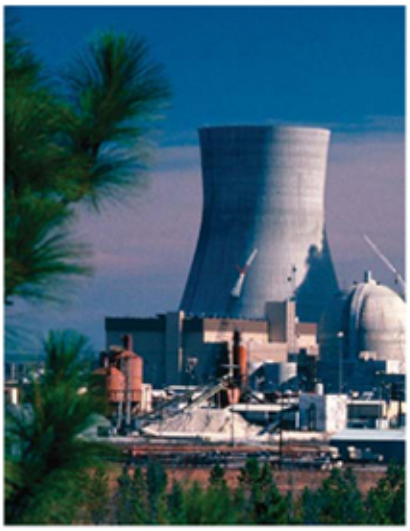
Hangers and Supports
User's Guide

Process, Power & Marine


INTERGRAPH

Version 2007 Service Pack 2 January 2007 / June 2007 DSP3D-PE-200015G - UPDATED

Copyright

Restricted Rights Legend

Intergraph Corporation
Huntsville, Alabama 35894-0001

Warranties and Liabilities

Trademarks

Table of Contents

Preface

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

SmartPlant 3D Documentation Set

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

The documentation set is divided into four categories:

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

Administrative Guides

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

SmartPlant 3D Database Integrity Guide - Provides information about the error messages in the database integrity reports, including meaning, cause, and possible corrective action.

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

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

SmartPlant 3D Interference Checking Guide - Provides information on installing, configuring, and using the interference detection service.

SmartPlant 3D Plant Design System (PDS) Guide - Provides all information needed to use PDS with SmartPlant 3D. Topics include referencing active PDS projects in SmartPlant 3D, exporting PDS data and importing that data into SmartPlant 3D, converting PDS reference data to SmartPlant 3D reference data, and converting EDEN symbols to Visual Basic symbols.

SmartPlant 3D Release Bulletin - Provides what's new, hardware/software requirements, and support information for the current release.

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

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

User's Guides

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Reference Data Guides

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

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

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

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

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

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

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

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

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

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

ISOGEN Guides

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

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

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

Documentation Comments

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

What's New in Hangers and Supports

The following changes have been made to the Hangers and Supports task.

Version 2007

- The **OffsetY** column has been added to **HS_System.xls**. This provides a way to define a rule or value for Y-direction (perpendicular to face) object placement. For more information, refer to *Face Position Selection Sheet* topic.
- Added information to the **Claim Command** topic**.**The software automatically claims children objects when you claim the parent object. For example, control points are claimed when you claim the parent object. Other examples include the automatic claiming of support components when you claim a support, and assembly connections when you claim a slab.
- You can define property restraint values for hangers. To view restraints values, select the hanger, right-click and select **Properties**. Click the **Definition** tab and select **Restraints** on the **Category** drop-down list.
- The **Definition Tab** is included on the **Support Properties** dialog box. This displays the support component properties as they are defined in the reference data.
- You can define loading property values for hangers using the **Support Properties** dialog box. Values can be defined for loads and pipe restraint stiffness.
- You can define pipe motion restraint direction using the **Support Properties** dialog box. Catalog supports automatically define any motion restraints. User-defined supports require the user to define the restraint support direction.
- A **Fabrication Requirement** property is now available for all supports and all components. You can select the fabrication requirement for the support component. If you want to add, edit, or remove values that are available for selection, edit the **Fabrication Type** sheet in the AllCodeLists.xls workbook. The property is found on the **General** tab of the **Support Properties** dialog box.
- A **Reporting Requirement and Reporting Type** property is now available for all supports. The property specifies whether the support object is included in reports. By default, this property is set to Undefined. The property is found on the **General** tab of the **Support Properties** dialog box.

- A **BOM Description** property is now available for designed supports. The property displays the Bill of Materials (BOM) description of the part. The description includes the size, finish, and length (where applicable). The default values are defined by the catalog and are read-only for standard supports. The values can be edited for designed supports.
- A new drawing template package is now available for use with drawing queries. It is delivered to *<product directory>\symbols\Drawings\Catalog\Packages\HngSup3View*. The package contains an orthographic Drawings by Query (DBQ) component and is available in the Drawings and Reports task.
- Added the ability to instance simple physical support geometry when a PDS physical support is translated into SP3D.
- Improved connecting standard supports HgrBeam steel.
- Hangers and Supports symbols and rules now support CAB files.

Hangers and Supports: An Overview

The primary purpose of hangers and supports is to support various types of distributive systems such as pipes, HVAC ducts, and cable trays. Usually, supports are connected to the supported object, such as a pipe, and to a supporting object, such as a beam. However, you can also place supports between two or more supported objects, such as placing a support between two pipes.

Notes

- Throughout this document, the word "support" is used as a generic way to refer to both supports and hangers.
- Throughout this document, the word "feature" is used as a generic way to refer to pipes, HVAC ducts, and cable trays.

The Hangers and Supports task treats supports as first class modeling components. The supports are actually connected to piping and structural systems instead of simple graphical references. Connections enable the supports to automatically react to design changes. For example, if the size of a pipe changes, the pipe supports automatically react. Additionally, popular support catalogs are included with the software. You can also model designed supports.

The Hangers and Supports task has these commands:

	**Select** - Used to select objects in the model. For more information, see *Editing Supports: An Overview*, page 45.
	**Place Support by Structure** - Places a support at the intersection of a supporting object and the supported object. For more information, see *Place Support by Structure Command*, page 16.
	**Place Support by Point** - Places a support at a location that you specify. For more information, see *Place Support by Point Command*, page 26.
	**Place Part** - Adds or modifies parts in a designed support. For more information, see *Place Part Command*, page 38.
	**Drop Standard** - Converts a standard support to a designed support. For more information, see *Drop Standard Command*, page 44.
	**Place Linear Member** - Places columns, beams, braces, and other linear members in the model. For more information, see *Place Linear Member System Command*, page 62.
	**Place Assembly Connection** - Places assembly connections between linear member systems. For more information, see *Place Assembly Connection Command*, page 92.

Related Topics

- *Hangers and Supports Common Tasks*, page 15

Understanding the Hangers and Supports Workflow: An Overview

Hangers and supports are placed in the model using information defined in the hangers and supports reference data. Using the reference data workbook, you can create parts, define assemblies, and define placement rules. Your first step is to review, edit, and otherwise customize the delivered hangers and supports reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

After the reference data is customized to suit your needs, go to the Systems and Specifications task and define the systems that you want in your model. While you are not required to create your systems first, doing so keeps you from having to edit your hangers and supports after placement to assign them to the correct system.

Because hangers and supports are dependent on both the supporting object and the supported object, you need to have those objects placed in the model before placing any hangers or supports. Examples of supporting objects include beams, plates, and slabs. Examples of supported objects include piping, HVAC, conduit, and cable trays. These objects are placed using the Piping, HVAC, and Electrical tasks.

Related Topics

- *Design a Support by Point*, page 42
- *Design a Support from a Structure*, page 41
- *Hangers and Supports: An Overview*, page 13
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34

Hangers and Supports Common Tasks

The following tasks are used frequently in the Hangers and Supports task.

Customize Reference Data

Create new hangers and supports by editing the applicable workbooks. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Placing Supports

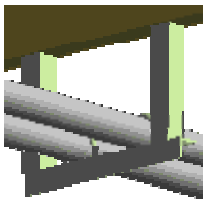
Place a hanger from a specific structure. For more information, see *Place a Support from a Structure*, page 23.

Place a hanger or support at a specific location. For more information, see *Place a Support at a Specific Location*, page 33.

Place multiple hangers or supports along a run. For more information, see *Place Multiple Supports along a Feature*, page 34.

Place Support by Structure Command

Places a support at the intersection of a selected feature and a selected structure. The software automatically determines the intersection point for you. Use this command when you want to support a feature that is not parallel to the supporting structure.



An example of when to use this command.

Related Topics

- *Hangers and Supports: An Overview*, page 13
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34

Support Placement Ribbon

Sets options for placing a support in the model.

Properties - Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box*, page 46.

Feature - Select the features for which you want to place a support. You must select at least one feature. Click **Reject** to clear all selected features and start over selecting features. Click **Accept** when you are finished selecting features.

If you are placing the support using the **Place Support by Point** command, the first feature that you select is the primary feature. The system uses the cross section from the primary feature to define the support plane. If you deselect the primary feature, the second feature that you select becomes the new primary feature.

Structure - Select the structure to support the feature you selected. Click **Reject** to clear all selected structures and start over selecting supporting structures. Click **Accept** when you are finished selecting supporting structures.

Position - Specify the location of the support on the primary feature. This option is available only when using the **Place Support by Point** command.

Finish - Places the support using the information that you provided.

Reject - Clears all selected objects and restarts the selection process.

Accept - Indicates to the software that you are finished selecting objects.

System - Select the parent system for the support that you are placing. You can create systems in the Systems and Specifications task.

Rule - Select if you want the system to display only those support types allowed by the rules defined in the reference data. When this option is selected, the software uses an Assembly Selection Rule to filter available supports in the **Type** box based on things like the pipe outside diameter, distance between the supported object and the supporting object, and whether the route is above or below the supporting object. You, or your project administrator, have complete control over how the rules are defined and what they check for. See the *Hangers and Supports Reference Data Guide* for more information. Clear the **Rule** option if you want to select from all available support types.

Type - Select the kind of support that you want to place. If you select the **Rule** option, the only types listed are those that are allowed by the rules defined in the reference data. If you do not select the **Rule** option, the last ten types that you placed appear. Select **More** to select a different type of support.

Tip

- When you place a designed support, the **Type** box is not available.

Toggle Connection to Feature - Cycles through the available attachment types based on the feature and the support type. This option may not be available if the type that you selected does not provide for multiple attachment types.

Toggle Connection to Structure - Cycles through the available attachment types based on the supporting structure and the support type. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option may not be available if the type you selected does not provide for multiple attachment types.

Toggle Face Position - Cycles through the available face positions for the supporting structure. For example, you can use this option to move a U-bolt so that it does not interfere with the web of a beam. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option may not be available if there are no optional face positions.

Copies - Type the number of supports to place at the currently defined offset. You must use the **Tools > Point Along** command to use this option. This option is available only when using the **Place Support by Point** command.

Design - Places the support as a designed support. You must check this box before selecting any supported features because this box is not available after that step.

If you select the **Design** option, the **Rule** box becomes unavailable. Likewise, if you deselect this option (thereby placing a standard support), the **Rule** box is available.

Note

- The **Design** option is only available at initial placement.

Related Topics

- *Drop Standard Command*, page 44
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26

Select Support Dialog Box

Allows selection of the type of support to be placed. This dialog box appears automatically when you select the **More...** option in the **Type** box on the ribbon. By browsing through the part hierarchy, you can find any support in the Catalog database. After you select a support, the software returns you to the model, where you can finalize placement.

Back - Returns you to the previously selected support type or node. Use this command to navigate through the support hierarchy to the specific type you need.

Forward - Sends you to the last selected support type or node that you moved away from by using the **Back** button. Use this command to navigate through the support hierarchy to the specific type you need.

Up One Level - Brings up the next highest level of the support catalog hierarchy. Use this command to navigate through the support hierarchy to the specific type you need.

Copy - Copies the selected object. This command is available only in the Catalog task.

Paste - Pastes a copied object. This command is available only in the Catalog task.

Delete - Deletes the selected object. This command is available only in the Catalog task.

Undo - Reverses the most recent operation. This command is available only in the Catalog task.

New Object - Creates a new object. This command is available only in the Catalog task.

Move Up - Moves up one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Move Down - Moves down one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Properties - Displays the properties of the selected support. Because you cannot modify any properties until the support is placed, all properties on the dialog box are read-only.

Preview - Displays a picture of the selected support. The image file must be assigned to the support in the reference data.

List View - Sets the dialog box to display supports in a list view.

Grid View - Sets the dialog box to display supports in a spreadsheet-style grid view.

Check Data - Checks the consistency of the data in the grid against other data in the Catalog. This command is available only in the Catalog task.

Address - Specifies your exact location within the displayed hierarchy.

Related Topics

- *Support Multiple Modify Ribbon*, page 21
- *Support Placement Ribbon*, page 16
- *Support Single Modify Ribbon*, page 20

Select System Dialog Box

This dialog box displays all of the defined systems so that you can select the system that you want. You can create new systems in the Systems and Specifications task.

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

Support Single Modify Ribbon

Displays options for the single support that you have selected to edit.

Properties - Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box*, page 46.

Feature - Select the features for which you want to place a support. You must select at least one feature. Click **Reject** to clear all selected features and start over selecting features. Click **Accept** when you are finished selecting features.

Structure - Select the structure that you want to use to support the feature that you selected. Click **Reject** to clear all selected structures and start over selecting supporting structures. Click **Accept** when you are finished selecting supporting structures.

Position - Specify the location of the support on the primary feature. This option is available only when using the **Place Support by Point** command.

Finish - Edits the support using the information that you provided.

Reject - Clears all selected objects, and restarts the selection process.

Accept - Indicates to the software that you are finished selecting objects.

System - Select the parent system for the support that you are placing.

Name - Displays the name of the selected support.

Rule - Select if you want the system to display only those support types allowed by the rules defined in the reference data. You, or your project administrator, have complete control over how the rules are defined in the reference data. Clear the **Rule** option if you want to select from all available support types.

Type - Select a new kind of support from the list. If you select the **Rule** option, the only types listed are those that are allowed by the rules defined in the reference data. If you do not select the **Rule** option, the last ten types that you placed appear. Select **More** to select a different type of support.

Note

- If the selected support is a designed support, the **Rule** and **Type** boxes are not available.

Toggle Connection to Feature - Cycles through the available attachment types based on the feature and the support type. This option may not be available if the type that you selected does not provide for multiple attachment types.

Toggle Connection to Structure - Cycles through the available attachment types based on the supporting structure and the support type. This option may not be available if the type that you selected does not provide for multiple attachment types.

Toggle Face Position - Cycles through the available face positions for the supporting structure. For example, you can use this option to move a U-bolt so that it does not interfere with the web of a beam. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option may not be available if there are no optional face positions.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Drop Standard Command*, page 44
- *Edit Support Name*, page 24
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26
- *Place Support by Structure Command*, page 16

Support Multiple Modify Ribbon

Displays options for the set of supports that you have selected to edit.

Properties - Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box*, page 46.

System - Displays the name of the parent system to which the supports belong.

Type - Select a new kind of support from the list.

Note

- If the selected supports are designed supports, the **Type** box is not available.

Spacing - Type the distance between adjacent supports on the same feature. The software locks the first support that you select in place and uses it as the reference point to calculate the position of the other selected supports.

Notes

- The **Spacing** option is available only when you are using the **Place Support by Point** command.
- When you select multiple supports, the software highlights the first selected support in cyan. You can easily see which support is the origin of the spacing as you space multiple supports along a feature.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Drop Standard Command*, page 44
- *Edit Support Name*, page 24
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26
- *Place Support by Structure Command*, page 16

Place a Support from a Structure

1. Click **Place Support by Structure** on the vertical toolbar.

   Tip

   - If you want to create a designed support, select **Design** on the ribbon. Otherwise, deselect it.
2. Select the feature to support.
3. Click **Accept**.
4. Select the supporting structure to use.
5. Click **Accept**.
6. Select the support type.
7. Click **Finish**.

Notes

- The software limits the choice of feature and structure connections to those available for the support type. This information is defined in the reference data.
- You can select more than one feature to support.
- You can select more than one structure from which to support the selected features.

Related Topics

- *Hangers and Supports: An Overview*, page 13
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34

Modify Support Load

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the support or hanger to edit.
4. Click **Edit > Properties**.
5. Type a new load value in the **Maximum Load** box.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Editing Supports: An Overview*, page 45

Modify Support Type

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the hanger or support to edit.
4. On the ribbon, select a new type from the list of available types.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Editing Supports: An Overview*, page 45

Edit Support Name

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the hanger or support to edit.
4. Click **Edit > Properties**.
5. Select the **General** tab.
6. Type a new name for the support.

Note

- You can also edit the name in the **Name** box on the ribbon for a support.

Related Topics

- *Delete Support*, page 25
- *Editing Supports: An Overview*, page 45

Change Structure Connection

1. Click **Select** on the vertical toolbar.
2. Select **Support** on the **Locate Filter** box.
3. Select the support or hanger to edit.
4. Click **Toggle Connection to Structure** on the ribbon until the connection that you want to use appears.

Note

- The software limits the available connections based on the supporting structure and the type of support.

Related Topics

- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45

Change Feature Connection

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the support or hanger to edit.
4. Click **Toggle Connection to Feature** on the ribbon until the connection to use appears.

Note

- The software limits the available connections based on the feature and the type of support.

Related Topics

- *Delete Support*, page 25
- *Editing Supports: An Overview*, page 45

Delete Support

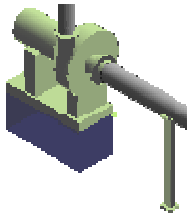
1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the support or hanger to delete.
4. Click **Delete**.

Related Topics

- *Delete Support*, page 25
- *Editing Supports: An Overview*, page 45

Place Support by Point Command

Places a support for a selected feature and a selected structure at a location that you specify. Use this command when the selected feature and the selected structure are parallel or when you are placing a support on a surface such as a floor. You can place a single support at a location that you specify, or you can place multiple supports at a given distance apart.



An example of when to use this command.

Tip

- This command also allows you to place a support without selecting a structural reference.

Related Topics

- *Drop Standard Command*, page 44
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26
- *Place Support by Structure Command*, page 16

Support Placement Ribbon

Sets options for placing a support in the model.

Properties - Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box*, page 46.

Feature - Select the features for which you want to place a support. You must select at least one feature. Click **Reject** to clear all selected features and start over selecting features. Click **Accept** when you are finished selecting features.

If you are placing the support using the **Place Support by Point** command, the first feature that you select is the primary feature. The system uses the cross section from the primary feature to define the support plane. If you deselect the primary feature, the second feature that you select becomes the new primary feature.

Structure - Select the structure to support the feature you selected. Click **Reject** to clear all selected structures and start over selecting supporting structures. Click **Accept** when you are finished selecting supporting structures.

Position - Specify the location of the support on the primary feature. This option is available only when using the **Place Support by Point** command.

Finish - Places the support using the information that you provided.

Reject - Clears all selected objects and restarts the selection process.

Accept - Indicates to the software that you are finished selecting objects.

System - Select the parent system for the support that you are placing. You can create systems in the Systems and Specifications task.

Rule - Select if you want the system to display only those support types allowed by the rules defined in the reference data. When this option is selected, the software uses an Assembly Selection Rule to filter available supports in the **Type** box based on things like the pipe outside diameter, distance between the supported object and the supporting object, and whether the route is above or below the supporting object. You, or your project administrator, have complete control over how the rules are defined and what they check for. See the *Hangers and Supports Reference Data Guide* for more information. Clear the **Rule** option if you want to select from all available support types.

Type - Select the kind of support that you want to place. If you select the **Rule** option, the only types listed are those that are allowed by the rules defined in the reference data. If you do not select the **Rule** option, the last ten types that you placed appear. Select **More** to select a different type of support.

Tip

- When you place a designed support, the **Type** box is not available.

Toggle Connection to Feature - Cycles through the available attachment types based on the feature and the support type. This option may not be available if the type that you selected does not provide for multiple attachment types.

Toggle Connection to Structure - Cycles through the available attachment types based on the supporting structure and the support type. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option may not be available if the type you selected does not provide for multiple attachment types.

Toggle Face Position - Cycles through the available face positions for the supporting structure. For example, you can use this option to move a U-bolt so that it does not interfere with the web of a beam. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option may not be available if there are no optional face positions.

Copies - Type the number of supports to place at the currently defined offset. You must use the **Tools > Point Along** command to use this option. This option is available only when using the **Place Support by Point** command.

Design - Places the support as a designed support. You must check this box before selecting any supported features because this box is not available after that step.

If you select the **Design** option, the **Rule** box becomes unavailable. Likewise, if you deselect this option (thereby placing a standard support), the **Rule** box is available.

Note

- The **Design** option is only available at initial placement.

Related Topics

- *Drop Standard Command*, page 44
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26
- *Place Support by Structure Command*, page 16

Select Support Dialog Box

Allows selection of the type of support to be placed. This dialog box appears automatically when you select the **More...** option in the **Type** box on the ribbon. By browsing through the part hierarchy, you can find any support in the Catalog database. After you select a support, the software returns you to the model, where you can finalize placement.

Back - Returns you to the previously selected support type or node. Use this command to navigate through the support hierarchy to the specific type you need.

Forward - Sends you to the last selected support type or node that you moved away from by using the **Back** button. Use this command to navigate through the support hierarchy to the specific type you need.

Up One Level - Brings up the next highest level of the support catalog hierarchy. Use this command to navigate through the support hierarchy to the specific type you need.

Copy - Copies the selected object. This command is available only in the Catalog task.

Paste - Pastes a copied object. This command is available only in the Catalog task.

Delete - Deletes the selected object. This command is available only in the Catalog task.

Undo - Reverses the most recent operation. This command is available only in the Catalog task.

New Object - Creates a new object. This command is available only in the Catalog task.

Move Up - Moves up one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Move Down - Moves down one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Properties - Displays the properties of the selected support. Because you cannot modify any properties until the support is placed, all properties on the dialog box are read-only.

Preview - Displays a picture of the selected support. The image file must be assigned to the support in the reference data.

List View - Sets the dialog box to display supports in a list view.

Grid View - Sets the dialog box to display supports in a spreadsheet-style grid view.

Check Data - Checks the consistency of the data in the grid against other data in the Catalog. This command is available only in the Catalog task.

Address - Specifies your exact location within the displayed hierarchy.

Related Topics

- *Support Multiple Modify Ribbon*, page 21
- *Support Placement Ribbon*, page 16
- *Support Single Modify Ribbon*, page 20

Select System Dialog Box

This dialog box displays all of the defined systems so that you can select the system that you want. You can create new systems in the Systems and Specifications task.

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

Support Single Modify Ribbon

Displays options for the single support that you have selected to edit.

Properties - Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box*, page 46.

Feature - Select the features for which you want to place a support. You must select at least one feature. Click **Reject** to clear all selected features and start over selecting features. Click **Accept** when you are finished selecting features.

Structure - Select the structure that you want to use to support the feature that you selected. Click **Reject** to clear all selected structures and start over selecting supporting structures. Click **Accept** when you are finished selecting supporting structures.

Position - Specify the location of the support on the primary feature. This option is available only when using the **Place Support by Point** command.

Finish - Edits the support using the information that you provided.

Reject - Clears all selected objects, and restarts the selection process.

Accept - Indicates to the software that you are finished selecting objects.

System - Select the parent system for the support that you are placing.

Name - Displays the name of the selected support.

Rule - Select if you want the system to display only those support types allowed by the rules defined in the reference data. You, or your project administrator, have complete control over how the rules are defined in the reference data. Clear the **Rule** option if you want to select from all available support types.

Type - Select a new kind of support from the list. If you select the **Rule** option, the only types listed are those that are allowed by the rules defined in the reference data. If you do not select the **Rule** option, the last ten types that you placed appear. Select **More** to select a different type of support.

Note

- If the selected support is a designed support, the **Rule** and **Type** boxes are not available.

Toggle Connection to Feature - Cycles through the available attachment types based on the feature and the support type. This option may not be available if the type that you selected does not provide for multiple attachment types.

Toggle Connection to Structure - Cycles through the available attachment types based on the supporting structure and the support type. This option may not be available if the type that you selected does not provide for multiple attachment types.

Toggle Face Position - Cycles through the available face positions for the supporting structure. For example, you can use this option to move a U-bolt so that it does not interfere with the web of a beam. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option may not be available if there are no optional face positions.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Drop Standard Command*, page 44
- *Edit Support Name*, page 24
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26
- *Place Support by Structure Command*, page 16

Support Multiple Modify Ribbon

Displays options for the set of supports that you have selected to edit.

Properties - Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box*, page 46.

System - Displays the name of the parent system to which the supports belong.

Type - Select a new kind of support from the list.

Note

- If the selected supports are designed supports, the **Type** box is not available.

Spacing - Type the distance between adjacent supports on the same feature. The software locks the first support that you select in place and uses it as the reference point to calculate the position of the other selected supports.

Notes

- The **Spacing** option is available only when you are using the **Place Support by Point** command.
- When you select multiple supports, the software highlights the first selected support in cyan. You can easily see which support is the origin of the spacing as you space multiple supports along a feature.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Drop Standard Command*, page 44
- *Edit Support Name*, page 24
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24
- *Place Part Command*, page 38
- *Place Support by Point Command*, page 26
- *Place Support by Structure Command*, page 16

Place a Support at a Specific Location

1. Click **Place Support by Point** on the vertical toolbar.

   Tip

   - If you want to create a designed support, select **Design** on the ribbon. Otherwise, deselect it.
2. Select the feature to support.
3. Click **Accept**.
4. Select the structure to use to support the selected feature.

   Tip

   - This step is optional. For example, if you are placing a support between two features only, then you do not need to select a supporting structure.
5. Click **Accept**. The allowed support types appear.
6. Select the support type.
7. Identify the location along the feature to place the support.

   Tip

   - You can use the **Tools > Point Along** command to identify this point. If you use the **Tools > Point Along** command when identifying the point, you must activate the **Point Along** command before selecting the **Place Support by Point** command.
8. Click **Finish**.

Notes

- The software limits the choice of feature and structure connections to those available for the support type. This information is defined in the reference data.
- You can select more than one feature to support.
- You can select more than one structure from which to support the selected features.

Related Topics

- *Hangers and Supports: An Overview*, page 13
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34

Place Multiple Supports along a Feature

1. Click **Tools** > **Point Along**.
2. Click **Reference** on the ribbon, and then select the feature along which you are planning to place supports.
3. Identify the point to measure from on the feature.
4. Click **Place Support by Point** on the vertical toolbar.
5. Select the feature to support.
6. Click **Accept**.
7. Select the structure to use to support the selected feature.
8. Click **Accept**.
9. In the **Distance** box on the **Point Along** ribbon, specify the distance from the measure-from point to the first support. You identified the measure-from point in step 2.
10. Click in the graphic view to position the first support.
11. In the **Copies** box, type the number of supports to place.
12. Select the support type.
13. Click **Finish**.

Notes

- The software limits the choice of feature and structure connections to those available for the support type. This information is defined in the reference data.
- You can select more than one feature to support.
- You can select more than one structure from which to support the selected features.
- The software optionally can limit the support types that are available by defining rules in the reference data.

Related Topics

- *Hangers and Supports: An Overview*, page 13
- *Place a Support at a Specific Location*, page 33
- *Place a Support from a Structure*, page 23
- *Place Multiple Supports along a Feature*, page 34

Modify Support Load

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the support or hanger to edit.
4. Click **Edit > Properties**.
5. Type a new load value in the **Maximum Load** box.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Modify Support Type

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the hanger or support to edit.
4. On the ribbon, select a new type from the list of available types.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Edit Support Name

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the hanger or support to edit.
4. Click **Edit > Properties**.
5. Select the **General** tab.
6. Type a new name for the support.

Note

- You can also edit the name in the **Name** box on the ribbon for a support.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Change Structure Connection

1. Click **Select** on the vertical toolbar.
2. Select **Support** on the **Locate Filter** box.
3. Select the support or hanger to edit.
4. Click **Toggle Connection to Structure** on the ribbon until the connection that you want to use appears.

Note

- The software limits the available connections based on the supporting structure and the type of support.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Change Feature Connection

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the support or hanger to edit.
4. Click **Toggle Connection to Feature** on the ribbon until the connection to use appears.

Note

- The software limits the available connections based on the feature and the type of support.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Delete Support

1. Click **Select** on the vertical toolbar.
2. Select **Support** in the **Locate Filter** box.
3. Select the support or hanger to delete.
4. Click **Delete**.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Editing Supports: An Overview*, page 45
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Place Part Command

Adds parts to a support. You can also re-size or replace parts within the support using this command.

This command is useful when you design a support. You can first place a standard support and then, using the **Place Part** command, "build it up" to customize it for your particular project. The software checks port compatibilities and sizes as you place the parts.

Note

The **Place Part** command can locate the ports that connect support components best with certain SmartSketch locate items turned off. You should turn off all SmartSketch check marks except for **Key Point** and **Reference Axis Aligned**. If you do not turn off all other SmartSketch options, the software locates other objects before it finds the component ports and this causes difficulties when the software builds the designed support.

Related Topics

- *Design a Support by Point*, page 42
- *Design a Support from a Structure*, page 41
- *Hangers and Supports: An Overview*, page 13

Place Part Ribbon

Allows you to add or modify parts in a support.

Properties - Opens the **Properties** dialog box for the part.

Select Part - Displays the **Select Part** dialog box. You can select a new part or part class to replace the active part.

Stretch - Extends a variable length part. This option appears only during modification (not placement). In addition, this option is available only if the active part can be extended.

Toggle Port - Toggles the port attachment of the part. This option is available only during placement (not modification).

Tip

- You can also press the Up and Down arrow keys or the T key to toggle the port attachment.

Name - Displays the part number from the reference data. You can edit this field.

Support Name - Displays the name of the designed support. This name also appears in the **Workspace Explorer** on the **System** tab.

Type - Displays the kind of part. After placement, this field displays the part number.

Related Topics

- *Design a Support by Point*, page 42
- *Design a Support from a Structure*, page 41
- *Place Part Command*, page 38

Select Part Dialog Box

Allows selection of the part to be placed. This dialog box appears when you select a support in the model. By browsing through the part hierarchy, you can find any part in the hangers and supports parts catalog. After you select a part, the software returns you to the model, where you can finalize placement.

Back - Returns you to the previously selected location. Use this command to navigate through the hierarchy to the specific part you need.

Forward - Sends you to the last selected location that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific part you need.

Up One Level - Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific part you need.

Copy - Copies the selected object. This command is available only in the Catalog task.

Paste - Pastes a copied object. This command is available only in the Catalog task.

Delete - Deletes the selected object. This command is available only in the Catalog task.

Undo - Reverses the most recent operation. This command is available only in the Catalog task.

New Object - Creates a new object. This command is available only in the Catalog task.

Move Up - Moves up one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Move Down - Moves down one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Properties - Displays the properties of the selected part. Because you cannot modify any properties until the part is placed, all properties on the dialog box are read-only.

Preview - Displays a picture of the selected support. The image file must be assigned to the part in the reference data.

List View - Sets the dialog box to display the parts in a list view.

Grid View - Sets the dialog box to display the parts in a spreadsheet-style grid view.

Check Data - Checks the consistency of the data in the grid against other data in the Catalog. This command is available only in the Catalog task.

Address - Specifies your exact location within the displayed hierarchy.

Related Topics

- *Support Multiple Modify Ribbon*, page 21
- *Support Placement Ribbon*, page 16
- *Support Single Modify Ribbon*, page 20

Design a Support from a Structure

1. Click **Place Support by Structure** on the vertical toolbar.
2. Select the **Design** box on the ribbon.
3. Select the feature to support.
4. Click **Accept**.
5. Select the supporting structure to use.
6. Click **Accept**.
7. Click **Finish**.
8. Click **Place Part** on the vertical toolbar.
9. Select the support that you just placed.
10. Select a support part from the catalog.
11. Place the part in the model.

    Tip

    - If the connection is successful, the software flashes the support in green. If the part is placed but the ports are not connected, the software highlights the support in yellow.
12. Continue to select the **Place Part** command until you have finished designing the support.

Notes

- While the part is on the end of your pointer, you can use the **Toggle Port** option to toggle the port attachment. Another way to toggle the port is by pressing the Up or Down arrow keys or the T key.
- You can use the **Stretch** option to adjust the length of the part, if the part can be of variable length.

Related Topics

- *Design a Support by Point*, page 42
- *Design a Support from a Structure*, page 41
- *Hangers and Supports: An Overview*, page 13

Design a Support by Point

1. Click **Place Support by Point** on the vertical toolbar.
2. Select the **Design** box on the ribbon.
3. Select the feature to support.
4. Click **Accept**.
5. Select the structure to use to support the selected feature.
6. Click **Accept**.
7. Identify the location along the feature to place the support. You can use the **Tools > Point Along** command when identifying this point.
8. Click **Finish**.
9. Click **Place Part** on the vertical toolbar.
10. Select the support that you just placed.
11. Select a support part from the catalog.
12. Place the part in the model.

Tip

- If the connection is successful, the software flashes the support in green. If the part is placed but the ports are not connected, the software highlights the support in yellow.

13. Continue to select the **Place Part** command until you have finished designing the support.

Notes

- While the part is on the end of your pointer, you can use the **Toggle Port** option to toggle the port attachment. Another way to toggle the port is by pressing the Up or Down arrow keys or the T key.
- You can use the **Stretch** option to adjust the length of the part, if the part can be of variable length.

Related Topics

- *Design a Support from a Structure*, page 41
- *Hangers and Supports: An Overview*, page 13

Dropping Standard Supports: An Overview

In some circumstances, you may want to modify standard supports delivered in the catalog in order to better suit the requirements of your particular support situation.

Before modifying a standard support, you must convert the support to a designed support to break the support into its constituent components. You can use the **Drop Standard** command for this purpose.

If the support is already within a designed support and you click this command, the software breaks it into its components. If the support is not within a designed support and you click this command, then the software converts it to a designed support and breaks it into its components.

Notes

- The software automatically converts a standard support to a designed support when you add a part to a standard support.
- In the **Workspace Explorer**, the icon for a designed support is . There are a few restrictions on a support hierarchy. A standard support cannot be the parent of a designed support, and a designed support cannot be the parent of another designed support.

Related Topic

- *Hangers and Supports: An Overview*, page 13

Drop Standard Command

Converts a standard support to a designed support and breaks the support into its constituent parts. You use this command to modify or delete individual parts of a standard support.

Note

- Associativity with the supported object is lost as a result of this command.

Related Topic

- *Hangers and Supports: An Overview*, page 13

Drop a Standard Support

1. Click **Drop Standard** on the vertical toolbar.
2. Select a standard support. The software converts the selected support to a designed support and breaks it into its parts.

Notes

- You can select more than one standard support to convert.
- Associativity with the supported object is lost as a result of this command.

Related Topics

- *Drop a Standard Support*, page 44
- *Hangers and Supports: An Overview*, page 13

Editing Supports: An Overview

All objects in the Hangers and Supports task have properties that you can edit. Using the **Select** command on the vertical toolbar, select the object that you want to edit.

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

The Hangers and Supports task includes these filters:

Support - Allows you to select supports in a graphic view or in the **Workspace Explorer**.

Support Component - Limits your selection in a graphic view or in the **Workspace Explorer** to the individual parts of a support system.

Support Connection - Limits your selection in a graphic view or in the **Workspace Explorer** to a support connection.

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

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

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

Note

- When you select multiple supports, the software highlights the first selected support in cyan. You can easily see which support is the origin of the spacing as you space multiple supports along a feature.

Related Topics

- *Change Feature Connection*, page 25
- *Change Structure Connection*, page 25
- *Delete Support*, page 25
- *Edit Support Name*, page 24
- *Modify Support Load*, page 23
- *Modify Support Type*, page 24

Support Properties Dialog Box

Specifies properties for the support that you have selected.

Related Topics
- *Configuration Tab*, page 49
- *Definition Tab*, page 49
- *General Tab (Support Properties Dialog Box)*, page 46
- *Notes Tab*, page 50
- *Relationship Tab*, page 49

General Tab (Support Properties Dialog Box)

Displays the support properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you select more than one support and then select the **Properties** command, only the common properties among the selected supports display.

When viewing properties for a single support, the following properties display. More properties may display depending on what you defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Category - Select the properties that you want to view for the support. Support properties are divided into several different categories: **Standard**, **Weight and CG**, and **Responsibility**. You select the category for which to define values by using the **Category** option.

Note

- Support and part properties can appear in any category. The custom interfaces in the reference data control how the properties are grouped into categories.

Type - Displays the type of support. You can select a different support type if you want.

Show Dimensional Legend - Click this button to view a picture of the support with dimensions labeled.

Standard

System - Displays the parent system of the selected support.

Type Selection Rule - Displays how this support was selected. If **True**, the software selected the support. If **False**, the user who placed the support selected the support from the catalog.

Name - Displays the unique name of the selected support. If you type a new name, the **Name Rule** value changes to **User Defined**. Otherwise, the name is defined using the specified naming rule.

Name Rule - Defines how the support was named. If you specified the support name, this box displays **User Defined**. If the software named the support, this box displays the name of the naming rule used to name the support.

Primary Run - Displays the name of the feature that the support is supporting.

Maximum Load - Specifies the maximum load that the support must carry.

BOM Description - Displays the Bill of Materials description of the part. The description includes the size, finish, and length (where applicable). The default values are defined by the catalog and are read-only for standard supports. The values can be edited for designed supports.

Reporting Requirement - Specifies whether the support object is included in reports. By default, this property is set to **Undefined**.

Reporting Type - Specifies how the objects are reported, depending on the **Reporting Requirement** setting. By default, this property is set to **Undefined**.

Support Discipline - Displays the discipline of the supported object, such as piping, duct, or another discipline. If you want to add, edit, or remove values that are available for selection, edit the **HngSupSupportType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Support Type - Specifies the general class to which the support belongs. If you want to add, edit, or remove values that are available for selection, edit the **HngSupSupportType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Weight and CG

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

Dry Weight - Displays the dry weight of the support.

Wet Weight - Displays the wet weight of the support.

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

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

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

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

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

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

Responsibility

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

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

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

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

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

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

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

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

Related Topics

- *Support Properties Dialog Box*, page 46

Definition Tab

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

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

Relationship Tab

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

Name - Displays the name of the related object.

Type - Displays the type of related object.

Go To - Displays the properties of the selected object.

Configuration Tab

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

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

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

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

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

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

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

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

Notes Tab

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

Note

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

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

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

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

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

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

Purpose of note - Specifies the purpose of the note.

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

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

New Note - Creates a new note on the object.

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

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

Delete Note - Deletes the currently displayed note.

Support Component Properties Dialog Box

Sets properties for support parts that you have placed in the model.

Related Topics

- *Configuration Tab*, page 49
- *Connections Tab (Support Component Properties Dialog Box)*, page 54
- *Occurrence Tab (Support Component Properties Dialog Box)*, page 51
- *Relationship Tab*, page 49

Occurrence Tab (Support Component Properties Dialog Box)

Displays the support component properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one support component, and then selected the **Properties** command, only the common properties among the selected support components display.

When viewing properties for a single support component, the following properties display. More properties may display depending on what you defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide*.

Category - Select the properties that you want to view for the support component. Support component properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, and **Responsibility**. You select the category for which to define values by using the **Category** option.

Note

- Support and part properties can appear in any category. The custom interfaces in the reference data control how the properties are grouped into categories.

Show Dimensional Legend - Click this button to view a picture of the component with dimensions labeled.

Standard

Name - Displays the unique name of the selected support component. If you type a new name, the **Name Rule** value changes to **User Defined**. Otherwise, the name is defined using the specified naming rule.

Name Rule - Defines how the support component was named. If you specified the support name, this box displays **User Defined**. If the software named the support, this box displays the name of the naming rule used to name the support.

Support Name - Displays the name of the parent support.

BOM Description - Displays the Bill of Materials description of the part. The description includes the size, finish, and length (where applicable). The default values are defined by the catalog and are read-only for standard supports. The values can be edited for designed supports.

Type - Displays the type of support component. The type displayed is the part number defined in the reference data.

Reporting Requirement - Specify whether or not this support component is reported.

Reporting Type - Select the reporting requirements code for the support component. Valid codes are defined in the **AllCodeLists.xls** workbook on the **Reporting Type** sheet.

Weight and CG

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

Dry Weight - Displays the dry weight of the support component.

Wet Weight - Displays the wet weight of the support component.

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

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

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

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

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

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

Fabrication and Construction

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

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

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

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

Surface Treatment and Coating

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

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

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

Exterior Coating Area - Enter the coating coverage.

Responsibility

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

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

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

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

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

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

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

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

Related Topics

- *Support Component Properties Dialog Box*, page 51

Definition Tab (Support Component Properties Dialog Box)

Displays the support component properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you select more than one support component, and then select the **Properties** command, only the common properties among the selected support components display.

The properties that display depend on what you defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Related Topics

- *Support Component Properties Dialog Box*, page 51

Connections Tab (Support Component Properties Dialog Box)

Displays the connection information for the object, the properties and their values, as defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Ports - Select the part port for which you want to view properties.

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

Value - Displays the value of the corresponding property.

Related Topics

- *Support Component Properties Dialog Box*, page 51

Hanger Connection Properties Dialog Box

Specifies properties for the connection that you have selected.

Related Topics
- *Configuration Tab*, page 49
- *General Tab (Hanger Connection Properties Dialog Box)*, page 55
- *Relationship Tab*, page 49

General Tab (Hanger Connection Properties Dialog Box)

Displays the connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you select more than one connection and then select the **Properties** command, only the common properties among the selected connections display.

When viewing properties for a single connection, the following properties display. More properties may display depending on what you defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Category - Select the properties that you want to view for the connection. Connection properties are in the **Standard** category.

Standard

Connection Type - Displays the type of connection. If you want to add, edit, or remove values that are available for selection, edit the **HngSupConnectionType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Connection Process - Displays the process associated with the connection. If you want to add, edit, or remove values that are available for selection, edit the **HngSupConnectionType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Related Topics
- *Hanger Connection Properties Dialog Box*, page 55

Working with Members: An Overview

You can place linear and curved members. All linear members are placed using the *Place Linear Member System Command*, page 62 . All curved members are placed using the *Place Curve Member Command* . Both commands provide options for selecting member type category, type, section name, cardinal point, and other options during placement.

In addition, SmartPlant 3D provides several member placement productivity commands in the Structure task that, depending on what you are doing, might be a better choice than the Place Linear Member System command. For example, to place a column at each grid intersection in one operation, use the *Place Columns at Grid Intersections Command* . If you want to place cross bracing, use the *Place Bracing Command* . Use the *Place Framing Members Command* to place secondary framing members in a bay. If you want to place support members around a vertical vessel, use the *Place Vessel Supports Command*, . Before you start placing members however, there are concepts that you need to know.

Note

- Although not required, we recommend that you place grid planes, elevations planes, and grid lines using the Grids task before placing structural members.

Member Systems

Member systems are logical collections of member parts that maintain the design basis and physical alignment of the member parts for analysis, design, and manufacturing. For example, in vertical cross-bracing, typically one of the vertical braces is split into two parts so that it does not interfere with the other vertical brace in the cross. The member system for that split vertical brace maintains the co-linear alignment of the two parts when you move either outside corner of the vertical brace. Another example of a member system would be a jacket leg. The leg is comprised of different parts, including cans that have different cross section sizes, but you want the entire leg to move as a single member. Use the *Place Split Command* to split member systems into member parts.

Member systems connect to other member systems using Frame Connections. For more information, see *Understanding Framing Connections: An Overview*, page 58.

Member Parts

Member parts represent the real, physical member parts in the model. Member parts connect logically to other member parts using Assembly Connections. For more information, see *Understanding Member Assembly Connections: An Overview*, page 89.

Member Type Category and Type

Member categories are broader groupings of member types. For example, the software delivers a member type category called column. In the column type category are member types called column and stud. When placing a member, you have to select a member type category and a member type. You can define your own member categories and member types by editing the **Structural Member Type** select list in the Catalog task. Refer to the *Catalog User's Guide* for more information.

Member Local Coordinate System and Orientation

The software uses the following convention to determine the local coordinate system of a member. The member's local x-axis is along the member axis from member start to member end. The member's local z-axis is the strong axis of the member cross section. The member's local y-axis is determined by the right-hand-rule using the local x- and z-axes.

When placing members, the software sets the local z-axis of the member parallel to the global Z-axis by default. However, if you rotate the member such that the local x-axis of the member becomes parallel to the global Z-axis, then the software switches the local z-axis of the member to be parallel to the positive global X-axis. When you select a member part, the software indicates the member's x- and z-axes with arrows that display at the member part start end.

Discrete or Contiguous Placement Methods

When placing members in the model, you can use either discrete placement or contiguous placement. The discrete placement method requires you to define both the start and the end points of the member, which is useful when placing columns. The contiguous placement method uses the end point of the previously placed member as the start point of the next member. This method is useful when placing multiple beams, because in most cases the end of the previous beam is where you want to start the next beam.

The **Start** and **End** commands on the ribbon are used to toggle between placement methods.

Related Topics

- *Delete a Member System*, page 88
- *Edit Member Part Properties*, page 87
- *Edit Member System Properties*, page 87
- *Place a Member using Finish Mode*, page 86
- *Place Members using Contiguous Placement*, page 85
- *Place Members using Discrete Placement*, page 84

Understanding Framing Connections: An Overview

Frame connections describe the positioning relationship between member systems. This positioning relationship defines the member orientation and offset of the supported member in relation to the supporting member. Two frame connections are placed when you place the member system, one at each end. Because frame connections define relationships between member systems, the frame connection may prevent you from moving a member. For help in moving members, see *Move One End of a Member*, and *Move a Member*. Refer to *Working with Members: An Overview*, page 56 for important related information about member systems.

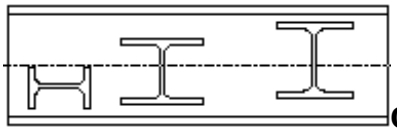
There are several basic types of frame connections:

Axis Along - An axis along frame connection aligns the cardinal point on the supported member system with the cardinal point on the supporting member system. Use this frame connection when the member systems are different types (a beam framing into a top of a column for example). Using this frame connection, the beam will slide along the length of the column, but will not cause the column to lengthen or shorten. You can specify an optional offset in all three directions.

Axis End - An axis end frame connection aligns the cardinal point on the supported member system with the cardinal point on the supporting member system. Use this frame connection when both member systems are of the same type (both columns, or both beams). If you move one member system end, this frame connection automatically moves the other member system end to maintain the connection. You can specify an optional offset in all three directions.

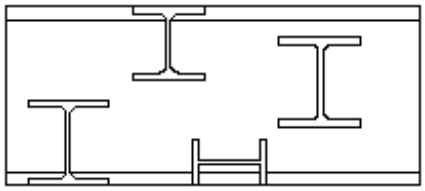
Important

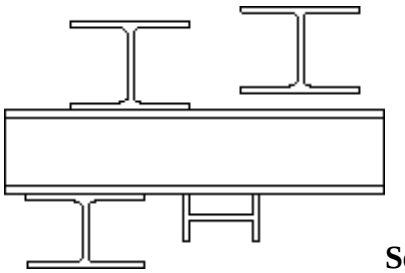
- The Axis End frame connection makes the members mutually editable. For example, if you move one member, the other member will extend or shorten to maintain the connection. Because of this, if you change the permission group of one member, the software automatically changes the permission group of the other member to match.



Centerline - A centerline frame connection uses the supporting member's centerline to position the supported member.

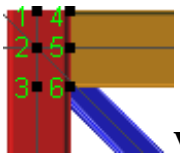
Flush - A flush frame connection uses the supporting member's top and bottom extent to position the supported member. The supported member typically lies within the body of the supporting member.





Seated - A seated frame connection uses the supporting member's top or bottom extent to position the supported member. The supported member typically rests against the supporting member, but can be offset.

Surface - A surface connection specifies the relationship between a supported member and the surface of the supporting object.



Vertical Corner Brace - A vertical corner brace connection specifies the location of a vertical brace that frames into a column-beam corner. You can define offsets in the X, Y, and Z-directions, and there are six work points to select from when using this connection.

Unsupported - An unsupported connection allows you to place a member in free space without defining any frame connection.

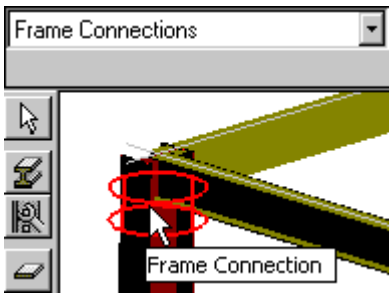
Selection of Frame Connections

During the placement of linear members, you can have the software determine frame connections by selecting the **By Rule** option. The software uses these rules based on supported member type category, type, permission group, and geometry as it connects to the supporting member to select frame connections:

- When the member connects to non-member objects, the software select the "Unsupported" frame connection unless the non-member object is a surface, in which case "Surface-Default" is chosen.
- When a member connects to a single member, the software selects "Axis Along" unless:
  - The two members have the same type category, are parallel, end-matched (connected end to end), and in the same Permission Group, then the software selects "Axis End".
  - The two members have the same type category and are end-matched, then the software selects "Axis End".
  - The member being placed has a type category of Brace and the two members are end-matched, then the software selects "Axis End".
  - The member being placed has a member type of Girt or Purlin and the two members are not parallel, then the software selects "Seated-Top".
- When placing a member you select another member's frame connection as the end point, the software reads both the frame connection's member and it's optional supporting member. If the member being placed is coplanar with those two members, then the software selects Vertical Corner Brace-WP2.
- When placing a member you select a split connection as the end point, the software reads the two members related to the split connection. If the member being placed is coplanar with those two members, then the software selects Vertical Corner Brace-WP2.

Locating Frame Connections

Frame connections do not display in the model except during member placement. However, if you set the **Locate Filter** to **Frame Connections**, you can locate and select frame connections for review and editing. Frame connections are located near the ends of member systems and appear as circles when you move the cursor over them.



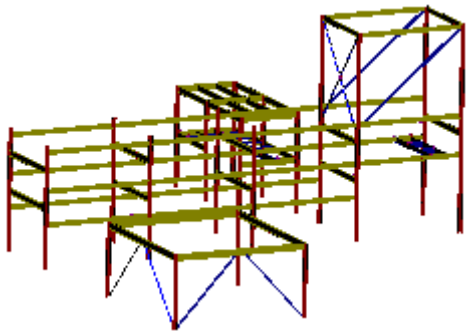
When you select a frame connection, the software displays the frame connection type in the ribbon. Select the **Edit > Properties** command to edit the frame connection properties. You cannot copy a frame connection using the **Edit > Copy** command.

Related Topics

- *Edit a Frame Connection*, page 87
- *Working with Members: An Overview*, page 56

Place Linear Member System Command

Places a linear member in the model. You can place beams, columns, braces, truss elements, or cable member types using this command. For additional information, see *Working with Members: An Overview*, page 56. Use this command when you want to place members by specifying the exact start and end points.



You can define custom member types by editing the **Structural Member Type** select list in the Catalog task.

Related Topics

- *Copy Framing Members*, page 88
- *Delete a Member System*, page 88
- *Edit Member Part Properties*, page 87
- *Edit Member System Properties*, page 87
- *Place a Member using Finish Mode*, page 86
- *Place Linear Member System Ribbon*, page 63
- *Place Members using Contiguous Placement*, page 85
- *Place Members using Discrete Placement*, page 84
- *Working with Members: An Overview*, page 56

Place Linear Member System Ribbon

Specifies the properties for the member that you are placing. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon*.

Member Properties - Activates the **Member Properties** dialog box. You can use this dialog box to specify additional member properties, such as material, material grade, and end releases, which you cannot set on the ribbon.

Start - Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see *Working with Members: An Overview*, page 56.

End - Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see *Working with Members: An Overview*, page 56.

Finish - Click to place the member in the model. The **Finish** button is active only when **Finish Mode** is selected.

Finish Mode - Specify whether or not the **Finish** button must be selected to place a member in the model. If the **Finish Mode** is selected, the software places the member in tentative mode after you identify the second end point. This tentative mode allows you to modify placement settings such as the offset, cardinal point, or frame connection properties before you commit the member to the model. If the **Finish Mode** is not selected, then the software automatically places the member in the model after you identify the second end point.

Connection - Select the frame connection type to use for the member that you are placing. If you select **By Rule**, the software determines the frame connection to use based on the geometry between the member that you are placing and existing members in the model. If you select **More**, all available frame connections display from which you can select the frame connection that you want to use. For more information about frame connections, see *Understanding Framing Connections: An Overview*, page 58. This option is not available if you are editing an existing member.

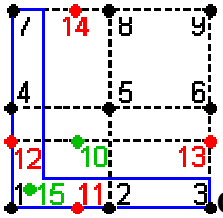
Connection Properties - Activates the **Connections Properties** dialog box, which is used to specify properties for the active frame connection. The properties that appear in this dialog box are described below under Connection Properties.

System - Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box*, page 19.

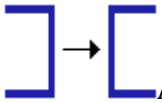
Type Category - Select the type category of the member that you want to place, such as a beam or a column. The properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** select list in the Catalog task.

Type - Select the type of member that you want to place, such as a horizontal brace, vertical brace, or knee brace. The properties change depending on the member type that you select. You can define a custom member type by editing the **Structural Member Type** select list in the Catalog task.

Section Name - Defines the cross-section for the member. If you know the section name that you want, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See *Structure Reference Data Guide* for more information about reference data.



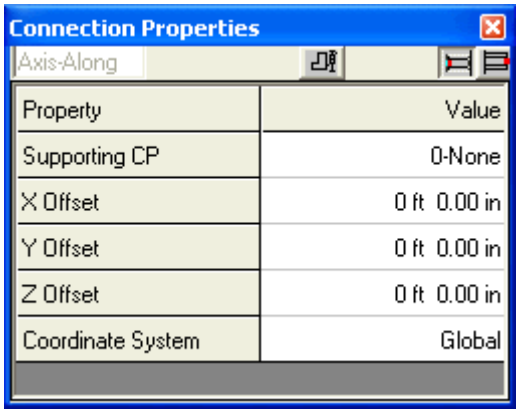
Cardinal Point - Specify the relative position of the structural section to the member placement line. Fifteen cardinal positions are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13.


Angle - Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Reflect - Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction.

Connection Properties

The **Connection Properties** appear only when you have selected the **Connection Properties** option. Connection properties change depending on the frame connection specified in the **Connection** option. To see the frame connection properties for the start of the member, select . To see the frame connection properties for the end of the member, select . Click to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.



Tip

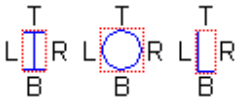
- The *supported* member is the member that you are placing. The *supporting* member is the existing member in the model to which you are connecting.

Seated, Flush, and Centerline Frame Connection Properties

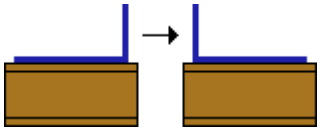
Side - Select the side of the supporting member on which you want to place the supported member.

Offset - Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

Edge - Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.



Reflect - Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option would be when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Axis Frame Connection Properties

X Offset - Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset - Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset - Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System - Specifies the coordinate system to use for the offset values.

Supporting CP - Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Surface Frame Connection Properties

X Offset - Specifies an offset to apply in the x-direction.

Y Offset - Specifies an offset to apply in the y-direction.

Z Offset - Specifies an offset to apply in the z-direction.

Coordinate System - Specifies the coordinate system to use for the offset values.

Vertical Corner Brace Frame Connection Properties

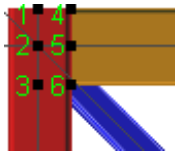
X Offset - Specifies the offset to apply in the x-direction.

Y Offset - Specifies the offset to apply in the y-direction.

Z Offset - Specifies the offset to apply in the z-direction.

Coordinate System - Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point - Specifies the work point location. There are six work point locations that you can choose.



1	Primary Center - Secondary Far Side
2	Primary Center - Secondary Center
3	Primary Center - Secondary Near Side
4	Primary Near Side - Secondary Far Side
5	Primary Near Side - Secondary Center
6	Primary Near Side - Secondary Near Side

Related Topics

- *Place Linear Member System Command*, page 62

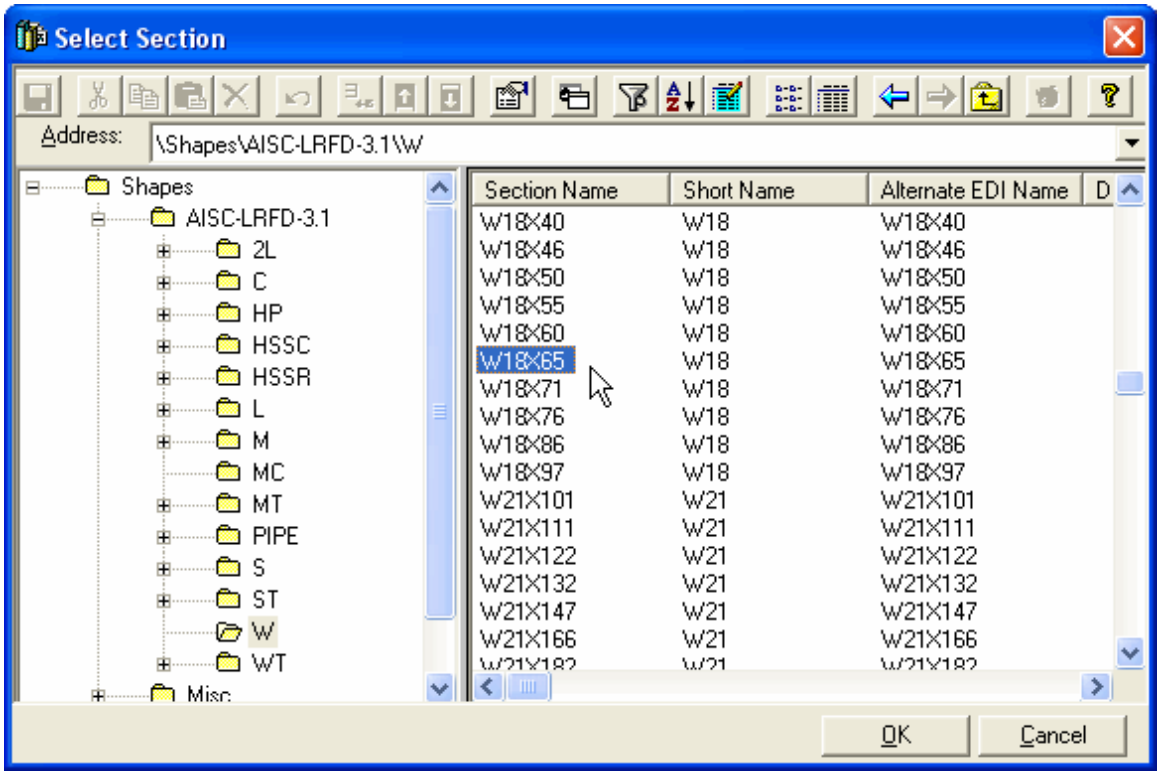
Select System Dialog Box

This dialog box displays all of the defined systems so that you can select the system that you want. You can create new systems in the Systems and Specifications task.

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

Select Section Dialog Box

Allows selection of the type of section to be placed. This dialog box appears when you click the **More** option on the **Section Name** list. By browsing through the hierarchy, you can find any section in the Catalog database. After you select a section, the software returns you to the model, where you can finalize placement.



Properties - Displays the properties of the selected section. Because you cannot modify any properties until the section is placed, all properties on the dialog box are read-only.

Preview - Displays a picture of the selected section. The image file must be assigned to the section in the reference data.

Filter - Allows you to filter catalog data to help find the subset of data that you want to work with, similar to Microsoft Excel.

Sort - Sorts the catalog data by column to help you find like items.

Customize Current View - Defines with columns in the data you want to see.

List View - Sets the dialog box to display sections in a list view.

Grid View - Sets the dialog box to display sections in a spreadsheet-style grid view.

Back - Returns you to the previously selected section type or node. Use this command to navigate through the hierarchy to the specific type that you need.

Forward - Sends you to the last selected section type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

Up One Level - Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Address - Specifies your exact location within the displayed hierarchy.

Member System Prismatic Properties Dialog Box

Specifies the properties for the member system that you are editing. For an explanation of the difference between a member system and a member part, see *Working with Members: An Overview*, page 56.

Related Topics

- *Configuration Tab*, page 49
- *Member System Tab (Member System Prismatic Properties Dialog Box)*, page 70
- *Notes Tab*, page 50
- *Place Linear Member System Command*, page 62
- *Relationship Tab*, page 49

Member System Tab (Member System Prismatic Properties Dialog Box)

Specifies the properties for the member system.

Category - Select the properties to view for the member system.

Standard

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

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

Parent System - Select the system to which the member system that you are placing belongs.

Type category - Select the type category of the member system, such as a beam or a column. You can define a custom member type category on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.

Type - Select the type of member, such as a beam or a column. You can define a custom member type on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.

Priority - Select the priority to assign to the member system. The priority is used to group members.

Continuity Type - Defines how the member system should react when it intersects another member system (your automatic splitting preference). Select **Continuous** to indicate that the member system should split the other member system. Select **Intercostal** to indicate that the member system should be split by the other member system.

Continuity Priority Number - Specify the continuity priority. This priority is used to select which member system is split when two member systems intersect, but have the same **Continuity Type** value. Member systems with a lower continuity priority (1, 2, 3, for example) will split member systems with a higher continuity priority (7, 8, 9, for example).

Align - If set to True, the software copies offsets from the frame connection at the member system end to the unsupported frame connection at the other member system end.

Start East - Displays the X-coordinates of the start of the member relative to the active coordinate system.

Start North - Displays the Y-coordinates of the start of the member relative to the active coordinate system.

Start Elevation - Displays the Z-coordinates of the start of the member relative to the active coordinate system.

End East - Displays the X-coordinates of the end of the member relative to the active coordinate system.

End North - Displays the Y-coordinates of the end of the member relative to the active coordinate system.

End Elevation - Displays the Z-coordinates of the end of the member relative to the active coordinate system.

Related Topics

- *Member System Prismatic Properties Dialog Box*, page 69

Relationship Tab

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

Name - Displays the name of the related object.

Type - Displays the type of related object.

Go To - Displays the properties of the selected object.

Configuration Tab

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

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

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

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

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

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

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

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

Notes Tab

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

Note

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

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

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

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

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

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

Purpose of note - Specifies the purpose of the note.

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

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

New Note - Creates a new note on the object.

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

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

Delete Note - Deletes the currently displayed note.

Member Part Prismatic Properties Dialog Box

Specifies the properties for the member part that you are editing. For an explanation of the difference between a member system and a member part, see *Working with Members: An Overview*, page 56.

Related Topics

- *Configuration Tab*, page 49
- *Cross Section Tab (Member Part Prismatic Properties Dialog Box)*, page 78
- *Member Part Tab (Member Part Prismatic Properties Dialog Box)*, page 74
- *Notes Tab*, page 50
- *Place Linear Member System Command*, page 62
- *Relationship Tab*, page 49

Member Part Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the member.

Category - Select the properties that you want to view for the member.

Member part properties are divided into several different categories: Standard, Weight and CG, Fabrication and Construction, Surface Treatment and Coating, Responsibility, and End Releases. You select which category that you want to define values for by using the **Category** option.

Standard

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

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

Parent System - Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Type category - Select the type category of the member part, such as a beam or a column. You can define a custom member type category on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.

Type - Select the type of member part, such as a beam or a column. You can define a custom member type on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.

Priority - Select the priority to assign to the member part. The priority is used to group members.

Length - Displays the length of the member without cutbacks applied. You cannot change this value.

Cut Length - Displays the length of the member with cutbacks applied. You cannot change this value. A cutback is that part of a member removed by an assembly connection or by a manually placed trim (a cope, for example).

Reporting Requirements - Specify whether or not this member part is reported.

Reporting Type - Select the reporting requirements code for the member part. Valid codes are defined in the **AllCodeLists.xls** workbook on the **Reporting Type** sheet.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density times the object's solid volume. Therefore, the material of the object will affect the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight - Displays the dry weight of the object.

Wet Weight - Displays the wet weight of the object.

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

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

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

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

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

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

Dry WCG Origin - Specifies how the dry weight center-of-gravity location is defined. Select **Computed** if you want the software to calculate the origin location. Select **Defined** if you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin - Specifies how the wet weight center-of-gravity location is defined. Select **Computed** if you want the software to calculate the origin location. Select **Defined** if you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

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

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

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

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

Surface Treatment and Coating

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

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

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

Exterior Coating Area - Enter the coating coverage.

Responsibility

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

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

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

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

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

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

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

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

End Releases

Start Member Release - Select the directions to release at the start of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions that you want to use is not available, select the **User Defined** option and define the releases yourself.

Start X Displacement - Defines if the X direction at the start of the member part is fixed or free.

Start Y Displacement - Defines if the Y direction at the start of the member part is fixed or free.

Start Z Displacement - Defines if the Z direction at the start of the member part is fixed or free.

Start X Rotation - Defines if the X moment direction at the start of the member part is fixed or free.

Start Y Rotation - Defines if the Y moment direction at the start of the member part is fixed or free.

Start Z Rotation - Defines if the Z moment direction at the start of the member part is fixed or free.

End Member Release - Select the directions to release at the end of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions that you want to use is not available, select the **User Defined** option and define the releases yourself.

End X Displacement - Defines if the X direction at the end of the member part is fixed or free.

End Y Displacement - Defines if the Y direction at the end of the member part is fixed or free.

End Z Displacement - Defines if the Z direction at the end of the member part is fixed or free.

End X Rotation - Defines if the X moment direction at the end of the member part is fixed or free.

End Y Rotation - Defines if the Y moment direction at the end of the member part is fixed or free.

End Z Rotation - Defines if the Z moment direction at the end of the member part is fixed or free.

Related Topics

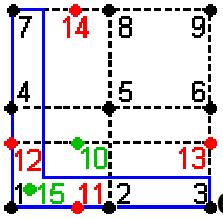
- *Member Part Prismatic Properties Dialog Box*, page 73

Cross Section Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the cross section of a member part.

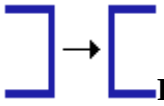
Section Standard - Select the section library from which you want to select a section for this member. Sections are defined in the reference data.

Section Name - Defines the cross-section for the member. If you know the section name that you want, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See *Structure Reference Data Guide* for more information about reference data.



Cardinal Point - Specify the relative position of the structural section to the member placement line. Fifteen cardinal positions are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13.

Angle - Defines the angle by which the section is rotated about the member axis.



Reflect - Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction.

Material - Select a material for the member. Materials are defined in the **AllCommon.xls** workbook or in the Catalog task.

Grade - Select a material grade for the member. Material grades are defined in the **AllCommon.xls** workbook or in the Catalog task.

Cross Section Type - Displays the cross-section type.

Related Topics
- *Member Part Prismatic Properties Dialog Box*, page 73

Frame Connection Properties Dialog Box

Specifies the properties for the frame connection that you are editing.

Related Topics
- *Edit a Frame Connection*, page 87
- *General Tab (Frame Connection Properties Dialog Box)*, page 79
- *Notes Tab*, page 50
- *Relationship Tab*, page 49
- *Understanding Framing Connections: An Overview*, page 58

General Tab (Frame Connection Properties Dialog Box)

Specifies the properties for the frame connection.

Category - Select the type of properties that you want to view for the selected frame connection.

Type - Displays the type of frame connection. This box is read-only.

Property - Displays all properties associated with the selected frame connection. The list of available properties depends on what was defined in the reference data for the frame connection type.

Value - Specifies the values for the frame connection properties.

Tip

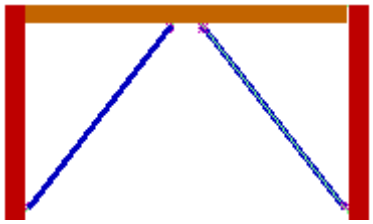
- The *supported* member is the member that you are placing. The *supporting* member is the existing member in the model to which you are connecting.

Name - Specifies the frame connection name.

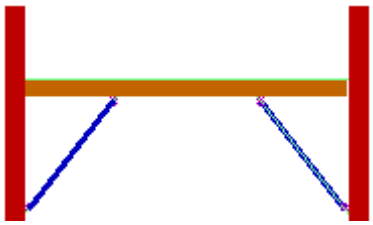
Name rule - Specifies the naming rule used to name the frame connection.

Position rule - Defines how the frame connection is to behave when the supporting member is moved. You can select one of three options:

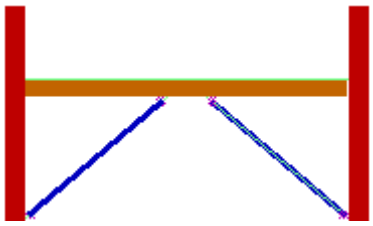
- **Intersection** - The member system lengthens or shortens to maintain the connections with the supporting member. The end of the supported member system will slide to a new location on the supporting member.
- **Ratio** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same relative position (that you can define) along the supporting member system. This option is similar to the **Distance Along** option except that you define a percentage ratio from the supporting member end.
- **Distance Along** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same position (that you can define) along the supporting member system. This option is similar to the **Ratio** option except that you define an actual distance from the supporting member end.



Original Position



Intersection Position Rule



Ratio Position Rule

Distance along - Enter the distance from the supporting member end that the supported member is positioned. You must include the units of measurement when defining this distance. Which supporting member end that is measured from is defined using the **End** option. This option is available when you set **Position Rule** to **Distance Along**.

Ratio - Enter the ratio of the supporting member length that the supported member is positioned. For example, enter .25 if you want the supported member a fourth of the way along the supporting member. Enter .333 if you want the supported member a third of the way, and so forth. Which supporting member end that is measured from is defined using the **End** option. This option is available when you set **Position Rule** to **Ratio**.

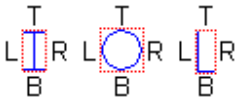
End - Specifies which end of the supporting member that the ratio or the distance-along distance is measured. You can select **Start**, **End**, or **Auto**. Start is the first member end that was placed. End is the second member end that was placed. If you select **Auto**, the software automatically selects the supporting member end that is closest to the frame connection. The **Auto** setting is recommended so that you do not have to worry whether the supporting member was modeled left-to-right, right-to-left, top-to-bottom, or bottom-to-top.

Seated, Flush, and Centerline Frame Connection Properties

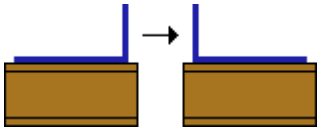
Side - Select the side of the supporting member on which you want to place the supported member.

Offset - Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

Edge - Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.



Reflect - Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option would be when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Surface Frame Connection Properties

X Offset - Specifies an offset to apply in the x-direction.

Y Offset - Specifies an offset to apply in the y-direction.

Z Offset - Specifies an offset to apply in the z-direction.

Coordinate System - Specifies the coordinate system to use for the offset values.

Axis Frame Connection Properties

X Offset - Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset - Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset - Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System - Specifies the coordinate system to use for the offset values.

Supporting CP - Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Vertical Corner Brace Frame Connection Properties

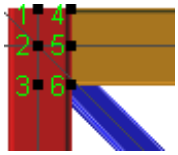
X Offset - Specifies the offset to apply in the x-direction.

Y Offset - Specifies the offset to apply in the y-direction.

Z Offset - Specifies the offset to apply in the z-direction.

Coordinate System - Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point - Specifies the work point location. There are six work point locations that you can choose.



1	Primary Center - Secondary Far Side
2	Primary Center - Secondary Center
3	Primary Center - Secondary Near Side
4	Primary Near Side - Secondary Far Side
5	Primary Near Side - Secondary Center
6	Primary Near Side - Secondary Near Side

Related Topics

- *Frame Connection Properties Dialog Box*, page 79

Place Members using Discrete Placement

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   Tip

   - If you are unsure of which frame connection type to use, review *Understanding Framing Connections: An Overview*, page 58. You can also select the **By Rule** option to allow the software to select automatically a frame connection type.
3. In the **Type category** box, select the member type category to place.
4. In the **Type** box, select the member type to place.
5. Specify the start location, or first point, of the member.
6. Specify the end location, or second point, of the member.
7. Click **Start** to specify the start location of the next member.
8. Specify the start location of the next member.

Notes

- In order to find the intersection of grid lines easily, verify that the SmartSketch **Intersection** option is selected. Click **Tools > Options**, and then select the **SmartSketch** tab to access the **Intersection** option.
- You can use the frame connection of another member as the start or end location of the member that you are placing.

Related Topics

- *Place Linear Member System Command*, page 62
- *Working with Members: An Overview*, page 56

Place Members using Contiguous Placement

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   Tip

   - If you are unsure of which frame connection type to use, review *Understanding Framing Connections: An Overview*, page 58. You can also select the **By Rule** option to allow the software automatically to select a frame connection type.
3. In the **Type category** box, select the member type category to place.
4. In the **Type** box, select the member type to place.
5. Specify the start location, or first point, of the member.
6. Specify the end location, or second point, of the member.
7. Click **End** to activate the contiguous placement mode.
8. Specify the end location of the next member.

Notes

- In order to find the intersection of grid lines easily, verify that the SmartSketch **Intersection** option is selected. Click **Tools > Options**, and then select the **SmartSketch** tab to access the **Intersection** option.
- You can use the frame connection of another member as the start or end location of the member that you are placing.

Related Topics

- *Place Linear Member System Command*, page 62
- *Working with Members: An Overview*, page 56

Place a Member using Finish Mode

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   Tip

   - If you are unsure of which frame connection type to use, review *Understanding Framing Connections: An Overview*, page 58. You can also select the **By Rule** option to allow the software to select automatically a frame connection type.
3. Click **Finish Mode** to activate the **Finish** button.
4. Click **Connection Properties** to activate the **Connection Properties** dialog box.
5. In the **Type category** box, select the member type category to place.
6. In the **Type** box, select the member type to place.
7. Specify the start location, or first point, of the member.
8. Specify the end location, or second point, of the member.
9. Edit the start and end frame connection properties in the **Connection Properties** dialog box.
10. Edit the member properties using the **Member Properties** dialog box.
11. Click **Finish**.

Notes

- In order to find the intersection of grid lines easily, verify that the SmartSketch **Intersection** option is selected. Click **Tools > Options**, and then select the **SmartSketch** tab to access the **Intersection** option.
- You can use the frame connection of another member as the start or end location of the member that you are placing.

Related Topics

- *Place Linear Member System Command*, page 62
- *Working with Members: An Overview*, page 56

Edit Member System Properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Systems** in the **Locate Filter**.
3. Select the member system to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed.

Related Topics

- *Member System Prismatic Properties Dialog Box*, page 69

Edit Member Part Properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed.

Related Topics

- *Member Part Prismatic Properties Dialog Box*, page 73

Edit a Frame Connection

1. Click **Select** on the vertical toolbar.
2. Select **Frame Connection** in the **Locate Filter**.
3. Select the frame connection to edit.
4. Edit the frame connection as needed.

Related Topics

- *Frame Connection Properties Dialog Box*, page 79
- *Understanding Framing Connections: An Overview*, page 58

Copy Framing Members

1. Click **Select** on the vertical toolbar.
2. Select **Member Systems** in the locate filter.
3. Select the members in the framing system to copy. Hold down the CTRL key as you select the members.
4. Press CTRL+C or click **Edit > Copy** to copy the members to the clipboard.
5. Select a reference point (a from point) for the members being copied.
6. Press CTRL+V or click **Edit > Paste** to paste the members from the clipboard.
7. In the **Paste** dialog box, select new systems and dependencies for the paste members.
8. Click **OK** on the **Paste Special** dialog box.
9. Select the new location of the reference point.

Notes

- Assembly Connections are not copied to the pasted members.
- If you do not select new dependencies for the pasted members in the **Paste** dialog box, the pasted members are placed in the same location as the copied members.

Related Topics

- *Place Linear Member System Command*, page 62
- *Working with Members: An Overview*, page 56

Delete a Member System

1. Click **Select** on the vertical toolbar.
2. Select **Member Systems** in the **Locate Filter**.
3. Select the members to delete.
4. Click **Delete**.

Notes

- All loads and boundary conditions placed in the Structural Analysis task on the deleted member system are also deleted. This could affect any Analytical Models that have been exported.
- All footings associated with the member system are also deleted.
- Ladders or stairs using the deleted member system as the defined top edge are sent to the To Do List.

Related Topics

- *Place Linear Member System Command*, page 62

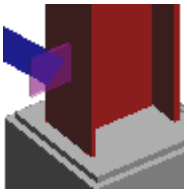
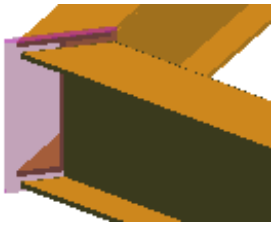
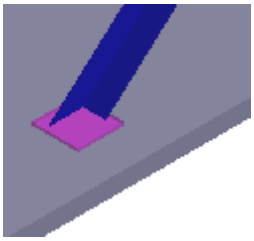
Understanding Member Assembly Connections: An Overview

Member assembly connections are similar to frame connections, but define the necessary trimming between member parts and the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections control member features including cutbacks, copes, notches, bolt holes, and slots. Whether or not features are placed depends on the member assembly connection type and the geometry of the connection between the members.

There are several basic assembly connections delivered with the software. You can create your own assembly connections by editing the **StructAssemblyConnections.xls** workbook, and then bulk loading the workbook. For more information on creating your own assembly connections, see the *Structure Reference Data Guide*.

For very complex nodal connections, instead of writing a custom assembly connection, you may want to try the *Trim Member Command* .

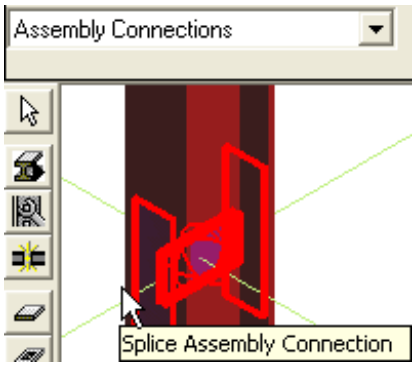
-  The base plate assembly connection places a plate at the end of an unsupported member. This assembly connection requires an unsupported frame connection on one member. For example, use this assembly connection to place a base plate at the bottom of a column.
-  The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.
-  The fitted assembly connection connects two members. This assembly connection requires a frame connection with two members, such as axis, seated, or flush. Examples of this connection include a beam framing into a column or a beam framing into another beam.

-  The gusset plate assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. This assembly connection requires a frame connection with two members, such as axis.
-  The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.
-  The splice assembly connection connects two members that are colinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
-  The general surface assembly connection connects a member end with a nonmember surface such as a slab. The member is cut to surface and a base plate is placed on the member end.

See *Working with Members: An Overview*, page 56 for important related information.

Locating Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Assembly Connections**, you can locate and select assembly connections for review and editing. Assembly connections are located at the ends of member parts and appear as circles when you move the cursor over them. Any assembly components, such as gusset plates, associated with the assembly connection also highlight.



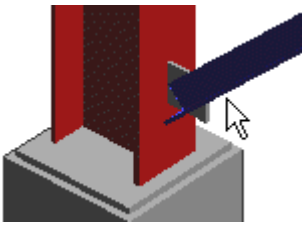
When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties.

Related Topics

- *Place an Assembly Connection*, page 99
- *Working with Members: An Overview*, page 56

Place Assembly Connection Command

 Places an assembly connection at the selected frame connection. Assembly connections define the necessary trimming between member parts and provides for the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections also control cutbacks, copes, notches, boltholes, and slots.



This command only places member assembly connections. The software automatically places slab assembly connections for you when you place the slab. See *Understanding Member Assembly Connections: An Overview*, page 89 and *Understanding Slab Assembly Connections: An Overview* for more information.

Related Topics

- *Delete an Assembly Connection*, page 100
- *Edit Assembly Connection Properties*, page 100
- *Place an Assembly Connection*, page 99
- *Understanding Member Assembly Connections: An Overview*, page 89

Place Assembly Connection Ribbon

Specifies the properties for the assembly connection that you are placing or editing.

Assembly Connection Properties - Activates the **Assembly Connections Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Assembly Connection Properties Dialog Box*, page 95.

Select Member/Connection - Activated automatically by the software so that you can select the frame connection or the member for which you want to place assembly connections.

Finish - Click to place the assembly connection the model.

Cancel - Rejects the selected object.

Accept - Confirms that the selected member, or members, is the member to place assembly connections for. The software displays in tentative mode the results of the assembly connection.

By Rule - Select to allow the software to select the assembly connection to use based on the selected member parts and their orientation to each other.

Condition - Specifies how you want the software to handle existing assembly connections when you try to place a new assembly connection at the same location. Select **Retain existing** to keep the existing assembly connection. Select **Update existing** to replace the existing assembly connection with the new assembly connection.

System - Select the system to which the member belongs. You can define new systems in the Systems and Specifications task.

Type - Select the assembly connection type to use. If you selected the **By Rule** option, the software determines the correct assembly connection to use based on the geometry between the member parts, and this option is disabled. If you select **More**, all available assembly connections display from which you can select the assembly connection to use. For more information about assembly connections, see *Understanding Member Assembly Connections: An Overview*, page 89.

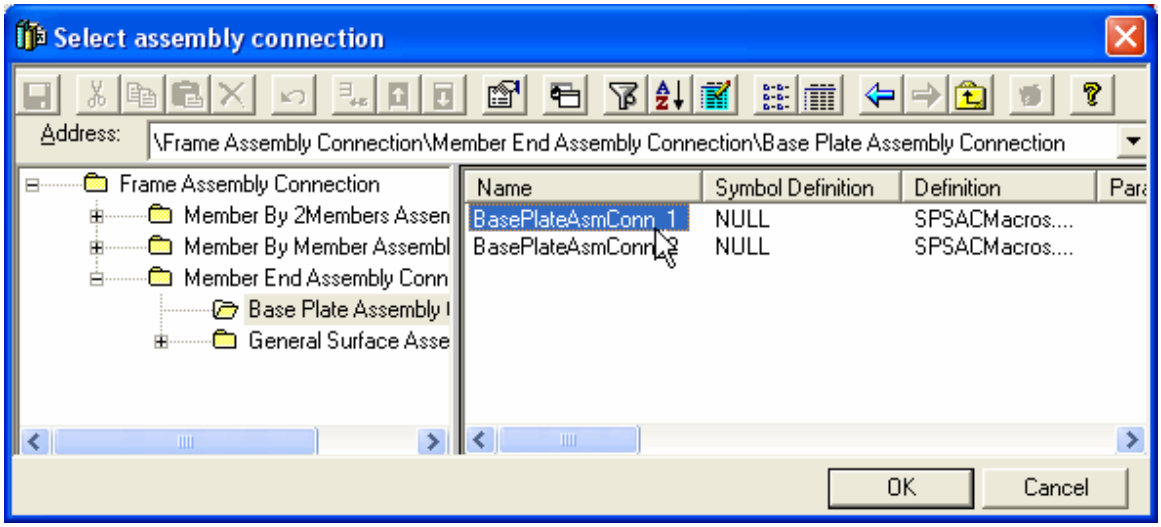
Name - Specify the name of the assembly connection.

Related Topics

- *Place an Assembly Connection*, page 99
- *Place Assembly Connection Command*, page 92

Select Assembly Connection Dialog Box

Allows selection of the type of connection to be placed. This dialog box appears when you click the **More** option on the **Type** list. By browsing through the hierarchy, you can find any connection in the Catalog database. After you select a connection, the software returns you to the model, where you can finalize placement.



Properties - Displays the properties of the selected connection. Because you cannot modify any properties until the connection is placed, all properties on the dialog box are read-only.

Preview - Displays a picture of the selected connection. The image file must be assigned to the connection in the reference data.

Filter - Allows you to filter catalog data to help find the subset of data that you want to work with, similar to Microsoft Excel.

Sort - Sorts the catalog data by column to help you find like items.

Customize Current View - Defines with columns in the data you want to see.

List View - Sets the dialog box to display connections in a list view.

Grid View - Sets the dialog box to display connections in a spreadsheet-style grid view.

Back - Returns you to the previously selected connection type or node. Use this command to navigate through the hierarchy to the specific type that you need.

Forward - Sends you to the last selected connection type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

Up One Level - Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Address - Specifies your exact location within the displayed hierarchy.

Assembly Connection Properties Dialog Box

Specifies the properties for the assembly connection that you are editing.

Related Topics

- *Configuration Tab*, page 49
- *Definition Tab (Assembly Connection Properties Dialog Box)*, page 99
- *Edit Assembly Connection Properties*, page 100
- *Notes Tab*, page 50
- *Occurrence Tab (Assembly Connection Properties Dialog Box)*, page 95
- *Relationship Tab*, page 49
- *Understanding Member Assembly Connections: An Overview*, page 89

Occurrence Tab (Assembly Connection Properties Dialog Box)

The **Occurrence** tab displays the assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

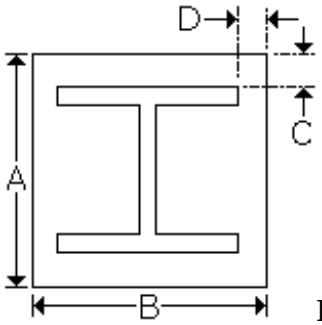
When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties.

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

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

System - Select the system to which the assembly connection that you are placing belongs. You can create new systems in the Systems and Specifications task.

Base Plate Assembly Connection Properties



Depth Clearance - Specify the clearance between the flange of the member and the edge of the base plate. This is dimension C in the figure.

Width Clearance - Specify the clearance between the flange of the member and the edge of the base plate. This is dimension D in the figure.

Sizing Rule - Select the sizing rule method for the base plate.

Plate Category - Select the plate category.

Plate Type - Select the plate type.

Miter Assembly Connection Properties

Top Distance - Specifies the distance between the top flange of the member section and the top of the plate.

Bottom Distance - Specifies the distance between the bottom flange of the member section and the bottom of the plate.

Left Distance - Specifies the distance between the left edge of the member section and the left edge of the plate.

Right Distance - Specifies the distance between the right edge of the member section and the right edge of the plate.

Symmetry - Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members should be cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

With Plates - Specifies whether or not a plate should be inserted between the member ends. Select **False** to not place the plate. Select **True** to place the plate.

Clearance - Specifies the distance between the member ends.

Sizing Rule - Select the sizing rule method for the base plate.

Slab by Member Boundary

Clearance - Enter a clearance distance between the edge of the slab and the member.

Port Face Position - Select the location on the member at which the slab is to stop. You can select the outmost plane, the centerline, or the in-most plane on the member.

Detailed Connection - Specifies if the assembly connection is a detailed connection.

Offset - Specify the distance between the selected **Port Face Position** and the edge of the slab. A negative value moves the edge into the body of the slab. A positive value moves the edge out from the body of the slab.

Slab Fee Edge Assembly

Reference Direction - Select the reference direction for the angle:

- **Normal** - The angle is measured from a vector perpendicular to the slab edge.
- **Horizontal** - The angle is measured from the global XY plane in the model.
- **Vertical** - The angle is measured from the Z-Axis in the model.

Angle - Enter a slope for the slab edge represented by the assembly connection. If the slab was place using the **Face Position Top**, the side face rotates about the top slab edge. If the slab was placed using the **Face Position Bottom**, the side face rotates about the bottom slab edge.

Offset - Specify the distance between the selected boundary object and the edge of the slab.

Splice Assembly Connection Properties

Symmetry - Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members should be cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

Clearance - Specifies the distance between the ends of the members.

Splice With - Select the plates that you want to use in the splice.

Web Plate Position - Select a web plate position.

Distance from flange gage line - Specifies the distance from the flange gage line.

Distance from web gage line - Specifies the distance from the web gage line.

Flange Plate Thickness - Specifies the thickness of the flange plates.

Flange Plate Length - Specifies the length of the flange plates.

Flange Plate Width - Specifies the width of the flange plates.

Flange Plate Category - Select the plate category for the flange plates.

Flange Plate Type - Select the plate type for the flange plates.

Web Plate Thickness - Specifies the thickness of the web plates

Web Plate Length - Specifies the length of the web plates.

Web Plate Width - Specifies the width of the web plates.

Web Plate Category - Select the plate category for the web plates.

Web Plate Type - Select the plate type for the web plates.

General Surface Assembly Connection Properties

Clearance - Type the distance between the end of the member and the face of the surface.

Related Topics

- *Assembly Connection Properties Dialog Box*, page 95

Definition Tab (Assembly Connection Properties Dialog Box)

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

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

Related Topics

- *Assembly Connection Properties Dialog Box*, page 95

Place an Assembly Connection

1. Click **Place Assembly Connection** on the vertical toolbar.
2. Select the **By Rule** option if you want the software to automatically select the type of assembly connection to place. Clear the **By Rule** option to select the assembly connection type yourself.
3. Select the frame connection nearest the member end to which you want to apply the assembly connection.
4. If you are selecting the assembly connection type yourself, do so now using the **Type** option.
5. Click **Accept**.

   The software automatically selects the assembly connection and displays the results.

6. Click **Finish**.

Notes

- For more information about the types of assembly connections, see *Understanding Member Assembly Connections: An Overview*, page 89.

Related Topics

- *Delete an Assembly Connection*, page 100
- *Edit Assembly Connection Properties*, page 100
- *Place Assembly Connection Command*, page 92
- *Understanding Member Assembly Connections: An Overview*, page 89

Edit Assembly Connection Properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the assembly connection to edit.
4. Click **Edit > Properties**.
5. Edit the assembly connection properties as needed.

Related Topics

- *Assembly Connection Properties Dialog Box*, page 95
- *Understanding Member Assembly Connections: An Overview*, page 89

Delete an Assembly Connection

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the assembly connection to delete.
4. Click **Delete**.

Related Topics

- *Edit Assembly Connection Properties*, page 100
- *Place an Assembly Connection*, page 99
- *Place Assembly Connection Command*, page 92
- *Understanding Member Assembly Connections: An Overview*, page 89

Hangers and Supports Glossary

A

assembly information rule

A program that pieces together individual parts to create a standard support assembly.

B

beam

A structural member type typically placed with the member axis in a nominal horizontal orientation.

brace

A diagonal member used to stiffen a framework.

C

cableway

Term to describe the volumetric path in a model design through which one or more cables pass from one location in the model to another. Cableway is synonymous with, and is used instead of, raceway or wireway.

channel

A structural shape referring to a three-sided member type with each of the sides joined at a right angle.

column

A vertical structural member usually attached to a footing and extending to the roof of a building.

G

girder

A horizontal support member similar to a beam. Some people maintain that girders span from column to column, and beams span from girder to girder. Other people maintain that beams span column to column and girders span from beam to beam.

girt

A beam, usually bolted to columns, to support the side covering or to serve as a window lintel.

grade

The material grade of the structural member.

grid

A network of uniformly spaced horizontal and perpendicular lines that help to identify either 2-D or 3-D relationships.

grid set
A group of grid lines placed within a plane that are linked. A grid set can be manipulated as a single unit.

I

I-Section
A structural shape referring to any member type in the form of an I.

item
A combination of an element and another type of data, such as a symbol or object.

J

joist
A horizontal structural members that support the floor or roof of a building.

K

knee brace
A corner brace used to prevent angular movement.

L

load group
A grouping in which all components feature uniform load limits and stress safety characteristics. For example, a pipe clamp from load group 5 will have a maximum nominal load of 20 kN and so will a threaded rod from load group 5.

lug (hangers and supports)
A plate with a bolt hole usually welded to the centerline of a pipe. Used to connect the pipe to the other parts of the hanger.

P

parameter
A property whose value determines the characteristics or behavior of something.

part class
A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part selection rule
A program that selects a particular part based on the supported and supporting attribute values entered by the user. For example, a part selection rule could select a 6 inch clamp to support a 4 inch pipe.

parts

The physical components that comprise a feature and are generally selected by the software. For example, the flanges, gaskets, and the gate valve itself are examples of the parts comprising the gate valve feature.

pipe run

A connected series of pipe components that have the same nominal piping diameter (NPD) and flow direction, and are governed by the same pipe specification.

pipeline

A set of graphically connected pipe runs including all branches.

R

rafters

Beams or truss members that support the purlins.

rebar

A term for steel reinforcing bars that are used to reinforce concrete.

run

Line or a portion of a line with no change in material properties or purpose.

S

section

A structural member whose parameters are defined in a table.

slab

A flat concrete area usually reinforced with wire mesh and rebar.

standard support

A single support object that can be ordered from a manufacturer. The contents of a standard support are an assembly. You cannot change or delete the parts of a standard support. This type of support is associative, meaning that if you change the size of a pipe, for example, the clamp on the pipe changes also.

stiffener

An angle, plate, or channel fastened to a member to prevent buckling.

strut

A compression member in a framework.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

W

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

Index