Structure

User's Guide

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Contents

Preface

This document is a user's guide for the SmartPlant 3D Structure 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.
- ISOGEN guides.

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 Global Workshare Guide - Provides instructions for setting up the software and the databases to work in a workshare environment.

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

SmartPlant 3D Installation Checklist - Provides a recommended installation workflow for installing SmartPlant 3D. The installation checklist, **SP3DInstall_Checklist.pdf** and **SP3DInstall_Checklist.xls**, is available in two file formats in the **Help** folder on the product CD.

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

SmartPlant 3D Integration Reference Guide - Provides information about installing, configuring, and using SmartPlant 3D in an integrated environment.

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

SmartPlant 3D Interpreting Human Piping Specifications - Provides information about how to interpret human piping specifications so that you can create the corresponding piping specification in the software.

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 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.

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.

Orthographic Drawings User's Guide - Provides information about creating and managing orthographic drawings.

Piping Isometric Drawings User's Guide - Provides information about creating and managing piping isometric drawings.

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

Reports User's Guide - Provides information about creating and managing spreadsheet reports.

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 2D Symbols Reference Data Guide - Provides information about the two-dimensional symbols used in all tasks.

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

We welcome comments or suggestions about this documentation. You can send us an email at: PPMdoc@intergraph.com.

What's New in Structure

The following changes have been made to the Structure task.

Version 2007 Service Pack 5

- Added two new how-to topics to the documentation that describe how to place openings in leaning and sloped walls. For more information, see *Place an Opening on a Leaning Wall* (on page 142) and *Place an Opening on a Sloped Wall* (on page 141). (P3 CP:135894)

Version 2007 Service Pack 4

- You can now add a vertex to a handrail path.
- You can now split handrails by using the new **Split** turn type when editing the handrail path. (P2 CP:116723)
- Walls will now be exported to the Physical Model CIS/2 file if they are part of the filter to export. (P3 CP:123014)
- The software now exports the operation geometry of stairs, ladders, and handrails to the CIS/2 file.
- Added several new how-to workflows for walls.

Version 2007 Service Pack 3

- The software now attempts to automatically connect frame connections when you move, copy/paste with move, or mirror copy members. For more information, see *Understanding Framing Connections* (on page 11). (P1 CP:45434)
- You can now import frame connections from a CIS file if that information is in the CIS file. (P2 CP:122652)
- Unconstrained openings placed in a slab now maintain their position relative to the global origin when the boundaries of the slab change. (P2 CP:118339)
- You can now export slabs and openings to a CIS file. (P3 CP:78471)
- You can now import and export curved members from a CIS file. (P3 CP:77513;CP:123016)

Version 2007 Service Pack 2

- The Import Structure command now tracks Global User Identities (GUIDs) to uniquely identify objects and manage electronic exchange with the other software package. In addition, you can now preview the contents of the CIS/2 file for problems. (P2 CP:97244)
- The Import Structure command now imports stairs, ladders, and handrails from the CIS/2 file. (P2 CP:116317)
- The *Cross Section Tab (Member Part Prismatic Properties Dialog Box)* (on page 30) now displays the cross section properties. (P3 CP:53052)
- You can now place side step ladders. For more information, see *Place a Side Step Ladder* (on page 187). (P3 CP:69885)
- The **Place Wall** command now allows you to draw more than wall path when placing walls by 2D Sketch. In addition, the **Place Wall** command remembers the previously selected

sketch plane, allowing you to place multiple walls quickly. For more information, see *Place a Wall by 2-D Sketch* (on page 127).

Version 2007 Service Pack 1

- Footings can now be placed either before or after the columns are placed in the model. For more information, see *Place Footing Command* (on page 168). (P2 CP:41094)
- The **Split Member** command no longer splits a member system that has a **Continuity Type** of **Continuous**. Only members systems with **Intercostal** set for the **Continuity Type** are able to be split. (P3 112882)
- Documented all of the optional handrail properties. Before, only the required handrail properties were documented. (P3 112774)

Version 2007

- The Vertical Corner Brace frame connection now supports offsets in the X, Y, and Z directions.
- You can now place doors and windows in the model. For more information, see *Place Equipment Command* (on page 151).
- You can now place walls in the model. For more information, see *Place Wall Command* (on page 118).
- Member system coordinate locations (on the properties dialog box) are now displayed relative to the active coordinate system rather than the global coordinate system as before.
- A new **Convert** option on the modify member part ribbon allows you to "convert" a member part into an independent member system. For more information, see *Modify Linear Member Part Ribbon* (on page 19).
- The weight center-of-gravity locations are now displayed relative to the active coordinate system instead of the global coordinate system.
- Frame connections that position a supported member along a supporting member (Axis-Along, Seated, Flush, Centerline) now allow the position to be governed by either a Ratio or Distance along the supporting member. For example, you can now control whether chevron braces stay at the midpoint of a beam or slide along the beam when the beam moves up or down. For more information, see *General Tab (Frame Connection Properties Dialog Box)* (on page 34).
- You can now place handrails by selecting a member instead of defining a handrail path. For more information, see *Place Handrail by Member Command* (on page 197).
- A new placement method called **Number and Spacing** has been added to the **Place Framing Members** command. The new method allows you to define the number of, spacing between, and starting location of the framing members. For more information, see *Place Framing Members Ribbon* (on page 50).
- You can now place curved members using Sketch 3D. For more information, see *Place Curve Member Command* (on page 44).
- You can now place arc's in handrail and curved member's paths using the new **Arc by End Points** option on the create path ribbon.
- When defining a section name for a member, you can now type the section name instead of having to browse the Catalog for it. The wildcard * is supported.

- The General Surface assembly connection now supports curved surfaces.
- General and details reports have been added for stairs, ladders, and handrails.
- Objects (such as ladders and equipment foundations) are automatically migrated to the new SmartMarine 3D plate parts as these plates get split.
- Base plates can now be created from members on curved surfaces (member perpendicular to the hull).

SECTION 1

Structure

The Structure task places and modifies structural objects. Using this task, you can place beams, columns, braces, truss elements, cables, equipment foundations, column footings, openings, slabs, walls, and connections in your model. You also can create custom section shapes using 2D Symbols and place those custom sections in the model.

The Structure task also provides for traffic needs by placing stairs, ladders, and handrails.

NOTE Although not required, we recommend that you place grids using the Grids task before placing structural members.

The Structure task has these commands:


Select - Used to select objects in the model. For more information, see *Selecting Objects* (on page 5).


Place Linear Member System - Places columns, beams, braces, and other linear members in the model. For more information, see *Place Linear Member System Command* (on page 14).


Place Curve Member - Places a curved column, beam, or brace member in the model. For more information, see *Place Curve Member Command* (on page 44).


Place X Bracing - Places vertical cross bracing or vertical chevron bracing in the model. For more information, see *Place Bracing Command* (on page 46).


Place Framing Members - Places secondary framing members between two supporting members that you specify. For more information, see *Place Framing Members Command* (on page 50).


Place Columns at Grid Intersections - Places columns at grid intersections. For more information, see *Place Columns at Grid Intersections Command* (on page 55).


Place Vessel Supports - Places structural members that support vertical vessels directly (typically four lugs that rest on the members) or that support the grating around the vessel. For more information, see *Place Vessel Supports Command* (on page 57).


Place Assembly Connection - Places assembly connections between linear member systems. For more information, see *Place Assembly Connection Command* (on page 70).


Place Split - Divides a member system into multiple member parts while maintaining the design intent of the original member system. For more information, see *Place Split Command* (on page 64).


Trim Member - Manually copes, snips, and planar trims member ends. For more information, see *Trim Member Command* (on page 78).


Place Slab - Places slabs, plates, and grates in the model. For more information, see

Place Slab Command (on page 103).

Place Wall - Places a wall in the model. For more information, see *Place Wall Command* (on page 118).

Place Equipment - Places equipment in the model. In the Structure task, use this command to place doors and windows. For more information, see *Place Equipment Command* (on page 151).

Place Opening - Places openings (holes) in slabs and linear member systems. For more information, see *Place Opening Command* (on page 135).

Place Stair - Places stairs in the model. For more information, see *Place Stair Command* (on page 178).

Place Ladder - Places ladders in the model. For more information, see *Place Ladder Command* (on page 183).

Place Handrail - Places a handrail in the model following a path that you specify. For more information, see *Place Handrail Command* (on page 189).

Place Handrail by Member - Places a handrail on selected members without having to define a path. For more information, see *Place Handrail by Member Command* (on page 197).

Place Footing - Places a column footing in the model. For more information, see *Place Footing Command* (on page 168).

Place Equipment Foundation - Places a foundation for a selected piece of equipment. For more information, see *Place Equipment Foundation Command* (on page 160).

Import Structure - Imports a CIS/2 file into the model. This command is on the **File** menu. For more information, see *Import Structure Command* (on page 86).

Export Structure - Exports the structural physical model to a CIS/2 file. This command is on the **File** menu. For more information, see *Export Structure Command* (on page 85).

New Mapping File - Creates an XML mapping file for the section and material names used in the software and third-party application. This command is on the **File** menu. For more information, see *New Mapping File Command* (on page 91).

SECTION 2

Understanding the Structure Workflow

All structure objects are placed in the model using information defined in the structure reference data. Your first step should be to review, edit, and otherwise customize the delivered structure reference data using the Catalog task. Refer to the Catalog documentation. If you prefer, you can still review, edit, and create structure reference data using Microsoft Excel workbooks. Refer to the *Structure Reference Data Guide* for more information on editing reference data using workbooks.

After the reference data is customized to suit your needs, consider going to the Systems and Specifications task and defining the systems that you want in your model. While not absolutely required that you create your systems first, doing so keeps you from having to edit your structural objects after placement to assign them to the correct system. In addition, we recommend that you create your elevations and grids using the Grids task before placing structural objects.

After the structure reference data and the needed systems and grids are defined, you can begin placing structural objects in your model.

See Also

Structure Common Tasks (on page 3)

Structure Common Tasks

The following tasks are used frequently in the Structure task.

Customize Reference Data

Create new member types by editing the **AllCodeLists.xls** workbook. For more information, see the *Structural Reference Data Guide*.

Review, add, and edit material information. For more information, see the *Structural Reference Data Guide*.

Place Members

Place columns in the model. For more information, see *Place Members using Discrete Placement* (on page 37).

Place beams in the model. For more information, see *Place Members using Contiguous Placement* (on page 38).

Places braces in the model. For more information, see *Place Vertical Cross Bracing* (on page 49), *Place Vertical Chevron Bracing* (on page 49), and *Position Framing Members at Equal Spacing* (on page 54).

Split member systems into member part as needed to resolve interferences, place column splices, and other modeling issues. For more information, see *Split a Member that Intersects another Member* (on page 66) and *Split Columns at a Plane* (on page 67).

Place assembly connections in the model. For more information, see *Place an Assembly Connection* (on page 77).

Place Slabs

Place slabs in the model. The software provides great flexibility in placing slabs. For more information, see:

Place a Slab by Selecting Boundary Objects (on page 113)
Place a Slab by Drawing 2-D Boundaries (on page 114)
Place a Slab by Drawing 3-D Boundaries (on page 114)
Place a Slab by Selecting Objects and Drawing 2-D Boundaries (on page 115)
Place a Slab by Selecting Objects and Drawing 3-D Boundaries (on page 116)

Place Walls

Place walls in the model. The software provides great flexibility in placing walls. For more information, see:

Place a Wall by 2-D Sketch (on page 127)
Place a Wall by 3-D Sketch (on page 132)

Place Openings

Place openings in slabs, walls, and members in the model. You can define openings by boundaries, by drawing the opening, or by selecting a shape from the catalog for the opening. You can place doors and windows by selecting a door or window from the catalog.

For more information on placing an opening by boundaries, see *Place an Opening by Boundaries* (on page 139).

For more information on placing an opening by drawing the opening, see *Place an Opening by Drawing* (on page 140).

For more information on placing an opening by using a shape from the catalog, see *Place an Opening by Shape* (on page 140).

For more information on placing doors, see *Place Doors from the Catalog* (on page 154).

For more information on placing windows, see *Place Windows from the Catalog* (on page 155).

Model Foundations

Place equipment foundations in your model. For more information, see *Place an Equipment Foundation* (on page 163).

Place column footings in your model. For more information, see *Place a Single Footing* (on page 172) or *Place a Combined Footing* (on page 172).

Provide for Traffic

Place stairs, or incline ladders, in your model. For more information, see *Place a Stair* (on page 182).

Place ladders in your model. For more information, see *Place a Ladder* (on page 187).

Place handrails in your model. For more information, see *Place a Handrail* (on page 195) and *Place a Handrail by Member* (on page 199).

Exporting the Physical Model

When all the primary and secondary members are modeled, you may want to export the physical model to a detailing application. For more information, see *Export Structure Model* (on page 86).

Selecting Objects

All objects in the Structure task have properties that you can edit. Using the **Select** command on the vertical toolbar, you 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 **Member Systems**, you can select only member systems in a graphic view or in the **Workspace Explorer**.

The Structure task includes these filters:

Structure - Allows you to select any object in a graphic view or in the **Workspace Explorer** that was placed using the Structure task. Objects placed using other tasks, such as equipment, cannot be selected using this filter.

Member Systems - Limits your selection in a graphic view or in the **Workspace Explorer** to member systems. A member system contains one or more member parts. You can place member systems using the **Place Linear Member** or the **Place Curved Member** commands. For more information, see *Place Linear Member System Command* (on page 14) and *Place Curve Member Command* (on page 44).

TIP If you not sure whether to use **Member Systems** or **Member Parts** when you want to edit properties, try **Member Parts** first.

Member Parts - Limits your selection in a graphic view or in the **Workspace Explorer** to member parts.

Frame Connections - Limits your selection in a graphic view or in the **Workspace Explorer** to frame connections. For more information about these connections, see *Understanding Framing Connections* (on page 11).

Split Connections - Limits your selection in a graphic view or in the **Workspace Explorer** to split connections. For more information about these connections, see *Splitting Members* (on page 62).

Member Assembly Connections - Limits your selection in a graphic view or in the **Workspace Explorer** to member assembly connections. For more information about these connections, see *Understanding Member Assembly Connections* (on page 68) and *Place Assembly Connection Command* (on page 70).

Slabs - Limits your selection in a graphic view or in the **Workspace Explorer** to slabs. You can place slabs using the **Place Slab** command. For more information, see *Place Slab Command* (on page 103).

Slab Assembly Connections - Limits your selection in a graphic view or in the **Workspace Explorer** to slab assembly connections. For more information about these connections, see *Understanding Slab Assembly Connections* (on page 102).

Openings - Limits your selection in a graphic view or in the **Workspace Explorer** to openings (holes). You can place openings using the **Place Opening** command. For more information, see *Place Opening Command* (on page 135).

Wall Systems - Limits your selection in a graphic view or in the **Workspace Explorer** to wall systems. A wall system contains one or more wall parts. You can place wall systems using the **Place Wall** command. For more information, see *Place Wall Command* (on page 118).

Wall Parts - Limits your selection in a graphic view or in the **Workspace Explorer** to wall parts.

Equipment - Limits your selection in a graphic view or in the **Workspace Explorer** to equipment. Remember that doors and windows are considered pieces of equipment. You can place doors and windows using the **Place Equipment** command.

Stairs - Limits your selection in a graphic view or in the **Workspace Explorer** to stairs (incline ladders). You can place stairs using the **Place Stair** command. For more information, see *Place Stair Command* (on page 178).

Ladders - Limits your selection in a graphic view or in the **Workspace Explorer** to ladders. You can place ladders using the **Place Ladder** command. For more information, see *Place Ladder Command* (on page 183).

Handrails - Limits your selection in a graphic view or in the **Workspace Explorer** to handrails. You can place handrails using the **Place Handrail** command or the **Place Handrail by Member** command. For more information, see *Place Handrail Command* (on page 189) or *Place Handrail by Member Command* (on page 197).

Footings - Allows you to select footings in a graphic view or in the **Workspace Explorer**. You can place footings using the **Place Footing** command. For more information, see *Place Footing Command* (on page 168).

Equipment Foundations - Limits your selection in a graphic view or in the **Workspace Explorer** to equipment foundations. You can place equipment foundations using the **Place Equipment Foundation** command. For more information, see *Place Equipment Foundation Command* (on page 160).

Assembly Components - Limits your selection in a graphic view or in the **Workspace Explorer** to assembly components placed by an assembly connection or by an assembly footing or foundation.

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.

See Also

Split Connection Properties Dialog Box (on page 65)
Footing Properties Dialog Box (on page 170)
Handrail Properties Dialog Box (on page 190)
Frame Connection Properties Dialog Box (on page 34)
Ladder Properties Dialog Box (on page 185)
Member Part Prismatic Properties Dialog Box (on page 26)
Stair Properties Dialog Box (on page 180)
Equipment Foundation Properties Dialog Box (on page 161)
Slab Properties Dialog Box (on page 107)
Assembly Connection Properties Dialog Box (on page 73)

SECTION 3

Working with Members

You can place linear and curved members. All linear members are placed using the *Place Linear Member System Command* (on page 14)  . All curved members are placed using the *Place Curve Member Command* (on page 44)  . 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* (on page 55)  . If you want to place cross bracing, use the *Place Bracing Command* (on page 46)  . Use the *Place Framing Members Command* (on page 50)  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* (on page 57)  . 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* (on page 64) to split member systems into member parts.

Member systems connect to other member systems using Frame Connections. For more information, see *Understanding Framing Connections* (on page 11).

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* (on page 68).

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.

See Also

Place Members using Contiguous Placement (on page 38)
Place Members using Discrete Placement (on page 37)
Place a Member using Finish Mode (on page 38)
Place a Curved Member (on page 45)
Convert a Member Part (on page 40)
Delete a Member System (on page 40)
Edit Member Part Properties (on page 39)
Edit Member System Properties (on page 39)
Modify the Cardinal Point of a Member (on page 40)
Modify the Angle of a Member (on page 41)
Modify the Cross Section of a Member (on page 41)
Modify the End Releases of a Member (on page 41)
Modify the Material of a Member (on page 42)
Modify the Material Grade of a Member (on page 42)
Modify the Type of Member (on page 42)

Move a Member (on page 43)
Move One End of a Member (on page 43)

Understanding Framing Connections

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* (on page 43) and *Move a Member* (on page 43). Refer to *Working with Members* (on page 9) for important related information about member systems.

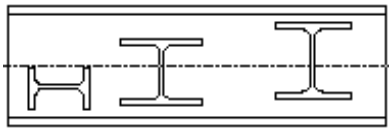
There are several basic types of frame connections:

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.

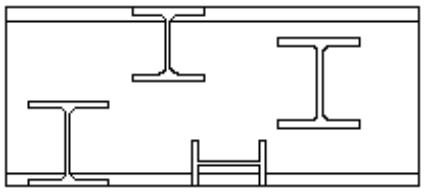
Axis End - An axis end frame connection aligns the cardinal point on the supported member system with the cardinal point on the supporting member system. Use this frame connection when both member systems are of the same type (both columns, or both beams). If you move one member system end, this frame 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.

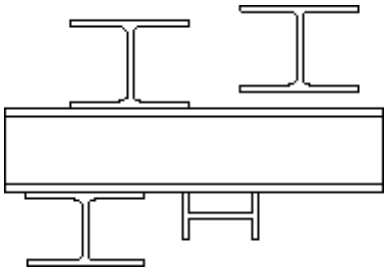
Centerline - A centerline frame connection uses the supporting member's centerline to position the supported member.



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.

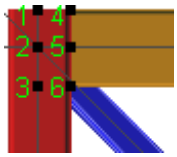


Seated - A seated frame connection uses the supporting member's top or bottom extent to position the supported member. The supported member typically rests against the supporting member, but can be offset.



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:

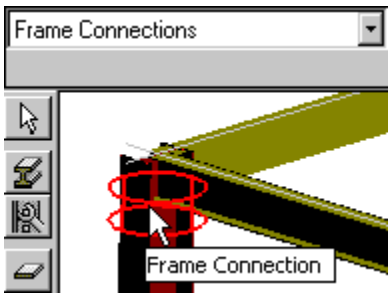
- When the member connects to non-member objects, the software select the "Unsupported" frame connection unless the non-member object is a surface, in which case "Surface-Default" is chosen.
- When a member connects to a single member, the software selects "Axis Along" unless:
- The two members have 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.



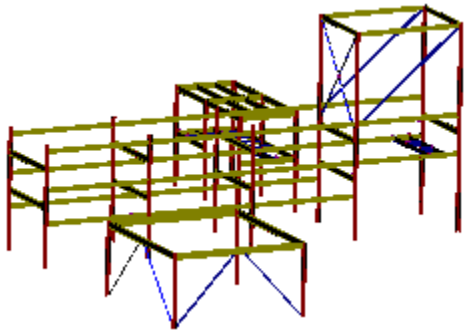
When you select a frame connection, the software displays the frame connection type in the ribbon. Select the **Edit > Properties** command to edit the frame connection properties. You cannot copy a frame connection using the **Edit > Copy** command.

See Also

Working with Members (on page 9)
Edit a Frame Connection (on page 40)

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* (on page 9). 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.

See Also

Place Linear Member System Ribbon (on page 15)
Modify Linear Member Part Ribbon (on page 19)
Working with Members (on page 9)
Place Members using Contiguous Placement (on page 38)
Place Members using Discrete Placement (on page 37)
Place a Member using Finish Mode (on page 38)
Convert a Member Part (on page 40)
Delete a Member System (on page 40)
Edit Member Part Properties (on page 39)
Edit Member System Properties (on page 39)
Modify the Cardinal Point of a Member (on page 40)
Modify the Angle of a Member (on page 41)
Modify the Cross Section of a Member (on page 41)
Modify the End Releases of a Member (on page 41)
Modify the Material of a Member (on page 42)
Modify the Material Grade of a Member (on page 42)
Modify the Type of Member (on page 42)
Move a Member (on page 43)
Move One End of a Member (on page 43)
Copy Framing Members (on page 55)
Modify Linear Member Part Ribbon (on page 19)
Place Linear Member System Ribbon (on page 15)

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* (on page 19).

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* (on page 9).

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* (on page 9).

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* (on page 11). 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* (see "Understanding Framing Connections" on page 11).

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* (on page 21).

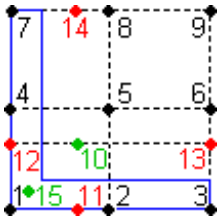
Type Category - Select the type category of the member that you want to place, such as a beam or a column. The properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** select list in the Catalog task.

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.

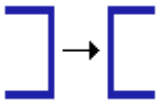
Section Name - Defines the cross-section for the member. If you know the section name that you want, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog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



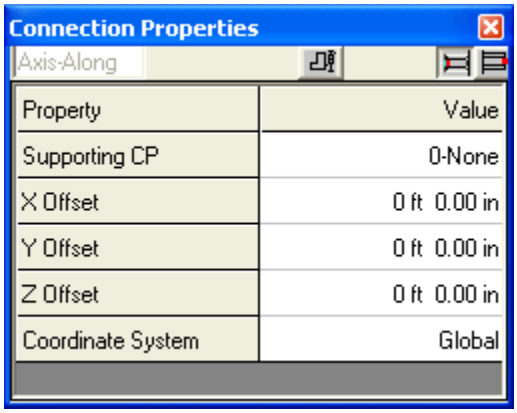
Angle - Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis depending on the member orientation.

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.



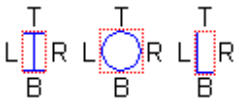
TIP The supported member is the member that you are placing. The supporting member is the existing member in the model *to which* you are connecting.

Seated, Flush, and Centerline Frame Connection Properties

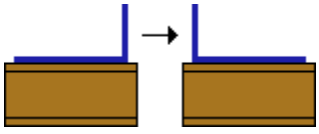
Side - Select the side of the supporting member on which you want to place the supported member.

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

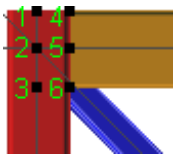
X Offset - Specifies the offset to apply in the x-direction.

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

See Also

Place Members using Contiguous Placement (on page 38)
Place Members using Discrete Placement (on page 37)
Place a Member using Finish Mode (on page 38)
Place Linear Member System Command (on page 14)

Modify Linear Member Part Ribbon

Displays the member part properties that you are edit when you select one or more linear member parts.

Member Properties - Activates the **Member Properties** dialog box. You can use this dialog box to specify additional member part properties, such as material, material grade, and end releases, which you cannot set on the ribbon. For more information, see *Member Part Prismatic Properties Dialog Box* (on page 26).

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* (on page 9).

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* (on page 9).

Convert - Translates a member part to have a single stand-alone member system. This option is only available when you select a member part that belongs to a member system that has been split.

When you place a member system, that member system has a single member part associated with it (for more information, see *Working with Members* (on page 9)). Using the *Place Split Command* (on page 64), you can split that single member part into multiple member parts that are each still associated with the original single member system.

ow you want to delete (or modify) just one of the member parts which you cannot do without deleting the member system and all the other member parts (deleting a member part deletes its parent member system which causes the sibling member parts to be deleted). However, using the **Convert** option you can cause a member part to have its own, new parent member system. You can then delete or otherwise modify that member part without affecting the other member parts of the original member system.

The software automatically translates the relevant split connections into frame connections. Permission groups of the original member system are used for the new member system.

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* (on page 21).

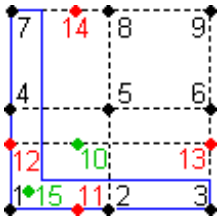
Type Category - Select the type category of the member that you want to place, such as a beam or a column. The properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** select list in the Catalog task.

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.

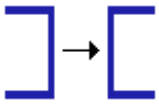
Section Name - Defines the cross-section for the member. If you know the section name that you want, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog 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.

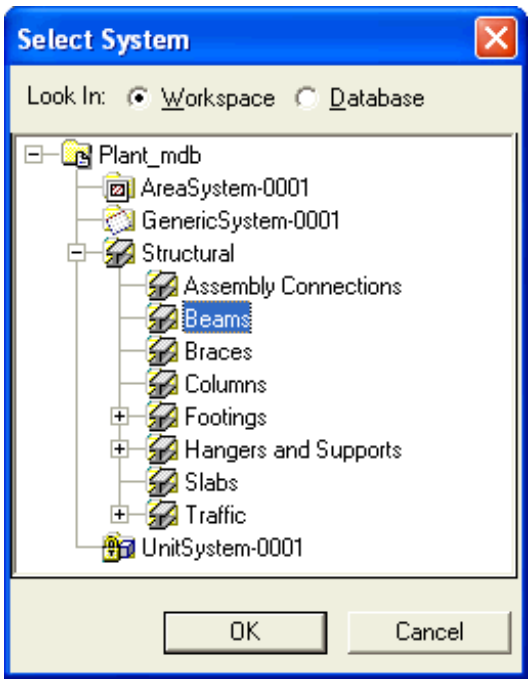


See Also

Convert a Member Part (on page 40)
Delete a Member System (on page 40)
Edit Member Part Properties (on page 39)
Edit Member System Properties (on page 39)
Modify the Cardinal Point of a Member (on page 40)
Modify the Angle of a Member (on page 41)
Modify the Cross Section of a Member (on page 41)
Modify the End Releases of a Member (on page 41)
Modify the Material of a Member (on page 42)
Modify the Material Grade of a Member (on page 42)
Modify the Type of Member (on page 42)
Move a Member (on page 43)
Move One End of a Member (on page 43)
Place Linear Member System Command (on page 14)

Select System Dialog Box

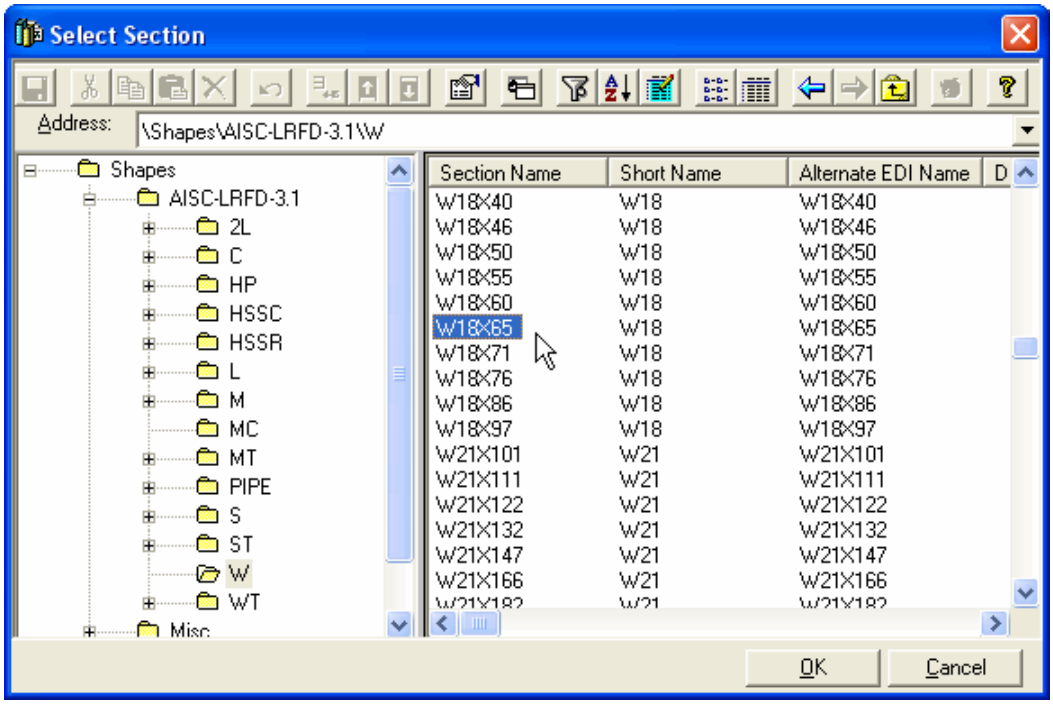
This dialog box displays when you select the **More..**. option in the **System** list. Use this dialog box to select the system that you want. You can create new systems in the Systems and Specifications task.



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* (on page 9).

See Also

Member System Tab (Member System Prismatic Properties Dialog Box) (on page 23)
Place Linear Member System Command (on page 14)
Configuration Tab (on page 24)
Relationship Tab (on page 24)
Notes Tab (on page 25)

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. You cannot split members that have a **Continuity Type** setting of **Continuous**.

Continuity Priority Number - Specify the continuity priority. This priority is used to select which member system is split when two member systems intersect, but both have **Intercostal** for the **Continuity Type**. 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.

See Also

Member System Prismatic Properties Dialog Box (on page 23)

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 model. 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.

Transfer - Re-assigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This button is only available if the active model/project is replicated in a workshare configuration. The button is not available if all of the

objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* (on page 25).

NOTE The **Transfer** option does not apply to the Surface Style Rules.

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.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location - Displays the name of the location that the current permission group is associated with. All of the objects in the select set must belong to the same location.

Current permission group - Displays the name of the permission group that the selected objects are currently associated with. If all of the objects in the select set do belong to the same permission group, this box appears blank.

New location - Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group - Specifies the new permission group to which you want to assign the selected objects. If you have specified a value in the **New location** box, this list displays all permission groups that you have write access to in the selected location. If you have not specified a value in the **New location** box, this list includes all permission groups that you have write access to in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE It is strongly recommended that administrators follow naming convention rules that include the location as a prefix in the permission group name.

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, then only one of the notes shows on the drawing. It is important to know about and to 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 that the note was created. The system automatically supplies the date.

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

Purpose of note - Specifies the purpose of the note.

Author - Displays the login 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.

Show dimension - Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

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 that 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* (on page 9).

See Also

Cross Section Tab (Member Part Prismatic Properties Dialog Box) (on page 30)
Member Part Tab (Member Part Prismatic Properties Dialog Box) (on page 26)
Place Linear Member System Command (on page 14)

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 Catalog task in the **Reporting Type** select list.

Piece Mark - Specifies the piece mark of the member. Piece marks are mainly used when importing and exporting member through CIMsteel to other software packages. If a piece mark is displayed here, more than likely it is the identity of the manufactured part from the other software package.

Assembly Mark - Specifies the assembly mark of the member. If the member was imported through CIMsteel, the assembly mark is more than likely the identity of the manufactured assembly to which this member belongs.

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.

See Also

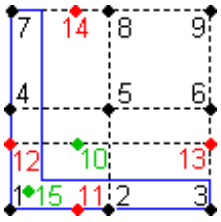
Member Part Prismatic Properties Dialog Box (on page 26)

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.

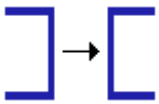
Section Name - Defines the cross-section for the member. If you know the section name that you want, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog 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.

Centroid X - Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Centroid Y - Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Moment of Inertia about X (Ixx) - Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Moment of Inertia about Y (Iyy) - Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Warping Statical Moment (Sw) - Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Elastic Section Modulus about X (Sxx) - Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Elastic Section Modulus about Y (Syy) - Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Torsional Moment of Inertia (J) - Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Warping Constant (Cw) - Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Flexural Constant (H) - Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro) - Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Radius of Gyration about X axis (Rxx) - Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy) - Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

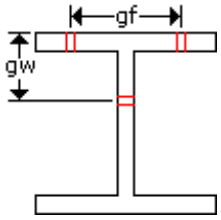
Radius of Gyration about Principle XY (Rxy) - Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Plastic Section Modulus about X (Zxx) - Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy) - Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Flange Gage (gf) - Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Gage (gw) - Displays the bolt gage for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.



Section Name - Displays the section name. This name appears when you label members. This property is read- only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Short Name - Displays the short name for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Alternate EDI Name - Displays the Electronic Data Interchange name for the section. This name is used when translating sections through CIMsteel. This property is currently not used.

Description - Displays the description. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Area - Displays the cross-section area for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Depth - Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Width - Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Perimeter - Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Unit Weight - Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Theoretical Maximum Yield Stress (Fy''') - Displays the maximum yield stress for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Point in Flange (Qf) - Displays the first moment of area for the flange. This property is read- only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Mid Depth of Section (Qw) - Displays the first moment of area for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Normalized Warping Function (Wno) - Displays the normalized warping function. The function is defined in square inches or square millimeters. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X1) - Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X2) - Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Thickness (tw) - Displays the web thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Depth (d) - Displays the web depth. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Flange Thickness (tf) - Displays the flange thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Flange Width (bf) - Displays the width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdetail) - Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdesign) - Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on the welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Group Id - Displays the material group identification for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

See Also

Member Part Prismatic Properties Dialog Box (on page 26)

Frame Connection Properties Dialog Box

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

See Also

General Tab (Frame Connection Properties Dialog Box) (on page 34)
Understanding Framing Connections (on page 11)
Edit a Frame Connection (on page 40)

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.

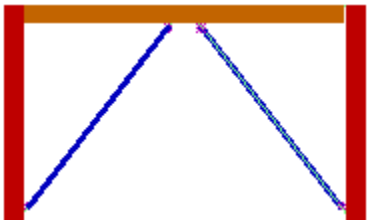


Figure 1: Original Position

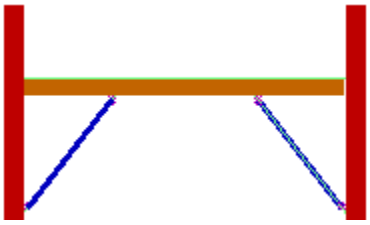


Figure 2: Intersection Position Rule

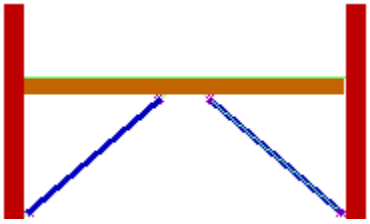


Figure 3: Ratio Position Rule

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**.

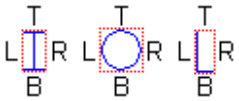
End - Specifies which end of the supporting member that the ratio or the distance-along distance is measured. You can select **Start**, **End**, or **Auto**. Start is the first member end that was placed. End is the second member end that was placed. If you select **Auto**, the software automatically 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

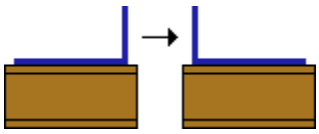
Side - Select the side of the supporting member on which you want to place the supported member.

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

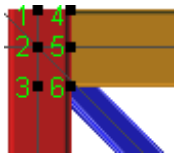
X Offset - Specifies the offset to apply in the x-direction.

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

See Also

Frame Connection Properties Dialog Box (on page 34)

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* (on page 11). 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.

See Also

Working with Members (on page 9)
Place Linear Member System Command (on page 14)

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* (on page 11). 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.

See Also

Working with Members (on page 9)
Place Linear Member System Command (on page 14)

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* (on page 11). 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.

See Also

Working with Members (on page 9)
Place Linear Member System Command (on page 14)

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.

See Also

Member System Prismatic Properties Dialog Box (on page 23)

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.

See Also

Member Part Prismatic Properties Dialog Box (on page 26)

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.

See Also

Understanding Framing Connections (on page 11)
Frame Connection Properties Dialog Box (on page 34)

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.

See Also

Place Linear Member System Command (on page 14)

Convert a Member Part

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member part to have its own member system.
4. Click **Convert** on the ribbon.

See Also

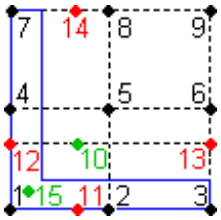
Place Linear Member System Command (on page 14)

Modify the Cardinal Point of a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to modify.

4. Using the **Cardinal Point** control on the ribbon bar, select a new cardinal point for the member part.

NOTE There are 15 cardinal positions 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.



See Also

Place Linear Member System Command (on page 14)

Modify the End Releases of a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Member Properties** on the ribbon.
5. Select the **Member Part** tab.
6. In the **Category** box, select **End Releases**.
7. Modify the member part end releases as required.

See Also

Place Linear Member System Command (on page 14)

Modify the Angle of a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Angle** control on the ribbon bar, edit the member angle.

See Also

Place Linear Member System Command (on page 14)

Modify the Cross Section of a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.

3. Select a member to modify.
4. Using the **Section name** box on the ribbon bar, type the new cross-section name for the member or select **More** to select the cross-section from the Catalog.

See Also

Place Linear Member System Command (on page 14)

Modify the Material of a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Material** property.

See Also

Place Linear Member System Command (on page 14)

Modify the Material Grade of a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Grade** property.

See Also

Place Linear Member System Command (on page 14)

Modify the Type of Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Type Category** and **Type** controls on the ribbon bar, change the type of the member part.

NOTE If the member type category or type that you want to use is not available, you can add it by editing the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook and bulkloading it into the Catalog database. Refer to the *SmartPlant 3D Reference Data Guide* for more information on editing the workbook and bulk loading.

See Also

Place Linear Member System Command (on page 14)

Move a Member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to move.
4. Click **Move** on the main ribbon.
5. Define the first point of a vector used to move the member.
6. Define the second point of the vector.

NOTE The frame connections may prevent you from moving the member as you would like. For example, a supported member has a seated frame connection to a supporting member. The seated frame connection prevents you from moving the supported member off of the supporting member. You can, however, slide the supported member along the supporting member as long as the seated frame connection is still valid. Consider copying and pasting a member that you want to move, and then deleting the original.

See Also

Place Linear Member System Command (on page 14)

Move One End of a Member

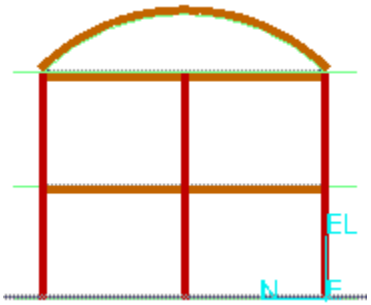
1. Click **Select** on the vertical toolbar.
2. Select **Frame Connections** in the **Locate Filter**.
3. Select the frame connection that is near the member end that you want to move.
4. In the **Type** option, select **By Rule**.
5. Define the new location for the member end.

See Also

Place Linear Member System Command (on page 14)

Place Curve Member Command

Places a curved member in the model. You can place beams, columns, or braces member types using this command. For additional information, see *Working with Members* (on page 9). Use this command when you want to place curved 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.

NOTE You cannot place loads on curved members, export curved members to a CIS file, or define frame connection offsets for curved members.

See Also

Place Curve Member Ribbon (on page 44)
Working with Members (on page 9)
Place a Curved Member (on page 45)
Place Curve Member Ribbon (on page 44)

Place Curve Member Ribbon

Specifies the properties for the curved member that you are placing.

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

Path - Activates the *Create Path Ribbon* (on page 202) with which you define the curved member path.

Finish - Click to place the member in the model.

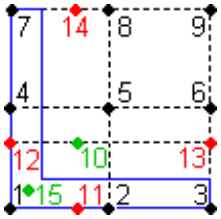
System - Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* (on page 21).

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.

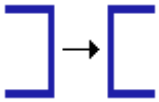
Section Name - Defines the cross-section for the member. If you know the section name that you want, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog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



See Also

Place a Curved Member (on page 45)
Place Curve Member Command (on page 44)

Place a Curved Member

1. Click **Place Curve Member** on the vertical toolbar.
2. Select an end location of the curved member.

   TIP You can select an existing member's frame connections to define the location, or define the location using a SmartSketch relationship such a Point Along.
3. Set **Path Type** to **Arc by End Points**.
4. Optionally, set the **Plane** option to match the plane of your curved member.
5. Select the opposite end location of the curved member.
6. Select a point along the curved member or define the radius or sweep of the curved member.

7. Click **Finish**.
8. Select a system for the curved member.
9. Select a member type and type category for the member.
10. Select a section size and cardinal point.
11. Click **Finish**.

See Also

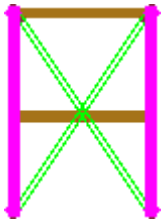
Place Curve Member Command (on page 44)
Define a Path (on page 209)
Sketching in 3-D (on page 201)

Place Bracing Command

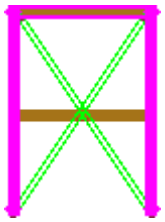
Places cross bracing between members that you select in the model. You can place vertical cross braces or vertical chevron braces with this command. Although the two cross bracing members are placed at the same time, after placement the two members can be edited or deleted individually if needed. For additional information about members, see *Working with Members* (on page 9).

Vertical X Bracing

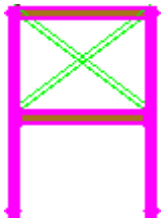
At a minimum, you must select two columns to define the cross bracing location. The two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. In this case, both ends of the braces are connected to the columns using the Axis-Along frame connection. (For more information about frame connections, see *Understanding Framing Connections* (on page 11).) In addition, the cross brace members are always placed with the starting end of the member at the lower elevation.



If you select two columns and a beam to define the cross bracing location, the bracing is placed at the intersection of the beam and the two columns to the two column bases. Again, the two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. In this case, the upper ends of the braces are connected to the columns and the beam using the Vertical Corner Brace frame connection. The bottom ends of the braces are connected to the column using the Axis-Along frame connection. Again, the cross brace members are always placed with the starting end of the member at the lower elevation.

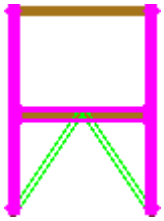


If you select two columns and two beams to define the cross bracing location, the bracing is placed in the intersection of the four members. Again, the two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. In this case, both ends of the braces are connected to the columns and the beams using the Vertical Corner Brace frame connection. Again, the cross brace members are always placed with the starting end of the member at the lower elevation.

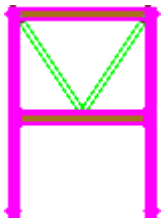


Vertical Chevron Bracing

At a minimum, you must select two columns and one beam to define the vertical chevron bracing location. The two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. The chevron braces are connected to the columns using the Vertical Corner Brace frame connection. The beam mid point is located to position the braces.



Optionally, you can select two columns and two beams to define the vertical chevron bracing location. The two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. The chevron braces are connected to the columns using the Vertical Corner Brace frame connection. The beam mid point is located to position the braces.



See Also

Place Bracing Ribbon (on page 47)
Working with Members (on page 9)
Place Vertical Chevron Bracing (on page 49)
Place Vertical Cross Bracing (on page 49)

Place Bracing Ribbon

Specifies the properties for the cross braces members that you are placing or editing.

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.

Select First Column - Select the first column that you want the cross bracing between.

Select Second Column - Select the second column that you want the cross bracing between. This column must be coplanar with the first column.

Select First Beam - Select the first beam that you want the cross bracing between.

Select Second Beam - Select the second beam that you want the cross bracing between.

Finish - Click to place the cross bracing members in the model.

Bracing Type - Specifies the type of bracing to place. Select **Cross** to place two vertical braces that cross. Select **Chevron** to place two vertical braces as a chevron brace.

Connection - Select the frame connection type to use for the cross bracing members 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* (on page 11).

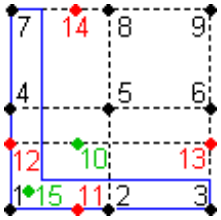
System - Select the system to which the cross bracing members belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* (on page 21).

Type Category - Select the type category of the cross bracing member that you want to place, such as brace. 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 cross bracing member that you want to place, such as a brace or vertical 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.

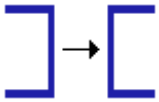
Section Name - Defines the cross-section for the member. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See the *Structure Reference Data Guide* for more information about reference data.

Cardinal Point - Specify the relative position of the structural section to the cross bracing 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.



See Also

Place Bracing Command (on page 46)

Place Vertical Cross Bracing

1. Click **Place X Bracing** on the vertical toolbar.
2. In the **Bracing Type** box, select **Cross**.
3. In the **Type category** box, select the cross bracing member type category to place.
4. In the **Type** box, select the cross bracing member type to place.
5. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.
6. Select the first column to support the cross bracing.
7. Select the second column to support the cross bracing.
8. Optionally, select one or two beams to support the cross bracing.
9. Click **Finish**.

See Also

Place Bracing Command (on page 46)

Place Vertical Chevron Bracing

1. Click **Place X Bracing** on the vertical toolbar.

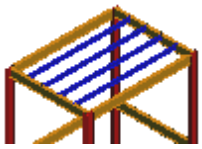
2. In the **Bracing Type** box, select **Chevron**.
3. In the **Type category** box, select the chevron bracing member type category to place.
4. In the **Type** box, select the chevron bracing member type to place.
5. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.
6. Select the first column to support the chevron bracing.
7. Select the second column to support the chevron bracing.
8. Select one beam to support the chevron bracing.
9. Optionally, select a second beam.
10. Click **Finish**.

See Also

Place Bracing Command (on page 46)

Place Framing Members Command

Places framing, secondary members, between two main supporting members typically to support a floor. The two supporting member must be coplanar but are not required to be of equal length or even the same member type thus providing lots of flexibility during placement. After placement, the framing members can be edited or deleted individually if needed.



For additional information about members, see *Working with Members* (on page 9).

See Also

Place Framing Members Ribbon (on page 50)
Working with Members (on page 9)
Position Framing Members using Best Fit (on page 53)
Position Framing Members using Count Method (on page 53)
Position Framing Members at Equal Spacing (on page 54)
Position Framing Members using Number and Spacing Method (on page 54)

Place Framing Members Ribbon

Specifies the properties for the framing members that you are placing or editing.

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

Select Supporting Member 1 - Select the first supporting member for the framing members.

Select Supporting Member 2 - Select the second supporting member for the framing members.

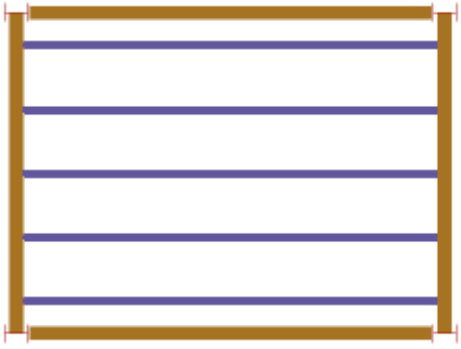
Select Start Point - Select the location of the first framing member along the first supporting member. This option is available only when you set **Placement** to **Number and Spacing**.

Finish - Click to place the framing members in the model.

Placement - Select how you want the software to place the framing members between the two supporting members.

- **By Count** - Select this option if you want to specify the exact number of framing members to place. Type the number of framing members in the **Count** box. The spacing between the framing members is determined by the length of the supporting member divided by the number of framing members plus one.
- **Equal Spacing** - Select this option to have the software determine the number of framing members to place based on the length of the supporting members and the value you specify in the **Maximum Spacing** box. The software places as many framing members as needed so that the spacing between the framing members does not exceed the **Maximum Spacing** value. All framing members are equally spaced.

- **Best Fit** - Select this option to have the software determine the number of framing members to place based on the length of the supporting members and the value you specify in the **Maximum Spacing** box. The software places all framing members except for the first and the last with a spacing equal to the **Maximum Spacing** value you specified. The spacing for the first and last framing member is automatically determined by the software.



- **Number and Spacing** - Select this option to place the exact number of framing members that you specify in the **Count** box the same distance apart that you define in the **Maximum Spacing** box. In addition, you can optionally specify the location of the first framing member along the first selected supporting member using **Select Start Point**. The supporting member end to measure from is the nearest end when you selected the supporting member.

Maximum Spacing - Type the framing member spacing. If **Placement** is set to **Equal Spacing**, then this value is the maximum spacing that you will allow between the framing members. If **Placement** is set to **Best Fit** or **Number and Spacing**, then this value is used as the exact spacing between the interior framing members. This option is not available when **Placement** is set to **By Count**.

Count - Type the number of framing members to place. This option is available only when **Placement** is set to **By Count** or **Number and Spacing**.

Connection - Select the frame connection type to use for the framing members 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* (on page 11).

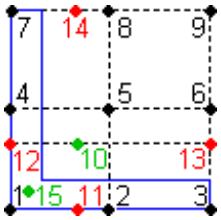
System - Select the system to which the framing members belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* (on page 21).

Type Category - Select the type category of the framing member that you want to place, such as brace. 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.

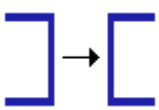
Type - Select the type of framing member that you want to place, such as a brace or horizontal brace. The properties change depending on the member type that you select. You can define a custom member type by editing the **Structural Member Type** select list in the Catalog task.

Section Name - Defines the cross-section for the member. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See the *Structure Reference Data Guide* for more information about reference data.

Cardinal Point - Specify the relative position of the structural section to the framing 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.

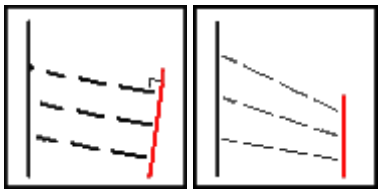


Reflect - Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction.



Offset - Specifies the framing member's offset from the supporting member's plane. This option is only available when **Connection** is set to a flush, seated, or centerline frame connection.

Position - Select how you want the framing members positioned with the two supporting members are not of equal length or are not parallel. Your choices are to make the framing members perpendicular to the first supporting member that you selected, or to skew the framing members by the ratio difference of the two supporting members.



Perpendicular By Ratio

Side - Select to flip the framing members to the opposite side.

See Also

Place Framing Members Command (on page 50)

Position Framing Members using Best Fit

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **Best Fit**.
3. In the **Max Spacing** box, type the distance to separate the interior framing members. The distance for the first and last framing members is determined by the software.
4. In the **Type category** box, select the framing member type category to place.
5. In the **Type** box, select the framing member type to place.
6. Set the **Section Name**, **Cardinal Point**, and **Offset** options as required.
7. Select the first supporting member.
8. Select the second supporting member.
9. Click **Finish**.

See Also

Place Framing Members Command (on page 50)

Position Framing Members using Count Method

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **By Count**.
3. In the **Count** box, type the number of framing members to place.
4. In the **Type category** box, select the framing member type category to place.
5. In the **Type** box, select the framing member type to place.
6. Set the **Section Name**, **Cardinal Point**, and **Offset** options as required.
7. Select the first supporting member.

8. Select the second supporting member.
9. Click **Finish**.

See Also

Place Framing Members Command (on page 50)

Position Framing Members at Equal Spacing

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **Equal Spacing**.
3. In the **Max Spacing** box, type the maximum distance that you will allow between the framing members. The distance between the framing members will be equal to or less than this distance.
4. In the **Type category** box, select the framing member type category to place.
5. In the **Type** box, select the framing member type to place.
6. Set the **Section Name**, **Cardinal Point**, and **Offset** options as required.
7. Select the first supporting member.
8. Select the second supporting member.
9. Click **Finish**.

See Also

Place Framing Members Command (on page 50)

Position Framing Members using Number and Spacing Method

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **Number and Spacing**.
3. In the **Count** box, type the number of framing members to place.
4. In the **Maximum Spacing** box, type the distance between the framing members.
5. In the **Type category** box, select the framing member type category to place.
6. In the **Type** box, select the framing member type to place.
7. Set the **Section Name**, **Cardinal Point**, and **Offset** options as required.
8. Select the first supporting member. If you are going to define a starting location for the framing members, select the first supporting member near the end from which you want to define the framing member's starting location.
9. Select the second supporting member.
10. Optionally, select the starting location of the first framing member along the first supporting member.
11. Click **Finish**.

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

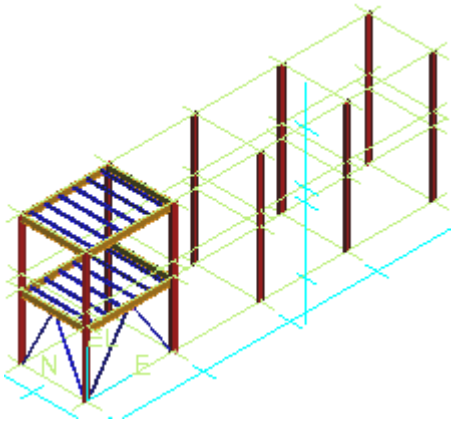
- Assembly Connections are not copied to the pasted members.
- If you do not select new dependencies for the pasted members in the **Paste** dialog box, the pasted members are placed in the same location as the copied members.

See Also

Working with Members (on page 9)
Place Linear Member System Command (on page 14)

Place Columns at Grid Intersections Command

Places columns between two selected elevations at selected grid intersections. You must have two elevation planes and one grid plane defined before you can use this command. Use the Grids task to create the planes. After placement, the columns can be edited or deleted individually if needed.



For additional information about members, see *Working with Members* (on page 9).

Place Columns at Grid Intersections Ribbon

Specifies the properties for the columns that you are placing or editing.

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

Select Elevation Plane 1 - Specify the bottom elevation for the columns.

Select Elevation Plane 2 - Specify the top elevation for the columns.

Select Grid Intersections - Drag a fence around all the grid intersections where you want columns placed. You can drag multiple fences if needed.

Finish - Click to place the columns in the model.

Cancel - Dismisses the active selection.

Accept - Accepts the active selection.

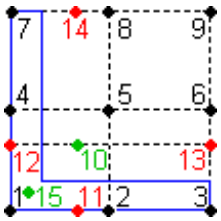
System - Select the system to which the columns belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* (on page 21).

Type Category - Select the type category of the columns that you want to place. 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 columns that you want to place. 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.

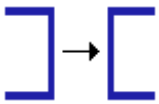
Section Name - Defines the cross-section for the member. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See *Structure Reference Data Guide* for more information about reference data.

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.



See Also

Place Columns at Grid Intersections Command (on page 55)

Place Columns at Grid Intersections

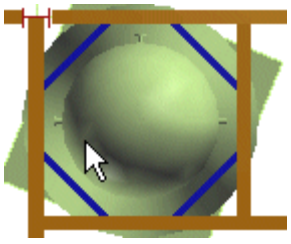
1. Click **Place Columns at Grid Intersections** on the vertical toolbar.
2. In the **Type category** box, select the member type category to place.
3. In the **Type** box, select the member type to place.
4. Select the bottom elevation plane for the columns.
5. Select the top elevation plane for the columns.
6. Drag a fence around the grid intersections where you want columns.
7. Click **Accept** ✓.
8. Click **Finish**.

See Also

Place Columns at Grid Intersections Command (on page 55)

Place Vessel Supports Command

Places structural members that support vertical vessels directly (typically four lugs that rest on the members) or that support the grating around the vessel. This command considers the vessel diameter and clearance, or the bolt circle diameter (lug hole-to-lug hole diameter) to determine member positions. Based on the cross section type chosen (I-section or channel), the command also considers the gage (the distance to the bolt hole location) in positioning the member. Because the equipment is often not centered within the main framing of the opening, the command accommodates such conditions and displays appropriate messages if the framing is not feasible. This command is currently limited to configurations where the lugs are positioned North, South, East, and West, or at 45 degrees to these axes.



For additional information about members, see *Working with Members* (on page 9).

See Also

Place Vessel Supports Ribbon (on page 58)
Working with Members (on page 9)
Support Vessels with 0 Degree Oriented Lugs (on page 61)
Support Vessels with 45 Degree Oriented Lugs (on page 61)

Place Vessel Supports Ribbon

Specifies the properties for the vessel support members that you are placing or editing.

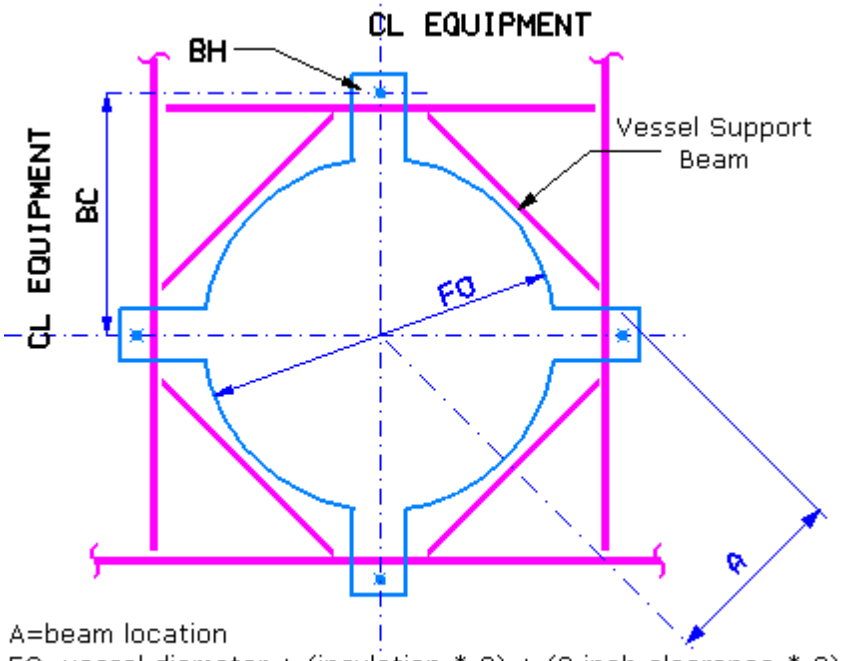
Select Vessel - Select the vessel around which to place the support members.

Select Members - Select the primary supporting members from which the vessel support members are placed.

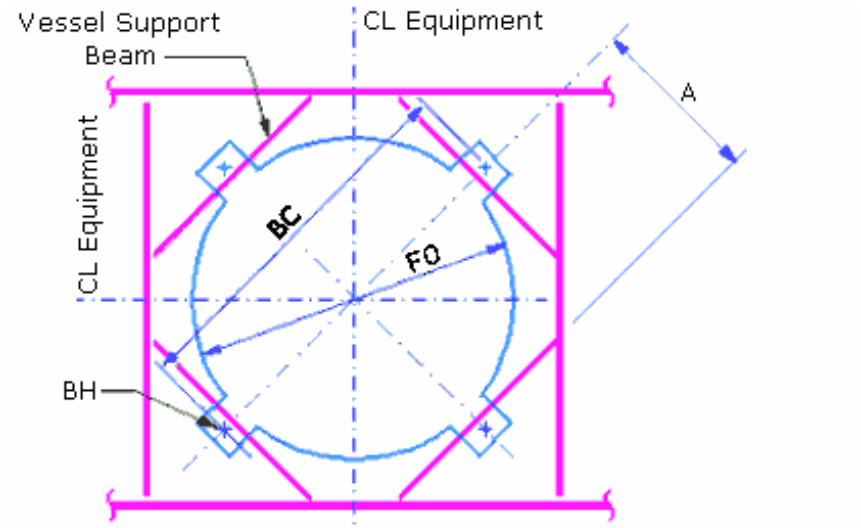
Finish - Click to place the vessel support members in the model.

Lug Orientation - Select the orientation of the vessel lugs relative to the coordinate system north.

Select **0 Degrees** if the vessel lugs are located like those in the figure below:



Select **45 Degrees** if the vessel lugs are located like those in the figure below:



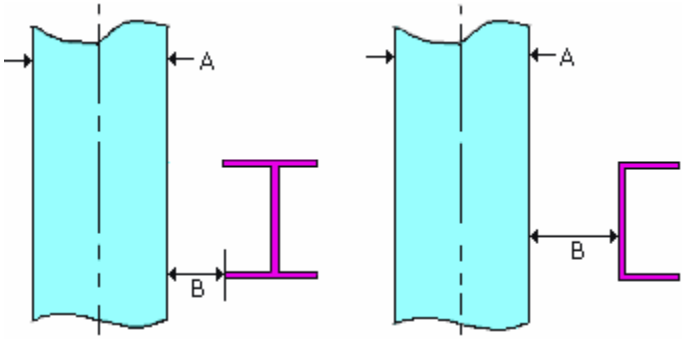
A=beam location
FO=vessel diameter + (insulation * 2) + (2 inch clearance * 2)
BC=bolt circle
BH=bolt hole diameter

Easting - Specifies the easting coordinate of the vessel center. If you select a vessel, the software automatically finds this value for you. If you are placing supports before the vessel is modeled, type the easting coordinate of the vessel in this box.

Northing - Specifies the northing coordinate of the vessel center. If you select a vessel, the software automatically finds this value for you. If you are placing supports before the vessel is modeled, type the northing coordinate of the vessel in this box.

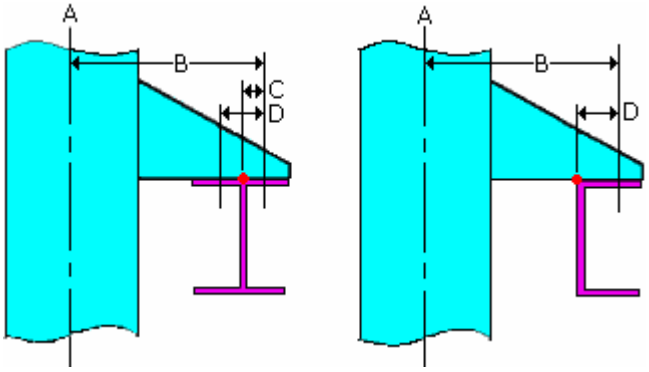
Vessel Diameter - Specifies the vessel diameter. If you select a vessel, the software automatically enters the diameter for you. If you are placing supports before the vessel is modeled, type the vessel diameter in this box. This option is only available when **Lug Orientation** is set to **0 Degrees**.

Clearance - Specifies the distance between the vessel support members and the vessel itself. This option is only available when **Lug Orientation** is set to **0 Degrees**.



A=Vessel Diameter
B=Clearance

Bolt Circle - Specifies the distance between the bolts on the opposite sides of the vessel. This value is used to calculate the member line location (red dot in figure below, figure shows cardinal point 8 being used for the I-beam and cardinal point 7 for the channel). This option is only available when **Lug Orientation** is set to **45 Degrees**.



A=Vessel Centerline
B=Bolt Circle divided by 2
C=Bolt Gage divided by 2
D=Bolt Gage

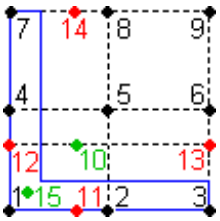
System - Select the system to which the vessel support members belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* (on page 21).

Type Category - Select the type category of the vessel support members that you want to place, such as brace. 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 vessel support member that you want to place, such as a brace or horizontal 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. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See the *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.



Support Vessels with 0 Degree Oriented Lugs

1. Click **Place Vessel Supports** on the vertical toolbar.
2. Select the vessel around which to place the supports.
   -OR-
   Key in the Easting and Northing coordinates of the vessel's center.
3. Set **Lug Orientation** to **0 Degrees**.
4. Key in the clearance between the side of the vessel and the support members being placed.
5. Select a system, type, section, and cardinal point for the support members. This command only supports W, M, HP, S, C, and MC sections.
6. Select the four main support members.
7. Click **Finish**.

See Also

Place Vessel Supports Command (on page 57)

Support Vessels with 45 Degree Oriented Lugs

1. Click **Place Vessel Supports** on the vertical toolbar.
2. Select the vessel around which to place the supports.
   -OR-
   Key in the Easting and Northing coordinates of the vessel's center.
3. Set **Lug Orientation** to **0 Degrees**.
4. Key in the bolt circle diameter.
5. Select a system, type, section, and cardinal point for the support members. This command only supports W, M, HP, S, C, and MC sections.
6. Select the four main support members.
7. Click **Finish**.

See Also

Place Vessel Supports Command (on page 57)

Splitting Members

When splitting members the software divides a member system into multiple member parts. The resulting member system is a set of continuous member parts that move as a single entity. This splitting is useful when you want to resolve the interference between two intersecting structural objects, such a pair of cross braces, or when you want to split the columns in your model at certain elevations. You split member systems at another member system, a grid plane, or an elevation plane.

Member End Releases

If a column is split, the upper end of the lower column and the lower end of upper column will have all end releases fixed. The lower end of the lower column and the upper end of the upper column do not change.

For beams and braces that are split, end releases on either side of the split will be pinned for rotation (RY and RZ). The free end of a cantilevered beam or brace has fully fixed member end releases.

Boundary Conditions

Boundary Conditions remain on the ends for which they were defined. For example, if end 1 of the original member was fully supported (X, Y, Z, RX, RY, RZ) and end 2 was supported only in RX, and RZ. Then end 1 of new member 1 gets fully supported and end 2 of new member 2 gets the RX and RZ support. The new interior ends (end 2 for member 1 and end 1 for member 2) do not have any supports defined.

For more information about boundary conditions, refer to the Structural Analysis task documentation.

Loads

Concentrated loads stay where they were placed (the physical location along the original member part length). The absolute or relative placement value is recalculated based on the new member part length.

Distributed loads are split into two distributed loads with the same magnitude (one distributed load for each new member part).

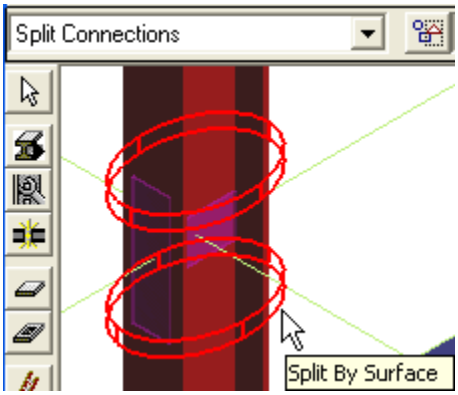
Partially distributed loads do one of two things based on the split location. If the split location is between the partially distributed load's end points, the load is split into two partially distributed loads one on each new member part on either side of the split location.

If the split location is outside of the partially distributed load's end points, the load is unaffected other than having the end points' absolute or relative placement values recalculated based on the new member part length.

For more information about loads, refer to the Structural Analysis task documentation.

Locating Split Connections

Split connections do not display in the model. However, if you set the **Locate Filter** to **Split Connections**, you can locate and select split connections for review and editing. Split connections are located at the ends of member parts where the split is located and appear as circles when you move the cursor over them.



When you select a split connection, the software displays the parent member system of the split connection in the ribbon. Select the **Edit > Properties** command to edit the split connection properties.

See Also

Remove a Split (on page 67)
Set Member Split Priority (on page 67)
Split Columns at a Plane (on page 67)
Split a Cross Brace (on page 66)
Split a Member that Intersects another Member (on page 66)
Place Split Command (on page 64)

Place Split Command

Divides a member system into multiple member parts. The resulting member system is a set of continuous member parts that move as a single entity. This splitting is useful when you want to resolve the interference between two intersecting structural objects, such a pair of cross braces, or when you want to split the columns in your model at certain elevations.

Only member systems can be split using the command. However, the object used to split the member system can be another member system, a grid plane, or an elevation plane.

To reconnect a split member system, simply delete the split connections.

If after you split a member system into multiple member parts you want to delete an individual member part, you will need to create member system parent for that member part using the **Convert** option. Set the locate filter to **Member Parts** and select the member part, then click **Convert** on the modify ribbon. For more information, see *Modify Linear Member Part Ribbon* (on page 19).

See Also

Place Split Ribbon (on page 64)
Remove a Split (on page 67)
Set Member Split Priority (on page 67)
Split Columns at a Plane (on page 67)
Split a Cross Brace (on page 66)
Split a Member that Intersects another Member (on page 66)
Place Split Ribbon (on page 64)

Place Split Ribbon

Specifies the properties for the split that you are placing or editing.

Split Properties - Activates the **Split Properties** dialog box. For more information, see *Split Connection Properties Dialog Box* (on page 65).

Split Members - Select the members to split. Remember, you cannot split members that have **Continuous** set for the **Continuity Type**. You can split only members set to **Intercostal**.

Splitting Members - Select the objects at which to split the member systems. The objects can be another member system, a grid plane, or an elevation plane.

Finish - Splits the member systems using the options that you have specified.

Cancel - Clears the selected objects.

Accept - Accepts the selected objects and move you to the next step in the command.

Split Status - Specifies the method to determine which member system splits which member system.

- **By Rule** - The software looks at the **Continuity Type** and **Continuity Priority Number** values defined on the *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 23) to determine which member system to split. Member systems set to **Continuous** will split the other member system. Member systems set to **Intercostal** will be

split by the other member system. The **Continuity Priority Number** is used to select which member system is split when two member systems intersect, but both have **Intercostal** for the **Continuity Type**. 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).

- **Split First** - The member systems that you identified during the **Split Members** step are split by the member systems that you identified during the **Splitting Members** step.
- **Split Second** - The member systems that you identified during the **Splitting Members** step are split by the member systems that you identified during the **Split Members** step.
- **Split Both** - All member systems that you have identified in both steps are split against each other.
- **Split None** - No member systems are split, but the interference between the two objects is suppressed.

See Also

Remove a Split (on page 67)
Set Member Split Priority (on page 67)
Split Columns at a Plane (on page 67)
Split a Cross Brace (on page 66)
Split a Member that Intersects another Member (on page 66)
Place Split Command (on page 64)

Split Connection Properties Dialog Box

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

See Also

General Tab (Split Connection Properties Dialog Box) (on page 65)

General Tab (Split Connection Properties Dialog Box)

Specifies the properties for the split connection.

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

Name - Specifies the name of the split connection.

Name Rule - Select the name rule to use to name the split connection. To name the split connection yourself, select **User Defined**.

Parent - Displays the parent member system to which the split belongs.

Split Parent Status - Specifies the method to determine which member system splits which member system.

- **By Rule** - The software looks at the **Continuity Type** and **Continuity Priority Number** values defined on the *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 23) to determine which member system to split. Member systems set to **Continuous** will split the other member system. Member systems set to **Intercostal** will be split by the other member system. The **Continuity Priority Number** is used to select which member system is split when two member systems intersect, but both have **Intercostal** for

the **Continuity Type**. 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).

- **Split First** - The member systems that you identified during the **Split Members** step are split by the member systems that you identified during the **Splitting Members** step.
- **Split Second** - The member systems that you identified during the **Splitting Members** step are split by the member systems that you identified during the **Split Members** step.
- **Split Both** - All member systems that you have identified in both steps are split against each other.
- **Split None** - No member systems are split, but the interference between the two objects is suppressed.

Offset - Specify the offset from the intersecting plane or surface to place the split. You can specify a positive or negative number. This option is only available after the split connection is place when defining a split using a grid plane, elevation plane, or surface as the splitting object.

See Also

Split Connection Properties Dialog Box (on page 65)

Split a Member that Intersects another Member

1. Click **Place Split** on the vertical toolbar.
2. In the **Split Status** box, select **Split First**.

   TIP If the **Continuity Type** and **Continuity Priority Number** values have been set for the member systems, you can use the **By Rule** option to have the software automatically determine which member system to split. For more information, see *Place Split Ribbon* (on page 64).
3. Select the member to split.
4. Click **Accept** ✓.
5. Select the splitting member.
6. Click **Finish**.

See Also

Place Split Command (on page 64)
Splitting Members (on page 62)

Split a Cross Brace

1. Click **Place Split** on the vertical toolbar.
2. In the **Split Status** box, select **Split First**.
3. Select the cross brace to split.
4. Click **Accept** ✓.
5. Select the other cross brace.
6. Click **Finish**.

Split Columns at a Plane

1. Click **Place Split** on the vertical toolbar.
2. In the **Split Status** box, select **By Rule**.
3. Select the columns to split.
4. Click **Accept**.
5. In the **Workspace Explorer**, select the elevation plane at which to split the columns.
   TIP You can also select the elevation plane by selecting it on the Z-axis ruler. Click **View > Rulers** to turn on rulers
6. Click **Finish**.

NOTE Use the Grids task to create elevation planes.

See Also

Place Split Command (on page 64)
Splitting Members (on page 62)

Set Member Split Priority

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Member Systems**.
3. Select the member system to edit.
4. Click **Properties**.
5. Set the **Continuity Type** and **Continuity Priority Number** options as required.
6. Click **OK**.

See Also

Place Split Command (on page 64)
Splitting Members (on page 62)
Member System Tab (Member System Prismatic Properties Dialog Box) (on page 23)

Remove a Split

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Split Connections**.
3. In a graphic view, select the split to remove.
4. Click **Delete**.

See Also

Place Split Command (on page 64)
Splitting Members (on page 62)

Understanding Member Assembly Connections

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* (on page 78).

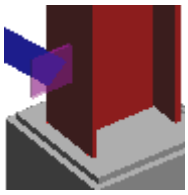
There are several basic assembly connections delivered with the software:

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.

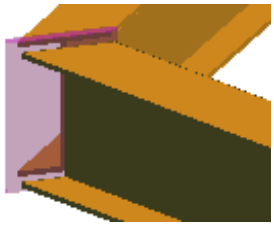
The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.

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.

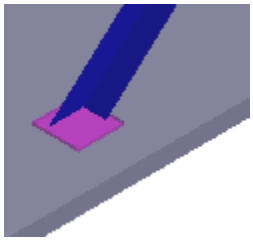


The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.



The splice assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.

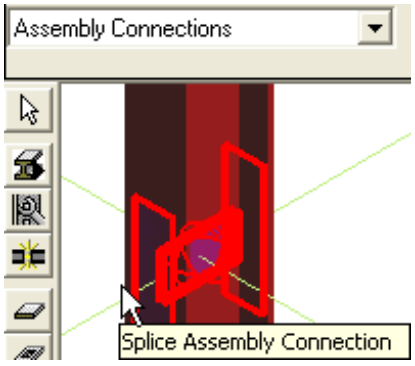
The general surface assembly connection connects a member end with a nonmember surface such as a slab. The member is cut to surface and a base plate is placed on the member end.



See *Working with Members* (on page 9) 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.



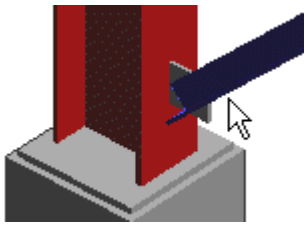
When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties.

See Also

Place an Assembly Connection (on page 77)
Working with Members (on page 9)

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* (on page 68) and *Understanding Slab Assembly Connections* (on page 102) for more information.

See Also

Place Assembly Connection Ribbon (on page 71)
Understanding Member Assembly Connections (on page 68)
Place an Assembly Connection (on page 77)

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* (on page 73).

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* (on page 68).

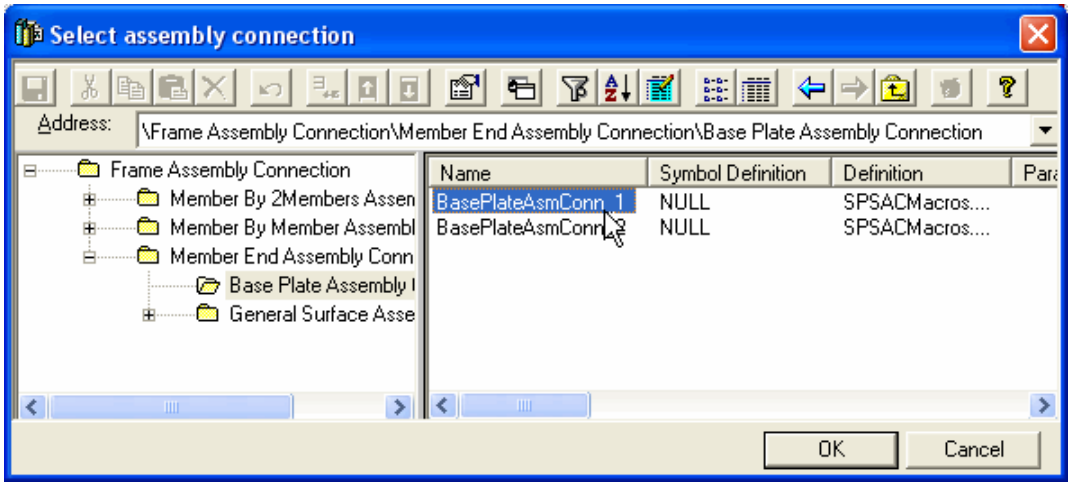
Name - Specify the name of the assembly connection.

See Also

Place an Assembly Connection (on page 77)
Place Assembly Connection Command (on page 70)

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.

See Also

Definition Tab (Assembly Connection Properties Dialog Box) (on page 76)
Occurrence Tab (Assembly Connection Properties Dialog Box) (on page 73)
Selection Tab (on page 76)
Understanding Member Assembly Connections (on page 68)
Edit Assembly Connection Properties (on page 77)

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.

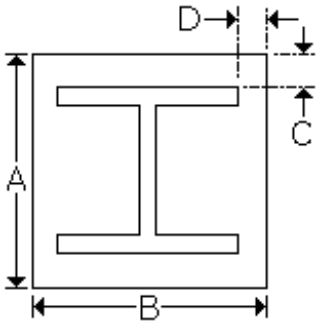
Name Rule - Specify the naming rule that you want to use to name this assembly connection. You can select one of the listed rules or select **User Defined** to specify the assembly connection name yourself in the **Name** box.

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

With Pad - Select **True** to place a steel pad between the member end and the surface. Select **False** to have the member end connect directly to the surface.

Pad Type - Select the shape of the pad.

Sizing Rule - Select whether you want to the software to automatically size the pad or if you want to size the pad.

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

See Also

Assembly Connection Properties Dialog Box (on page 73)

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.

See Also

Assembly Connection Properties Dialog Box (on page 73)

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

Class - Displays the name of the rule class.

User Answers

Select answers to questions asked by the rule class, and then select from a list of **Results** that match all of the answers. The default answers are determined by the rule class.

Valid - Click after making any changes to the user answers so that valid results for the answers are displayed in the **Results** box. Also click after changing the selection in a **Results** box.

IMPORTANT

- The available questions, answers, and results vary depending upon the detailed parts associated with the connection.
- The questions in each Selection tab represent the default rules delivered with the software. User customized rules may have different questions.

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* (on page 68).

See Also

Understanding Member Assembly Connections (on page 68)
Place Assembly Connection Command (on page 70)
Edit Assembly Connection Properties (on page 77)
Delete an Assembly Connection (on page 77)

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.

See Also

Understanding Member Assembly Connections (on page 68)
Assembly Connection Properties Dialog Box (on page 73)

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**.

See Also

Understanding Member Assembly Connections (on page 68)
Place Assembly Connection Command (on page 70)
Place an Assembly Connection (on page 77)
Edit Assembly Connection Properties (on page 77)

Trim Member Command

Manually copes and snips member parts based on objects and planes that you specify. Use this command when assembly connections are not sufficient to resolve the interference conflicts due to the position, orientation, and complexity of the intersecting members. You can place a trim and an assembly connection at the same member end. However, the results of the trim may be overwritten by the assembly connection results making the trim redundant. Both the trim and the assembly connection are listed in the **Workspace Explorer** under the member part.

See Also

Trim Member Ribbon (on page 78)
Cope a Member Web (on page 79)
Delete a Member Trim (on page 81)
Trim Member to Surface (on page 80)
Trim Member Ribbon (on page 78)

Trim Member Ribbon

Specifies the properties for the trim that you are placing or editing.

Trim Feature Properties - Activates the **Trim Feature Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Trim Feature Properties Dialog Box* (on page 78).

Select Members - Select the members that you want to cut.

Select Cutters - Select the objects to use to cut the members.

Finish - Trims the members using the cutters you have defined.

Cancel - Rejects the selected objects.

Accept - Confirms that the selected objects are the one you want use.

Trim Feature - Select the type of trim you want to place. Click **More** to select from all available trimming types defined in the catalog.

Name - Specify the name of the trim feature.

See Also

Cope a Member Web (on page 79)
Delete a Member Trim (on page 81)
Trim Member to Surface (on page 80)
Trim Member Command (on page 78)

Trim Feature Properties Dialog Box

Specifies the properties for the trim feature that you are placing.

See Also

Definition Tab (Trim Feature Properties Dialog Box) (on page 79)

Definition Tab (Trim Feature Properties Dialog Box)

The **Definition** tab displays the trim feature 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 trim feature and then selected the properties command, only the common properties between the selected trim features display.

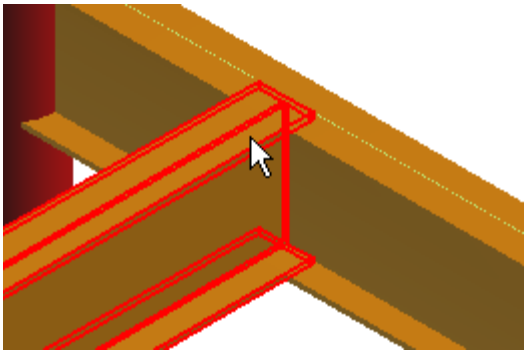
The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also

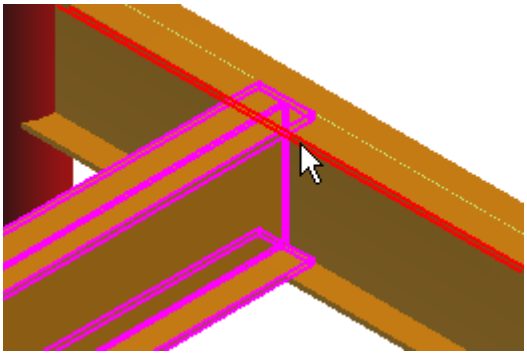
Trim Feature Properties Dialog Box (on page 78)

Cope a Member Web

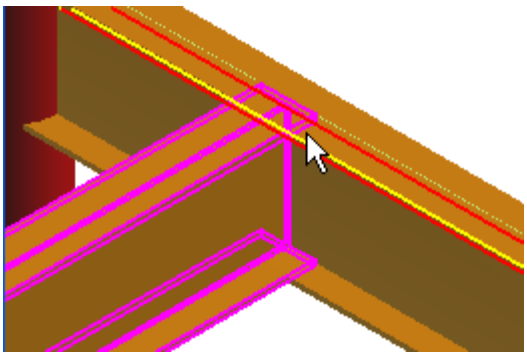
1. Click **Trim Member** on the vertical toolbar.
2. In the **Trim Feature** box on the ribbon, click **More**.
3. In the tree view, expand **Member End Trim Feature > Web Corner Cope Feature**.
4. Select **Web Cope 1/2in Clearance** in the **Name** column.
5. Click **OK**.
6. Select the member whose web you want to cope.

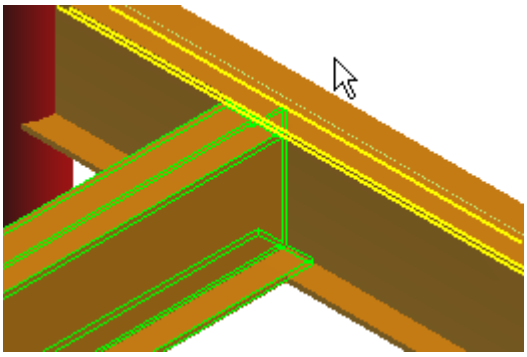


7. Click **Accept** ✓.
8. Select the plane that defines the cut into length of the web.

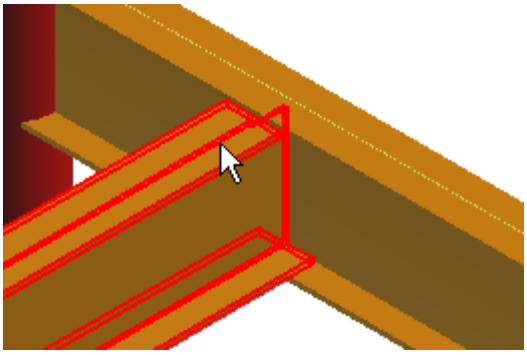


9. Select the plane that defines the cut into the height of the web.





10. Click **Finish**.



See Also

Trim Member Command (on page 78)

Trim Member to Surface

1. Click **Trim Member** on the vertical toolbar.
2. In the **Trim Feature** box on the ribbon, click **More**.
3. In the tree view, expand **Member End Trim Feature > Surface Trim Feature**.
4. Select **SurfaceTrim_1** in the **Name** column.
5. Click **OK**.
6. Select the member to trim.

7. Click **Accept** ✓.
8. Select the surface to trim the member.
9. Click **Finish**.

See Also

Trim Member Command (on page 78)

Delete a Member Trim

1. Click **Select** on the vertical toolbar.
2. Select **Structure** in the **Locate Filter**.
3. In the Workspace Explorer, expand the member prismatic part branch.
4. Select the trim feature node under the member part branch to delete.
5. Click **Delete** ✕.

NOTE You can also use the **Tools > Select by Filter** command to select the member trims to delete. Define your filter using the **Object Type** tab and the **Structure > Features > Surface Trim Feature** or **Structure > Features > Web Corner Cope Feature** objects.

See Also

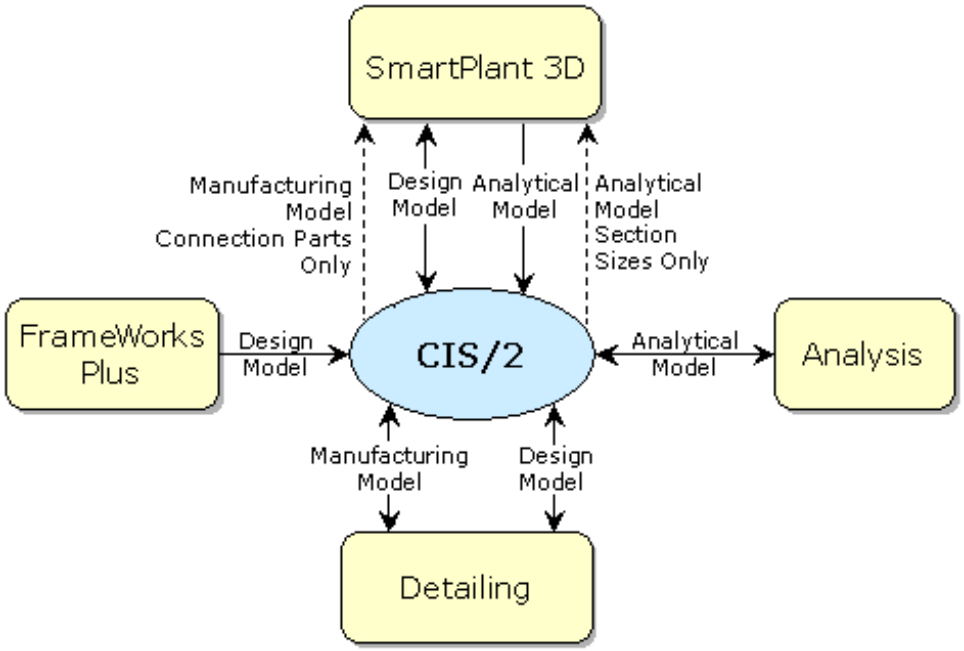
Trim Member Command (on page 78)

Importing and Exporting Structure

Structural objects are imported and exported from SmartPlant 3D using the CIMsteel Integration Standards 2.0 (CIS/2). CIS/2 was authored by The Steel Construction Institute and the Computer Aided Engineering Group School of Civil Engineering at the University of Leeds to facilitate a more integrated method of working through the sharing and management of information within and between companies involved in the planning, design, analysis, and construction of steel framed building and similar structures. For more information about CIS/2 in general, go to *http://www.cis2.org* (http://www.cis2.org).

SmartPlant 3D provides limited support for:

- importing of manufacturing models
- partial importing and exporting the analytical model (available only when in the Structural Analysis task)
- importing and exporting the design model



Limitations

There are some limitations when importing and exporting the design model and analysis model through CIS/2:

- Members imported from a design model CIS/2 file are standard SmartPlant 3D members; however, there are no relationships defined for the members. Frame connections are created during import if the connection information is provided in the CIS/2 import file. Frame connections will be set to Axis-Along with X, Y, and Z offsets defined. Assembly connections are created during import if the connection information is provided in the CIS/2 import file.

- When importing members from a CIS/2 file exported from FrameWorks Plus®, arc and tapered members are ignored. Members with workpoint offsets are read in at the physical location of the workpoint offset, but no analytical information is retained. In addition, all member end releases are set to User defined - Free. However, the FrameWorks Plus member name is assigned to the SmartPlant 3D member and the rotation and reflection settings of the member are maintained.
- FrameWorks Plus solids are ignored. FrameWorks Plus slabs are imported with bottom face orientation regardless of the face setting in FrameWorks Plus.
- Only member section size changes can be imported from an analytical model CIS/2 file.
- The software exports only the operation geometry for handrails, stairs, and ladders to the design model.
- Curved members that are exported and then re-imported will lose their associativity to the curve member's path.
- Slabs are always read into the model using a sketch 3-D path. You can edit the slab path after the import if needed. No grid lines associated with the slab are imported and there are no constraints assigned to the slab.

Based-on CIS/2 Statement for Import

Application Name:	SmartPlant 3D
Application Version: Version 2008 Service Pack 5	**Date:** January 2007 / April 2008
Translator Version: Version 2008 Service Pack 5	**Date:** January 2007 / April 2008
Software Vendor:	Intergraph Process, Power & Marine 300 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC312+CC110+CC118+CC249, CC312+CC111+CC118+CC259, CC278

Type of CIS Translator:	**Basic** | DMC | IDI | PMR-enabled
Data exchange capabilities:	**Import** | Export | Import & Export
Level of implementation:	**File Exchange** | In memory | DBMS | KBS
Flavors supported:	EU | **US** | UK
Unit Systems supported:	**SI** | US Imperial

The vendor places the following riders on the operation of the translators:

- No riders.

Date of Statement:	January 2007 / April 2008
Statement made by:	Intergraph Process, Power & Marine

Based-on CIS/2 Statement for Export

Application Name:	SmartPlant 3D
Application Version: Version 2008 Service Pack 5	**Date:** January 2007 / April 2008
Translator Version: Version 2008 Service Pack 5	**Date:** January 2007 / April 2008
Software Vendor:	Intergraph Process, Power & Marine 300 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC312, CC110, CC118, CC255, CC331, CC100, CC308

Type of CIS Translator:	Basic | **DMC** | IDI | PMR-enabled
Data exchange capabilities:	Import | **Export** | Import & Export
Level of implementation:	**File Exchange** | In memory | DBMS | KBS
Flavors supported:	EU | **US** | UK
Unit Systems supported:	**SI** | US Imperial

The vendor places the following riders on the operation of the translators:

- managed_data_deleted, managed_data_creation, and managed_data_transaction are not exported.

Date of Statement:	January 2007 / April 2008
Statement made by:	Intergraph Process, Power & Marine

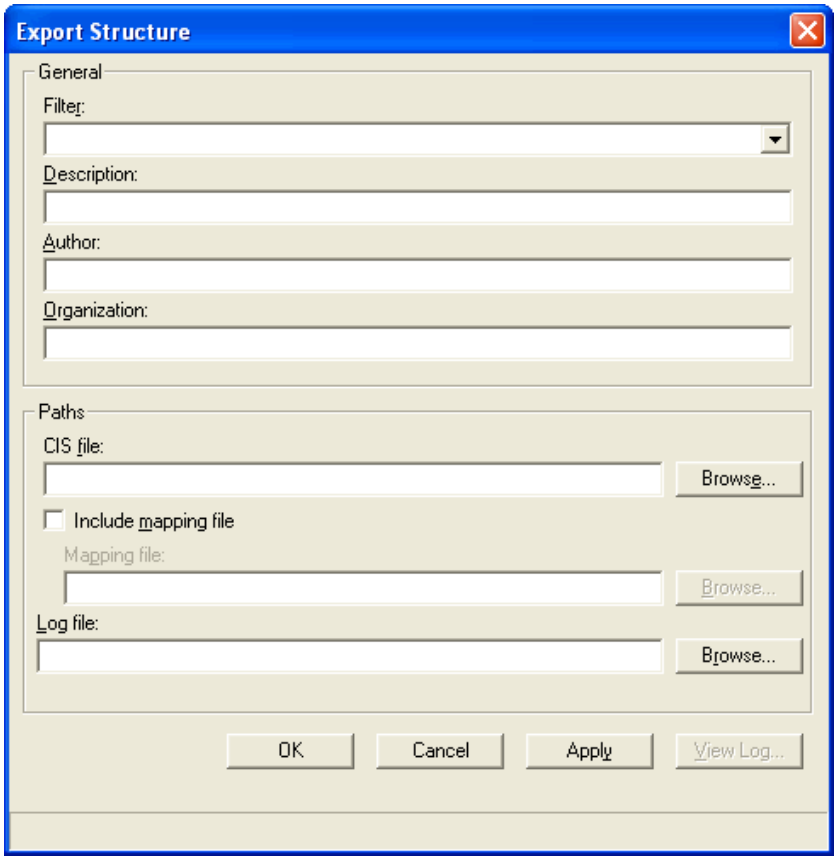
If you have any questions about using this translator, please contact Intergraph support. You can find support information on our web site *http://support.intergraph.com* (http://support.intergraph.com/).

See Also

Import Structure Command (on page 86)
Export Structure Command (on page 85)

Export Structure Command

The **File > Export > Structure** command exports the structural physical model to a CIS/2 file. The CIS/2 file contains all the necessary physical data to allow third-party software, such as a detailing application, to import the structure.



Filter - Select the filter to use to identify the model objects to export.

Description - Type a description of the model. This description is included in the CIS/2 file.

Author - Specifies the person who created the CIS/2 file. The default is the current system user name.

Organization - Specifies your company or organization name to use in the CIS/2 file.

CIS file - Define the file name and folder path of the CIMsteel Integration Standard file to write.

Include mapping file - Select this option to use a section name mapping file when exporting the members to the CIS file. A mapping file swaps the SmartPlant 3D name for a section (for example, L3-1/2X2-1/2X1/4) with the third-party software name for a section (for example,

L3.5X2.5X1/4). You must create the mapping file using the **File > New Mapping File Command** before you can use the mapping file in this command.

Mapping file - Specify the mapping file to use if the **Include mapping file** option is selected.

Log file - Specify a log file name. You can view the log file by clicking **View Log**.

NOTE The International System of Units (SI) are used in the exported CIS file regardless of the settings on the **Tools > Options > Units of Measure** tab.

If you have any questions about using this translator, please contact Intergraph Support. You can find support information on our web site: *http://support.intergraph.com* (http://support.intergraph.com/).

See Also

Importing and Exporting Structure (on page 82)
Export Structure Model (on page 86)
New Mapping File Command (on page 91)

Export Structure Model

1. Click **File > Export > Structure**.
2. Select the filter to use to select the objects to export.
3. Type a description.
4. Type your name in the **Author** box and your company name in the **Organization** box.
5. Specify the file name and folder for the CIS file.
6. Define a mapping file, if needed.
7. Define a log file name and folder.
8. Click **OK**.

NOTES

- You must create the filter before you can export the physical model.
- You must create a mapping file using the **File > New Mapping File Command** before you can use that mapping file when exporting the physical model.

See Also

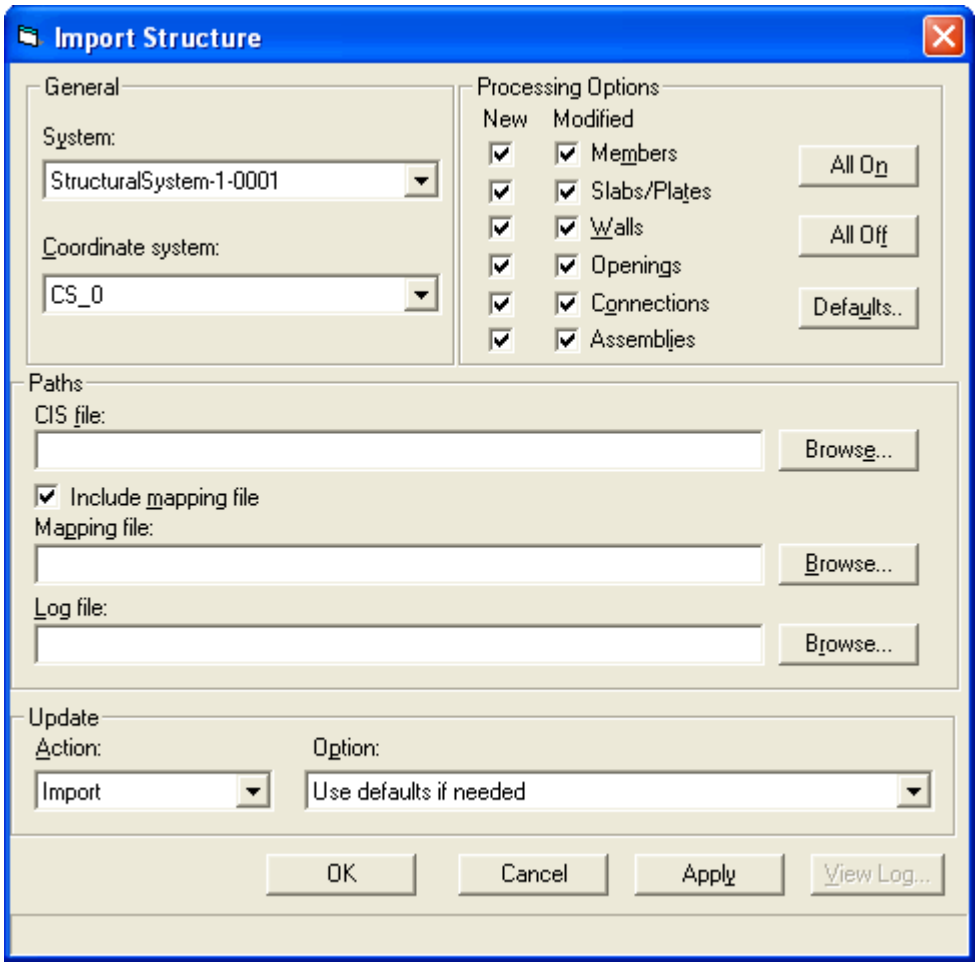
Export Structure Command (on page 85)
New Mapping File Command (on page 91)

Import Structure Command

The **File > Import > Structure** command in SmartPlant 3D imports a CIS/2 file into the model. This command recognizes Global User Identities (GUIDs) to uniquely identify objects and manage the electronic exchange with the other software package.

NOTE Members imported using this command are standard SmartPlant 3D members. Frame connections and assembly connections are created if that information is provided in the CIS/2 import file. No relationships are created for the members.

If you have any questions about using this translator, please contact Intergraph Support. You can find support information on our web site: *http://support.intergraph.com* (http://support.intergraph.com/).



System - Select the system to which to assign the objects being read. 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* (on page 21).

Coordinate System - Select the coordinate system to which to assign the objects being read. If needed, you can create a new coordinate system in the Grids task.

New - Loads any object in the CIS/2 file that is not in the model. Use the **All On** or **All Off** commands to select or clear all checkboxes.

Modified - Loads any object that exists in both the CIS/2 file and the model that has changed in the CIS/2 file. Use the **All On** or **All Off** commands to select or clear all checkboxes.

Members - Select to read into the model the linear members defined in the CIS/2 file. Openings on members are not imported.

Slabs/Plates - Select to read into the model the slabs and plates defined in the CIS/2 file. Slabs are always read into the model using a sketch 3-D path. You can edit the slab path after the

import if needed. No grid lines associated with the slab are imported and there are no constraints assigned to the slab.

Walls - Select to read into the model the walls defined in the CIS/2 file. All walls are imported as slabs.

Openings - Select to read into the model the openings (holes) defined in the CIS/2 file for slabs, walls, grating, and checker plates.

Connections - Select to read into the model the gusset plates, clip angles, and other plates and standard sections used to connect members. The detailing software marks these objects as connection parts in the CIS/2 schema during export.

Assemblies - Select to read into the model the assemblies defined in the CIS/2 file.

All On - Click to activate all **New** and **Modified** check boxes.

All Off - Click to clear all **New** and **Modified** check boxes.

Defaults - Activates a dialog box where you define properties to use when the object being imported does not have a recognized property.

- **Type Category** - Select the type category to use when a member being imported does not have a recognized type category. You can define a custom member type category on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.
- **Type** - Select the type to use when a member being imported does not have a recognized type. The properties change depending on the member type that you select. You can define a custom member type on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.
- **Section Name** - Select the cross-section to use when a member being imported does not have a recognized section. Sections are defined in the reference data. See the *Structure Reference Data Guide* for more information about reference data.
- **Material Name** - Select the material to use when a member being imported does not have a recognized material.
- **Material Grade** - Select the material grade to use when a member being imported does not have a recognized material grade.
- **Type** - Select the slab type to use when a slab being imported does not have a recognized slab type.
- **Composition** - Select the slab composition to use when a slab being imported does not have a recognized slab composition.

CIS file - Define the file name and folder path of the CIMsteel Integration Standard file to read.

Include mapping file - Select this option to use a section name mapping file when importing the members from the CIS file. A mapping file swaps the third-party software name for a section (for example, L3.5X2.5X1/4) with the SmartPlant 3D name for a section (for example, L3-1/2X2-1/2X1/4). You must create the mapping file using the **File > New Mapping File Command** before you can use the mapping file in this command. Sample mapping files for the FrameWorks Plus AISC table are delivered with the software in [Pr*oduct Directory]\3D\S*martPlantStructure\Symbols\StructureImportExportMaps. For more information, see *New Mapping File Command* (on page 91).

Mapping file - Specify the mapping file to use if the **Include mapping file** option is selected.

Log file - Specify a log file name. You can view the log file by clicking **View Log**.

Action - Select whether to import or preview the contents of the CIS/2 file. Select **Import** to import objects from the CIS/2 into the model. Select **Preview** to identify problems with the CIS/2 file before the actual import is attempted. Objects in the CIS/2 file that have not been imported cannot be previewed.

Filter - Specifies how you want to preview the CIS/2 file. This option is only available when **Action** is set to **Preview**.

- **Objects in CIS file with unknown sections, material, or type** - Select this option to help identify potential problems with the CIS/2 file before you attempt the import. After using option, you can add unknown sections, materials, and types to the mapping file.
- **Objects in DB that exist in CIS File** - Select this option to identify objects that are in both the model and the CIS/2 file.
- **Objects in DB that are modified in CIS File** - Select this option to identify objects that are in both the model and the CIS/2 but that have different attributes (such as section size) in the CIS/2 file. This option is useful when "round tripping" between SmartPlant 3D and other software package.

Option - Specifies how you want to import objects from the CIS/2 file. This option is only available when **Action** is set to **Import**.

- **Use no defaults** - Select this option to import only those objects with known to SmartPlant 3D (known either in the software or in the specified mapping file) sections. Objects with unknown sections are not imported into the model but are noted in the log file.
- **Use defaults if needed** - Select this option to import all objects in the CIS/2 to the model. Objects with sections not found in the software or in the mapping file will be given the sections defined in the **Defaults** dialog box. Objects imported with default sections are noted in the log file.

See Also

Importing and Exporting Structure (on page 82)
Import Structure Model (on page 89)
Preview a CIS File (on page 90)
New Mapping File Command (on page 91)

Import Structure Model

1. Click **Tasks > Structure**.
2. Click **File > Import > Structure**.
3. Select the system in which to place the imported objects. You can create new systems in the Systems and Specifications task.
4. Select the coordinate system to which to associate the imported objects. You can create new coordinate systems in the Grids tasks.
5. Specify which objects to import by selecting the **New** box or to update by selecting the **Modified** box.

6. Click **Defaults** and define the default properties for imported objects that do not have recognized properties.
7. Specify the file name and folder for the CIS file.
8. Define a mapping file, if needed.
9. Define a log file name and folder.
10. In the **Action** box, select **Import**.
11. In the **Option** box, select whether the software should use default properties.
12. Click **Apply** to import the CIS file.
13. Click **View Log** to review the log file.

NOTE You must create a mapping file using the **File > New Mapping File Command** before you can use that mapping file when importing a structural model.

See Also

Import Structure Command (on page 86)

Preview a CIS File

1. Click **Tasks > Structure**.
2. Click **File > Import > Structure**.
3. Select the system in which to place the previewed objects. You can create new systems in the Systems and Specifications task.
4. Select the coordinate system to which to associate the previewed objects. You can create new coordinate systems in the Grids tasks.
5. Specify which objects to preview by selecting corresponding **New** or **Modified** boxes.
6. Click **Defaults** and define the default properties for imported objects that do not have recognized properties.
7. Specify the file name and folder for the CIS file.
8. Define a mapping file, if needed.
9. Define a log file name and folder.
10. In the **Action** box, select **Preview**.
11. In the **Filter** box, select the preview option to use.
12. Click **Apply** to preview the CIS file.
13. Click **View Log** to review the log file.

NOTE You must create a mapping file using the **File > New Mapping File Command** before you can use that mapping file when importing a structural model.

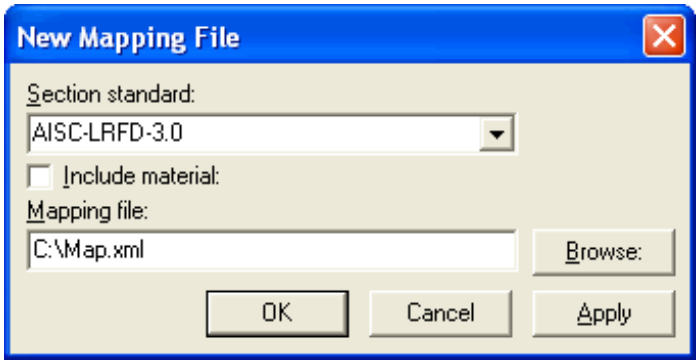
See Also

Import Structure Command (on page 86)

New Mapping File Command

The **File > New Mapping File** command creates an XML mapping file for the section names, and optionally material names, used in the software and third-party application. Many times, the software and the third-party application use different names for the same section or material. The mapping file solves the naming conflicts by mapping section names in the software to section names in the other applications. The mapping file must contain each section standard table that you have used in the model.

CAUTION The mapping file created by this command is a template. The software does not write known-to-be-different section names to the mapping file. You are responsible for verifying, editing, updating, and maintaining the third-party application section names in the file.



Section standard - Select the section standard table for which to create a mapping file.

Include material - Select to include material name mappings in addition to the section name mappings.

Mapping file - Specify a name and folder path for the XML mapping file.

See Also

Create a Mapping File (on page 94)
Export Structure Command (on page 85)
Import Structure Command (on page 86)

Mapping File Format

The mapping file is an XML-formatted file with which you can define mappings for section names, material names, member types, and slab types between the software and third-party software. The format for each mapping is given below. You can include all four mappings in a single XML file, or you can define the mappings in separate XML files and reference the four files in a single master XML file using include statements. Using separate files that are pulled together using include statements may be a better workflow in that you can quickly mix and match files for different requirements.

You cannot nest include files. Only the master XML mapping file can call an include file. You cannot call another include file inside an include file. An example of a master XML mapping file is shown:

```
<xml>
<!-- Comment Line -->
    <IncludeXML href="SectionStandard.xml" />
    <IncludeXML href="Material.xml" />
    <IncludeXML href="MemberTypes.xml" />
    <IncludeXML href="SlabTypes.xml" />
</xml>
```

Section Mapping

The <SectionStandard> area maps the software section names to the third- party section names. The software section names are labeled "section name". The third-party section names are labeled "externalname". When the XML file is created, the software section name is duplicated for the third-party section name. You must verify that the correct third-party section name is defined for "externalname" by manually editing the XML file.

```
<xml>
<!-- Comment Line -->
<SectionStandard>
    <Standard name="AISC-LRFD-3.0" externalname="AISC"
externalorganization="user" externaldate="2002" externalversion="1.0"
/>
        <Sections>
            <Section name="W10x39" externalname="W10x39" />
            <Section name="W10x33" externalname="W10x33" />
            <Section name="W10x30" externalname="W10x30" />
            ...
        </Sections>
</SectionStandard>
</xml>
```

Material Mapping

The <MaterialStandard> area maps the software material grade names to the third-party material grade names. You must have selected the **Include material** option when you created the XML file to see the material grade name mappings. The software materials are labeled "Material type" and "grade". The third-party material names are labeled "externalname". You must verify that the correct third-party material grade name is defined for "externalname" by manually editing the XML file.

```
<xml>
<!-- Comment Line -->
<MaterialStandard>
    <Standard name="" externalname="" externalorganization=""
externaldate="" externalversion="" />
        <Materials>
            <Material type="Steel - Carbon" grade="A36"
externalname="A36" />
            <Material type="Steel - Carbon" grade="A529"
externalname="A529" />
            <Material type="Steel - Carbon" grade="A588"
externalname="A588" />
            ...
        </Materials>
</MaterialStandard>
</xml>
```

Member Type Mapping

The <MemberTypes> area maps the software member types to the third-party member types. You have to create this section in a text editor. The software member types are labeled "Member type". The third-party member types are labeled "externaltype" and "externalrole".

```
<xml>
<!-- Comment Line -->
<MemberTypes>
    <Member type="Beam" externaltype="Beam" externalrole="" />
    <Member type="Girder" externaltype="Beam"
externalrole="gantry_girder" />
    <Member type="Joist" externaltype="Beam" externalrole="joist" />
    ...
</MemberTypes>
</xml>
```

Slab Type Mapping

The <SlabTypes> area maps the software slab types to the third-party slab types. You have to create this section in a text editor. The software slab types are labeled "Slab type" and "composition". The third-party slab types are labeled "externaltype".

```
<xml>
<!-- Comment Line -->
<SlabTypes>
   <Slab type="4&quot; Cast in Place" composition="CIP_4&quot;_Fc3"
externaltype="slab" />
   <Slab type="4&quot; Cast in Place" composition="CIP_4&quot;_Fc4"
externaltype="flat_slab" />
   <Slab type="5&quot; Cast in Place" composition="CIP_5&quot;_Fc3"
externaltype="wall" />
    ...
</SlabTypes>
</xml>
```

See Also

Create a Mapping File (on page 94)
Exporting and Importing Analytical Data

See Also

Create a Mapping File

1. Click **File > New Mapping File**.
2. In the **Section standard** box, select the section standard for the mapping file.
3. Optionally, select **Include material** to write material names to the mapping file.
4. Click **Browse**, and then specify a name and folder location for the mapping file.
5. Click **OK**.
6. Edit the mapping file using a text editor such as Notepad, and define the third-party standard section, material names, member types, and slab types.

See Also

New Mapping File Command (on page 91)

Translator Setup

The CIS/2 translator for FrameWorks Plus requires the lease of "SmartPlant 3D PDS Project Translators" (SEBY801AA), which is a separate lease from FrameWorks Plus, PDS, and SmartPlant 3D. Contact Intergraph for software availability. These setup instructions cover translating FrameWorks Plus data to SmartPlant 3D only. For instructions on translating PDS Piping, Equipment, or HVAC data to SmartPlant 3D, see *Exporting Data from PDS*.

NOTE You must have FrameWorks Plus and PD_Shell installed on the computer where you install the translator.

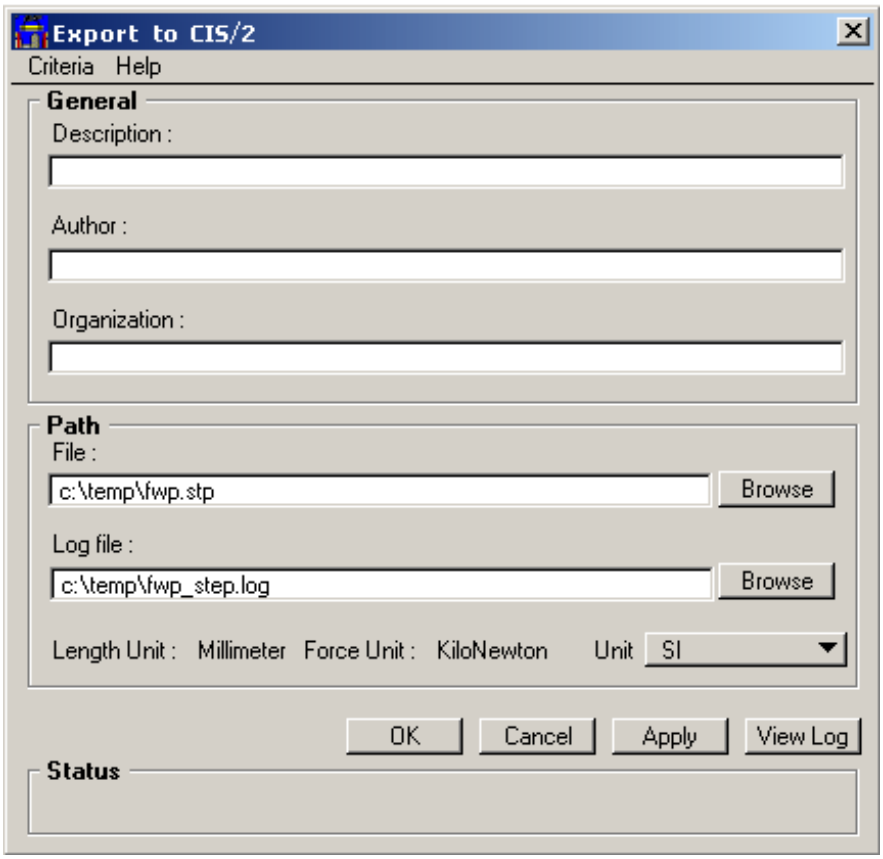
1. On the computer where you have FrameWorks and PD_Shell installed, insert the SmartPlant 3D CD-ROM.
2. Close the SmartPlant 3D installation menu.
3. Using Windows Explorer, browse to the **PDS Export to SP3D** folder on the CD.
4. Open the sub-folder that corresponds to your PDS version. For example, if you are using PDS Version 8.0, open the **PDSExport_PDS8.0** folder.
5. Double-click setup.exe
6. Click **Continue**.
7. Click **I Agree** on the License Agreement.
8. Enter your name, company name, and the SmartPlant 3D PDS Project Translators serial number.

9. Click **Continue** to verify the information.

Exporting from FrameWorks Plus

The CIS/2 translator for FrameWorks Plus is delivered as part of a FrameWorks Plus. To use this command, open the model that you want to export to SmartPlant 3D. Be sure to attach all reference models that are needed. Then, type **mdl load SPExport** in the MicroStation key-in field.

NOTE FrameWorks Plus does not write tapered member, arc members, or generic solid members to the CIS/2 file.



Criteria - Activate a dialog box used to select what members to write to the CIS/2 file. For more information, see *Member Criteria* (on page 97).

Description - Enter a description for the CIS/2 file.

Author - Enter your name.

Organization - Enter your organization name.

File - Enter a name and location for the CIS/2 file.

Log file - Enter a name and location for the log file.

Units - Specify the units for the CIS/2 file.

OK - Writes the CIS/2 file and closes the dialog box. You cannot review the log file using the **View Log** command if you click **OK**. Use **Apply** if you want to review the log file at the end of the process.

Cancel - Exits the dialog box without writing the CIS/2 file.

Apply - Writes the CIS/2 file without closing the dialog box. Use this command if you want to review the log file at the end of the process.

View Log - Opens the log file for review.

Status - Displays the CIS/2 writing progress.

Based-on CIS/2 Statement for Export

Application Name:	SmartPlant 3D PDS Project Translators
Application Version: 2007 SP3	**Date:** September 2007
Translator Version: 2007 SP3	**Date:** September 2007
Software Vendor:	Intergraph Process, Power & Marine 300 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC312, CC110, CC118, CC255, CC331, CC100, CC308

Type of CIS Translator:	Basic | **DMC** | IDI | PMR-enabled
Data exchange capabilities:	Import | **Export** | Import & Export
Level of implementation:	**File Exchange** | In memory | DBMS | KBS
Flavors supported:	EU | **US** | UK
Unit Systems supported:	**SI** | **US Imperial**

The vendor places the following riders on the operation of the translators:

- managed_data_deleted, managed_data_creation, and managed_data_transaction are not exported.

Date of Statement:	September 2007
Statement made by:	Intergraph Process, Power & Marine

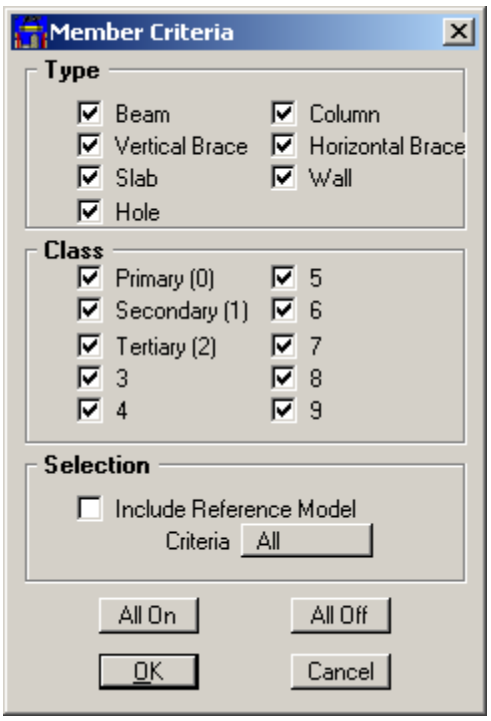
If you have any questions about using this translator, please contact Intergraph support. You can find support information on our web site *http://support.intergraph.com* (http://support.intergraph.com/).

See Also

Exporting Models from FrameWorks Plus (on page 98)
Member Criteria (on page 97)

Member Criteria

The **Criteria > Member Criteria** command is used to define which members you want to write to the CIS/2 file. Because you define the criteria for what members to write to the file here, the settings in the **Select Filter Settings** command are ignored.



Type - Toggle on the member design types you want to write to the file.

Class - Toggle on the member classes you want to write to the file.

Include Reference Model - Defines whether to write members from attached models to the file in addition to the members in the active model. You can attach models using the **File > Attach Models** command. You can detach models using the **File > Detach Models** command.

Criteria - Defines whether to write members from a selection set or to write all the members in the model to the file.

All On - Toggles on all **Design Type** and **Member Class** settings. The default is that all settings are selected.

All Off - Toggles off all **Design Type** and **Member Class** settings.

OK - Saves any changes you have made and exits the dialog box.

Cancel - Ignores any changes you have made and exits the dialog box.

See Also

Exporting from FrameWorks Plus (on page 95)
Exporting Models from FrameWorks Plus (on page 98)
Member Criteria (on page 97)

Translator Setup (on page 94)

Exporting Models from FrameWorks Plus

1. Start FrameWorks Plus.
2. Open the model that you want to export to SmartPlant 3D.
3. Attach all reference models that are needed.
4. Type **mdl load SPExport** in the MicroStation key-in field.
5. Define the **Criteria** for member selection.
6. Type a description for the file.
7. Type your name and your organization's name.
8. In the **File** box, enter a name and location for the CIS/2 step file.
9. In the **Log file** box, enter a log file name.
10. Select whether to write the SI units or Imperial Units to the file.
11. Click **Apply**.
12. Click **View Log**.
13. Carefully review the log file for errors.

See Also

Exporting from FrameWorks Plus (on page 95)
Exporting Models from FrameWorks Plus (on page 98)
Member Criteria (on page 97)
Translator Setup (on page 94)

SECTION 4

Working with Walls and Slabs

Slabs

Slabs are used to model solid surfaces, such as floors or steel grating, in your model. The software places slabs on a supporting plane, or multiple planes, that you select and between selected boundaries. The supporting plane of the slab can be an elevation plan, a grid plane, or a plane that you define during placement. You can define planes at an angle to an existing elevation or grid plane, or you can define a plane in space using three points.

The slab boundaries that you can select include grid lines, members, edges of members, edges of equipment, faces of other slabs, and many other objects in the model. When an object that is used as a boundary moves, the software automatically updates the slab to reflect the move.

You also have the option to sketch the boundaries of the slab in the model's 3D or 2D environments, or combine the selecting of boundaries for some sides of the slab and sketching the other sides.

NOTE We recommend that you place grid planes using the Grids task before placing slabs so that supporting planes are available for selection.

Walls

Walls are used to model solid surfaces, such as retaining wall or bearing wall, in your model. The software places walls on a supporting plane that you select. The supporting plane of the wall can be an elevation plan, a grid plane, or a plane that you define during placement. You can place straight walls, curved walls, or a wall with both curved and straight areas.

When placing walls, you define a path that the wall is to follow. You can define this path in the 3-D environment using the same path commands that you use to define handrails paths, or you can define this path in the 2-D environment using lines connected and grouped together to define the path. When you need to modify the walls that you have placed, keep these things in mind:

- All walls paths created in the 2-D environment should be modified in the 2-D environment unless you are modifying the entire area with a rotate or move, in which case you should do the modification in the 3-D environment. For example, you placed a wall using a 2-D drawn path. To move, rotate, or add on to that individual wall, you would go back to the 2-D environment. However, if you were going to rotate the wall along with everything else in the general area, you would do the rotate all the objects in the 3-D environment.
- When moving or rotating a wall path in the 2-D environment, you may have to delete some constraints before you can move or rotate. You need to display the relationships layer to delete them if needed.
- When a 2-D sketched wall is selected in the 3-D environment, all the other walls that were created during that same 2-D session are also selected. If you move or rotate one wall, all the walls from the session will move or rotate also. To move or rotate an individual wall, you need to either place each wall individually in the 2-D environment (coming back out to the 3-

D environment using Finish) or you need to go into the 2-D environment and perform the move or rotate on the individual wall in the 2-D environment.

- If you import grid lines into the 2-D environment as references and then sketch a wall path on top of the grid lines, the software will create horizontal or vertical constraints to the grid lines.

See Also

Delete a Slab (on page 118)
Edit Slab Object Boundaries (on page 117)
Edit 2-D Drawn Slab Boundaries (on page 117)
Edit 3-D Drawn Slab Boundaries (on page 117)
Place a Slab by Drawing 3-D Boundaries (on page 114)
Place a Slab by Drawing 2-D Boundaries (on page 114)
Place a Slab by Selecting Boundary Objects (on page 113)
Place a Slab by Selecting Objects and Drawing 2-D Boundaries (on page 115)
Place a Slab by Selecting Objects and Drawing 3-D Boundaries (on page 116)
Add End Segment to Existing Wall using 2-D Sketch (on page 127)
Add Fillet Corner to Existing Wall using 2-D Sketch (on page 128)
Add Middle Segments to Existing Wall using 2-D Sketch (on page 129)
Add End Segment to Existing Wall using 3-D Sketch (on page 133)
Create Gap in Existing Wall using 2-D Sketch (on page 131)
Delete a Wall (on page 134)
Modify Wall Length using 2-D Sketch (on page 130)
Place a Wall by 2-D Sketch (on page 127)
Place a Wall by 3-D Sketch (on page 132)
Rotate an Existing Wall using 2-D Sketch (on page 132)
Toggle Wall Corner (on page 133)
Understanding Slab Assembly Connections (on page 102)

Understanding Slab Assembly Connections

Slab assembly connections are similar to member assembly connections, but define the trimming and edges of slabs. Slab assembly connections are placed automatically when the slab is created.

The slab assembly connections are:

- The Slab by Member Boundary assembly connection is placed by the software when you select a member as a slab boundary.
- The Slab Free Edge Assembly connection is placed by the software when you select a grid line as a slab boundary.

See *Working with Walls and Slabs* (on page 101) for important related information.

Locating Slab Assembly Connections

Assembly connections do not display in the model. However, if you set the Locate Filter to Slab Assembly Connections, you can locate and select assembly connections for review and editing. Assembly connections are located at the edges of slabs.

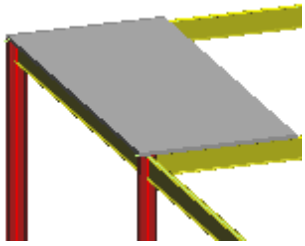
When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties.

See Also

Working with Walls and Slabs (on page 101)

Place Slab Command

Places a slab in the model. You can use this command to model floors, roofs, and other volume type members in your model. For more information, read *Working with Walls and Slabs* (on page 101).



See Also

Slab Properties Dialog Box (on page 107)
Working with Walls and Slabs (on page 101)
Delete a Slab (on page 118)
Edit Slab Object Boundaries (on page 117)
Edit 2-D Drawn Slab Boundaries (on page 117)
Edit 3-D Drawn Slab Boundaries (on page 117)
Place a Slab by Drawing 3-D Boundaries (on page 114)
Place a Slab by Drawing 2-D Boundaries (on page 114)
Place a Slab by Selecting Boundary Objects (on page 113)
Place a Slab by Selecting Objects and Drawing 2-D Boundaries (on page 115)
Place a Slab by Selecting Objects and Drawing 3-D Boundaries (on page 116)
Place Slab Ribbon (on page 103)
Edit Slab Properties (on page 118)

Place Slab Ribbon

Specifies the properties for the slab that you are placing.

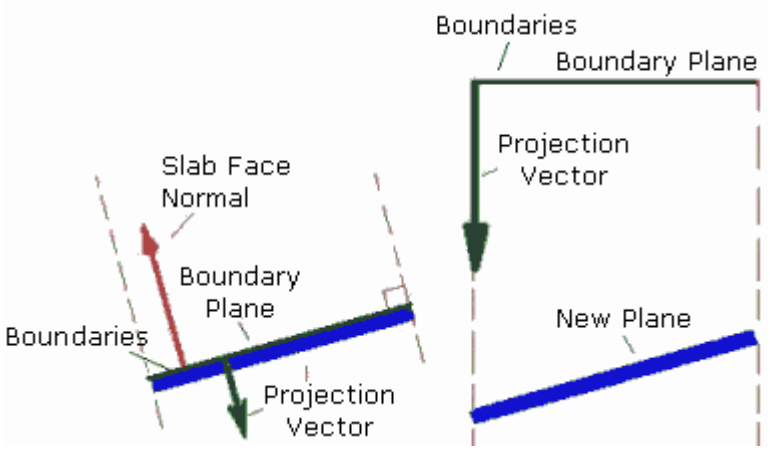
Slab Properties - Activates the **Slab Properties** dialog box. You can use this dialog box to specify additional slab properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Slab Properties Dialog Box* (on page 107).

Define Plane - Select the support plane of the slab. You can select multiple planes to place multiple slabs at the same time. For example, if you have a multi-story building with the same floor geometry and you want the same size and type of slab on each floor, you can select the plane for each floor. You can select each floor elevation plane, and then define the boundaries on one floor. The software uses the same boundaries for each elevation level and places the slabs. After placement, the slabs are unrelated to each other and can be edited as individual entities.

Custom Sketching Plane - When the support plane is sloped (you used **3 Point Plane** or **Angle to Plane** to define the support plane), you can select a custom sketching plane to on which to define the boundaries.

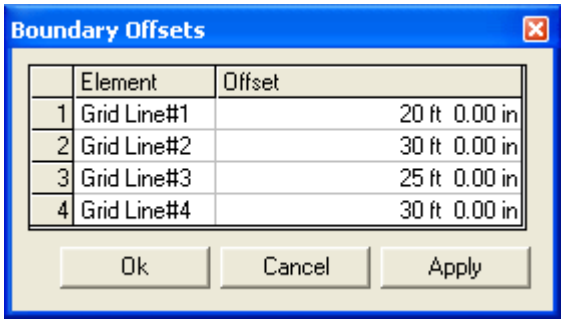
When placing a sloped slab, the default boundary plane is normal to the slab face (left side of the figure). The boundaries that you define are projected normal to sloped slab face.

However, if you select the **Custom Sketching Plane** option on the ribbon, the software prompts you to select a boundary plane (usually an elevation plane). The boundaries that you defined on the custom boundary plane are projected parallel to the Z-axis (right side of the figure).



Define Boundaries - Define the outside boundaries of the slab. You can define the slab boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the slab to maintain the boundary relationship. For example, you can select the outside edge of a beam flange as a boundary along one side of a slab. Later, if the section size for the beam is changed and the flange edge moves out an inch, the software automatically moves the edge of the slab an inch to match the new location of the flange edge.

Define Boundary Offsets - Activates the **Boundary Offsets** dialog box. This dialog box allows you to define an offset distance for each boundary that you have defined. Select a row to highlight the boundary in the model. Then, type the distance from the boundary to place the edge of the slab. Type 0 to place the slab edge on the boundary. Type a negative number to offset the slab inside the boundary. Type a positive number to offset the slab outside the boundary.



Finish - Places the slab using the defined parameters.

Plane Method - Specifies how you want to define the support plane. This option is only available when you are defining the support plane.

- Select **Coincident** to specify that you want the slab placed on the support plane.
- Select **Offset from a Plane** to place the slab a specified distance from the support plane. If you choose this option, you must define the offset distance.
- Select **Angle to plane** to place the slab at a specified angle or slope to the support plane. If you choose this option, you must define an axis of rotation and the angle or slope.
- Select **Vector & Point** to specify the support plane using two points to define a vector normal to the plane and a third point to define the plane position along the vector.
- Select **3 Point Plane** to specify the support plane using three points that you specify in the model.

Offset - Specify the offset distance for the slab from the selected support plane. You can specify the offset dynamically in graphics or by typing the distance. This option is only available when **Plane Method** is set to **Offset from a Plane**. Click **Lock/Unlock** to lock the offset distance.

Angle - Specify the angle at which to place the slab relative to the support plane. You have to define the axis of rotation using two points before you can define the angle. This option is only available when **Plane Method** is set to **Angle to plane**. Click **Lock/Unlock** to lock the angle.

Slope - Specify the slope at which to place the slab relative to the support plane. You have to define the axis of rotation using two points before you can define the slope. This option is only available when **Plane Method** is set to **Angle to plane**. Click **Lock/Unlock** to lock the slope.

Select Boundaries - Select this option to select objects in the model to define the slab boundaries. This option is only available when you are defining the slab boundaries.

Add References to Sketch 2D - Allows you to select which objects in the 3-D environment to see in the sketch 2-D environment when you are drawing the boundaries of the slab. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model. This option is only available when you are drawing the slab boundaries.

Draw - Opens the sketch 2-D environment in which you can draw the boundaries of the slab. You can only draw one slab at a time in the 2-D environment. This option is only available when you are defining the slab boundaries.

Select Patches to Compose the Slab - If the boundaries that you selected can form the boundary for the slab in more than one way, then you have defined an ambiguous boundary. The software prompts you to select one or more bounded areas to clarify the desired slab boundary. For more information, see *Solve Ambiguous Boundaries* (on page 116).

Sketch 3D - Select this option to sketch the boundaries of the slab in the 3-D environment.

Cancel - Clears all selected planes or boundary objects.

Accept - Accepts all selected planes or boundary objects.

System - Select the parent system for the slab that you are placing. Systems are defined using the Systems and Specifications task.

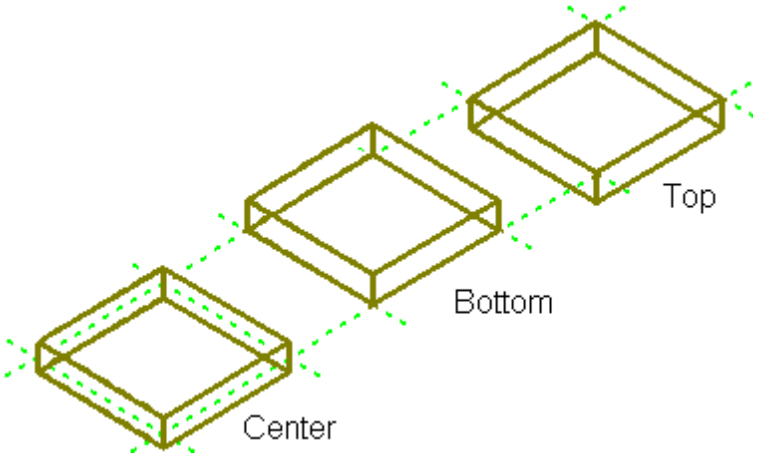
Type - Select the type of slab to place, such as a slab on grade or composite deck. The properties change depending on the slab type that you select. You can define a custom slab type in the **StructSlabGeneral.xls** workbook in the reference data. Refer to the *Structure Reference Data Guide* for more information.

Composition - Select a slab composition from the list. Available compositions depend on the type of slab being placed. You can define a custom composition using the **StructSlabGeneral.xls** and **StructSlabLayer.xls** workbooks in the reference data. Refer to the *Structure Reference Data Guide* for more information.

Name - Displays the name of the slab that you are placing.

Priority - Select the priority to assign to the slab.

Face Position - Specifies how the slab is placed in the model with respect to the support plane. If this option is set to **Center**, the slab is placed with respect to its centroidal axis. If this option is set to **Top**, the slab is placed with respect to its top face. If this option is set to **Bottom**, the slab is placed with respect to its bottom face.



Total Thickness - Type the thickness of the slab. The default thickness is based on the selected slab type and composition.

NOTE If the slab has more than one layer, then the **Total Thickness** box is read-only. To edit the slab thickness, use the **Layer** tab of the **Slab Properties** dialog box. For more information, see *Edit Slab Properties* (on page 118).

See Also

Delete a Slab (on page 118)
Edit Slab Object Boundaries (on page 117)
Edit 2-D Drawn Slab Boundaries (on page 117)
Edit 3-D Drawn Slab Boundaries (on page 117)
Place a Slab by Drawing 3-D Boundaries (on page 114)
Place a Slab by Drawing 2-D Boundaries (on page 114)
Place a Slab by Selecting Boundary Objects (on page 113)
Place a Slab by Selecting Objects and Drawing 2-D Boundaries (on page 115)

Place a Slab by Selecting Objects and Drawing 3-D Boundaries (on page 116)
Place Slab Command (on page 103)

Slab Properties Dialog Box

Specifies the properties for the slab that you are editing. You can define the default values in the **StructSlabGeneral.xls** and **StructSlabLayer.xls** workbooks.

See Also

General Tab (Slab Properties Dialog Box) (on page 107)
Layer Tab (Slab Properties Dialog Box) (on page 111)
Edit Slab Properties (on page 118)

General Tab (Slab Properties Dialog Box)

The **General** tab displays the slab 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 slab and then selected the properties command, only the common properties between the selected slabs display.

When viewing properties for a single slab, 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.

Category - Select the properties that you want to view for the slab. Slab properties are divided into several different categories: *Standard* (see "General Tab (Slab Properties Dialog Box)" on page 107), *Dimensions* (see "General Tab (Slab Properties Dialog Box)" on page 107), *Weight and CG* (see "General Tab (Slab Properties Dialog Box)" on page 107), *Fabrication and Construction* (see "General Tab (Slab Properties Dialog Box)" on page 107), *Surface Treatments and Coatings* (see "General Tab (Slab Properties Dialog Box)" on page 107), and *Responsibility* (see "General Tab (Slab Properties Dialog Box)" on page 107). You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the slab. Because slab reference data is fully customizable, only the common properties are documented.

Parent System - Select the system to which the slab belongs.

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

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

Type - Select the slab type.

Piece Mark - Specifies the piece mark of the slab. Piece marks are mainly used when importing and exporting slabs through CIMsteel to other software packages. If a piece mark is displayed here, more than likely it is the identity of the manufactured part from the other software package.

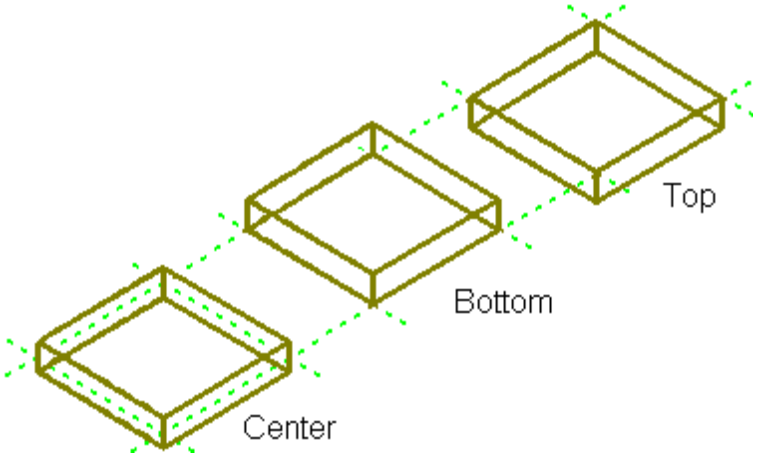
Assembly Mark - Specifies the assembly mark of the slab. If the slab was imported through CIMsteel, the assembly mark is more than likely the manufactured assembly identity to which this slab belongs.

Priority - Select the priority to assign to the slab. The priority is used to group objects.

Continuity - Select the continuity type for the slab.

Fire Rating - Enter the fire rating time for the slab.

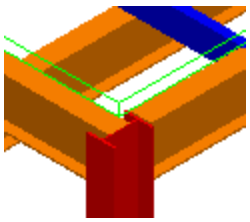
Face Position - Specifies how the slab is placed in the model with respect to the support plane. If this option is set to Center, the slab is placed with respect to its centroidal axis. If this option is set to Top, the slab is placed with respect to its top face. If this option is set to Bottom, the slab is placed with respect to its bottom face.



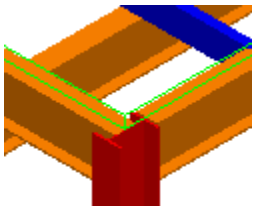
Normal Offset - Displays the offset distance for the slab from the selected support plane.

Boundary Offset Reference - When a member is used as a boundary for a slab, you can use this option to specify which part of the member the slab should use as the boundary.

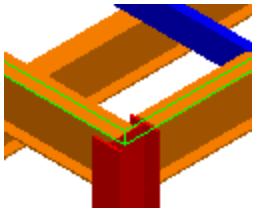
Select **Inner port-face of a bounding member** when you want the slab to stop at the inner most plane of the bounding member.



Select **Centerline (axis) of a bounding member** when you want the slab to stop at the member-line axis of the bounding member. In the picture below, the beams were placed using cardinal point 8, which means the member- line axis is in the top center of the beams.



Select **Outer port-face of a bounding member** when you want the slab to stop at the out-most plane of the bounding member.



Boundary Offset - Displays the boundary offsets for the slab.

Angle - Displays the angle of the slab relative to the reference plane that was selected during placement.

Slope - Displays the slope of the slab relative to the reference plane that was selected during placement.

Thickening Direction - Select the thickening direction.

Boundaries Projection Direction - Select the boundaries projection direction.

Dimensions

Total Volume - Displays the volume of the slab (length times width times height)

Net Volume - Displays the volume of the slab minus the volumes of all openings in the slab.

Total Surface - Displays the surface area of the slab (top surface plus the bottom surface plus the side surfaces).

Net Surface - Displays the surface area of the slab minus the surface area of all the openings in the slab.

Total Projected Area - Displays the area of the slab (length times width).

Net Projected Area - Displays the area of the slab minus the area of all the openings in the slab.

Number of Openings - Displays the number of openings in the slab.

Angle - Displays the angle at which the slab is placed from the plane.

Low Point - Displays the lowest elevation of the slab.

High Point - Displays the highest elevation of the slab.

Bottom Face - Displays the lowest elevation of the slab.

Top Face - Displays the highest elevation of the slab.

Weight and CG

Displays the center-of-gravity and the weight of the selected slab. 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 slab.

Wet Weight - Displays the wet weight of the slab.

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

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

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

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

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

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

Fabrication and Construction

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

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

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

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

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.

See Also

Slab Properties Dialog Box (on page 107)

Layer Tab (Slab Properties Dialog Box)

Displays the layers for the selected slab type on the **General** tab. Layers are defined in the **StructSlabLayer.xls** workbook. Refer to the Struc*ture Reference Data Guide for* more information about this workbook.

Composition - Select the composition that you want to use for the slab type. The layers in the select composition appear.

Total Thickness - Displays the total thickness of all the layers in the slab.

Layer - Select the layer of the composition to view or edit properties for.

Property - Displays the defined properties for the active layer.

Value - Specify a value for the layer property.

See Also

Slab Properties Dialog Box (on page 107)

Understanding Plane Methods

A common activity while working with slabs is defining planes. These planes are used to define the location of the slab or as a sketching plane for the slab boundaries. Usually, but not necessarily, these planes are based from an existing object or from a grid plane created using the Grids task. There are six methods for defining planes:

- **Coincident** - Defines a plane coincident to another plane. For more information, see *Define Plane as Coincident* (on page 112).

- **Offset from Plane** - Defines a plane at a specified offset distance from another plane. For more information, see *Define Plane using Offset from Plane* (on page 112).
- **Angle from Plane** - Defines a plane at a specified angle or slope to another plane. You must define an axis of rotation and the angle or slope. For more information, see *Define Plane using Angle from Plane* (on page 112).
- **Plane by Point and Vector** - Defines a plane using a vector normal to the plane being defined and a third point to define the plane position along the vector. For more information, see *Define Plane using Vector and Point* (on page 113).
- **Plane by Three Points** - Defines a plane using three points that you identify in the model. For more information, see *Define Plane using Three Points* (on page 113).

See Also
Place Slab Ribbon (on page 103)

Define Plane using Angle from Plane

1. Select **Angle from Plane**.
2. In the model or **Workspace Explorer**, select the base plane from which to angle the slab that you are defining.
3. Select a rotation axis.

   TIPS

   - The rotation axis can be a grid line or a member edge.
   - The rotation axis must be parallel to the surface of the plane selected in step 2.
4. In the **Angle** box, specify the rotation angle of the slab relative to the selected base plane.

See Also
Place Slab Ribbon (on page 103)

Define Plane as Coincident

1. Select **Coincident**.
2. In the model or **Workspace Explorer**, select the plane to place the new slab coincident with.

See Also
Place Slab Ribbon (on page 103)

Define Plane using Offset from Plane

1. Select **Offset from a Plane**.
2. In the model or **Workspace Explorer**, select the base plane to offset the new slab from.
3. In the **Offset** box, type the offset distance from the plane.

TIP By moving the pointer back and forth over the base plane in the graphic view, you can change the offset direction. You can also change the direction by changing the sign in the **Offset** box.

4. To use the **Step** box, enter a value, and then move the pointer in the graphics view. The offset follows the pointer, incrementing by the step value that you specified.

See Also

Place Slab Ribbon (on page 103)

Define Plane using Three Points

1. Select **Plane by Three Points**.
2. Specify the first point that defines the plane.
3. Specify the second point that defines the plane.
4. Specify the third point that defines the plane.

NOTE You can define points on the surface of objects or on grid planes. Locate points on grid planes, especially at intersections, by using the **Tools > Add to SmartSketch List** command.

See Also

Place Slab Ribbon (on page 103)

Define Plane using Vector and Point

1. Select **Vector & Point**.
2. Define the first vector point.
3. Define the second vector point.
4. Specify the plane location normal to the vector at which to place the slab.

See Also

Place Slab Ribbon (on page 103)

Place a Slab by Selecting Boundary Objects

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 112)
   Define Plane using Offset from Plane (on page 112)
   Define Plane using Angle from Plane (on page 112)
   Define Plane using Vector and Point (on page 113)
   Define Plane using Three Points (on page 113)
3. Click **Accept**.
4. Click **Select Boundaries**.

5. Select objects in the model to define the boundaries of the slab. You can select members, grid lines, edges of equipment, edges of other slabs, or most anything else in the model to define the slab boundaries. The selected objects must roughly define a close shape.
6. Click **Accept** ✓.
7. Set the slab system, type, priority, face position, and other properties.
8. Click **Finish**.

See Also

Place Slab Command (on page 103)

Place a Slab by Drawing 2-D Boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 112)
   Define Plane using Offset from Plane (on page 112)
   Define Plane using Angle from Plane (on page 112)
   Define Plane using Vector and Point (on page 113)
   Define Plane using Three Points (on page 113)
3. Click **Accept** ✓.
4. Click **Add References to Sketch 2D**.
5. Select objects in the model that you want to see when drawing the slab boundaries in the 2-D environment.
6. Click **Draw**.
7. In the 2-D environment, draw the boundaries of the slab. The boundaries that you draw must be closed shape. You can only draw one slab at a time in the 2-D environment.
8. In the 2-D environment, click **Close**.
9. Click **Accept** ✓.
10. Set the slab system, type, priority, face position, and other properties.
11. Click **Finish**.

See Also

Place Slab Command (on page 103)

Place a Slab by Drawing 3-D Boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 112)
   Define Plane using Offset from Plane (on page 112)
   Define Plane using Angle from Plane (on page 112)
   Define Plane using Vector and Point (on page 113)
   Define Plane using Three Points (on page 113)

3. Click **Accept** ✓.
4. Click **Sketch 3D** 3D.
5. Sketching a path that defines the outside boundary of the slab.

   Define a Path (on page 209)
6. Click **Finish**.
7. Click **Accept** ✓.
8. Set the slab system, type, priority, face position, and other properties.
9. Click **Finish**.

See Also

Place Slab Command (on page 103)
Sketching in 3-D (on page 201)

Place a Slab by Selecting Objects and Drawing 2-D Boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 112)
   Define Plane using Offset from Plane (on page 112)
   Define Plane using Angle from Plane (on page 112)
   Define Plane using Vector and Point (on page 113)
   Define Plane using Three Points (on page 113)
3. Click **Accept** ✓.
4. Click **Select Boundaries**.
5. Select objects in the model to define part of the slab boundaries. You can select members, grid lines, edges of equipment, edges of other slabs, or most anything else in the model to define the slab boundaries.
6. Click **Add References to Sketch 2D**.
7. Select objects in the model that you want to see when drawing the slab boundaries in the 2-D environment.
8. Click **Draw**.
9. In the 2-D environment, draw the remaining boundaries of the slab. The object boundaries that you selected are displayed in black. The objects that you selected for sketch landmarks display in blue. The boundaries that you selected and the boundaries that you draw must form closed shape.
10. In the 2-D environment, click **Close**.
11. Click **Accept** ✓.
12. Set the slab system, type, priority, face position, and other properties.
13. Click **Finish**.

See Also

Place Slab Command (on page 103)

Place a Slab by Selecting Objects and Drawing 3-D Boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 112)
   Define Plane using Offset from Plane (on page 112)
   Define Plane using Angle from Plane (on page 112)
   Define Plane using Vector and Point (on page 113)
   Define Plane using Three Points (on page 113)
3. Click **Accept** ✓.
4. Click **Select Boundaries**.
5. Select objects in the model to define part of the slab boundaries. You can select members, grid lines, edges of equipment, edges of other slabs, or most anything else in the model to define the slab boundaries.
6. Click **Sketch 3D**.
7. Sketching a path that defines the remaining part of the slab boundary.
8. *Define a Path* (on page 209)
9. Click **Finish**.
10. Click **Accept** ✓.
11. Set the slab system, type, priority, face position, and other properties.
12. Click **Finish**.

See Also

Place Slab Command (on page 103)
Sketching in 3-D (on page 201)

Solve Ambiguous Boundaries

The command switches to the solve ambiguity mode automatically if ambiguous boundaries exist after clicking **Accept** ✓ or **Finish**. All possible bounded areas appear in the graphic view outlined in green.

1. Move the pointer over a bounded area, and then click to select that bounded area. The selected area highlights in yellow.
2. Continue to select bounded areas until you have defined the entire bounded area that you want. A correct solution to ambiguity meets these conditions:
   - Each bounded area must have a common edge with at least one other bounded area.
   - Each boundary must be used by at least one bounded area.

NOTE To remove a bounded area, select the area again.

See Also

Place Slab Command (on page 103)

Edit Slab Object Boundaries

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Slabs**.
3. Select the slab to edit.
4. On the ribbon, click **Define Boundaries**.
5. Redefine the boundaries by clicking existing boundaries to clear them and by clicking new objects to use as the new boundaries.
6. Click **Accept**.
7. Click **Finish**.

See Also

Place Slab Command (on page 103)
Solve Ambiguous Boundaries (on page 116)

Edit 2-D Drawn Slab Boundaries

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Slabs**.
3. Select the slab to edit.
4. On the ribbon, click **Define Boundaries**.
5. On the ribbon, click **Draw**.
6. Click **Select Tool** in the 2-D environment.
7. Select the slab boundary to edit.
8. Click **Edit > Ungroup Objects** and ungroup the boundary to edit. Do **not** ungroup all the boundaries.
9. Edit the boundary as needed.
10. Click **Close** in the 2-D environment when you are finish editing the boundary objects.
11. Click **Accept**.
12. Click **Finish**.

See Also

Place Slab Command (on page 103)
Solve Ambiguous Boundaries (on page 116)

Edit 3-D Drawn Slab Boundaries

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Slabs**.
3. Select the slab to edit.
4. On the ribbon, click **Define Boundaries**.
5. On the ribbon, click **Sketch 3D**.
6. Edit the path.

Modify a Straight Segment in a Path (on page 211)
Modify an Arc in a Path (on page 212)
Modify a Turn in a Path (on page 213)
Move Segments of a Path (on page 211)

7. Click **Finish**.
8. Click **Accept** ✓.
9. Click **Finish**.

See Also

Place Slab Command (on page 103)
Solve Ambiguous Boundaries (on page 116)

Edit Slab Properties

1. Click **Select** on the vertical toolbar.
2. Select **Slabs** in the locate filter.
3. Select the slab to edit.
4. Click **Edit > Properties**.
5. Edit the slab properties as needed.

See Also

Slab Properties Dialog Box (on page 107)
Place Slab Command (on page 103)

Delete a Slab

1. Click **Select** on the vertical toolbar.
2. Select **Slabs** in the **Locate Filter**.
3. Select the slab to delete.
4. Click **Delete** ✕.

See Also

Place Slab Command (on page 103)

Place Wall Command

Places curved and straight walls in the model. Use this command to place parapets, retaining walls, bearing walls, nonbearing walls, and foundation walls. Like slabs, you can cut openings in walls using the *Place Opening Command* (on page 135). You can use the *Place Equipment Command* (on page 151) to place doors and windows in walls.

For more information about walls, read *Working with Walls and Slabs* (on page 101).

See Also

Wall System Properties Dialog Box (on page 120)
Working with Walls and Slabs (on page 101)
Add End Segment to Existing Wall using 2-D Sketch (on page 127)

Add Fillet Corner to Existing Wall using 2-D Sketch (on page 128)
Add Middle Segments to Existing Wall using 2-D Sketch (on page 129)
Add End Segment to Existing Wall using 3-D Sketch (on page 133)
Create Gap in Existing Wall using 2-D Sketch (on page 131)
Delete a Wall (on page 134)
Modify Wall Length using 2-D Sketch (on page 130)
Place a Wall by 2-D Sketch (on page 127)
Place a Wall by 3-D Sketch (on page 132)
Rotate an Existing Wall using 2-D Sketch (on page 132)
Toggle Wall Corner (on page 133)
Place Wall Ribbon (on page 119)
Edit Wall Properties (on page 134)

Place Wall Ribbon

Specifies the properties for the wall that you are placing.

Wall Properties - Activates the **Wall Properties** dialog box. You can use this dialog box to specify additional wall properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Wall System Properties Dialog Box* (on page 120).

Sketching Plane - Select the support plane for the bottom of the wall.

Select a Path - Select the **Sketch 2D** or **Sketch 3D** option.

Select Boundaries - Select additional graphic objects to be boundaries for the wall. You can select members, other walls, grid planes, and surfaces. An example of when to use this option would be when placing a wall under a pitched roof.

Finish - Places the wall using the defined parameters.

Cancel - Clears all selected planes or boundary objects.

Accept - Accepts all selected planes or boundary objects.

Sketch 2D - Opens the sketch 2-D environment in which you can draw the path of the wall. You can draw more than one wall at a time in the 2-D environment. However, if you draw more than one wall path in the 2-D environment, the walls will move as a single unit if you move any of the walls regardless of whether or not the walls are connected to each other. If you will need to move any of the walls individually, you must draw that wall's path in 2-D environment by itself.

Sketch 3D - Activates the *Create Path Ribbon* (on page 202) with which you define the wall path. Select this option to sketch the path of the wall in the 3-D environment.

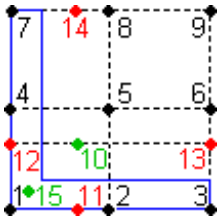
System - Select the parent system for the wall that you are placing. Systems are defined using the Systems and Specifications task.

Type - Select the type of wall to place. The properties change depending on the wall type that you select.

Composition - Select a wall composition from the list. Available compositions depend on the type of wall being placed.

Name - Displays the name of the wall that you are placing.

Position - Specify the relative position of the wall to the path. Fifteen positions are available. The location of positions 10 (center-of-gravity) and 15 (shear center) depend on the wall's section shape. The local z- axis of the wall and the center-of-gravity point of the section define positions 11 and 14. The local y-axis of the wall and the center-of-gravity point of the section define positions 12 and 13.



Total Thickness - Type the thickness of the wall. The default thickness is based on the selected wall type and composition.

NOTE If the wall has more than one layer, then the **Total Thickness** box is read-only. To edit the wall thickness, use the **Layer** tab of the **Wall Properties** dialog box. For more information, see *Edit Wall Properties* (on page 134).

Maximum Height - Enter the maximum height for the wall.

Reflect - Reflects or mirrors the cross-section about the wall's path. This parameter affects both symmetric and asymmetric sections.

See Also

Add End Segment to Existing Wall using 2-D Sketch (on page 127)
Add Fillet Corner to Existing Wall using 2-D Sketch (on page 128)
Add Middle Segments to Existing Wall using 2-D Sketch (on page 129)
Add End Segment to Existing Wall using 3-D Sketch (on page 133)
Create Gap in Existing Wall using 2-D Sketch (on page 131)
Delete a Wall (on page 134)
Modify Wall Length using 2-D Sketch (on page 130)
Place a Wall by 2-D Sketch (on page 127)
Place a Wall by 3-D Sketch (on page 132)
Rotate an Existing Wall using 2-D Sketch (on page 132)
Toggle Wall Corner (on page 133)
Place Wall Command (on page 118)

Wall System Properties Dialog Box

Specifies the properties for the wall system that you are editing. You can define the default values in the **StructWallGeneral.xls** and **StructWallLayer.xls** workbooks.

See Also

General Tab (Wall System Properties Dialog Box) (on page 121)
Layer Tab (Wall System Properties Dialog Box) (on page 121)
Section Tab (Wall System Properties Dialog Box) (on page 122)
Edit Wall Properties (on page 134)

General Tab (Wall System Properties Dialog Box)

The **General** tab displays the wall 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 wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, 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.

Category - Select the properties that you want to view for the wall. Wall properties are divided into several different categories: *Standard* (see "General Tab (Wall System Properties Dialog Box)" on page 121). You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Parent System - Select the system to which the wall belongs.

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

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

Type - Select the wall type.

Composition - Select a wall composition from the list. Available compositions depend on the type of wall being placed.

Bearing - Indicates whether or not the wall is a load bearing wall.

Priority - Select the priority to assign to the wall system. The priority is used to group walls.

Continuity - Select the wall's continuity type.

Layer Tab (Wall System Properties Dialog Box)

Displays the layers for the selected wall type on the **General** tab. Layers are defined in the **StructWallLayer.xls** workbook. Refer to the Stru*cture Reference Data Guide for* more information about this workbook.

Composition - Select the composition that you want to use for the wall type. The layers in the select composition appear.

Total Thickness - Displays the total thickness of all the layers in the wall.

Layer - Select the layer of the composition to view or edit properties for.

Property - Displays the defined properties for the active layer.

Value - Specify a value for the layer property.

See Also
Wall System Properties Dialog Box (on page 120)

Section Tab (Wall System Properties Dialog Box)

The **Section** tab displays the properties for the wall's cross section. 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 wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, 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.

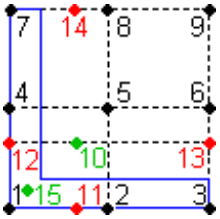
Category - Select the properties that you want to view for the cross section. Cross section properties are divided into several different categories: *Standard* (see "Section Tab (Wall System Properties Dialog Box)" on page 122). You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Wall Cross Sections Name - Displays the name of the wall cross section.

Position - Specify the relative position of the wall to the path. Fifteen positions are available. The location of positions 10 (center-of-gravity) and 15 (shear center) depend on the wall's section shape. The local z- axis of the wall and the center-of-gravity point of the section define positions 11 and 14. The local y-axis of the wall and the center-of-gravity point of the section define positions 12 and 13.



Horizontal Offset - Type the horizontal offset between the defined wall path and the wall's selected **Position** location.

Vertical Offset - Type the vertical offset between the defined wall path and the wall's selected **Position** location.

Thickness - Type the thickness of the wall.

Height - Type the height of the wall.

Reflect - Reflects or mirrors the cross-section about the wall's path. This parameter affects both symmetric and asymmetric sections.

Angle - Enter the angle of the wall from vertical. You can enter both positive and negative angles.

Wall Part Properties Dialog Box

Specifies the properties for the wall part that you are editing. You can define the default values in the **StructWallGeneral.xls** and **StructWallLayer.xls** workbooks.

See Also

General Tab (Wall Part Properties Dialog Box) (on page 123)
Layer Tab (Wall Part Properties Dialog Box) (on page 125)
Section Tab (Wall Part Properties Dialog Box) (on page 126)
Edit Wall Properties (on page 134)

General Tab (Wall Part Properties Dialog Box)

The **General** tab displays the wall 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 wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, 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.

Category - Select the properties that you want to view for the wall. Wall properties are divided into several different categories: *Standard* (see "General Tab (Wall Part Properties Dialog Box)" on page 123), *Weight and CG* (see "General Tab (Wall Part Properties Dialog Box)" on page 123), and *Dimensions* (see "General Tab (Wall Part Properties Dialog Box)" on page 123). You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Parent System - Select the system to which the wall belongs.

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

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

Composition - Displays the composition of the wall.

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

Reporting Type - Select the reporting requirements code for the wall part. Valid selections are defined in the Catalog task in the Reporting Type select list.

Dimensions

Total Volume - Displays the volume of the wall (length × width × height)

Net Volume - Displays the volume of the wall minus the volumes of all openings in the wall.

Total Surface - Displays the surface area of the wall (top surface plus the bottom surface plus the side surfaces).

Net Surface - Displays the surface area of the wall minus the surface area of all the openings in the wall.

Projected Surface - Displays the surface area of the wall's bottom surface.

Net Projected Surface - Displays the surface area of the wall's bottom surface minus the surface area of all openings in the wall.

Length - Displays the wall length.

Weight and CG

Displays the center-of-gravity and the weight of the selected wall. 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 wall.

Wet Weight - Displays the wet weight of the wall.

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

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

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

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

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

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

Fabrication and Construction

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

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

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

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

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.

Layer Tab (Wall Part Properties Dialog Box)

Displays the layers for the selected wall type on the **General** tab. Layers are defined in the **StructWallLayer.xls** workbook. Refer to the Struc*ture Reference Data Guide for* more information about this workbook.

Composition - Select the composition that you want to use for the wall type. The layers in the select composition appear.

Total Thickness - Displays the total thickness of all the layers in the wall.

Layer - Select the layer of the composition to view or edit properties for.

Property - Displays the defined properties for the active layer.

Value - Specify a value for the layer property.

See Also

Wall Part Properties Dialog Box (on page 123)

Section Tab (Wall Part Properties Dialog Box)

The **Section** tab displays the properties for the wall's cross section. 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 wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, 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.

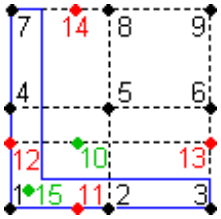
Category - Select the properties that you want to view for the cross section. Cross section properties are divided into several different categories: *Standard* (see "Section Tab (Wall Part Properties Dialog Box)" on page 126). You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Wall Cross Sections Name - Displays the name of the wall cross section.

Position - Specify the relative position of the wall to the path. Fifteen positions are available. The location of positions 10 (center-of-gravity) and 15 (shear center) depend on the wall's section shape. The local z- axis of the wall and the center-of-gravity point of the section define positions 11 and 14. The local y-axis of the wall and the center-of-gravity point of the section define positions 12 and 13.



Horizontal Offset - Type the horizontal offset between the defined wall path and the wall's selected **Position** location.

Vertical Offset - Type the vertical offset between the defined wall path and the wall's selected **Position** location.

Thickness - Type the thickness of the wall.

Height - Type the height of the wall.

Reflect - Reflects or mirrors the cross-section about the wall's path. This parameter affects both symmetric and asymmetric sections.

Angle - Enter the angle of the wall from vertical. You can enter both positive and negative angles.

Place a Wall by 2-D Sketch

1. Click **Place Wall** on the vertical toolbar.
2. Define the sketching plane. The sketching plane defines the bottom elevation of the wall.
3. Click **Sketch 2D**.
4. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.
5. Click **Draw**.
6. In the 2-D environment, draw the path of the wall.

   TIP You can draw more than one wall path at a time in the 2-D environment.
7. In the 2-D environment, click **Close**.
8. Click **Finish**.
9. Set the wall system, type, composition, and other properties.
10. Optionally, click **Select Boundaries** to limit the height of the wall.
11. Click **Finish**.

    The software places the walls in the model.
12. The software maintains the same sketching plane and prompts you to select references for the next wall. Go back to step 4.

    -OR-

    Click **Cancel** to return to the main ribbon to define a new sketching plane. Go back to step 2.

NOTE If you draw more than one wall path in the 2-D environment, the walls will move as a single unit if you move any of the walls regardless of whether or not the walls are connected to each other. If you will need to move any of the walls individually, you must draw that wall's path in 2-D environment by itself.

See Also

Place Wall Command (on page 118)

Add End Segment to Existing Wall using 2-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to add to.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.
7. Click **Draw**.
8. In the 2-D environment, add on to the path of the existing wall path.

TIP If you exit the 2D environment now, you will have two walls that are connected. If you want a single wall, you need to continue on to the next steps.

9. Click **Tools > Custom Commands**.
10. Browse to *[Product Directory]*\3D\Common2D\Symbol2D\Bin folder. By default, the product direction is c:\Program Files\SmartPlant.
11. Select **GroupModify.ocx** in the folder, and then select **Open**.
12. Click **Select Graphic Group** on the ribbon.
13. Select the original wall path.
14. Click **Select elements to add to group or remove from it** on the ribbon.
15. Select the new wall path segments that you just placed.
16. Click **Add** on the ribbon.
17. In the 2-D environment, click **Close**.
18. Click **Finish**.
19. Set the wall system, type, composition, and other properties.
20. Optionally, click **Select Boundaries** to limit the height of the wall.
21. Click **Finish**.

    The software places the new wall segments in the model.

See Also

Place Wall Command (on page 118)

Add Fillet Corner to Existing Wall using 2-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to add the fillet corner to.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.
7. Click **Draw**.
8. Click **Fillet** on the vertical toolbar.
9. Place the fillet on the wall path by identifying the two sides for the fillet and then the radius.
10. Click **Tools > Custom Commands**.
11. Browse to *[Product Directory]*\3D\Common2D\Symbol2D\Bin folder. By default, the product direction is c:\Program Files\SmartPlant.
12. Select **GroupModify.ocx** in the folder, and then select **Open**.
13. Click **Select Graphic Group** on the ribbon.
14. Select the original wall path.
15. Click **Select elements to add to group or remove from it** on the ribbon.

16. Select the fillet segment of the wall path that you just placed.
17. Click **Add** on the ribbon.
18. In the 2-D environment, click **Close**.
19. Click **Finish**.
20. Set the wall system, type, composition, and other properties.
21. Optionally, click **Select Boundaries** to limit the height of the wall.
22. Click **Finish**.

    The software places the fillet in the wall.

See Also

Place Wall Command (on page 118)

Add Middle Segments to Existing Wall using 2-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to add to.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.
7. Click **Draw**.
8. Click **Split** on the vertical toolbar.
9. Identify the two locations on the wall between which you want to add to the wall.
10. Click **Tools > Custom Commands**.
11. Browse to *[Product Directory]*\3D\Common2D\Symbol2D\Bin folder. By default, the product direction is c:\Program Files\SmartPlant.
12. Select **GroupModify.ocx** in the folder, and then select **Open**.
13. Click **Select Graphic Group** on the ribbon.
14. Select the original wall path.
15. Click **Select elements to add to group or remove from it** on the ribbon.
16. Select the segment of the wall path between the two points that you identified during the **Split** command.
17. Click **Remove** on the ribbon.
18. Click **Select** on the vertical toolbar.
19. Select the segment between the two points that you just removed from the group.
20. Press **Delete** on the keyboard.

    You should now have a wall path with a gap in it.
21. Using commands on the vertical toolbar, like the **Line** command, draw the new wall segment in the gap. Make sure you connect to the two end of the gap.

22. Click **Tools > Custom Commands**.
23. Browse to *[Product Directory]*\3D\Common2D\Symbol2D\Bin folder.
24. Select **GroupModify.ocx** in the folder, and then select **Open**.
25. Click **Select Graphic Group** on the ribbon.
26. Select the original wall path.
27. Click **Select elements to add to group or remove from it** on the ribbon.
28. Select the new wall path segments that you just placed.
29. Click **Add** on the ribbon.
30. In the 2-D environment, click **Close**.
31. Click **Finish**.
32. Set the wall system, type, composition, and other properties.
33. Optionally, click **Select Boundaries** to limit the height of the wall.
34. Click **Finish**.

    The software places the new wall segments in the model.

See Also

Place Wall Command (on page 118)

Modify Wall Length using 2-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to modify the length of.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when modifying the wall length in the 2- D environment.
7. Click **Draw**.
8. Click **Bottom Up** on the ribbon.
9. Select the wall segment to modify.
10. In the **Length** box, key in the new length for the segment.
11. In the 2-D environment, click **Close**.
12. Click **Finish**.
13. Optionally, edit the wall system, type, composition, and other properties.
14. Optionally, click **Select Boundaries** to limit the height of the wall.
15. Click **Finish**.

See Also

Place Wall Command (on page 118)

Create Gap in Existing Wall using 2-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to place the gap in.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see in the 2-D environment.
7. Click **Draw**.
8. Click **Split** on the vertical toolbar.
9. Identify the two locations on the wall between which you want to place the gap.
10. Click **Tools > Custom Commands**.
11. Browse to *[Product Directory]*\3D\Common2D\Symbol2D\Bin folder. By default, the product direction is c:\Program Files\SmartPlant.
12. Select **GroupModify.ocx** in the folder, and then select **Open**.
13. Click **Select Graphic Group** on the ribbon.
14. Select the original wall path.
15. Click **Select elements to add to group or remove from it** on the ribbon.
16. Select the segment of the wall path where the gap should be.
17. Click **Remove** on the ribbon.
18. Click **Select** on the vertical toolbar.
19. Select the segment between the two points that you just removed from the group.
20. Press **Delete** on the keyboard.

    You should now have a wall path with a gap in it.
21. Click **Tools > Custom Commands**.
22. Browse to *[Product Directory]*\3D\Common2D\Symbol2D\Bin folder. By default, the product direction is c:\Program Files\SmartPlant.
23. Select **GroupModify.ocx** in the folder, and then select **Open**.
24. Click **Select Graphic Group** on the ribbon.
25. Select the original wall path.
26. Click **Select elements to add to group or remove from it** on the ribbon.
27. Select all the segments on one side of the gap.
28. Click **Remove** on the ribbon.

    By removing all the segments on one side of the gap from the original wall group, you create a new second wall. The software currently does not support having a gap in a single wall.
29. In the 2-D environment, click **Close**.
30. Click **Finish**.
31. Set the wall system, type, composition, and other properties.

32. Optionally, click **Select Boundaries** to limit the height of the wall.
33. Click **Finish**.

    The software removes the segment from the wall thus creating the gap.

See Also

Place Wall Command (on page 118)

Rotate an Existing Wall using 2-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to rotate.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.
7. Click **Draw**.
8. Click **Select** on the vertical toolbar.
9. Select the wall path that you want to rotate.
10. In the **Change** toolbar, click **Rotate**.
11. Rotate and move the wall as needed.
12. In the 2-D environment, click **Close**.
13. Click **Finish**.
14. Set the wall system, type, composition, and other properties.
15. Optionally, click **Select Boundaries** to limit the height of the wall.
16. Click **Finish**.

See Also

Place Wall Command (on page 118)

Place a Wall by 3-D Sketch

1. Click **Place Wall** on the vertical toolbar.
2. Define the sketching plane.
3. Click **Sketch 3D**.
4. Sketch a path that defines the position of the wall.

   Define a Path (on page 209)
5. Click **Finish**.
6. Set the wall system, type, composition, and other properties.
7. Optionally, click **Select Boundaries** to limit the height of the wall.
8. Click **Finish**.

The software places the wall in the model.

9. The software maintains the same sketching plane and resets itself for you to define the next wall path. Go back to step 4.

   -OR-

   Click **Cancel** to return to the main ribbon to define a new sketching plane. Go back to step 2.

See Also

Place Wall Command (on page 118)
Sketching in 3-D (on page 201)

Add End Segment to Existing Wall using 3-D Sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to add to.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 3D**.
6. Click **Create** on the ribbon.
7. Select one of the boxes on the existing wall path as your starting point. If you can add to the wall path from that path point, the box will turn red to indicate that you have selected it. If the box does not turn red, you cannot add on to the wall from that path point.
8. Define the addition to the path using the path commands. For more information, see *Define a Path* (on page 209).
9. Click **Finish**.
10. Edit the wall system, type, composition, and other properties if needed.
11. Optionally, click **Select Boundaries** to limit the height of the wall.
12. Click **Finish**.

    The software places the wall extension in the model.

See Also

Place Wall Command (on page 118)
Sketching in 3-D (on page 201)

Toggle Wall Corner

When two separate walls join at a corner, one wall extends past the other wall to form the corner. You can use this procedure to toggle which wall extends past the other.

1. Click **Tools > Custom Commands** from the menu.

   TIP If you have already added the custom command, you can skip ahead to step 6.

2. Click **Add** on the **Custom Commands** dialog box.
3. In the **Command Progid** box, type in **WallCornerConnCmds.CmdRevertCornerConn**.
4. In the **Command Name** box, type **Wall Corner Command**.
5. Click **OK**.

6. Select **Wall Corner Command** from the list of command names.
7. Click **Run**.
8. Select the first wall at the corner.
9. Select the other wall at the corner.
10. Click **Finish**.

See Also

Place Wall Command (on page 118)

Modify Wall Cross Section

1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the locate filter.
3. Select the wall to edit.
4. On the ribbon bar, select a new cross section using the **Composition** option.

See Also

Wall System Properties Dialog Box (on page 120)
Place Wall Command (on page 118)

Edit Wall Properties

1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the locate filter.
3. Select the wall to edit.
4. Click **Edit > Properties**.
5. Edit the wall properties as needed.

See Also

Wall System Properties Dialog Box (on page 120)
Place Wall Command (on page 118)

Delete a Wall

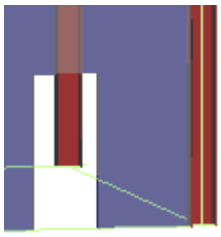
1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the **Locate Filter**.
3. Select the wall to delete.
4. Click **Delete**.

See Also

Place Wall Command (on page 118)

Place Opening Command

Places an opening (hole) in an existing slab, wall, or member. The shape of the opening can be defined by placing a pre-defined shape from the catalog, sketching the opening outline, or defining the boundaries for the opening. In addition, you can control the depth of the opening to create a fully penetrating hole or a recessed opening.



See Also

Change a Recess Opening to a Through Opening (on page 145)
Change a Through Opening to a Recess Opening (on page 145)
Delete an Opening (on page 145)
Place an Opening by Boundaries (on page 139)
Place an Opening by Drawing (on page 140)
Place an Opening by Shape (on page 140)
Place Opening Ribbon (on page 135)

Place Opening Ribbon

Specifies the properties for the opening that you are placing or editing.

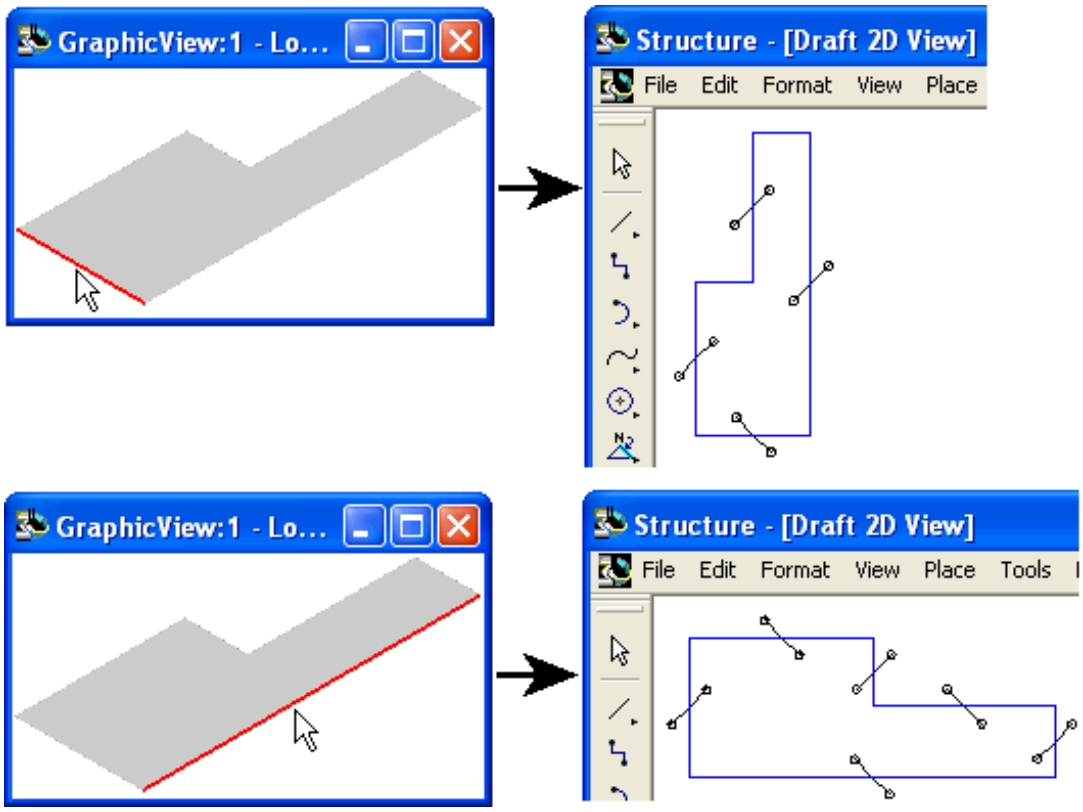
Opening Properties - Activates the **Opening Properties** dialog box. You can use this dialog box to specify opening properties that you cannot set on the ribbon. For more information, see *Opening Properties Dialog Box* (on page 138).

Structure - Select the object in which to place an opening. You can select members or slabs.

Sketching Plane - Select the plane on the object on which to draw the opening. If you are placing an opening that does not go all the way through the object, the sketching plane defines the side of the object in which the partial opening is placed.

Define Orientation - Select this option if you plan to sketch the opening and want to define the orientation of the object, in which the opening is being placed, in the 2D environment. Most of the time, the default orientation that the software uses is appropriate for the opening. However, use this option if you are placing an opening in an multi- sided slab, for example, and you need to know that the correct side of the slab is selected for the opening.

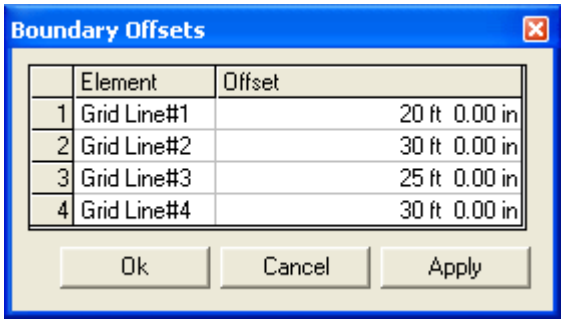
Edge for 2D X-axis - Select this option to specify the object edge that should be oriented along the bottom of the 2D environment. This option is only available when you select the **Define Orientation** option.



End Point - Select this option to specify the lower left corner of the object as you want it to appear in the 2D environment. This option is only available when you select the **Define Orientation** option.

Boundary - Specify the boundaries of the opening. If you select objects in the model and those objects are moved, the software automatically resizes the opening to maintain the boundary relationship. For example, you can select the outside edge of a beam flange as a boundary along one side of an opening. Later, if the section size for the beam is changed and the flange edge moves out an inch, the software automatically moves the edge of the opening an inch to match the new location of the flange edge.

Boundary List - Activates the **Boundary Offsets** dialog box. This dialog box allows you to define an offset distance for each boundary that you have defined. Select a row to highlight the boundary in the model. Then, type the distance from the boundary to place the edge of the opening. Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary.



Finish - Places the opening using the defined parameters.

Cancel - Rejects the objects that you have selected.

Accept - Confirms the objects that you have selected.

Select Boundaries - Select this option to select objects in the model to define the boundaries of the opening. This option is only available when you are specifying boundaries.

Add References to Sketch 2D - Adds an object as a reference object in the 2-D environment. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model. This option is only available when you are drawing the boundaries of the opening.

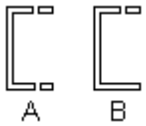
Draw - Activates the Structure Draft 2D View environment. Use this environment to sketch the outline of the opening in the structure that you have selected.

Shape - Specifies how you want to define the opening shape. Select **Sketch** to draw the opening shape. Select **More** to select an opening shape from the catalog. This option is only used when you are not using boundaries to define the opening shape.

Cutting Limit - Select the cutting depth method for the opening.

- Select **Through-All** to completely penetrate the object in which the opening is placed. This option is available for all objects in which you can place an opening.

- Select **Through-Next** to penetrate only one side of an object. For example, a channel section member in which you want an opening only on the top flange. This option is available only when the object that you select for the opening is a member whose cross section has parallel webs or flanges or a wall that turns back on itself. In the figure, A is the **Through-All** option and B is the **Through-Next** option.



- Select **User Defined** if you want to specify the penetration depth of the opening. This option is available for all objects in which you can place an opening.

Cutting Depth - Specify the cutting depth for the opening. The depth is measured from the sketching plane that you specified.

See Also

Opening Properties Dialog Box (on page 138)
Place Opening Command (on page 135)

Opening Properties Dialog Box

Specifies the properties for the opening that you are editing.

See Also

General Tab (Opening Properties Dialog Box) (on page 138)
Place Opening Ribbon (on page 135)

General Tab (Opening Properties Dialog Box)

The **General** tab displays the opening 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 opening and then selected the properties command, only the common properties between the selected openings display.

When viewing properties for a single opening, 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.

Category - Select the properties that you want to view for the opening. Opening properties have one category: Standard. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the opening. Because opening reference data is fully customizable, only the common properties are documented.

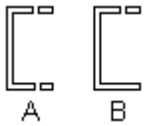
Name - Specify a name for the opening.

Naming Rule - Select the naming rule to use to name the opening. Set this option to **User Defined** if you want to name the opening yourself.

Description - Type a description for the opening.

Cutting Limit - Select the cutting depth method for the opening.

- Select **Through-All** to completely penetrate the object in which the opening is placed. This option is available for all objects in which you can place an opening.
- Select **Through-Next** to penetrate only one side of an object. For example, a channel section member in which you want an opening only on the top flange. This option is available only when the object that you select for the opening is a member whose cross section has parallel webs or flanges or a wall that turns back on itself. In the figure, A is the **Through-All** option and B is the **Through-Next** option.



- Select **User Defined** if you want to specify the penetration depth of the opening. This option is available for all objects in which you can place an opening.

Cutting Depth - Specify the cutting depth for the opening. The depth is measured from the sketching plane that you specified.

See Also

Opening Properties Dialog Box (on page 138)

Place an Opening by Boundaries

1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which you want to place the opening. You can select walls, slabs, and members.
3. Select the side of the object in which you want to place the opening.

   TIP If you are placing an opening that completely penetrates the object, the side (surface) selection is not important. However, if you are placing an opening that does not completely penetrate the object, then the surface selection is important because the penetration depth is measured from the selected surface.
4. Click **Select Boundaries**.
5. Select objects in the model to use as boundaries of the opening.

   NOTE You must select the boundary objects in a clockwise or counterclockwise order.
6. Click **Accept**.
7. Optionally, define any offset from the boundaries that you have selected.

   TIP Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary.
8. Click **OK** on the **Boundary** dialog box.
9. Specify the cutting limit to use.
10. Click **Finish**.

Place an Opening by Shape

1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which to place the opening. You can select walls, slabs, and members.
3. Select the side of the object in which to place the opening.

   TIP If you are placing an opening that completely penetrates the object, the side (surface) selection is not important. However, if you are placing an opening that does not completely penetrate the object, then the surface selection is important because the penetration depth is measured from the selected surface.
4. In the **Shape** option, select **More...**.
5. Select the catalog shape that you want to place, and then click **OK**.
6. Drag the shape to where you want the opening on the object.
7. Click **Close** on the ribbon bar.
8. Specify the cutting limit to use.
9. Click **Finish**.

See Also

Place Opening Command (on page 135)

Place an Opening by Drawing

1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which to place the opening. You can select walls, slabs, and members.
3. Select the side of the object in which to place the opening.

   TIP If you are placing an opening that completely penetrates the object, the side (surface) selection is not important. However, if you are placing an opening that does not completely penetrate the object, then the surface selection is important because the penetration depth is measured from the selected surface.
4. Click **Add References to Sketch 2D**.
5. Select objects in the 3-D model near the object that you are placing the opening in. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model.
6. Click **Draw**.
7. Using the available drawing commands in the 2-D environment, draw the opening.
8. Click **Close** on the ribbon bar.
9. Specify the cutting limit to use.
10. Click **Finish**.

Place an Opening on a Sloped Wall

Hole Parallel with Wall Slope

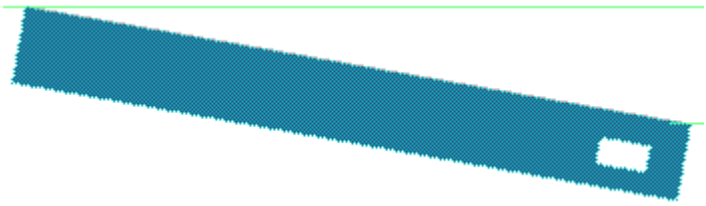
1. Click **Place Opening** on the vertical toolbar.
2. Select the sloped wall.
3. Select the side of the wall in which to place the opening.

   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.
4. Click **Draw**.

   TIP Regardless of the slope, the wall displays as level in the 2-D environment.
5. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, a small rectangular hole was drawn at the wall end.

6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The system places the hole in the sloped wall parallel to the slope.



Hole Parallel with Another Model Object

1. Click **Place Opening** on the vertical toolbar.
2. Select the sloped wall.
3. Select the side of the wall in which to place the opening.

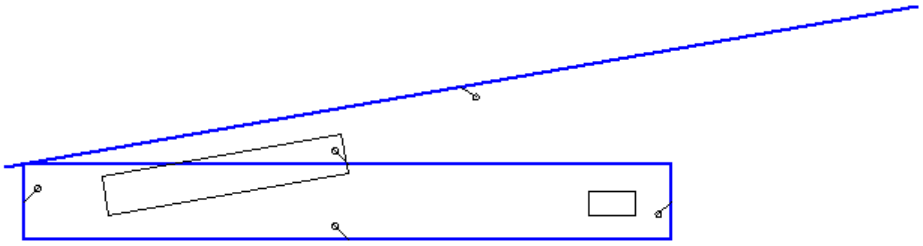
   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.
4. Click **Add References to Sketch 2D**.
5. Select an object in the 3-D model near the wall that you want the hole parallel to. For example, select a grid line to make the hole parallel to a plane. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and

dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model.

6. Click **Draw**.

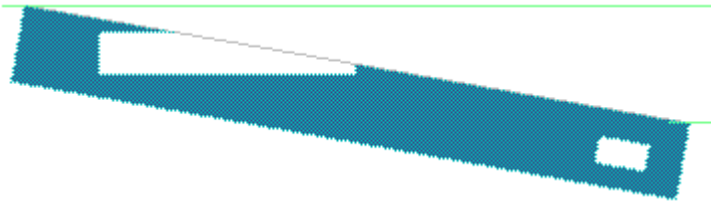
   The wall and the selected object displays in the 2-D environment.

7. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, the opening was placed using the parallel SmartSketch relationship with a grid line that was selected as a reference.



8. Click **Close** on the ribbon bar.
9. Specify the cutting limit to use.
10. Click **Finish**.

    The system places the hole in the sloped wall parallel to the referenced grid line.



See Also

Place Opening Command (on page 135)

Place an Opening on a Leaning Wall

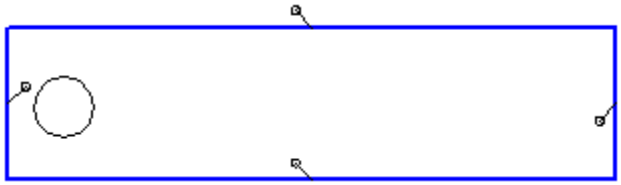
Opening Normal to Side of Wall

1. Click **Place Opening** on the vertical toolbar.
2. Select the wall.
3. Select the side of the wall in which to place the opening.

   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.

4. Click **Draw**.

5. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, a circular hole was drawn at the wall end.



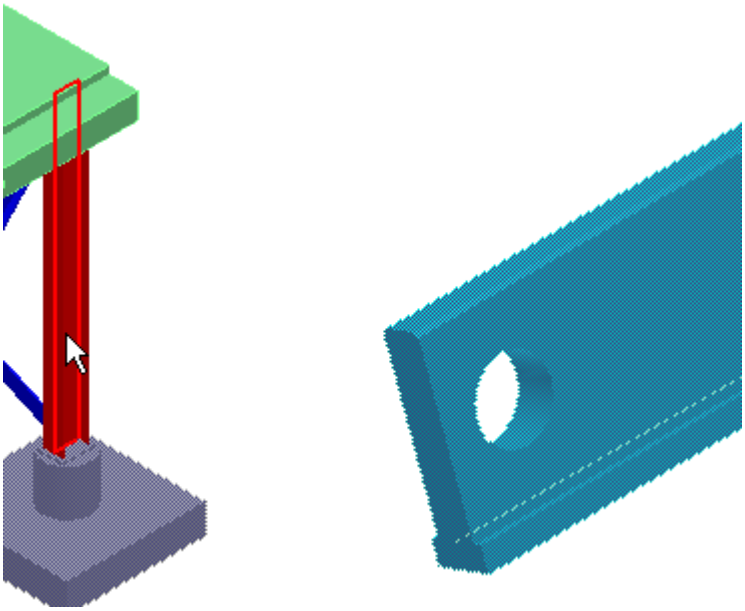
6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The system places the opening normal to the wall face.

Opening Normal to Another Model Object

1. Click **Place Opening** on the vertical toolbar.
2. Select the wall.

3. Select a plane that is normal to the opening that you want. In the example below, the web of a nearby column is selected.

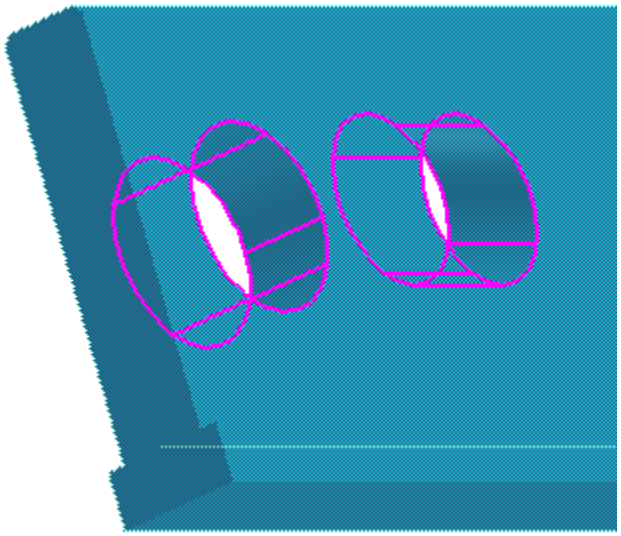


4. Click **Draw**.

   The wall displays in the 2-D environment.

5. Using the available drawing commands in the 2-D environment, draw the opening.
6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The system places the opening normal to the column's web (right opening), not the wall face (left opening).



Change a Recess Opening to a Through Opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the recess opening to change to a through opening.
4. In the **Cutting Limit** option, select **Through-All**.

See Also

Place Opening Command (on page 135)

Change a Through Opening to a Recess Opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the through opening to change to a recess opening.
4. In the **Cutting Limit** option, select **User Defined**.
5. In the **Cutting Depth** option, specify the new depth of the opening.

See Also

Place Opening Command (on page 135)

Delete an Opening

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

See Also

Place Opening Command (on page 135)

SECTION 5

Placing Equipment from the Catalog

You place equipment objects into the model by selecting the equipment from the **Select Equipment** dialog box and positioning the equipment in the model using the available relationships. When you select an equipment object from the Catalog, you can define a default surface so that, when the equipment is placed into the model, the software automatically creates a relationship to any other surface or reference element that you select. For example, you might define the bottom of a pump as the default surface because you want the software to mate that surface to the floor. If the software cannot find a suitable mating surface for the equipment, the equipment object is placed in free space, pending additional design of the structures or other reference elements.

You can also provide an optional offset distance from the surface or reference elements. The default offset distance for any new piece of equipment is zero or the last offset used in the current session. The software maintains the offset relationship between the default surface and the surface or reference element in the workspace in the event of any changes to their position (for example, if a slab is lowered, then the equipment follows the slab, maintaining the offset). You can further define the equipment position and orientation by mating, aligning, or connecting equipment to other objects in the model, or by moving or rotating the equipment.

In addition to placing equipment from the **Select Equipment** dialog box, you can also drag and drop equipment directly from the **Catalog Browser** in another session opened on the same computer. In this case, if you have defined a default surface, the software will place the equipment directly onto a surface or reference element that you select. If you do not define a default surface or reference element, the software places the equipment in free space, with no relationship to other elements in the model.

Cable and Equipment

If you are going to use an object for routing cable, you must not model that object as equipment.

Although it may sometimes be desirable to model a piece of raceway, cable tray, or cable bus as a piece of equipment, you should not do this if there is any possibility of ever needing to route cable through the object. The reason is because cable routes cannot traverse equipment; they can only be terminated by equipment. A cable cannot be properly routed through a piece of equipment. The cable can only interconnect two pieces of equipment.

An example is a pull box. This object should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used.

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An example is a pull box. This object should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used.

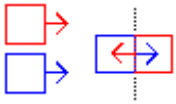
See Also

Rotating Equipment
Place Equipment from the Catalog (on page 155)
Replace Equipment
Set Positioning Relationships for Equipment (on page 156)
Positioning Relationships (on page 148)

Positioning Relationships

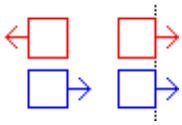
The following positioning relationships between equipment and reference elements are available.

Mate - The mate relationship is applied between a surface of the equipment and another equipment surface, structural surface, elevation plane, or grid plane. The surfaces are constrained to be parallel with the indicated offset distance between them. The normal vector pointing out from the surface of one solid points toward the other solid:



The mate relationship can also be applied between the axis of a cylindrical surface and a planar surface. The offset in this situation is applied in the direction of the surface normal. You can use this relationship to orient the nozzle axis of an equipment parallel to a wall and offset from the wall a given distance.

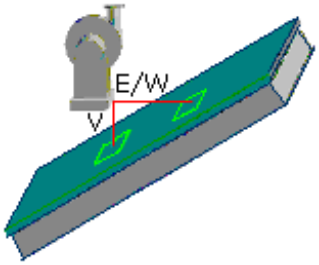
Align - The align relationship makes the axes of two cylindrical surfaces collinear or constrains two planar surfaces to be parallel. When planar surfaces are aligned, the surface normals point in the same direction with the indicated offset distance between them:



Connect - The connect relationship forces a point on an equipment to be coincident with a point on another equipment, structure, piping part, or arbitrary coordinate in space. The software adds the connect relationship to the relationship list only if the two points connected are both nozzles. Otherwise, the two points are made coincident, but no relationship is saved to the database. Offsets are disabled when establishing a connect relationship.

NOTE Press F3 to turn on or off the Surface Locate option. This option makes it easier to create Connect relationships.

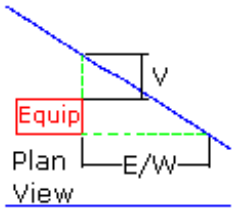
The minimum distance constraint is used to locate a point on an equipment object at a specified distance (horizontal or vertical) from a sloped surface, edge, or point.



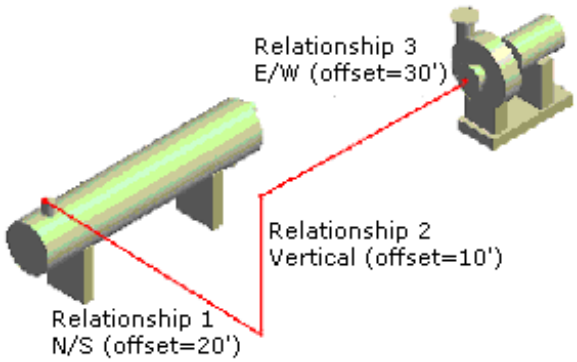
Three constraint options are available: **E/W Distance** (horizontal along the East/West axis), **N/S Distance** (horizontal along the North/South axis), and **Vertical Distance** (vertical along Up/Down axis). These constraints are not available for Designed Equipment.

NOTE The referenced axes are in the active coordinate system.

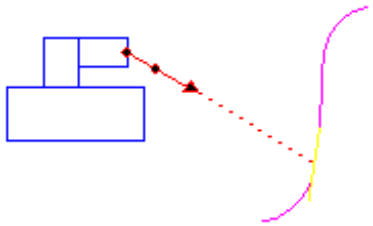
Choose a control point on the equipment that you are placing or editing and a reference surface, edge, or point. Any selectable point on the equipment (SmartSketch points, connect points) may be used in the relationship, and the sloped surface may be at any orientation relative to the equipment.



As illustrated in the following example, the three minimum distance relationships can be used collectively to define separate relationships to fully constrain equipment objects.



Mate to Tangent Plane - Using the mate to tangent plane relationship, you can create a tangent plane at the intersection of a user-defined vector and a selected surface. You establish the relationship by first selecting a reference on an equipment object and then selecting two points in the model to define a vector and identify a surface (in the model). The vector is used to locate an intersection point in the model. A plane tangent to the surface (at the intersection point) is created and used as the mating surface.



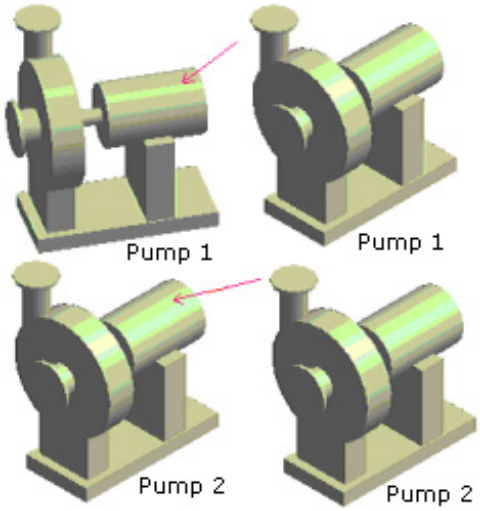
Both the vector and the surface are associative inputs; if the referenced surface is modified, the point and the ensuing tangent plane are re-computed.

Parallel - Parallel relationships can be set between the following objects:

- Edge to planar surface
- Surface to surface
- Edge to edge
- Edge to axis (implied axis of cylindrical objects)
- Axis (implied axis of cylindrical objects) to axis (implied axis of cylindrical objects)

In terms of behavior, the parallel relationship is similar to align. However, while the align relationship rotates and moves an equipment object, the parallel relationship only rotates the object. As such, if you orient a pump by setting a particular nozzle axis parallel to a nozzle axis on a different equipment object, only the orientation of the equipment is affected, not its location. This option is not available for Designed Equipment.

In the following illustration, the motor of the P1 object is selected as an input for the parallel constraint, and the motor of the P2 object is selected as the reference. The result is that the P1 object is rotated (but not moved) so that the two motors are parallel.



See Also

Rotating Equipment
Place Equipment from the Catalog (on page 155)
Replace Equipment
Set Positioning Relationships for Equipment (on page 156)

Place Equipment Command

Specifies any piece of equipment from the Equipment folder of the catalog and places an occurrence of it inside the model. You can modify the offset of the equipment, its relationships to other equipment, and other properties during or after placement. Using positioning relationships, the **Place Equipment** command allows you to mate, connect, or align equipment, and you can use common tools like the **PinPoint** command for precise positioning of the equipment.

If you are going to use an object for routing cable, you must not model that object as equipment. For example, a pull box should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object renders conduit networks useless for cable routing every time it is used.

If you are going to use an object for routing cable, you must not model that object as equipment. For example, a pull box should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object renders conduit networks useless for cable routing every time it is used.

Select Equipment Dialog Box (on page 152)
Place Equipment Ribbon (on page 153)
Equipment Properties Dialog Box (on page 154)

Equipment with Occurrence Properties Modeled Using Solid Edge

You cannot place equipment with occurrence properties modeled using Solid Edge unless Solid Edge is installed on your computer. If you try to place such an object, the following dialog box appears:

As a workaround to the Solid Edge requirement, you can use the Bulkload utility with the flavors option. Designers can create dozens of variations for any equipment part imaginable. Also, by creating several variations of a part, rather than using occurrence properties to create the variations automatically, designers can implement custom Solid Edge equipment without having to install Solid Edge on every designer's computer.

Each variation of a part represents some minor deviation from the catalog part, be it on the basis of size, operational specifications, or material. With Solid Edge, new variations can be created

on the basis of size, while the software allows manipulation of the operating parameters or material of construction in the reference data. For more information on bulkloading with flavors, refer to the *SmartPlant 3D Reference Data Guide*, accessible from the **Help** > **Printable Guides** command in the software.

See Also

Select Equipment Dialog Box (on page 152)
Place Equipment from the Catalog (on page 155)
Replace Equipment
Placing Equipment from the Catalog (on page 147)
Place Equipment Ribbon (on page 153)
Equipment Properties Dialog Box (on page 154)

Select Equipment Dialog Box

Specifies the equipment needed for placement. This dialog box appears automatically when you click the **Place Equipment** command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. After you select a piece of equipment and click **OK**, the **Equipment Properties** dialog box appears so you can define properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so you can finalize configuration and placement.

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

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

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

Properties - Displays the equipment properties as defined in the catalog.

Preview - Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

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

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

See Also

Place Equipment Command (on page 151)
Replace Equipment Command
Placing Equipment from the Catalog (on page 147)
Place Equipment from the Catalog (on page 155)
Replace Equipment
Equipment Properties Dialog Box (on page 154)

Place Equipment Ribbon

Sets options for adding equipment to your model. This ribbon appears automatically after you select the **Place Equipment** command and then select an equipment object, or when you select an existing piece of equipment.

TIP To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Equipment Properties - Edits the occurrence properties and reviews the static properties of an existing piece of equipment. Equipment properties can be set only after an equipment object is placed in the model. Equipment properties can be edited only after the object is placed in the model.

NOTE Any object modeled in Solid Edge that has occurrence properties cannot be placed or modified unless your computer has a copy of Solid Edge installed.

Relationship List - Lists all relationships for the selected equipment and provides an option for creating a new relationship if the equipment is not already fully constrained. An equipment part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships - Displays the available options for types of positioning relationships. Some options may not be available for all equipment types. See *Positioning Relationships* (on page 148) for more information.

Delete Relationship - Removes the selected relationship from the equipment model and the database. Using the **Relationships** list box, select a previously existing relationship for the equipment, and click **Delete Relationship**. You can use this command only when modifying existing equipment.

Equipment Reference - Prompts you for the reference on the equipment to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference - Prompts you for the reference on the equipment object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first equipment part chosen with respect to the second part selected in the definition of the relationship.

Offset - Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed.

Name - Displays the equipment name, as dictated by your predefined name rules, and accepts changes to that name.

System - Specifies the system with which to associate the selected equipment. The default system is the plant itself.

See Also

Place Equipment from the Catalog (on page 155)
Replace Equipment
Placing Equipment from the Catalog (on page 147)
Place Equipment Command (on page 151)
Replace Equipment Command

Equipment Properties Dialog Box

Displays equipment properties for review and editing.

See Also

Occurrence Tab (Equipment Properties Dialog Box)
Definition Tab
Definition Tab (Equipment Component Properties Dialog Box)
Connection Tab
Place Equipment Ribbon (on page 153)

Select System Dialog Box

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

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

Place Doors from the Catalog

1. Click **Place Equipment** on the vertical toolbar.
2. On the **Select Equipment** dialog box, locate the door to place under the Equipment > Architectural > Doors node.
3. In the list view, select the door to place.
4. Click **OK** on the **Select Equipment** dialog box. The **Equipment Properties** dialog box appears so you can define properties for the new door.
5. Define properties as needed, then click **OK** to return to the workspace and place the door. For more information, see *Equipment Properties Dialog Box*.
6. Click in a graphic view to select the mounting surface and approximate position for the door.

   TIP You can press the left and right arrow keys to rotate the door by 90-degree increments at any time during the placement of the door. Press the up arrow key to scroll through the three possible axes of rotation.
7. Using the *Place Equipment ribbon* (on page 153), do any of the following, if necessary:
8. Set an offset for the door relationship by entering the distance in the **Offset** box.

9. Click **Properties** on the ribbon, and enter any necessary property information in the grid provided on the **Equipment Properties** dialog box.
10. If necessary, add or change a positioning relationship by selecting it from the **Relationship** dropdown list on the **Place Equipment** ribbon.
11. Choose a system with which to associate the new door in the **System** box.

Place Windows from the Catalog

1. Click **Place Equipment** on the vertical toolbar.
2. On the **Select Equipment** dialog box, locate the window to place under the Equipment > Architectural > Windows node.
3. In the list view, select the window part to place.
4. Click **OK** on the **Select Equipment** dialog box. The **Equipment Properties** dialog box appears so you can define properties for the new window.
5. Define properties as needed, then click **OK** to return to the workspace and place the window. For more information, see *Equipment Properties Dialog Box*.
6. Click in a graphic view to select the mounting surface and approximate position for the window.

   TIP You can press the left and right arrow keys to rotate the window by 90-degree increments at any time during the placement of the window. Press the up arrow key to scroll through the three possible axes of rotation.
7. Using the *Place Equipment ribbon* (on page 153), do any of the following, if necessary:
8. Set an offset for the window by entering the distance in the **Offset** box.
9. Click **Properties** on the ribbon, and enter any necessary property information in the grid provided on the **Equipment Properties** dialog box.
10. If necessary, add or change a positioning relationship by selecting it from the **Relationship** dropdown list on the **Place Equipment** ribbon.
11. Choose a system with which to associate the new window in the **System** box.

Place Equipment from the Catalog

1. Click **Place Equipment** on the vertical toolbar.
2. On the **Select Equipment** dialog box, locate the necessary equipment type using the tree view. Expand the nodes for the general type of equipment that you need, continuing until a list of available parts appears in the catalog window.
3. In the list view, select the equipment object.
4. Click **OK** on the **Select Equipment** dialog box. The **Equipment Properties** dialog box appears so you can define properties for the new piece of equipment.
5. Define properties as needed, then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 154).
6. Click in a graphic view to select the mounting surface and approximate position for the equipment.

TIP You can press the left and right arrow keys to rotate the equipment by 90- degree increments at any time during the placement of the equipment. Press the up arrow key to scroll through the three possible axes of rotation.

7. Using the **Place Equipment** *ribbon* (on page 153), do any of the following, if necessary:
8. Set an offset for the equipment relationship by entering the distance in the **Offset** box.
9. Click **Properties** on the ribbon, and enter any necessary property information in the grid provided on the **Equipment Properties** dialog box.
10. If necessary, add or change a positioning relationship by selecting it from the **Relationship** dropdown list on the **Place Equipment** ribbon.
11. Choose a system with which to associate the new equipment in the **System** box.

NOTES If you are going to use an object for routing cable, you must not model that object as equipment. For example, a pull box should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used. If you are going to use an object for routing cable, you must not model that object as equipment. For example, a pull box should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used. For information on placing equipment with occurrence properties modeled using Solid Edge, see *Place Equipment Command* (on page 151).

See Also

Select Equipment Dialog Box (on page 152)
Placing Equipment from the Catalog (on page 147)
Place Equipment Command (on page 151)
Replace Equipment Command
Place Equipment Ribbon (on page 153)

Set Positioning Relationships for Equipment

1. Select the equipment for which you need to define a relationship.
2. In the **Relationship List** box on the *Place Equipment ribbon* (on page 153), choose <**New Relationship**>.
3. Select one of the available positioning relationships in the **Positioning Relationships** list box and follow the status bar prompts.

NOTE Depending on the type of positioning relationship defined in the previous step, surfaces or points need to be defined to finish configuring the relationship. For more information, see *Positioning Relationships* (on page 148).

See Also

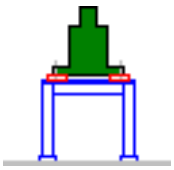
Place Equipment from the Catalog (on page 155)
Replace Equipment
Placing Equipment from the Catalog (on page 147)

SECTION 6

Managing Equipment Foundations

An equipment foundation makes the installation of equipment easier. The foundation is typically connected to the floor or wall, and the equipment is then bolted to the foundation.

In this figure, green indicates the equipment, red indicates the equipment pad, blue indicates the legs and supports of the foundation, and gray indicates the floor.



When you place foundations from the catalog, many properties of the foundations are already defined in the reference data. However, the software calculates some of the properties, including the length of its legs, when you place the foundation in the model. For example, when you select a piece of equipment, the software matches the bolt hole patterns of the foundation and the foundation port and automatically orients and sizes the foundation and its legs accordingly.

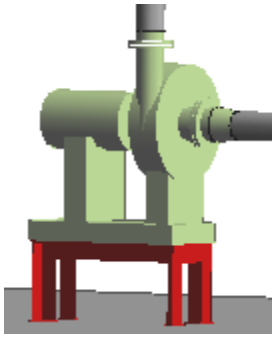
When you place equipment in the model, you position it in space as required to serve the design needs. Each piece of equipment is expected to have at least one (and sometimes more) foundation port. The foundation port determines the placement of the foundation to appropriately support the equipment. The foundation is then designed, using the bolt hole pattern from the foundation port of the equipment, to support the equipment in the functionally required position. You always place the foundation used to support the equipment after you have placed the equipment in the model.

See Also

Delete an Equipment Foundation (on page 164)
Edit an Equipment Foundation (on page 164)
Edit Equipment Foundation Properties (on page 164)
Place an Equipment Foundation (on page 163)

Place Equipment Foundation Command

Places equipment foundations in the model. Equipment foundations are defined in the catalog. You place these foundations relative to an equipment foundation port. The equipment for which you are placing a foundation must already be placed in the model. You can use the Equipment and Furnishings task to place equipment in the model. See *Managing Equipment Foundations* (on page 159) for more information.



See Also

Managing Equipment Foundations (on page 159)
Delete an Equipment Foundation (on page 164)
Edit an Equipment Foundation (on page 164)
Edit Equipment Foundation Properties (on page 164)
Place an Equipment Foundation (on page 163)
Place Equipment Foundation Ribbon (on page 160)

Place Equipment Foundation Ribbon

Specifies the properties for the equipment foundation that you are placing or editing.

Properties - Activates the **Equipment Foundation Properties** dialog box. You can use this dialog box to specify additional properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Equipment Foundation Properties Dialog Box* (on page 161).

Select Equipment - Select the equipment for which to place a foundation. You can select more than one piece of equipment to place a single foundation under all the selected equipment.

Select Support Surface - Select the surface to support the foundation, and thus the equipment.

Finish - Places the foundation.

Cancel - Rejects the selected object.

Accept - Confirms that the selected object is the object that you want to use.

By Rule - Select to allow the software to select the equipment foundation to use based on the default foundation defined for the equipment part. The default foundation for the equipment appears in the **Type** box. The default foundation for the equipment part is defined in the

equipment reference data. The foundation name specified in the equipment reference data must exactly match an equipment foundation part defined in the equipment foundation reference data.

System - Select the system in which to place the foundation.

Type - Select the type of foundation to place. Select **More..**. to select from all available equipment foundations in the catalog.

Name - Type a name for the foundation.

See Also

Delete an Equipment Foundation (on page 164)
Edit an Equipment Foundation (on page 164)
Edit Equipment Foundation Properties (on page 164)
Place an Equipment Foundation (on page 163)
Place Equipment Foundation Command (on page 160)

Select Equipment Foundation Dialog Box

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

Back - Returns you to the previously selected equipment foundation 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 equipment foundation 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.

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

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

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

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

Address - Specifies your exact location within the displayed hierarchy.

Equipment Foundation Properties Dialog Box

Specifies the properties for the equipment foundation that you are editing.

See Also

Definition Tab (Equipment Foundation Properties Dialog Box) (on page 163)
Occurrence Tab (Equipment Foundation Properties Dialog Box) (on page 162)

Occurrence Tab (Equipment Foundation Properties Dialog Box)

The **Occurrence** tab displays in a grid the equipment foundation 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 equipment foundation, and then selected the properties command, only the common properties between the selected equipment foundations display.

When viewing properties for a single equipment foundation, 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.

Category - Select the properties that you want to view for the equipment foundation. Equipment foundation properties are divided into several different categories: Standard, Weight and CG, and Fabrication and Construction. You select the category to define values for by using the **Category** option.

Standard

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

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

System - Select the system to which the equipment foundation that you are placing belongs.

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.

See Also

Equipment Foundation Properties Dialog Box (on page 161)

Definition Tab (Equipment Foundation Properties Dialog Box)

The **Definition** tab displays the equipment foundation 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 equipment foundation, and then selected the properties command, only the common properties between the selected equipment foundations 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.

See Also

Equipment Foundation Properties Dialog Box (on page 161)

Place an Equipment Foundation

1. Click **Place Equipment Foundation** on the vertical toolbar.
2. Select the equipment for which to place a foundation.
3. Click **Accept** ✓.

4. Select the supporting plane for the equipment foundation.
5. Click **Accept** ✓.
6. Verify that the **Type** and **System** options are set to your satisfaction. Clear the **By Rule** option if you want to manually select the equipment foundation that you want.
7. Click **Finish**.

See Also

Place Equipment Foundation Command (on page 160)

Edit an Equipment Foundation

1. Click **Select**.
2. Select **Foundations** in the **Locate Filter**.
3. Select the foundation to edit.
4. Using the ribbon, edit the selected foundation as needed.

See Also

Place Equipment Foundation Command (on page 160)

Edit Equipment Foundation Properties

1. Click **Select** on the vertical toolbar.
2. Select **Equipment Foundations** in the **Locate Filter**.
3. Select the equipment foundation to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed.

See Also

Equipment Foundation Properties Dialog Box (on page 161)

Delete an Equipment Foundation

1. Click **Select**.
2. Select **Equipment Foundations** in the **Locate Filter**.
3. Select the foundation to delete.
4. Click **Delete** ✕.

See Also

Place Equipment Foundation Command (on page 160)

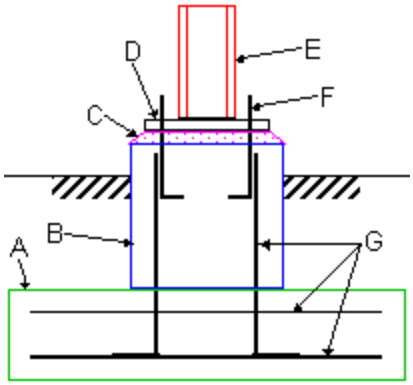
SECTION 7

Implementing Footings

A footing is used to transfer the loads of a column to the ground or sub-structure and to provide mounting stability for the supported column. A typical footing is comprised of:

- A base that supports one or more piers. Some footings for light-weight structures, such as a light pole, do not have a base.
- A pier that rests on the base and supports the grout layer. Some footings do not have a pier in which case the grout is put directly on the base.
- A grout layer that rests on the pier and supports the base plate.

A base plate that rests on the grout layer and is welded to the supported column. You can place base plates using the *Place Assembly Connection Command* (on page 70).



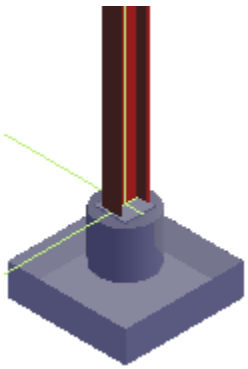
A-Base B-Pier C-Grout D-Base Plate
E-Column F-Anchor Bolts G-Reinforcing Bars

See Also

Place a Combined Footing (on page 172)
Place a Combined Footing by Point (on page 173)
Place a Single Footing (on page 172)
Place a Single Footing by Point (on page 172)
Remove Column from a Combined Footing (on page 174)

Place Footing Command

Places column footings in the model. You can place footings before or after columns are placed in the model. The software creates a frame connection between the footing and the member end to which the footing connects. For more information about frame connections, see *Working with Members* (on page 9).



There are two types of footings that you can place: single and combined. Single footings create a one-to-one relationship between the column and the footing. Combined footings create a many-to-one relationship between a multiple columns and a singe footing.

Helpful things to know about footings:

- Combined footings require a minimum of two columns for placement. If you remove columns from a combined footing such that a single column remains, the software sends the combined footing to the To Do List.
- When placing a combined footing with a merged pier, all columns must have the same bottom elevation. The software will not allow a column with a different bottom elevation to be added to the merged pier.
- The software places key points and control points that can be selected as layout points in drawings on combined footings. Key points are placed at the vertices of slabs and piers. Control points are placed at column locations.

See Also

Implementing Footings (on page 167)
Place a Combined Footing (on page 172)
Place a Combined Footing by Point (on page 173)
Place a Single Footing (on page 172)
Place a Single Footing by Point (on page 172)
Remove Column from a Combined Footing (on page 174)
Place Footing Ribbon (on page 169)

Place Footing Ribbon

Specifies the properties for the footing that you are placing or editing.

Properties - Activates the **Footing Properties** dialog box. You can use this dialog box to specify additional properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Footing Properties Dialog Box* (on page 170).

Select Member - If **Placement** is set to **By Member**, select the column, or columns, in the model for which to place footings. When you are done selecting columns, click **Accept**. If you accidentally select a column that you did not want, click **Cancel**, and then re-select the columns.

If **Placement** is set to **By Point**, identify the point, or points, in the model at which you want a footing. The point identifies the bottom elevation location of a future column.

Select Bottom Plane - Select the bottom plane, or supporting surface, for the footing. The software disables this option if the footing type that you have selected does not require that a supporting surface be defined.

Finish - Places the footing in the model.

Cancel - Rejects the objects that you have selected.

Accept - Confirms the objects that you have selected.

Placement - Select **By Member** to place footings at the bottom of members that you specify. Select **By Point** to place footings at a point you specify in the model. Use the **By Point** option when you want to place footings before the columns in the model.

System - Select the system in which to place the footing. You can create new systems in the Systems and Specifications task.

Type - Select the type of footing to place. Select **More** to select from all available footings in the catalog. If you start to select columns and then change the footing type, the software clears the selected columns because most footing types require different column orientations and the columns that you selected may not work with the new footing type.

Name - Type a name for the footing.

See Also

Place a Combined Footing (on page 172)
Place a Combined Footing by Point (on page 173)
Place a Single Footing (on page 172)
Place a Single Footing by Point (on page 172)
Remove Column from a Combined Footing (on page 174)
Place Footing Command (on page 168)

Select Footing Dialog Box

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

Back - Returns you to the previously selected footing 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 footing 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.

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

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

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

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

Address - Specifies your exact location within the displayed hierarchy.

Footing Properties Dialog Box

Specifies the properties for the footing that you are editing.

See Also

Definition Tab (Footing Properties Dialog Box) (on page 171)
Occurrence Tab (Footing Properties Dialog Box) (on page 170)
Place Footing Command (on page 168)

Occurrence Tab (Footing Properties Dialog Box)

The **Occurrence** tab displays the footing 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 footing, and then selected the properties command, only the common properties between the selected footings display.

When viewing properties for a single footing, 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.

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

Standard

The standard properties that display depend on the reference data for the footing. Because footing reference data is fully customizable, only the common properties are documented.

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

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

System - Displays the name of the parent system. You can create new systems in the Systems and Specifications task.

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.

See Also

Footing Properties Dialog Box (on page 170)

Definition Tab (Footing Properties Dialog Box)

The **Definition** tab displays the footing 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 footing and then selected the properties command, only the common properties between the selected footings 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.

See Also

Footing Properties Dialog Box (on page 170)

Place a Single Footing

1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Member**.
3. Using the **Type** option, select the footing type to place.
4. Select the system for the footing.
5. Select the column where the footing is to be placed.
6. Click **Accept** ✓.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.
8. Click **Finish**.

See Also

Place Footing Command (on page 168)

Place a Single Footing by Point

1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Point**.
3. Using the **Type** option, select the footing type to place.
4. Select the system for the footing.
5. Identify a point in the model where you want the footing. The point identifies the bottom elevation of a future column.
6. Click **Accept** ✓.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.
8. Click **Finish**.

See Also

Place Footing Command (on page 168)

Place a Combined Footing

1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Member**.

3. Using the **Type** option, select the footing type to place.
4. Select the system for the footing.
5. Select the columns where the footing is to be placed. You must select at least two columns.
6. Click **Accept** ✓ when you are done selecting the columns.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.
8. Click **Finish**.

See Also

Implementing Footings (on page 167)
Place Footing Command (on page 168)
Place a Combined Footing by Point (on page 173)
Remove Column from a Combined Footing (on page 174)
Delete a Footing (on page 174)

Place a Combined Footing by Point

1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Point**.
3. Using the **Type** option, select the footing type to place.
4. Select the system for the footing.
5. Identify points in the model where you want the footing. You must identify at least two points.
6. Click **Accept** ✓ when you are done.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.
8. Click **Finish**.

See Also

Implementing Footings (on page 167)
Place Footing Command (on page 168)
Place a Combined Footing (on page 172)
Place a Combined Footing by Point (on page 173)
Remove Column from a Combined Footing (on page 174)
Delete a Footing (on page 174)

Add Column to a Combined Footing

1. Click **Select** on the vertical toolbar.
2. Select **Footings** in the **Locate Filter**.
3. Select the footing to which you want to add the column.
4. Click **Select Member** on the ribbon bar.
5. Select the column, or columns, to add to the footing.
6. Click **Accept** ✓ when you are done selecting columns.
7. Click **Finish**.

See Also

Implementing Footings (on page 167)
Place Footing Command (on page 168)
Place a Combined Footing (on page 172)
Place a Combined Footing by Point (on page 173)
Remove Column from a Combined Footing (on page 174)
Delete a Footing (on page 174)

Remove Column from a Combined Footing

1. Click **Select** on the vertical toolbar.
2. Select **Footings** in the **Locate Filter**.
3. Select the footing from which you want to remove the column.
4. Click **Select Member** on the ribbon bar.
5. Select the column, or columns, to remove from the footing.
6. Click **Accept** when you are done selecting columns.
7. Click **Finish**.

NOTE Combined footings must support a minimum of two columns. Combined footings with only a single column are set to the To Do List.

See Also

Implementing Footings (on page 167)
Place Footing Command (on page 168)
Place a Combined Footing (on page 172)
Place a Combined Footing by Point (on page 173)
Remove Column from a Combined Footing (on page 174)
Delete a Footing (on page 174)

Edit Footing Properties

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

See Also

Footing Properties Dialog Box (on page 170)

Delete a Footing

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

NOTE The software automatically deletes a single footing when you delete the associated column.

See Also

Place Footing Command (on page 168)

SECTION 8

Providing for Traffic

The software provides several commands that you can use to model needed traffic items to provide quick and safe access for personnel. The placement of ladders, stairs, and handrails are very important for efficient operations.

Handrails are usually needed for stairways, walkways, ramps, platforms, and bridges. Default placement parameters for handrails are based on OSHA requirements for height, rail and post spacing, and toe plates.

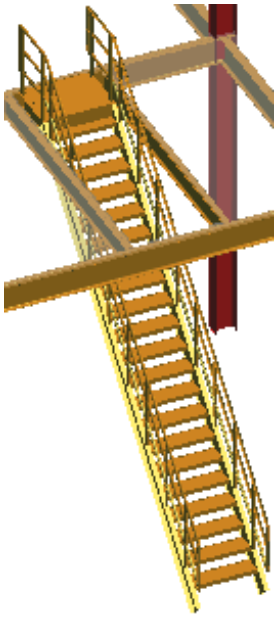
You can mount vertical ladders on a wall, or simply place the vertical ladder between two platforms. The mounting wall can be a flat surface, a piece of equipment, or an equipment tank.

See Also

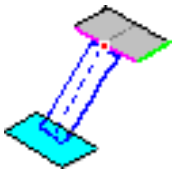
Delete a Ladder (on page 188)
Edit Ladder Placement (on page 187)
Edit Ladder Properties (on page 188)
Place a Ladder (on page 187)
Place a Side Step Ladder (on page 187)
Delete a Stair (on page 182)
Edit Stair Placement (on page 182)
Edit Stair Properties
Place a Stair (on page 182)
Delete a Handrail (on page 196)
Edit Handrail Placement (on page 195)
Edit Handrail Properties (on page 196)
Place a Handrail (on page 195)
Place a Handrail by Member (on page 199)

Place Stair Command

Places a stair in the model. You attach the top of the stair to an edge of any object. You attach the bottom of the stair to a plane. The bottom plane must be lower than the top edge. You must also define the position of the stair along the top edge. There are two methods for defining this position: 1) Select a reference edge or plane that intersects with the top edge and set the horizontal offset distance along the top edge from the intersection point. You can position the stair on either side of the reference. 2) Define a position for the stair graphically.



After you define the position of the stair, the stair appears in the model. The software automatically calculates the number of steps.



In this figure, the purple line is the top edge, the light blue plane is the bottom plane, the green line is the reference edge, and the red dot is the location point.

NOTE Because the stair, the top edge, and the bottom plane are connected, the interference checker service will not find any hard interferences between any part of the stair (stringer, handrail, tread, and so forth) and the top edge object or the bottom plane.

See Also

Providing for Traffic (on page 177)
Delete a Stair (on page 182)
Edit Stair Placement (on page 182)
Edit Stair Properties

Place Stair Ribbon

Specifies the properties for the stair that you are placing or editing.

Stair Properties - Activates the **Stair Properties** dialog box. You can use this dialog box to specify additional stair properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Stair Properties Dialog Box* (on page 180).

Select Top Edge - Select the reference plane or edge that defines the top elevation of the stair. If this edge is deleted after the stair has been placed, the software puts the stair on the To Do List.

Select Bottom Plane - Select the reference plane that defines the bottom elevation of the stair. This plane must be lower than the top edge that you selected. The reference plane can be flat or sloped up to 45 degrees. If this place is deleted after the stair has been placed, the software puts the stair on the To Do List.

Select Reference Edge - Select a plane or edge from which to place the stair along the top edge. This input is optional.

Select Position - Graphically define the location of the stair along the selected top edge.

Finish - Places the stair in the model.

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

Type - Select the type of stair to place.

Name - Specify the name for the stair that you are placing.

Width - Specify the width of the stair.

Angle - Specify the slope of the stair.

Pitch - Specify the distance between the stair steps.

Horizontal Offset - Specify the distance from the selected reference edge.

Vertical Offset - Specify the offset, if needed, from the selected top edge.

Side - Places the stair on the opposite side of the top edge.

See Also

Delete a Stair (on page 182)
Edit Stair Placement (on page 182)
Edit Stair Properties
Place a Stair (on page 182)
Place Stair Command (on page 178)

Select Stair Dialog Box

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

Back - Returns you to the previously selected stair 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 stair 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.

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

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

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

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

Address - Specifies your exact location within the displayed hierarchy.

Stair Properties Dialog Box

Specifies the properties for the stair that you are editing.

See Also

Definition Tab (Stair Properties Dialog Box) (on page 181)
Occurrence Tab (Stair Properties Dialog Box) (on page 180)
Place Stair Ribbon (on page 179)

Occurrence Tab (Stair Properties Dialog Box)

The **Occurrence** tab displays the stair 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 stair, and then selected the properties command, only the common properties between the selected stairs display.

When viewing properties for a single stair, 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.

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

Standard

The standard properties that display depend on the reference data for the stair. Because stair reference data is fully customizable, only the common properties are documented.

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

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

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

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.

See Also

Stair Properties Dialog Box (on page 180)

Definition Tab (Stair Properties Dialog Box)

The **Definition** tab displays the stair 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 stair, and then selected the properties command, only the common properties between the selected stairs 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.

See Also

Stair Properties Dialog Box (on page 180)

Place a Stair

1. Click **Place Stair** on the vertical toolbar.
2. Select the stair to place from the **Select Stair** dialog box. This selection becomes the default selection the next time you place a stair. You can change the default using the **Type** option.
3. Select the top edge for the stair. This edge defines the top elevation of the stair.
4. Select the bottom plane for the stair. This plane defines the bottom elevation of the stair.
5. Select the optional reference edge for the stair.
6. Using the **Horizontal Offset**, **Width**, **Angle**, **Pitch**, **Side**, and **Vertical Offset** options, define the location of the stair along the top edge.
7. Click **Finish**.

See Also

Place Stair Command (on page 178)

Edit Stair Placement

1. Click **Select** on the vertical toolbar.
2. Select **Stairs** in the **Locate Filter**.
3. Select the stairs to edit.
4. Using the ribbon bar options, edit the stair placement as needed.

See Also

Place Stair Command (on page 178)

Delete a Stair

1. Click **Select** on the vertical toolbar.
2. Select **Stairs** in the **Locate Filter**.
3. Select the stairs to delete.
4. Press **Delete**.

See Also

Place Stair Command (on page 178)

Place Ladder Command

Places a ladder in the model. You attach the top of the ladder to an edge of any object. You attach the bottom of the ladder to a plane. The bottom plane must be lower than the top edge. You must also define the position of the ladder along the top edge. There are two methods for defining this position: 1) Select a reference edge or plane that intersects with the top edge and set the horizontal offset distance along the top edge from the intersection point. You can position the ladder on either side of the reference. 2) Define a position for the ladder graphically.

After you define the position of the ladder, the ladder appears in the model. The software automatically calculates the number of rungs.

NOTE Because the ladder, the top edge, and the bottom plane are connected, the interference checker service will not find any hard interferences between any part of the ladder (safety cage, rung, and so forth) and the top edge object or the bottom plane.

See Also

Providing for Traffic (on page 177)
Delete a Ladder (on page 188)
Edit Ladder Placement (on page 187)
Edit Ladder Properties (on page 188)
Place a Ladder (on page 187)
Place a Side Step Ladder (on page 187)
Place Ladder Ribbon (on page 183)

Place Ladder Ribbon

Specifies the properties for the ladder that you are placing or editing.

Ladder Properties - Activates the **Ladder Properties** dialog box. You can use this dialog box to specify additional ladder properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Ladder Properties Dialog Box* (on page 185).

Select Top Edge - Select the reference plane or edge that defines the top elevation of the ladder. If this edge is deleted after the ladder has been placed, the software will put the ladder on the To Do List.

Select Bottom Plane - Select the reference plane that defines the bottom elevation of the ladder. This plane must be lower than the top edge that you selected. The reference plane can be flat or sloped up to 45 degrees. If this plane is deleted after the ladder has been placed, the software will put the ladder on the To Do List.

Select Reference Edge - Select a plane or edge from which to place the ladder along the top edge. This input is optional.

Select Position - Graphically define the location of the ladder along the selected top edge.

Finish - Places the ladder in the model.

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

Type - Select the type of ladder to place.

Name - Specify the name for the ladder that you are placing.

Width - Specify the width of the ladder.

Angle - Specify the slope of the ladder. The default angle is 90 degrees.

Pitch - Specify the distance between the ladder rungs.

Horizontal Offset - Specify the distance from the selected reference edge.

Vertical Offset - Specify the offset, if needed, from the selected top edge.

Side - Places the ladder on the opposite side of the top edge.

See Also

Delete a Ladder (on page 188)
Edit Ladder Placement (on page 187)
Edit Ladder Properties (on page 188)
Place a Ladder (on page 187)
Place a Side Step Ladder (on page 187)
Place Ladder Command (on page 183)

Select Ladder Dialog Box

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

Back - Returns you to the previously selected ladder 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 ladder 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.

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

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

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

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

Address - Specifies your exact location within the displayed hierarchy.

Ladder Properties Dialog Box

Specifies the properties for the ladder that you are editing.

See Also

Definition Tab (Ladder Properties Dialog Box) (on page 186)
Occurrence Tab (Ladder Properties Dialog Box) (on page 185)
Place Ladder Command (on page 183)

Occurrence Tab (Ladder Properties Dialog Box)

The **Occurrence** tab displays the ladder 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 ladder, and then selected the properties command, only the common properties between the selected ladders display.

When viewing properties for a single ladder, 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.

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

Standard

The standard properties that display depends on the reference data for the ladder. Because ladder reference data is fully customizable, only the common properties are documented.

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

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

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

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.

See Also

Ladder Properties Dialog Box (on page 185)

Definition Tab (Ladder Properties Dialog Box)

The **Definition** tab displays the ladder 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 ladder, and then selected the properties command, only the common properties between the selected ladders 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.

See Also

Ladder Properties Dialog Box (on page 185)

Place a Ladder

1. Click **Place Ladder** on the vertical toolbar.
2. Select the ladder to place from the **Select Ladder** dialog. This selection becomes the default selection the next time that you place a ladder. You can change the default using the **Type** option.
3. Select the top edge for the ladder. This edge defines the top elevation of the ladder.
4. Select the bottom plane for the ladder. This plane defines the bottom elevation of the ladder.
5. Select the optional reference edge for the ladder.
6. Using the **Horizontal Offset**, **Width**, **Angle**, **Pitch**, **Side**, and **Vertical Offset** options, define the location of the ladder along the top edge.
7. Click **Finish**.

See Also

Place Ladder Command (on page 183)

Place a Side Step Ladder

1. Click **Place Ladder** on the vertical toolbar.
2. Select the ladder to place from the **Select Ladder** dialog. This selection becomes the default selection the next time that you place a ladder. You can change the default using the **Type** option.
3. Select the top edge for the ladder. This edge defines the top elevation of the ladder.
4. Select the bottom plane for the ladder. This plane defines the bottom elevation of the ladder.
5. Select the optional reference edge for the ladder.
6. Using the **Horizontal Offset**, **Width**, **Angle**, **Pitch**, **Side**, and **Vertical Offset** options, define the location of the ladder along the top edge.
7. Click **Properties** on the ribbon.
8. On the **Occurrence** tab, set the **Hoop Opening** property to either left or right, depending on your needs.
9. Click **OK** on the properties dialog box.
10. Click **Finish**.

See Also

Place Ladder Command (on page 183)

Edit Ladder Placement

1. Click **Select** on the vertical toolbar.
2. Select **Ladders** in the **Locate Filter**.
3. Select the ladder to edit.
4. Using the ribbon bar options, edit the ladder position as needed.

Edit Ladder Properties

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

See Also

Ladder Properties Dialog Box (on page 185)

Delete a Ladder

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

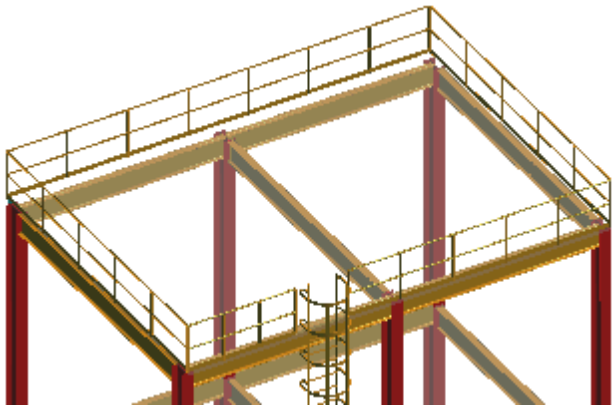
See Also

Place Ladder Command (on page 183)

Place Handrail Command

Places a handrail along a path that you specify in the model. The handrail path can be straight, curved, or a combination of both and can be on the same plane or transverse multiple elevations.

NOTE Because handrails are placed along a defined path, moving other objects in the model may not affect the position of the handrail. Handrails will move when other objects in the model move if you have defined the handrail path using the Point on Surface or Point on Curve constraints. Use the **Tools > Options > SmartSketch** command to make these two constraints available for handrail path placement.



See Also

Providing for Traffic (on page 177)
Delete a Handrail (on page 196)
Edit Handrail Placement (on page 195)
Edit Handrail Properties (on page 196)
Place a Handrail (on page 195)
Place a Handrail by Member (on page 199)
Place Handrail Ribbon (on page 189)

Place Handrail Ribbon

Specifies the properties for the handrail that you are placing or editing.

Handrail Properties - Activates the **Handrail Properties** dialog box. You can use this dialog box to specify additional handrail properties, such as rail section and toe plate section, which you cannot set on the ribbon. For more information, see *Handrail Properties Dialog Box* (on page 190).

Create Handrail Path - Activates the *Create Path Ribbon* (on page 202) with which you define the handrail path.

Toggle Side - Select to place the handrail on the other side of the path.

Finish - Places the handrail in the model.

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

Type - Select the type of handrail to place from the catalog.

Name - Specify the name for the handrail that you are placing. This box is read-only unless you are using the **User Defined** naming rule.

Beginning Treatment - Specify the end treatment for the beginning of the handrail.

End Treatment - Specify the end treatment for the end of the handrail.

See Also

Delete a Handrail (on page 196)
Edit Handrail Placement (on page 195)
Edit Handrail Properties (on page 196)
Place a Handrail (on page 195)
Place a Handrail by Member (on page 199)
Place Handrail Command (on page 189)
Place Handrail by Member Command (on page 197)

Select Handrail Dialog Box

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

Back - Returns you to the previously selected handrail 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 handrail 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.

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

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

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

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

Address - Specifies your exact location within the displayed hierarchy.

Handrail Properties Dialog Box

Specifies the properties for the handrail that you are editing.

Occurrence Tab (Handrail Properties Dialog Box)

The **Occurrence** tab displays in a grid the handrail 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 handrail, and then selected the properties command, only the common properties between the selected handrails display.

When viewing properties for a single handrail, 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.

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

Standard

The standard properties that display depend on the reference data for the handrail. Because handrail reference data is fully customizable, only the common properties are documented.

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

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

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

Begin Extension Length - Enter the distance between the beginning of the handrail and the first handrail post. This distance must be greater than zero.

Begin Treatment - Specify the handrail end treatment for the beginning of the handrail.

Column Clearance - Enter the distance between the column, if one exists, and the handrail. If a column does not exist, this value is ignored and the **End Clearance** value is used instead.

End Clearance - Enter the distance between the member end and the end of the handrail. This value is only used when a column does not exist at that end of the handrail.

End Column Offset - Enter the offset distance between the column at the end of the handrail and the handrail. This offset is in addition to the **Column Clearance** offset. Use this option when you want an offset only at the end of the handrail.

End Extension Length - Enter the distance between the end of the handrail and the last handrail post. This distance must be greater than zero.

End Treatment - Specify the handrail end treatment for the end of the handrail.

Floor Thickness - Enter the thickness of the floor. The thickness is used to calculate the correct height for the toe plate.

Grade - Specify the handrail grade.

Handrail Orientation - Defines the orientation of the post when the handrail is on a slope. Select **Always Vertical** to keep the posts vertical even when the handrail is on a slope. Select **Perpendicular to Slope** to keep the posts perpendicular to the slope.

Height - Enter the height of the handrail.

Horizontal Offset - Enter the horizontal offset from the defined handrail path to the handrail.

Is Assembly - Not used by the software. Set to **False**.

Is System - Not used by the software. Set to **False**.

Material - Specify the handrail material.

Maximum Clearance at Post Turn - Enter the maximum distance from a handrail turn that the software can place a post.

Mid Rail Section Angle - Defines the angle, in degrees or radians, by which the mid rail 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.

Mid Rail Section CP - 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.

Mid Rail Section Name - Enter the cross-section name for the mid rail.

Mid Rail Section Reference Standard - Enter the section library name that contains the mid rail section.

Mid Rail Spacing - Specify the distance between the mid rail spacing.

Minimum Clearance at Post Turn - Enter the minimum distance from a handrail turn that the software can place a post.

Number of Mid Rails - Enter the number of mid-rails for the handrail.

Offset Reference - Select from where on the member you want to measure the handrail horizontal offset. You can select to use the inner port-face, the centerline (axis), or the outer port-face of a bounding member.

Path Horizontal Offset - Specify the offset distance to place the handrail from the defined path.

Path Horizontal Offset Type - Select the offset direction.

Post Section Angle - Defines the angle, in degrees or radians, by which the post 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.

Post Section CP - 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.

Post Section Name - Enter the cross-section name for the top rail.

Post Section Reference Standard - Enter the section library name that contains the post's section.

Segment Maximum Spacing - Enter the maximum distance allowed between the posts.

Sloped Segment Maximum Spacing - Enter the maximum distance allowed between post for that part of a handrail on a slope.

Start Column Offset - Enter the offset distance between the column at the start end of the handrail and the handrail. This offset is in addition to the **Column Clearance** offset. Use this option when you want an offset only at the start of the handrail.

Toe Plate Section Angle - Defines the angle, in degrees or radians, by which the toe plate 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.

Toe Plate Section CP - 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.

Toe Plate Section Name - Enter the cross-section name for the mid rail.

Toe Plate Section Reference Standard - Enter the section library name that contains the mid rail section.

Top of Toe Plate Dimension - Specify the height of the toe plate.

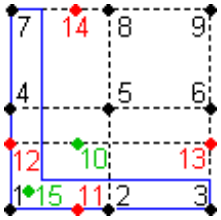
Top of Mid Rail Dimension - Specify the height of the mid rail.

Top Rail Section Name - Enter the cross-section name for the top rail.

Top Rail Section Reference Standard - Enter the section library name that contains the top rail section.

Top Rail Section Angle - Defines the angle, in degrees or radians, by which the top rail 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.

Top Rail Section CP - 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.



Side Mount Connection Type - Specifies the type of side mount. You can select a side mount with a pad, or a side mount with a bracket.

Side Mount Connection Vertical Offset - Specify the offset for the side mount.

With Post At Turn - Specifies if you want a post in the handrail corner. Select **True** to place a post in the corner. Select **False** to not require a post in the corner.

With Toe Plate - Select **True** to attach a toe plate to the handrail. Select **False** to place a handrail without a toe plate.

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.

See Also

Handrail Properties Dialog Box (on page 190)

Definition Tab (Handrail Properties Dialog Box)

The **Definition** tab displays the handrail 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 handrail and then selected the properties command, only the common properties between the selected handrails 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.

See Also

Handrail Properties Dialog Box (on page 190)

Place a Handrail

1. Click **Place Handrail** on the vertical toolbar.
2. Select the handrail to place from the **Select Handrail** dialog. This selection becomes the default selection the next time you place a handrail. You can change the default using the **Type** option.
3. Define the handrail path.

   Define a Path (on page 209)
4. Select a system for the handrail.
5. Click **Finish**.

See Also

Place Handrail Command (on page 189)
Place Handrail by Member Command (on page 197)
Define a Path (on page 209)
Sketching in 3-D (on page 201)

Edit Handrail Placement

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to edit.
4. Using the ribbon option, edit the handrail placement as needed.

Edit Handrail Properties

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

See Also

Handrail Properties Dialog Box (on page 190)

Delete a Handrail

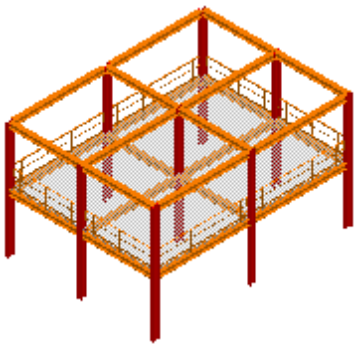
1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to delete.
4. Press **Delete**.

See Also

Place Handrail Command (on page 189)
Place Handrail by Member Command (on page 197)

Place Handrail by Member Command

Places a handrail that is associated with a member part's frame connections. Because the handrail is associated with the member part's frame connections, the handrail length automatically resizes if the member part length changes. You can also define clearances between the handrail-end and the member-end or a column located at the member end.



You can use this command to place a handrail on a single member or multiple members at a time. If you select multiple members, each handrail placed is independent of the others after placement.

If you split a member after placing a handrail on it, the handrail will still be associated to the original placement frame connections, one on either side of the split. If the two new member parts (created by the split) need their own handrails, you will need to delete the original handrail and then place the two new handrails, one on each new member part.

See Also

Providing for Traffic (on page 177)
Delete a Handrail (on page 196)
Edit Handrail Placement (on page 195)
Edit Handrail Properties (on page 196)
Place a Handrail (on page 195)
Place a Handrail by Member (on page 199)
Place Handrail by Member Ribbon (on page 197)

Place Handrail by Member Ribbon

Specifies the properties for the handrail that you are placing or editing.

Handrail Properties - Activates the **Handrail Properties** dialog box. You can use this dialog box to specify additional handrail properties, such as rail section and toe plate section, which you cannot set on the ribbon. For more information, see *Handrail Properties Dialog Box* (on page 190).

Select Members - Select the members on which to place a handrail.

Walking Surface - Select the surface on which people will be walking. The software using this point to know on which side of the member to place the handrail and to calculate the toe-plate and railing height.

Finish - Places the handrail in the model.

Cancel - Clears your selections from the selection set.

Accept - Confirms your selections as the ones to process.

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

Type - Select the type of handrail to place from the catalog.

Name - Specify the name for the handrail that you are placing. This box is read-only unless you are using the **User Defined** naming rule.

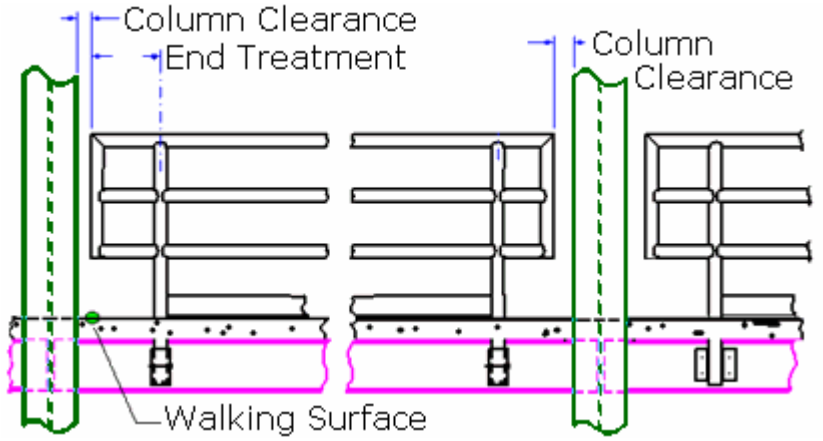
Beginning Treatment - Specify the end treatment for the beginning of the handrail.

End Treatment - Specify the end treatment for the end of the handrail.

Offset Reference - Select how the horizontal offset of the handrail is measured from the member. You can edit the horizontal offset value by editing the **Horizontal Path Offset Distance** value in the handrail Standard category properties.

Column Clearance - Enter the distance between the column, if one exists, and the start of the handrail. If a column does not exist, this value is ignored.

End Clearance - Enter the distance between the member end and the end of the handrail. This value is only used when a column does not exist at that end.



See Also

Delete a Handrail (on page 196)
Edit Handrail Placement (on page 195)
Edit Handrail Properties (on page 196)
Place a Handrail (on page 195)
Place a Handrail by Member (on page 199)
Place Handrail Command (on page 189)
Place Handrail by Member Command (on page 197)

Place a Handrail by Member

1. Click **Place Handrail by Member** on the vertical toolbar.
2. Select the handrail to place from the **Select Handrail** dialog. This selection becomes the default selection the next time you place a handrail. You can change the default using the **Type** option.
3. Select the members for the handrail.
4. Click **Accept**
5. Select a system for the handrail.
6. Enter a distance for the column clearance or the end clearance as needed.
7. Click **Finish**.

See Also

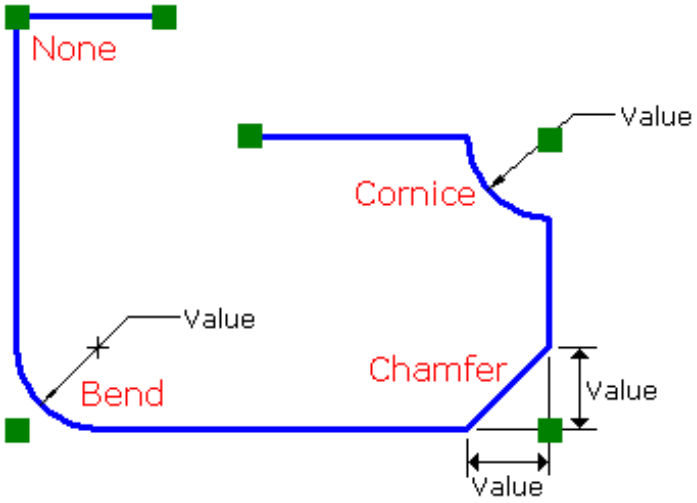
Place Handrail Command (on page 189)
Place Handrail by Member Command (on page 197)

SECTION 9

Sketching in 3-D

To place handrails, curved members, and one of the methods for placing slabs and walls in the model is to sketch a path in the 3-D environment. For handrails, the sketched 3-D path specifies the handrail location. For curved members, the sketched 3-D path specifies the curved member location. For slabs, the sketched 3-D path specifies the slab boundaries or a portion of the slab boundaries. For walls, the sketched 3-D path specifies the wall location. You can use **Tools > PinPoint**, **Tools > Point Along**, and the SmartSketch relationship indicators when defining the path to help with precise placement.

Path segments can be straight lines or three-point arcs with the ability to switch between the two types at any time using the **Path Type** option. In addition, you can define four types of corners for your path using the **Turn Type** option. The following graphic shows examples of the available turn types (paths display in the model as yellow lines (blue below) with yellow squares (green below) at the vertices):



When placing a path for a handrail or curved member, the path can transverse multiple elevations and not required to be a closed shape. However, when placing a path that defines the boundaries of a slab or the path of a wall, the software limits the path to the plane defined for the slab or wall.

See Also

Place Handrail Command (on page 189)
Place Handrail by Member Command (on page 197)
Place Slab Command (on page 103)
Place Curve Member Command (on page 44)

Create Path Ribbon

Sets options for defining a new path. The path can be used to define the location of a handrail, curved member, the boundaries of a slab, or the path of a wall.

Properties - Displays the **Sketch Properties** dialog box, in which you can view properties for the path. For more information, see *Sketch Properties Dialog Box* (on page 208).

Finish - Displays the path in the active view and returns control to the **Place Handrail**, **Place Slab**, **Place Wall**, or **Place Curved Member** command.

Cancel - Cancels the path placement, closes the ribbon, and returns control to the activating command.

Show Cross Section View - Opens a new graphics window that is oriented normal to the sketch plane.

Edit - Allows you to modify and move the existing path. When you initially create a path, this option is only available after you place at least two points in the path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create - Allows you to sketch the path or add segments to an existing path.

First Point - Specifies that you are currently defining the first point of the path segment.

Second Point (Straight Line) - Specifies that you are currently defining the second point of a straight path segment.

Second Point (Arc) - Specifies that you are currently defining the second point of an arc. This option only appears when **Arc by 3 Points** is selected in the **Path Type** list.

Third Point (Arc)- Specifies that you are defining the final point of an arc.

Insert Vertex - Allows you to add a vertex to a straight segment. After the vertex has been inserted, you can move that vertex to edit the segment path. You cannot insert a vertex on arc path segments.

Path Type - Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list. You can select from the following line types:

- **Line** - Defines the line type for the segment to be a straight line.
- **Arc by 3 Points** - Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.
- **No Line** - This option is not available for handrails, walls, or slabs.
- **Arc by End Points** - Defines the line type for the segment to be an arc that you have defined by the arc's end points. To define the arc, you must define the arc's two end points and then a point along the arc.
- **Elliptical Arc** - Defines the line type for he segment to be an elliptical arc.

Plane - Activates options for selecting a working plane for the path. This option is disabled when defining a path for a slab.

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

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

Angle - Specifies the angle for the turn.

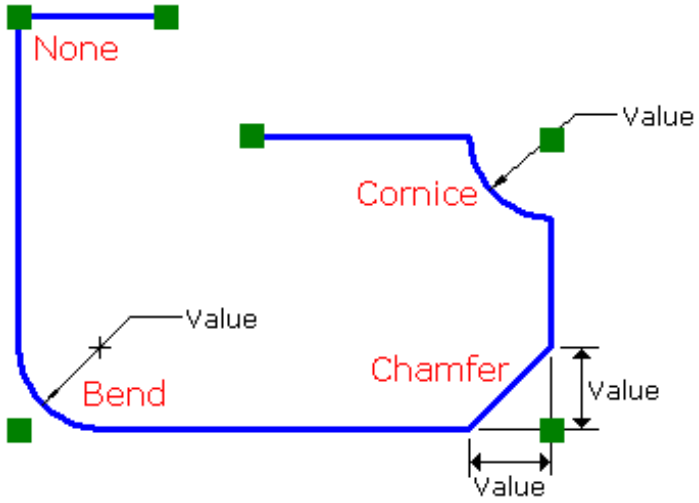
Lock Length - Locks or unlocks the Length box.

Length - Specifies the length of the selected path segment.

Turn Type - Specifies the type of turn associated with the current path segment. The **Turn Type** option is unavailable if you select **Arc by 3 Points** or **No Line** in the **Path Type** list. The following turn types are available:

- **None** - Indicates that no special turn type will be applied to the turn.
- **Bend** - Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Value** box.
- **Cornice** - Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Value** box.
- **Chamfer** - Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for setback value of the chamfer in the **Value** box. The dimensions of setback A and setback B for the chamfer must be the same.
- **Split** - Specifies that you want to split the path you are defining into multiple individual paths between each set of vertices. This option is only available when **Path Type** is set to **Line** and the previous vertex is not the end vertex of an arc. Use this option when, for example, you want individual handrails for each straight segment of the path.

The following graphic includes an example of each of the available turn types:



Value - Defines dimensions for the selected turn type.

Edit Path Straight Segment Ribbon

Sets options for modifying a straight segment of an existing path. This ribbon appears when you select a straight segment in the existing path.

Properties - Displays the **Sketch Properties** dialog box, in which you can view properties for the path.

Finish - Displays the path in the active view and returns control to the **Place Handrail** or **Place Slab** command.

Cancel - Cancels the path placement, closes the ribbon, and returns control the activating command.

Show Cross Section View - Opens a new graphics window that is oriented normal to the sketch plane.

Edit - Allows you to modify and move the existing path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create - Displays the **Create Path** ribbon bar to allow you to add segments to an existing path. You cannot add segments to slab boundary paths.

First Point - Specifies that you are selecting the starting location of the move vector.

Second Point - Specifies that you are selecting the ending location of the move vector.

Insert Vertex - Allows you to add a vertex to the selected straight segment. After you define the location of the vertex along the segment, click **Finish** to insert the vertex. After the vertex has been inserted, you can move that vertex to edit the segment path. You cannot insert a vertex on arc path segments.

Path Type - Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list. You can select from the following line types:

- **Line** - Defines the line type for the segment to be a straight line.
- **Arc by 3 Points** - Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.
- **No Line** - This option is not available for handrails or slabs.
- **Arc by End Points** - Defines the line type for the segment to be an arc that you defined by the arc's end points. To define the arc, you must define the arc's two end points and then a point along the arc.
- **Elliptical Arc** - Defines the line type for the segment to be an elliptical arc

Plane - Activates options for selecting a working plane for the path. This option is disabled if you are editing a path for a slab boundary. Six options are available:

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

Delete Selected Items - Deletes the selected path segments.

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

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

When not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

See Also

Modify a Straight Segment in a Path (on page 211)

Edit Path Arc Ribbon

Sets options for modifying an arc that is part of an existing path. This ribbon appears when you select an arc in the existing path.

Properties - Displays the **Sketch Properties** dialog box, in which you can view properties for the path.

Finish - Displays the path in the active view and returns control to the **Place Handrail** or **Place Slab** command.

Cancel - Cancels the path placement, closes the ribbon, and returns control the activating command.

Show Cross Section View - Opens a new graphics window that is oriented normal to the sketch plane.

Edit - Allows you to modify and move the existing path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create - Displays the **Create Path** ribbon bar to allow you to add segments to an existing path. You cannot add segments to slab boundary paths.

First Point - Specifies that you are selecting the starting location of the move vector.

Second Point - Specifies that you are selecting the ending location of the move vector.

Path Type - Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list. You can select from the following line types:

- **Line** - Defines the line type for the segment to be a straight line.
- **Arc by 3 Points** - Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.
- **No Line** - This option is not available for handrails.

Plane - Activates options for selecting a working plane for the path. This option is disabled if you are editing a path for slab boundaries. Six options are available:

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

Delete Selected Items - Deletes the selected path segments.

See Also

Modify an Arc in a Path (on page 212)

Edit Path Turn Ribbon

Sets options for modifying a turn in an existing path. This ribbon appears when you select a turn in the existing path.

Properties - Displays the **Sketch Properties** dialog box, in which you can view properties for the path.

Finish - Displays the path in the active view and returns control to the **Place Handrail** or **Place Slab** command.

Cancel - Cancels the path placement, closes the ribbon, and returns control the activating command.

Show Cross Section View - Opens a new graphics window that is oriented normal to the sketch plane.

Edit - Allows you to modify and move the existing path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create - Displays the **Create Path** ribbon bar to allow you to add segments to an existing path. You cannot add segments to slab boundary paths.

First Point - Specifies that you are selecting the starting location of the move vector.

Second Point - Specifies that you are selecting the ending location of the move vector.

Plane - Activates options for selecting a working plane for the path. This option is disabled if you are editing a path for a slab boundary. Six options are available:

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

Delete Selected Items - Deletes the selected path segments.

Angle 1 - Displays the first angle used in the turn, if one exits. This option is read-only.

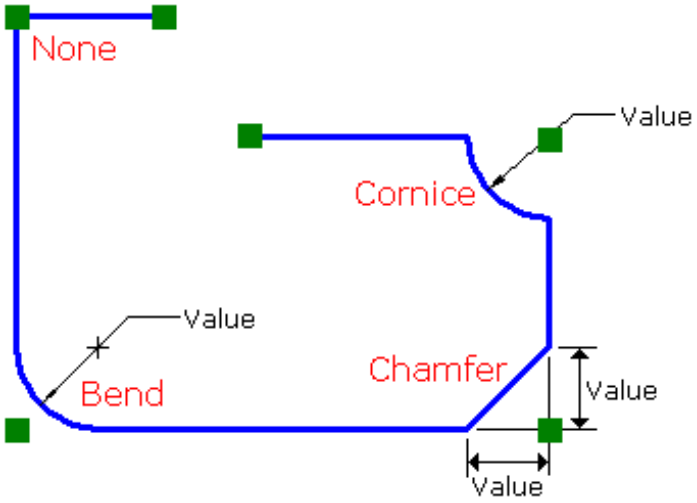
Angle 2 - Displays the second angle used in the turn, if one exits. This option is read-only.

Angle 3 - Displays the third angle used in the turn, if one exits. This option is read-only.

Turn Type - Specifies the type of turn. You can change the turn type by selecting another type in the list. The following turn types are available:

- **None** - Indicates that no special turn type will be applied to the turn.
- **Bend** - Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Value** box.
- **Chamfer** - Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for the chamfer in the **Value** box. The dimensions of setback A and setback B for the chamfer must be the same.
- **Cornice** - Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Value** box.

The following graphic includes an example of each of the available turn types:



Value - Defines dimensions for the selected turn type.

See Also

Modify a Turn in a Path (on page 213)

Sketch Properties Dialog Box

Sets options for paths. You cannot edit the different applicable properties.

See Also

General Tab (Sketch Properties Dialog Box) (on page 208)
Configuration Tab (on page 24)
Relationship Tab (on page 24)
Notes Tab (on page 25)

General Tab (Sketch Properties Dialog Box)

Displays the points that make up a path, their X, Y, and Z coordinates, turn types, and turn type dimensions.

Point No - Displays point number that identifies the selected point.

X - Displays the location of the point on the X-axis.

Y - Displays the location of the point on the Y-axis.

Z - Displays the location of the point on the Z-axis.

Turn Type - Displays the type of turn associated with the point. Turn types include none, bend, chamfer, and cornice.

Value - Specifies dimensions for the selected turn type. For bends and cornices, the value specifies the radius of the bend. For chamfers, the value specifies the dimensions for setback A and setback B of the chamfer.

See Also

Sketch Properties Dialog Box (on page 208)

Define a Path

1. Click to place the first point for the path.

   TIPS

   - You can use Pin Point, Point Along, and the SmartSketch relationship indicators when defining your path.
   - For handrails, you can change the path plane at any time by selecting a new plane in the **Plane** list.
   - To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click 3 points to define the arc.
   - To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.

2. Click the second point for the path.

   TIPS

   - To change turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or offset dimensions for the turn in the **Value** box
   - Use the **Split** turn type to place individual walls or handrails for each straight segment of the path.

3. Click to place other segments of the path as needed.
4. After you place all the points that define the path, click **Finish**.

NOTES

- The software does not require that you close the path for a handrail. You can end the path wherever you want.
- The software does require a closed shape for a slab boundary. If you do not define a closed shape path, then you need to specify additional objects in the model for the missing boundary sides.
- After you place a segment of the path by defining two points, you can click **Edit** on the ribbon to change the segment.

- To manually set the length and angle for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

See Also

Create Path Ribbon (on page 202)

Add Segments to a Path

1. Click **Select**.
2. Select the handrail for which you want to modify the path.
3. On the ribbon, click **Path**.
4. To add one or more segments to the path, click **Create**.
5. Click the point on the existing path where you want to insert the new segment.

   TIPS

   - You can use Pin Point, Point Along, and the SmartSketch relationship indicators when defining your path.
   - You can change the plane for the path at any time by selecting the plane in the **Plane** list.
   - To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click 3 points to define the arc.
   - To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.
6. Click to place other points and add to the path as needed.

   TIP

   - To change turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or dimensions for the turn in the **Feature Value** box.
7. After you place all the points for the new segments, click **Finish**.

NOTES

- You cannot add segments to slab boundary paths.
- You can click **Edit** on the ribbon to change the segment or modify the path further.
- To manually set the length and angle for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

See Also

Create Path Ribbon (on page 202)

Add Vertices to a Path

1. Click **Select**.
2. Select the handrail for which you want to modify the path.
3. On the ribbon, click **Path**.

4. Select the segment to add the vertex to.
5. On the ribbon, click **Insert Vertex**.
6. Identify the location on the path segment for the new vertex.
7. Click **Finish**.

See Also

Create Path Ribbon (on page 202)

Move Segments of a Path

1. Click **Select**.
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which you want to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D**.
5. Select the segments to move.
6. If you want the length of a straight segment to remain constant while you move the segment click **Length Locked**.

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of adjacent segments, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.
7. Click to specify the starting location of the move vector.
8. Click to specify the ending location of the move vector.
9. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** and then **Finish** on the ribbon.

Modify a Straight Segment in a Path

1. Click **Select**.
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D**.

5. Select the straight segment to modify.
   TIP You can select multiple segments by holding down the **Ctrl** key and clicking the segments.
6. To change the line type for the segment, click a new type in the **Path Type** list.
7. To change the plane for the segment, click a new plane in the **Plane** list.
8. To delete the segment, click **Delete**.
9. If you want the length of a straight segment to remain constant when you move the segment click **Length Locked**.

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

10. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** and then **Finish** on the ribbon.
- You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 211).

See Also

Edit Path Straight Segment Ribbon (on page 204)

Modify an Arc in a Path

1. Click **Select**.
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D**.
5. Select the arc to modify.
   TIP You can select multiple segments by holding down the **Ctrl** key and clicking the segments.
6. To change the line type for the segment, click a new type in the **Path Type** list.
7. To change the plane for the segment, click a new plane in the **Plane** list.
8. To delete the segment, click **Delete**.

9. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** ✓ and then **Finish** on the ribbon.
- You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 211).

See Also

Edit Path Arc Ribbon (on page 205)

Modify a Turn in a Path

1. Click **Select** .
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D** 3D.
5. To change the plane for the segment, click a new plane in the **Plane** list.
6. To delete the turn, click **Delete** ✕.
7. To change the turn type, select a new type in the **Turn Type** list.
8. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** ✓ and then **Finish** on the ribbon.
- You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 211).

See Also

Edit Path Turn Ribbon (on page 207)

Glossary

A

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

ACI

American Concrete Institute.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

actuator

A device used to operate a valve using electric, pneumatic, or hydraulic pressure.

airway

A user-defined path for routing cables outside of a cableway. Examples include jumping between trays, drop-offs to equipment, and so forth.

AISC (American Institute of Steel Construction)

An organization responsible for defining American steel construction standards.

AISI

American Iron and Steel Institute

alternative text (AText)

An ISOGEN feature that allows you to change or remove any standard ISOGEN text on an isometric drawing.

ampacity

The current-carrying capacity, expressed in amperes, of a wire or cable under stated thermal conditions.

analysis

The process of modeling a structure to study its physical behavior, such as mechanical (static and dynamic), thermal, and so forth. The most commonly-used analysis is finite element.

analytical member

A mathematical object derived from the logical member used to perform finite element analysis and design.

anchor bolts

Bolts used to connect structural members to concrete footings.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

anisotropic material

A material that has heterogeneous material properties represented by a material property matrix in three directions.

annotations

Dimensions, notes, symbols, or reports placed in a drawing to provide information or comments.

applet

Set of tools or procedures attached to an application, normally held in a dynamic-link library (DLL) and usually exposed through appropriate interfaces from a Component Object Model (COM) server or set thereof. An applet does not support a complete framework user interface but can define user interface items appropriate to its functionality.

application

A program that can execute as a stand-alone entity and that provides the framework and base navigation tools to access all portions of the main program and any attached applets.

application component

An object, like drawings and folders, in the Drawings and Reports Management Console. Components allow you to perform various drawing management tasks, including organizing

drawings in folders and creating different drawing types. The Microsoft Management Console in the operating system also has this concept, called snap-ins.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern the ability of users to alter specific data about a product.

arc element

A structural member of constant radius used to model curved elements.

area

A group of work that is organized primarily by geographic position relative to a named volume or area to which you can assign a relationship.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

arrangement (electrical)

Electrical system arranged in three-dimensional space with accurate dimensional representation for installation. This arrangement is generally shown as a wireway or trunk that contains cable from multiple systems.

arrangement (HVAC)

Components and ductwork of a ventilation system arranged in three-dimensional space with accurate dimensional representation.

arrangement (machinery)

Machinery arranged in three-dimensional space with accurate dimensional representation for installation.Machinery arranged in three-dimensional space with accurate dimensional representation for installation.

arrangement (outfitting)

Outfitting structure arranged in three-dimensional space with accurate dimensional representation.

arrangement (piping)

Components of a piping system arranged in three-dimensional space with accurate dimensional representation.

as-built

Describes the computer model intended to accurately represent the physical plant. An as-built model contains a group of objects that already have been constructed. Objects in the as-built model contain property values (for example, serial numbers) that associate the model objects to physical objects in the plant. Accuracy of this model depends on the incorporation of changes based on changes made in the actual plant.

as-designed

Describes the computer model that depicts the most accurate design of the physical plant. This model does not use property values (that is, serial numbers) but identifies objects by a tag number or actual location. Currently, the authoring tools update the as-designed model, not the as-built model.

as-is

Describes the set of physical objects that actually exist in the plant. The as-is model is not a computer model but a physical entity.

assembly

Unit composed of a collection of parts or other assemblies. Assembly creation consumes the individual part names and provides the unit a unique identification in the fabrication process.

assembly information rule

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

assembly process

Processes and sequences used to manufacture and assemble parts into block units.

associativity

A model architecture where the integrity and consistency of the model is guaranteed by the relationships between model entities.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

auto-nesting

Automatic selection and location of various shapes and sizes of steel plate parts to fit on a standard steel plate, minimizing scrap while optimizing cutting time and minimizing thermal distortion.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x, y, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

B

bar

A steel rod used to reinforce concrete.

basic design

Engineering definition of the model and its systems.

batch extraction

A method of extracting drawings in which many drawings are extracted at a time. You can schedule the extraction process and set its recurrence.

bay

The distance between two trusses.

BCSA (British Constructional Steelwork Association)

An organization responsible for defining British steel construction standards.

beam

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

bearing plate

A steel plate used to distribute a load over a larger area. Usually used at the base of a column.

bent

A vertical framework usually consisting of a truss or beam supported at the ends on columns.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

block decomposition

Breakdown of block units from the major parts to the piece part level. This process includes the complete assembly tree of a structural block that reaches to the lowest level structural part.

block division

Process of establishing boundaries of the major units or blocks to use in building a plant.

boundary condition

A property that defines the restriction on the allowable direction of movement (degree of freedom) at a particular node.

brace

A diagonal member used to stiffen a framework.

branch point

A place where at least three segments of pipe intersect or where a pipe run extends out of a header.

build strategy

Combination of block division, block decomposition, erection network, and assembly process that, when taken as a whole, represents the assembly plan and sequence for building a plant. Note that the plan does not include the schedule but does include the activities needed to develop the schedule.

built-up member

A member built from multiple standard shapes to create a single, usually stronger, member.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

bus

A conductor, or group of conductors, that serve as a common connector for two or more circuits.

busbar

A conducting bar that carries heavy current to supply several electric circuits.

C

cabinet

An enclosure designed either for surface or flush mounting and provided with a frame, mat, or trim in which a swinging door or doors may be hung. See also enclosure.

cable

A conductor with insulation, or a stranded conductor with or without insulation and other coverings (single-conductor cable) or a combination of conductors insulated from one another (multiple-conductor cable). See also optical cable.

cable core binder

A wrapping of tapes or cords around the several conductors of a multiple-conductor cable used to hold them together. Note: Cable core binder is usually supplemented by an outer covering of braid, jacket, or sheath.

cable filler

The material used in multiple-conductor cables to occupy the interstices formed by the assembly of the insulated conductors, thus forming a cable core of the necessary shape (usually circular).

cable hanger

Description of all physical cableway supports.

cable jacket

A protective covering over the insulation, core, or sheath of a cable.

cable schedule

A list of cables for a given unit or project. A cable schedule includes cable names, cable parts, termination information, and electrical service levels.

cable shielding

A nonmagnetic, metallic material applied over the insulation of the conductor or conductors to confine the electric field of the cable to the insulation of the conductor or conductors.

cable terminal

A device which provides insulated egress for the conductors.

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.

cableway load

Weight per unit length supported by a cableway segment.

callout

Label and leader combination that identifies the area on the main drawing view from which a detail view has been derived.

camber

Slight upward curve given to trusses and girders to avoid the effect of sag or self-weight.

can

A reinforcing connection piece placed at member intersections.

cantilever

A beam, girder, or truss that overhangs one or both supports.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog task

User interface that allows you to view the Catalog hierarchy.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

channel

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

channel (electrical)

1) A single path for transmitting electric signals, usually in distinction from other parallel paths. 2) A band of frequencies.

chord

The principal member of a truss. Can be on either the top or bottom of the truss.

circuit

A conductor or system of conductors through which an electric current is intended to flow.

circuit breaker

A device used to open and close a circuit by nonautomatic means, and to open the circuit automatically on a predetermined overload of current, without injury to itself when properly applied within its rating.

circular bar

A structural shape referring to a cylindrical solid.

circular tube

A structural shape referring to a hollow cylindrical member type or pipe.

CISC (Canadian Institute of Steel Construction)

An organization responsible for defining Canadian steel construction standards.

claim

To identify objects as part of a project.

class

Grouping of individual objects that share some very significant, common characteristics.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

clip angle

A small angle-shaped piece of steel used for fastening members together.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

column

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

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

A piece of a pipe feature or pipe run.

component

Physical part that a feature generates.

compound document

Document that contains files with various formats. An example is a document composed of AutoCAD and MicroStation documents.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

condition rule

The condition that must be met before any rule is applied in a drawing view style.

conductor

1) A substance or body that allows a current of electricity to pass continuously along it. 2) A wire or combination or wires not insulated from one another, suitable for carrying an electric current. It may be bare or insulated.

conduit (flexible)

Conduit built up of spiral metal strips that interlock. It is not moisture proof and hence cannot be used where the action of any considerable amount of moisture is objectionable.

conduit (rigid)

Conduit that is fireproof, moisture proof, reliable, and mechanically strong. This type of conduit is generally employed when wires are installed in cableways.

conduit body

A separate portion of a conduit or tubing system that provides access through a removable cover(s) to the interior of the system at a junction of two or more sections of the system or at a terminal point of the system.

conduit fitting

An accessory that serves to complete a conduit system, such as bushings and access fittings.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraint

An analytical boundary condition applied to a node in the model. Constraints can be applied in any valid degree of freedom in the model. Constraints force zero movement at the node and degree of freedom of application.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

construction profile

Principal structural plan for the plant that generally includes a cross section; also an initial structural plan.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cope

To cut out the top or bottom flanges and possibly the web so that one member will frame into another.

cover plate

A plate used in building up flanges, in a built-up member, to give greater strength and area, or for protection.

critical flow

The state of flow for a given discharge at which the specific energy is minimum.

cross section

The shape of a member when viewed along the member line.

cut pipe report

A list that shows the length of each piece of pipe in the pipeline.

cutback

An axial offset that typically represents the distance from a member centerline to its face. Cutbacks are used to account for the difference between how a structure is modeled and constructed in terms of lengths of members and quantities of materials.

cutting plane

A plane that cuts through an object.

D

d/D

The maximum ratio of flow depth over pipe diameter.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes since the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

date or time last revised

Date and time of day when data was last changed in the model.

degree of freedom

An allowable direction of movement, either translation or rotation. There are six possible degrees of freedom (DOFs): translation X, Y, and Z, and rotation RX, RY, and RZ.

de-rating factor

A factor that reduces the current-carrying capacity (ampacity) of a cable due to its method of installation. This factor is affected by the size and type of tray or conduit, whether or not the cables are installed in air or laid in the ground, ambient temperature, proximity of cables with one another, and so forth. This de-rating factor does not apply to cable tray; it is a factor applied to each cable depending on the method of installation.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so the user can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

designed support

A customized support that can contain numerous parts and assemblies. You can change and delete the individual parts of a designed support. For example, you can start with a standard support from the catalog and add to it. This type of support is not associative.

detail

A small part of a structure, drawn separated from the structure to better explain information.

detail envelope

Shape on the main drawing view that determines the 3-D model objects contained in the drawing view that the detail view also includes. Detail envelopes can be circles or polygons in shape.

detail schedule

Lowest level of schedule used to manage and track work progress.

detail sketch

A small drawing inset on an isometric drawing that provides more information about the fabrication or erection of a component represented in the isometric drawing.

detail view

Drawing view that has been extracted from a main drawing view or another detail view, instead of the 3-D model. Detail views are more than simply enlargements of the main drawing view; they

often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information.

diagonals

A member used for stiffening and wind bracing.

diagram (2-D)

Diagram that shows the topology, functional components, wiring connections, and special requirements of a electrical or electronics system. Generally represents the engineering design of the system.

dimension rule

The dimension processing for a drawing view style. This rule controls the style, units, and placement of dimensions in a drawing view.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

division

Intersection in a cableway at which the cross section divides into two or more individual cableways.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

double line

A representation of a structural member that shows the true (or scaled) width of a structural section on a member, including hidden lines, if any. Double line representation is often used in plan and elevation model views.

Draw toolbar

Contains many of the commands you can use to draw elements.

drawing

A graphic file that contains data about a process. Each drawing has a unique drawing number within the unit to which the drawing belongs.

drawing frame

An option category that controls text in the title block area of an isometric drawing.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

drawing view

A two-dimensional representation of three-dimensional geometry from the model. A template drawing view is associated with a drawing volume or multiple volumes in the model. After you generate actual drawings, each drawing view has a one-to-one relationship with a volume. Each view has a view style that applies rules to the object query during drawing generation.

drawing view style

A set of rules that control how three-dimensional objects appear in a two-dimensional drawing view.

drawing volume

A clipping volume that is associated with a specific drawing view in a document.

driftpin

A tapered steel pin used to align bolt or rivet holes when assembling steel.

E

easting

A term that describes an east coordinate location in a coordinate system.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

edge reinforcement axis

A curve that you place along the edge reinforcement. This axis can exist as part of the free edge or an offset of a part of the free edge.

edge reinforcement boundary

An object that limits the edge reinforcement axis.

electric

Electrical system used to distribute electric power and instrumentation signals.

electromagnetic interference (EMI)

Undesirable coupling of electrical, magnetic, or radio wave energy between electrical circuits or cables causing unwanted effects on systems and on electrical or electronic components.

element

Primitive geometric shape such as a line, circle, or arc.

elevation

A viewing orientation normal to a vertical section through a structure, such as a single bay, or along a grid line. Also known as a sectional view of a structure.

elevation (grids)

The height, or value along the Z-axis of the coordinate system, of a point.

enclosure

A surrounding case or housing used to protect the contained conductor or equipment and protect personnel from contacting live parts.

end releases

Physical member properties that define the connection between a member and its nodes. End releases (degrees of freedom) are used to simulate pinned members as well as other special modeling situations.

The member will not contribute stiffness to the node if the degree of freedom (end release) is released. Similarly, the node will not transfer forces or moments to the member through a release degree of freedom. End releases can be defined in any valid degree of freedom for the model.

engineering drawing

Basic layout drawing of a structure used for design and engineering purposes.

equipment

Pieces that a foundation supports. Examples are engines, generators, pumps, fans, consoles, large valves, large strainers, and winches. Usually, you can find these pieces on a machinery arrangement plan. Equipment is most often associated with a system.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

equipment modeler

Facility of the software to create three-dimensional representations of equipment and components for use in defining arrangements.

equipment trim drawing

A type of drawing that contains bill-of-material data, but no graphics, to describe all of the trim lines from a piece of equipment. The purpose of this type of drawing is to produce a bill-of-material that is on a single drawing and is grouped by nozzle connection.

erection clearance

The amount of space left between members to ease assembly.

erection diagrams

Structural drawings prepared specifically for use in the field for erecting a structure. These drawings are used in steel and precast concrete construction to show how the structure fits together, and in what order each piece is to be erected. The drawings include the location of every part of the structure, the number of pieces in members, and the approximate weights of heavy members to assist in construction.

erection network

Sequence of activities needed to erect block units.

exit elevation

The lowest downstream elevation point on the internal diameter of a pipe.

F

fabricate

To cut, punch, and sub-assemble members in the shop.

fabrication plans

Structural detailed drawings of individual structural members, describing exactly how they are to be manufactured.

face-to-face

The overall length of a component from the inlet face to the outlet face.

facilities

Data objects that describe the characteristics of the plant facilities and equipment to manufacture or handle the parts of the plant.

fasteners

Bolts and rivets used to connect structural members.

feature

A logical collections of parts driven by the piping specification. There are four basic features: straight, turn, branch, and inline component.

feeder

1) A set of conductors originating at a main distribution center and supplying one or more secondary distribution centers, one or more branch-circuit distribution centers, or any combination of these two types of equipment. 2) All circuit conductors between the service equipment, or the generator switchboard, and the final branch-circuit overcurrent device.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fill calculations

Computations that find the percent fill of cable tray or conduit. In the software, these calculations comply with the National Electric Code. It is possible to modify the delivered algorithm to satisfy other standards. Fill information can be viewed on fill reports. In addition, maximum fill is stored on the straight features in the model, and you can view this information on ribbons or property pages.

fill efficiency

A factor that provides for future expansion of cable. The user enters this factor. It allows the user to ensure that there is always spare capacity in the cable tray or conduit. For example, a fill efficiency set to 80% denotes that a run of cable tray should be considered full when 80% of the allowable cable is contained in the tray.

fill factor

The percentage of the tray or conduit that you allow to be filled with cables. A percentage of free space is defined to allow for power cable spacing needs, snaking of cables, allowances for bending radii on the tray, and for future expansion.

finite element

A simple geometric shape defined by a specific number of nodes in a specific order. Elements are dependent on all the nodes defining their shape; if any node is deleted, the element is also deleted. Elements are the building blocks of finite element models. Elements can be one of three types: linear or one-dimensional, plate or two-dimensional, or solid or three-dimensional.

fitting

An accessory such as a locknut, bushing, or other parts of a wiring system that is intended primarily to perform a mechanical rather than an electrical function.

flange

The projecting portion of a beam, channel, or column.

flat face

A flange surface on which the gasket sealing area is the entire surface from the inside face diameter to the outside face diameter.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

flexure

A term used for describing bending behavior.

flow rate

The quantity of fluid flowing per unit of time.

flow time

The time it takes for the flow, from the head of the piped system, to reach a downstream point.

fluid flow analysis

Computational fluid dynamics.

focus of rotation

A point or line about which an object or view turns.

footing

An enlargement at the base of a column, or at the bottom of a wall, that distributes the load over a greater portion of ground and thereby prevents settling.

footprint

A graphical cross-section of a structural member. Footprints are used to represent members that intersect a model view plane at or near a perpendicular angle of intersection.

foundation

The bottom-most portion of a wall, or that part of a wall that rests on the footing, upon which the rest of the wall is built.

foundation or masonry plans

Structural drawings that include detail drawings of all foundations and walls that support the structure. The plans show the loads on the foundations, the depth of footings, the proportions of concrete, the quality of masonry, the allowable bearing on the soil, and all data necessary for accurately locating and constructing the foundations.

frame

Single workspace that provides a standard menu bar, status bar, graphical view area, and tree view area.

frame connection

A way of establishing connectivity between member parts to represent physical relationships such as work point offsets, and logical relationships such as end releases. A placement positioning mechanism to represent physical relationships between member systems that establishes and maintains connectivity.

framing plan

A structural drawing plan view, drawn to scale, providing an overhead view of the structural components of a building. Columns, beams, and girders, roof members, floor members, and wall members all require separate framing plans.

free edge

A plate edge that is not bounded by another structural object.

Fu

Ultimate strength

full penetration weld

A type of weld where the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts, like movable articles and fittings, that normally are not associated with a system (for example, a chair).

Fy

Yield strength.

G

gap element

An analytical element used to simulate one-way compression-only behavior. A gap can be closed and in compression. In addition to the compression, it can then conduct (or not conduct) shearing stress depending on whether or not the gap is sliding. When open, the gap does not participate in

the structure stiffness. Typical uses of gap elements in modeling include beams on elastic foundation and the contact of two initially unconnected regions.

gasket

A component used to seal a joint between two other components.

general plans

Structural drawings that include a profile of the ground; location of the structure; elevations of ruling points in the structure; clearances; grades; direction of flow; and all other substructure and superstructure design data.

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

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.

graphic rule

The graphic processing rule for a drawing view style. A graphic rule could be vector hidden line (VHL), for example.

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.

ground

A conducting connection, whether intentional or accidental, by which an electric circuit or equipment is connected to the earth, or to some conducting body of relatively large extent that serves in place of the earth.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

gusset plate

A plate used to connect various members, such as in a truss.

H

handwheel

A wheel-shaped operator intended to be turned by hand to operate the valve stem or operator shaft to which it is attached.

head loss

The loss of pressure due to friction or shape of a structure.

header

The portion of a pipeline topology associated with the primary fluid flow.

HITS report

A diagnostic tool generated by the ISOGEN interface used to analyze the data collected from the 3-D piping model when a problem extracting an isometric occurs.

hook element

An analytical element or boundary condition used to simulate one-way tension-only behavior. A hook element is a backwards gap with the ability to resist tension only when the hook is closed. It resists neither tension nor compression when it is open. Typical uses of hook elements in modeling include structural bracing, cables, tension, bolts, and the separation of two connected regions.

host location

The first Location created for a Site. This Host Location is defined when the Database Wizard creates the Site database.

host server

The database server where the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

hull

The outside surface or envelope of a member.

HVAC

Acronym for heating, ventilation and cooling. This system is the distribution system design for heating or cooling.

HVAC analysis

Analysis routines that address heating and cooling loads required for the compartments and that size or evaluate ventilation ducts and blower requirements.

hydrostatic test

A pressure test in which water is used to detect leaks in a component.

I

I-Section

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

ISOGEN

A software component that generates isometric drawings. Alias, Ltd develops ISOGEN.

isometric

Relating to or being a drafting system characterized by three equal axes at right angles; a view in which the horizontal lines of an object are drawn at an angle to the horizontal and all verticals are projected at an angle from the base.

isometric drawing

A line drawing, always shown in an isometric perspective, that is used for fabricating and erecting piping systems. An isometric drawing usually shows a complete line from one piece of equipment to another and provides all information necessary for fabrication and erection of piping.

isometric drawing style

A set of options that control the drawing output, including format and content. Each style has a unique set of options stored in reference data. You can use the Isometric Style Options Browser to edit the options.

isotropic material

A material that behaves with the same material properties in all direction. An example of an isotropic material is steel.

item

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

J

jacket

A three- or four-faced vertical or sloped tower.

jigs and fixtures

Design of, or data for, devices that position work or hold work in position for joining, transport, or erection.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joist

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

junction box

A box with a blank cover that serves the purpose of joining different runs of cableway or cable and provides space for the connection and branching of enclosed conductors.

K

kips

Kilo pounds.

knee brace

A corner brace used to prevent angular movement.

L

label rule

The label processing rule for a drawing view style. A label rule could locate white space in a drawing view, for example.

leg length analysis

Preferred term is welding length analysis.

lever

A handle type operator.

library

Resource of reference information that a software user can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

lifting

Analysis of the units that must be moved or positioned to determine weight and center of gravity (CG), unit height (including roll over requirements), and the appropriate number and location of lifting pads.

line

Maze of paths identifying connecting elements; synonymous with a network. Fluid comes in at one point and exits through all other connected points. See also pipeline.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load (electrical)

1) A device that receives power or the power or apparent power delivered to such a device. 2) The electric power used by devices connected to an electrical generating system.

load (structure)

A force vector applied to a member.

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.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the SQL Server where the Site databases reside for that Location. One Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

lug (electrical)

A wire connector device to which the electrical conductor is attached by mechanical pressure or solder.

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.

lug and eye piece lifting

Pads used to move structural assemblies.

M

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

manhole (electrical)

More accurately termed splicing chamber or cable vault, a subsurface chamber, large enough for a man to enter, in the route of one or more conduit runs and affording facilities for placing and maintaining in the runs, conductors, cables, and any associated apparatus.

manufacturing (electrical)

Information needed to manufacture the electrical components such as pans, hangers, and the cut lengths of the required cables.

manufacturing (HVAC)

Information needed to manufacture the ventilation piece parts and accomplish assembly.

manufacturing (joiner)

Information needed to manufacture the joiner components.

manufacturing (machinery)

Information needed to manufacture the machinery components or special parts, if any, required to install the equipment.

manufacturing (outfitting)

Information needed to manufacture the outfitting structure piece parts and accomplish assembly.

manufacturing (piping)

Information needed to manufacture the pipe piece parts and accomplish assembly.

marking and cutting

Marking lines used to align any parts for assembly; also marks used to identify parts and cutting of standard plates into needed shapes, usually by N/C equipment.

master run

The cableway run along which a set of cableways is routed.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

material properties

Properties of the material useful in the analysis process.

member name

A user-definable alphanumeric code used to uniquely identify individual members in the model.

member part

A model object derived from the logical model that represents the manufactured physical member parts.

member system

A logical collection of member parts that can be moved as a single entity.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

Microsoft Jet Database Engine

Database management system that retrieves and stores data within a user database or a system database.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A nongraphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

N

network

Maze of paths identifying connecting elements; synonymous with line. Fluid enters at one point and exits through all other connected points.

node

1) One of the set of discrete points in a flow graph. 2) A terminal of any branch of a network or a terminal common to two or more branches of a network. 3) An end point of any branch or a network or graph, or a junction common to two or more branches.

normal vector

In drawing extraction commands, the normal vector is the view orientation of a drawing view set.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

O

object

A type of data other than the native graphic format of the application.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with oa: in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

off-line modification batch

Processing access to the database.

olet

A type of branching fitting that is preshaped to the curvature of the run pipe. Types of olets include sockolets, nipolets, and elbolets.

open interface

Open architecture and utilities allowing the user to develop individual applications that access and return information to the database.

optical cable

A fiber, multiple fibers, or fiber bundle in a structure fabricated to meet optical, mechanical, and environmental specifications.

option

A predefined alternative to the default part in the pipe specification. There can be more than one option.

options file

A set of options that drives the ISOGEN interface.

orientation vector

A vector used to define the local y-x plane of a physical member when combined with the local x-axis defined by the start and end nodes.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

orthotropic material

A material that has two material directions that are orthogonal to one another. An example of an orthotropic material is wood.

oversized spool

A spool with dimensions greater than the maximum allowed dimensions. You can define the maximum dimensions in the reference data.

P

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

panel

The space between adjacent floor supports, or purlins, in a roof.

parallel cable

A group of cables routed together. The child cables all have the same part number, terminating devices, and path.

paralleled cable

A cable that has identical siblings that have the same part number and follow the same path. A paralleled cable must have at least one sibling and be the child of a parallel cable object.

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 number

Unique identifier of a part.

part override

An option used to place a component not defined in the pipe specification.

part ports

An intelligent connection point on a support part. Ports are used to connect parts together in a physically meaningful manner.

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.

parts family

Collection of similar parts. Parts families are gathered into a parts library.

parts library

Identified set of data, and possibly programs, that can generate information about a set of parts.

path

Single path along a continuous curve connecting two ports (nozzles, attach points). It can be made up of numerous runs. For example, pipe path connects nozzle B of equipment 1 to nozzle C of equipment 2.

path (electrical)

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

PCF (Piping Component File)

The intermediary file that the host software generates and delivers to the Alias ISOGEN software with the goal of creating an isometric piping drawing.

PDS (Plant Design System)

A comprehensive, intelligent computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

peak flow

The maximum flow rate of water through a specific size pipe.

penetration spool

A spool that can include parts from one or more pipeline systems. The common factor among all the systems is that each pipeline is welded to a common penetration plate.

physical member

A concept that helps to eliminate modeling errors by providing a straightforward method for defining structure geometry by placing members in a model in much the same way that they would be framed in the field. During analysis pre-processing, physical members are split into finite elements (segments) while still maintaining the collective status of the physical member during model revisions. The software assigns member releases at each end of the true physical member, while preventing release assignments from being made to interior nodes of the finite elements.

pile

A double-battered vertical member placed at each corner of a jacket or at an intermediate position.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

pipe

Piping part that is hollow and approximately cylindrical; may have a constant cross section along its length. Pipe conveys a working media (fluid or gas).

pipe (Structure)

A hollow, cylindrical structural steel shape.

pipe run

Type of interconnection where a single path through a portion of a piping system has a common specification, common property values, and one start and one end point.

pipe specification

A collection of the allowed types of piping commodities and requirements that can be used in the design of a piping system to which the specification applies. These commodities are also known as specification parts. Each individual piping specification includes additional rules that determine the types of parts that must be used in certain design circumstances as well as suggestions for parts that could be used in other circumstances.

pipe stress analysis

Analysis routines that provide stress and deflection data for piping designs. Loading conditions can be both static (thermal or displacement) and dynamic.

pipeline

A set of graphically connected pipe runs including all branches.

piping

Pipe, tubing, and support structure used to move liquids.

piping system

Type of distribution system that allocates and controls the dispersion of a working media (fluid or gas) among functional devices. The piping system may be comprised of other piping systems, parts, devices, pipelines, and pipe runs.

pitch

The ratio of rise to run for roofs. Pitch is also the center distance between bolts or rivets, parallel to the axis of the member.

plant

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Plant points to a Catalog (optionally shared with other Plants) and can reference a PDS project. Access control is managed at the Plant level.

Plant Breakdown Structure (PBS)

The composition of the plant based on the grouping of physical objects by their function in the plant. The plant occupies the top level of the hierarchy, followed by areas and units.

plant configuration

The set of databases and files required for work in a particular Plant. Each Plant must have the following databases: Site database, Site Schema database, Plant database, Report database, Report Schema database, Catalog database, and Catalog Schema database. Each Plant also must have one shared file location for Catalog symbols and output files, such as drawings, specific to the Plant.

plate

A flat, rectangular steel shape.

pneumatic test

A pressure test in which air is used to detect leaks in a component.

port

A connection point to a pipe or a component such as a valve.

post tensioning

A method of pre-stressing concrete, by stressing the steel strands after the concrete has been poured and allowed to harden.

precast concrete

Concrete members that are poured in forms at a plant or factory and allowed to harden. Two types of precast products exist: pre-stressed products and reinforced products.

pre-stressed concrete

Concrete products that are stressed by passing high-strength steel strands through the form and applying stress to the strands either before or after the concrete is poured.

pretensioning

Stressing the steel strands in a pre-stressed member before the concrete is poured into the form.

primary member

A main structural support member.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product model

Collection of the representations and properties of all of the objects and their versions that make up a plant design.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

project

The scope of work approved for capital expenditure; a financed set of work (that is, a job). Normally, a project begins in the design world and then progresses to the physical world when the approval for actual construction is approved. You can modify a property for an object to associate it to a project.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

pull box

A box with a blank cover that is inserted in one or more runs of cableway to facilitate pulling in the conductors, and may also serve the purpose of distributing the conductors.

pulling tension

The longitudinal force exerted on a cable during installation.

purlin

Horizontal structural member extending between trusses, used as beams for supporting the roof.

Q

query select sets

Set of objects that are selected in a query or queries on the database.

R

rafters

Beams or truss members that support the purlins.

raised face

The raised area of a flange face that is the gasket sealing surface between two mating flanges.

reactance

1) The imaginary part of impedance. 2) Opposition to the flow of alternating electric current caused by the inductance and capacitance in a circuit.

rebar

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

rectangular bar

A structural shape referring to a four-sided solid bar.

rectangular tube

A structural shape referring to a four-sided hollow member type.

reel

The quantity of wire wound on a reel.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

reference ports

A location on a beam, pipe, or duct that defines the connection relationship between the parts in the assembly and the beam, pipe, or duct.

reflect

A parameter that affects the placement orientation of member cross sections. At your option, the sections can be placed with their geometry reflected or mirrored about the local section y-axis.

reinforced concrete plan

A framing plan that identifies all beams and slabs by letter and number. The slab, beam, and bend schedules give all details for preparation and placement (number, size, spacing, bending, and location) of the reinforcing steel.

repeatability

A process in which re-extracted drawings only change where modifications have been made to the model. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers.

resistance

That physical property of an element, device, branch, network, or system that is the factor by which the mean-square conduction current must be multiplied to give the corresponding power lost by dissipation as heat or as other permanent radiation or loss of electromagnetic energy from the circuit.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

retaining wall

A structural wall, either gravity or cantilever, used to hold back dirt or other materials.

revision cloud

A set of arcs used to enclose changes that have occurred since the last revision.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

rule-based joint

A feature that enables you to offset the work point of two members that intersect at a node. The rules vary depending on the structural type, member orientation, and construction practices. Ruled joints are typically used when a connection detail calls for the physical member ends to be noncoincident with the centerline of an intersection member.

run

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

S

saddle

The multi-curved shape that appears when a pipe section is connected to another pipe section.

sag ties

Tie rods between purlins in the plane of a roof, used to carry the component of the roof load parallel to the roof. Tie rods are used to support girts.

SAISC (South African Institute of Steel Construction)

An organization responsible for defining South African steel construction standards.

satellite location

Location command.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

scenario

Part of the requirements documentation providing detailed descriptions of a set of functionality placed in the context of a user performing a task. Scenarios establish a set of function points that the software must include.

schedule

A system for indicating the wall thickness of a pipe. The higher the schedule number, the thicker the wall for a certain pipe size.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after users modify or add to the database structure.

scope document

Part of the requirements documentation providing a high level view of the functionality that a user task contains. A use case describes the detailed functionality.

seal weld

A weld that seals or prevents leakage from a joint but does not contribute to the strength of the joint.

secondary member

A member that is not a significant load-bearing member.

section

A structural member whose parameters are defined in a table.

section name

An alphanumeric code used to refer to a particular member in a table; for example W18X35.

section orientation

The position of physical member cress section relative to the physical member local coordinate system.

section table

A library of standard structural shapes containing the necessary parametric properties of each section size, such as depth, width, and inertias.

service

The conductors and equipment for delivering electric energy from the secondary distribution or street main, or other distribution feeder, or from the transformer, to the wiring system of the premises served.

session

Series of commands or functions that you carry out in a document.

shear stiffness

A physical member property that indicates whether to include shear stiffness in the analytical element stiffness matrix of a member.

shear walls

A wall designed to resist lateral loading from winds, underground disturbances, or blasts.

shell structure

External portion of the surface of the plant.

shield

As normally applied to instrumentation cables, refers to metallic sheath (usually copper or aluminum), applied over the insulation of a conductor or conductors for the purpose of reducing electrostatic coupling between the conductors, which may be susceptible to or which may be generating unwanted electrostatic fields.

shop drawings

Drawings prepared for shop personnel to fabricate structural members. Usually includes fabrication details and a bill of materials.

side-wall pressure

The crushing force exerted on a cable during installation.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

skewed member

A structural member that is not at right angles to its connecting members.

slab

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

slave run

The cableway run(s) that follow a master run as it is routed, following the same path offset by a constant distance.

slope

The degree of incline of a roof, expressed as a ratio of the vertical rise to the horizontal run.

span

A group of contiguous physical members that are not intersected by a brace- or column-type physical member.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specific

Unique object that resides in a catalog or library that a user can reference. See also generic specific and specific object.

specific energy

Depth of flow plus velocity head.

specific object

Object with properties that are fully defined but lacking a usage context (occurrence); for example, a part that has a complete definition including make and model number. See also specific.

specifications

Contracted requirements for the plant.

splice (electrical)

The physical connection of two or more conductors to provide electrical continuity.

splice (structure)

To fasten together two physical members, usually columns end-to-end, to form a single continuing length.

split

A feature that represents a break and a joint in the piping, for example, butt-weld, coupling, flange set, union, and so forth.

spool

A prefabricated portion of a piping system that is an assembly of fittings, flanges, and pipe. A spool does not include bolts, gaskets, valves, or instruments.

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.

station

User-defined point with a unique name on an object, such as a trunk, that identifies where other objects can pass through it; also called route numbers.

steel reinforced concrete

An association of concrete and steel where the concrete is there to stand the compression forces and the steel the traction forces. The resulting structure can withstand traction and flexion.

stem

A rod that transmits the motion from the operator to the closure element of the valve.

stiffener

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

stress

Forces acting on structural members due to various types of loads. These forces can be shear, tension, compression, or torsion.

stringer

A longitudinal member used to support loads directly.

structure analysis

Analysis routines that provide stress and deflection data for structural designs. Loading conditions can be both static and dynamic. Finite element analysis is the most common type of structure analysis.

strut

A compression member in a framework.

stud

A bolt, threaded on both ends, used to connect components.

style

Appearance of geometry and annotations on the drawing sheet (for example, color and line weight, font used in a text box, and so forth); collection of formats or properties that you name and store as a group. When you apply a style to a selected object, the software applies all the formats or properties in the style to the object. The style types include: fill, dimension, line, and text.

support assembly

A grouping of individual parts, such as beam clamps, rods, and pipe clamps, that comprise a support.

suspended floor

A concrete floor system built above and off the ground.

switchgear

A general term covering switching and interrupting devices and their combination with associated control, metering, protective, and regulating devices; also assemblies of these devices with associated interconnections, accessories, enclosures and supporting structures, used primarily in connection with the generation, transmission, distribution, and conversion of electric power.

symbol key (SKEY)

A code for a symbol on an isometric drawing. For example, FLSO is the SKEY for a slip on flange.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

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.

T

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

task

Various design environments in the software application; an ActiveX object that you can plug into an application framework that represents a set of commands, toolbars, ribbons, and views necessary to perform a set of functionality. Previously called applet and user environment.

task area

Area of the workspace that displays the list of currently available tasks.

terminal block

An insulating base equipped with terminals for connecting secondary and control wiring.

tilt-up walls

Concrete walls that are poured in forms on the ground and then tilted up into place by cranes or hoists.

toolsets

A set of commands along with related user interface components for users to create, manipulate, review, and delete objects. The interface components include menu items, toolbars, ribbons, and dialog boxes.

transition

A cross-sectional type that results in a tapered length of member and involves a gradual change in section size. See also uniform.

trim

A common term that refers to the working parts of a valve and the associated materials.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

truss

A rigid framed structure consisting of straight members joined to form a pattern of interconnecting triangles for carrying loads.

turn

A feature that represents an angular change in direction of a pipe run; for example, an elbow or an angle valve.

U

uniform

A cross-section type that results in a uniform length of member and involves an abrupt change in section size. See also transition.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

update rule

The rule for a drawing view style that determines when to update the view.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

V

valve

A component used to control the flow of fluid contained in a pipeline.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

viewer

Graphic or non-graphic views of the query results of select sets. Examples include a 3-D graphic view, a tree view showing the system hierarchy, or even a report on the selected objects.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

voltage drop

The difference of voltages at the two terminals of a passive impedance.

W

web

The center section separating the flanges of an I-section, tee, or channel.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

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

wire

A slender rod or filament of drawn metal.

wireways

Sheet-metal troughs with hinged or removable covers for housing and protecting electric wires and cables and in which conductors are laid in place after the wireway has been installed as a complete system.

wizard

Software routine attached to an application that provides guidance and expert help to a user to complete one of the functionalities of the application.

Work Breakdown Structure (WBS)

The composition of the plant based on the construction work to be completed. The plant occupies the top level of the hierarchy, followed by projects, contracts, and documents.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

X

X-section

Cross section; a graphically placed cross section representing the member section size.

Y

yield strength

The stress beyond which a material sustains permanent deformation.

Z

zone

A spatial object that relates a functional requirement with a physical space in the model. A zone can refer to one or more space systems and areas, while an area can be entirely or partially part of several different zones.

Index

D

E

F

G

N

O

P

T

V

W