to continue the API flange insertion, dialog boxes prompt you for gasket thickness, number of bolts, bolt diameter and bolt length.

Nozzles

To insert a nozzle...

1. From the **Flanges** tab, select **Nozzle**.

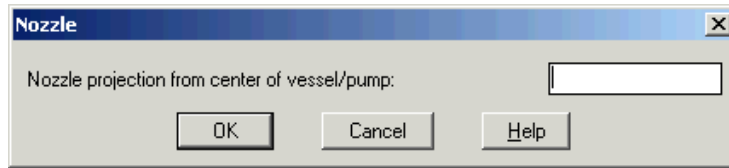


The **Nozzle Insertion Point** dialog box appears.

2. From the options, select the required insertion point option.
3. Click **OK**.

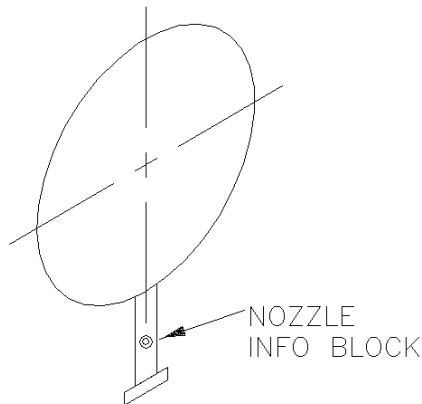
4. In the drawing, click to select the insertion point.
5. Click to select the direction.

The **Nozzle** dialog box appears.




6. In the input box, type a value for the nozzle projection.
7. Click **OK**.

The nozzle appears in the drawing.

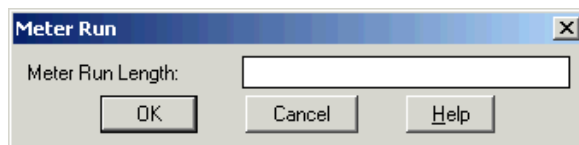


Meter Run

To insert a meter run...

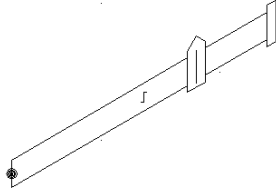
1. From the **Flanges** tab, select **Senior Meter Run**.

2. In the drawing, click to select the insertion point option.
3. Click to select the direction.
4. Click to select the end point.

The **Meter Run** dialog box appears.



5. In the **Meter Run Length** box, type the length value.
6. Click **OK**.

The meter run appears in the drawing.



Gaskets

The current spec determines the gasket type and thickness. To override the gasket thickness, see *Gasket Thickness Override*.

To insert a gasket...

1. From the **Gaskets and Bolts** tab, select **Gaskets**



2. In the drawing, click to select the start point.
3. Click to select the line direction.

The gasket appears in the drawing.

- **Notes:** When PROCAD SPOOLCAD inserts the gasket, it creates a space between flanges to indicate the presence of a gasket.
- PROCAD SPOOLCAD inserts the gasket spacer the DEFPOINTS layer, which AutoCAD does not plot.

Gasket and Bolts

To insert a gasket and associated bolts...

1. From the **Gaskets and Bolts** tab, select **Gasket + Bolts**.



2. In the drawing, click to select the start point.
3. Click to select the line direction.

The gasket and bolts appear in the drawing.

Inserting Gasket Tick Marks

After dimensioning, you can insert gasket tick marks to indicate on which dimension side to include the gasket's thickness.

To insert gasket thickness tick marks...

1. From the **Gasket and Bolts** tab, select **Gasket Tick Marks**.

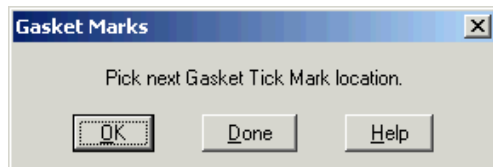


The **Gasket Marks** isoplane orientation dialog box appears.

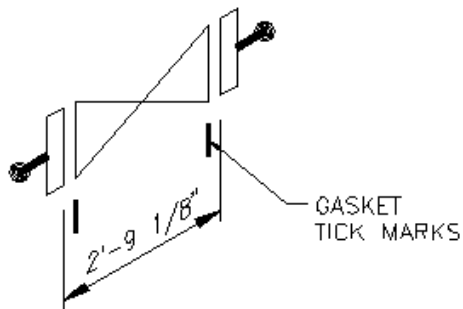
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the first tick mark location.

The tick mark appears in the drawing.

The **Gasket Marks** dialog box appears.



4. Choose one of the following options:
 - Click **OK** to insert another tick mark
 - Click **Done** to exit the tool



Overriding the Gasket Thickness

See *Setting Spec Overrides* and *Gasket Thickness Override*.

Bolts

When you insert components like bleed rings and spectacle blinds, you will need to add that component's width plus one gasket thickness to the bolt length.

If you do not know the required bolt length when you insert the component, use the **Dynamic Attribute Edit** tool later to edit bolt length information in the bolt's INFO block. See *Dynamic Attribute Edit*.

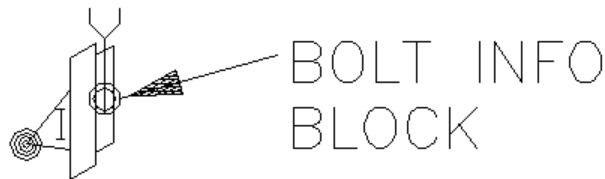
To insert a bolt...

1. From the **Gasket and Bolts** tab, select **Bolts**.



2. In the drawing, click to select the insertion point.

The bolt INFO block appears in the drawing.



Overriding Bolt Length

When you require longer bolts for such items as butterfly valves, you can override the length.

See *Bolt Length Override*.

Changing the Bolt Type

Use this procedure when you require bolts that differ from the type set in the current spec.

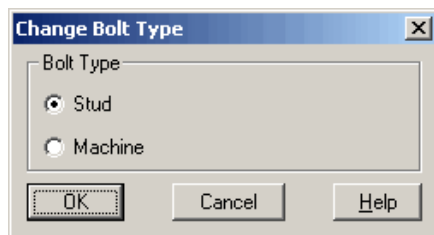
When you insert a bolt after changing the bolt type, an override dialog box appears if the new bolt type is out-of-spec.

To change the bolt type...

1. From the **Gasket and Bolts** tab, select **Change Bolt Type**.



The **Change Bolt Type** dialog box appears.



2. From the **Bolt Type** options, select the required bolt type.
3. Click **OK**.

API Gaskets and Bolts

To insert API gaskets and bolts...

See the *API Flange* section.

Valves

Valve Selection

The following valves are available in PROCAD SPOOLCAD:

- Gate Valves
- Globe Valves
- Ball Valves
- Check Valves
- Plug Valves
- Angle Valves
- Needle Valves
- 3-Way Valves
- 4-Way Valves
- Butterfly Valves
- Thermal Valves
- Pressure Safety Valves
- Control Valves

➤ **Note:** See *Appendix C: Valve Types* for details.

Valve Ends

PROCAD SPOOLCAD extracts valve dimensions from the dimensional data files. To modify valve dimensions, use *Spec Generator*. Refer to the *Spec Generator User Guide*.

PROCAD SPOOLCAD selects the valve type based on the current drawing settings and specifications, except when you set certain overrides. The following settings determine which valve type appears in the drawing:

Fittings Size Range	Overrides Set	Valve Inserted
Butt weld	None	Flanged
Butt weld	Welded valve	Welded
Screwed/socket weld	Small Fittings-set to Screwed	Screwed
Screwed/socket weld	Small Fittings-set to Socket weld	Socket weld
Screwed/socket weld	Small Fittings-set to Socket weld; Screwed by socket weld	Screwed by socket weld

Inserting Valves

Inserting General Valves

Select the correct isoplane setting before inserting valves.

Use this procedure to insert the following valves:

- Flanged
- Welded
- Screwed
- Socket Welded

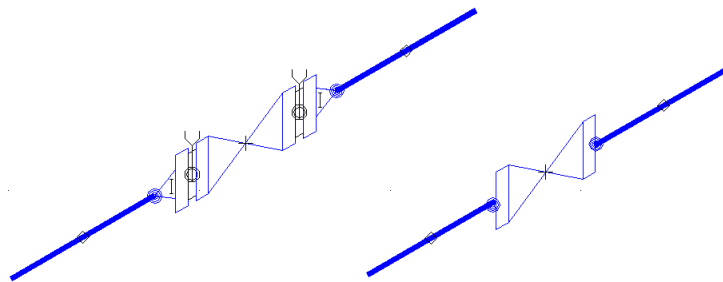
To insert a valve...

1. From the **Valves** tab, select a valve type.
2. In the drawing, click to select the insertion point.
3. Click to select the direction.

The valve appears in the drawing.

If the **Valve Stem** toggle is active, the **Add Stem?** dialog box appears (see *Inserting Valve Stems*).

- **Note:** Socket weld valves symbols appear with socket extensions on both sides of the valve.



Flanged Valves

Welded Valves



Screwed Valves



Socket Weld Valves

Inserting Bolt-Through Valves

Insert bolt-through valves, such as butterfly valves, in conjunction with a valve assembly (see *Inserting Valve Assemblies*).

When you insert a bolt-through valve, a notification box reminds you to add the valve's thickness and one gasket thickness to the bolt length. Inserting bolt-through valves activates the bolt length override, and deactivates it when you complete the valve insertion.

Inserting Screwed by Socket Weld Valves

You must click the **Screwed/SW Valve** override button on the *Settings* tab every time you want to insert a screwed by socket weld valve.

You can insert only the following valves as screwed by socket weld: gate, globe, ball, plug and check.

Select the correct isoplane setting before inserting socket weld valves.

To insert a screwed by socket weld valve...

Ignore Steps 1-3 if the current spec specifies socket weld fittings.

1. From the *Settings* tab, select **Spec Override**.



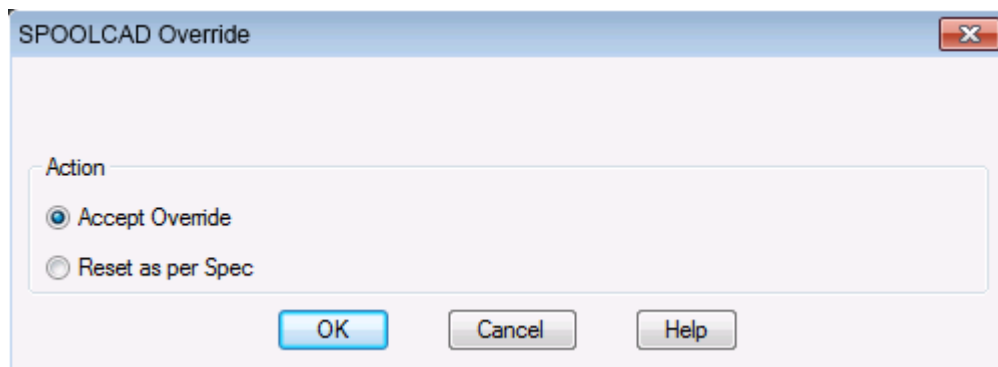
The **Spec Overrides** dialog box appears.

2. In the **Small Fitting Overrides** options, select **Socket Welded**.
3. Click **OK**.
4. On the *Settings* tab, click **Screwed/SW Valve**.



5. On the *Valves* tab, select the valve you want to insert.

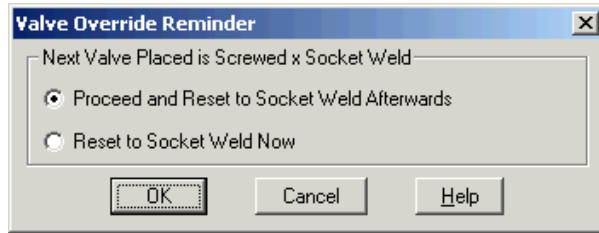
The **SPOOLCAD Override** dialog box appears.



6. Under **Action**, click **Accept Override**.

7. Click **OK**.

The **Valve Override Reminder** dialog box appears.



8. From the options, click the **Proceed and Reset to Socket Weld Afterwards** option.
9. Click **OK**.
10. In the drawing, click to select the start point.
11. Click to select the direction.

The valve appears in the drawing.

If the **Valve Stem** toggle is active, the **Add Stem?** dialog box appears (see *Inserting Valve Stems*).

The valve appears as socket weld on both sides of the valve.

12. Use AutoCAD's **Trim** function to trim the lines on the threaded side.
 - If valve rating information is not available in the data files, a dialog box prompts you to select the appropriate valve rating.

Inserting Out-of-spec Valves

When you insert out-of-spec valves, a dialog box prompts you to either accept the override or reset the settings to conform to the spec file.

Inserting Valve Stems

When the **Valve Stem** toggle is active, a dialog box prompts you to insert stems after placing a valve.

Use this procedure to insert valve stems manually.

To insert a valve stem...

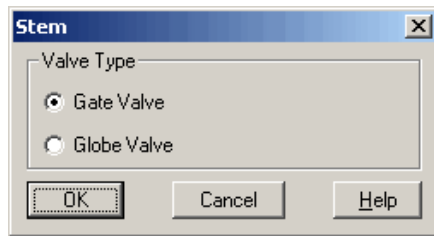
1. From the **Valves** tab, select **Valve Stem**.



2. In the drawing, click to select the valve's center point.

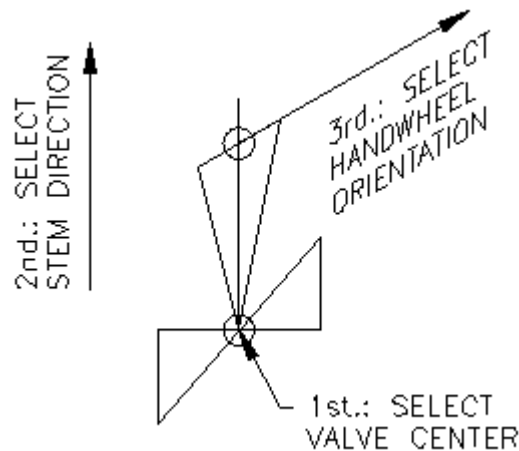
3. Click to select the stem direction.
4. Click to select the hand wheel direction.

The **Stem** dialog box appears.



5. From the options, select the valve type.
6. Click **OK**.

The stem appears in the drawing.



Inserting Valve Stems Automatically

Activate the valve stem toggle. See *Valve Stem Toggle*. When you insert valves, a dialog box prompts you to insert the valve stem.

Draw gate and angle valves with a rising stem. Draw globe valves with a non-rising stem.

Inserting Valve Assemblies

Spec and drawing settings determine which type of valve appears in the drawing. You must click a valve assembly button each time you want to insert a valve assembly.

Select the correct isoplane setting before inserting valve assemblies.

To insert a valve assembly...

1. From the **Valves** tab, select one of the valve assembly options.



Flange + Gasket + Bolts + Valve



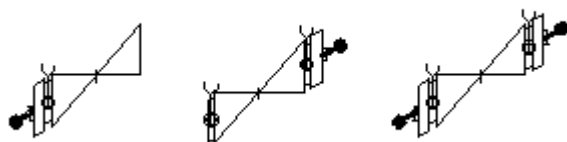
Gasket + Bolts + Valve + Bolts + Gasket + Flange



Flange + Gasket + Bolts + Valve + Bolts + Gasket + Flange

2. On the **Valves** tab, click the valve type you want to insert.
3. In the drawing, click to select the valve's start point.
4. Click to select the direction.

The valve assembly appears in the drawing.



Inserting Control Valves

Select the correct isoplane setting before inserting control valves.

You can insert control valves when spec and drawing settings indicate flanged, welded, or screwed fittings.

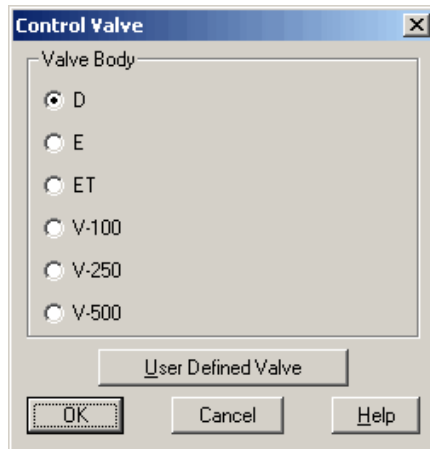
You can also use the valve assembly tools when inserting control valves. See *Inserting Valve Assemblies*.

To insert a control valve...

1. From the **Valves** tab, select **Control Valve**.

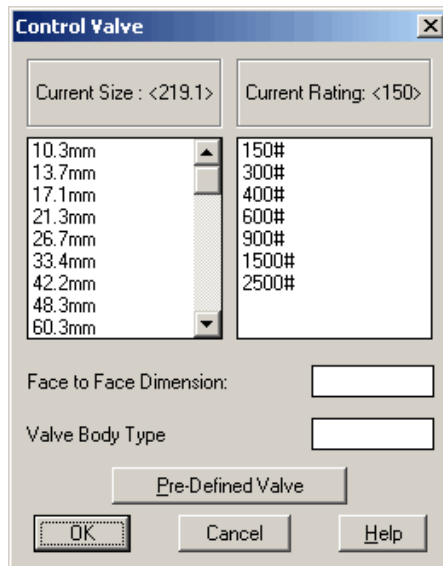


The **Control Valve** dialog box appears.



2. From the **Valve Body** options, select one of the following options:
 - Click the required valve body type
 - Click **User Defined Valve**

If you click **User Defined Valve**, the **Control Valve** user-defined valve dialog box appears.



- **Note:** When placing V100 control valves, type V10 in the **Valve Body Type** box.

3. In this dialog box, do the following:

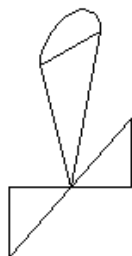
In this...	Do this...
Current Size scroll list	Select the appropriate size
Current Rating list	Select the required valve rating
Face to Face Dimension input box	Type the required face-to-face dimension value
Valve Body Type input box	Type the required valve body description

4. Click **OK**.
5. In the drawing, click to select the start point.
6. Click to select the direction.

The valve appears in the drawing.

If the valve stem toggle is active, the **Add Actuator?** dialog box appears. See *Inserting a Control Valve Actuator Manually* below.

➤ **Notes:** Control valves appear in the Bill of Material.



Inserting a Control Valve Actuator Manually

When the valve stem toggle is active, a dialog box usually prompts you to insert an actuator when you insert the control valve. See *Valve Stem Toggle*.

To manually insert a control valve actuator...

1. From the **Valves** tab, select **Control Valve Actuator**.



2. In the drawing, click to select the center point.
3. Click to select the stem direction.
4. Click to select the actuator direction.

The actuator appears in the drawing.

Inserting Pressure Safety Valves

PROCAD extracts dimensional data for pressure safety valves (PSV) from Farris catalogues.

You can modify PSV component data in *Spec Generator* to conform to another manufacturer's PSV dimensional data. Refer to the *Spec Generator User Guide*. Refer to *Appendix D: Pressure Safety Valve List* for available PSV types.

You can use the valve assembly tools when inserting pressure safety valves. See *Inserting Valve Assemblies*.

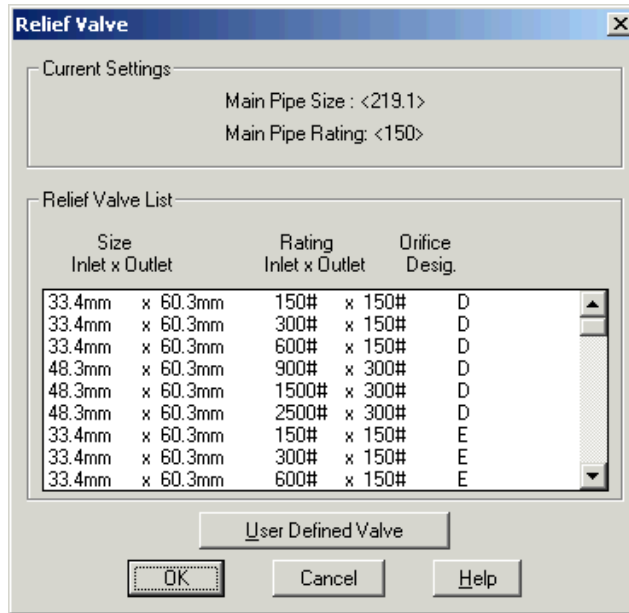
Select the correct isoplane setting before inserting pressure safety valves. The current isoplane sets only the inlet flange orientation.

To insert a Pressure Safety Valve...

1. From the **Valves** tab, select **PSV Valve**.

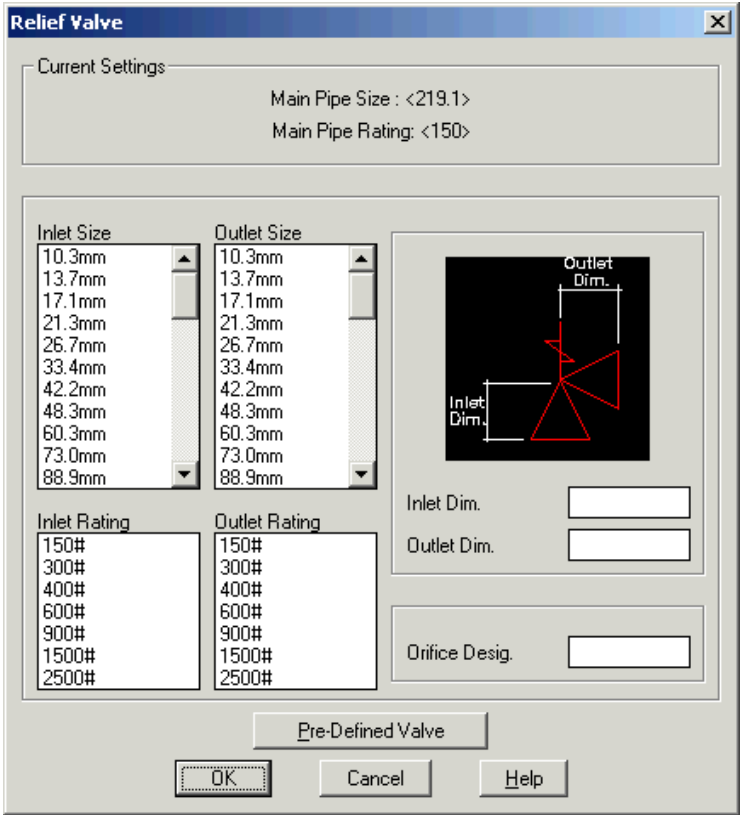


The **Relief Valve** dialog box appears.



2. Choose one of the following options:
 - From the list, select a PSV type
 - Click **User Defined Valve**

If you click **User Defined Valve**, the **Relief Valve** Current Settings dialog box appears.

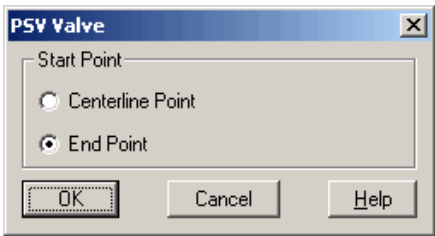


3. In this dialog box, do the following:

In this...	Do this...
Inlet Size and Outlet Size scroll lists	Select the appropriate inlet and outlet sizes
Inlet Rating and Outlet Rating lists	Select the required inlet and outlet rating values
Inlet Dim. input box	Type the required inlet dimension value
Outlet Dim. input box	Type the required outlet dimension value
Orifice Desig. input box	Type the required orifice designation description

4. Click **OK**.

For both spec-defined and user-defined valves, the **PSV Valve Start Point** dialog box appears.



5. From the options, select a start point option.

6. Click **OK**.
7. In the drawing, click to select the insertion point.
8. Click to select the inlet flange direction.

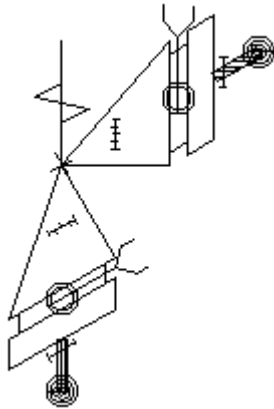
The **PSV Outlet Flange Orientation** dialog box appears.

9. From the dialog box options, select the outlet orientation.
10. Click **OK**.
11. In the drawing, click to select the outlet direction.

The valve appears in the drawing.

The PROCAD SPOOLCAD Message box appears, indicating the new pipe size setting.

- **Notes:** Use longer bolts for the inlet side. PSV inlets usually have thicker than standard flanges.
- PSVs appear in the Bill of Material.



Valve Overrides and Toggles

Flanged or Welded Valves Override

When fittings are in the butt weld range, PROCAD SPOOLCAD places flanged valves. To place welded valves, activate the welded valve override. See *Welded Valves Override*.

Face to Face Dimension Override

See *Valve Face to Face Dimension Override*.

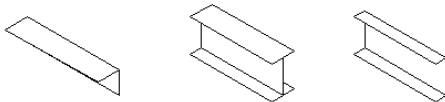
Valve Stem Toggle

See *Valve Stem*.

Supports

Steel Members

PROCAD SPOOLCAD contains three steel member drafting routines.



Steel members show support locations only. They do not appear in the Bill of Material.

Beam or Wide Flange

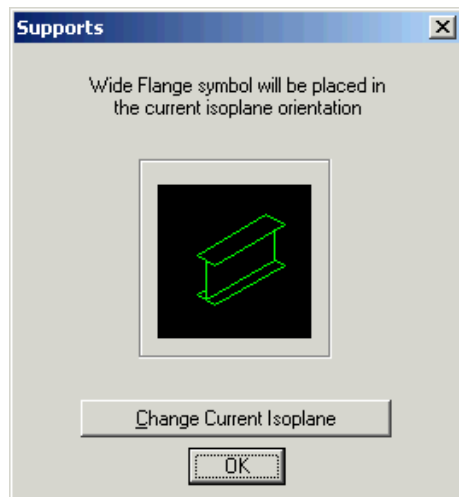
Select the correct isoplane setting before inserting a beam.

To insert a beam or wide flange...

1. From the *Structural Steel* tab, select **Wide Flange**.



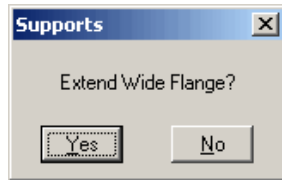
The **Supports** isoplane orientation dialog box appears.



2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting

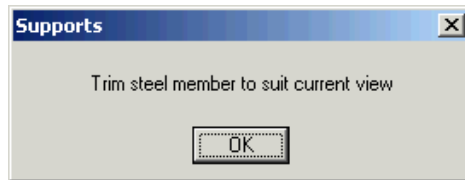
3. In the drawing, click to select the insertion point of the beam member. The beam section appears in the drawing.

The **Supports** dialog box appears.



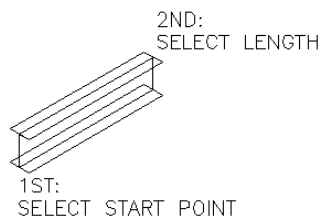
4. Choose from the following options:
 - Click **Yes** to extend the flange; go to Step 5
 - Click **No** to end the procedure; the support section appears in the drawing
5. In the drawing, click to select the flange length.

The **Supports** dialog box appears.



6. Click **OK**.
- The beam appears in the drawing.
7. Use AutoCAD's TRIM command to trim the hidden lines of the beam extension side if necessary.

➤ **Note:** You must disable Object Grouping before you can trim lines.



Channel

Select the correct isoplane setting before inserting a channel.

To insert a channel...

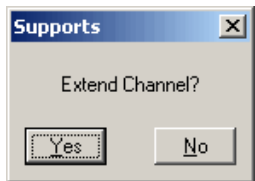
1. From the **Structural Steel** tab, select **Channel**.



The **Supports** isoplane orientation dialog box appears.

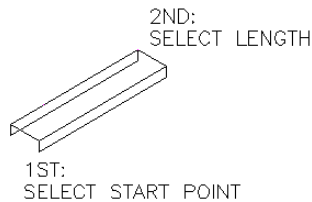
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the insertion point. The channel section appears in the drawing.

The **Supports** dialog box appears.



4. Choose from the following options:
 - Click **Yes** to extend the channel; go to Step 5
 - Click **No** to end the procedure; the channel section appears in the drawing
 5. In the drawing, click to select the channel length.
- The **Supports** dialog box appears.
6. Click **OK**.
- The channel appears in the drawing.
7. Use AutoCAD's TRIM command to trim the hidden lines of the beam extension side if necessary.

➤ **Note:** You must disable Object Grouping before you can trim lines.



Angle

Select the correct isoplane setting before inserting an angle support.

To insert an angle...

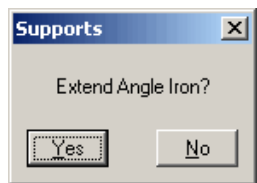
1. From the **Structural Steel** tab, select **Angle**.



The **Supports** isoplane orientation dialog box appears.

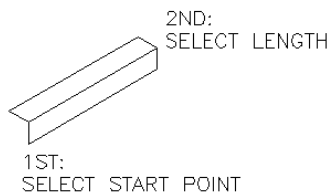
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the insertion point. The angle section appears in the drawing.

The **Supports** dialog box appears.



4. Choose from the following options:
 - Click **Yes** to extend the angle; go to Step 5
 - Click **No** to end the procedure; the angle section appears in the drawing
 5. In the drawing, click to select the angle length.
- The **Supports** dialog box appears.
6. Click **OK**.
- The channel appears in the drawing.
7. Use AutoCAD's TRIM command to trim the hidden lines of the beam extension side if necessary.

➤ **Note:** You must disable Object Grouping before you can trim lines.



Column Identification Balloons

Use this procedure to insert column identification balloons.

Select the correct isoplane setting before inserting column identification balloons.

To insert a column identification balloon...

1. From the **Pipe Supports** tab, select **Column I.D. Balloon**.



The **Column I.D.** isoplane dialog box appears.

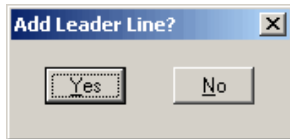
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the balloon insertion point.

The **Column I.D.** Number dialog box appears.



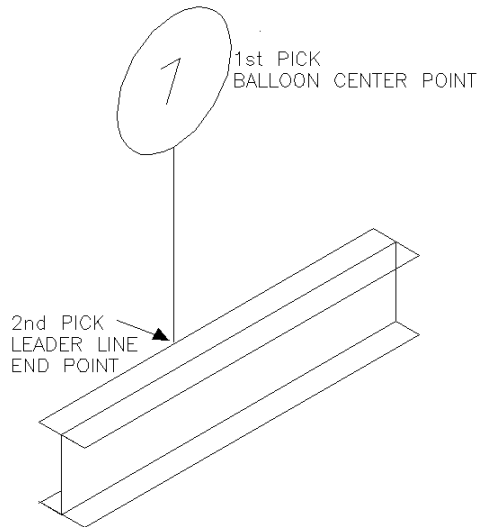
4. In the **Column ID Number** box, type a column number.
5. Click **OK**.

The **Add Leader Line?** dialog box appears.



6. Choose one of the following options:
 - Click **No** to insert the balloon without a leader line
 - Click **Yes** to insert the balloon with a leader line
7. If you click **Yes**, click in the drawing to select the leader line end point.

The balloon and leader line appear in the drawing.



Pipe Supports

PROCAD SPOOLCAD extracts material information from the currently selected spec. To modify support specification information, use *Spec Generator*. Refer to the *Spec Generator User Guide*.

Pipe supports appear in the Bill of Material.

Pipe Shoe

Select the correct isoplane setting before inserting a pipe shoe.

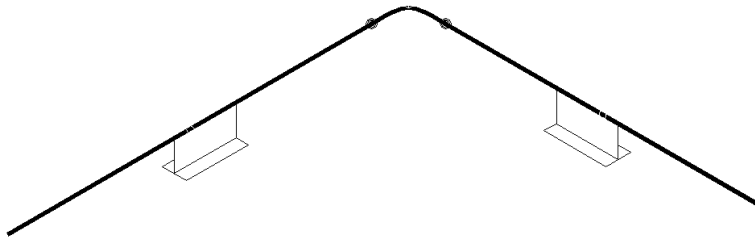
To insert a pipe shoe...

1. From the *Pipe Supports* tab, select **Pipe Shoe**.



2. In the drawing, click the pipe where you want to insert the shoe.

The pipe shoe appears in the drawing.



Pipe Guide

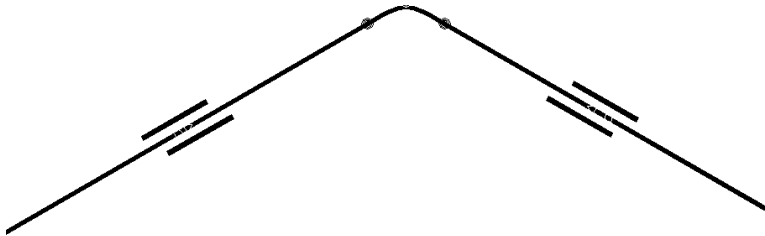
Select the correct isoplane setting before inserting a pipe guide.

To insert a pipe guide...

1. From the *Pipe Supports* tab, select **Pipe Guide**.



2. In the drawing, click the pipe where you want to insert the guide.
The pipe guide appears in the drawing.



Pipe Anchor

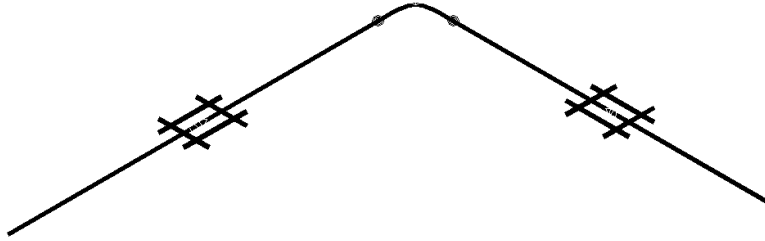
Select the correct isoplane setting before a pipe anchor.

To insert a pipe anchor...

1. From the *Pipe Supports* tab, select **Pipe Anchor**.



2. In the drawing, click the pipe where you want to insert the anchor.
The pipe anchor appears in the drawing.



Base Support

Select the correct isoplane setting before inserting a base support.

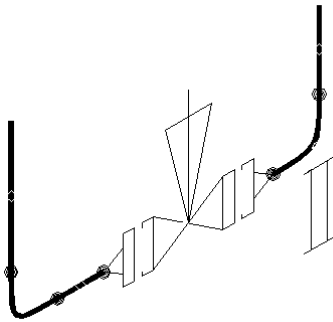
To insert a base support...

1. From the *Pipe Supports* tab, select **Base Support**.



2. In the drawing, click the center point of the fitting where you want to insert the base support.
3. Click to select the end point of the base.
4. Click to select the side of the support.

The base support appears in the drawing.



Base Guide

Select the correct isoplane setting before inserting base guides.

To insert a base guide...

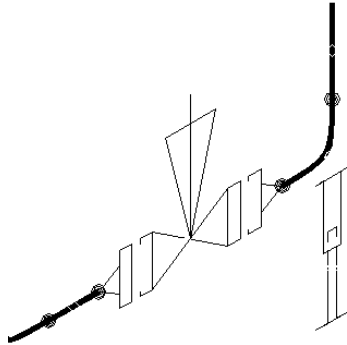
1. From the *Pipe Supports* tab, select **Base Guide**.



This icon is similar to the **Base Support** icon. Move your mouse over the button image to view the description text.

2. In the drawing, click the center point of the fitting where you want to insert the base guide.
3. Click to select the end point of the base.
4. Click to select the side of the support.

The support appears in the drawing.



Spring

Select the correct isoplane setting before inserting a spring.

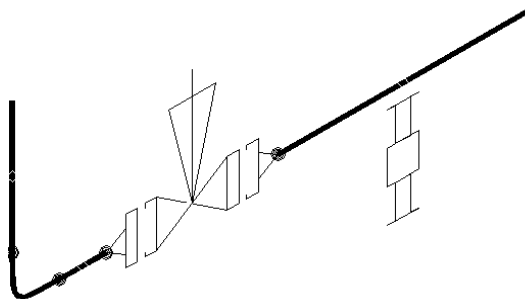
To insert a spring...

1. From the *Pipe Supports* tab, select **Spring**.



2. In the drawing, click the pipe or fitting where you want to insert the spring.
3. Click to select the end point of the base.
4. Click to select the side of support.

The spring appears in the drawing.



Dummy Leg

Select the correct isoplane setting before inserting a dummy leg support.

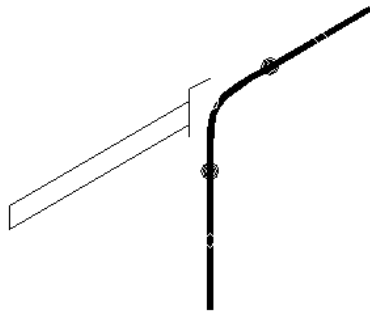
To insert a dummy leg...

1. From the **Pipe Supports** tab, select **Dummy Leg**.



2. In the drawing, click the center point of the fitting where you want to insert the dummy leg.
3. Click to select the end point of the base.
4. Click to select the top of the support.

The dummy leg appears in the drawing.



Pipe Hanger

Select the correct isoplane setting before inserting a pipe hanger.

To insert a pipe hanger...

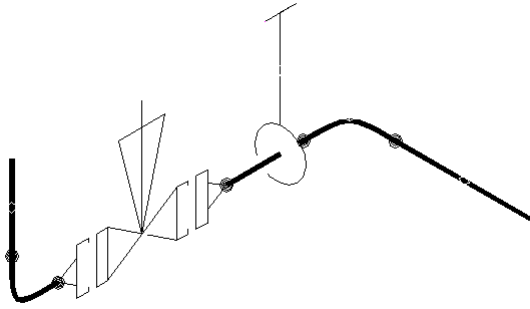
1. From the **Pipe Supports** tab, select **Pipe Hanger**.



2. In the drawing, click the pipe where you want to insert the hanger.
3. Click to select the hanger clamp location.

The pipe hanger appears in the drawing.

- **Note:** PROCAD SPOOLCAD breaks the pipe when you insert a pipe hanger.



Spring Hanger

Select the correct isoplane setting before inserting a spring hanger.

To insert a spring hanger...

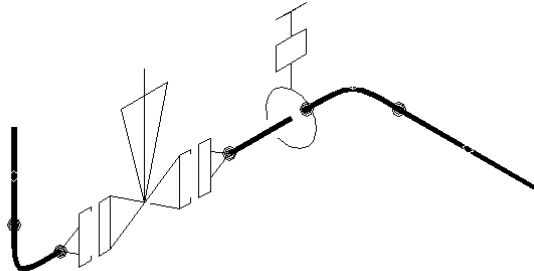
1. From the **Pipe Supports** tab, select **Spring Hanger**.



2. In the drawing, click the pipe where you want to insert the spring hanger.
3. Click to select the attachment point.

The spring hanger appears in the drawing.

- **Note:** PROCAD SPOOLCAD breaks the pipe when you insert a spring hanger.



U-Bolt

Select the correct isoplane setting before inserting a u-bolt.

To insert a u-bolt...

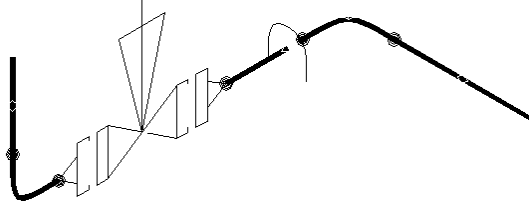
1. From the **Pipe Supports** tab, select **U-Bolt**.



2. In the drawing, click the pipe where you want to insert the u-bolt.

The u-bolt appears in the drawing.

➤ **Note:** PROCAD SPOOLCAD breaks the pipe when you insert a u-bolt.



Trunnion

Select the correct isoplane setting before inserting a trunnion.

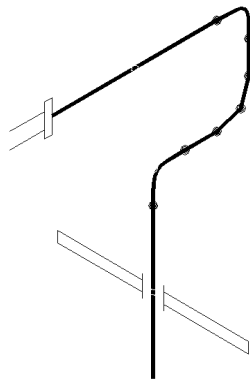
To insert a trunnion...

1. From the **Pipe Supports** tab, select **Trunnion**.



2. In the drawing, click the pipe where you want to insert the trunnion.
3. Click to select the end point of one side of support.
4. Click to select the top of the support.

The trunnion appears in the drawing.



PROCAD SPOOLCAD Dimensioning

Dimension Text

The dimensioning system uses the following text styles:

Text Tool	Font	Compression
Standard	ROMANS	100%
Standard-80 (compressed)	ROMANS	80%
ISO- L	ROMANS	100%
ISO-L80 (compressed)	ROMANS	80%
ISO-R	ROMANS	100%
ISO-R80 (compressed)	ROMANS	80%

See *Dimension Text Utilities*.

Setting a Weld Root Gap

When you insert a butt weld fitting, PROCAD SPOOLCAD adds half of the root gap to the fitting's dimension. When dimensioning butt weld fittings, the two half root gaps added to each fitting add up to a full root gap in the dimension. The default setting is 0.

Dimensioning

General Dimensioning Tips

Type **'u'** in the command line to delete the last dimension inserted.

Type **'c'** in the command line to change the isoplane setting.

Press **Enter** to complete the dimension string and exit the dimensioning tool.

Dimensioning Valves

When dimensioning valves, click to select the face of mating flanges to include the gaskets within the valve dimension. You can insert gasket ticks to indicate the dimension includes gasket thickness. See *Inserting Gasket Tick Marks*.

If you want to include a gasket in the dimension, pick the face of the flange beyond the gasket.

If you do not want to include a gasket in the dimension, pick the face of the flange before the gasket.

When dimensioning small fitting (screwed and socket weld) valves, include the entire valve within the dimension origin points. Never dimension to the center of a small fitting valve.

Dimensioning Pipe Lengths

When dimensioning pipe lengths, a dialog box prompts you to supply the overall dimension between two selected origin points. PROCAD SPOOLCAD subtracts the dimensions of any fittings in the selection and inserts the resulting pipe length in the pipe's INFO block. PROCAD SPOOLCAD uses this data when generating the Bill of Material.

You can dimension only one pipe length at a time. If more than one pipe length occurs between origin points, PROCAD SPOOLCAD cancels the second point and prompts you to re-select the point to include one pipe segment only.

If you set the weld root gap setting to more than zero, PROCAD SPOOLCAD deducts the full root gap from the pipe's cut length. See *Setting a Weld Root Gap*.

Drawing settings determine the required minimum pipe length between fittings. When the user-supplied dimension does not accommodate the required pipe length, a dialog box prompts you for a different value.

If you have already calculated the pipe length using the **Calculate Pipe Length** tool, no dialog box prompts appear.

Changing the Isoplane During Dimensioning

If you select an origin point that conflicts with the current isoplane, the following message appears in the command line:

```
Direction incompatible with current isometric plane setting.  
Select a different direction.
```

If the second dimension origin point includes an elbow, the **Select Isometric Dimension Plane** dialog box appears. From this dialog box you can do the following:

- Select a different isoplane and continue dimensioning
- Undo the last dimension
- Insert an offset dimension
- Insert a rolling offset dimension
- Exit the dimensioning tool

You can also change the isoplane manually by typing 'c' at the command line prompt. This option also displays the **Select Isometric Dimension Plane** dialog box.

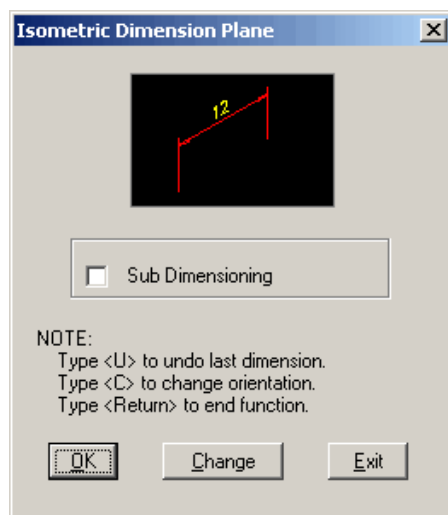
Dimensioning a Drawing

To insert dimensioning...

1. From the *Utilities* tab, select **Iso Dimensioning**.



The **Isometric Dimension Place** dialog box appears.



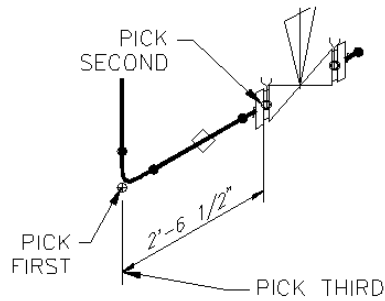
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the first extension line origin point.
4. Click to select the second extension line origin point.
5. Click to select the dimension line location. PROCAD SPOOLCAD extracts the dimensional data and inserts the dimension lines and text.

If there is no dimensional information available, the **SPOOLCAD Input** box appears. In the input box, type the dimension value and click **OK**.

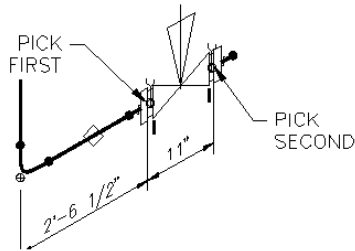
The dimension lines and text appear in the drawing.

6. In the drawing, continue selecting origin points.
7. Press **Enter** to complete the dimensioning string.

Selecting the first dimension segment:



Selecting subsequent dimension segments:



Sub Dimensioning

Sub dimensioning (such as from an elbow to a Thredolet®) prevents PROCAD SPOOLCAD from extracting the dimensional data from component INFO and DIMENSION blocks, and prompts you to enter the value manually.

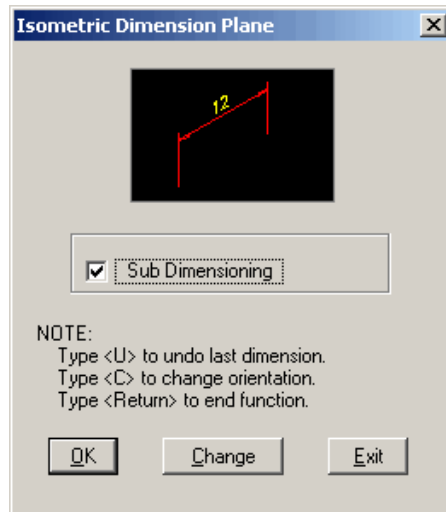
If you want to insert a second level of sub-dimensioning, exit the dimensioning tool and repeat the sub-dimensioning procedure.

To insert sub-dimensioning...

1. From the *Utilities* tab, select **Iso Dimensioning**.



The **Isometric Dimension Plane** dialog box appears.



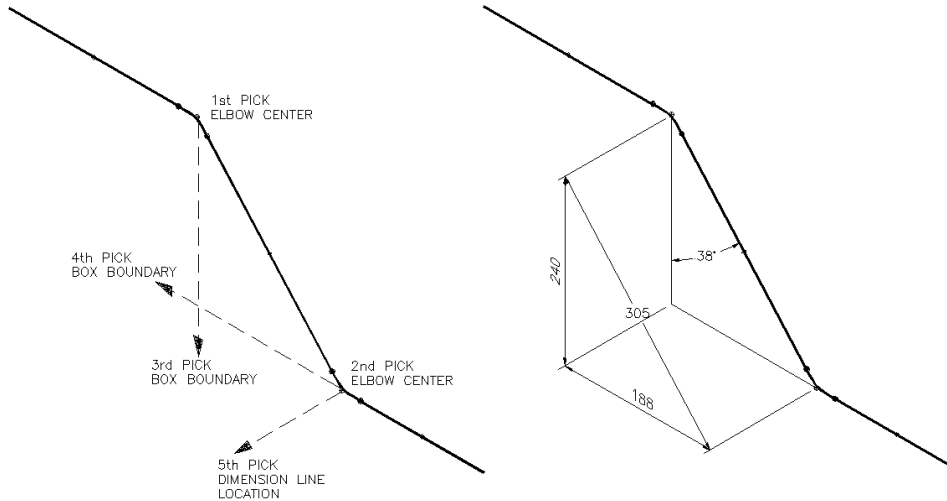
2. Click in the **Sub Dimensioning** checkbox.
3. Click **OK**.
4. In the drawing, click to select the first extension line origin point.
5. Click to select the second extension line origin point.
The **SPOOLCAD Input** dialog box appears.
6. In the input box, type the overall dimension value.
7. Click **OK**.
The dimension lines and text appear in the drawing.
8. In the drawing, continue selecting origin points.
9. Press **Enter** to complete the dimensioning string.

Offset Dimensioning

You can insert two types of offset dimensioning:

- Fitting to Fitting
- Fittings + Pipe

Important: Select the correct isoplane before inserting an offset dimension. If you select an incorrect isoplane, the dimensioning will display an incorrect dimension or no dimension.



Fitting to Fitting Offset Dimensioning

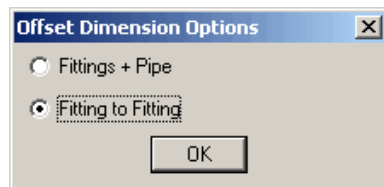
Select the correct isoplane before inserting dimensioning.

To insert a fitting to fitting offset dimension...

1. From the **Utilities** tab, select **Offset**.

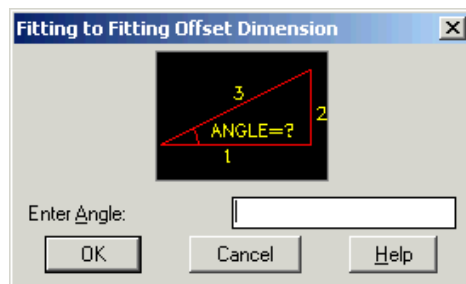


The **Offset Dimension Options** dialog box appears.



2. From the options, select **Fitting to Fitting**.
3. Click **OK**.

The **Fitting to Fitting Offset Dimension** dialog box appears.



4. In the **Enter Angle** box, type the angle value.
 5. Click **OK**.
 6. In the drawing, click to select the center point of the first fitting (origin point).
 7. Click to select the center point of the second fitting (origin point).
 8. Click to select the first boundary of the dimension box.
 9. Click to select the second boundary of the dimension box.
- The cursor changes to a different isoplane setting.
10. Click to select the dimension line location.

The offset dimension appears in the drawing.

Fitting and Pipe Offset Dimensioning

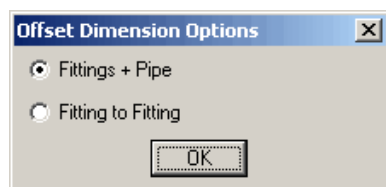
Select the correct isoplane before inserting dimensioning.

To insert a fitting and pipe offset dimension...

1. From the *Utilities* tab, select **Offset**.

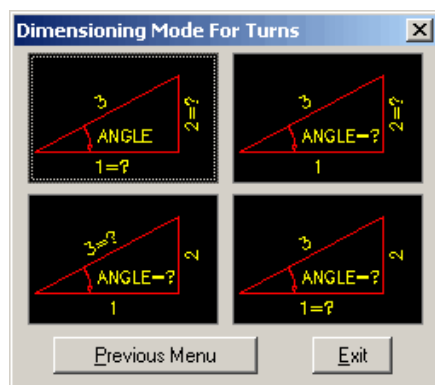


The **Offset Dimension Options** dialog box appears.



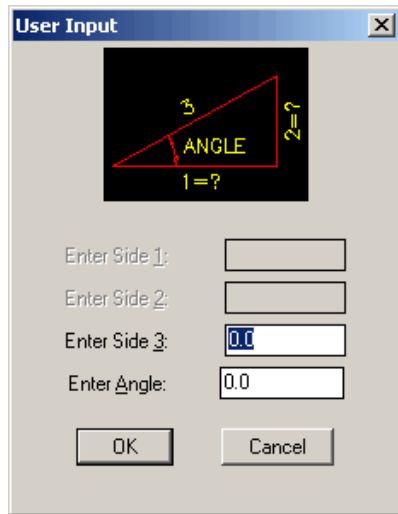
2. From the options, select **Fittings + Pipe**.
3. Click **OK**.

The **Dimensioning Mode for Turns** dialog box appears.



4. In this dialog box, select a dimensioning mode

The **User Input** dialog box appears.



Boxes for unknown values appear solid.

5. In the input boxes, type the known values.
6. Click **OK**.
 - **Note:** If one of the sides contains a pipe segment, PROCAD SPOOLCAD inserts the cut length in its INFO block.
7. Click to select the center point of the first fitting (origin point).
8. Click to select the center point of the second fitting (origin point).
9. Click to select the first boundary of the dimension box.
10. Click to select the second boundary of the dimension box.
11. Click to select the dimension line location.

The offset dimension appears in the drawing.

If you typed a pipe length value less than the minimum required length in the **User Input** dialog box, the **SPOOLCAD Message** box appears, notifying you the pipe length is less than the minimum allowed. Click **OK**.

The **Dimensioning Mode for Turns** dialog box reappears.

12. Repeat Steps 4-6. The dimension appears in the drawing.
 - **Note:** PROCAD SPOOLCAD inserts the minimum pipe length value in pipe segments between elbows.

Rolling Offset Dimensioning

Use this procedure to dimension piping that changes direction in two planes.

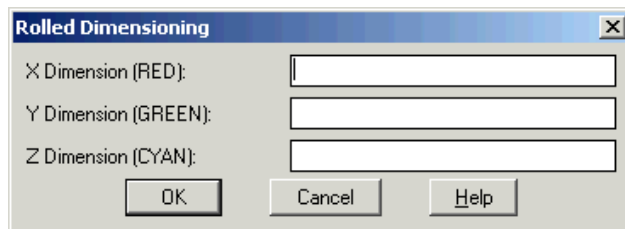
To insert a rolling offset dimension...

1. From the *Utilities* tab, select **Rolling Offset**.

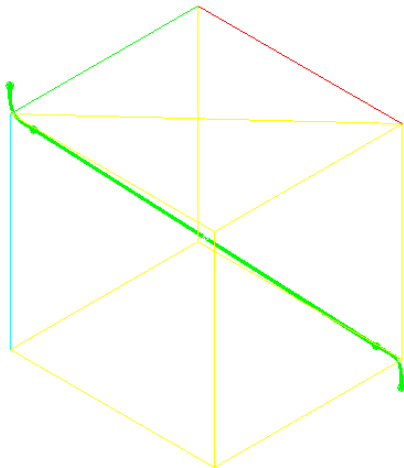


2. Click to select the center point of the first elbow (origin point).
3. Click to select the center point of the second elbow (origin point).

The **Rolled Dimensioning** dialog box appears.

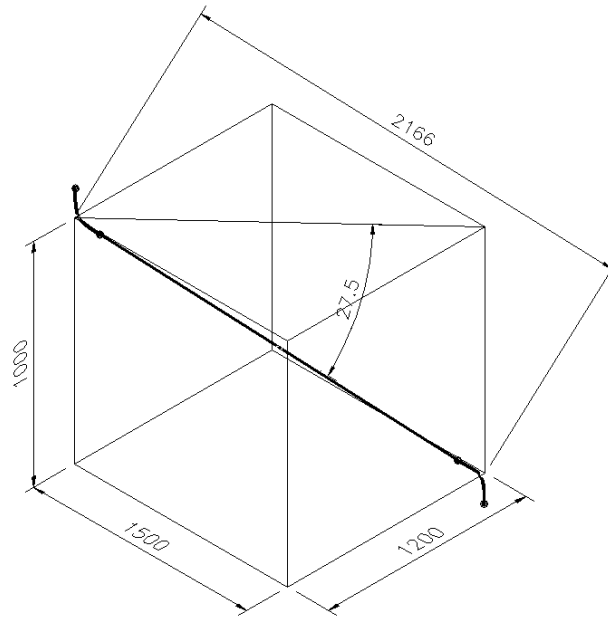


A construction box showing the three different planes appears in the drawing.



4. In the input boxes, type values for the X, Y and Z dimensions. The color-coding on the screen identifies the different sides.
5. Click to select the dimension line distance from the construction box.

The pipe length and construction box dimensions and the rotation angle appear in the drawing.



Bill of Material

Overview

When you complete the isometric, you can generate a Bill of Material for the drawing. The Bill of Material can be grouped by the component types listed below:

- pipe
- flanges
- weld fittings
- screwed and socket weld fittings
- gaskets
- bolts
- valves
- supports and miscellaneous

Caution: Do not deactivate the Material ON/OFF toggle if you plan to generate a Bill of Material. See *Material ON/OFF Toggle*.

Modify the size and appearance of the Bill of Material and lists using the BOM Settings Manager.

Bill of Material Example

BILL OF MATERIAL							
PIPE							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	S/F
1	4m	219.1mm	STD.		PIPE SMLS	A106 GR. B	S
FLANGES							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	S/F
2	2	219.1mm	300#	STD.	FLANGE RF WN	A106 GR. II	S
WELD FITTINGS							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	S/F
3	2	219.1mm	STD.		ELBOW 45 DEG BW - TRIM TO 40	A234 GR. WPB	S
4	1	219.1mm	STD.		ELBOW 45 DEG BW	A234 GR. WPB	S
5	2	219.1mm	STD.		ELBOW 90 DEG LR BW - TRIM TO 40	A234 GR. WPB	S
6	2	219.1mm	STD.		ELBOW 90 DEG LR BW	A234 GR. WPB	S
7	1	219.1mm X 88.9mm	STD.		REDUCING CROSS BW	A234 GR. WPB	S
GASKETS							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	S/F
8	2	219.1mm	300#		GASKET RF 1/8" SPIRAL WOUND	304 SS	F
BOLTS							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	S/F
9	2 SETS	7/8"x5 1/2"			STUD BOLTS GW/2 NUTS-12/SET	A193 B7 /A194 2H	F
VALVES							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	VALVE TAG S/F
10	1	219.1mm	300#		GATE VALVE RF	CS	GA301 F
SUPPORTS & MISC							
WK	QTY	SIZE	RATING	SCHEDULE	DESCRIPTION	MATERIAL	S/F
11	1	219.1mm			BASE SUPPORT	A106 GR. A	F

To ensure your Bill of Material contains accurate pipe length information, dimension your drawing or use the **Calculate Pipe Length** tool.

Generating a New Bill of Material

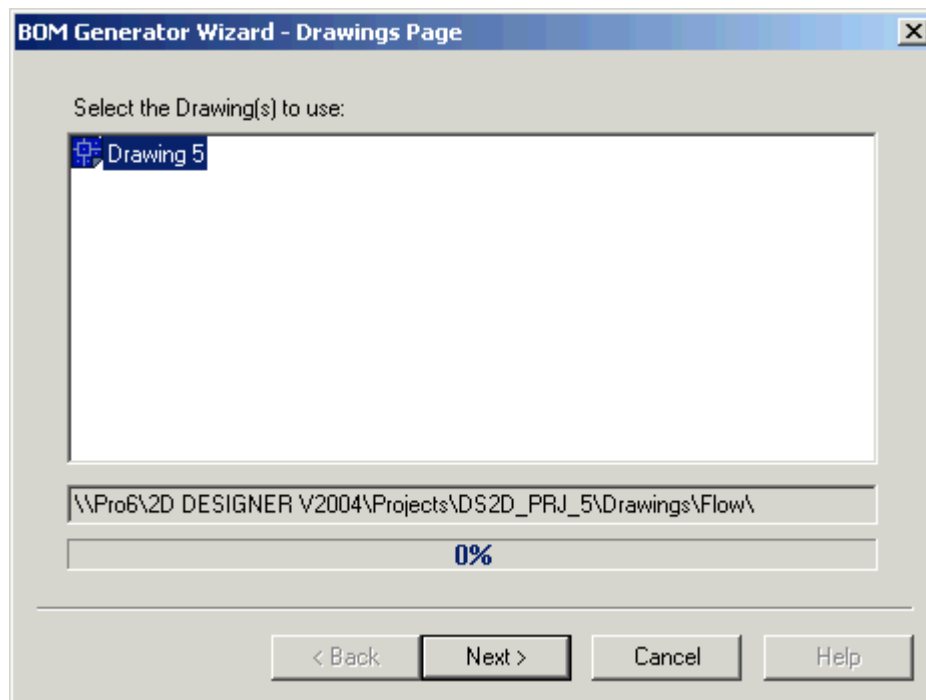
This situation occurs when a BOM is being created for the very first time, or when a BOM is being generated after the complete removal of a previous BOM and the related balloons.

To generate a Bill of Material...

1. From the **BOM and Lists** tab, select **Generate BOM**.

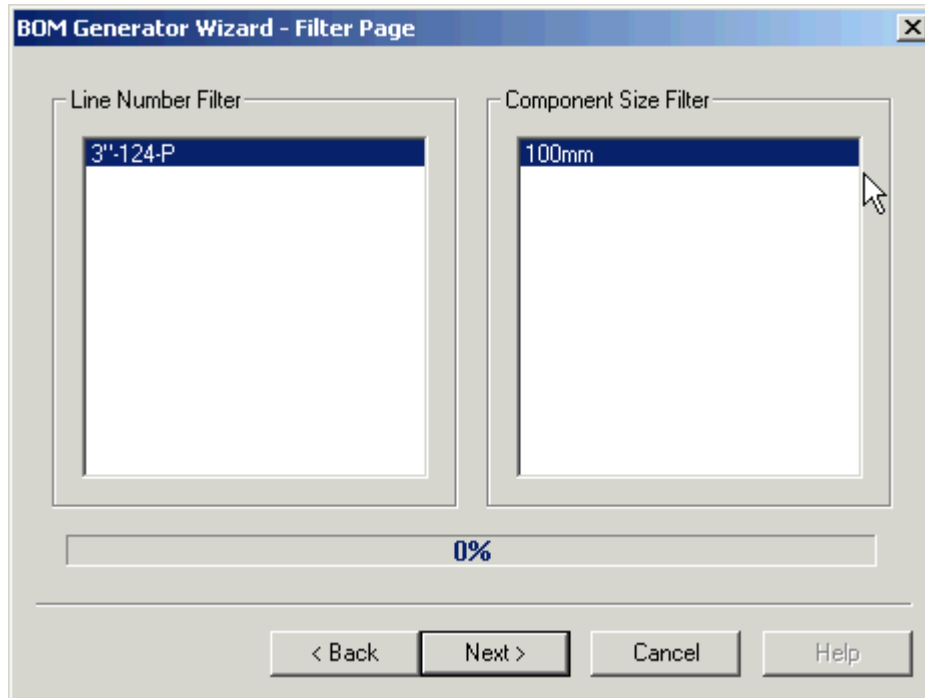


The **BOM Generator Wizard** dialog box appears.



2. Select the drawing you want to use
3. Click **Next**.

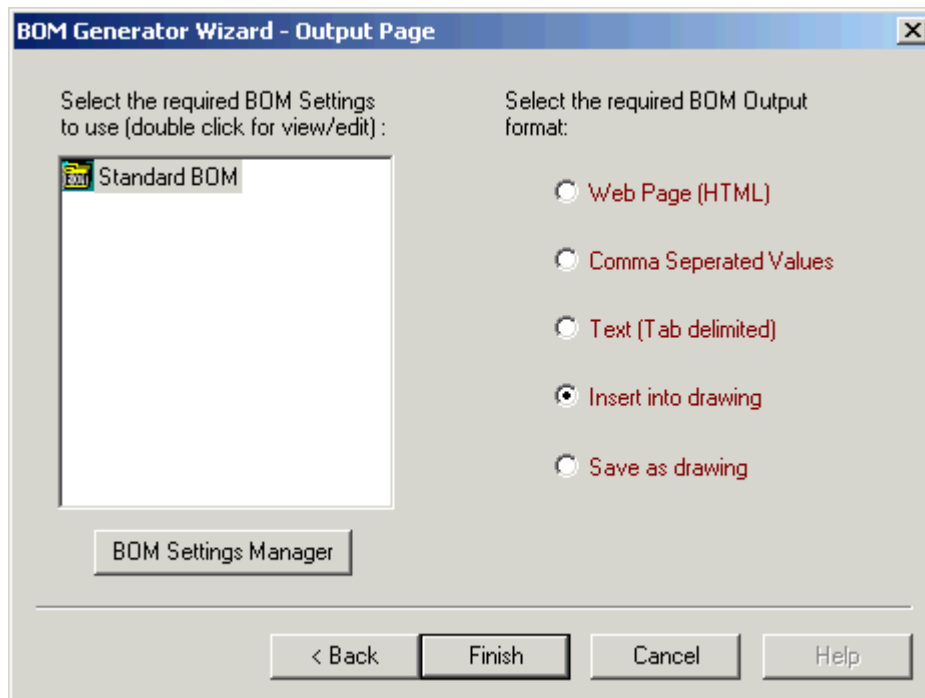
The **BOM Generator Wizard – Filter Page** dialog box appears.



4. Select the line numbers and component sizes to use.
5. Click **Next**.

The **BOM Generator Wizard – Output Page** dialog box appears.

- **Note:** The **BOM Settings Manager** button will only be enabled if you have the BOM Settings Manager installed.



6. Select the settings and output format.
 - Click on Web Page (HTML) to display the BOM as a web page.
 - Click on Comma Separated Values to display as a file with commas separating the columns/fields. This is the most useful for importing information into a database or spreadsheet.
 - Click on Text (Tab delimited) to display the BOM as a text file with tabs separating the columns/fields.
 - Click on Insert into Drawing to display the BOM in an already existing drawing.

Click Save as Drawing to display the BOM in a new drawing.

7. Click **Finish** to continue.
8. If you selected the option Insert in Drawing, click where you want to insert the Bill of Material.
9. The **Generate Bill of Material-Balloons** dialog box appears.
10. In this dialog box, do one of the following:
 - Click **Yes** to insert material balloons.
 - Click **No** to insert the BOM without material balloons.

The Bill of Material appears in the drawing.

Inserting Material Balloons Automatically

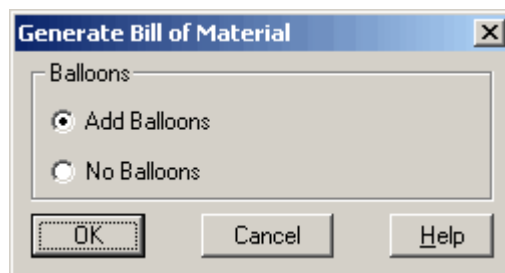
Use this procedure to insert material balloons immediately after inserting the Bill of Material.

To insert material balloons automatically...

See also *Inserting Material Balloons Manually*.

1. Generate a **Bill of Material**.

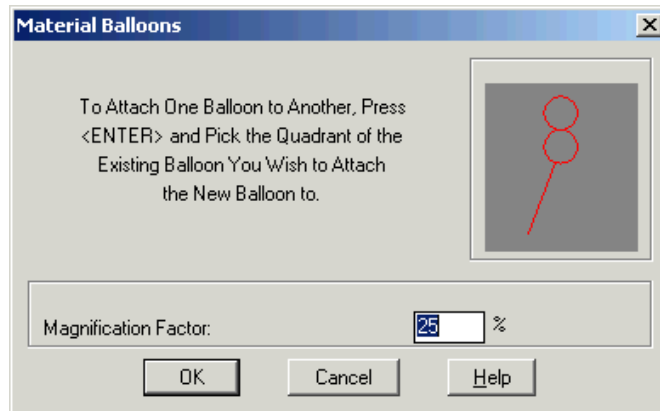
The **Generate Bill of Material Balloons** dialog box appears.



2. From the **Balloons** options, select **Add Balloons**.

3. Click **OK**.

The **Material Balloons** dialog box appears.



In the **Magnification Factor** box, type a magnification value. This value determines how close you zoom into the drawing when placing balloons.

4. Click **OK**.

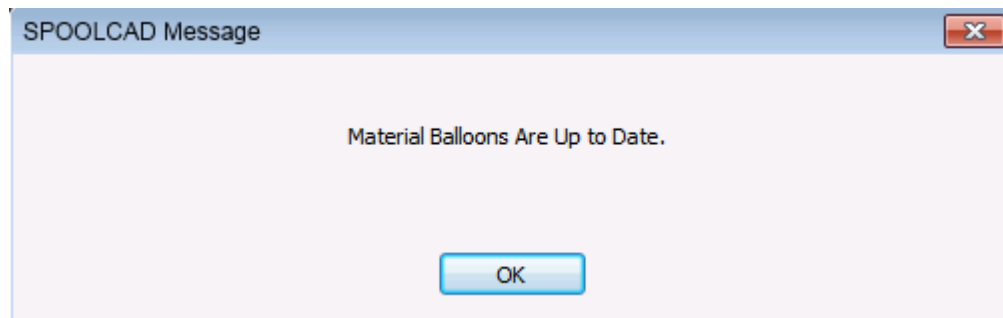
A leader line appears for the first component.

5. In the drawing, click where you want to insert the material balloon.

A balloon containing that component's Bill of Material number appears in the drawing.

6. Repeat Step 6 until all the material balloons appear in the drawing.

When you have inserted all the material balloons, the **SPOOLCAD Message** dialog box appears.



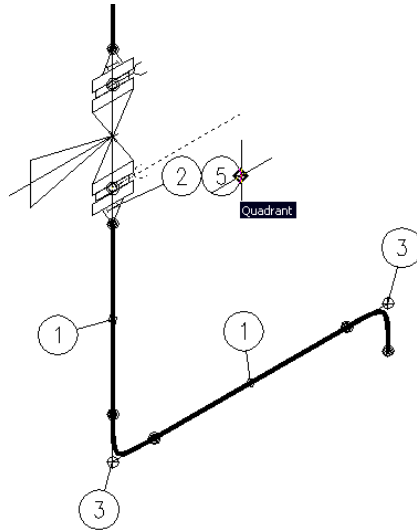
7. Click **OK**.

Stacking Material Balloons

This is a sub-procedure that lets you stack material balloons during the *Inserting Material Balloons* procedure. Stack material balloons for complex components such as valves.

To stack material balloons...

1. In the drawing, click where you want to insert the first material balloon of the stack.
The balloon appears in the drawing.
2. Press **Enter**.
3. Click the quadrant on the first balloon where you want to stack the next balloon.
The second balloon appears, stacked on the first.
4. Repeat Steps 2 and 3 until the stack is complete.



Inserting Material Balloons Manually

Use this procedure if you did not insert material balloons immediately after inserting the Bill of Material.

See also *Inserting Material Balloons Automatically*.

To insert material balloons manually...

1. From the **Bill of Material** tab, select **Component Numbers**.



The **Material Balloons** dialog box appears.

2. In the **Magnification Factor** box, type a magnification value. This determines how close you zoom in on the drawing when placing the material balloons...
3. Click **OK**.
4. In the drawing, click where you want to insert the material balloon.

A balloon containing that component's Bill of Material number appears in the drawing.

5. Repeat Step 4 until all the material balloons appear in the drawing.

Updating the Bill of Material

There are four different options available for updating the Bill of Material:

1. Completely remove the Bill of Material and the related material balloons and generating a new complete overall Bill of Material and material balloons. This is the procedure listed in *Generating a New Bill of Material*.
2. Generate a new Bill of Material and material balloons for all components without removing the previous objects. This is the procedure listed in *Generating a New Bill of Material*.
3. Appending the components added to the drawing since the previous Bill of Material generation to the bottom of the existing Bill of Material. This will create a second Bill of Material with only the newly added components and will continue from the last used material balloon tag number. This is the procedure listed in *Appending New Components to the Existing Bill of Material*.
4. Inserting the components added to the drawing since the previous Bill of Material generation into any empty and available numbers not used in the previous Bill of Material. This will create a new Bill of Material including all of the components selected during the generation process. This is the procedure listed in *Inserting New Components into a Bill of Material and Maintaining Existing Ballooning*.

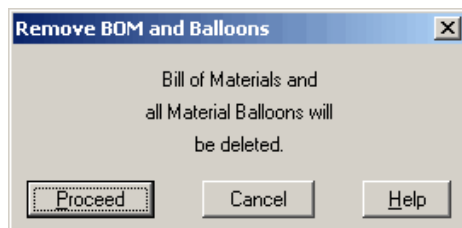
Removing the Bill of Material and Material Balloons

To remove the existing Bill of Material...

1. From the **Bill of Material** tab, select **Remove BOM**.



The **Remove BOM and Balloons** dialog box appears.



2. Click **Proceed**.

The Bill of Material and material balloons disappear from the drawing.

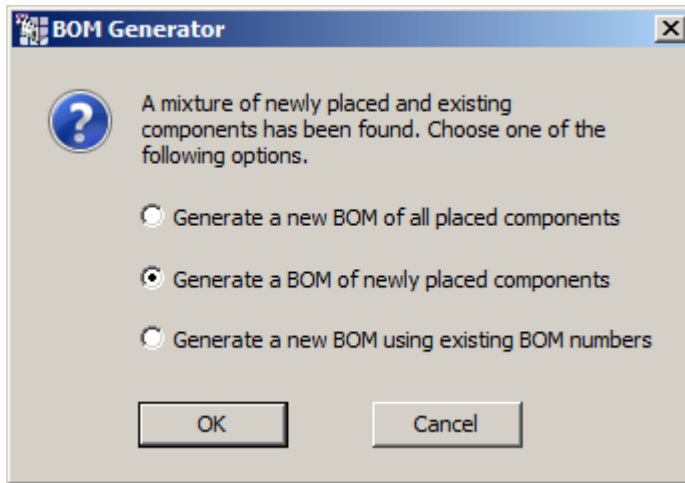
3. Generate a new **Bill of Material** as required.

Appending New Components to the Existing Bill of Material

It may be desirable to append recently added components to the bottom of the existing Bill of Material when the changes are extensive and there are not enough available numbers for the appended components.

To append new components to the existing Bill of Material...

1. Begin the procedure for generating a Bill of Material. The program will automatically detect that new items have been added after the existing Bill of Material was generated and the following dialog box will appear.



2. Select the **Generate a BOM of newly placed components** option.
3. Continue through the remainder of the BOM Generation wizard options.

When you are prompted to place the BOM, keep in mind that it will only include the recently placed components. The balloon numbering will continue on from the last number used in the original Bill of Material, and will use the numbering system defined in the Bill of Material Settings template.

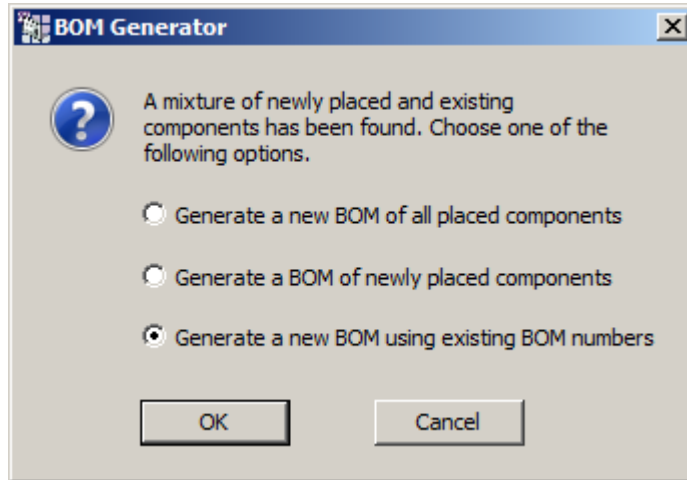
All of the appended components will get new balloon numbers. For example, if the drawing had a 3" Schedule 80 regular tee that was originally tagged as item number 23, any newly placed 3" Schedule 80 regular tees will have a different number. Existing material balloons will remain.

Inserting New Components into a Bill of Material and Maintaining Existing Ballooning

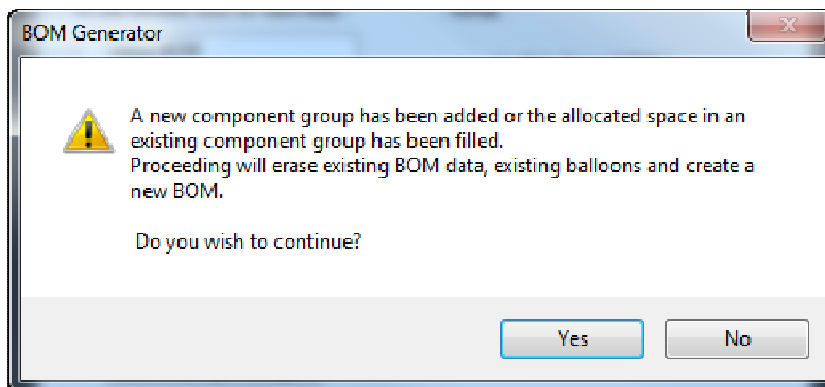
The Bill of Material Settings Manager allows a numbering scheme to be defined which may include a minimum number of unused item numbers within a component group. It may be desirable to insert recently added components into these empty slots to avoid changing the item numbers and balloons of existing components.

To insert new components into a Bill of Material...

1. Begin the procedure for generating a Bill of Material. The program will automatically detect that new items have been added after the existing Bill of Material was generated and the following dialog box will appear.



2. Select the **Generate a BOM using existing BOM numbers** option.
3. Continue through the remainder of the BOM Generation wizard options.
4. A check will be done to ensure that there are enough blank item numbers for all of the newly placed components. If there are not, or if a component from a new group has been added, the following message will appear.



5. Click **No** to exist out of the command. Click **Yes** to erase the Bill of Material and all material balloons and to regenerate a Bill of Material as if for the first time.

When you are prompted to place the BOM, it will generate a BOM for all components, which may be placed beside the original for comparison purposes. Only newly placed components will require ballooning.

Components that have already received an item number will keep the same item number for any newly placed identical items.. For example, if the drawing had a 3" Schedule 80 regular tee that was originally tagged as item number 23, any newly placed 3" Schedule 80 regular tees will also be tagged as item number 23. The quantity of this item in the Bill of Material will be updated.

It is not recommended to use the insertion process for very large modification or for multiple iterations, as problems may eventually arise with the numbering.

Inserting a Special Item

Use this procedure to add newly drawn special components to the **Bill of Material**.

To add a special component to the Bill of Material...

1. Insert or draw the special component in the isometric drawing.
2. From the **Bill of Material** tab, select **Special Item**.



The **Special Item** dialog box appears.

The **Special Item** dialog box contains the following fields and controls:

- Main Pipe Size:** A scroll list with values: 10.3mm, 13.7mm, 17.1mm, 21.3mm, 26.7mm, 33.4mm, 42.2mm, 48.3mm, 60.3mm, 73.0mm, 88.9mm, 101.6mm, 114.3mm, 141.3mm, 168.3mm.
- Branch Pipe Size:**
 - ☒ Not Required
 - ☐ Required
 - When **Required** is selected, a scroll list with values: 10.3mm, 13.7mm, 17.1mm, 21.3mm, 26.7mm, 33.4mm, 42.2mm, 48.3mm, 60.3mm, 73.0mm.
- Rating:** A scroll list with values: 150#, 300#, 400#, 600#, 900#, 1500#, 2000#, 2500#, 3000#.
- Destination:**
 - ☒ Shop
 - ☐ Field
- Spec:** A list box with values: A, B, C, D, E, F.
- Description:** A text input field.
- Material:** A text input field.
- Schedule:** A text input field.
- Manufacturer:** A text input field.
- Dimension:** A text input field.
- Catalog Number:** A text input field.
- Tag:** A text input field.
- Buttons:** OK, Cancel, Help.

3. In this dialog box, do the following:

In this...	Do this...
Main Pipe Size scroll list	Select the required main pipe size value
Branch Pipe Size options and scroll list	Select Not Required or Required If you select Required , select the branch pipe size value from the scroll list
Rating scroll list	Select the required rating value
Spec list	Select the required specification type
Description input box	Type a description for the special item

Material input box	Type the item's material information
Schedule input box	Type the item's schedule information
Manufacturer input box	Type the manufacturer name
Dimension input box	Type the item's dimension data
Catalog Number input box	Type the item's catalog number
Tag input box	Type the item's tag value/description

4. Click **OK**.
5. In the drawing, click in the newly drawn or inserted component.
An INFO block appears, containing the information defined in Step 3.
6. Update the Bill of Material to show the new component
A REVISION block listing appears in the Bill of Material displaying the new component information.

Single Material Balloon

Use this procedure to replace any erased material balloons. For this tool to function, the Bill of Material must already exist.

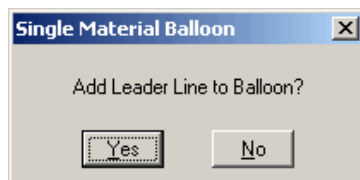
To insert a single material balloon...

1. From the **Bill of Material** tab, select **Single Material Balloon**.



2. In the drawing, click the INFO block of the component for which you want to insert a material balloon. Zoom in to drawing if necessary.

The **Single Material Balloon** dialog box appears.



3. Choose one of the following options:
 - Click **Yes** to insert the balloon with a leader line
 - Click **No** to insert the balloon without a leader line
4. In the drawing, click where you want to insert the material balloon.
The material balloon appears in the drawing.

Bill of Material Settings and Appearance

See the *BOM Settings User Guide*.

Generating Lists

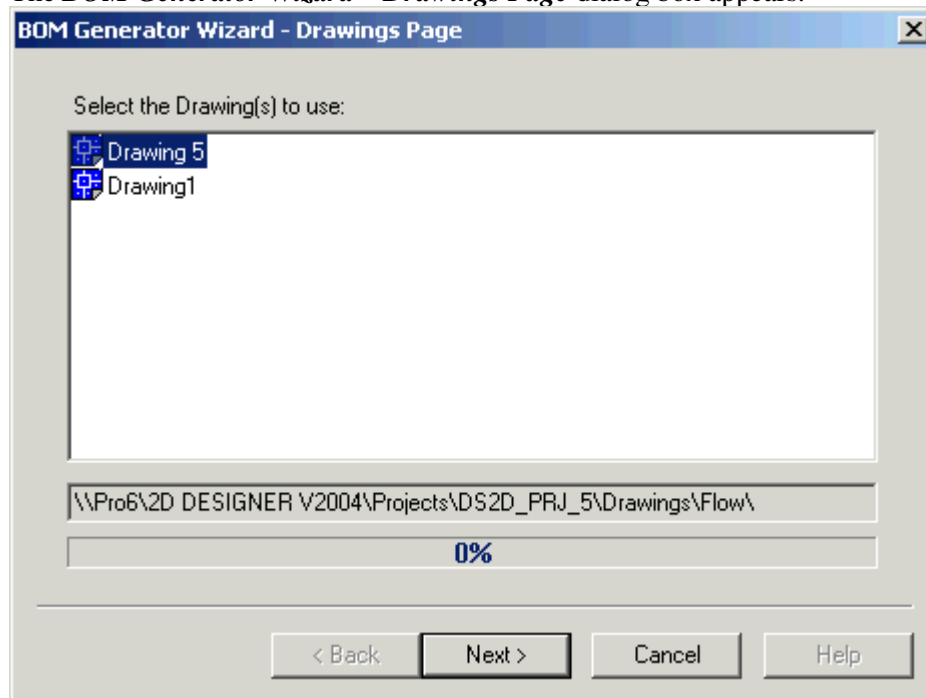
Generating a List

Use this procedure to generate a list in the drawing.

To generate a list...

1. From the **BOM and Lists** tab, select **Generate List**.

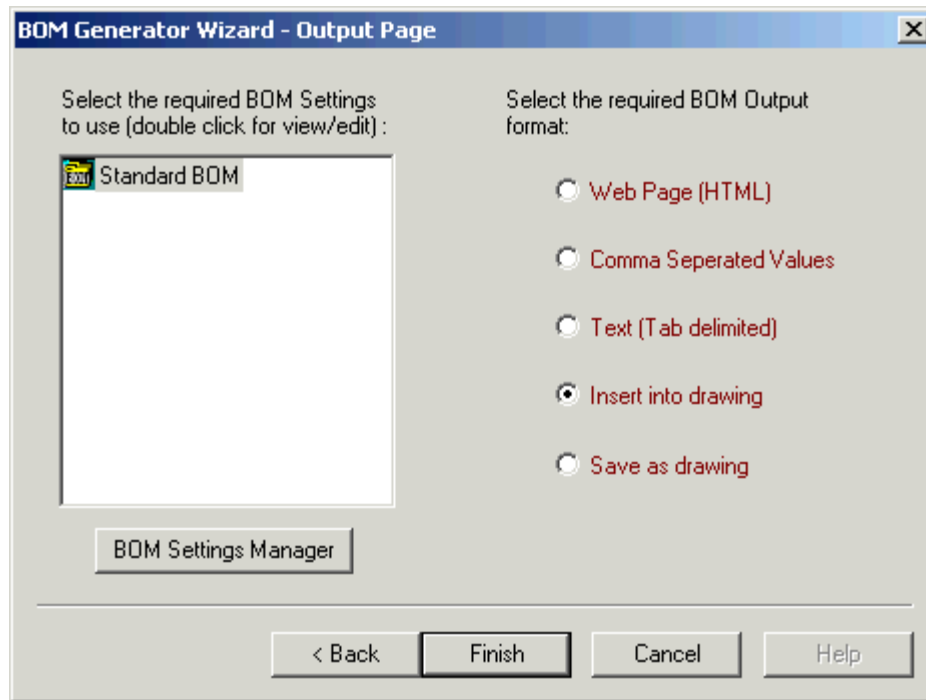
The **BOM Generator Wizard – Drawings Page** dialog box appears.



2. Select the drawing you want to use.
3. Click **Next**.

The **BOM Generator Wizard – Output Page** dialog box appears.

- **Note:** The **BOM Settings Manager** button will only be enabled if you have the BOM Settings Manager installed.



4. Select the type of list you want to use, as well as the output format.
 - Click on Web Page (HTML) to display the BOM as a web page.
 - Click on Comma Separated Values to display as a file with commas separating the columns/fields. This is the most useful for importing information into a database or spreadsheet.
 - Click on Text (Tab delimited) to display the BOM as a text file with tabs separating the columns/fields.
 - Click on Insert into Drawing to display the BOM in an already existing drawing.
 - Click Save as Drawing to display the BOM in a new drawing.
5. Click **Finish** to continue.
6. If you selected the option **Insert into Drawing**, click where you want to insert the list.

The list appears in the drawing.

AutoDATA and Database Interface

In the Generating Lists procedure, the user has the ability to export data into a database of their choice.

There are two available methods to export the data. The first uses the AutoDATA format. The second allows for more custom settings to be selected.

To export the AutoDATA information...

1. From the **Bill of Material** tab, select **AutoDATA**.



2. Select the drawings you want to use
 - **Note:** It is recommended that all drawings be selected.
3. Click **Open**.

The **Save AutoDATA info as** dialog box appears.
4. Select the location for the datafile and click **Save**.
5. Open the database program you want to use and import datafile. I.e. Excel

To export data in a customized setting...

1. In the **BOM Generator Wizard – Drawings Page** dialog box of step one in the Generating Lists procedure, select of the drawings you want to use.
 - **Note:** It is recommended that all drawings be selected.
2. Click **Next**.

The **BOM Generator Wizard – Output Page** dialog box appears.
3. In this dialog box, select one of the available BOM Settings listed to use during the data export. If you wish to create or modify one of the BOM Settings, click **BOM Settings Manager**.

See the *BOM Settings User Guide*.
4. Select **Comma Separated Values** as the required BOM Output Format.
5. Click **Finish**.

The **Save As** dialog box appears.

6. Select the location for the datafile and click **Save**.
7. Open the database program you want to use and import datafile.

Fab Tools

Update XRay and Welder ID

Use this procedure to add and Welder ID to your welds.

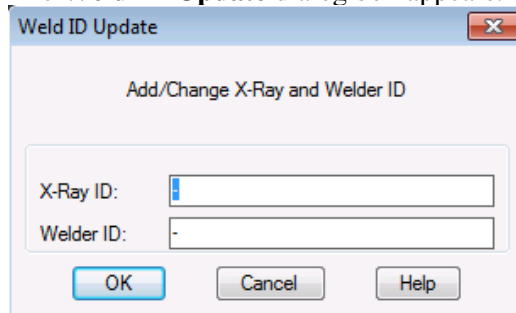
To update the XRay and Welder ID...

1. Insert or draw components that requires welding.
2. From the **Fab Tools** tab, select **Update XRay and Welder ID**.



3. Select a weld symbol. A weld symbol is the round symbol at the end of a connected pipe.

The **Weld ID Update** dialog box appears.



4. Enter the X-Ray ID and Welder ID and click **OK**.
5. These number are used when you generate **Weld List**.

See *Weld List Section*.

Update Weld Info

Use this procedure to edit attributes related to weld info. This includes the Welder ID or Name, XRay Number, Weld ID, etc.

To update Weld Info...

1. Insert or draw components that requires welding.

- From the **Fab Tools** tab, select **Update Weld Info**.



- Select a weld symbol. A weld symbol is the round symbol at the end of a connected pipe.

The **Edit Attributes** dialog box appears. (There are 2 pages to the dialog box but only one is shown)

The **Edit Attributes** dialog box is shown with the following fields and values:

Field	Value
Block name:	WLD_BLK
Weld ID	1
Pipe Size	4.00
Weld Schedule	STD.
Welder Name or ID	321312
X-Ray	121222
Line number	Line 1
Weld Inches	
Misc 1	

Buttons at the bottom: OK, Cancel, Previous, Next, Help.

- Edit the information according to your requirements.
- Click **Next** to go to the next page. There are two pages in total.
- Edit the information on all pages as required.
- Click **OK** to save the changes and return to the drawing.

Field Weld Symbol

Use this procedure to add field weld symbols to your drawing. This procedure removes the weld from the weld list and shows a symbol representing a weld that is done on the field.

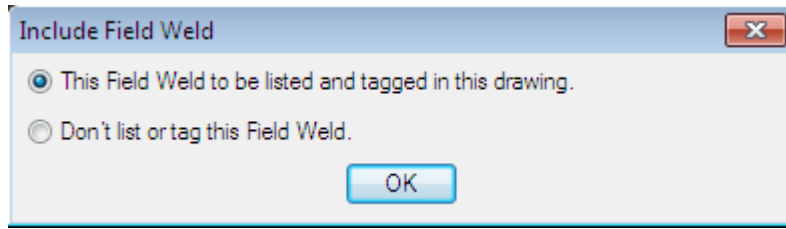
To insert a Field Weld Symbol...

- Insert or draw components that requires welding.
- From the **Fab Tools** tab, select **Field Weld Symbol**.



- Select a weld and an orientation to insert the field weld symbol.

The **Include Field Weld** dialog box will appear.



4. Choose one of the following options:
 - This Field Weld to be listed and tagged in this drawing
 - Don't list or tag this Field Weld
5. Click the **OK** Button to continue.

Remove Field Weld Symbol

Use this procedure to remove a field weld symbols in your drawing. This procedure removes the field weld and adds the weld back into the weld list.

To Remove a Field Weld Symbol...

1. Insert or draw components that requires welding.
2. From the **Fab Tools** tab, select **Field Weld Symbol**.



3. Select a weld to remove the field weld symbol.

Do Not Include Weld in List

Use this procedure to remove a weld from the weld list. When you have a weld that you do not wish to place in a weld list, you can add a mark and specify that this weld is not part of the weld list.

To insert a Field Weld Symbol...

1. Insert or draw components that requires welding.
2. From the **Fab Tools** tab, select the **Do Not Include Weld in List**.



3. Select a weld to remove the weld from the weld list.

Weld Count

The weld count is a material chart that shows type of weld, the number of that type of weld, size and schedule of the weld.

To insert a Weld Count Chart...

1. From the **Fab Tools** tab, select **Weld Count**.



2. Follow the **Weld Count wizard** that appears. With the weld count wizard, you are able select the drawing to produce a weld count list for, any filters as well as the output for the weld count list.

We will choose insert into drawing for this example.

3. Select a location to place the chart and PROCAD SPOOLCAD will automatically traverse the isometric drawing and produce a weld count chart for the welds in your drawing.

Here is an example of a **Weld Count** list.

WELD COUNT			
Num	SIZE	SCHEDULE	Weld Type
1	4"		Pipe Shoe
6	4"	STD.	Butt Weld

Weld List

The weld list is a collection of weld ID's and the corresponding Welder and X-Ray ID. It will show for each weld, which welder and which X-Ray number is used.

To insert a Weld List...

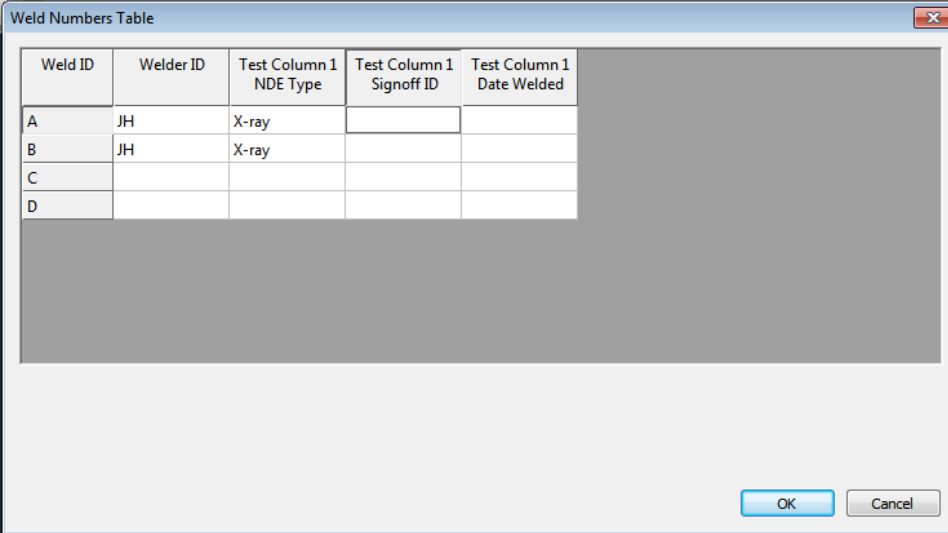
1. From the **Fab Tools** tab, select **Weld List**.



2. Follow the **Weld List wizard** that appears. With the weld list wizard, you are able select the drawing to produce a weld list for, any filters as well as the output for the weld count list.

We will choose insert into drawing for this example.

The **Weld Numbers Table** dialog box appears.



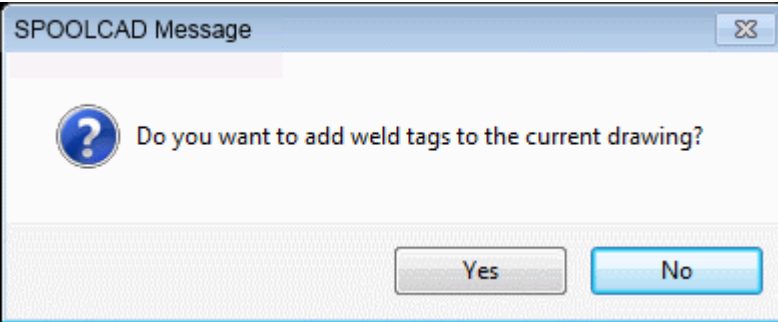
The 'Weld Numbers Table' dialog box contains a table with the following data:

Weld ID	Welder ID	Test Column 1 NDE Type	Test Column 1 Signoff ID	Test Column 1 Date Welded
A	JH	X-ray		
B	JH	X-ray		
C				
D				

At the bottom right of the dialog are 'OK' and 'Cancel' buttons.

3. Edit the numbers if needed.
4. Click **OK** to continue.
5. Select a location to place the cahrt and PROCAD SPOOLCAD prompt to add weld tags to the current drawing. You can do this now or add the weld tags later.

You will see the **Weld Tag** option dialog box.



The 'SPOOLCAD Message' dialog box displays a question mark icon and the text: 'Do you want to add weld tags to the current drawing?'. At the bottom are 'Yes' and 'No' buttons.

- a. Click **Yes** to add **weld tags**.

PROCAD SPOOLCAD will assist you in placing weld tags by automatically traversing the isometric and prompting the user for the location of weld tags.

- b. Click **No** to continue.

Here is an example of a **Weld List**.

WELD LIST				
Drawing1				
Weld ID	Welder ID	NDE Type	Signoff ID	Date Welded
A	JH	X-ray		
B	JH	X-ray		
C				
D				

Remove Weld Tags and Weld List

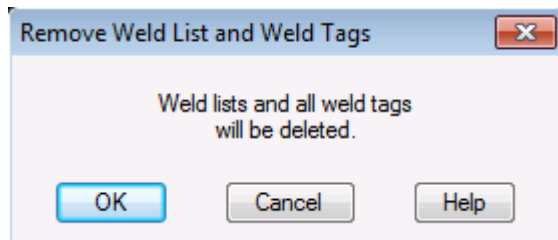
The remove weld tags and weld list removes both the weld list and weld tags in your drawing. It will also remove the weld count chart.

To Remove Weld Tags and Weld List...

1. From the **Fab Tools** tab, select **Remove Weld Tags and Weld List**.



The **Remove Weld List and Weld Tags** dialog box appears.



2. Click **OK** to continue with the removal procedure.
3. You will see that the **Weld List** and **Weld Tags** will be removed.

Single Weld Tag

Use this procedure to add a single weld tag to your drawing.

To insert a Single Weld Tag...

1. From the **Fab Tools** tab, select **Single Weld Tag**.



2. Select a weld in your drawing.
3. Select a location for the **weld tag**.

Place Weld Tags

Use this procedure to add a single weld tag to your drawing.

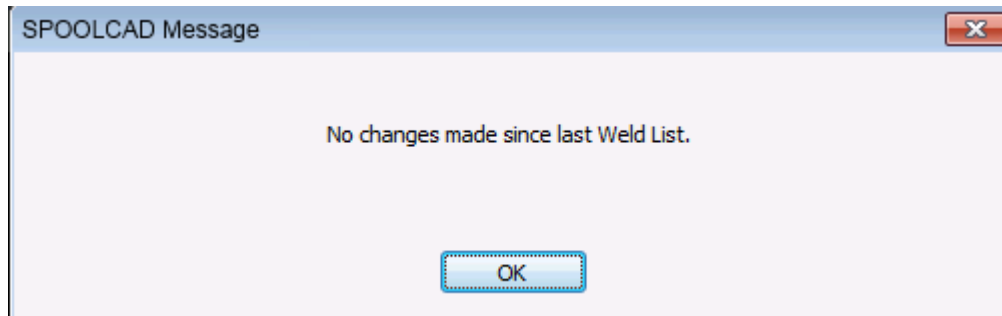
To insert a Single Weld Tag...

1. From the *Fab Tools* tab, select **Single Weld Tag**.



Weld Tags can only be added after you have generated a **weld list**.

2. PROCAD SPOOLCAD will automatically cycle through all the welds to place tags.
3. Select a location for the **weld tag**.
4. After PROCAD SPOOLCAD finishes cycling through the welds, weld tag message will appear.



5. Click **OK** to continue.

Heat Number List

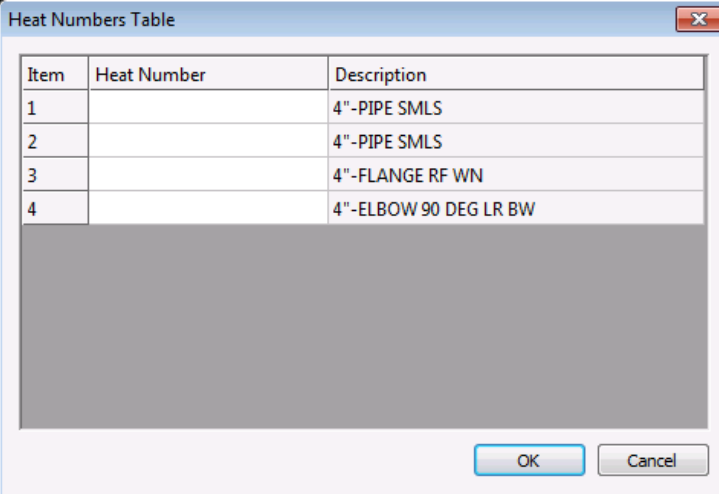
Use this procedure to add a Heat Number List to your drawing. Before a heat number list can be made, a Bill of Material needs to be placed first. The heat number list uses the numbers generated from the BOM as reference.

To insert a Heat Number List...

1. After a BOM is generated, from the *Fab Tools* tab, select **Heat Number List**.



The **Heat Numbers Table** dialog box appears.



The dialog box titled "Heat Numbers Table" contains a table with three columns: Item, Heat Number, and Description. The table has four rows of data. Below the table is a large grey rectangular area. At the bottom right of the dialog are "OK" and "Cancel" buttons.

Item	Heat Number	Description
1		4"-PIPE SMLS
2		4"-PIPE SMLS
3		4"-FLANGE RF WN
4		4"-ELBOW 90 DEG LR BW

2. Enter in the heat numbers in the table.
3. Click **OK** to continue.
4. The heat numbers will be populated in the BOM.

Calculate Labor Cost Codes

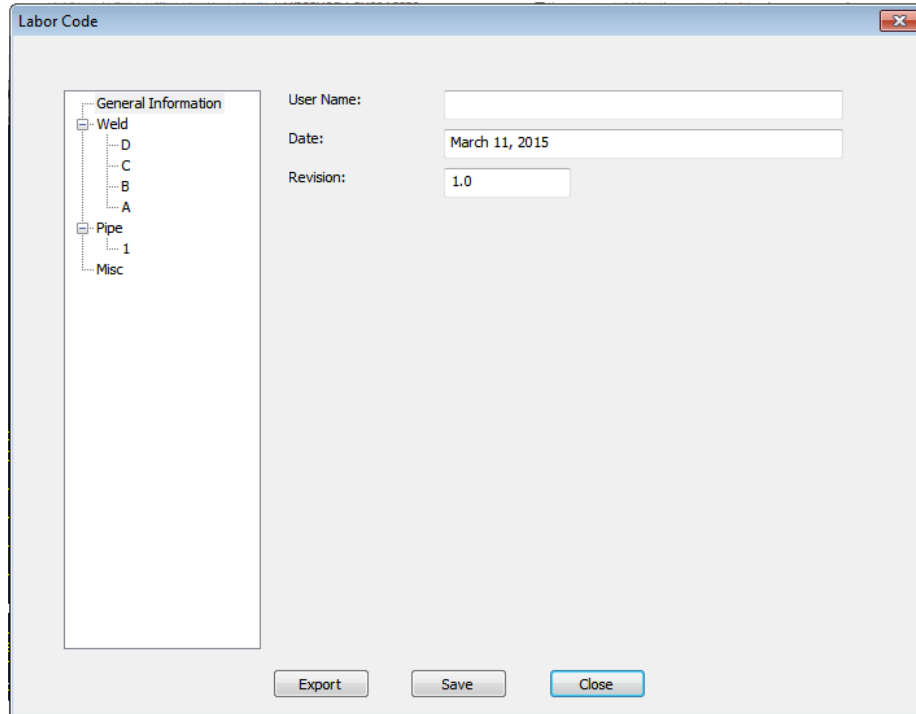
Use this procedure to calculate the labor cost code for the created spool. This function will take information from the Shop Settings Manager to add to the cost code file.

To calculate the labor cost codes...

1. From the **Fab Tools** tab, select **Labor Cost Codes**.



The **Labor Cost Codes** dialog box appears.



The Labor Code dialog box is shown. It has a title bar 'Labor Code' with a close button. On the left is a tree view under 'General Information' with nodes: Weld (expanded), D, C, B, A, Pipe (expanded), 1, and Misc. On the right are input fields for 'User Name:', 'Date:' (containing 'March 11, 2015'), and 'Revision:' (containing '1.0'). At the bottom are 'Export', 'Save', and 'Close' buttons.

2. You can enter the information under the General information.
3. You can check each weld and pipe to see if the labor code is correct.
4. Click **Export** to export the Labor Code. This will save the labor code list to a cvs file.
5. Click the **Save** to save the labor code to a file.

Diameter Inch

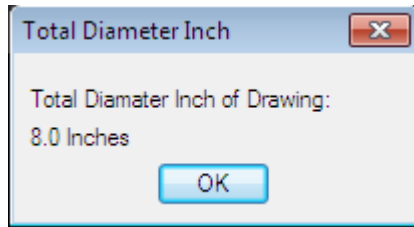
Use the Diameter Inch function to calculate the diameter inch for this spool.

To calculate the Diameter Inch...

1. From the **Fab Tools** tab, select **Diameter Inch**.



The **Total Diameter Inch** dialog box appears.



2. Click **OK** to continue.

Note: The **Diameter Inch** factor will be displayed in the border if there is an attribute for diameter inch. Otherwise, it will be displayed in the **Total Diameter Inch** dialog box.

Calculate Spool Weight

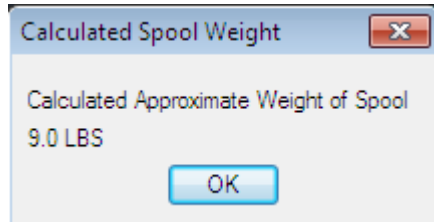
Use the Calculate Spool Weight to calculate the total weight of the spool.

To calculate the Spool Weight...

1. From the **Fab Tools** tab, select **Calculate Spool Weight**.



The **Calculated Spool Weight** dialog box appears.



2. Click **OK** to continue.

Note: The spool weight will be displayed in the border if there is an attribute for spool weight. Otherwise, it will be displayed in the **Calculated Spool Weight** dialog box

Edit Spool Weight

Use the Edit Component Weight to modify a component's weight.

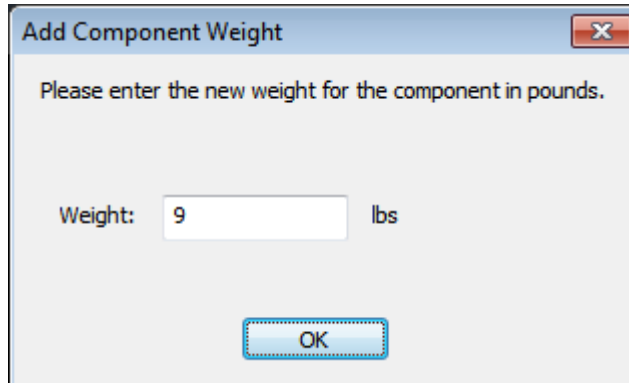
To edit the Spool Weight...

1. From the **Fab Tools** tab, select **Edit Component Weight**.



2. Select the component you would like to edit the weight on.

The **Add Component Weight** dialog box appears.



3. Change the weight to the desired weight for this component. Click **OK** to continue.

Pipe End Prep

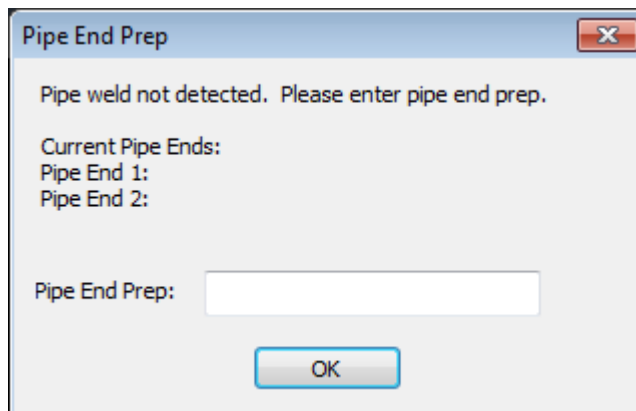
Use the Pipe End Prep to label open pipe ends with a preparation method.

To label all pipe ends...

1. From the **Fab Tools** tab, select **Pipe End Prep**.



If there are open ended pipes, the **Pipe End Prep** dialog box appears.



2. Type in the pipe end prep for the pipe that is centered to the screen.
3. Click **OK** to apply the changes.

Edit Pipe End Prep

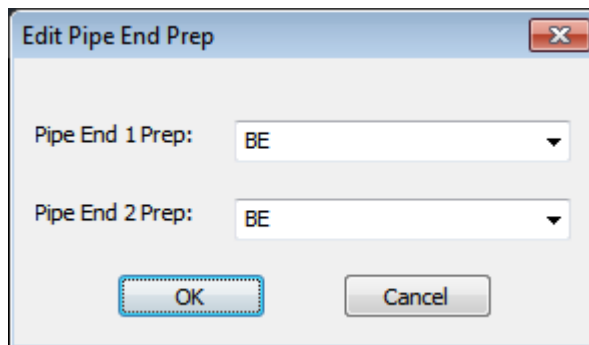
Use the Edit Pipe End Prep to manually change the pipe end preparation of pipe.

To label edit pipe ends...

1. From the **Fab Tools** tab, select **Pipe End Prep**.



2. Click on a existing pipe. The **Edit Pipe End Prep** dialog box will appear.



3. Select the appropriate pipe end prep.
4. Click **OK** to apply the changes.

Calculate Surface Area

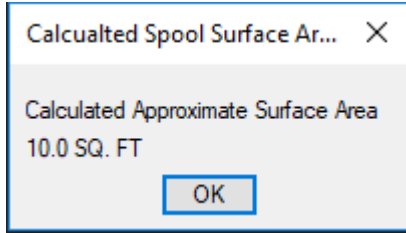
Use the Calculate Surface Area to calculate the estimated total surface area of the spool.

To calculate the Surface Area ...

1. From the **Fab Tools** tab, select **Calculate Surface Area**.



The **Calculated Surface Area** dialog box appears.



2. Click **OK** to continue.

Note: The surface area will be displayed in the border if there is an attribute for the surface area. Otherwise, it will be displayed in the **Calculated Surface Area** dialog box

Edit Component Surface Area

Use the Edit Component Surface Area to modify a component's weight.

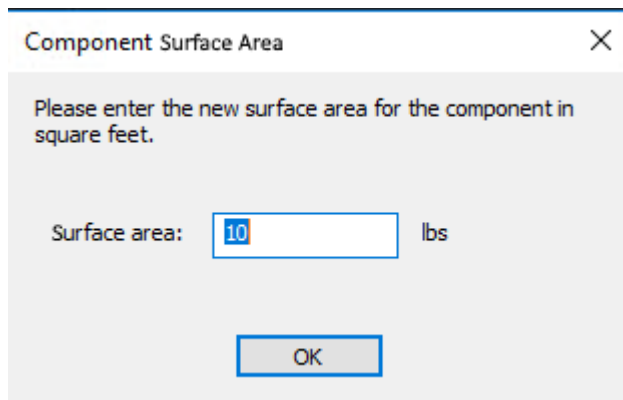
To edit the Surface Area...

1. From the **Fab Tools** tab, select **Edit Component Surface Area**.



2. Select the component you would like to edit the surface area on.

The **Edit Component Surface Area** dialog box appears.



3. Change the surface area to the desired number for this component. Click **OK** to continue.

Miscellaneous Drafting Functions

Centerline Symbol

To insert a centerline symbol...

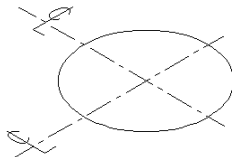
1. From the *Miscellaneous* tab, select **Centerline Symbol**.



The **Center Line Symbol** isoplane dialog box appears.

2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click where you want to insert the centerline symbol.

The symbol appears in the drawing.



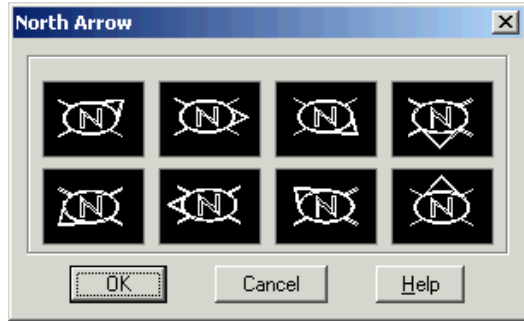
North Arrow

To insert a north arrow...

1. From the *Miscellaneous* tab, select **North Arrow**.



The **North Arrow** dialog box appears.



2. In the dialog box, click to select the appropriate orientation.
3. Click **OK**.
4. In the drawing, click where you want to insert the north arrow.

The north arrow appears in the drawing.

Line Breaks

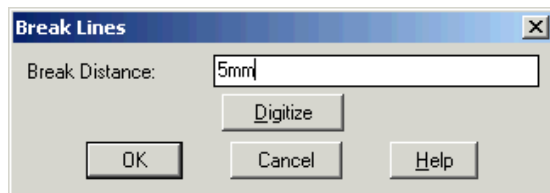
Use this procedure to show line breaks in the drawing where components cross over or under other components.

To insert a crossing line break...

1. From the *Miscellaneous* tab, select **Trim Lines**.

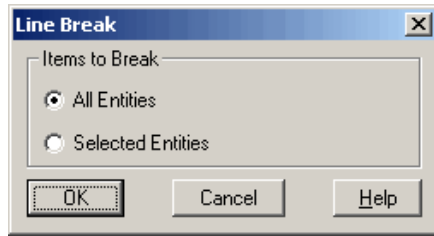


The **Break Lines** dialog box appears.



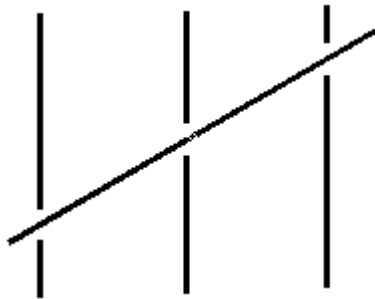
2. In the **Break Distance** box, type how far you want the line to break from the reference line.
3. Click **OK**.
4. In the drawing, click to select the reference line (the line you do not want to break).

The **Line Break** dialog box appears.



5. From the **Items to Break** options, select which items to break:
 - Click **All Entities** to break all lines crossing the reference line
 - Click **Selected Entities** to break only the user selected lines
6. Click **OK**.
7. If you clicked **Selected Entities**, click the lines in the drawing you want to break.
8. Press **Enter**.

The line breaks appear in the drawing.

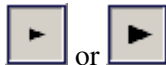


Flow Direction Arrows

Use this procedure to insert a small or large flow direction arrow.

To insert a flow direction arrow...

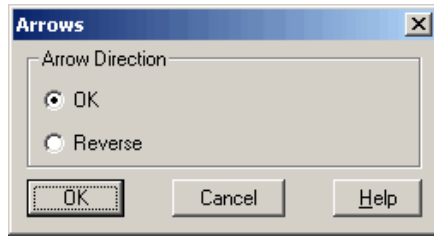
1. From the *Miscellaneous* tab, select **Small Arrow** or **Large Arrow**.



2. In the drawing, click the pipe or line where you want to insert the arrow.

The arrow appears in the drawing, pointing in the direction in which the component was originally drawn.

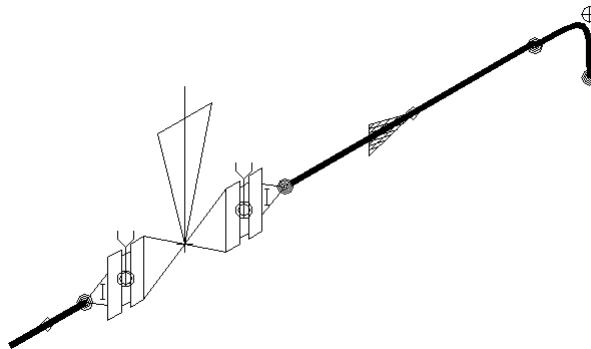
The **Arrows** dialog box appears.



3. From the **Arrow Direction** options set the arrow direction:
 - Click **OK** to accept the default arrow direction
 - Click **Reverse** to change the arrow direction

4. Click the **OK** button.

The arrow appears in the direction selected above.



Revision Triangle

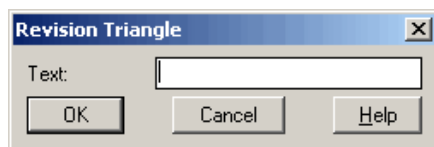
To insert a revision triangle...

1. From the *Miscellaneous* tab, select **Revision Triangle**.



2. In the drawing, click where you want to insert the revision triangle.

The **Revision Triangle** dialog box appears.



3. In the **Text** box, type a revision number.
4. Click **OK**.

The revision triangle appears in the drawing.

Revision Cloud

Use this tool to draw a revision cloud around edited parts of a drawing.

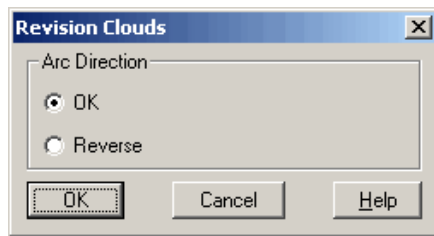
To draw a revision cloud...

1. From the **Miscellaneous** tab, select **Revision Cloud**.



2. In the drawing, click at least four points around the edited items, moving *counter-clockwise*.
3. Press **Enter**.

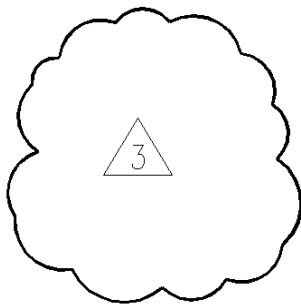
The **Revision Clouds** dialog box appears.



4. From the **Arc Direction** options, select one of the following:
 - Click **OK** to accept the current default direction
 - Click **Reverse** to reverse the arc direction

➤ **Note:** If you select points in a *clockwise* direction, a star-shaped polyline appears. Reverse the arcs to form a cloud by selecting **Reverse** from the **Revision Clouds** dialog box.
5. Click the **OK** button.

The revision cloud appears in the drawing.



Modify Revision Cloud Polyline Thickness

Modify the values in the **Continuation Arrows and Rev Blocks** box on the **Scaling Factors** dialog box.

Instrument and Identification Balloons

- **Note:** Balloons break after a set number of characters.

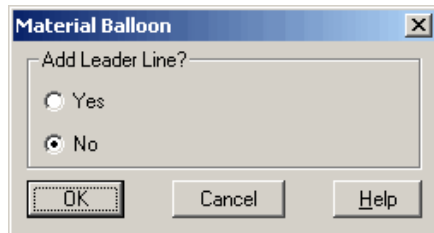
Single Line Round Balloons

To insert single line round balloons...

1. From the *Miscellaneous* tab, select **Single Line Balloon**.

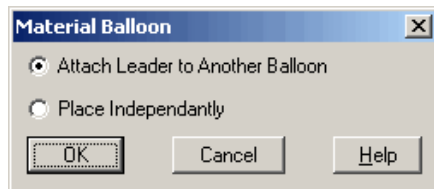


The **Material Balloon** dialog box appears.



2. From the **Add Leader Line** options, select one of the following:
 - Select **Yes** to insert the balloon with a leader line
 - Select **No** to insert the balloon without a leader line
3. Click **OK**.

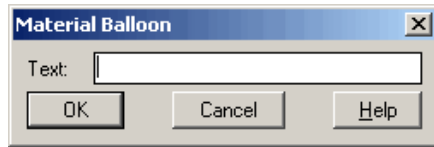
If you select **No**, the **Material Balloon** dialog box appears.



4. Select a balloon placement option:
 - **Attach Leader to Another Balloon** prompts you to select an existing balloon as an insertion point
 - **Place Independantly** lets you insert the balloon anywhere in the drawing

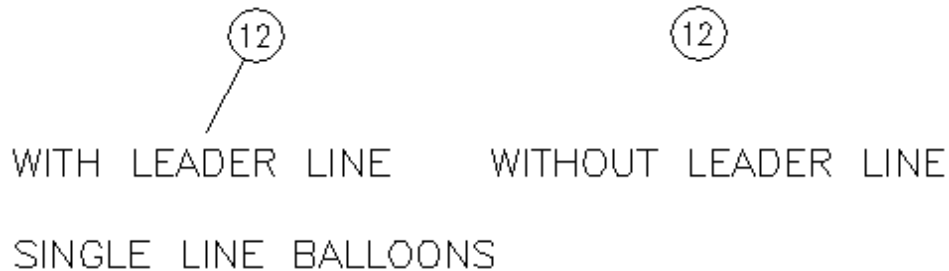
5. In the drawing, click to select the leader line start point (if applicable).
6. Click where you want to insert the balloon.

The **Material Balloon** text dialog box appears.



7. In the text input box, type the required balloon text.
8. Click **OK**.

The balloon and leader line (if applicable) appear in the drawing.



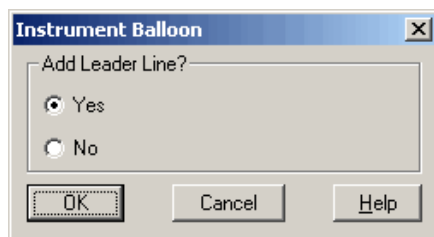
Double Line Round Balloons

To insert double line round balloons...

1. From the *Miscellaneous* tab, select **Balloon with Leader**.



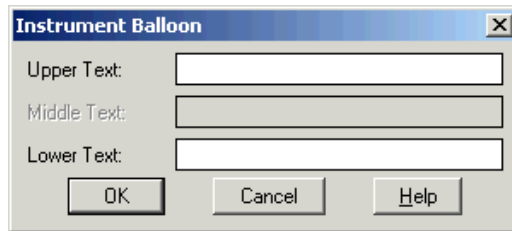
Instrument Balloon dialog box appears.



2. From the **Add Leader Line** options, select one of the following:
 - Select **Yes** to insert the balloon with a leader line
 - Select **No** to insert the balloon without a leader line
3. Click **OK**.
4. In the drawing, click to select the leader line start point (if applicable).

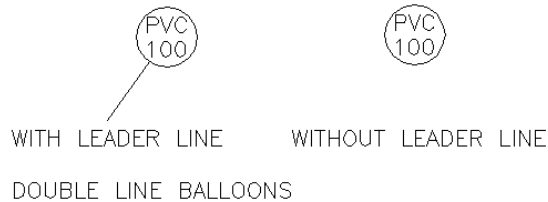
- Click where you want to insert the balloon.

The **Instrument Balloon** text dialog box appears.



- In the text input boxes, type the required text.
- Click **OK**.

The balloon and leader line (if applicable) appear in the drawing.



Hexagonal Balloons

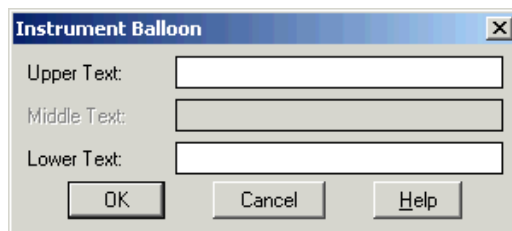
To insert hexagonal balloons...

- From the **Miscellaneous** tab, select **Hexagonal Balloon** or **Hexagonal w/Line**



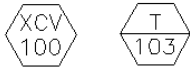
- In the drawing, click where you want to insert the balloon.

The **Instrument Balloon** dialog box appears.



- In the text input boxes, type the required text.
- Click **OK**.

The balloon appears in the drawing.



HEXAGONAL BALLOONS

Editing Balloon Text

See *Editing Instrument Balloon Text* .

Continuation Arrows

Select the correct isoplane setting before inserting continuation arrows. When you set the isoplane to standard (flat), you must also supply the direction.

To insert a continuation arrow...

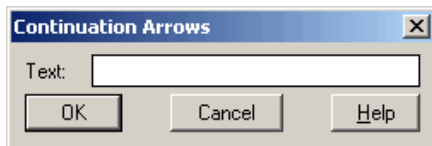
1. From the *Miscellaneous* tab, select **Dual Direction**, **Arrow Left** or **Arrow Right**.



The **Continuation Arrows** isoplane dialog box appears.

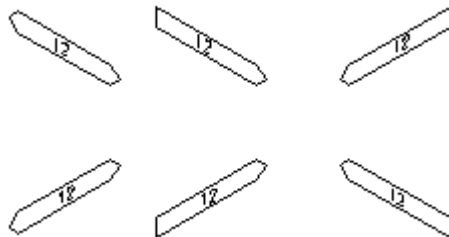
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click where you want to insert the arrow.

The **Continuation Arrows** text dialog box appears.



4. In the **Text** box, type the text you want to appear in the arrow.
5. Click **OK**.

The continuation arrow appears in the drawing.



Spool Numbers

Use this tool to insert spool number tags.

To insert spool numbers...

1. From the *Miscellaneous* tab, select **Spool Mark Number**.

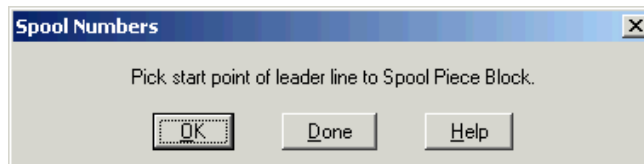


The **Spool Numbers** prefix dialog box appears.



2. In the **Spool Number Prefix** box, do one of the following:
 - Accept the default prefix value
 - Type a new value
3. Click **OK**.

The **Spool Numbers** pick start point dialog box appears.



4. In this dialog box, do one of the following:
 - Click **OK** to insert a spool piece block
 - Click **Done** to exit the tool
5. In the drawing, click to select the start point of the leader line.
6. Click to select the block location.

The **Spool Numbers** piece number dialog box appears.

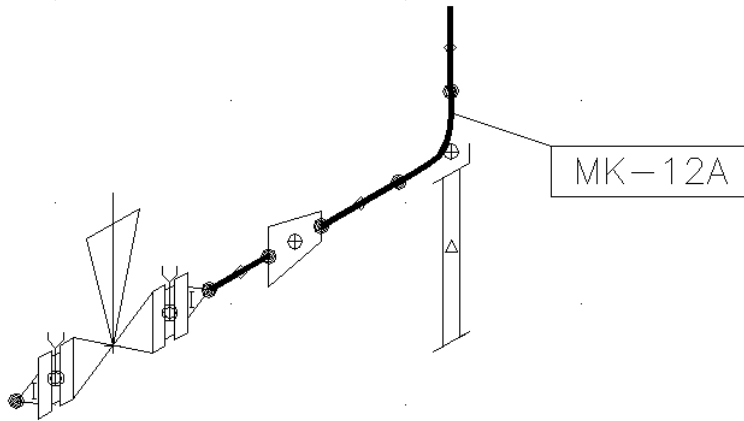


7. In the **Spool Piece Number** box, type a number.
8. Click **OK**.

The block appears in the drawing.

The **Spool Numbers** dialog box reappears.

9. In this dialog box, do one of the following:
 - Click **OK** to insert another spool piece block; repeat Steps 5-8
 - Click **Done** to exit the tool



Modify the Spool Piece Prefix Text

Modify the default prefix value in the **Prefix Text for Spool Piece Number Blocks** field on the **General Parameters** dialog box.

Field Weld Symbol

To insert a field weld symbol...

1. From the *Miscellaneous* tab, select **Field Weld Symbol**.



2. In the drawing, click where you want to insert the field weld.

The field weld appears in the drawing.

Weld Dot Symbol

To insert a weld dot symbol...

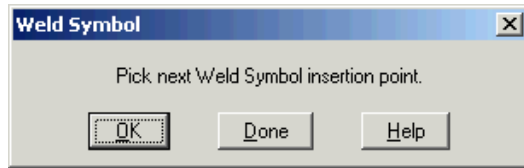
1. From the *Miscellaneous* tab, select **Weld Symbol**.



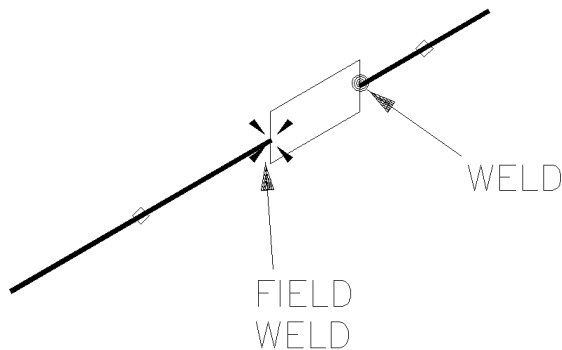
2. In the drawing, click where you want to insert the weld dot.

The weld dot appears in the drawing.

The **Weld Symbol** dialog box appears.



3. Choose one of the following options:
 - Click **OK** to insert another weld dot
 - Click **Done** to exit the tool without inserting another weld dot



Pipe Insulation

Before you insert pipe insulation, move the pipe's INFO block to the outside of the pipe insulation symbol. To do this, turn off object grouping in AutoCAD (Ctrl+Shift+A).

Set the scaling factor and the break distance for the pipe insulation symbol on **Component Sizing – Page 2**.

To insert pipe insulation...

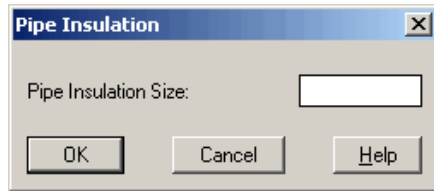
1. From the **Miscellaneous** tab, select **Pipe Insulation**.



➤ **Notes:** The left icon draws **Traced Pipe** Insulation.

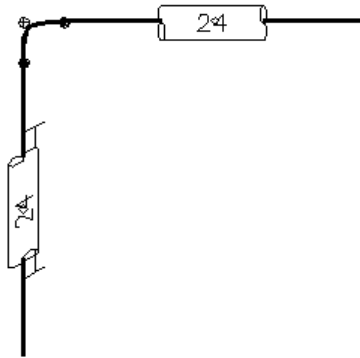
2. In the drawing, click the pipe where you want to insert the pipe insulation.

The **Pipe Insulation** dialog box appears.



3. In the **Pipe Insulation Size** box, type the insulation thickness.
4. Click **OK**.

The pipe insulation appears in the drawing.



Slope Symbol

To insert a slope symbol...

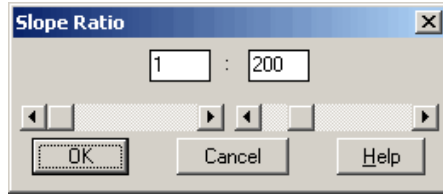
1. From the *Miscellaneous* tab, select **Slope**.



The **Slope** isoplane dialog box appears.

2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click where you want to insert the slope symbol.
4. Click to select the direction.

The **Slope Ratio** dialog box appears.



5. Type the slope ratio, or use the arrows to change the values.
6. Click **OK**.

The slope symbol appears in the drawing.



Grade

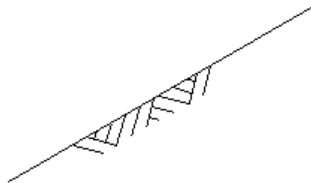
To insert a grade symbol...

1. From the *Miscellaneous* tab, select **Grade**.



2. In the drawing, click where you want to insert the grade symbol.
3. Click to select the direction.

The grade symbol appears in the drawing.



Specialty Items

To insert a specialty item...

1. From the *Miscellaneous* tab, select **Specialty Item**.



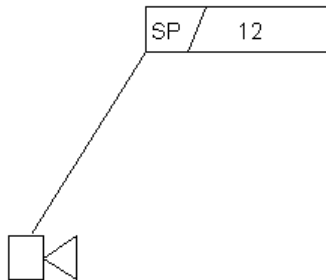
2. In the drawing, click to select the leader line start point.
3. Click to select the specialty item block location.

The **Specialty Item** dialog box appears.



4. In the **Specialty Item number** box, type a value.
5. Click **OK**.

The specialty item block appears in the drawing.



- **Note:** This tool deactivates the ORTHO toggle until the block appears in the drawing. This allows you to insert the leader line in any direction.

Spec Breaks

To insert a spec break...

1. From the *Miscellaneous* tab, select **Spec Break**.



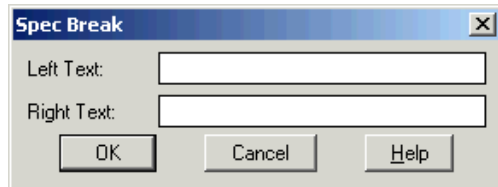
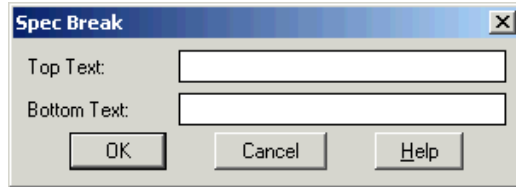
The **Spec Break** isoplane dialog box appears.

2. Choose one of the following options:

- Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click where you want to insert the spec break.
 4. Click to select the direction of the spec break line.

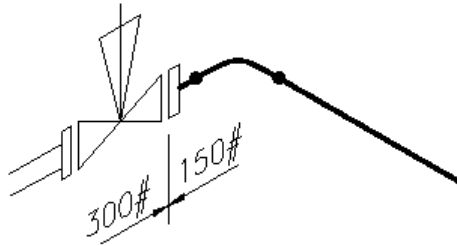
The **Spec Break** text dialog box appears.

PROCAD SPOOLCAD detects the isoplane orientation and prompts for top/bottom text or left/right text.




5. In the text input boxes, type the required text.
6. Click **OK**.
7. In the drawing, click to select the end point of the leader line.






The spec break appears in the drawing.



Lines

PROCAD SPOOLCAD provides the following linetypes:

Symbol	Name	Use For...
	Pipe Routing Polyline	Pipe routing to insert AutoELBOW

	Limits Border Line	Drawing and battery limits polyline (phantom linetype) set at same width as that of pipe thickness
	Dash Line	Inserting a Dashed linetype
	Center Line	Inserting a Center linetype
	Hidden Line	Inserting a Hidden linetype
	Dotted Line	Inserting a Dotted linetype

Inserting Lines

To insert a line...

1. From the **Line Types** tab, select the appropriate line type icon.
2. In the drawing, click to select the first point.
3. Click to select another point.
4. Continue clicking to select line points as required.
5. Press **Enter** to end the line.

PIPE ROUTING POLYLINE

BORDER LIMITS LINE

DASH LINE

CENTER LINE

HIDDEN LINE

DOTTED LINE

Text and Text Editing

Isometric Text

PROCAD SPOOLCAD provides a variety of isometric text styles with preset orientation and rotation angles.

Inserting Isometric Text

Select the correct isoplane setting before inserting text.

To insert isometric text...

1. From the **Text** tab, select the text tool required.



Standard Text



ISO Text R-30



ISO Text R-90



ISO Text L-30



ISO Text L-90



ISO Text R+30



ISO Text R+90

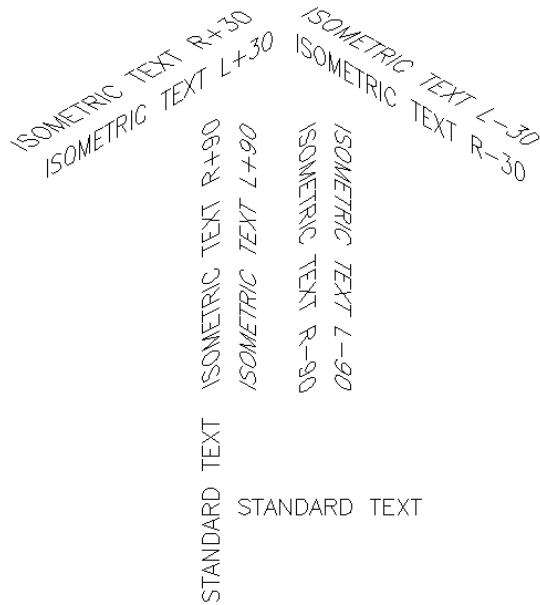


ISO Text L+30



ISO Text L+90

2. In the drawing, click to select the text start point.
3. Type the text.
4. Choose one of the following options:
 - Press **Enter** to type another line
 - Press **Enter** twice to exit the tool



Text Editing

Replacing Text

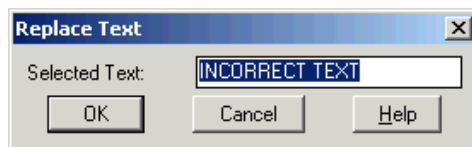
To replace text...

1. From the **Change** tab, select **Replace Text**.



2. In the drawing, click the text you want to replace.
3. Press **Enter**.

The **Replace Text** dialog box appears.



4. In the **Selected Text** box, type the new text.
5. Click **OK**.

The text change appears in the drawing.

Changing Text from Upper to Lower Case

To change text from upper to lower case...

1. From the **Change** tab, select **Change Uppercase to Lowercase**.



2. In the drawing, click the text you want to change.
3. Press **Enter**.

All selected upper case text changes to lower case.

Change Text from Lower to Upper Case

To change text from lower to upper case...

1. From the **Change** tab, select **Change Lowercase to Uppercase**.



2. In the drawing, click the text you want to change.
3. Press **Enter**.

All selected lower case text changes to upper case.

Editing Instrument Balloon Text

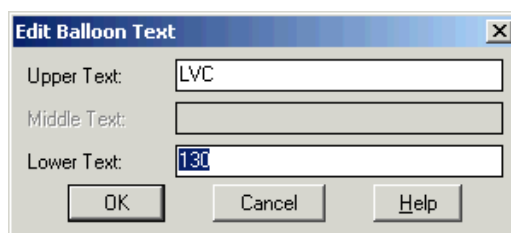
To edit instrument balloon text...

1. From the **Miscellaneous** tab, select **Edit Balloon Text**.



2. In the drawing, click the balloon you want to edit.

The **Edit Balloon Text** dialog box appears:



3. Edit the text as required.
4. Click **OK**.

The new balloon text appears in the drawing.

Dimension Text Utilities

This tool allows you to do the following:

- Change Dimension Text Width
- Move Dimension Text
- Change Dimension Text

Changing the Dimension Text Width

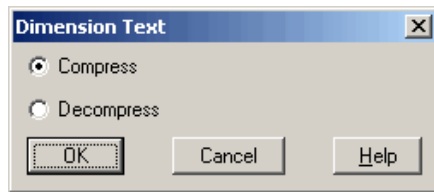
To change the dimension text width...

1. From the *Utilities* tab, select **Dimension Text Utilities**.



2. In the drawing, click the dimension text you want to modify.

The **Dimension Text** dialog box appears.



2. In this dialog box, select the required dimension width option:
 - Click **Compress** to compress the text to 80% of the original
 - Click **Decompress** to return the text to 100% of the original
3. Click **OK**.

The text modification appears in the drawing.

Moving Dimension Text

1. From the *Utilities* tab, select **Dimension Text Utilities**.



2. Select the dimension text that you wish to move.

1. From the command prompt the following options are available:
 - Left – Moves the text close to the left extension line.
 - Right – Moves the text close to the right extension line.
 - Centre – Moves the text to the middle of the extension lines
 - Home – Moves the text to the last position specified when using the Dimension Text Utility.
 - Angle – Rotates the text the specified angle about its current position.

2. At the command prompt, enter the desired option.
If you have selected the Angle option, enter the desired rotation angle.

The text modification appears in the drawing.

Changing the Dimension Text Value

Use this procedure to modify a dimension text value. PROCAD SPOOLCAD does not verify the validity of the changed dimension.

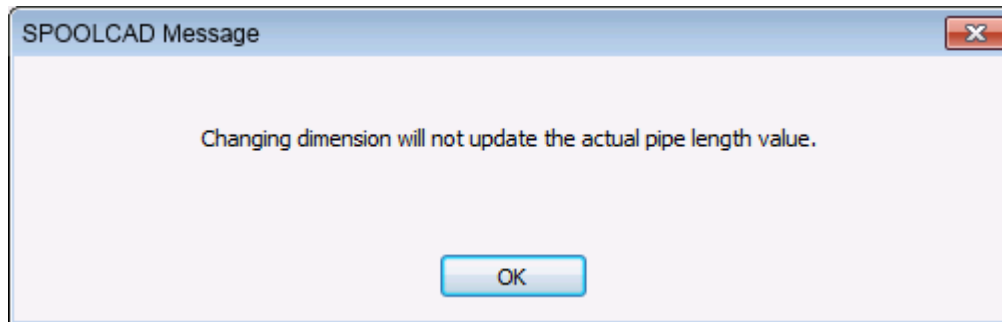
If the changed dimension contains a pipe segment, the system will not update pipe length. Use **Change Pipe Length** instead.

To change the dimension text...

1. From the *Utilities* tab, select **Edit Text**.

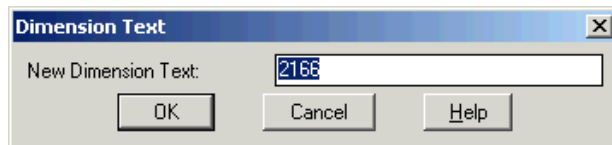


The **SPOOLCAD Message** box appears.



2. Click **OK**.
3. In the drawing, click the dimension text you want to change.

The **Dimension Text** dialog box appears.



4. In the **New Dimension Text** box, type a new dimension value.
5. Click **OK**.

The text change appears in the drawing.

Utilities and Change Tools

Freeze DEFPOINTS Layer

INFO blocks, gasket symbols, and bolt symbols appear in the DEFPOINTS layer, which does not plot.

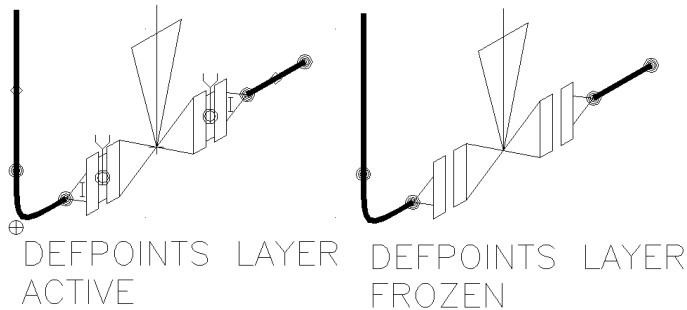
To hide the DEFPOINTS layer...

1. From the *Utilities* tab, select **Freeze\Thaw Defpoints Layer**.



All components on this layer disappear from the drawing.

2. Click **Freeze\Thaw Defpoints Layer** again to display the components.



Borders Blocks

See *Border Drawing Blocks*.

ISO Dimensioning, Offset Dimensioning and Dimension Text

See *PROCAD SPOOLCAD* Dimensioning and *Dimension Text Utilities*.

Equipment Tags

Use this procedure to insert equipment tags in your drawing.

To insert an equipment tag...

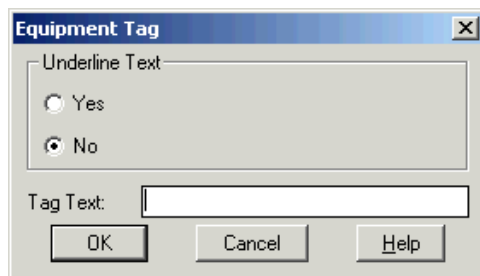
1. From the *Utilities* tab, select **Equipment Tag**.



The **Equipment Tag** isoplane dialog box appears.

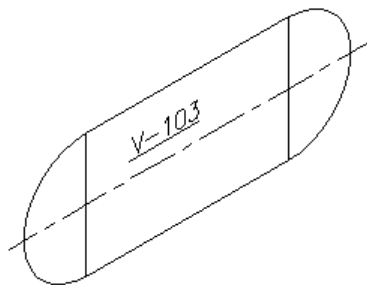
2. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
3. In the drawing, click to select the tag text center point.

The **Equipment Tag** dialog box appears.



4. From the **Underline Text** options, select one of the following:
 - Click **Yes** to underline the text
 - Click **No** to leave it plain
5. In the **Tag Text** box, type the text you want to insert.
6. Click **OK**.

The equipment tag appears in the drawing.



Valve Tags

Select the correct isoplane setting before inserting valve tags.

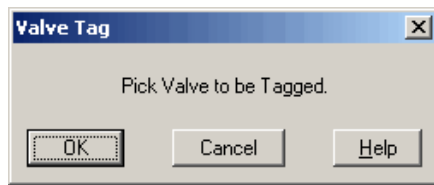
PROCAD SPOOLCAD extracts valve tag information from the valve's INFO block.

To insert a valve tag...

1. From the **Utilities** tab, select **Valve Tag**.



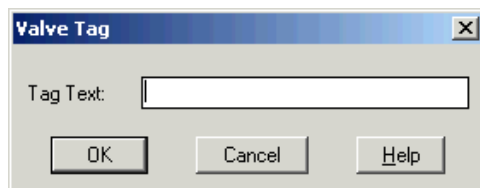
The **Valve Tag** dialog box appears.



2. Click **OK**.
3. In the drawing, click the valve you want to tag.
4. Press **Enter**.
5. In the drawing, click to select the center point for the tag text.

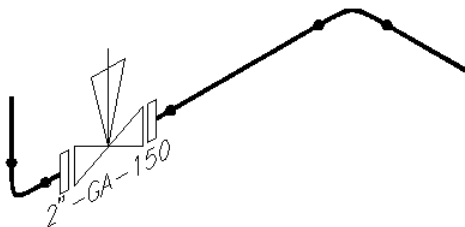
The tag appears in the drawing.

If the valve INFO block does not contain a valve tag, the **Valve Tag** dialog box appears.



6. In the **Tag Text** box, type the valve tag information.
7. Click **OK**.

The tag appears in the drawing.



Line Number Tags

Select the correct isoplane setting before inserting line number tags.

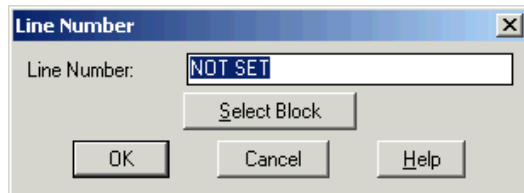
To insert a line number tag...

1. From the *Utilities* tab, select **Line Number**.



The **Line Number** dialog box appears.

The dialog box displays the current line number or **NOT SET**.



2. In this dialog box, do one of the following:
 - In the **Line Number** box, type a line number and click **OK**; go to Step 5
 - Click **Select Block** to extract the line number from an existing INFO block

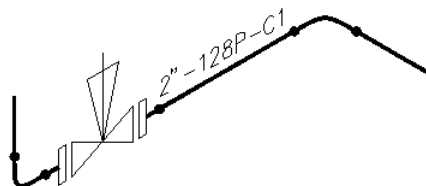
If you click **Select Block**, the cursor changes to a square.

3. In the drawing, click the INFO block of a component containing the required line number.
4. Press **Enter**.

The **Line Number** isoplane dialog box appears.

5. Choose one of the following options:
 - Click **OK** to accept the current isoplane
 - Click **Change Current Isoplane** and select a new isoplane setting
6. In the drawing, click to select the text center point.

The line number tag appears in the drawing.



Branch Fitting Query

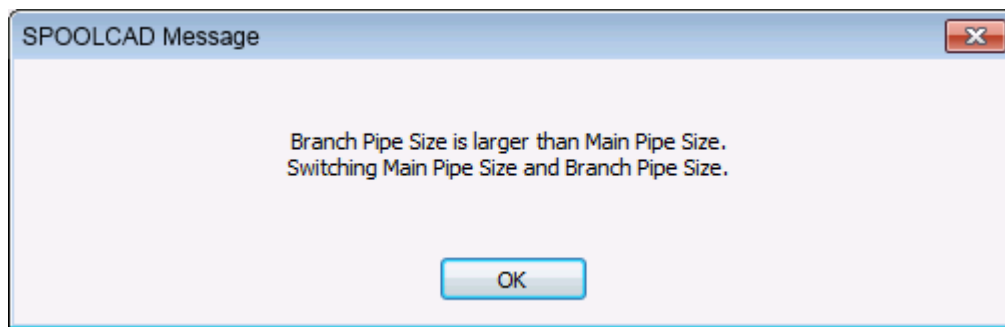
Use this tool to view the branch fitting type specified in the piping spec for the current main pipe size and branch size.

To view the branch information settings...

1. From the *Utilities* tab, select **Branch Fitting Query**.

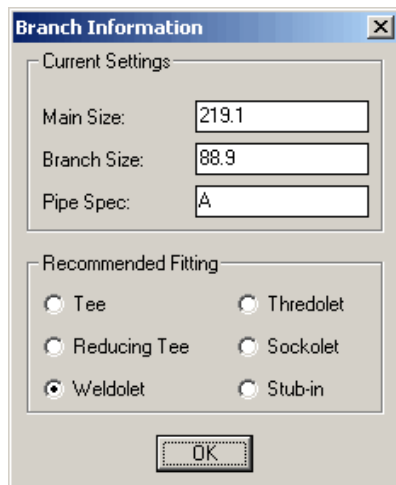


If the main pipe size is smaller than branch pipe size, the **SPOOLCAD Message** box appears.



2. Click **OK**.

The **Branch Information** dialog box appears.



3. Review the settings. You cannot edit data in this dialog box.
4. Click **OK**.

Data File Names

See *Finding a Data File*.

Making Charts

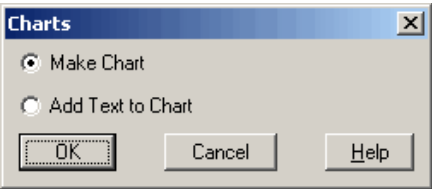
Use this tool to draw charts and add text to charts. The chart appears in the drawing's current active layer and uses AutoCAD's 'continuous' line type.

To draw a chart...

1. From the *Utilities* tab, select **Make Chart**.

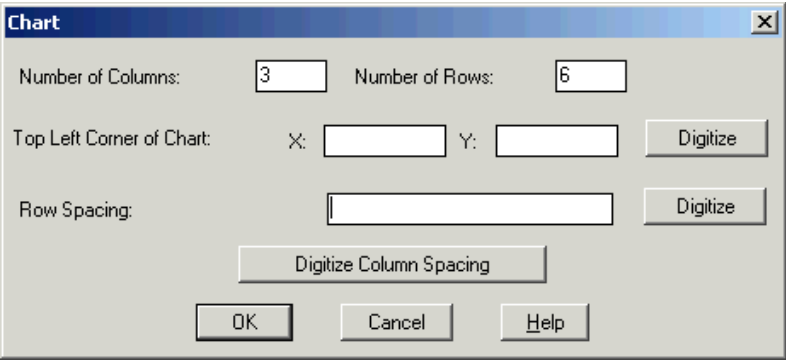


The **Charts** dialog box appears.



2. From the options, select **Make Chart**.
3. Click **OK**.

The **Chart** dialog box appears.



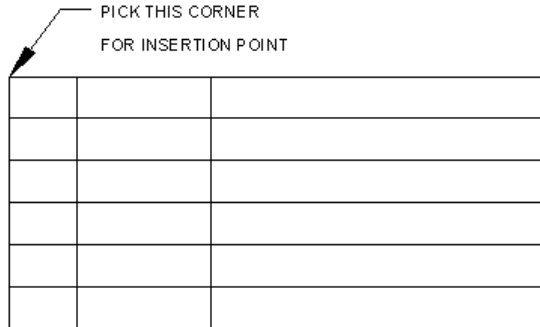
4. In this dialog box, do the following:

In this...	Do this...
Number of Columns input box	Type the required number of columns
Number of Rows input box	Type the required number of rows
Top Left Corner of Chart	Type coordinates in the input boxes, or click

X and Y input boxes	Digitize and click in the drawing to select the coordinates
Row Spacing input box	Type the row spacing value, or click Digitize and click in the drawing to select the row spacing
Digitize Column Spacing button	Click the button, then click in the drawing to select the start point for each column

- Click **Draw Chart**.

The chart appears in the drawing.



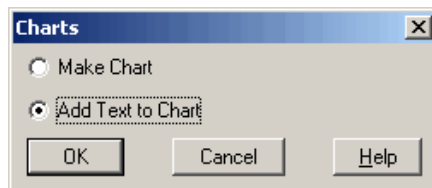
Adding Text to Chart

To add text to a chart...

- From the *Utilities* tab, select **Make Chart**.

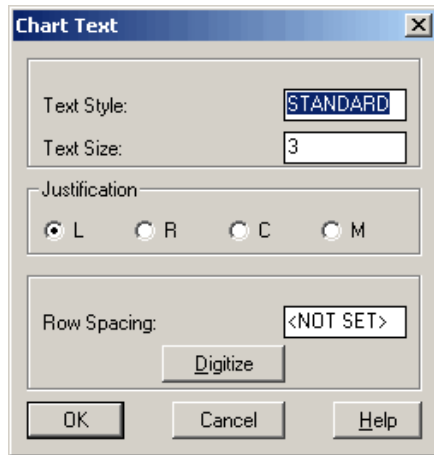


The **Charts** dialog box appears.



- From the options, select **Add Text to Chart**.
- Click **OK**.

The **Chart Text** dialog box appears.



4. In this dialog box, do the following:

In this...	Do this...
Text Style input box	Accept the default style, or type a new style name; the two styles available are STANDARD and ROMAND
Text Size input box	Accept the default text size, or type a new text size value
Justification options	Select a text justification setting: L: insertion point = bottom left of text block R: insertion point = bottom right of text block C: insertion point = bottom center of text M: insertion point = middle of text block
Row Spacing input box	Do one of the following: Type a value in the input box to set the row height and click OK ; in the drawing, click where you want to insert the first line of text or Click Digitize , and click two vertical points in the drawing to set the row height; the first point you select becomes your text insertion point

The **Chart Text** dialog box appears.



5. In the **Text Line** box, type the required text.
6. Click **OK**.

The text appears in the drawing.

The **Chart Text** dialog box reappears. This lets you add another line of text.

7. When you are finished inserting text, click **Cancel** to exit the tool.

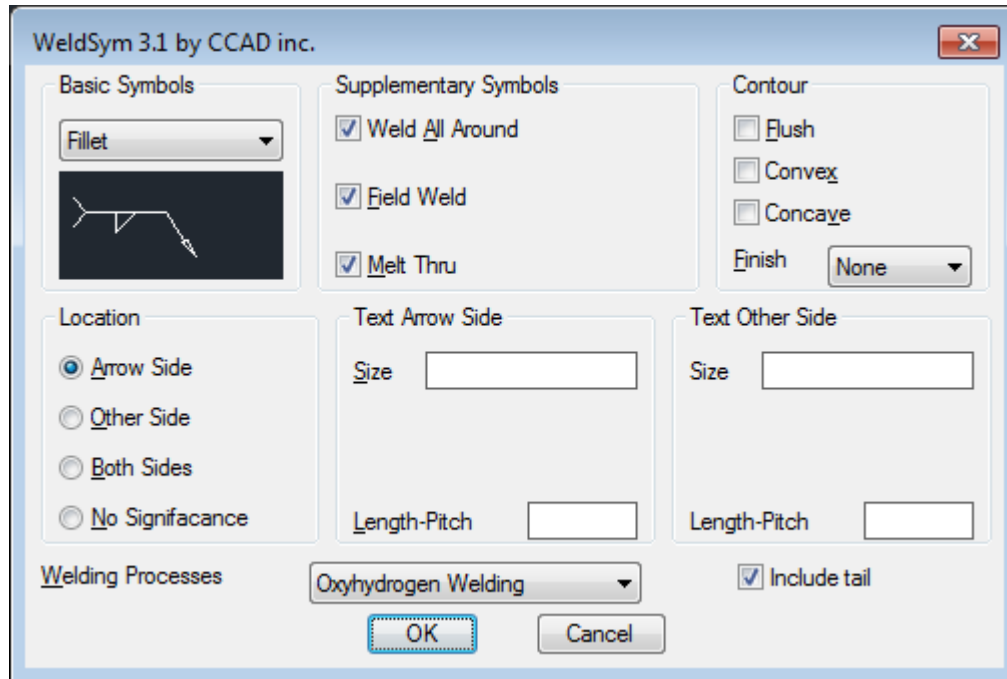
Weld Symbol

The **Weld Symbol** tool provides a quick way insert wend symbols.

1. From the *Changes* tab, select **Weld Symbol**.



2. The **WeldSym** dialog box will appear



3. Enter the information required for this weld symbol.
4. Click **OK** when you have entered the weld symbol information.
5. Click a location on the drawing to insert the weld symbol.
6. Click the location and length for the leader line.
7. Use the keyboard and press the **Enter** key to place the Weld Symbol.

User Defined Grid

The **User Defined Grid** tool provides a quick way to change the drawing grid increments.

1. From the *Changes* tab, select **User Defined Grid**.



2. In the drawing, click to select the first grid point.

3. Click to select the second grid point.

The grid changes to reflect the new settings.

➤ **Note:** You can also use AutoCAD's GRID command.

User Snap Settings

The **User Snap Setting** tool provides a quick way to change the snap value.

1. From the **Changes** tab, select **User Snap Setting**.



2. In the drawing, click to select the first snap point.
3. Click to select the second snap point.

The snap changes to reflect the new settings.

➤ **Note:** You can also use AutoCAD's SNAP command.

Layer Tools

Use layer tools to move drawing entities from one layer to another.

Important: If you move piping components from one layer to another, the component INFO blocks also moves to the new layer and will appear when you plot the drawing.

Changing Entities to the Current Layer

To change entities to the current layer...

1. From the **Changes** tab, select **Change Items to Current Layer**.



2. In the drawing, click to select the components to move to the current layer.
3. Press **Enter**.

The selected components appear in the current layer.

Changing Entities To the Selected Entity's Layer

To change entities to the selected entity's layer...

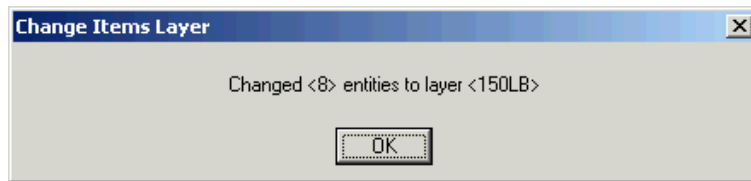
1. From the **Changes** tab, select **Change Item to Selected Layer**.



2. In the drawing, click to select a component in the destination layer.
3. Click to select the components you want to move.
4. Press **Enter**.

The selected components appear in the destination layer.

The **Change Items Layer** notification box appears.



5. Click **OK**.

Changing Layers to the Selected Entity's Layer

To change the layer to the selected entity's layer...

1. From the **Changes** tab, select **Change Layer to Entity's**.



2. In the drawing, click to select a component in the layer you want to make current.

That component's layer becomes the current layer.

Moving Layer Contents

To move the layer contents...

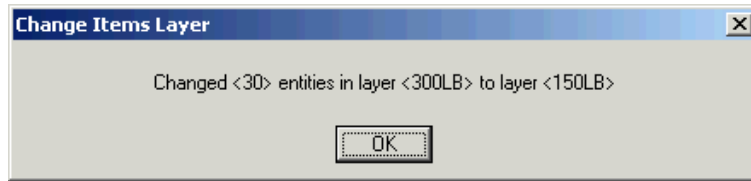
1. From the **Changes** tab, select **Move Layer Contents to New Layer**.



2. In the drawing, click to select a component in the source layer.
3. Click to select a component in the destination layer.

The component from the source layer appears in the destination layer.

The **Change Items Layer** notification box appears.



4. Click **OK**.

Dynamic Attribute Edit

Use AutoCAD's Dynamic Attribute Edit (DDATTE) command to edit the information in a component's INFO block.

Unless the changes are minor, such as editing the BOM number, Line number or bolt length, PROCAD recommends you change component information using *Spec Generator* instead. Refer to the *Spec Generator User Guide*.

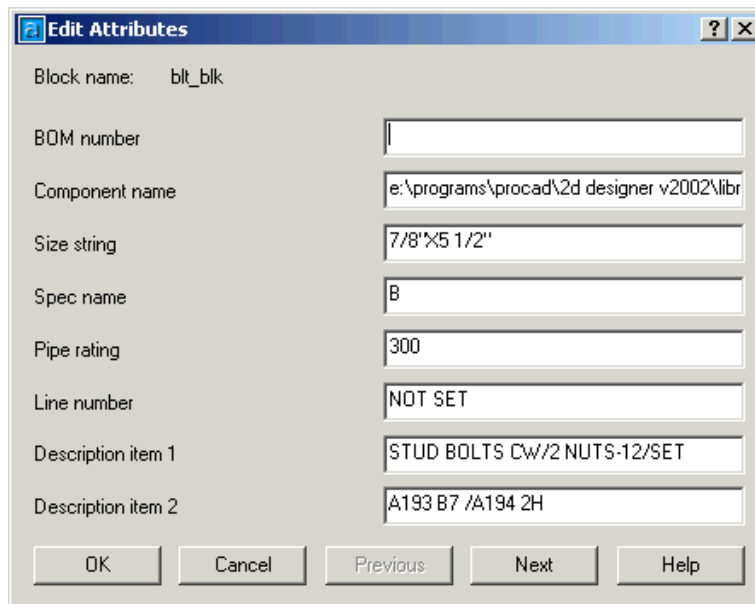
To edit attributes...

1. From the **Change** tab, select **Dynamic Attribute Edit**.



2. In the drawing, click the INFO block of the component you want to edit. Press **Enter**.

The **Edit Attribute** dialog box appears.



3. Edit the information according to your requirements.

4. Click **Next** to go to the next page. There are four pages in total.
5. Edit the information on all pages as required.
6. Click **OK** to save the changes and return to the drawing.

Global Block Attribute Changes

Use these tools to change global drawing information for material destination, drawing number and line number blocks. Object Grouping must be active for these tools to work properly.

Changing the Material Destination

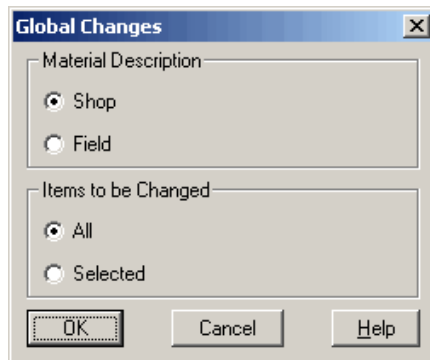
Use this tool to change the shipping destination for components.

To change the material destination...

1. From the **Changes** tab, select **Shop or Field Destination**.



The **Global Changes** dialog box appears.



2. From the **Material Description** options, select **Shop** or **Field**.
3. From the **Items to be Changed** options, select one of the following:
 - Click **All** to change destination data for the entire drawing
 - Click **Selected** to change the destination only for selected components

4. Click **OK**.

If you clicked **All**, PROCAD SPOOLCAD changes the destination information for all INFO blocks.

5. If you click **Selected**, click the INFO blocks of drawing components you want to change.
6. When you are finished selecting components, press **Enter**.

PROCAD SPOOLCAD inserts the new destination information in the component INFO blocks.

Changing Line Numbers

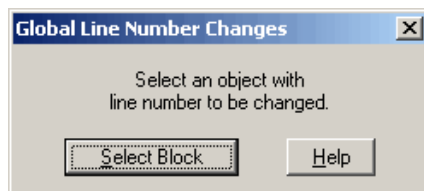
Use this tool to change the line number attribute in all component INFO blocks.

To change the line number...

1. From the **Changes** tab, select **Line Number**.

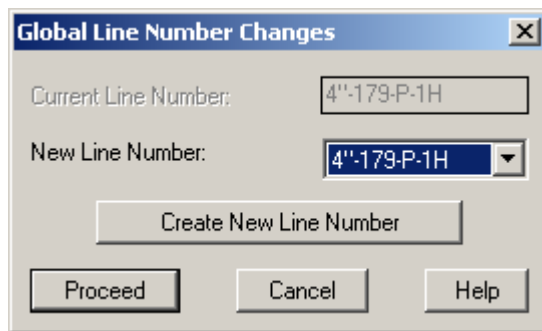


The **Global Line Number Changes** dialog box appears.



2. Click **Select Block**.
3. In the drawing, click the INFO block of a component within the line you want to change.
4. Press **Enter**.

The **Global Line Number Change** dialog box appears.



5. Use the **New Line Number** drop down box to select a previously created line number. If the line number you need has not yet been generated, click the **Create New Line Number** button and type the new line number.
6. Click **Proceed**.

All component INFO blocks within that line now contain the new line number.

Changing Drawing Numbers

Use this tool when reusing existing drawings from previous standards for new standards.

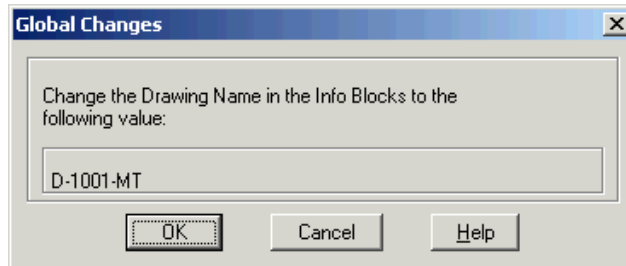
This tool inserts the drawing *file name* in the drawing *number* attribute field of all affected INFO blocks. This ensures AutoDATA extracts accurate drawing information.

To change the drawing number...

1. From the **Changes** tab, select **Drawing Number**.



The **Global Changes** dialog box appears.



2. Click **OK**.

The new drawing name/number appears in all affected INFO blocks.

Component Changes

Match Settings to Selected Component

Use this procedure to change the drawing's main pipe size, branch pipe size and spec settings to match a selected component.

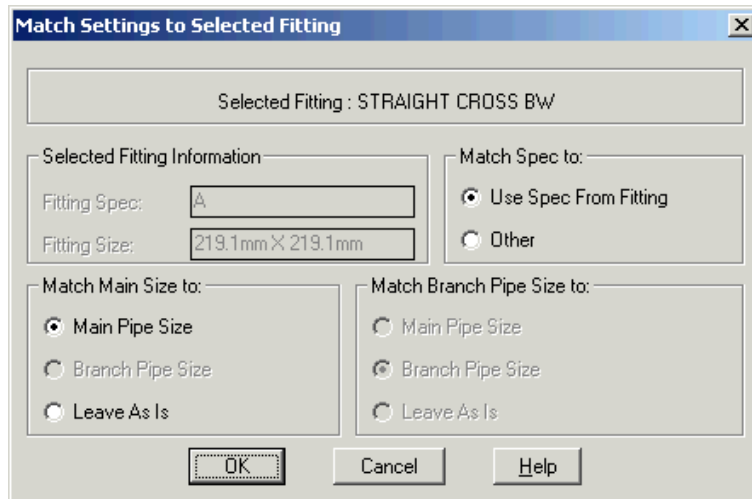
To match settings to an existing component...

1. From the **Settings** tab, select **Match Settings to Item**.



2. In the drawing, click the component that has the settings you want.
3. Press **Enter**.

The **Match Settings to Selected Fitting** dialog box appears.



4. From the **Match Spec to** options, select **Use Spec From Fitting**.
5. From the **Match Main Size to** options, select one of the following
 - Click **Main Pipe Size** to set the size to that of the selected fitting
 - Click **Leave As Is** to leave the main pipe size unchanged
6. Click **OK**.

The drawing settings change to match the component.

Switch Main Pipe Size and Branch

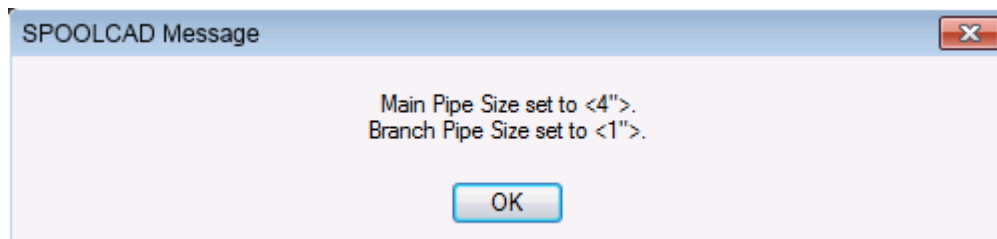
Use this tool to switch the main pipe and branch pipe sizes.

To switch the main and branch pipe sizes...

1. From the **Settings** tab, select **Switch Main and Branch Sizes**.



The **SPOOLCAD Settings** notification box appears.



2. Click **OK**.

Plotting

Plotting a Drawing

To plot a drawing, use AutoCAD's PLOT command. Refer to your AutoCAD manual for details.

Layering System

The Standard Manager controls all the program's generic layers. This layering system places entity groups on different layers, and assigns each group a color and line thickness. This makes it easy to configure the application for pen plotters or printer-type plotters such as laser and ink jets.

Use the Standard Manager to change layer colors, line type and line thickness.

Modifying Drawing Component Sizing

To modify drawn line thickness for pipes and fittings, change the values in the **BW Fitting Polyline Width** and **SCRD/SW Fitting Polyline Width** fields on the **Component Sizing – Page 1** dialog box.

Laser and Ink Jet Plotters

PROCAD SPOOLCAD assigns these types of plotters as the default.

Line width is plotted according to the polyline's default thickness. Control plot appearance by assigning line widths to layers or colors.

Pen Plotters

If using a multi-pen plotter, assign specific line colors to pens with different thicknesses or set all the polyline widths to zero and allow the pen thickness to determine the plotted width of that line.

If using a single pen plotter, set all pen widths to zero, and use polyline widths to control line appearance.

Appendix A: Pipe Sizes

You can display pipe sizes in English, Metric NPS (Nominal Pipe Size), English NPS, and Metric OD (Outside Diameter).

English	English NPS	Metric NPS	Metric O.D.
1/8"	NPS 1/8	3	10.3
1/4"	NPS 1/4	5	13.7
3/8"	NPS 3/8	10	17.1
1/2"	NPS 1/2	15	21.3
3/4"	NPS 3/4	20	26.7
1"	NPS 1	25	33.4
1 1/4"	NPS 1 1/4	32	42.2
1 1/2"	NPS 1 1/2	40	48.3
2"	NPS 2	50	60.3
2 1/2"	NPS 2 1/2	65	73
3"	NPS 3	80	88.9
3 1/2"	NPS 3 1/2	90	101.6
4"	NPS 4	100	114.3
5"	NPS 5	125	141.3
6"	NPS 6	150	168.3
8"	NPS 8	200	219.1
10"	NPS 10	250	273.1
12"	NPS 12	300	323.9
14"	NPS 14	350	355.6
16"	NPS 16	400	406.4
18"	NPS18	450	457.2
20"	NPS 20	500	508
22"	NPS 22	550	558.8
24"	NPS 24	600	609.6
26"	NPS 26	650	660.4
28"	NPS 28	700	711.2
30"	NPS 30	750	762
32"	NPS32	800	812.8
34"	NPS 34	850	863.6
36"	NPS 36	900	914.4
38"	NPS 38	950	965
40"	NPS 40	1000	1016
42"	NPS 42	1050	1066.8

44"	NPS 44	1100	1118
46"	NPS 46	1150	1168
48"	NPS 48	1200	1219
52"	NPS 52	1300	1321
56"	NPS 56	1400	1422
60"	NPS 60	1500	1524
64"	NPS 64	1600	1626
68"	NPS 68	1700	1727
72"	NPS 72	1800	1829
76"	NPS 76	1900	1930
80"	NPS 80	2000	2032

API Pipe Sizes

PROCAD SPOOLCAD uses a limited number of API pipe sizes. These sizes are assigned ANSI pipe sizes in the dimensional data files. If any changes of dimensional data are required, refer to the ANSI sizes for identification.

API Size	ANSI Size
2 1/16"	2"
2 9/16"	2 1/2"
3 1/8"	3"
4 1/16"	4"
7 1/16"	8"
9"	10"
11"	12"
13 5/8"	14"
16 3/4"	16"
18 3/4"	18"

Appendix B: PROCAD SPOOLCAD Layers

The following chart shows the standard layer names, colors, and linetypes used by PROCAD SPOOLCAD.

You can modify PROCAD SPOOLCAD's standard layering system to accommodate your in-house requirements. See the *Standard Manager*.

AutoCAD does not plot the DEFPOINTS layer.

Layer Name	Color	Line Type
0	WHITE	CONTINUOUS
1500LB	RED	CONTINUOUS
150LB	YELLOW	CONTINUOUS
2000LB	RED	CONTINUOUS
2500LB	YELLOW	CONTINUOUS
3000LB	YELLOW	CONTINUOUS
300LB	GREEN	CONTINUOUS
4000LB	CYAN	CONTINUOUS
400LB	CYAN	CONTINUOUS
6000LB	GREEN	CONTINUOUS
600LB	BLUE	CONTINUOUS
900LB	MAGENTA	CONTINUOUS
5000LB	WHITE	CONTINUOUS
10000LB	WHITE	CONTINUOUS
ATTRIB	WHITE	CONTINUOUS
BORDER1	CYAN	CONTINUOUS
BORDER2	YELLOW	PHANTOM
CENTER	RED	CENTER
CONSTR	WHITE	CONTINUOUS
CONTIN	YELLOW	CONTINUOUS
DASH	BLUE	DASH
DEFPOINTS	WHITE	CONTINUOUS
DIMENSION	YELLOW	CONTINUOUS
DOT	WHITE	DOT
EQUIP	YELLOW	CONTINUOUS
HIDDEN	MAGENTA	HIDDEN
INSTR	MAGENTA	CONTINUOUS
MISC	WHITE	CONTINUOUS

Layer Name	Color	Line Type
B.O.M.	YELLOW	CONTINUOUS
MTO-BAL	RED	CONTINUOUS
SUPTS	GREEN	CONTINUOUS
TEXT1	YELLOW	CONTINUOUS
TEXT2	GREEN	CONTINUOUS

Appendix C: Valve Types

The valve type placed in the drawing depends on the current spec and drawing settings. See *Viewing Drawing Settings*.

To insert a valve that is different from that set in the spec, activate the Spec Check toggle. See *Spec Check Toggle*.

PROCAD SPOOLCAD uses the following ratings for screwed/socket weld valves:

- 800#
- 1500#
- 2500#

PROCAD SPOOLCAD uses the following patterns for flanged and welded valves:

Ball Valves

- Reducing Port
- Full Port
- Venturi

Check Valves

- Swing
- Piston
- Wafer

Plug Valves

- Short Pattern
- Regular Pattern
- Venturi Pattern

Appendix D: Pressure Safety Valve List

The PSV sizes included in the PSV.DAT file are listed below. User can add own sizes. See *Inserting Pressure Safety Valves*.

Inlet Size X Outlet Size (Inches)	Orifice Designation	Inlet Rating (Pounds)	Outlet Rating (Pounds)
1" x 2"	D	150	150
1" x 2"	D	300	150
1" x 2"	D	600	150
1 1/2" x 2"	D	900	300
1 1/2" x 2"	D	1500	300
1 1/2" x 2 1/2"	D	2500	300
1" x 2"	E	150	150
1" x 2"	E	300	150
1" x 2"	E	600	150
1 1/2" x 2"	E	900	300
1 1/2" x 2"	E	1500	300
1 1/2" x 2 1/2"	E	2500	300
1 1/2" x 2"	F	150	150
1 1/2" x 2"	F	300	150
1 1/2" x 2"	F	600	150
1 1/2" x 2 1/2"	F	900	300
1 1/2" x 2 1/2"	F	1500	300
1 1/2" x 2 1/2"	F	2500	300
1 1/2" x 2 1/2"	G	150	150
1 1/2" x 2 1/2"	G	300	150
1 1/2" x 2 1/2"	G	600	150
1 1/2" x 2 1/2"	G	900	300
2" x 3"	G	1500	300
2" x 3"	G	2500	300
1 1/2" x 3"	H	150	150
1 1/2" x 3"	H	300	150
2" x 3"	H	300	150
2" x 3"	H	600	150
2" x 3"	H	900	150
2" x 3"	H	1500	300

Inlet Size X Outlet Size (Inches)	Orifice Designation	Inlet Rating (Pounds)	Outlet Rating (Pounds)
2" x 3"	J	150	150
2" x 3"	J	300	150
2 1/2" x 4"	J	300	150
2 1/2" x 4"	J	600	150
3" x 4"	J	900	150
3" x 4"	J	1500	300
3" x 4"	K	150	150
3" x 4"	K	300	150
3" x 4"	K	600	150
3" x 6"	K	900	150
3" x 6"	K	1500	300
3" x 4"	L	150	150
3" x 4"	L	300	150
4" x 6"	L	300	150
4" x 6"	L	600	150
4" x 6"	L	900	150
4" x 6"	L	1500	150
4" x 6"	M	150	150
4" x 6"	M	300	150
4" x 6"	M	600	150
4" x 6"	M	900	150
4" x 6"	N	150	150
4" x 6"	N	300	150
4" x 6"	N	600	150
4" x 6"	N	900	150
4" x 6"	P	300	150
4" x 6"	P	600	150
4" x 6"	P	900	150
6" x 8"	Q	300	150
6" x 8"	Q	600	150
6" x 8"	R	300	150
6" x 10"	R	600	150
8" x 10"	T	300	150

Appendix E: Generic Pipe Specs

Generic spec files are located in the [Network]\Datafiles\Specs directory.

PROCAD supplies these generic specifications as samples only. They cover ANSI standards for 150# to 600#. Before using generic specs, review them thoroughly.

Use **Spec Generator** to generate specs that conform to your own specification requirements. Refer to the **Spec Generator User Guide**.

Specifications for Generic Spec A

ANSI Rating:	150#
Flange Face Type:	Raised Face
WOL & EOL weight:	Standard
Lap joint stub end weight:	Standard
Straight/Reducing Cross weight:	Standard
Ball valve type:	Reducing Port
Plug valve type:	Short Pattern
Check valve type:	Swing
Small fittings type:	Screwed
Small fittings rating:	3000#
Small valve rating:	800#
Start size for small fittings:	1 1/2" and smaller
Gasket thickness:	0.125" (use inch sizes only)
Pipe nipple length:	4" (use inch sizes only)
Bolt type:	Stud Bolts

Specifications for Generic Spec B

ANSI Rating:	300#
Flange Face Type:	Raised Face
WOL & EOL weight:	Standard
Lap joint stub end weight:	Standard
Straight/Reducing Cross weight:	Standard
Ball valve type:	Reducing Port
Plug valve type:	Short Pattern
Check valve type:	Swing
Small fittings type:	Screwed

Small fittings rating:	3000#
Small valve rating:	800#
Start size for small fittings:	1 1/2" and smaller
Gasket thickness:	0.125" (use inch sizes only)
Pipe nipple length:	4" (use inch sizes only)
Bolt type:	Stud Bolts

Specifications for Generic Spec C

ANSI Rating:	600#
Flange Face Type:	Raised Face
WOL & EOL weight:	XS
Lap joint stub end weight	XS
Straight/Reducing Cross weight:	XS
Ball valve type:	Reducing Port
Plug valve type:	Short Pattern
Check valve type:	Swing
Small fittings type:	Screwed
Small fittings rating:	3000#
Small valve rating:	1500#
Start size for small fittings:	1 1/2" and smaller
Gasket thickness:	0.125" (use inch sizes only)
Pipe nipple length:	4" (use inch sizes only)
Bolt type:	Stud Bolts

Branch Fittings Files

The program uses the BRANCH file located in that spec's directory to check branch fitting type. For the generic specs A to F, the branch fittings are based on the following criteria.

Screwed/socket Weld Branch Fittings

1/8" to 1 1/2"

Run equals Branch size	Use TEE
Run does not equal Branch size	Use REDUCING TEE

Butt Weld Branch Fittings

2" and up

Run equals Branch size	Use TEE	
Run does not equal Branch size	Up to 1/2 Run size	Use REDUCING TEE



















Below 1/2 Run size

Use Weldolet® &
Thredolet®

- **Note:** Branch fittings are based on the reduction size. Butt Weld fittings require a Weldolet®; screwed and socket welded fittings require a Thredolet®.


























Appendix F: Button References

Tabs








	Settings		Fittings
	Equipment		Miscellaneous Fittings
	Miscellaneous Drafting Functions		Pipe Supports
	Line Types		Structural Steel
	AutoRoute Feature		Text
	Flanges		Fabrication Tools
	Gaskets and Bolts		Bill of Material
	Pipe		Utilities
	Valves		Changes






Tool Buttons

Settings Tools

























			
	Set Isoplane Orientation		Bolt Length Override
	Select Current Main Pipe Size		Gasket Thickness Override
	Select Current Branch Size		Valve Face to Face Dimension Override
	Select Pipe Spec		Welded Valves Override
	Override Specs		Screwed/Socket Weld Valve Override
	Switch Main and Branch Sizes		Spec Check Toggle
	Match Settings to Item		Branch Check Toggle
	View Component Information		Valve Stem Toggle
	Change Drawing Settings		Material ON/OFF Toggle
	List Current Settings		Delete Entire Line
	Shop/Field Material Destination Override		Set/Change Line Number
	Nipple Length Override		Line Filter

Equipment Tools








			
	Pig Launcher		Vertical Vessel-Full View
	Orifice Tap		Iso Rectangle-Center
	Orifice Tap		Iso Rectangle-Base

	Horizontal Vessel-Flat View		Center Rectangle
	Horizontal Vessel-Full View		Corner Rectangle
	Vertical Vessel-Flat View		





Miscellaneous Drafting Tools

			
	Centerline Symbol		Dual Direction Off Page Arrow
	North Arrow		Left Off Page Arrow
	Trim Lines		Right Off Page Arrow
	Small Directional Arrow		Spool Mark Number
	Large Directional Arrow		Field Weld
	Revision Triangle		Weld Dot
	Revision Cloud		Pipe Insulation-Traced
	Balloon with Leader Line		Pipe Insulation
	Single Line Balloon		Slope
	Hexagonal Balloon		Grade
	Hexagonal Balloon with Line		Specialty Item
	Edit Balloon Text		Spec Break

















Line Type Tools

			
	Pipe Routing Line		Center Line
	Limits Border Line		Hidden Line
	Dash Line		Dotted Line







AutoRoute Tools

	
	Enable AutoRoute Toggle
	Undo Last Item
	Show Current Position

Flange Tools







			
	Insert flange by weld point		Blind Flange
	Insert by face-of-flange		Nozzle
	Flange		Orifice Flange Set
	Flange, Gasket and Bolts		Lap Joint Flange
	Flange Set		API Flange
	Slip-on Flange		Senior Meter Run
	Long Weld Neck Flange		MSS Flange
	MSS Blind Flange		

Gasket and Bolt Tools

























			
	Gasket		Gasket Tick Marks
	Bolt		Change Bolt Type
	Gasket and Bolts		

Pipe Tools










			
	Isoplane Setting		AutoELBOW and Pipe




















	Pipe Ends – Insert Pipe Break		Pipe Nipple
	Pipe End (single)		Pipe
	Ground/Wall Penetration		Stub-in

Valve Tools






















			
	Insert Flange, Gasket and Valve		Plug Valve
	Insert Valve, Gasket and Flange		Angle Valve
	Insert Flange, Gasket, Valve, Gasket, and Flange		Needle Valve
	Pipe		PSV Valve
	Welded by Flanged Gate Valve		Thermal Valve
	Welded by Flanged Globe Valve		Butterfly Valve
	Welded by Flanged Ball Valve		3-Way Valve
	Welded by Flanged Plug Valve		4-Way Valve
	Gate Valve		Control Valve
	Globe Valve		Control Valve Actuator
	Ball Valve		Valve Stem
	Check Valve		

Fittings Tools

			
	Insert Fitting by Weld Point		Eccentric Reducer
	Insert Fitting by Center Point		Concentric Reducer
	Pipe		Reducing Lateral
	Mitered Elbow		Straight Lateral

	Pipe Bend		Reducing Cross
	Reducing Elbow		Straight Cross
	180° Return		Reducing Tee
	Trimmed Elbow		Straight Tee
	45° Elbow		Elbolet®
	90° 3dia Elbow		Thredolet®
	90° SR Elbow		Sockolet®
	90° LR Elbow		Weldolet®
	Eccentric Swage		AutoELBOW and Pipe
	Concentric Swage		

Miscellaneous Fittings Tools

			
	Insert Fitting by Weld Point		Plug
	Insert Fitting by Center Point		Cap
	Flange Spool		Union
	Flange Isolation Kit		Half Coupling
	Quick Connector		Reducing Coupling
	Transition Piece		Full Coupling
	Re-Pad		Y-Strainer
	Bleed Ring		T-Strainer
	Expansion Joint		Conical Strainer
	Spectacle Blind		End Cover

Pipe Support Tools



Pipe Shoe



Pipe Guide



Pipe Anchor



Column Identification Balloon



Base Support



Base Guide



Spring



Dummy Leg



Pipe Hanger



Spring Hanger



U-Bolt



Trunnion

Structural Steel Tools



Wide Flange Support



Channel Support



Angle Support

Text Tools



Standard Text



ISO Text R-90



ISO Text L-90



ISO Text R+90



ISO Text L+90



ISO Text R-30



ISO Text L-30



ISO Text R+30



ISO Text L+30

Bill of Material Tools



Generate BOM



Generate List



Remove BOM



Component Numbers



Single Material Balloon



Special Item



Calculate Pipe Length



Change Pipe Length



Delete INFO Block



AutoDATA

Fab Tools



Update XRay and Welder ID



Update Weld Info



Field Weld Symbol



Remove Field Weld Symbol



Do Not Include Weld in List



Heat Number List



Calculate Diameter Inch



Edit Spool Weight



Edit Pipe End Prep



Edit Component Surface Area



Weld Count



Weld List



Remove Weld Tags and Weld List



Single Weld Tag



Place Weld Tags



Cost Code



Calculate Spool Weight



Pipe End Preparation



Calculate Surface Area













Utilities Tools

















Freeze/Thaw Defpoints Layer



Equipment Tag

	Make Chart		Valve Tag
	Insert Border		Line Number Tag
	Dimension Text Utilities		Branch Fitting Query
	Rolling Offset Dimensioning		Data File Names
	Offset Dimensioning		Edit Balloon Text
	ISO Dimensioning		Weld Symbol

Change Tools

			
	User Defined Grid		Change Lower Case Text to Upper Case
	User Snap Settings		Change Upper Case Text to Lower Case
	Change Layer to Entity's		Replace Text
	Move Layer Contents to New Layer		Shop or Field Destination-Change Globally
	Change Item to Selected Layer		Line Number-Change Globally
	Change Items to Current Layer		Drawing Number-Change Globally
	Dynamic Attribute Edit		

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