Intergraph Smart™3D: Working with Structure

Student Workbook

Version 2014

Document # TSP3D-TP-100108A

PART OF HEXAGON

INTERGRAPH®

Copyright

Copyright © 2002 – 2014 Intergraph® Corporation. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2014. All Rights Reserved.

Portions of the user interface copyright 2012-2014 Telerik AD.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
300 Intergraph Way
Huntsville, AL 35813

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

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

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION’S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation’s software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.

b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www.bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, http://www.treas.gov/offices/enforcement/ofac/, and the U.S. Department of State Debarred List, http://www.pmddtc.state.gov/compliance/debar.html.

c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation’s Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, PDS, SmartPlant, FrameWorks, I-Sketch, SmartMarine, IntelliShip, ISOGEN, SmartSketch, SPOOLGEN, SupportManager, SupportModeler, Sapphire, and Intergraph Smart are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2013, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

Classroom Training vs Online Training

We expect your Intergraph Smart™3D classroom experience to be rewarding and exciting. This student workbook will allow you to apply your in-class instruction to practical examples in order to learn Smart3D. A variant of this document is used in our Intergraph Smart™3D Online Training courses. If your employer has chosen to enroll you in one of these programs, you will notice differences between this documentation and the online tutorials. The following information provides a brief synopsis of our online training and how it differs from the classroom.

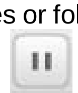
Nature of Online Training

Each Intergraph Smart™3D Online Training session is comprised of a series of informative video clips, a short quiz, and one PDF-based tutorial viewable on any Windows or Android-OS computer/tablet. These videos are presented through a structured, learning management system, or LMS, which logs your time and monitors all videos you have watched. In our live classroom courses, you listen to an instructor's lecture and are given the chance to practice what has been relayed using your text book and the software. The recommended student workflow for our online training course is much the same.

After logging in to the LMS, you watch a video demonstration of specific topics/techniques, complete a practice tutorial and then sit a short quiz about the session to obtain credit in the system.

Please read the following information about the Intergraph Smart™3D Online Training Series.

Videos

Videos are meant to provide visual demonstrations of specific designer tasks performable in the software. They may be viewed for note-taking purposes or followed step-by-step as you explore Smart3D. You can play/pause a video by pressing the  button or Space bar on your keyboard. Pressing the  button or R on your keyboard will rewind the video to the beginning. Videos can only be viewed using the latest Flash-compliant browsers such as Internet Explorer, Firefox, Opera, or Google Chrome.

Quizzes

A Quiz will be given at the end of each session to test what you have just learned. Once you answer the question, you will be given your score.

Tutorials

Tutorials are meant to provide information and step-by-step practice for performing specific designer tasks. Although tutorials and videos are related by subject, video and tutorial content may differ in certain areas. As a guideline, try to follow the steps noted in each tutorial while using previously viewed video content as reference to your learning experience.

If you choose to follow a video task step-by-step and the same task is listed in its tutorial, note that section as followed and proceed to the next task until you finish the tutorial.

© Copyright 2014 Intergraph Corporation

Credit for Viewing

To obtain credit for viewing a series, watch every video session from begin to end, complete its tutorial, and answer the quiz question at the end of the video. Then close the video window using the EXIT button at the top- right-hand side. You should see a check mark appear for that session in the LMS.

© Copyright 2014 Intergraph Corporation

Contents

© Copyright 2014 Intergraph Corporation

© Copyright 2014 Intergraph Corporation

Grids: An Overview

Objective:

By the end of this session, you will be able to:

- Identify the tasks that can be performed using the **Grids** task in SP3D.

Prerequisite Session:

- SP3D Overview

Overview:

The **Grids** task allows you to create and manipulate coordinate systems and reference planes and cylinders defined relative to a coordinate system. For a rectangular grid system, you define vertical grid planes parallel to the x (east)- and y (north)-axes and horizontal elevation planes. For a radial grid system, you define concentric vertical cylinders, radial vertical planes passing through the center of the cylinders, and horizontal elevation planes. A coordinate system has only one reference plane or cylinder at a given position. For example, only one grid plane can be created at x = 10 ft on the coordinate system named CS-0.

Grid lines are displayed at the intersection of the elevation planes and the vertical grid planes. Grid arcs are displayed at the intersection of the elevation planes with the reference cylinders. You can choose the elevation planes on which you want to show grid lines/arcs for a given vertical grid plane/arc.

The reference planes and cylinders associated with the given coordinate system are displayed in the **Workspace Explorer** nested under the coordinate system. You can select any system as the parent of the coordinate system. The reference planes are also displayed for graphic selection on rulers. You can turn these rulers off/on by using the **View** > **Rulers** command. The positions of the reference planes are shown as check marks on the rulers. You can drag these rulers to any position you want in the graphic window.

You can use the coordinate systems, reference planes, and the grid lines/arcs when positioning your design objects in the 3D model. Use any number of different reference grid systems for pipe racks, buildings, or other areas of the plant. If you modify the position of the reference planes/cylinders later, then the associated grid lines/arcs move and all design objects whose positions depend on these reference elements also move.

Figure below shows a rectangular grid.

© Copyright 2014 Intergraph Corporation

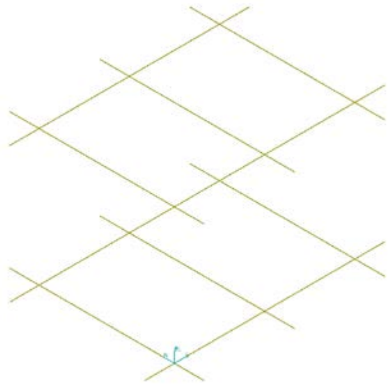
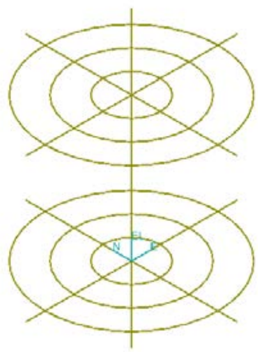
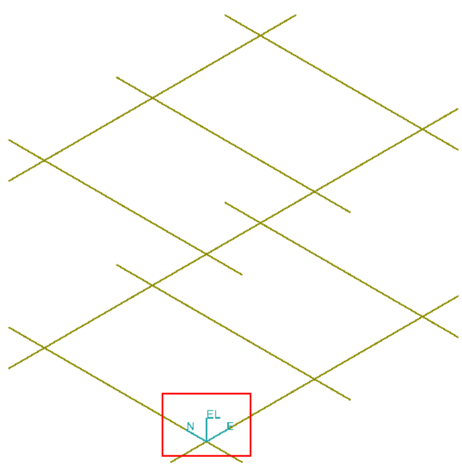


Figure below shows a radial grid.



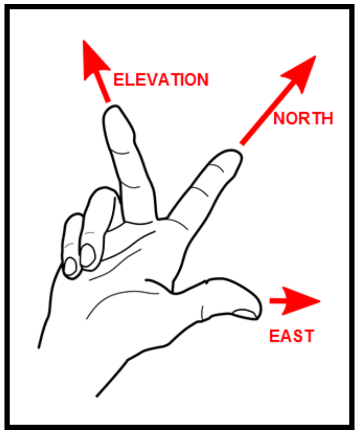
To open the **Grids** task, click the **Tasks** > **Grids** command.

<u>Coordinate Systems:</u>



© Copyright 2014 Intergraph Corporation

Right-handed coordinate systems can be defined for positional reference. In a right-handed coordinate system, the positive direction of the axes are defined, as shown in Figure 4, using the thumb pointing to positive x (east), the index finger pointing to y (north), and the middle finger pointing to z (elevation).



In the modeling environment, you can interact with the coordinate system in rectangular, spherical, or cylindrical coordinates by selecting the desired coordinate input/output option on the **PinPoint** ribbon. It is the same coordinate system, but just a different coordinate mode. There are two types of coordinate systems:

- Global coordinate system
- Design coordinate system (local coordinate system)

Global Coordinate System:

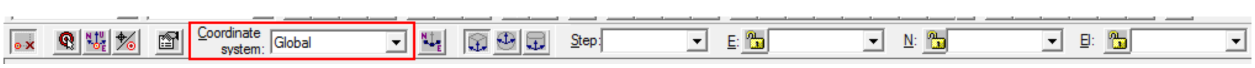
Each model contains one global coordinate system, which you cannot edit or delete. All data you create is stored in the database relative to this coordinate system.

You can view coordinates while you model and output coordinates to drawings relative to the global coordinate system or any local or design coordinate system.

The positive y-axis designates the north axis and the positive z-axis designates the elevation axis.

Note:

- *Model your graphics within 10,000 meters of the global origin to avoid problems with round-off errors in calculations.*
- *To visually reference the global coordinate system, you can activate the global coordinate display by using the **Coordinate system** drop-down list on the **PinPoint** ribbon, as shown in Figure 5.*



Design Coordinate System:

Design coordinate systems are created in relation to the global coordinate system by specifying the origin and orientation of the new coordinate system axes. You can have any number of named coordinate systems to aid in the design of localized constructions such as separate buildings.

The coordinate system used for modeling, the active coordinate system, is selected on the **PinPoint** ribbon.

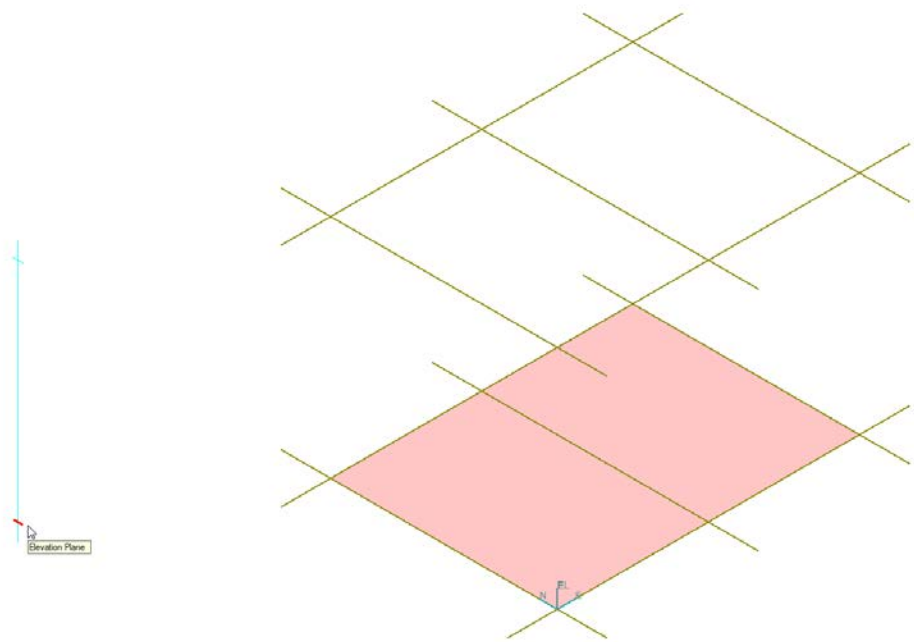
Note:

- *Create a design coordinate system for your plant monument instead of using the global coordinate system. Create the design coordinate system for your plant monument even if the coordinate systems are directly on top of each other. This coordinate system should not have a grid system defined relative to it. The coordinate system will be used only to report coordinates in drawings and reports. If you want to move the entire plant relative to the plant monument, you only have to move the coordinate system representing the plant monument.*

Elevation Planes:

Elevation planes are parallel to the X-Y plane (the plane defined by the x- and y-axes) of the coordinate system. Elevation planes are used in both rectangular and radial grid systems. Grid lines are displayed on the elevation plane at the intersection of the elevation plane with the grid planes, radial cylinders, and radial planes.

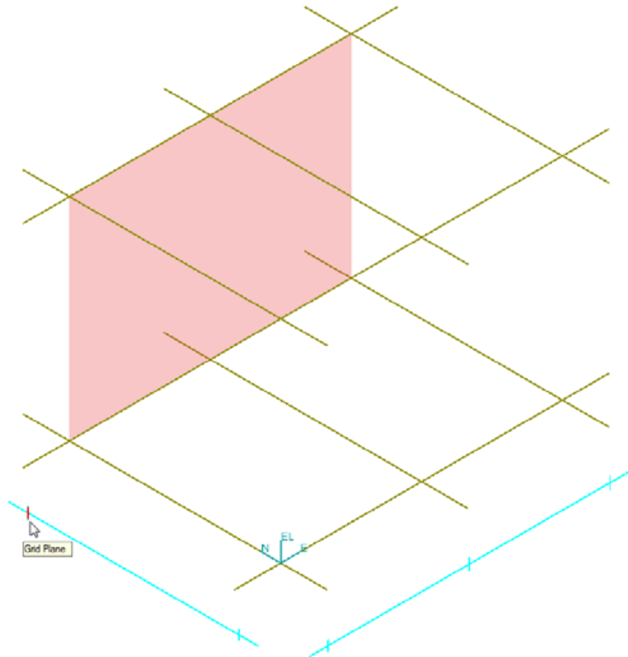
You can create and edit multiple elevation planes with a single command.



Rectangular Grid Planes:

In a rectangular grid system, grid planes are vertical planes parallel to either the x-axis or the y-axis of the coordinate system. A smartstep command on the **Create** and **Edit** ribbons of the grid plane controls the elevation planes that will display a grid line at the intersection with the grid plane.

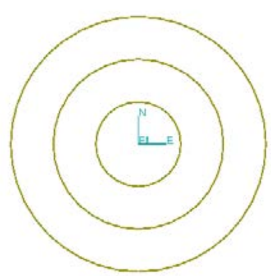
You can create and edit multiple grid planes with a single command.



Radial Grid Cylinders:

For a radial grid system, cylinders are defined with their axis coincident with the z-axis of a coordinate system. Any number of cylinders can be defined with a radius greater than zero. A smartstep command on the ribbon of the grid cylinder controls the elevation planes that will display a grid arc at the intersection with the cylinder. An arc is created for each quadrant of the circle rather than a circle.

You can create and edit multiple radial grid cylinders with a single command—this procedure is similar to the creation of the rectangular grid plane.

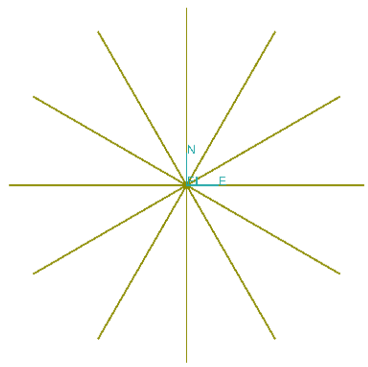


Radial Grid Planes:

© Copyright 2014 Intergraph Corporation

In a radial grid system, radial planes are vertical planes passing through the coordinate system origin and positioned by an angle with respect to the north axis being 0 degrees. Radial planes are infinite in size. Therefore, you cannot place a radial plane that is equal to or greater than 180 degrees.

You can create and edit multiple radial grid planes with a single command—this procedure is similar to the creation of the rectangular grid plane.



Grid Wizard:

You can use the **Grid Wizard** command to quickly create a design coordinate system and an entire set of related elevation planes, grid planes, radial cylinders, and/or radial planes. Once you create the grid system by using the wizard, you can add reference objects to the grid system or edit the existing objects.

For more information related to grids, refer to following topics in the user guide *GridsUsersGuide.pdf*:

- *Understanding Grids: An Overview*
- *Understanding the Grids Workflow: An Overview*
- *Understanding Coordinate Systems: An Overview*
- *Understanding Elevation Planes: An Overview*
- *Understanding Grid Planes: An Overview*

You can access *GridsUsersGuide.pdf* from *GridsPrintGuide.htm*.

Quiz:

1. Which task enables you to design coordinate systems and create elevation planes, grid planes, radial cylinders, and radial planes?
2. Which command do you use to reference the global coordinate system?
3. What is the difference between global and design coordinate systems?
4. What is a radial grid plane?

© Copyright 2014 Intergraph Corporation

LAB-1A: Placing Grids/Coordinate Systems - U02

Objectives

After completing this lab, you will be able to:

- Understanding the grid entities and relationships
- Place grids / coordinate systems, using Grid Wizard command

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Grids: An Overview

Overview:

The **Grid Wizard** command enables you to:

- Create coordinate systems, elevation planes, rectangular grid planes, radial grid cylinders, and radial grid planes in a model.
- Specify when grid lines and arcs should be displayed for each.

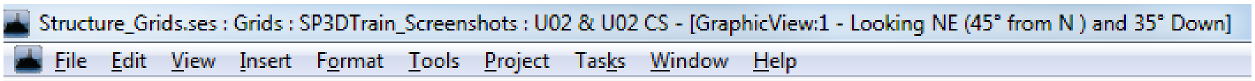
Before going through this and the remaining SP3D Structure sessions, define your workspace to include all objects in the SP3Dtrain model database:

- Start **SmartPlant 3D**. Select the **EnglishUnits** template and click **OK**.
- Click the **File** menu and select the **Define Workspace…** command.
- In the **Filter** drop-down list of the **Define Workspace** dialog box, select the **More…**
- option.
- In the **Select Filter** dialog box, select **All** under **Plant Filters** and click **OK**.
- Select **View** > **Fit**.
- Now, you are going to delete the existing modeled objects from the workspace before starting the session.
- Use the filter mechanism to select the existing modeled objects.
- Select the **Tools** > **Select by Filter** command to open the **Select Filter** dialog box.
- Select and expand the **For Instructors Only** folder.
- Select the **Structural Classroom Session - Select and Delete** filter and click **OK**.
- Click the **Delete** command to delete the selected objects.

1. Open or create a session file and define an “U02 & U02 CS” filter for your workspace.

   Note: Session file stores settings from the last time you were in SmartPlant 3D. The name of the current session file appears in the title bar of the application, along with the name of the task, model and filter. One of the settings saved in the session file is the workspace. See the previous common labs on how to define a workspace.

© Copyright 2014 Intergraph Corporation



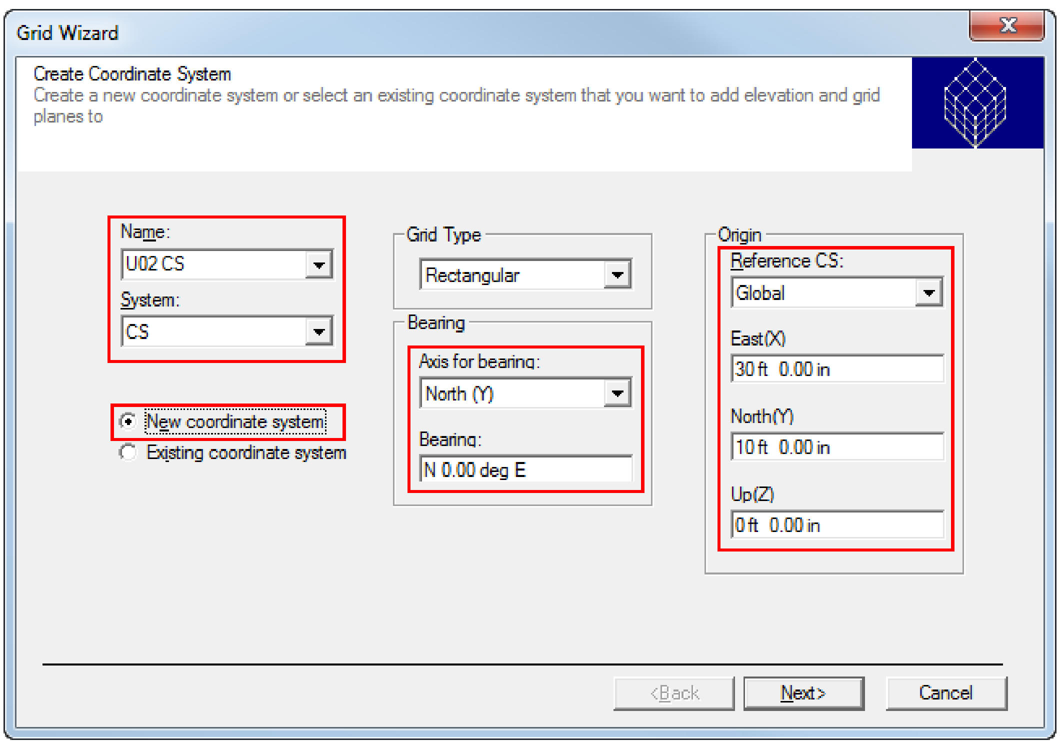
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*

   Note: Objects that you place directly in the model are assigned to the active permission group which is located on the main toolbar. Therefore, you are responsible of making sure the object is assigned to the appropriate Permission Group.

3. Select the Grid Wizard command from the vertical toolbar.

4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:	U02 CS
Axis for bearing:	North (Y)
Bearing:	N 0.00 deg E
Origin:	
Reference CS:	Global
East (X):	30 ft
North (Y):	10 ft
Up (Z):	0 ft



Next, click "Next" button.

After defining the coordinate system, the user defines the elevation planes, grid X-plane, and grid Y-plane. The settings that you specify in these planes are as follows:

- **Reference CS** - It defines the coordinate system with reference to which all planes are placed.
- **Start plane** - It defines the location of the first plane.

- **Copies** - It specifies the number of planes to be created from the start plane.
- **Spacing** - It specifies the distance between the copied planes.
- **End plane** - It specifies the last location of the created plane.
- **Name rule** - It defines the naming convention of the planes. The common naming conventions are as follows:
    - **Index name rule**: This option provides the root labels (GPX for x-axis and GPY for y-axis) for primary planes and then appends onto this root, the sequential order of the planes. For example, 1 for the first plane, 2 for the second plane, and subsequently for next planes. Secondary planes are suffixed with an additional decimal indication of the sequential order of the planes between two primary planes, and so forth for the tertiary level.

Primary	*Secondary*	*Tertiary*
GPX1		
GPX2		
	GPX2.1	
		GPX2.1.1
		GPX2.1.2
	GPX2.2	
GPX3		

    - **Position name rule**: This option provides the base labels (E for the Easting axis, N for the Northing axis) for the primary planes and then appends onto this base, the physical location of the plane on the parent coordinate system. The position is defined in meters.

Primary	*Secondary*	*Tertiary*
E0.00 m		
E8.00 m		
	E10.25 m	
		E12.78 m
	E13.25 m	
E16.50 m		

    - **Imperial position name rule**: This option provides the base labels (E for the Easting axis, N for the Northing axis) for the primary planes and then appends onto this base, the physical location of the plane on the parent coordinate system. The position is defined in feet.

Primary	*Secondary*	*Tertiary*
E0.00 ft		
E8.00 ft		
	E10.25 ft	
		E12.78 ft
	E13.25 ft	
E16.50 ft		

    - **Index and percent name rule**: This option provides the root labels (GPX for x-axis and GPY for y-axis) for the primary planes. Sub-nesting levels are a percentage (in decimal form) of the bounding and higher nesting levels.

© Copyright 2014 Intergraph Corporation

Example:
Grids are usually prefixed with a letter and suffixed with a decimal distance away (A, B, C, C.6, D, and E). The number to the right of the decimal place, such as C.6, for a secondary plane is the relative position between the previous and the next primary plane. In other words, it is 60% away from the previous primary elevation plane.

Primary	*Secondary*	*Tertiary*
GPX1		
GPX2		
	GPX2.5	
		GPX2.5.3
		GPX2.5.8
	GPX2.6	
GPX3		

- **Alphanumeric and percent name rule**: This option provides the root labels (A, B, C… for the x-axis and 1, 2, 3…for the y-axis) for the primary planes. Sub-nesting levels are a percentage (in decimal form) of the bounding and higher nesting levels.

  Example:
  Grids are usually prefixed with a letter and suffixed with a decimal distance away (A, B, C, C.6, D, and E). The number to the right of the decimal place, such as C.6, for a secondary plane is the relative position between the previous and the next primary plane. In other words, it is 60% away from the previous primary elevation plane.

Primary	*Secondary*	*Tertiary*
A		
B		
	B.5	
		B.5.3
		B.5.8
	B.6	
C		

- **User defined name rule**: With this option, the user defines the name by key in. If the name rule is changed to user defined after a name was automatically generated by one of the above-mentioned name rules, the existing name is kept by default but does not automatically change on editing the object properties.

- **Nesting level** - It specifies the levels of divisions to be placed in a grid. There are three nesting levels: primary, secondary, and tertiary. Tertiary planes can be in between two secondary planes. Secondary planes can be in between two primary planes. In a large model, it enables you to organize planes to sublevels and name them to sublevels automatically using the given name rules, such as 1, 1.1, and 1.2. Primary is the default nesting level.
- **Type** - It specifies the type of elevation planes to be placed. There are different types of elevation planes, such as bottom of baseplate, bottom of concrete, top of steel, and grade elevation. A user has to define the type of elevation to be created. This information is used for reporting purposes.

Note: *The value in the **End plane** box is always grayed out. This value is determined by value of the other settings. If the number of copies is set to 0, the end plane takes the value of the start plane. Otherwise, the value of the end plane is determined by the start plane, the number of copies, and the spacing between each plane.*

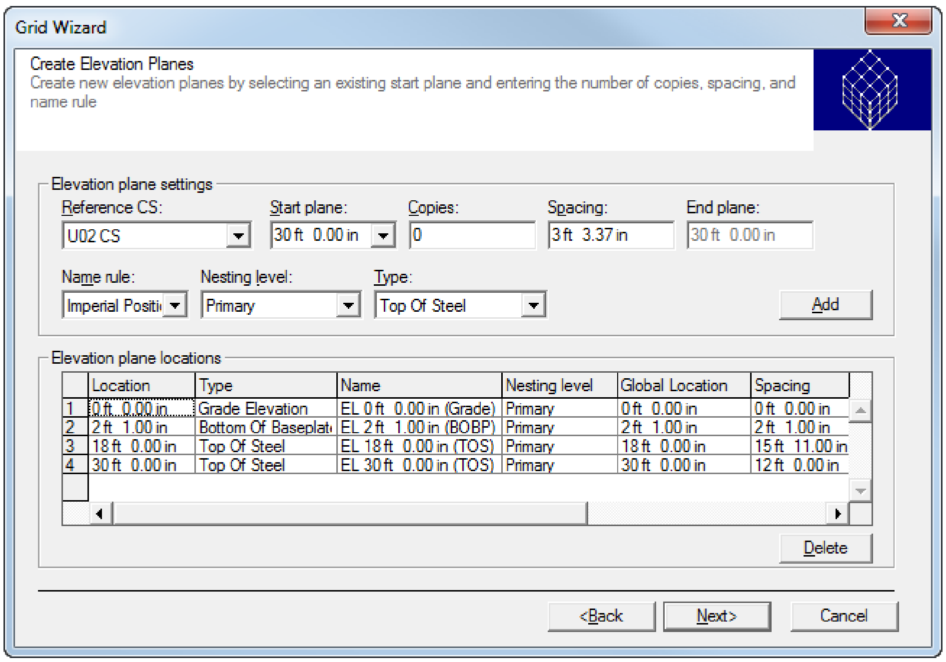
© Copyright 2014 Intergraph Corporation

5. **Elevation Planes**: Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

   Set the Nesting level to Primary for All Elevation Planes.

Step	Start Plane	Copies	Spacing	Name Rule	End Plane	Type
1	0 ft 0 in	0	Ignore	Imperial Position	0 ft 0 in	Grade Elevation
2	2 ft 1 in	0	Ignore	Imperial Position	2 ft 1 in	Bottom of Baseplate
3	18 ft	0	Ignore	Imperial Position	18 ft	Top of Steel
4	30 ft	0	Ignore	Imperial Position	30 ft	Top of Steel

   Select the “Add” button.



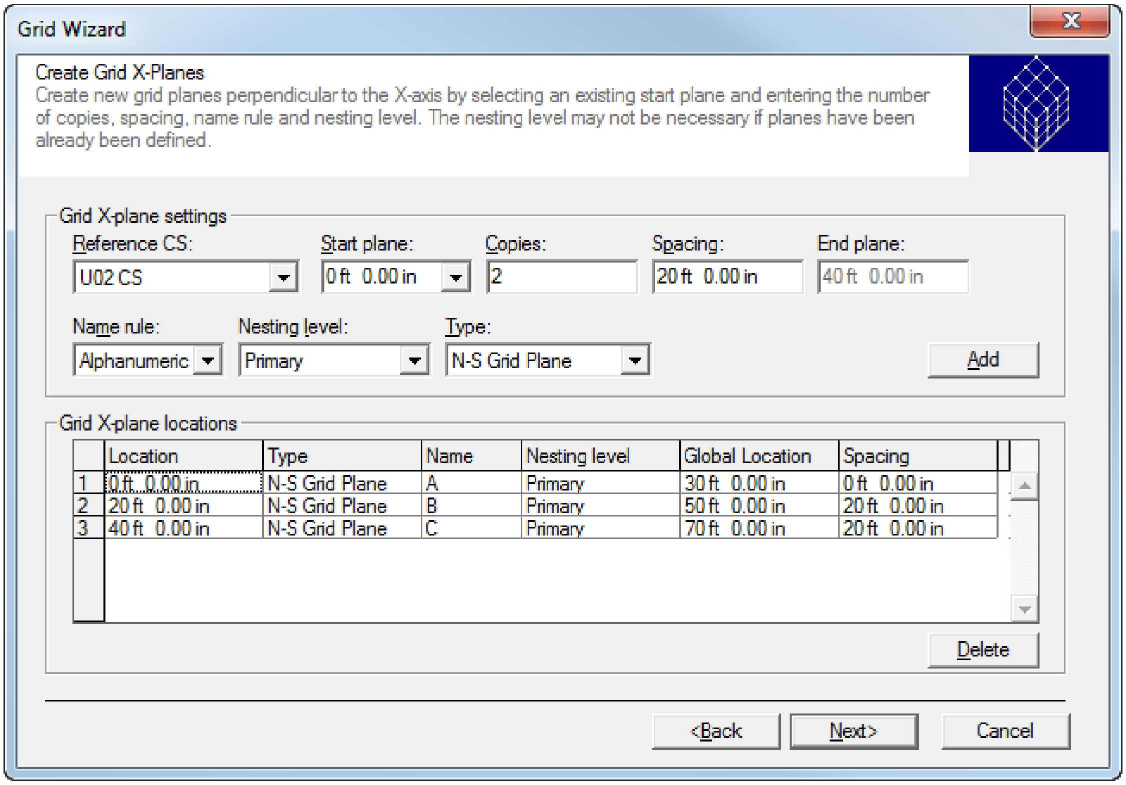
   Next, click “Next” button.

6. **Grid X-Planes:** Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:
   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U02 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	Nesting Level	End Plane	Type
1	0 ft 0 in	2	20 ft 0 in	Primary	40 ft 0 in	N-S Grid Plane

   Select the “Add” button.

© Copyright 2014 Intergraph Corporation



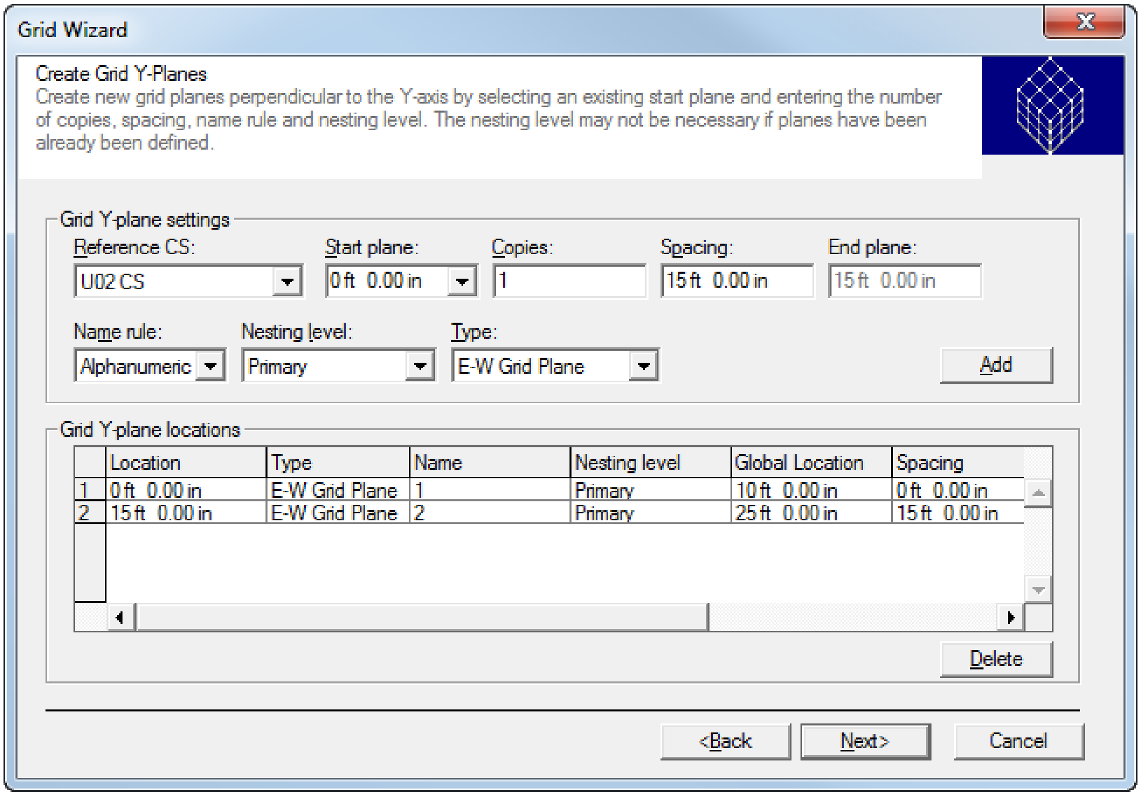
Next, click “Next” button.

7. **Grid-Y Planes**: Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U02 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	15 ft 0 in	15 ft 0 in	Primary	E-W Grid Plane

Select the “Add” button.

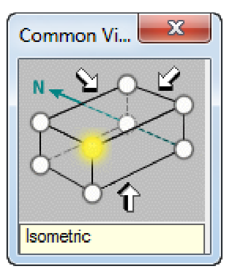


Next, click "Next" button.

8. Hit "Finish" button on Associated Elevation Plane form. The wizard will create the grids lines at all elevations.

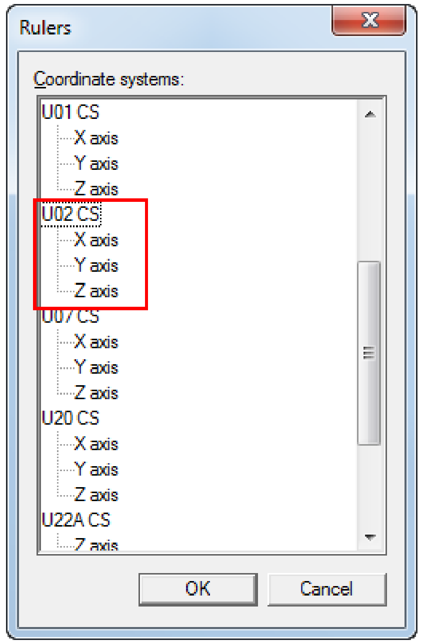
9. Use the Fit command from the main toolbar to fit all graphics into the view. 

10. Use the Common View control to look at an ISO view. Select the yellow corner indicated below.

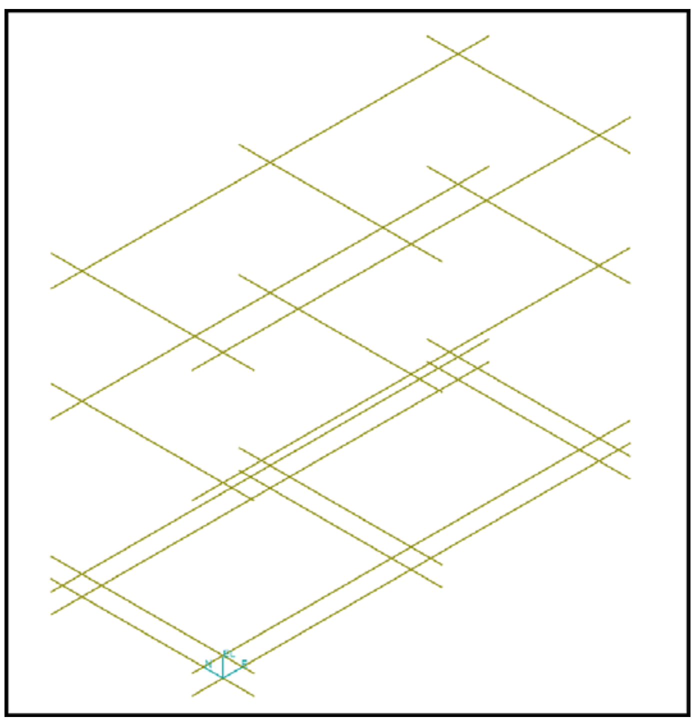


11. Go to the main menu and select View → Ruler option to open the ruler dialog box.

12. Hold the <***Control***> key down and de-highlight the coordinate system rulers called U02 CS. Select the "OK" button.



13. Your View should now resemble the following graphic.



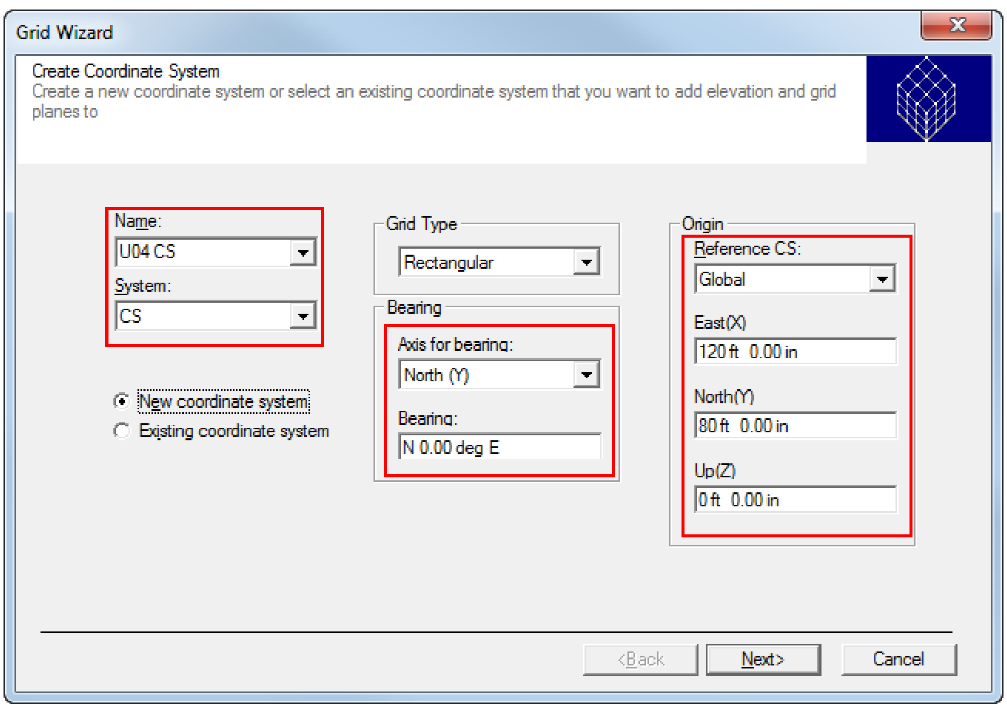
LAB-1B: Placing Grids – U04

Objectives

After completing this lab, you will be able to:

- Understanding the grid entities and relationships
- Place Grids / Coordinate systems, using Grid Wizard Command

1. Open or create a session file and define an "U04 & U04 CS" filter for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command from the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:
   Name: U04 CS
   Axis for bearing: North (Y)
   Bearing: N 0.00 deg E
   Origin:
   Reference CS: Global
   East (X): 120 ft
   North (Y): 80 ft
   Up (Z): 0 ft



Next, click "Next" button

© Copyright 2014 Intergraph Corporation

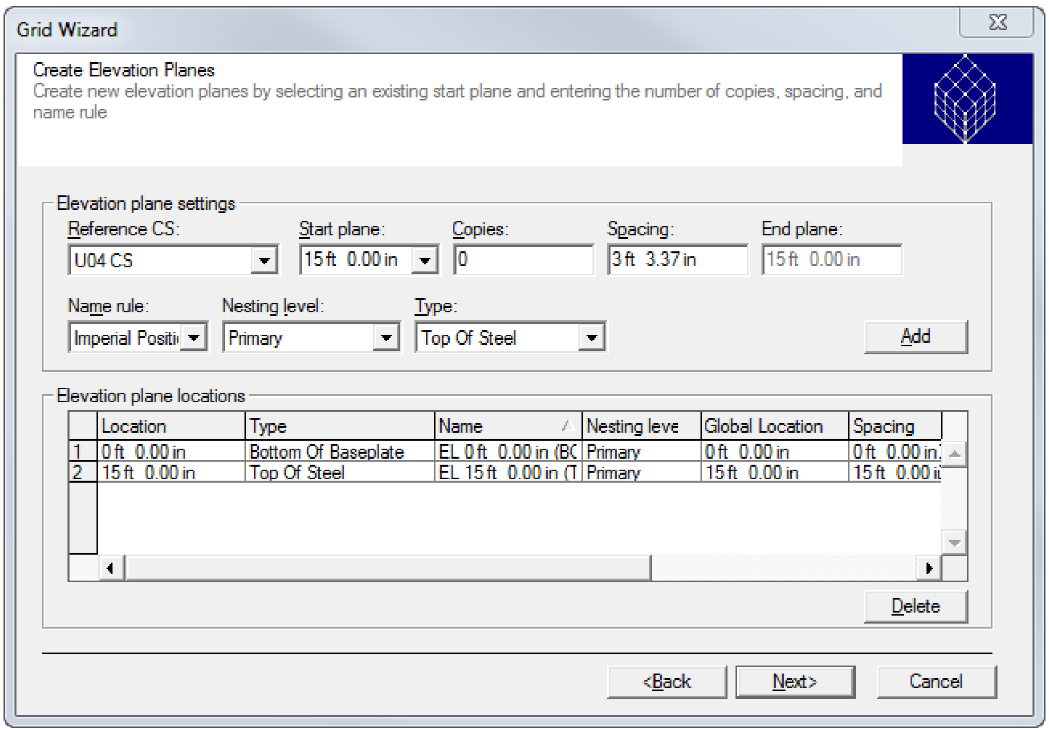
5. **Elevation Planes:** Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

   Set the Name rule to Imperial Position for All Grid Planes.

   Set the Nesting level to Primary for All Elevation Planes.

Step	Start Plane	Copies	Spacing	Name Rule	End Plane	Type
1	0 ft 0 in	0	Ignore	Imperial Position	0 ft 0 in	Bottom of Baseplate
2	15 ft 0 in	0	Ignore	Imperial Position	15 ft 0 in	Top of Steel

   Select the "Add" button.



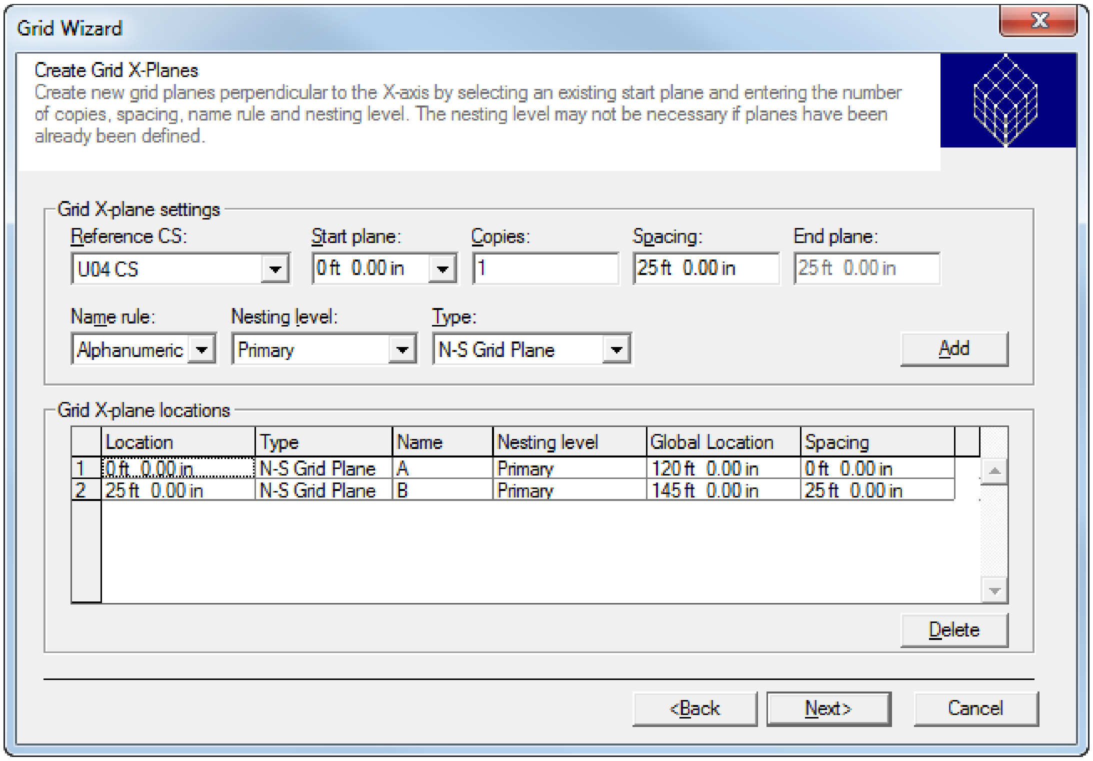
   Next, click "Next" button.

6. **Grid X-Planes**: Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:
   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U04 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	25 ft 0 in	25 ft 0 in	Primary	N-S Grid Plane

   Select the "Add" button.

© Copyright 2014 Intergraph Corporation

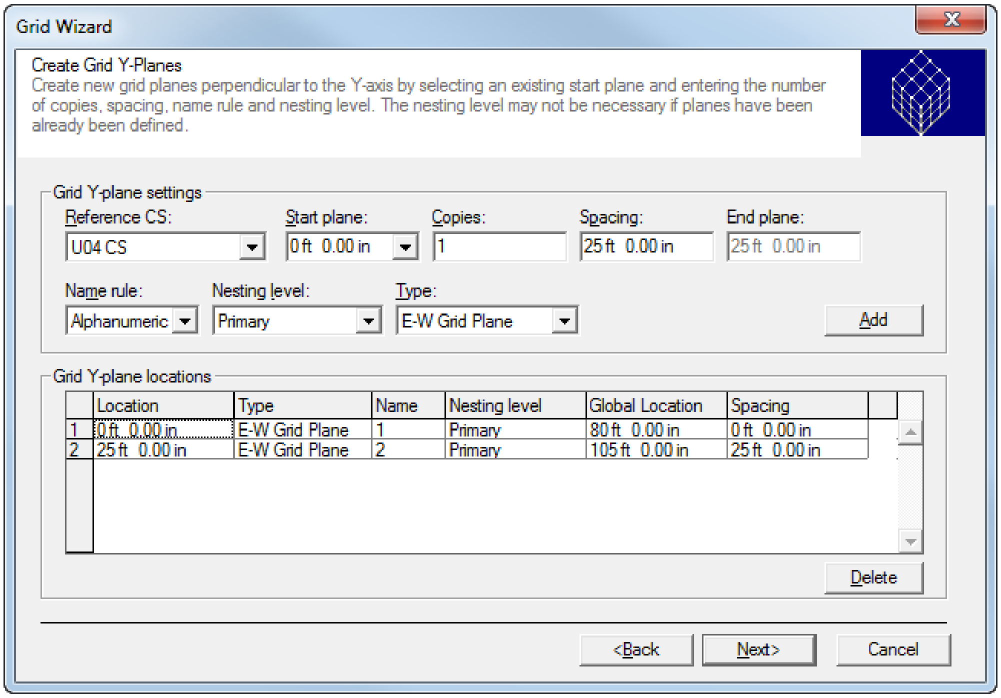


Next, click "Next" button.

7. **Grid-Y Planes**: Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:
Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U04 CS in the Reference CS pull down menu.

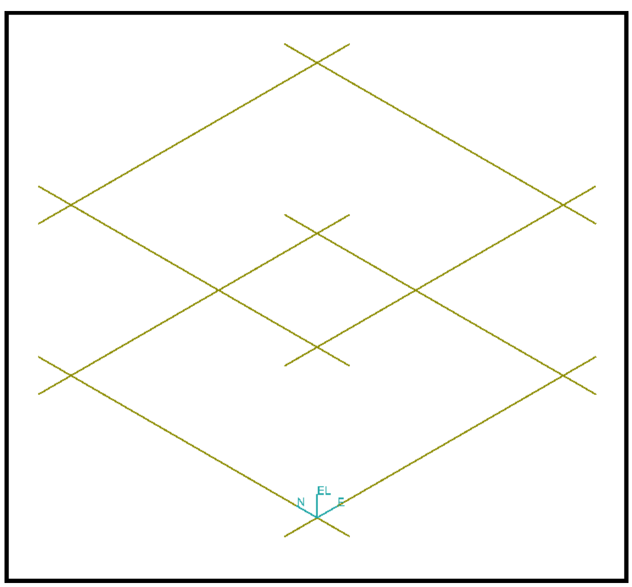
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	25 ft 0 in	25 ft 0 in	Primary	E-W Grid Plane

Select the "Add" button.



Next, click “Next” button.

8. Hit “Finish” button on Associated Elevation Plane form.
9. Use the Fit command from the main toolbar to fit all graphics into the view.
10. Go to the main menu and select View → Ruler option to open the ruler dialog box.
11. Press the <***Control***> key and unselect all items. Select the “OK” button to turn off the rulers.
12. Your View should now resemble the following graphic.



© Copyright 2014 Intergraph Corporation

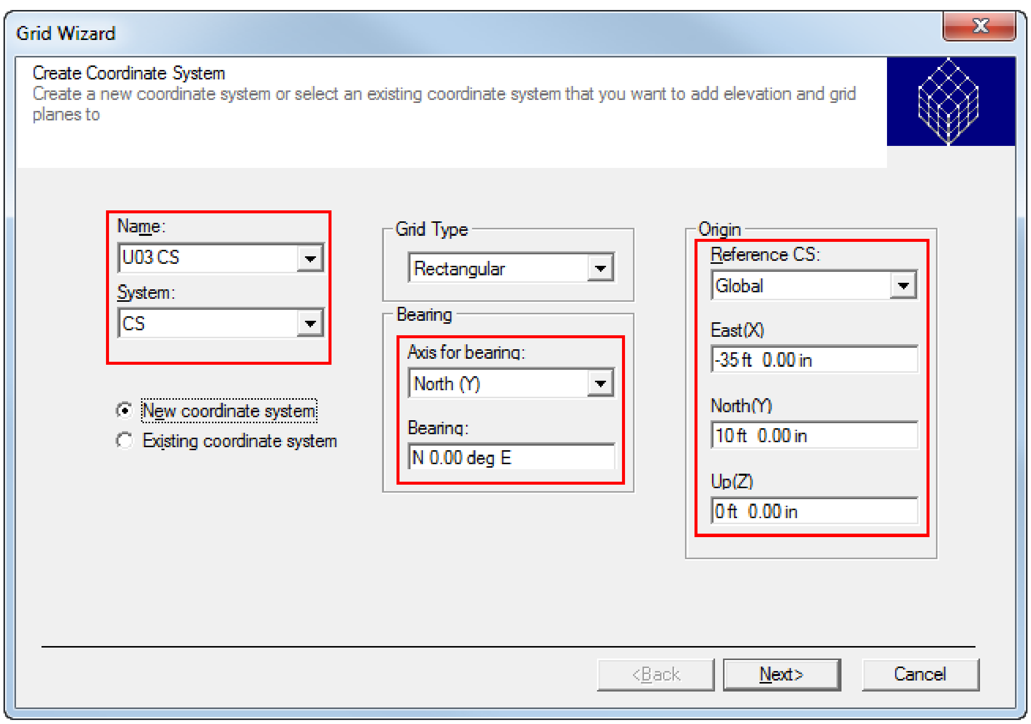
LAB-1C: Placing Grids – U03

Objectives

After completing this lab, you will be able to:

- Understanding the grid entities and relationships
- Place Grids / Coordinate systems, using Grid Wizard Command

1. Open or create a session file and define an "U03 & U03 CS" filter for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command on the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:
   Name: U03 CS
   Axis for Bearing: North (Y)
   Bearing: N 0.00 deg E
   Origin:
   Reference CS: Global
   East (X): -35 ft
   North (Y): 10 ft
   Up (Z): 0 ft



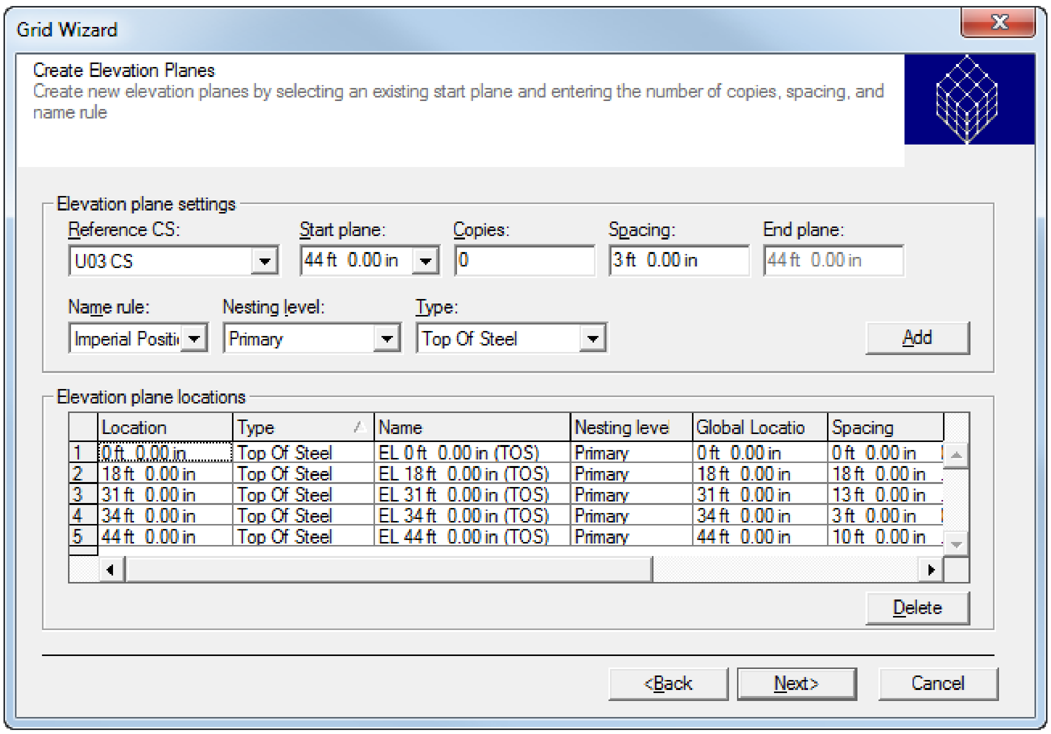
Next, click "Next" button.

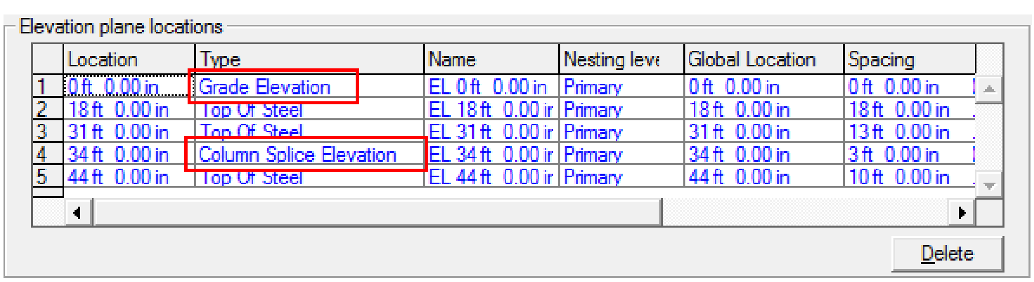
5. **Elevation Planes:** Create the Elevation Planes based on the following information:
Set the Name rule to Imperial Position for All Grid Planes.

Step	Start Plane	Copies	Spacing	End Plane	Nesting level	Type
1	0 ft 0 in	1	18 ft 0 in	18 ft 0 in	Primary	Top of Steel
2	31 ft 0 in	1	3 ft 0 in	34 ft 0 in	Primary	Top of Steel
3	44 ft 0 in	0	3 ft 0 in	44 ft 0 in	Primary	Top of Steel

Select the "Add" button.

6. In the Elevation Plane Locations setting, edit the Type as follows:





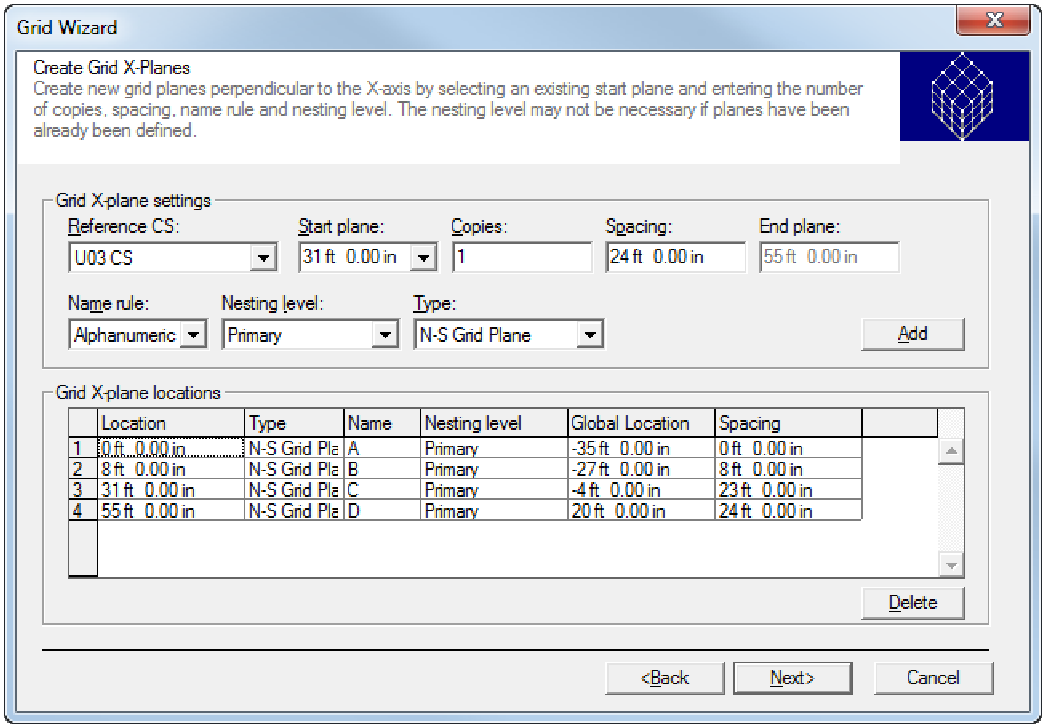
Next, click "Next" button.

7. **Grid X-Planes:** Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:
Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U03 CS in the Reference CS pull down menu.

© Copyright 2014 Intergraph Corporation

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	8 ft 0 in	8 ft 0 in	Primary	N-S Grid Plane
2	31 ft 0 in	1	24 ft 0 in	55 ft 0 in	Primary	N-S Grid Plane

Select the “Add” button.



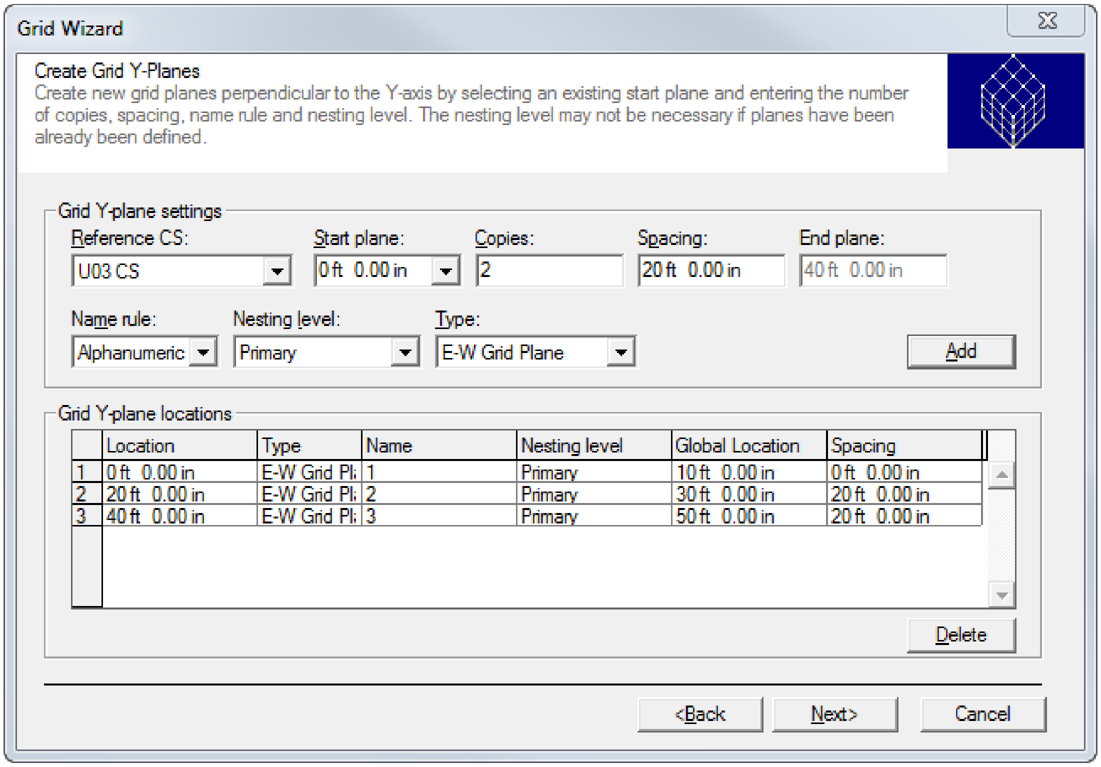
Next, click “Next” button.

8. **<u>Grid-Y Planes</u>:** Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:
   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U03 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	2	20 ft 0 in	40 ft 0 in	Primary	E-W Grid Plane

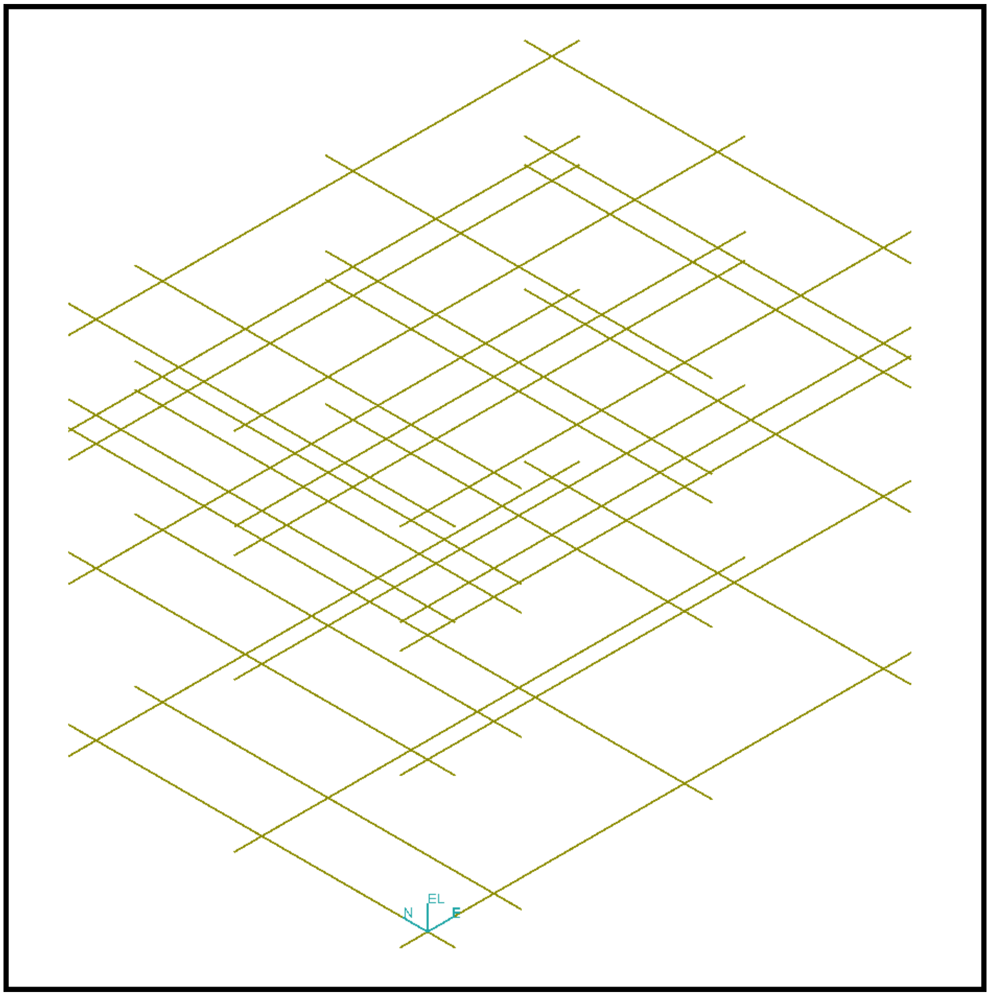
Select the “Add” button.

© Copyright 2014 Intergraph Corporation



Next, click “Next” button.

9. Hit “Finish” button on Associated Elevation Plane form.
10. Go to the main menu and select View → Ruler option to open the ruler dialog box.
11. Press the <Control> key and unselect all items. Select the “OK” button to turn off the rulers.
12. Your View should now resemble the following graphic.


EL
N
E

LAB-2: Editing Grid Planes

Objectives

After completing this lab, you will be able to:

- Add grid planes to existing grid system
- Use SmartStep Plane Ribbon Bar

Overview:

A grid system is a coordinate system and a set of reference planes and/or cylinders defined relative to that coordinate system. You can create, edit, or delete the reference planes/cylinders.
The **Place Elevation Plane** command creates one or more elevation planes.
The **Place Grid Plane** command creates one or more planes along either the x (or East) axis or the y (or North) axis.
The **Place Radial Grid** command places one or more concentric cylinders or radial planes passing through the center of the cylinders.
When you select one or multiple consecutive planes (or cylinders) on the same coordinate system axis, you can edit the position of the planes using the commands displayed on the **Edit** ribbon.

1. Open or create a session file and define an “U04 & U04 CS” filter for your workspace.
2. Go to the Grid Task environment.
3. Make sure the Active Permission Group is set to *Misc***.**

<u>Placing Elevation Plane:</u>

4. Select Place Elevation Plane command button from the vertical toolbar.

The next step is to specify values for the **Reference**, **Offset**, **Copies,** and **Nesting Level** parameters on the **Elevation Plane Horizontal** ribbon:

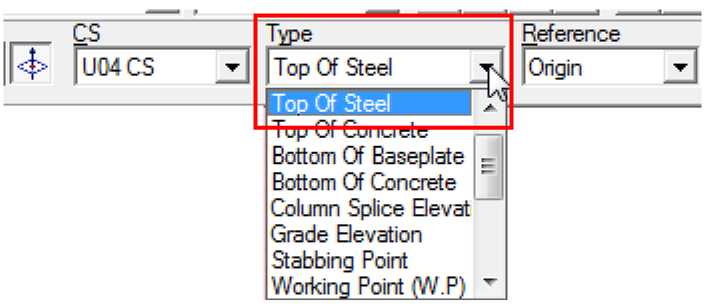
- The **Reference** can be the origin or a selected elevation plane.
- When you are in the **Elevation Plane Position** smartstep of the **Elevation Plane Placement** command, the software prompts you to enter a point to position the plane. The **Offset** box dynamically indicates the distance from the reference defined by your current cursor position. You can click in the graphic view to create the plane at a graphically selected location. If you use the step function of the **PinPoint** command, this can be a very easy way to position the planes without keying in the values. *If you key in an offset, the plane is placed when you press the **<u>Enter</u>** key*. After the elevation plane is created, the command cycles to again accept input for the next plane position.
- The command defaults to the last used nesting level. You must set the nesting level before entering an offset by data point or by key in since that creates the plane.
- The nesting level is used by some of the naming rules provided in the default catalog. Nesting level can be **Primary**, **Secondary**, or **Tertiary**. This represents a hierarchy along an axis. Between two adjacent planes, a difference of only one level is allowed. Therefore, you can place secondary planes between two primary planes and tertiary planes between two secondary planes.

© Copyright 2014 Intergraph Corporation

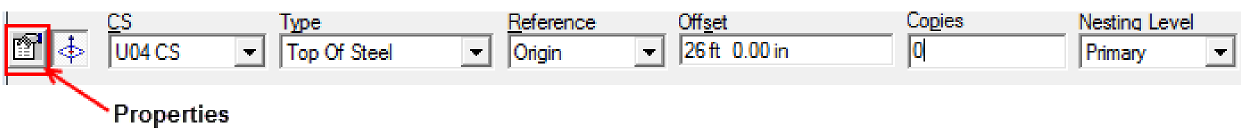
- The **Copies** option allows you to place multiple elevation planes. It indicates the number of additional elevation planes to be placed in the grid. By default, the value in this box is 0. With this value, a single elevation plane is placed by using the specified reference plane as the starting point and the offset as the distance to the new elevation plane.

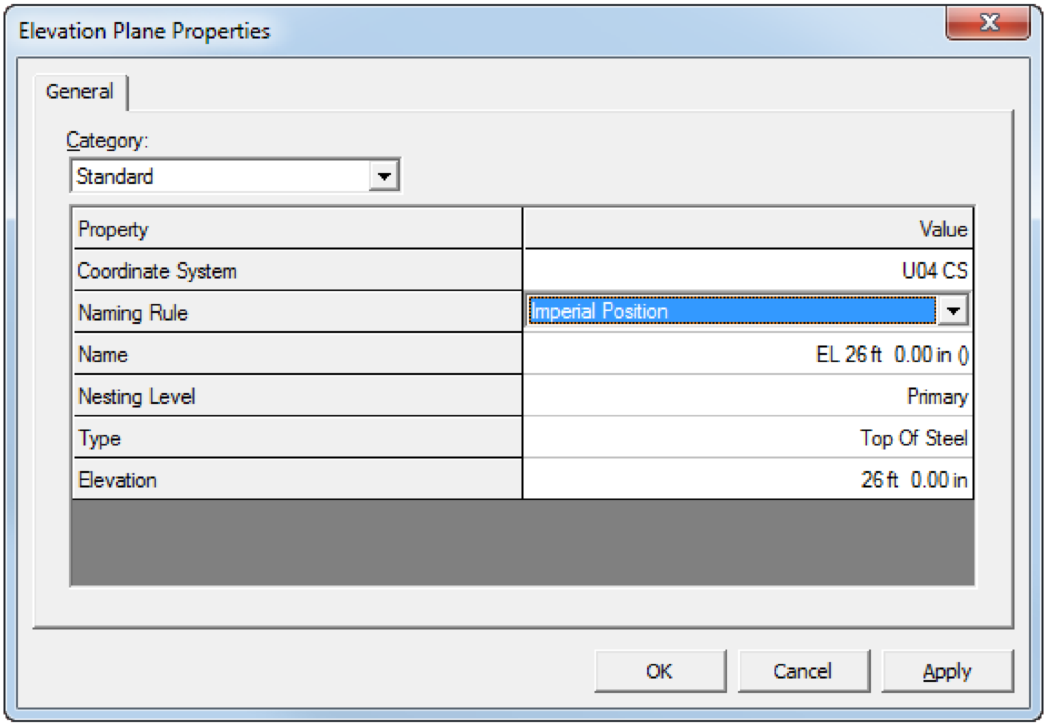
5. In the Elevation Plane horizontal ribbon bar enter the following parameters:

Nesting level:	Primary
Reference:	Origin
Copies:	0
Coordinate System:	Pick U04 CS
Type:	Pick Top of Steel



Name: Open the setting properties page and change the name rule to Imperial Position and click OK.





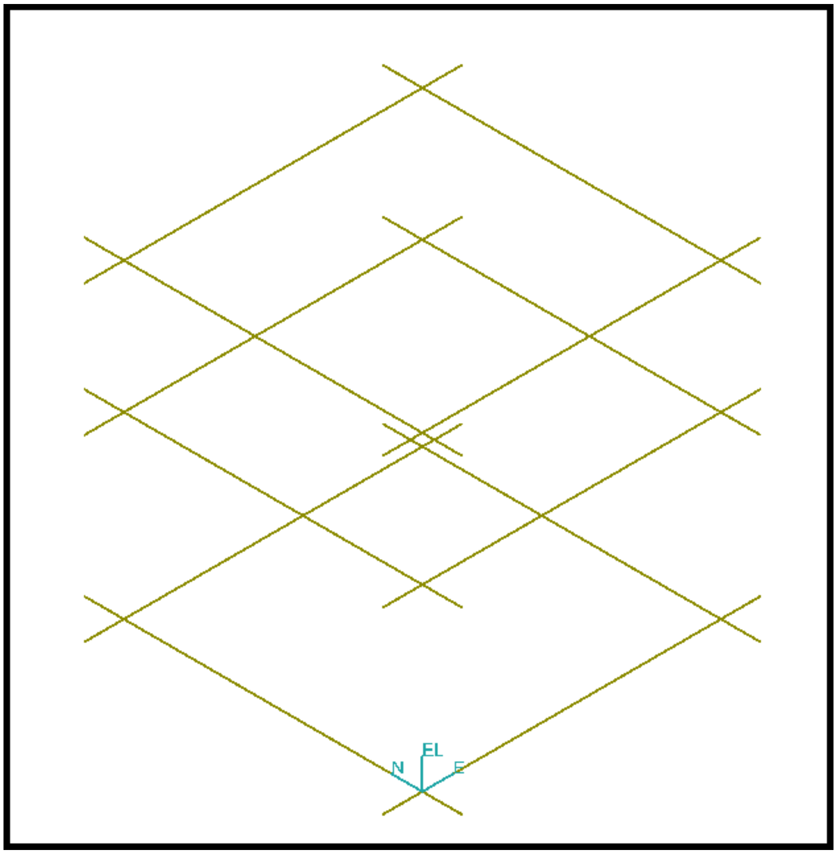
6. In the Elevation Plane horizontal ribbon bar, key-in the offset value and hit **<Enter> to commit the transaction**.

   Offset value: 26 ft

7. The system places the elevation plane marker in the Ruler after entering the above parameters. **<u>Right mouse click in the workspace to exit the command.</u>**

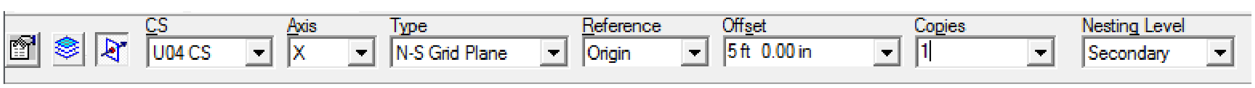
   Your View should now resemble the following graphic.

   Note: The labeling of elevation planes is for better explanation.



Placing Grid Planes: X-Axis

8. Select Place Grid Plane command button from the vertical toolbar.
9. In the Grid Plane horizontal ribbon bar enter the following parameters:



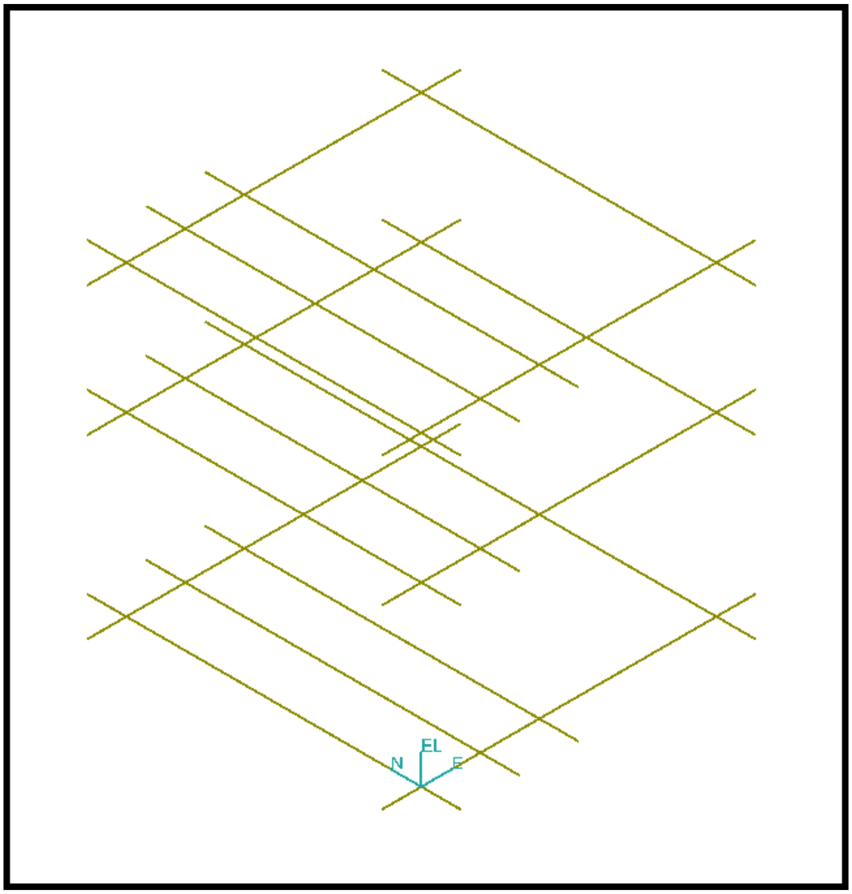
Coordinate System:	U04 CS
Axis:	X
Type:	N-S Grid Plane
Reference:	Origin
Copies:	1
Nesting Level:	Secondary
Name:	Open the properties page and change the name rule to Alphanumeric and Percent

10. Key in the offset value and hit <**Enter**> to commit the transaction.

    Offset value: 5 ft

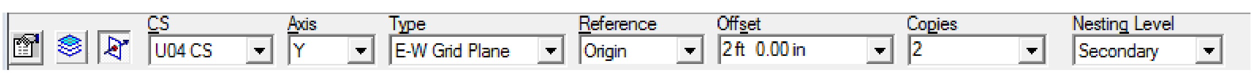
11. The system places the X-Planes marker in the Ruler and the gridlines. Right mouse click to exit the command.

© Copyright 2014 Intergraph Corporation



Placing Grid Planes: Y-Axis

12. Select Place Grid Plane command button from the vertical toolbar.
13. In the Grid Plane horizontal ribbon bar enter the following parameters:



Coordinate System:	U04 CS
Type:	E-W Grid Plane
Reference:	Origin
Copies:	2
Nesting Level:	Secondary
Name:	Open the properties page and change the name rule to Alphanumeric and Percent

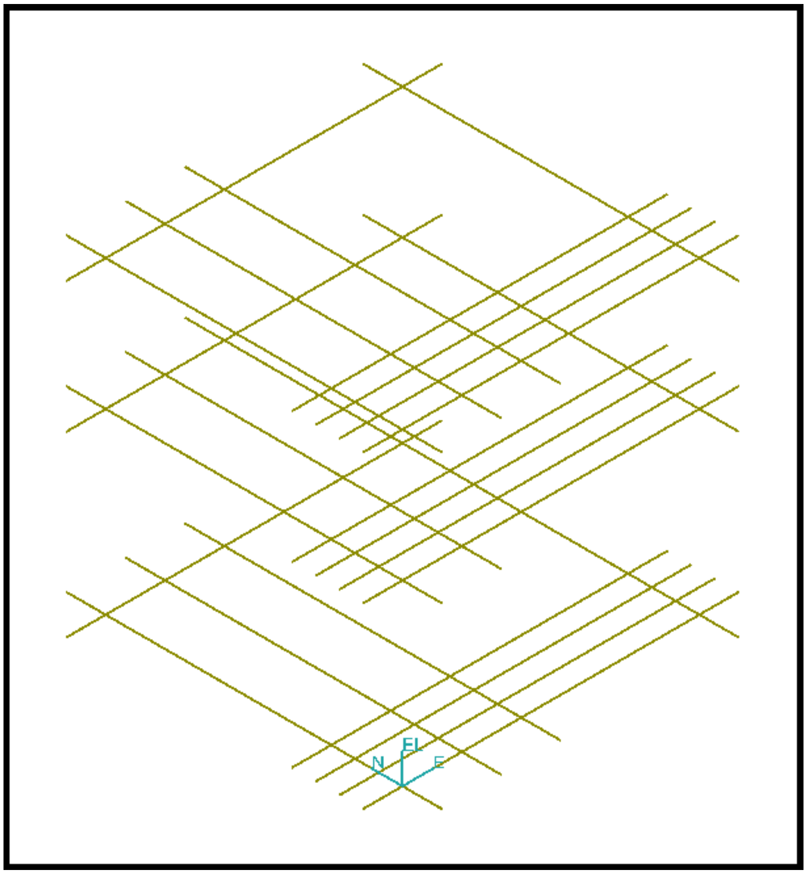
14. Key-in the offset value and hit <**Enter**> to commit the transaction.

    Offset value: 2 ft

15. The system places the Y-Planes marker in the Ruler and the gridlines. Right mouse click to exit the command.

    Your View should now resemble the following graphic.

© Copyright 2014 Intergraph Corporation


EL
N
E

LAB-3: Offshore Coordinate System - U05

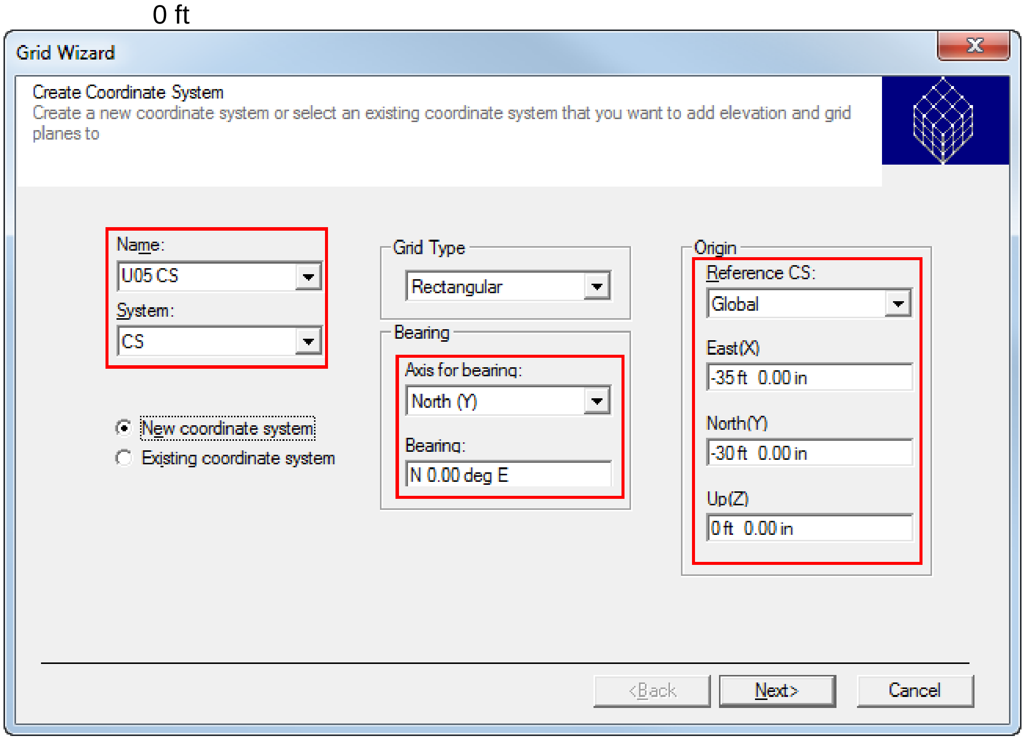
Objectives

After completing this lab, you will be able to:

- Place Grids / Coordinate systems, using Grid Wizard Command
- Familiarize with the grid plane properties

1. Open or create a session file and define an “U05 & U05 CS” filter for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command from the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:	U05 CS
Axis for Bearing:	North (Y)
Bearing:	N 0.00 deg E
Origin:	
Reference CS:	Global
East (X):	-35 ft
North (Y):	-30 ft
Up (Z):	0 ft

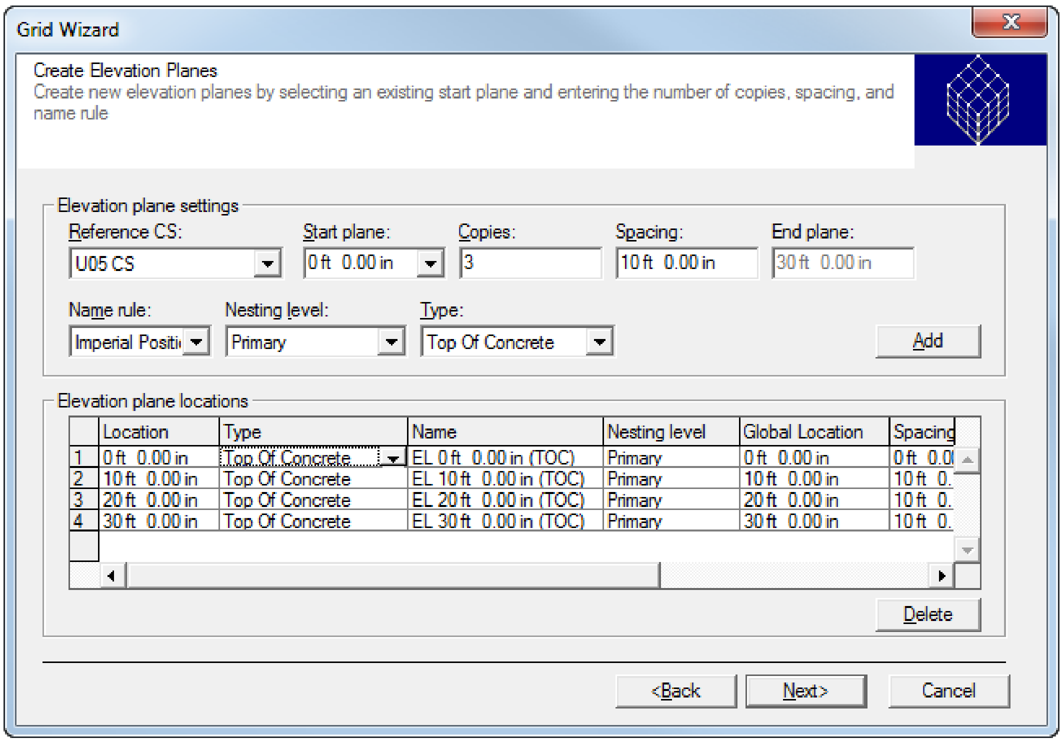


Next, click "Next" button.

5. Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

   Use Imperial Position as Name Rule.

Step	Start Plane	Copies	Spacing	End Plane	Nesting level	Type
1	0 ft 0 in	3	10 ft 0 in	30 ft 0 in	Primary	Top of Concrete

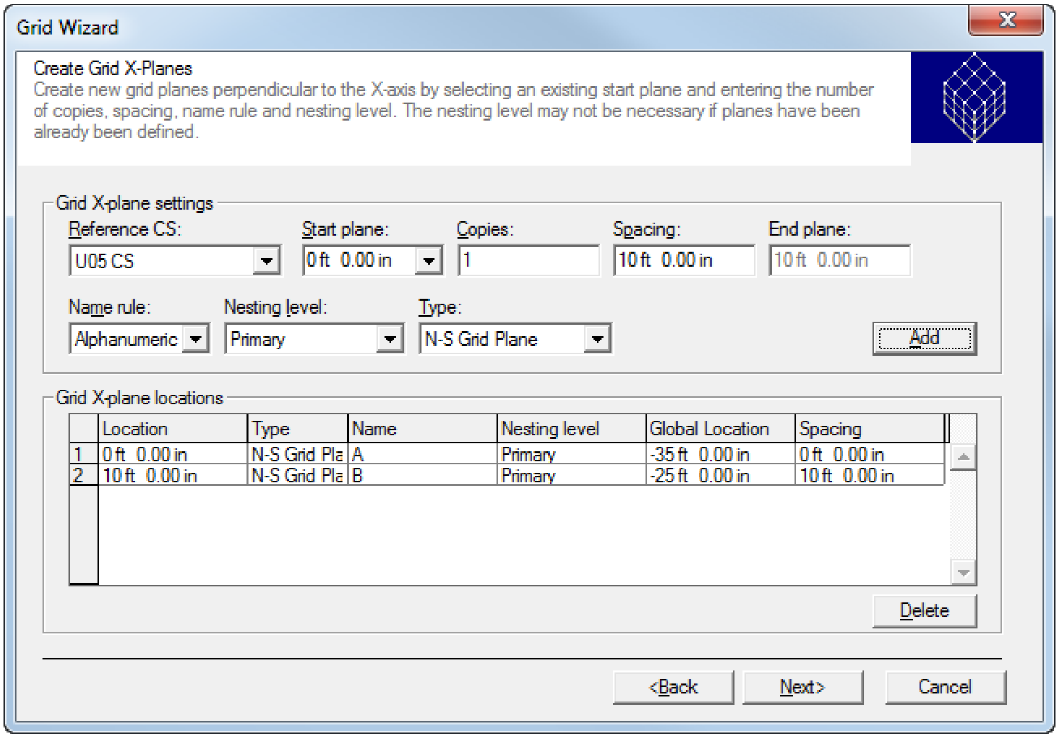


Next, click "Next" button.

6. Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:

   Use Alphanumeric and Percent as Name Rule.

Step	Start Plane	Copies	Spacing	End Plane	Nesting level	Type
1	0 ft 0 in	1	10 ft 0 in	10 ft 0 in	Primary	N-S Grid Plane



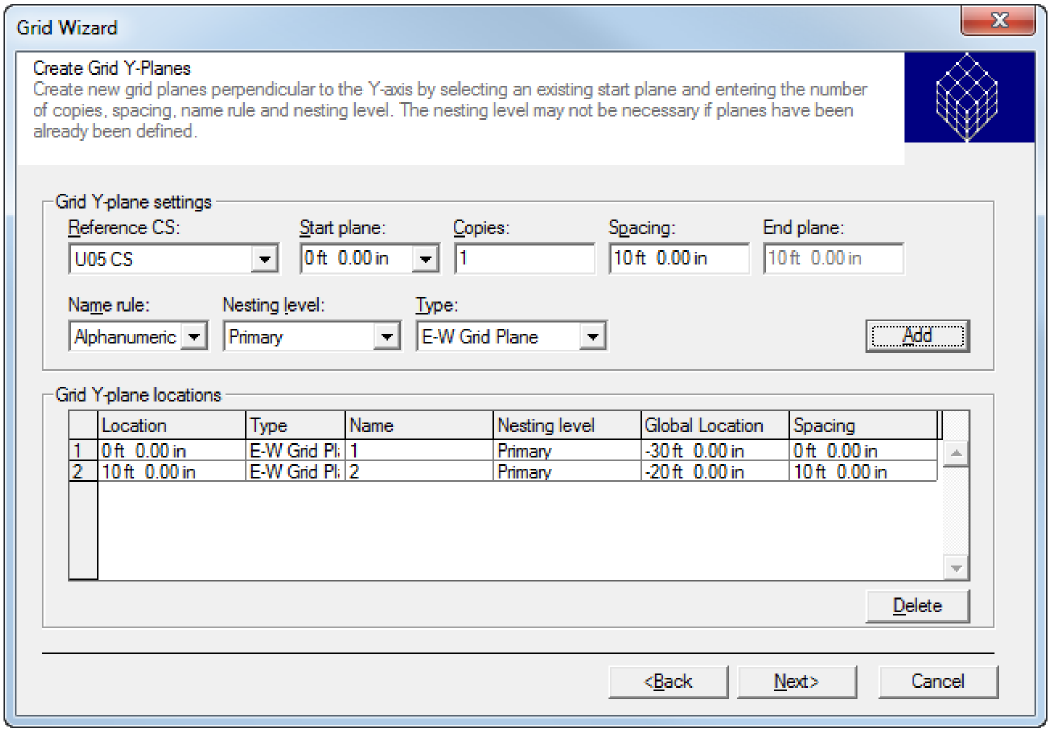
Next, click “Next” button.

7. Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

   Use Alphanumeric and Percent as Name Rule.

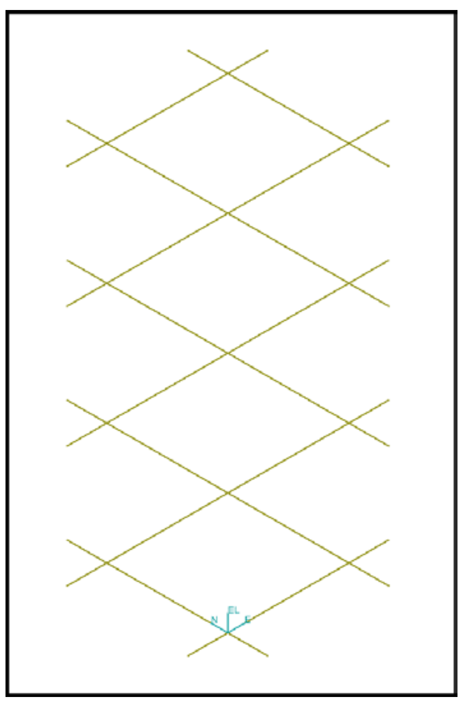
Step	Start Plane	Copies	Spacing	End Plane	Nesting level	Type
1	0ft 0in	1	10 ft 0 in	10 ft 0 in	Primary	E-W Grid Plane

© Copyright 2014 Intergraph Corporation



Next, click “Next” button.

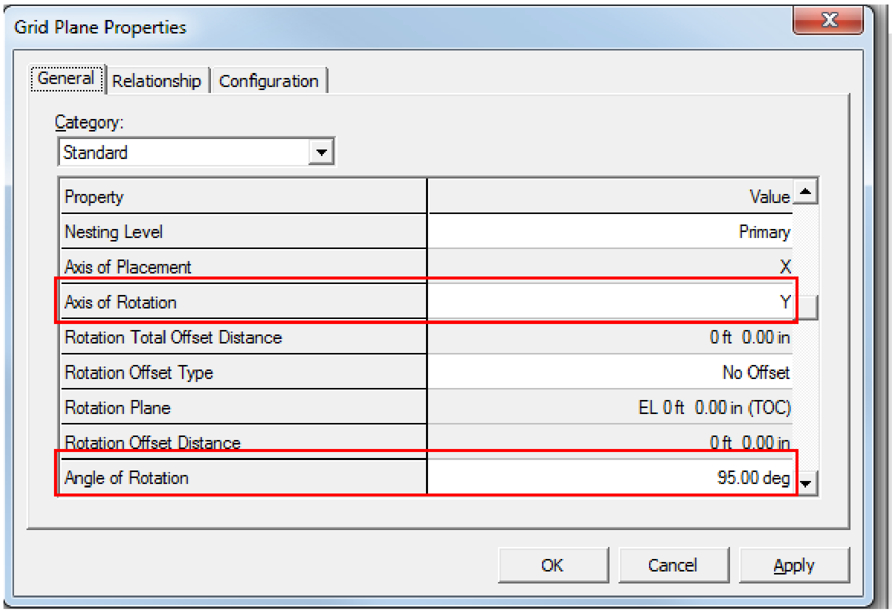
8. Select the “Finish” Button.
9. Turn off all Rulers. Your View should now resemble the following graphic.

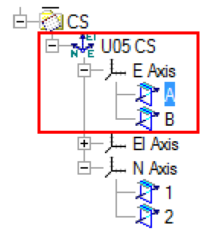


10. Select Grid Plane A in the Workspace Explorer to open the Edit Grid Plane ribbon bar.
11. Select the properties icon to open the properties page.

12. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation:	Y
Angle of Rotation:	95.0 deg

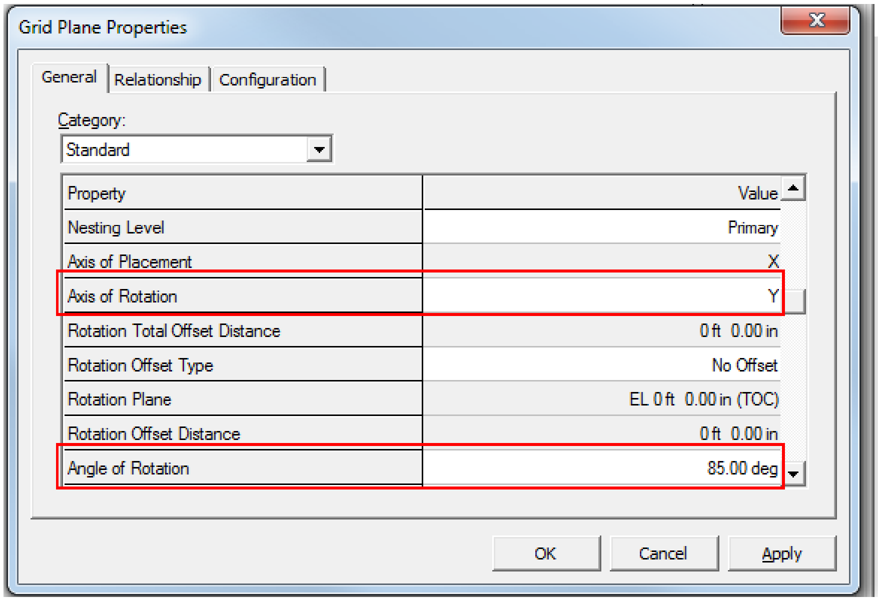


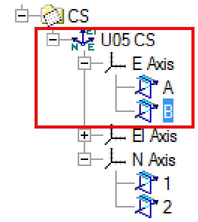


15. Hit “Apply” button to commit the transaction.
16. Select Grid Plane B in the Workspace Explorer.
17. Select the properties icon to open the properties page.
18. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation:	Y
Angle of Rotation:	85.0 deg

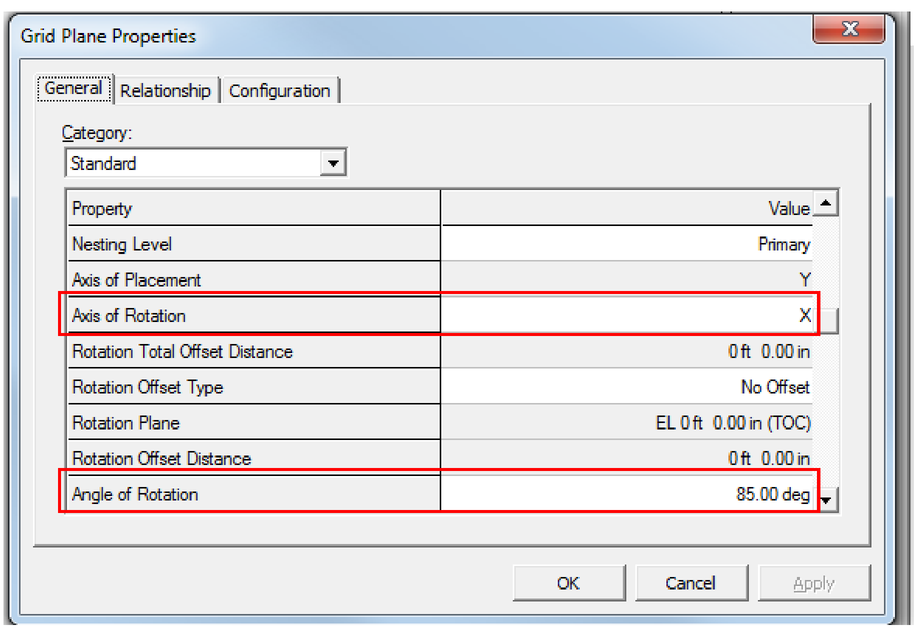
© Copyright 2014 Intergraph Corporation

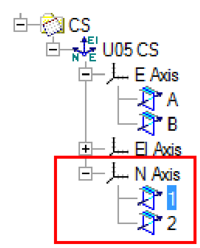




19. Hit “Apply” button to commit the transaction.
20. Select Grid Plane 1 in the Workspace Explorer.
21. Select the properties icon to open the properties page.
22. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation:	X
Angle of Rotation:	85.0 deg



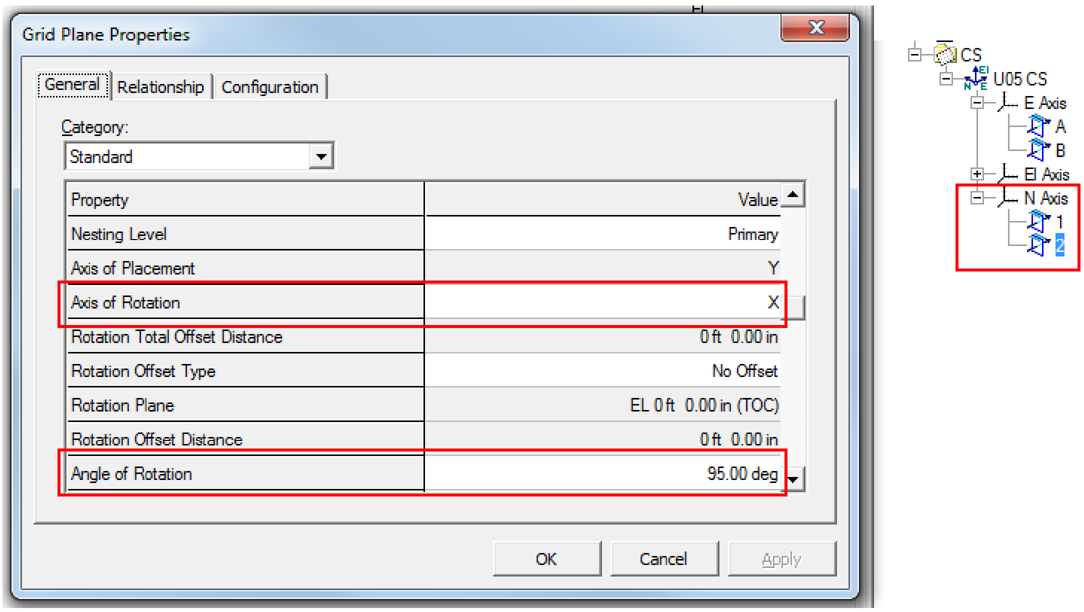


23. Hit “Apply” button to commit the transaction.

© Copyright 2014 Intergraph Corporation

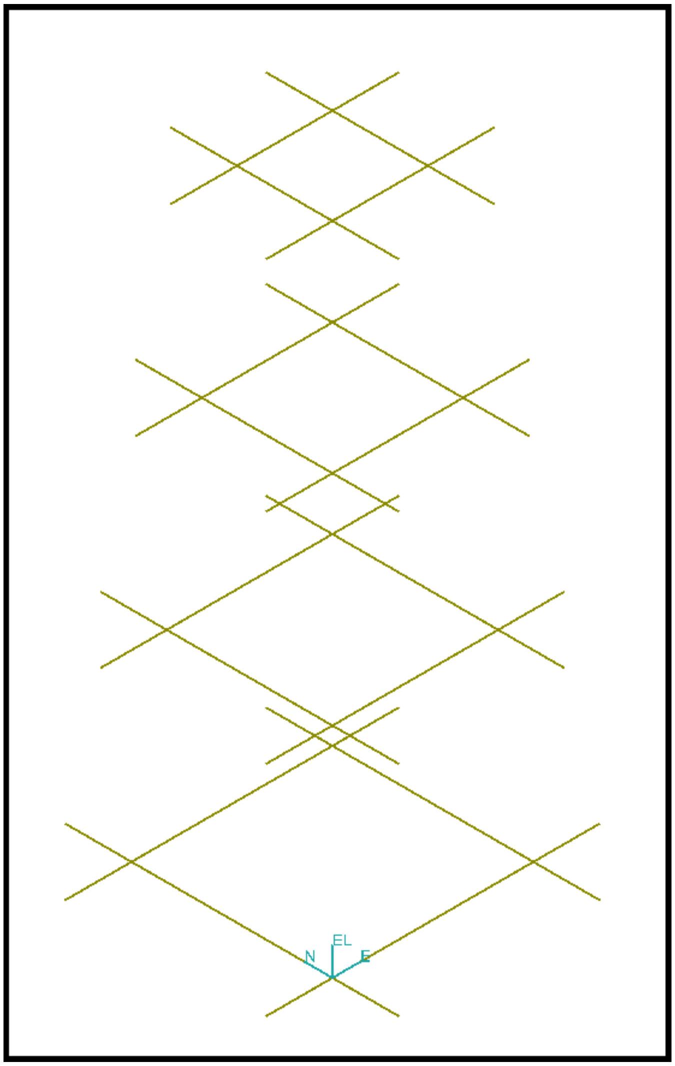
24. Select Grid Plane 2 in the Workspace Explorer.
25. Select the properties icon to open the properties page.
26. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation:	X
Angle of Rotation:	95.0 deg



27. Hit “OK” button to commit the transaction.

    Your View should now resemble the following graphic.


EL
N
E

LAB-4: Rotated Grid Coordinate System - U06

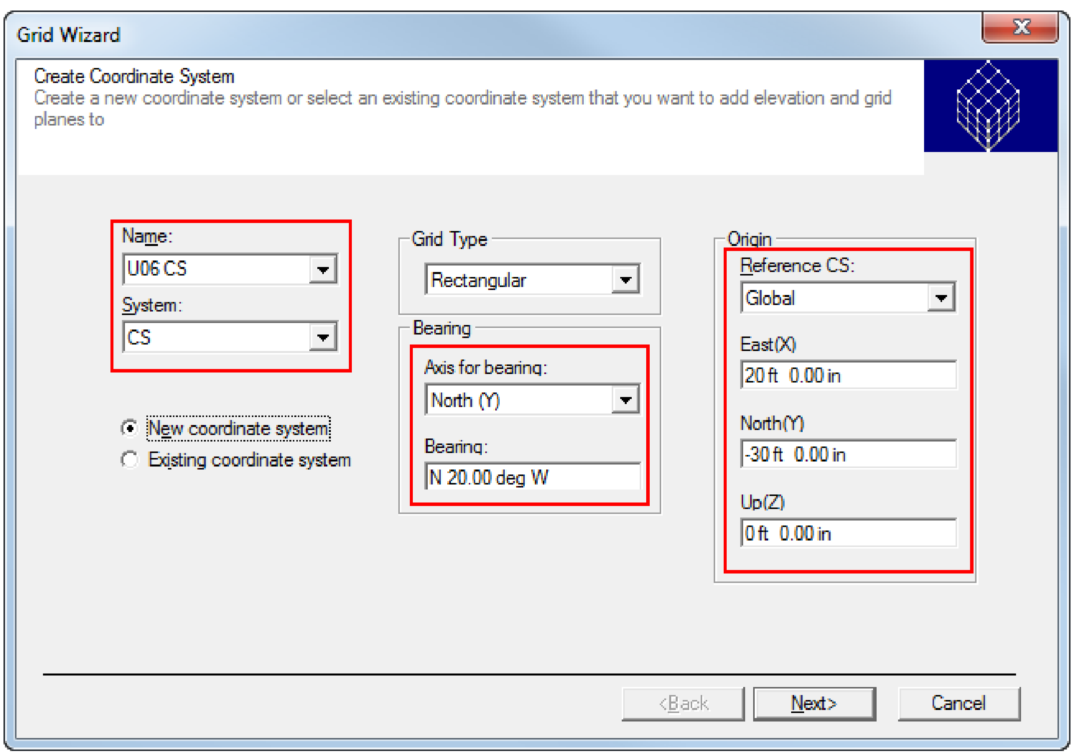
Objective

After completing this lab, you will be able to:

- Place Rotated Grids / Coordinate systems, using Grid Wizard Command

1. Open or create a session file and define an “U06 & U06 CS” filter for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command from the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:	U06 CS
Axis for Bearing:	North (Y)
Bearing:	N 20.00 deg W
Origin:	
Reference CS:	Global
East (X):	20 ft
North (Y):	-30 ft
Up (Z):	0 ft

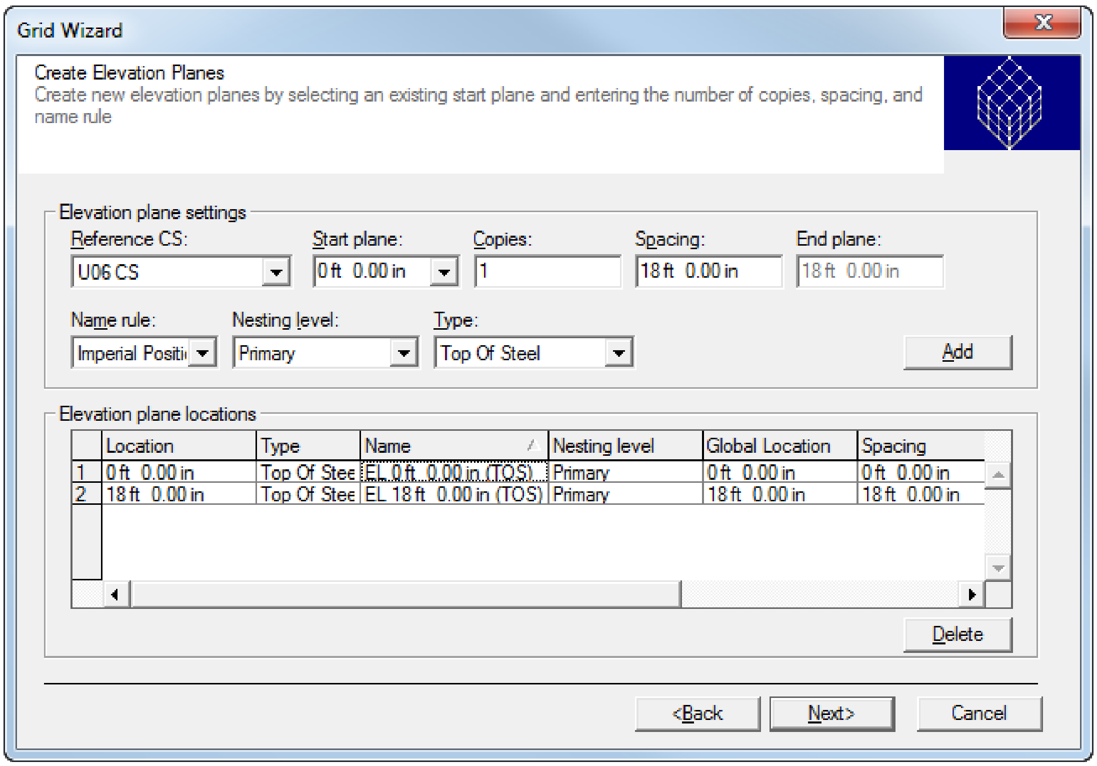


Next, click “Next” button.

5. Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

   Use Imperial Position as Name Rule.

Step	Start Plane	Copies	Spacing	End Plane	Nesting level	Type
1	0 ft 0 in	1	18 ft 0 in	18 ft 0 in	Primary	Top of Steel

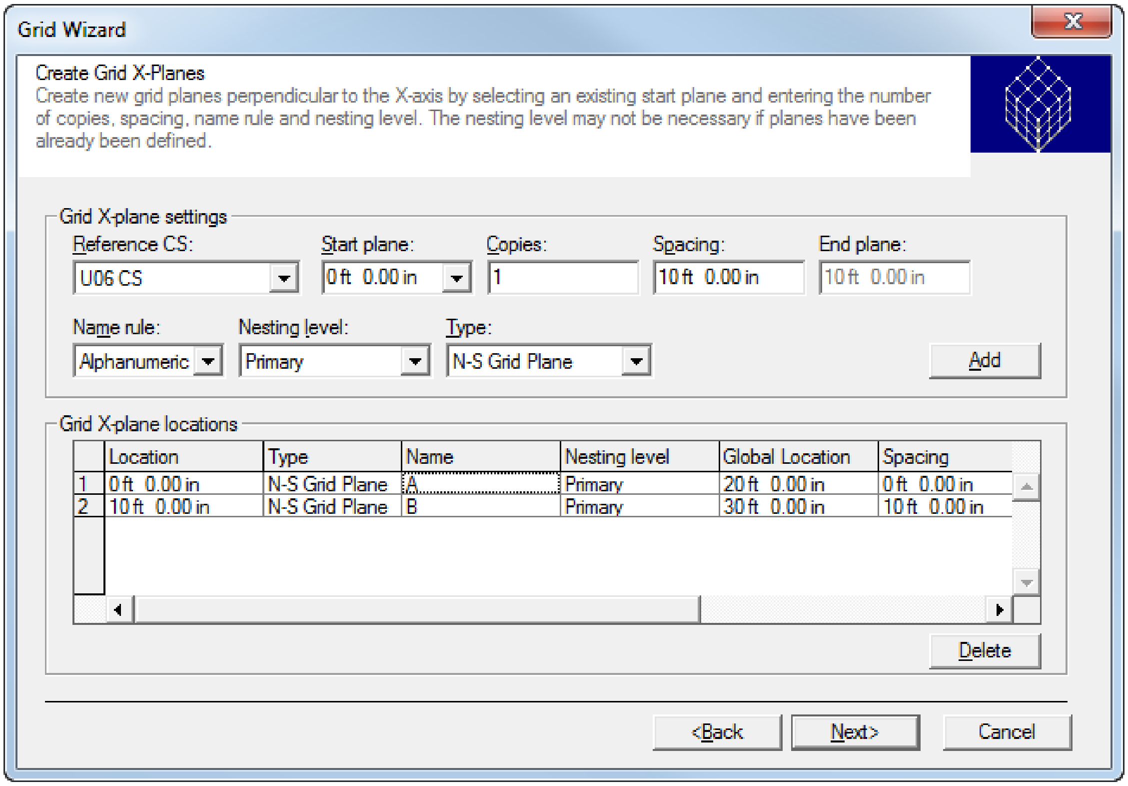


   Next, click “Next” button.

6. Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:

   Use Alphanumeric and Percent as Name Rule.

Step	Start Plane	Copies	Spacing	End Plane	Nesting level	Type
1	0 ft 0 in	1	10 ft 0 in	10 ft 0 in	Primary	N-S Grid Plane



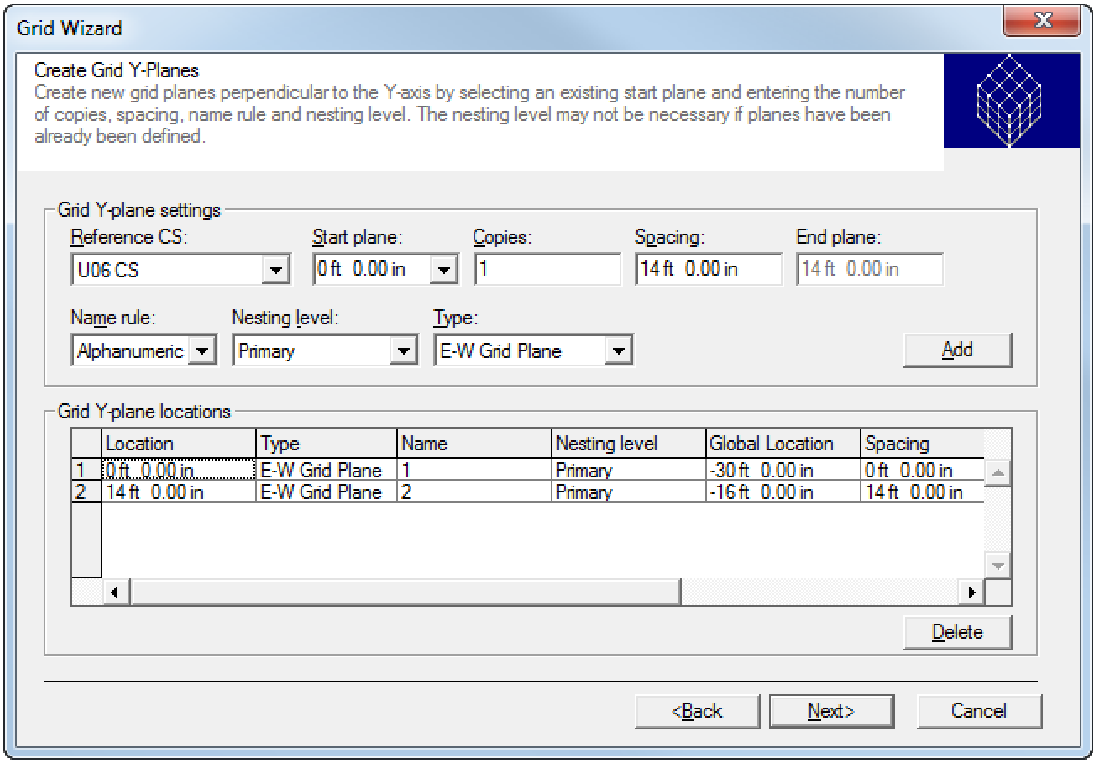
Next, click “Next” button.

7. Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

   Use Alphanumeric and Percent as Name Rule.

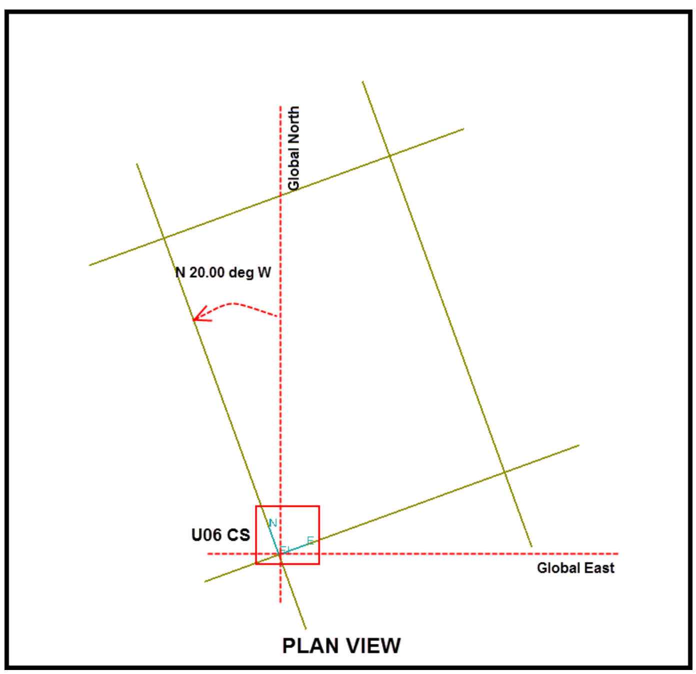
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	14 ft 0 in	14 ft 0 in	Primary	E-W Grid Plane

© Copyright 2014 Intergraph Corporation



Next, click “Next” button.

8. Hit “Finish” button on Associated Elevation Plane form.
9. Turn off all Rulers and switch to the plan view. Your View should now resemble the following graphic.



10. Turn off all Rulers.
11. Save your session. Select File -> Save.

© Copyright 2014 Intergraph Corporation

LAB-5: Moving Grids/Coordinate Systems - U09 CS

Objectives

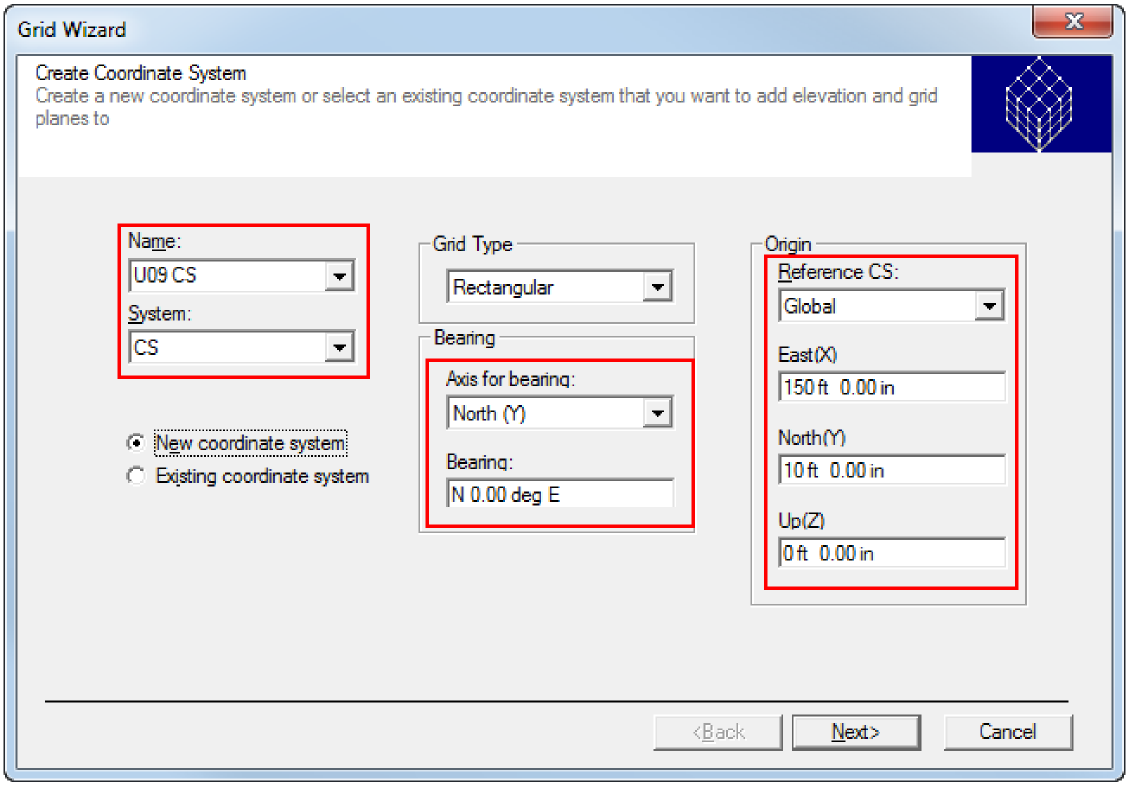
After completing this lab, you will be able to:

- Move the grids along with the coordinate systems (include planes).
- Move the coordinate system and maintain the same location of the grids (exclude planes).

1. Open or create a session file and define an “U09 & U09 CS” filter for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command from the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:	U09 CS
Axis for bearing:	North (Y)
Bearing:	N 0.00 deg E
Reference CS:	Global
Origin:	
East (X):	150 ft
North (Y):	10 ft
Up (Z):	0 ft

© Copyright 2014 Intergraph Corporation



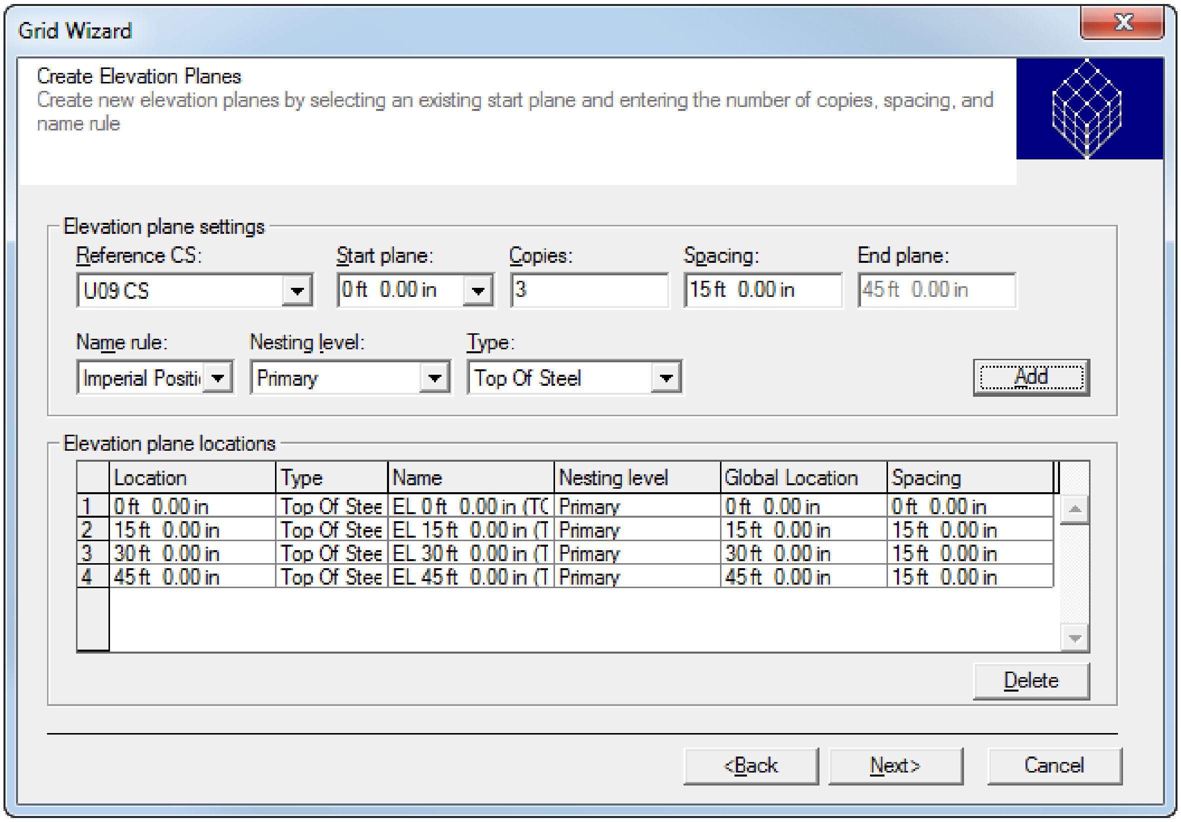
Next, click “Next” button

5. **<u>Elevation Planes</u>**: Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

   Set the Nesting level to Primary for All Elevation Planes.

Step	Start Plane	Copies	Spacing	Name Rule	End Plane	Type
1	0 ft 0 in	3	15 ft 0 in	Imperial Position	45 ft 0 in	Top of Steel

Select the “Add” button.



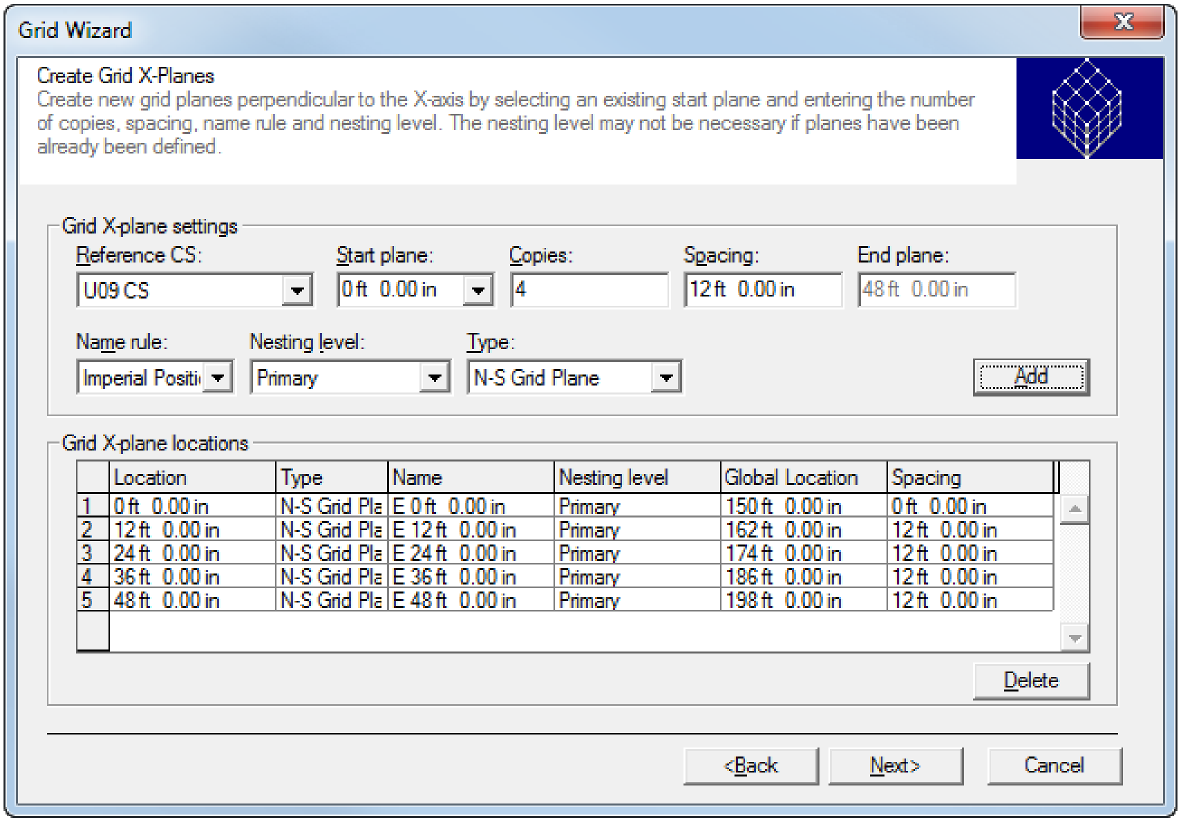
Next, click “Next” button.

6. **<u>Grid X-Planes</u>:** Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:
   Set the Name rule to Imperial Position for All Grid Planes. Select U09 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	Nesting Level	End Plane	Type
1	0 ft 0 in	4	12 ft 0 in	Primary	48 ft 0 in	N-S Grid Plane

Select the “Add” button.

© Copyright 2014 Intergraph Corporation



Next, click "Next" button.

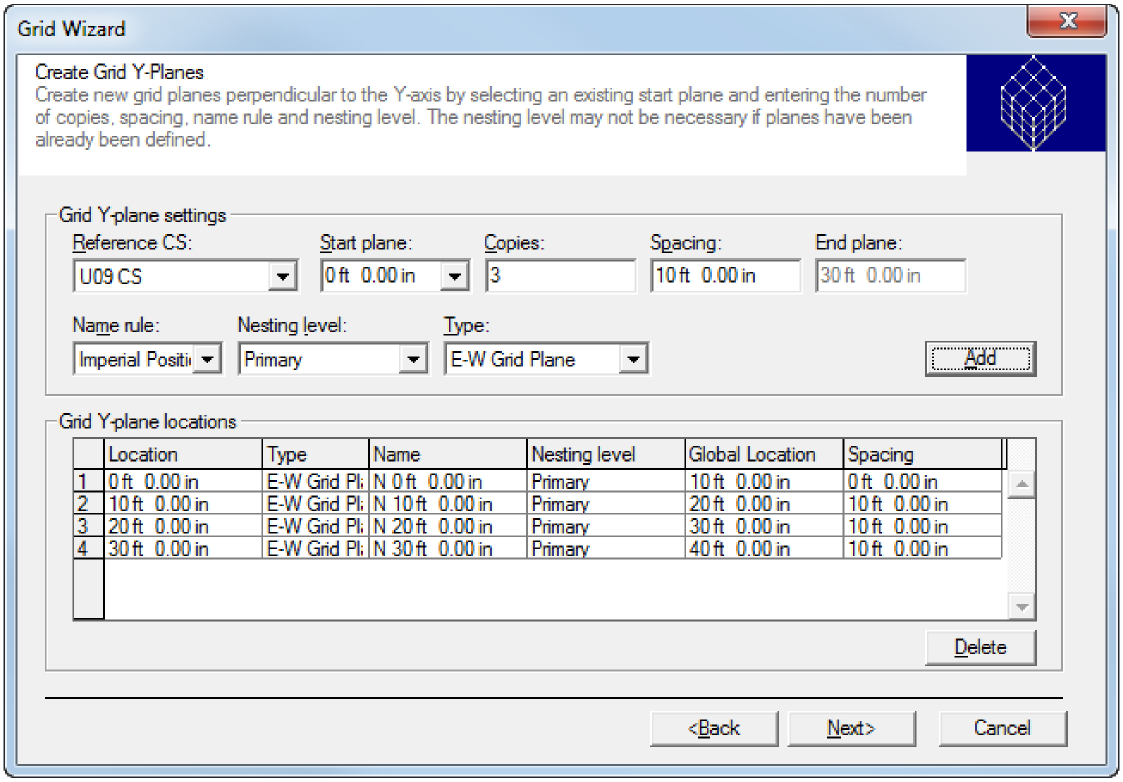
7. **Grid-Y Planes**: Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

   Set the Name rule to Imperial Position for All Grid Planes. Select U09 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	3	10 ft 0 in	30 ft 0 in	Primary	E-W Grid Plane

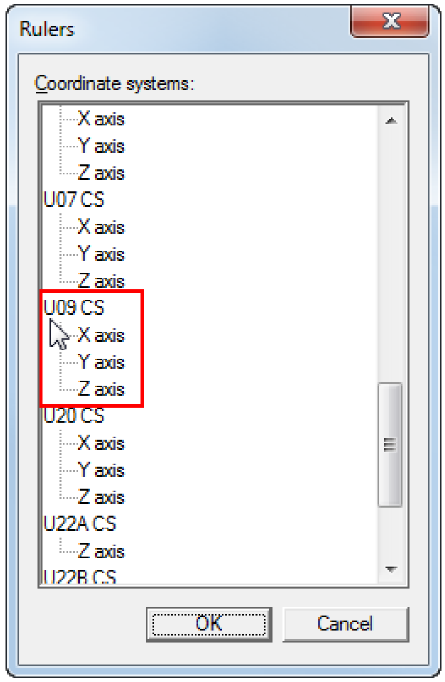
   Select the "Add" button.

© Copyright 2014 Intergraph Corporation

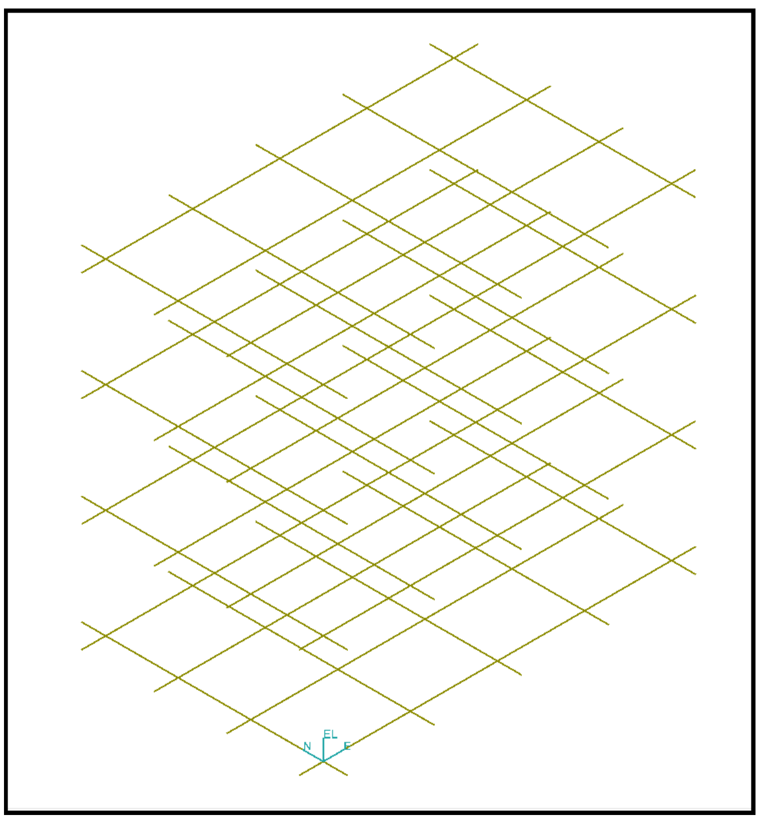


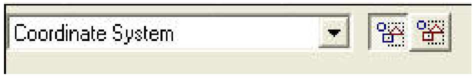
Next, click “Next” button.

8. Hit “Finish” button on Associated Elevation Plane form. The wizard will create the grids lines at all elevations.
9. Use the Fit command from the main toolbar to fit all graphics into the view.
10. Use the Common View control to look at an Isometric view..
11. Go to the main menu and select View -> Ruler option to open the ruler dialog box.
12. Hold the <*Control*> key down and de-highlight the coordinate system rulers called U09 CS. Select the “OK” button.

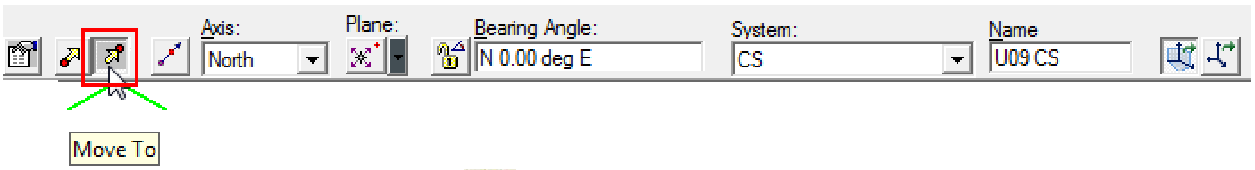


Your View should now resemble the following graphic.

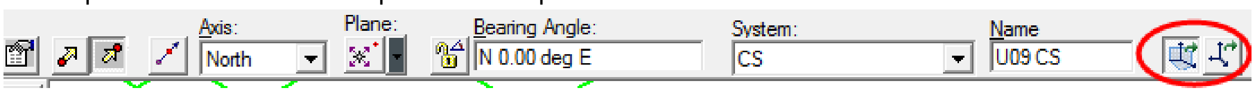


13. Set the locate filter to 

14. Select the coordinate system U09 CS from the workspace explorer.
15. On the ribbon bar, select the “Move To” option as shown in the image below,



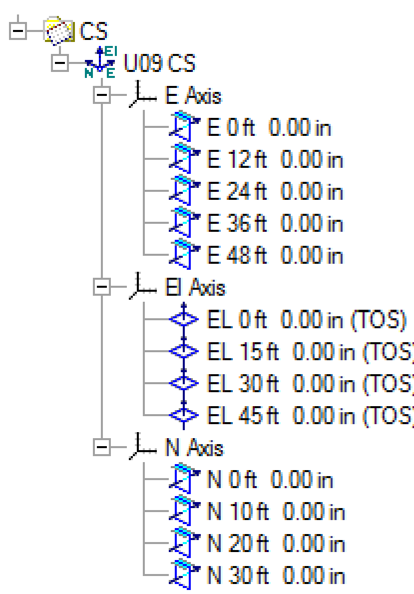
16. Include planes  and exclude planes  option is now active as shown in the ribbon bar.



Note:

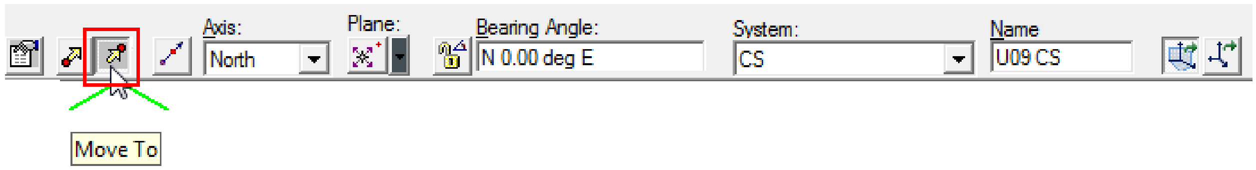
Include Planes *– This option will move the grid planes along with the coordinate system origin.This option is only available in modify mode when you are moving a coordinate system's origin.*
Exclude Planes *- This option will leave the grid planes at their current location and move the coordinate system to a new location. However,it will recalculate the grid plane relative position to the coordinate system.This option is only available in modify mode when you are moving a coordinate system's origin.*

17. By default, include planes option is selected. Move the mouse cursor in the workspace. Observe the preview of grids / coordinate systems moving along with mouse movement.
18. Left mouse click anywhere in the workspace. Grids and coordinate systems have been moved to a new location in the workspace.
19. Go to Edit → Undo Edit Coordinate System. The coordinate system and grids will return to the original location.
20. Select the coordinate system U09 CS from the workspace explorer.
21. Observe the values of the elevation plane in the workspace explorer,

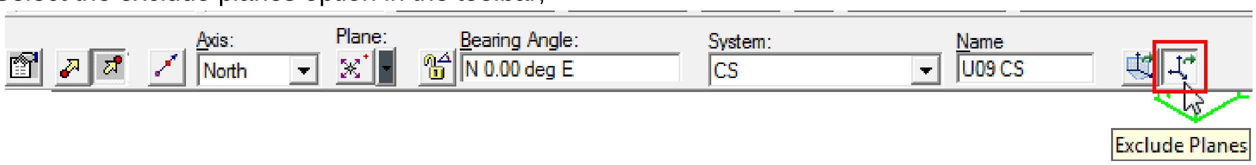


© Copyright 2014 Intergraph Corporation

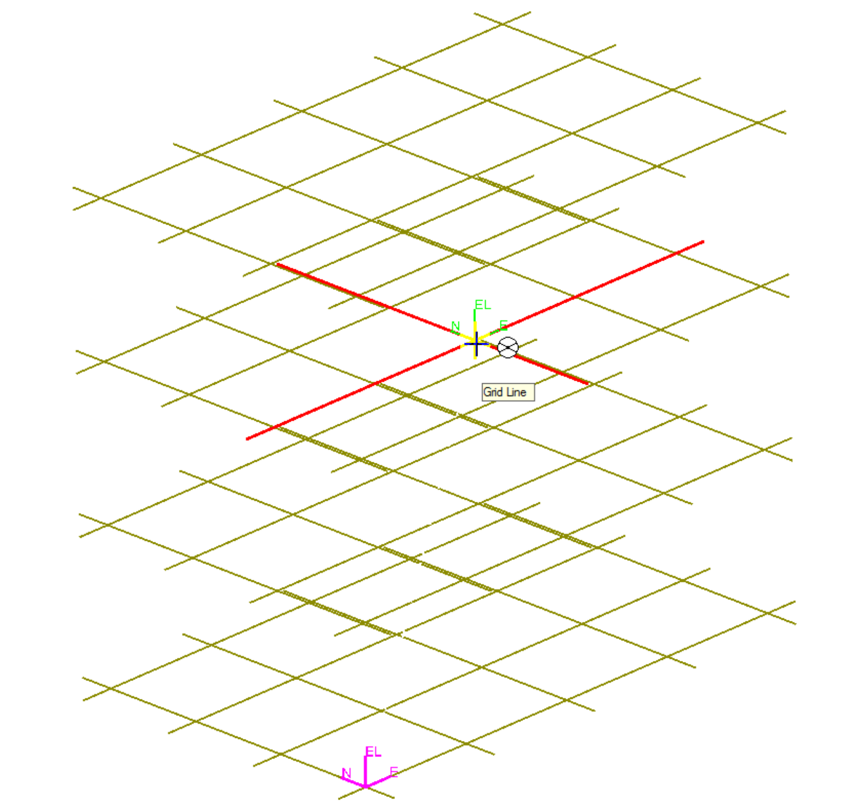
22. On the ribbon bar, select the “Move To” option as shown in the image below,



23. Select the exclude planes option in the toolbar,

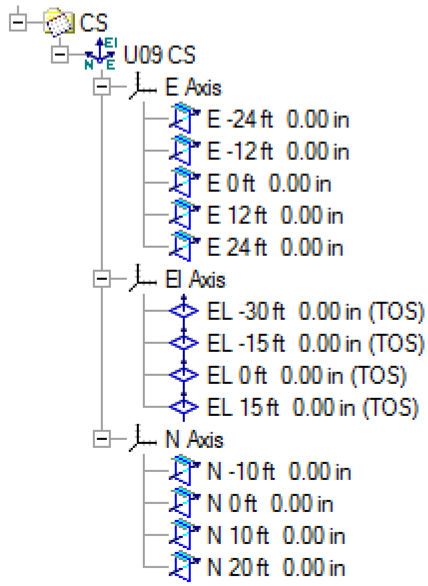


24. Move the mouse cursor in the workspace. Observe that the preview of the coordinate system moving along with the mouse movement. However, the grids do not move.
25. Move the mouse to the new location as shown in the image below,



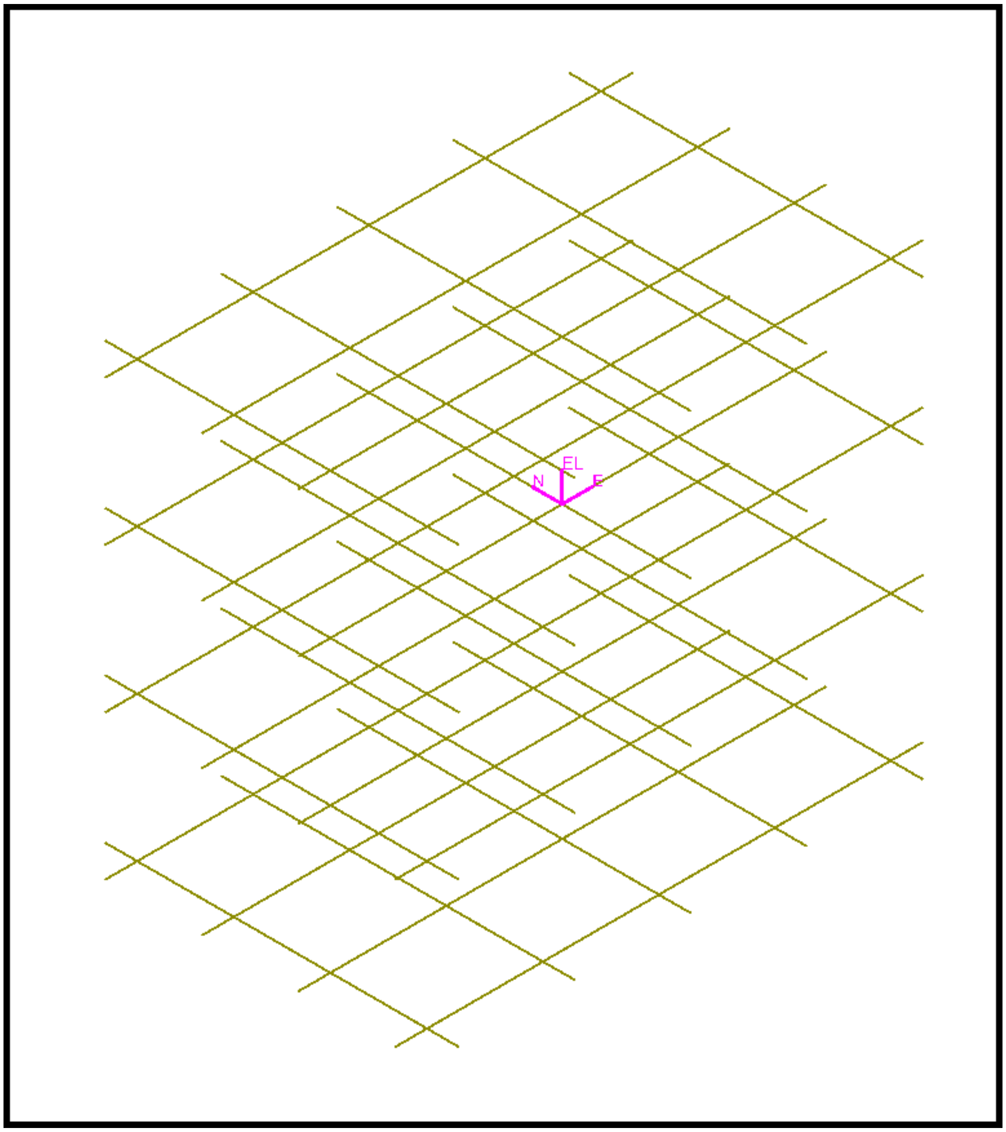
28. Left click on the grid intersection. The coordinate system has moved to a new location.
29. Observe the values of the elevation plane in the workspace explorer,

© Copyright 2014 Intergraph Corporation



Your view should resemble the following graphic.

© Copyright 2014 Intergraph Corporation


EL
N
E

LAB-6: Rotate Grid planes – U10 CS (Optional)

Objectives

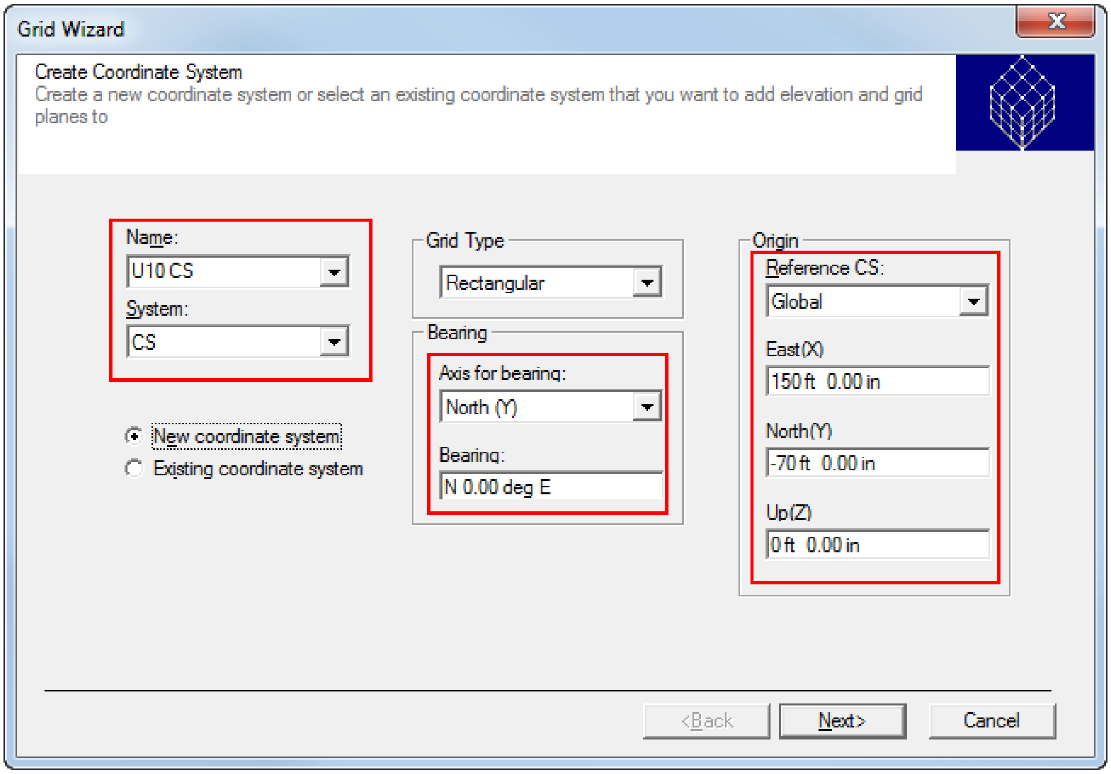
After completing this lab, you will be able to:

- Rotate the grid plane from different elevation planes.
- Rotate the grid plane using various rotation offset types.
- Rotate the grid planes by providing slope values about the axis of rotation.

1. Open or create a session file and define an “U10 & U10 CS” for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command from the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:	U10 CS
Axis for bearing:	North (Y)
Bearing:	N 0.00 deg E
Reference CS:	Global
Origin:	
East (X):	150 ft
North (Y):	-70 ft
Up (Z):	0 ft

© Copyright 2014 Intergraph Corporation



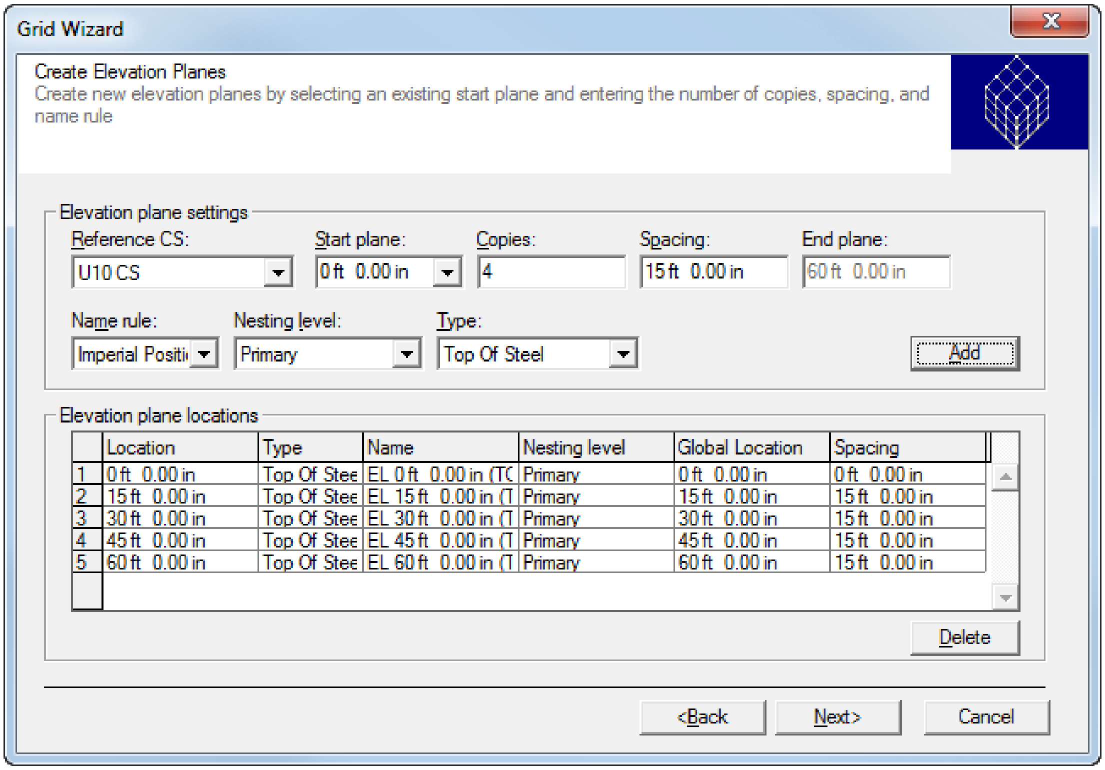
Next, click "Next" button

5. **Elevation Planes**: Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

Set the Nesting level to Primary for All Elevation Planes.

Step	Start Plane	Copies	Spacing	Name Rule	End Plane	Type
1	0 ft 0 in	4	15 ft 0 in	Imperial Position	60 ft 0 in	Top of Steel

Select the "Add" button.

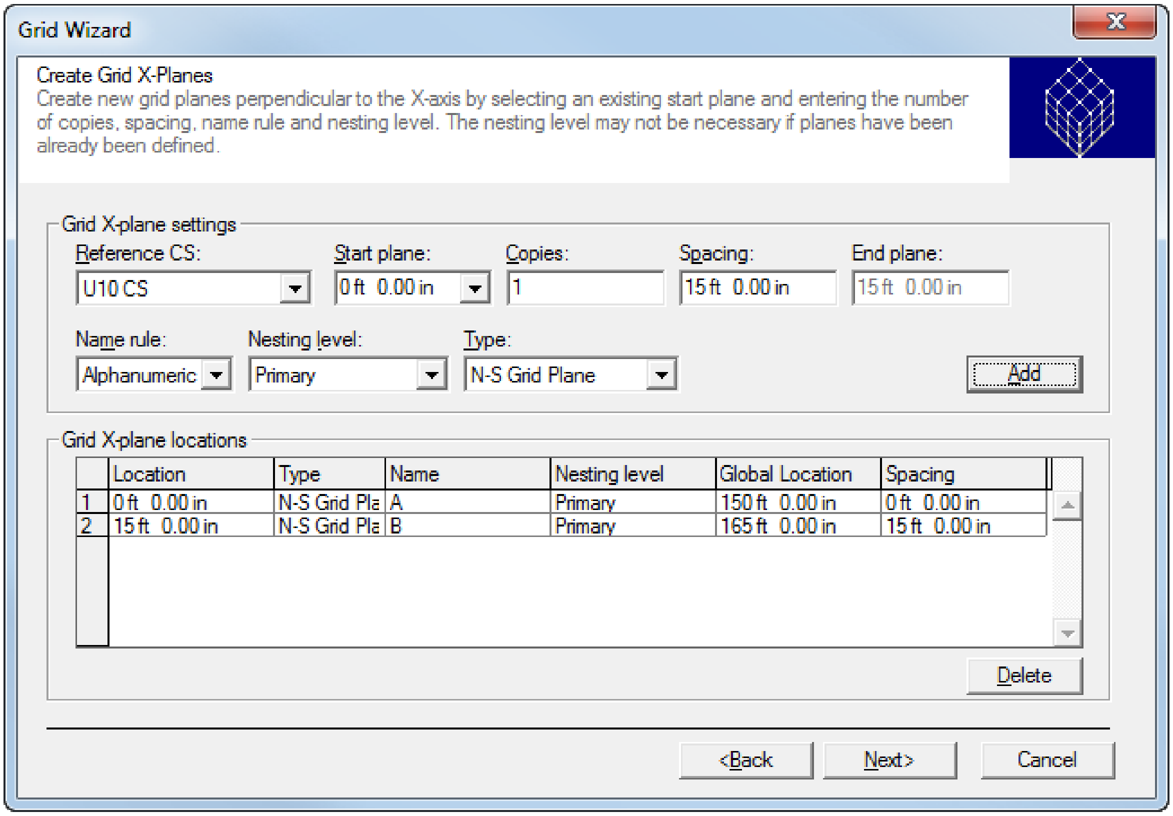


Next, click “Next” button.

6. **<u>Grid X-Planes</u>:** Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:
   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U10 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	Nesting Level	End Plane	Type
1	0 ft 0 in	1	15 ft 0 in	Primary	15 ft 0 in	N-S Grid Plane

Select the “Add” button.



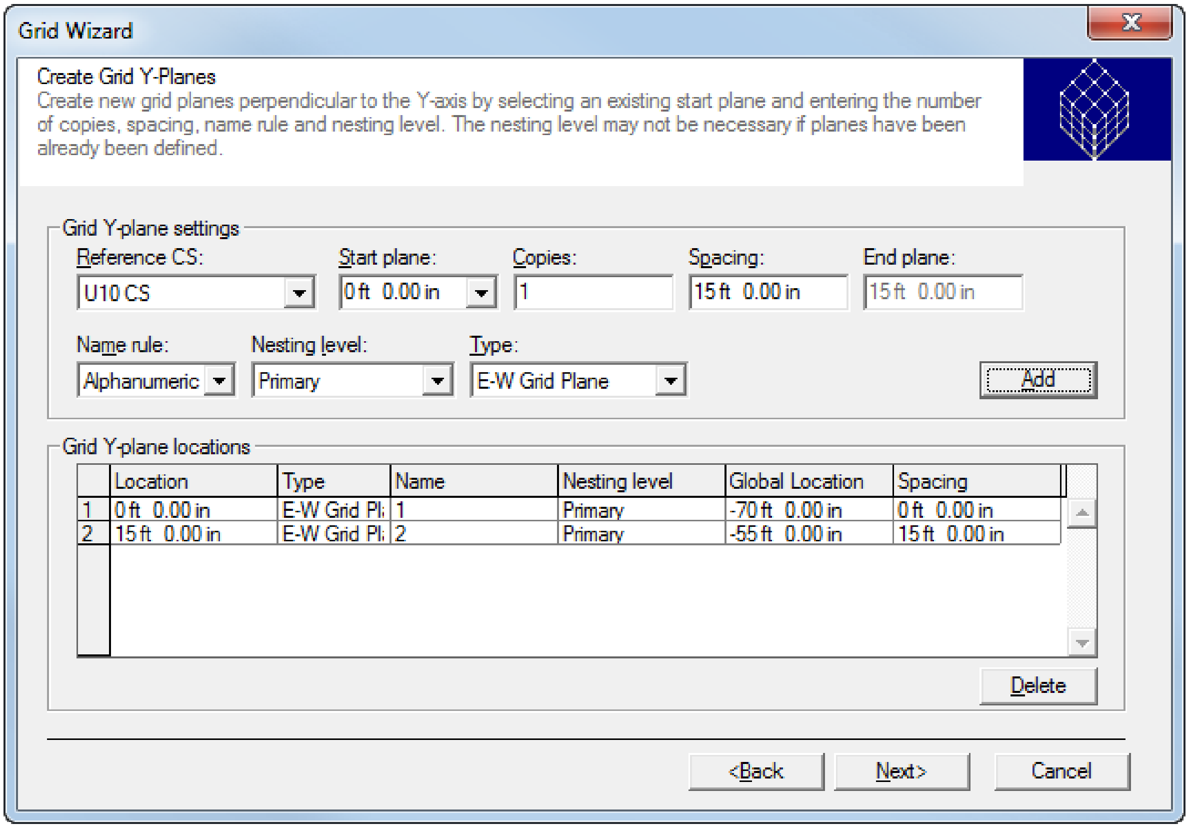
Next, click “Next” button.

7. **Grid-Y Planes**: Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U10 CS in the Reference CS pull down menu.

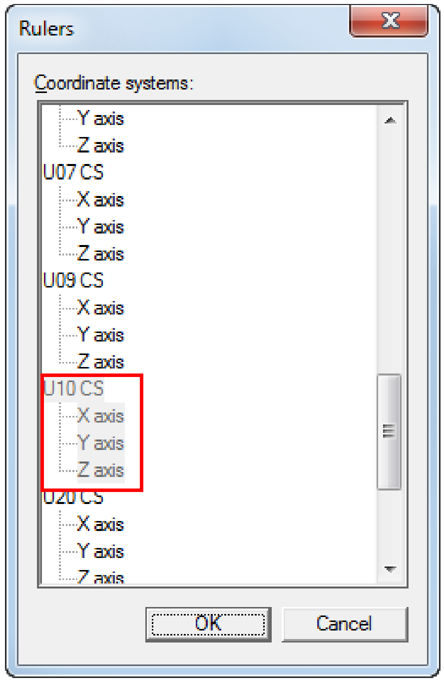
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	15 ft 0 in	15 ft 0 in	Primary	E-W Grid Plane

   Select the “Add” button.

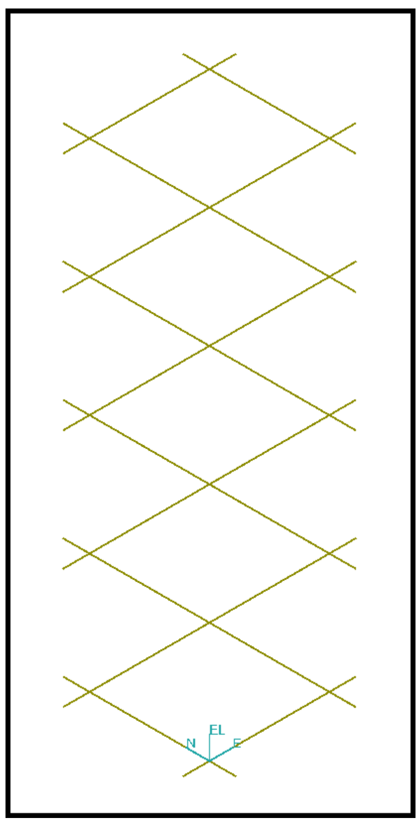


Next, click “Next” button.

8. Hit “Finish” button on Associated Elevation Plane form. The wizard will create the grids lines at all elevations.
9. Use the Fit command from the main toolbar to fit all graphics into the view.
10. Use the Common View control to look at an Isometric view.
11. Go to the main menu and select View -> Ruler option to open the ruler dialog box. Hold the <*Control*> key down and de-highlight the coordinate system rulers called U10 CS. Select the “OK” button.

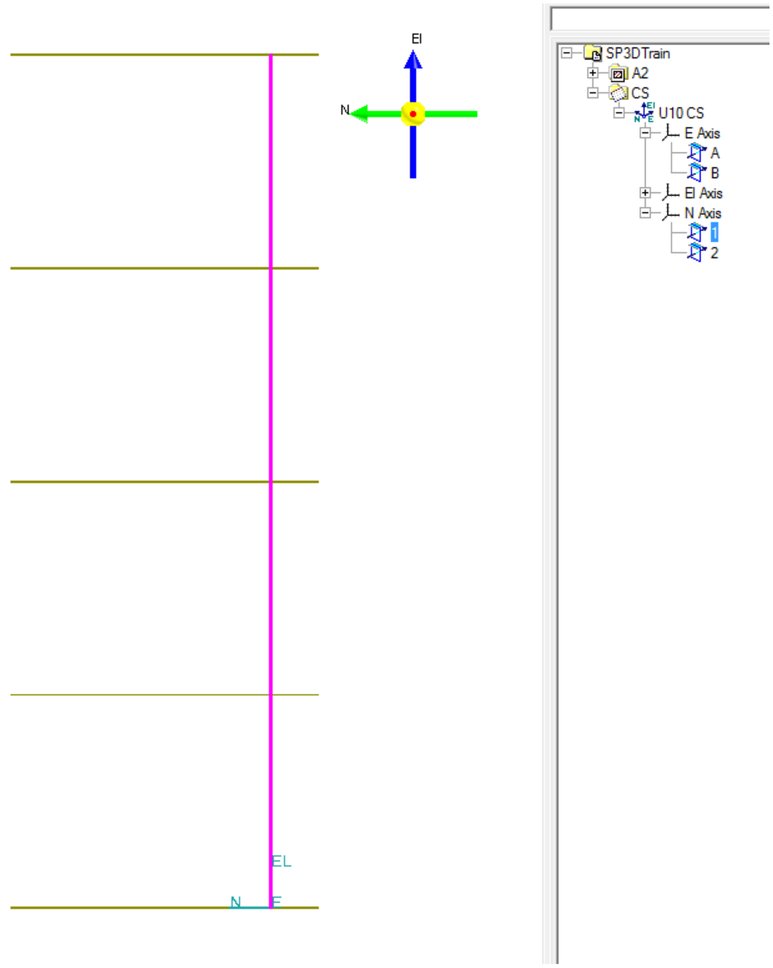


12. Your View should now resemble the following graphic.

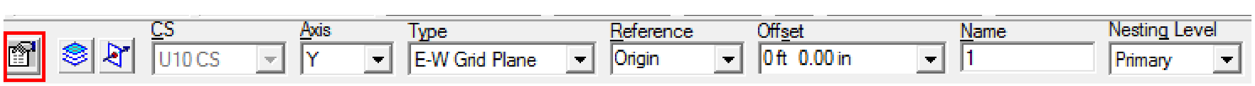


Angle Method

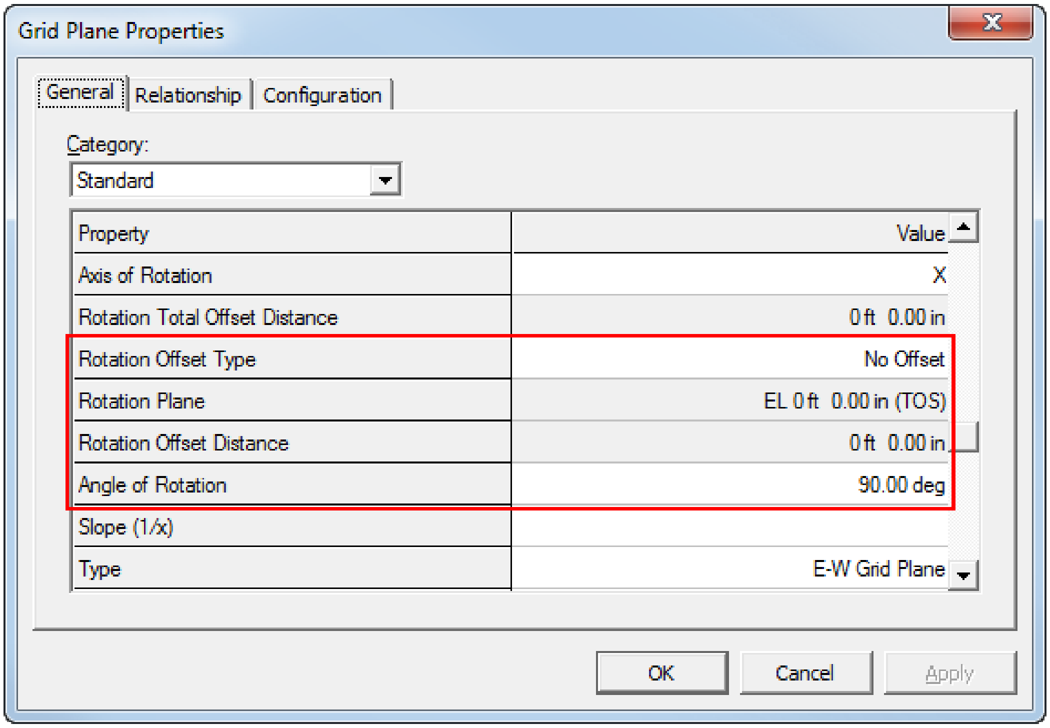
13. Expand the U10 CS in the workspace explorer. Expand the N-Axis and select "1" grid plane.
14. Use the Common View control for the Looking East view. Your view should resemble the following graphic.



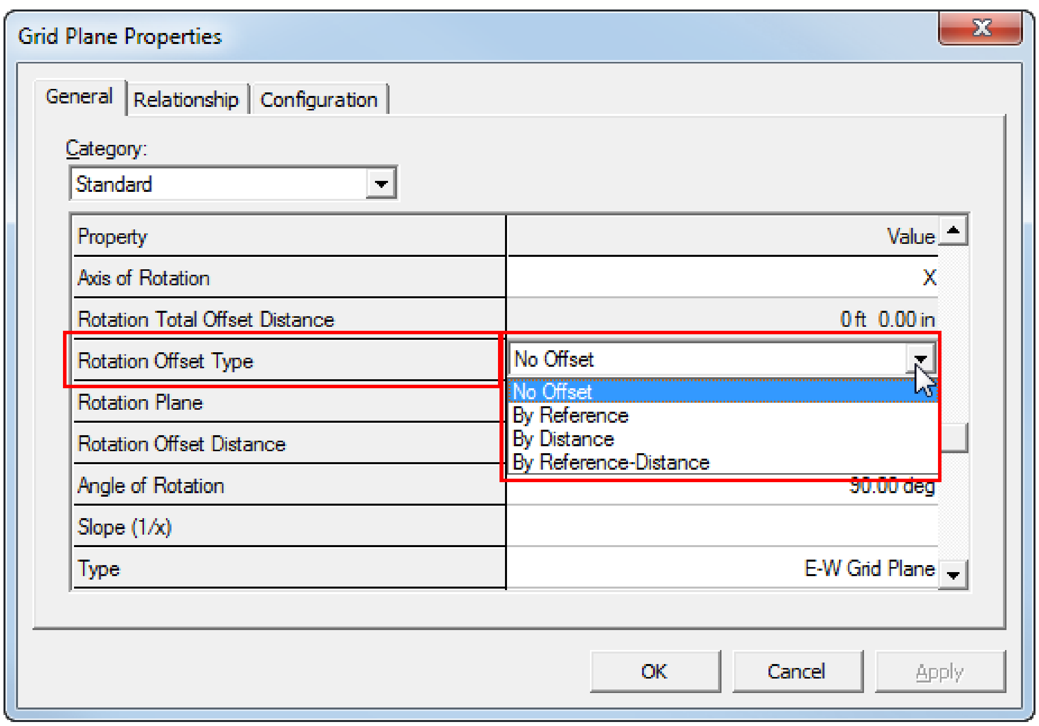
15. Click on the properties option provided on the ribbon bar,



16. Properties dialog box will appear.
17. Scroll down to view the rotation offset type / rotation plane / rotation offset distance options.



18. There are four rotation offset type options as shown in the image below.

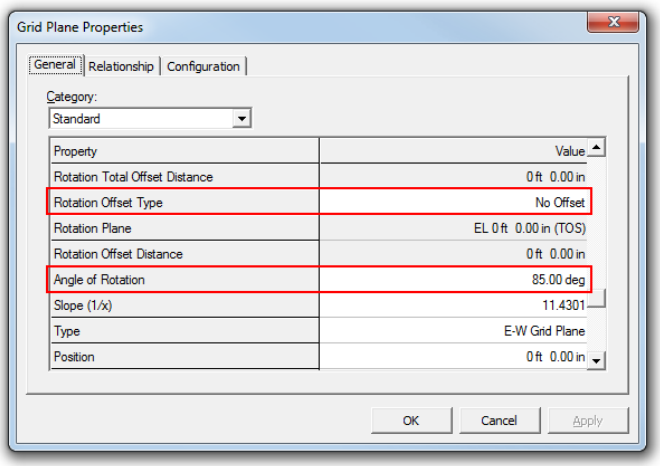


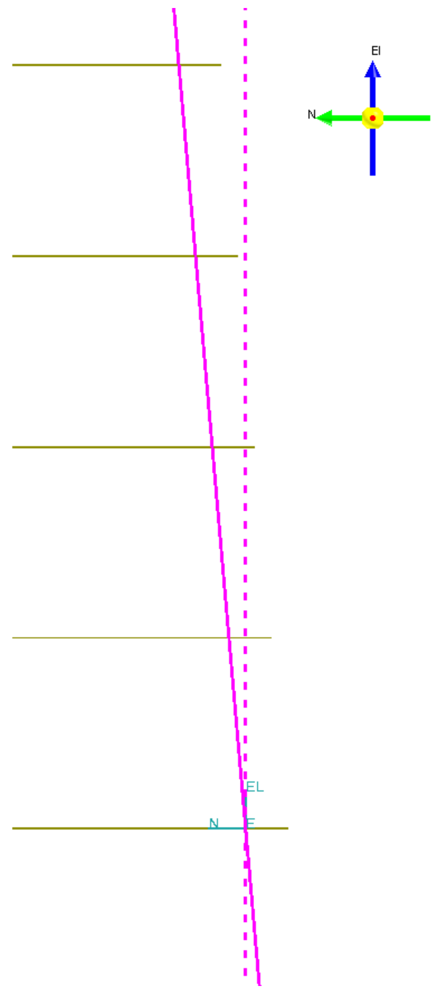
Notes:
No Offset *- Select this option to rotate the grid plane about the base elevation plane.*
By Reference *- Select this option to specify an elevation plane to use as the rotation axis.*

© Copyright 2014 Intergraph Corporation

*You select the elevation plane using the **Rotation Plane** box.*
***By Distance** - Select this option to specify the axis of rotation location relative to the base elevation plane. Type the distance in the **Rotation Offset Distance** box.*
***By Reference-Distance** - Select this option to specify an elevation plane and an offset from that plane to use as the rotation axis. You select the elevation plane using the **Rotation Plane** box, and define the offset distance using the **Rotation Offset Distance** box.*

19. Select the “No Offset” option. This will rotate the grid plane along the elevation plane. Currently the elevation plane is EL 0ft 0.00in (TOS) and key in Angle of rotation as 85.00 deg. Please see the image below,



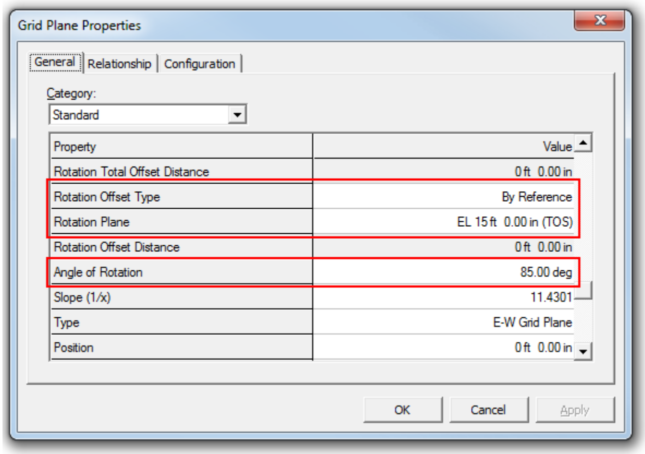


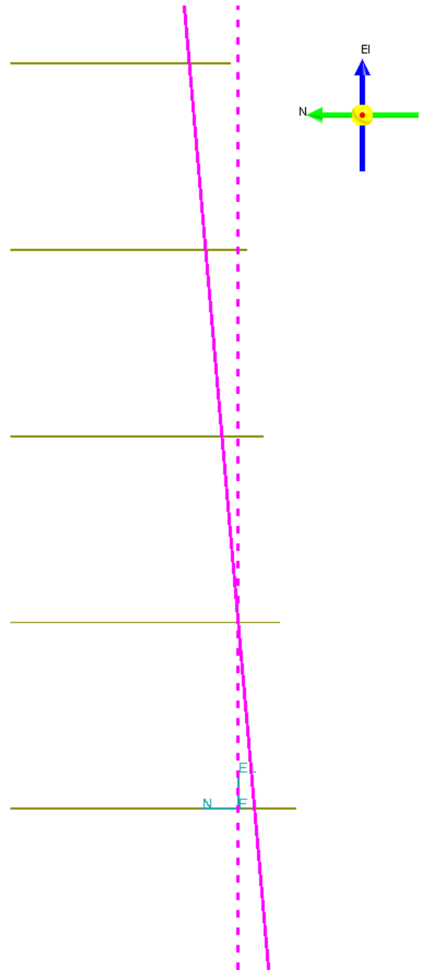
20. Click Apply.

21. Click Cancel on the grid plane properties window.
22. Go to Edit → Undo Edit Properties.
23. Select the “1” grid plane under the N-Axis in the workspace explorer and click the properties window on the toolbar.

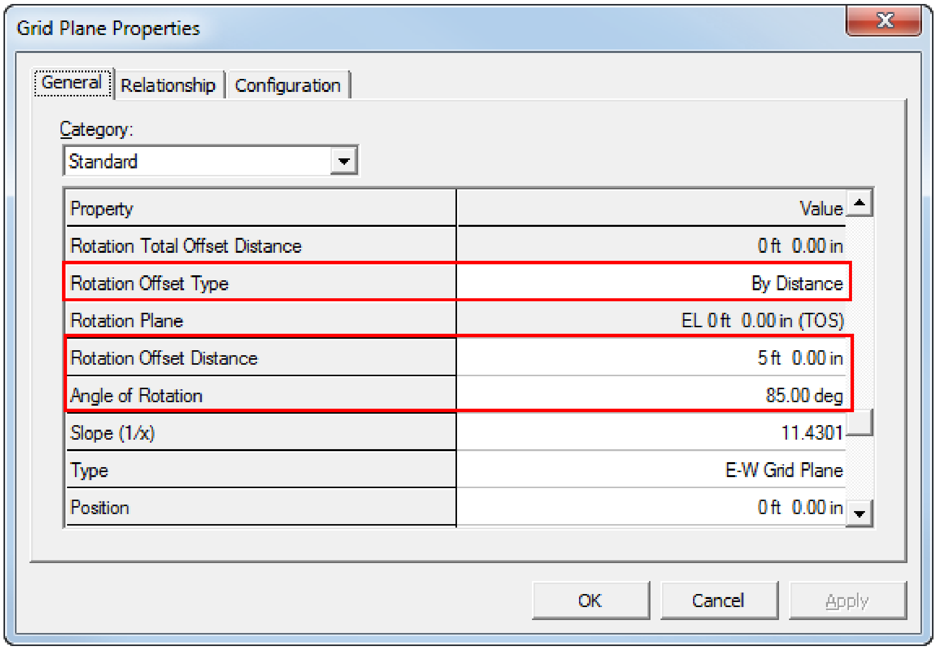
© Copyright 2014 Intergraph Corporation

24. Select the “By Reference” option. This will rotate the grid plane along the elevation plane. Select the elevation plane EL 15ft 0.00in (TOS) from the rotation plane pull down menu and key in Angle of rotation as 85.00 deg. Click Apply.
25. Your view should resemble the following graphic,

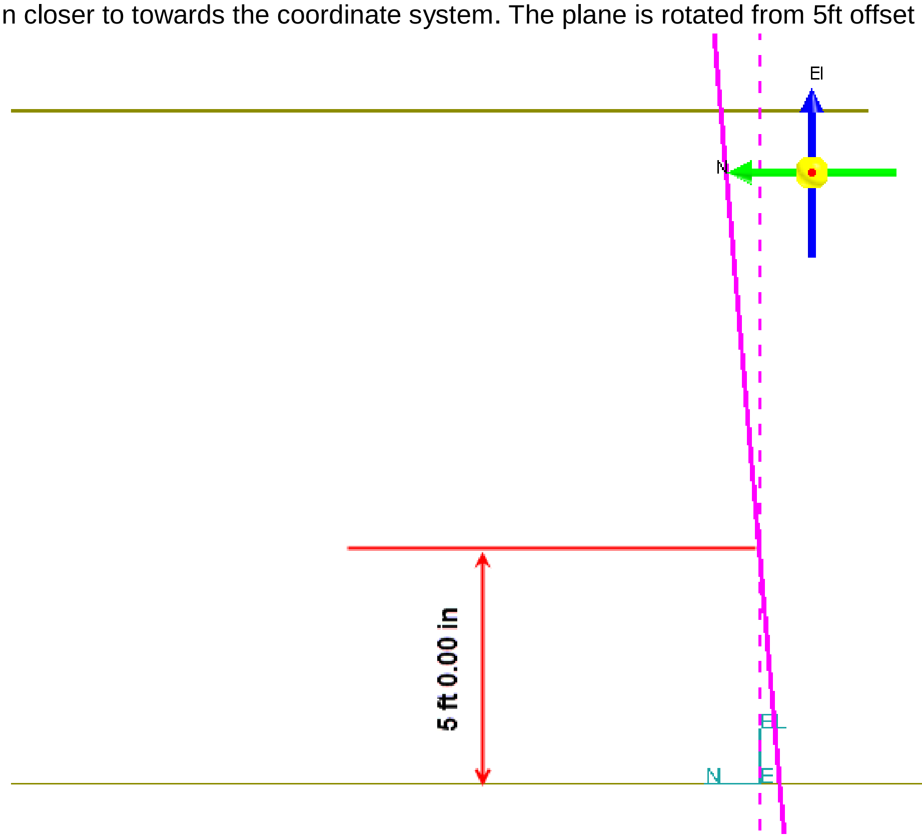




26. Click Cancel on the grid plane properties window.
27. Go to Edit → Undo Edit Properties.
28. Select the “1” grid plane under the N-Axis in the workspace explorer and click the properties window on the toolbar.
29. Select the “By Distance” option. Key in Rotation Offset Distance as 5 ft 0.00in. Key in Angle of rotation as 85.00 deg. Please see the image below,

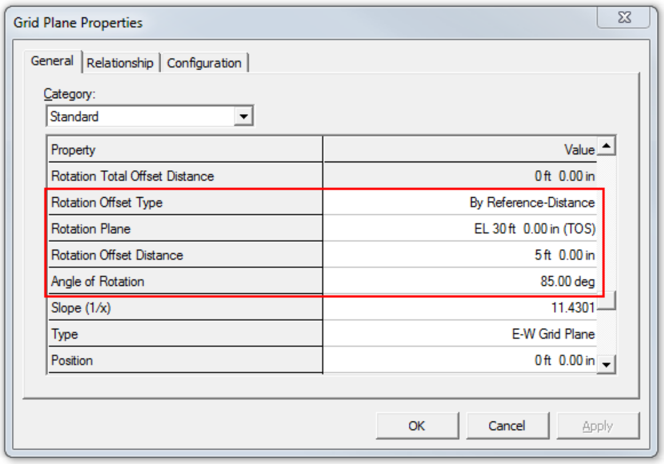


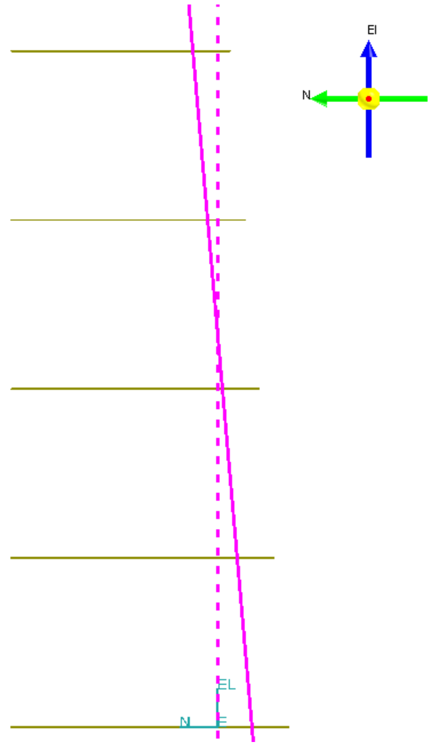
30. Click Apply.
31. Zoom in closer to towards the coordinate system. The plane is rotated from 5ft offset distance.



32. Click Cancel on the grid plane properties window.
33. Go to Edit → Undo Edit Properties.

34. Select the "1" grid plane under the N-Axis in the workspace explorer and click the properties window on the toolbar.
35. Select the "By Reference-Distance" option. Select in 30ft 0.00in (TOS) and key in 5ft as offset value and 85 deg as angle of rotation and click Apply,

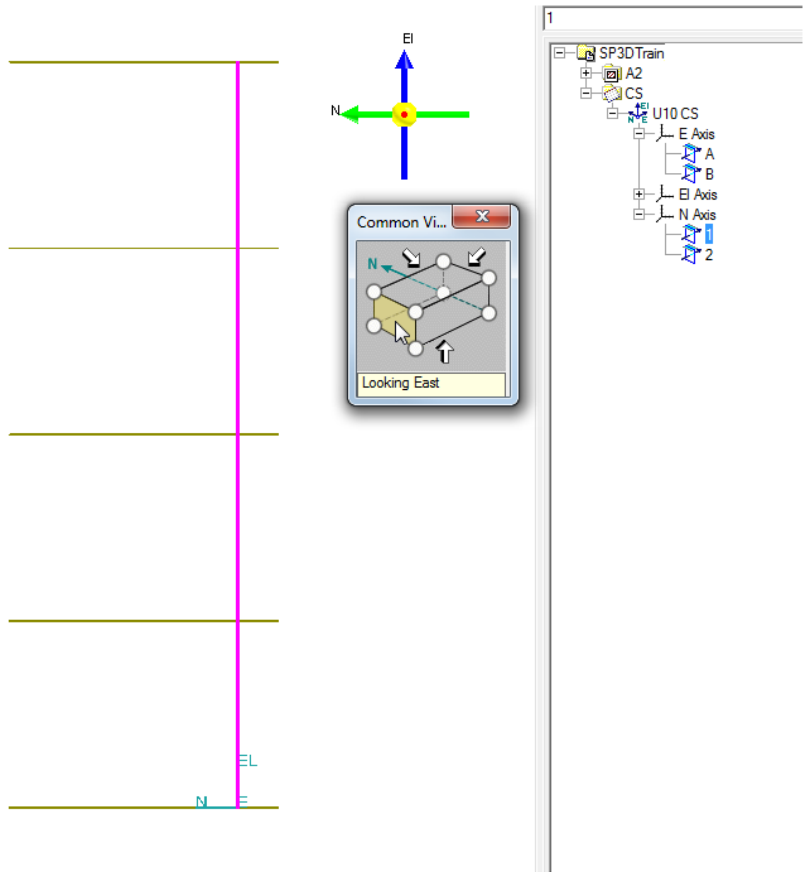




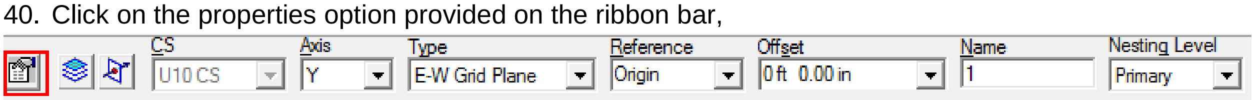
36. Zoom in closer to towards the coordinate system. The plane is rotated from 5ft offset distance from elevation plane 30ft 0.00in (TOS).
37. Go to Edit → Undo Edit Properties.

Slope Method

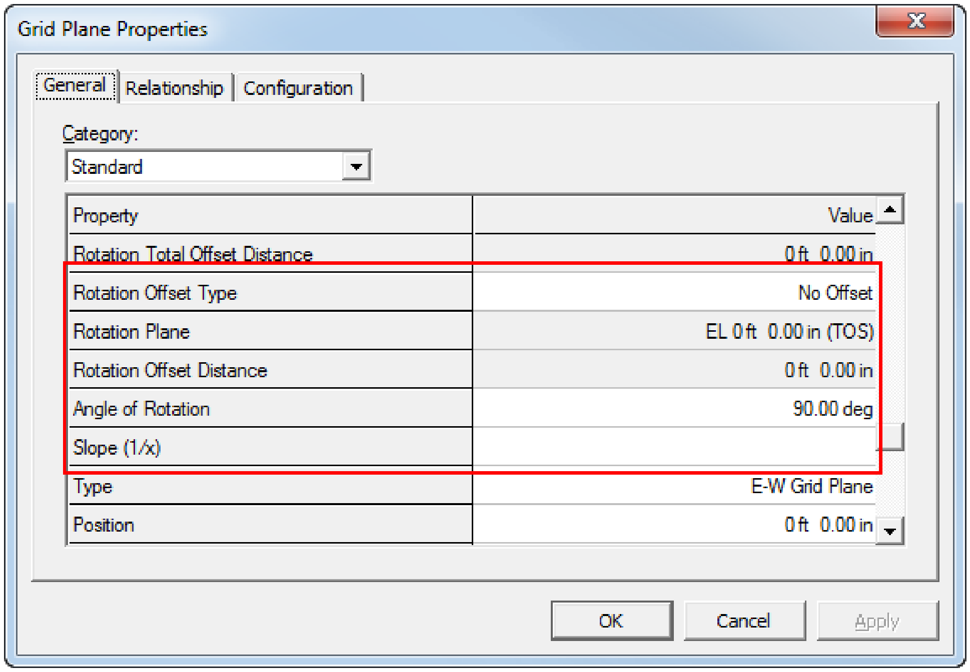
38. Expand the U10 CS in the workspace explorer. Expand the N-Axis and select "1" grid plane.
39. Use the Common View control for the Looking East view. Your view should resemble the following graphic.



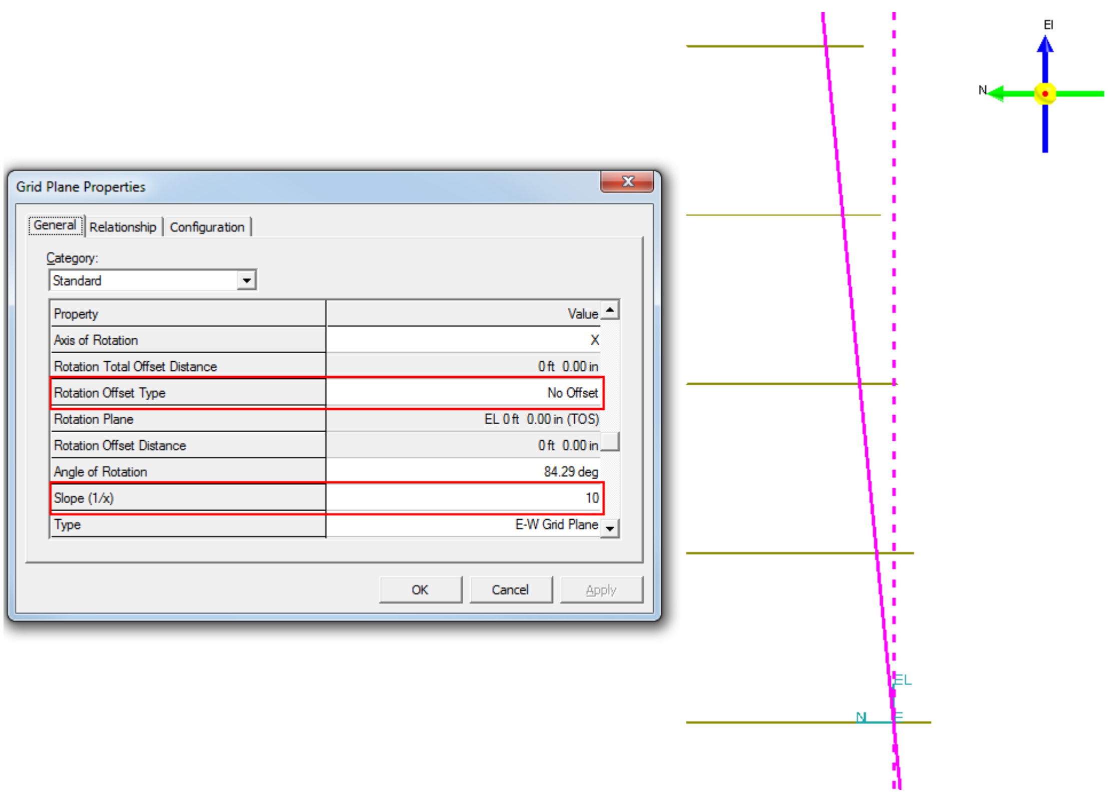
40. Click on the properties option provided on the ribbon bar,



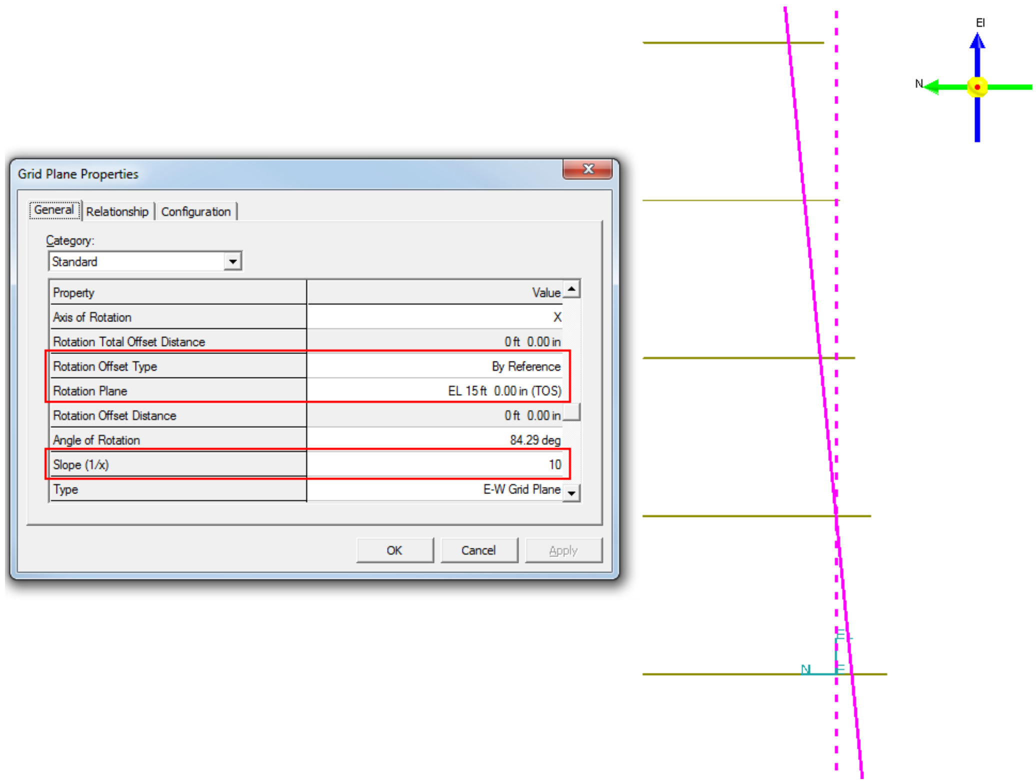
41. Properties dialog box will appear. Scroll down to view the rotation offset type / rotation plane / rotation offset distance and slope options. See the image below,



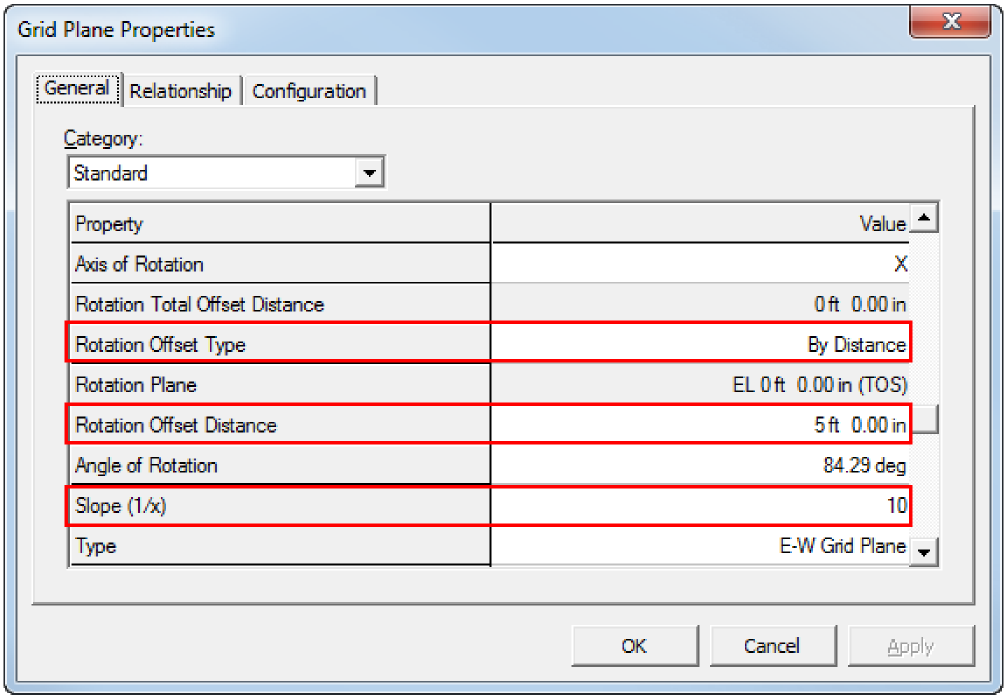
42. Select the “No Offset” option. This will rotate the grid plane along the elevation plane. Currently the elevation plane is EL 0ft 0.00in (TOS) and key in slope values as 10 and click Apply. Your view should resemble the following graphic,



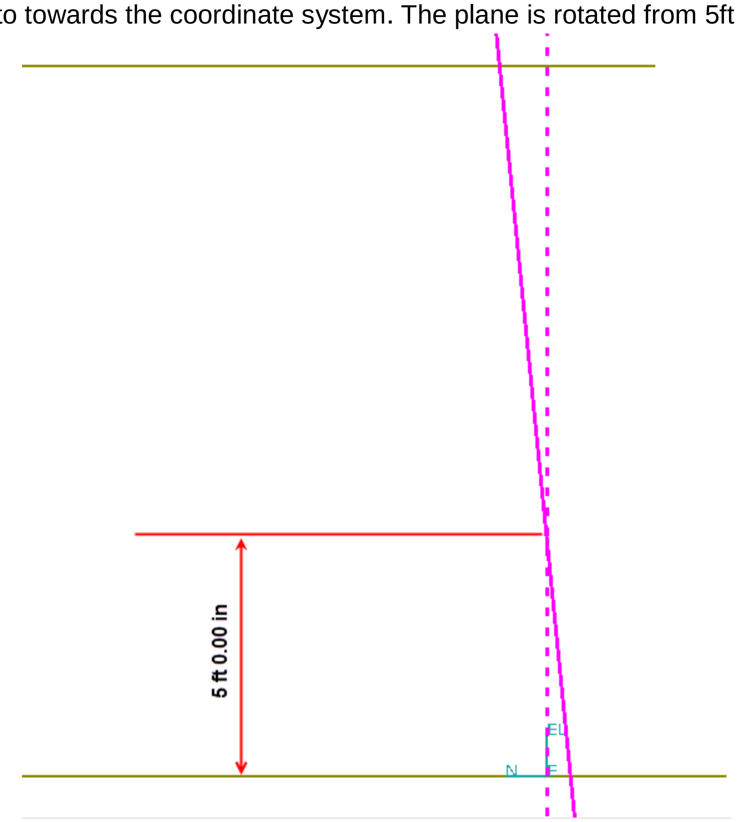
43. Click Cancel on the grid plane properties window.
44. Go to Edit → Undo Edit Properties.
45. Select the “1” grid plane under the N-Axis in the workspace explorer and click the properties window on the toolbar.
46. Select the “By Reference” option. Select the elevation plane EL 15ft 0.00in (TOS) from the rotation plane pull down menu and key in 10 as the slope value and click Apply. Your view should resemble the following graphic.



47. Click Cancel on the grid plane properties window.
48. Go to Edit → Undo Edit Properties.
49. Select the “1” grid plane under the N-Axis in the workspace explorer and click the properties window on the toolbar.
50. Select the “By Distance” option. Key in Rotation Offset Distance as 5 ft 0.00in. Key in 10 as the slope value. Please see the image below,

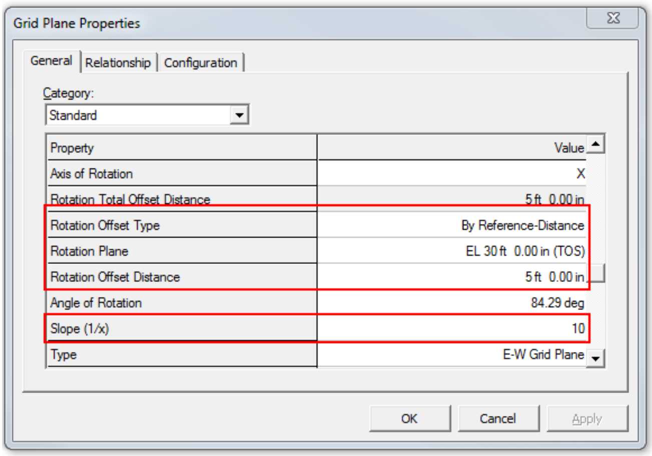


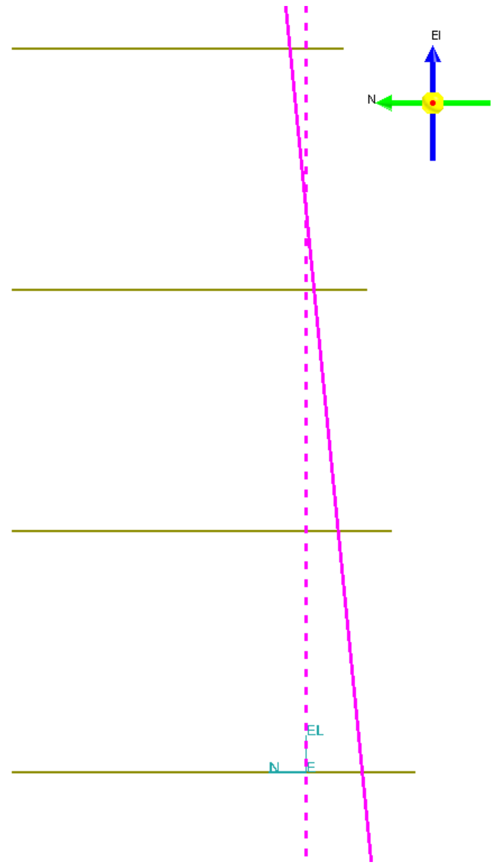
51. Click Apply.
52. Zoom in closer to towards the coordinate system. The plane is rotated from 5ft offset distance.



53. Click Cancel on the grid plane properties window.
54. Go to Edit → Undo Edit Properties.

55. Select the “1” grid plane under the N-Axis in the workspace explorer and click the properties window on the toolbar.
56. Select the “By Reference-Distance” option. Select in 30ft 0.00in (TOS) and key in 5ft as offset value and 10 as slope value and click Apply,





57. Zoom in closer to towards the coordinate system. The plane is rotated from 5ft offset distance from elevation plane 30ft 0.00in (TOS).
58. Go to Edit → Undo Edit Properties.

© Copyright 2014 Intergraph Corporation

LAB-7: Grid line Extensions

Objectives

After completing this lab, you will be able to:

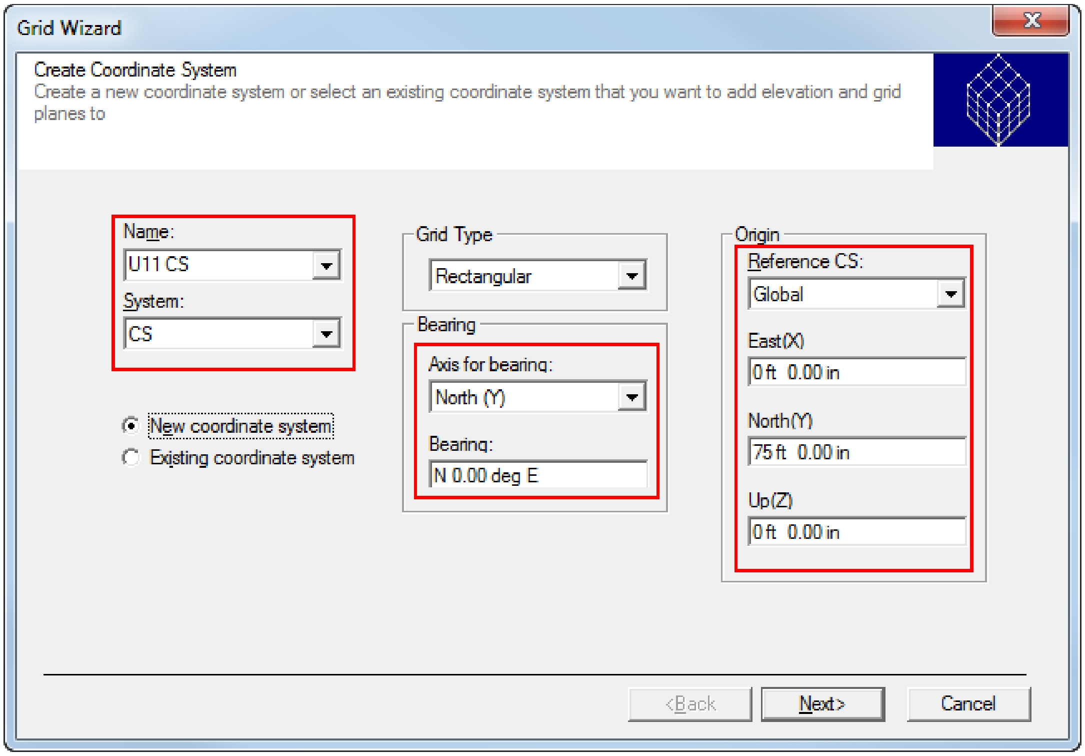
- Ability to specify grid line extensions.

1. Open or create a session file and define an “U11 & U11 CS” filter for your workspace.
2. Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
3. Select the Grid Wizard command from the vertical toolbar.
4. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name: U11 CS
Axis for bearing: North (Y)
Bearing: N 0.00 deg E
Origin:

Reference CS: Global
East (X): 0 ft
North (Y): 75 ft
Up (Z): 0 ft
Next, click “Next” button

© Copyright 2014 Intergraph Corporation



5. **Elevation Planes:** Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

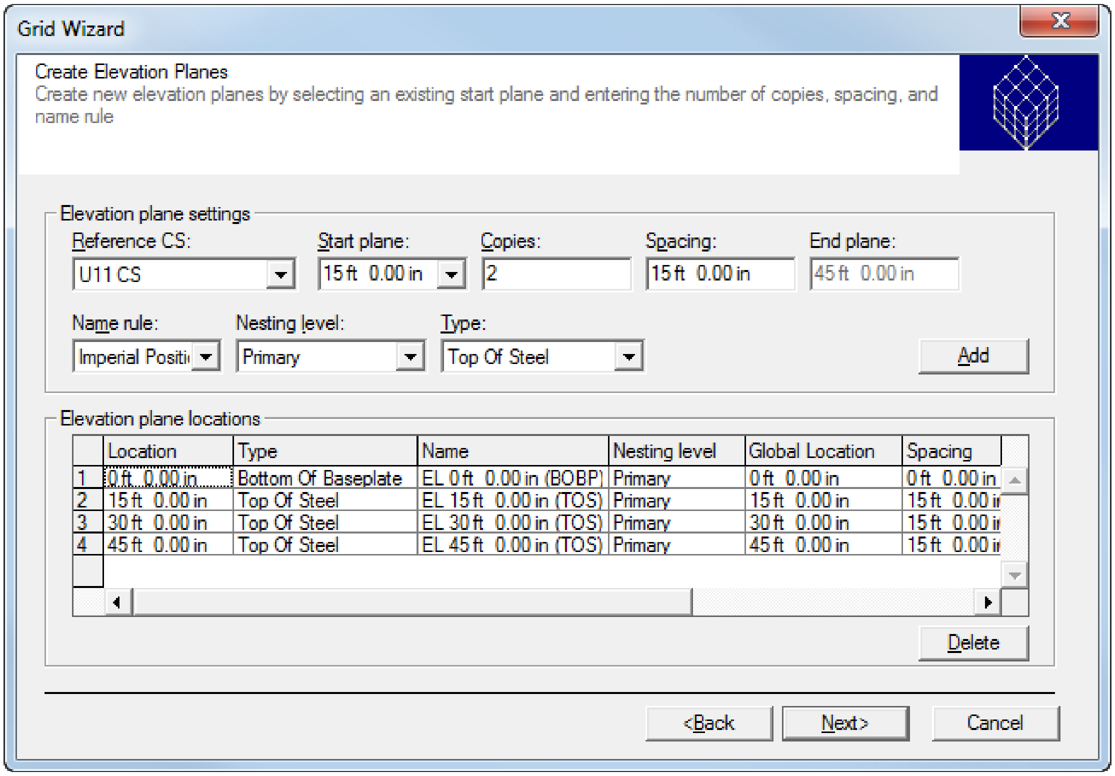
   Set the Name rule to Imperial Position for All Grid Planes.

   Set the Nesting level to Primary for All Elevation Planes.

Step	Start Plane	Copies	Spacing	Name Rule	End Plane	Type
1	0 ft 0 in	0	Ignore	Imperial Position	0 ft 0 in	Bottom of Baseplate
2	15 ft 0 in	2	15 ft 0 in	Imperial Position	45 ft 0 in	Top of Steel

   Select the “Add” button.

© Copyright 2014 Intergraph Corporation

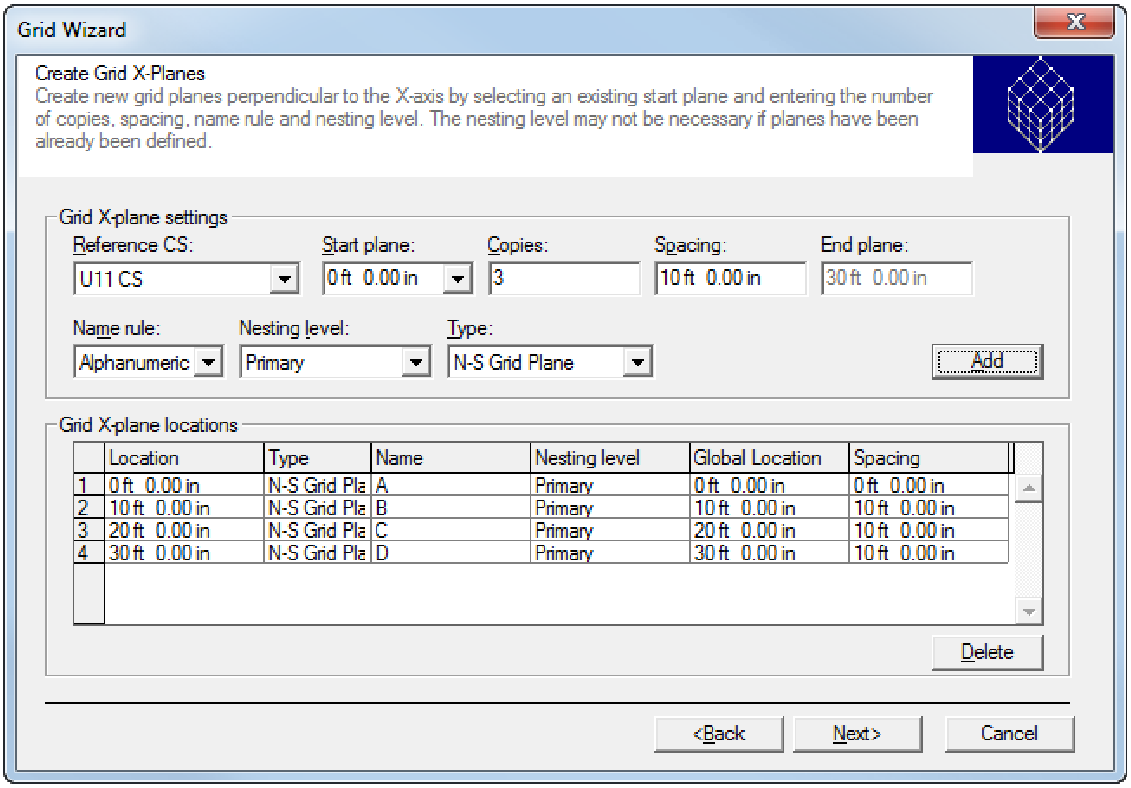


6. **Grid X-Planes**: Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information.
   Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U11 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	3	10 ft 0 in	30 ft 0 in	Primary	N-S Grid Plane

   Select the “Add” button.

© Copyright 2014 Intergraph Corporation



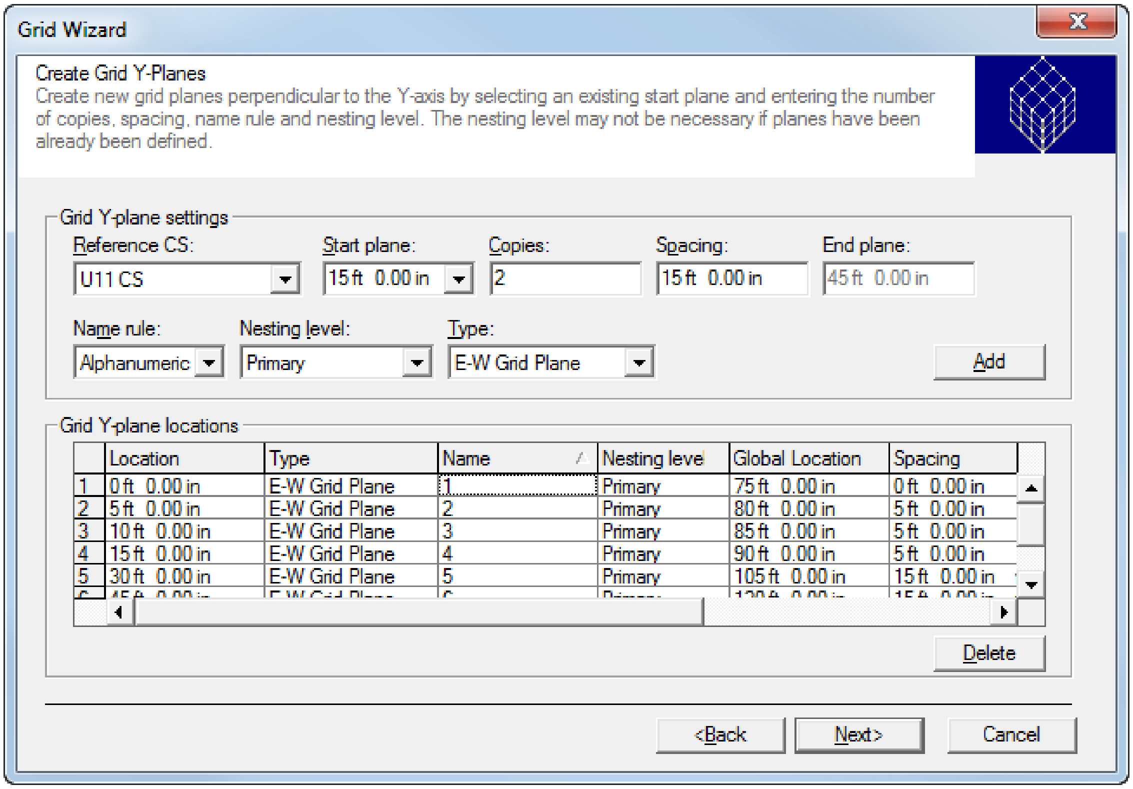
Next, click “Next” button.

7. **Grid-Y Planes**: Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:
Set the Name rule to Alphanumeric and Percent for All Grid Planes. Select U11 CS in the Reference CS pull down menu.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	3	5 ft 0 in	15 ft 0 in	Primary	E-W Grid Plane
2	15 ft 0 in	2	15 ft 0 in	45 ft 0 in	Primary	E-W Grid Plane

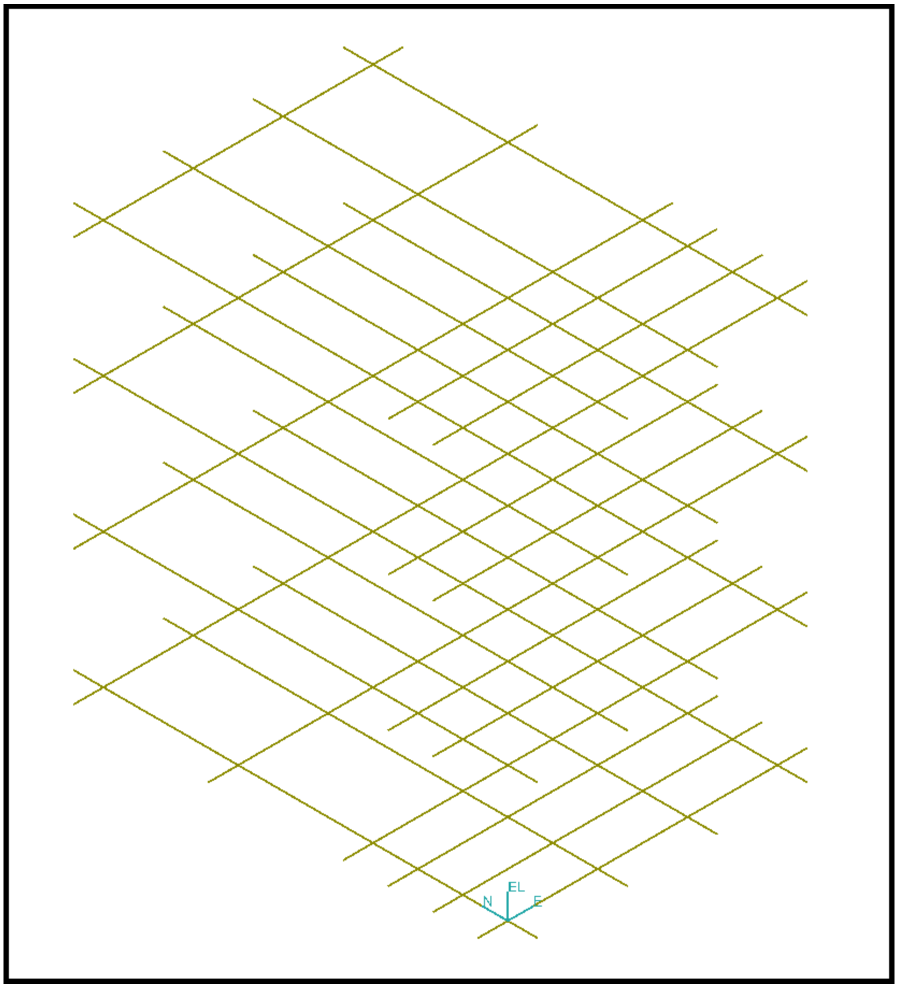
Select the “Add” button.

© Copyright 2014 Intergraph Corporation

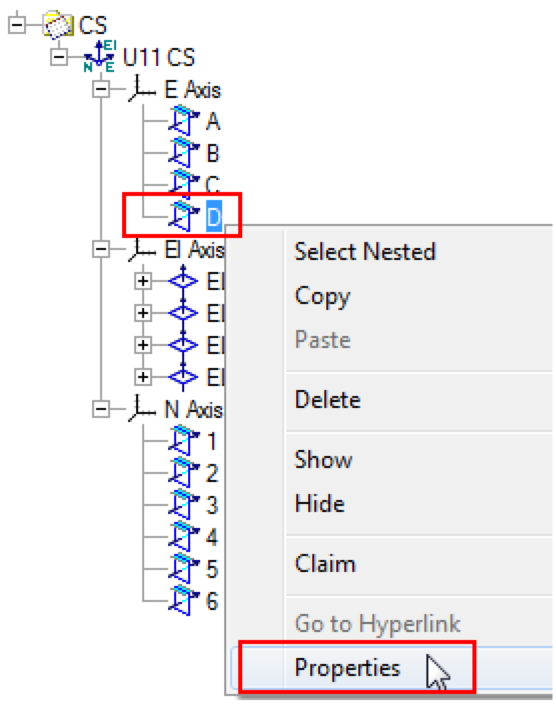


Next, click “Next” button and then click Finish.

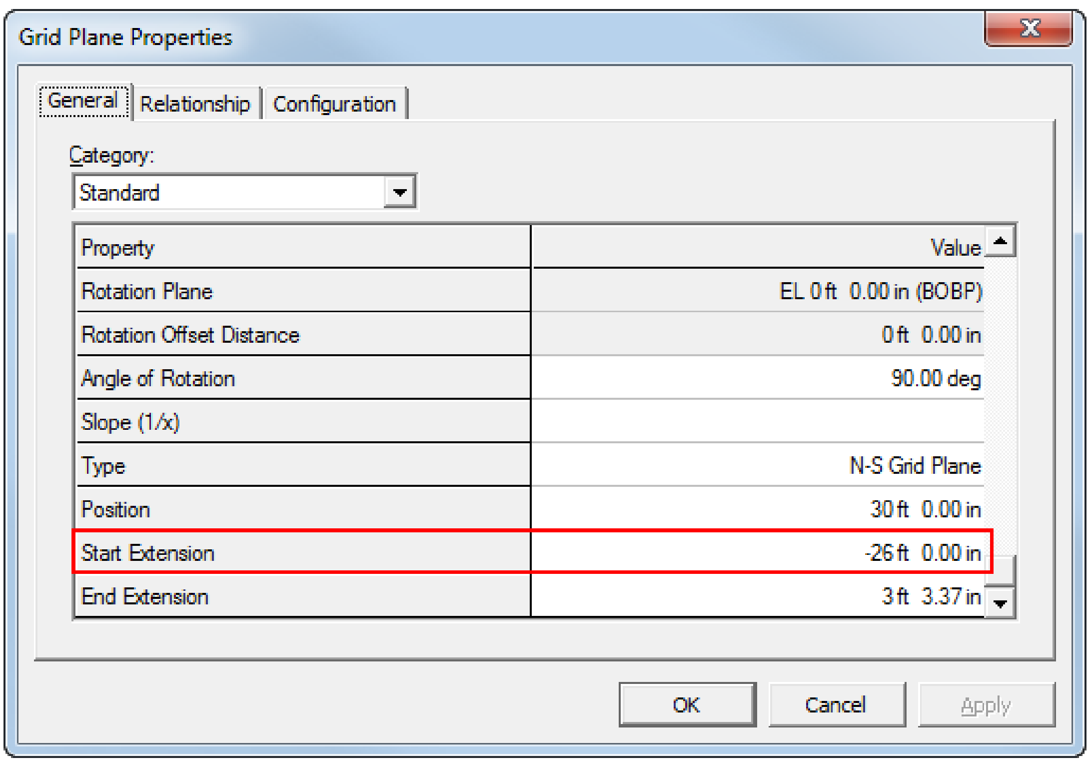
8. Use the Fit command from the main toolbar to fit all graphics into the view.
9. Go to the main menu and select View → Ruler option to open the ruler dialog box.
10. Press the <***Control***> key and unselect all items. Select the “OK” button to turn off the rulers.
11. Your View should now resemble the following graphic.



12. Set the locate filter to ALL and go to workspace explorer, select D plane from E-Axis, right click and select properties.

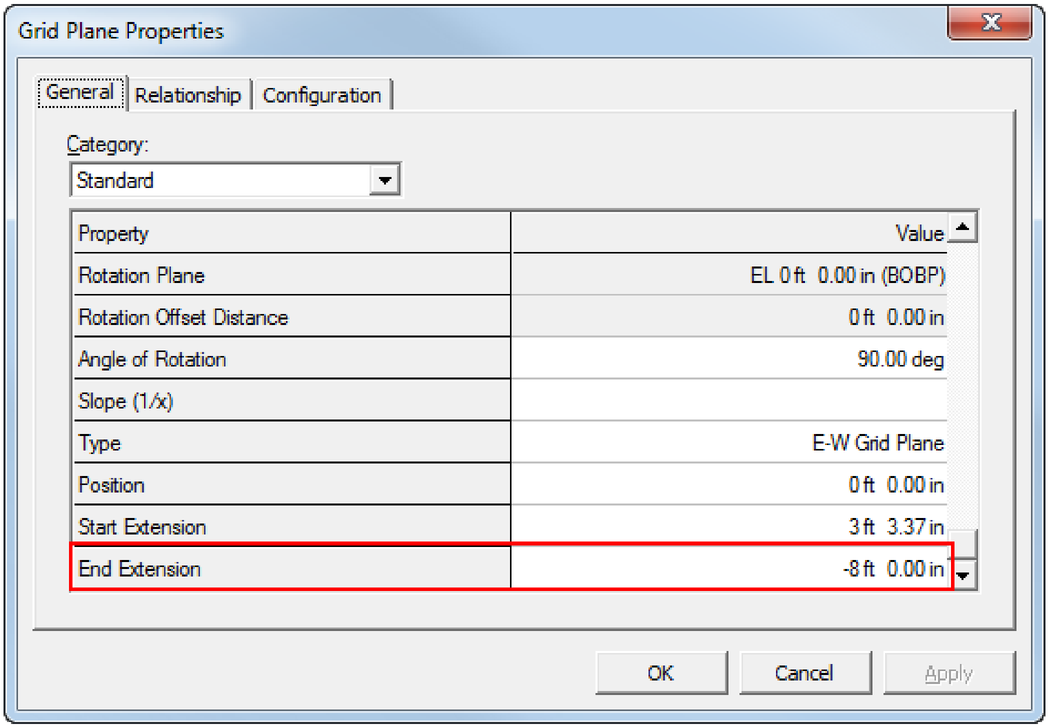


13. Scroll down and find the Start Extension value property and key in -26 ft 0.00 in as the new value and click Apply.

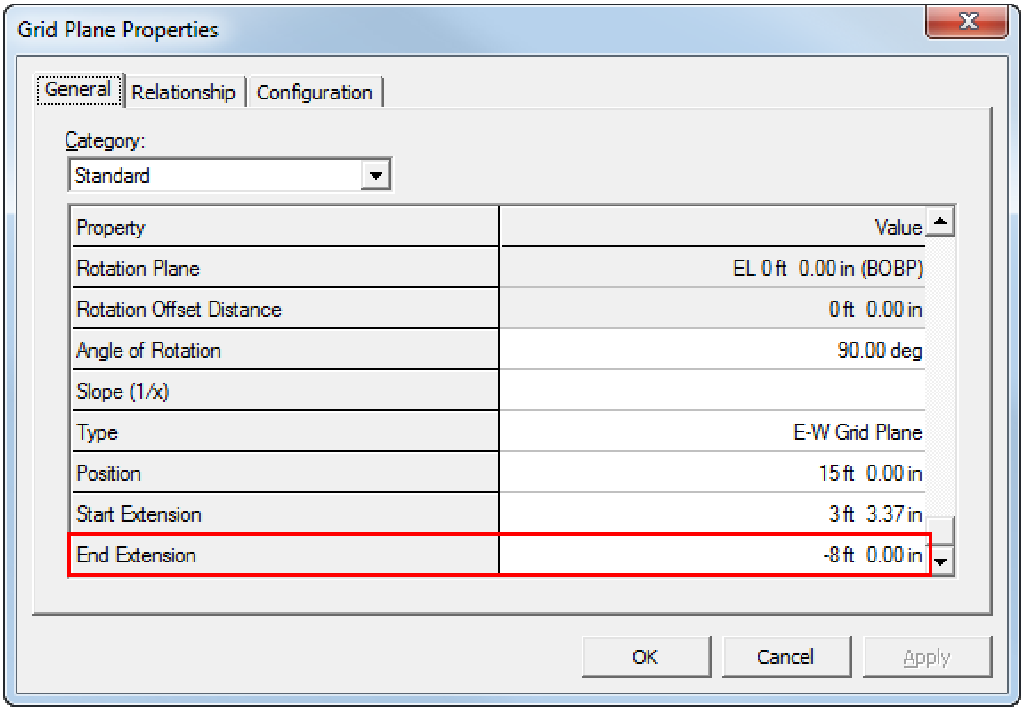


14. Go to workspace explorer, select 1 plane from N-Axis, right click and select properties.

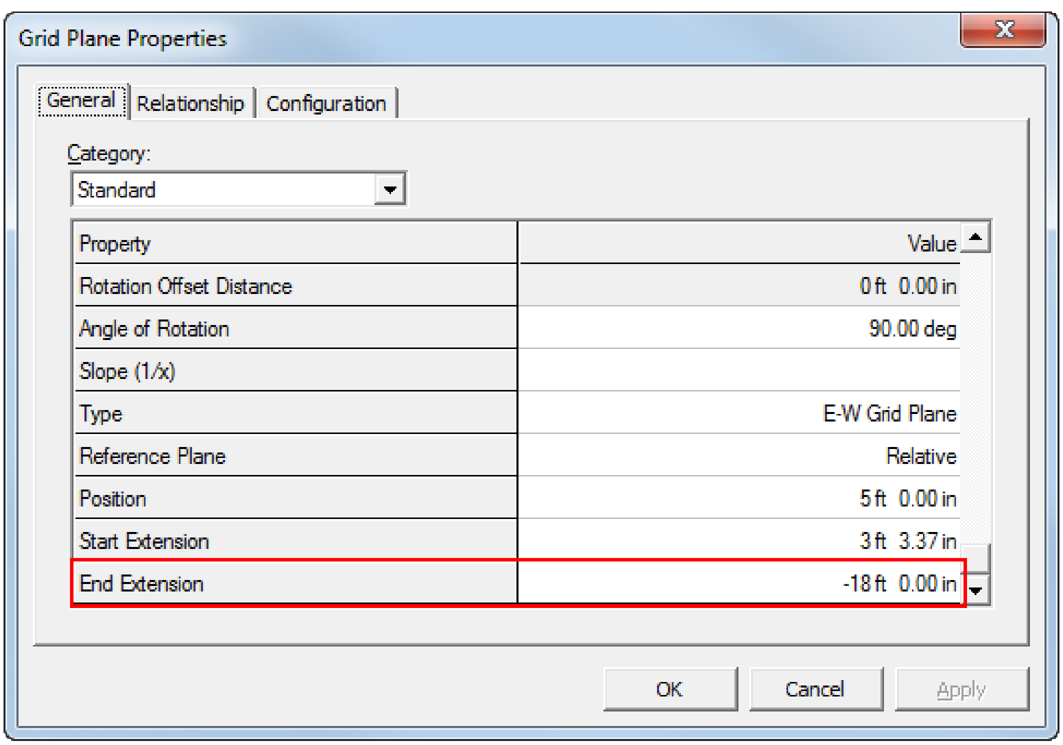
15. Scroll down and find the End Extension value property and key in -8 ft 0.00 in as the new value and click Apply.



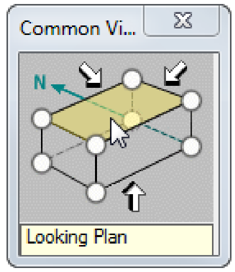
16. Go to workspace explorer, select 4 plane from N-Axis, right click and select properties.
17. Scroll down and find the End Extension value property and key in -8 ft 0.00 in as the new value and click Apply.



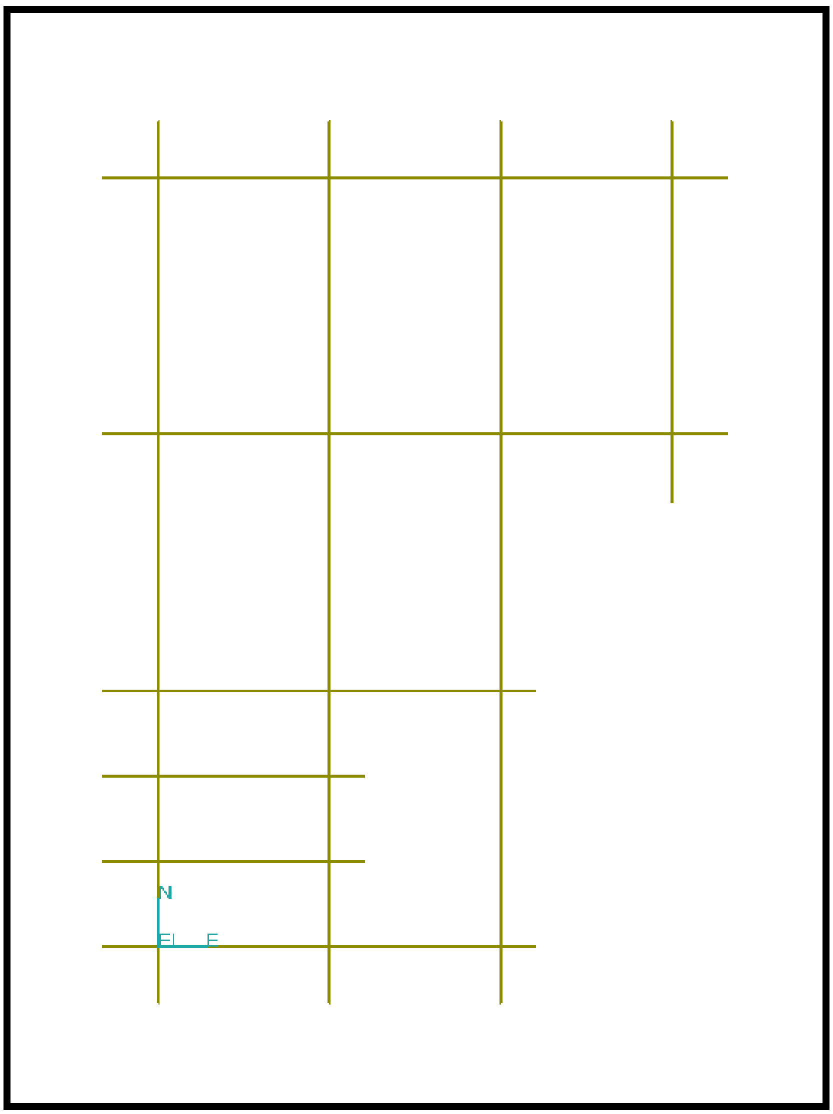
18. Go to workspace explorer, select 2&3 plane from N-Axis using the *ctrl key*, right click and select properties.
19. Scroll down and find the End Extension value property and key in -18 ft 0.00 in as the new value and click Apply.



20. Click OK and switch to the Plan view.



21. Your View should now resemble the following graphic.



© Copyright 2014 Intergraph Corporation

LAB-8: Grids Export-Import

Objectives

After completing this lab, you will be able to:

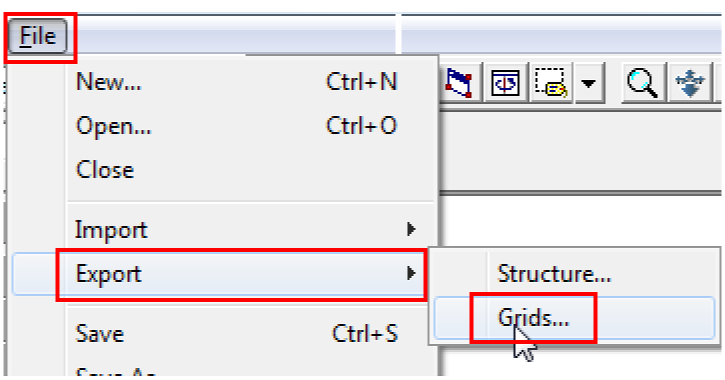
- Export grids as .XLS file.
- Import grids as .XLS file.

Overview

The File > Export > Grids command exports the coordinate system to a XLS or XML file. The exported file contains all needed information to import the coordinate system into a different model. All the distance and angle values are exported with the Units and Precision as set in Tools → Options... → Units of Measure window.

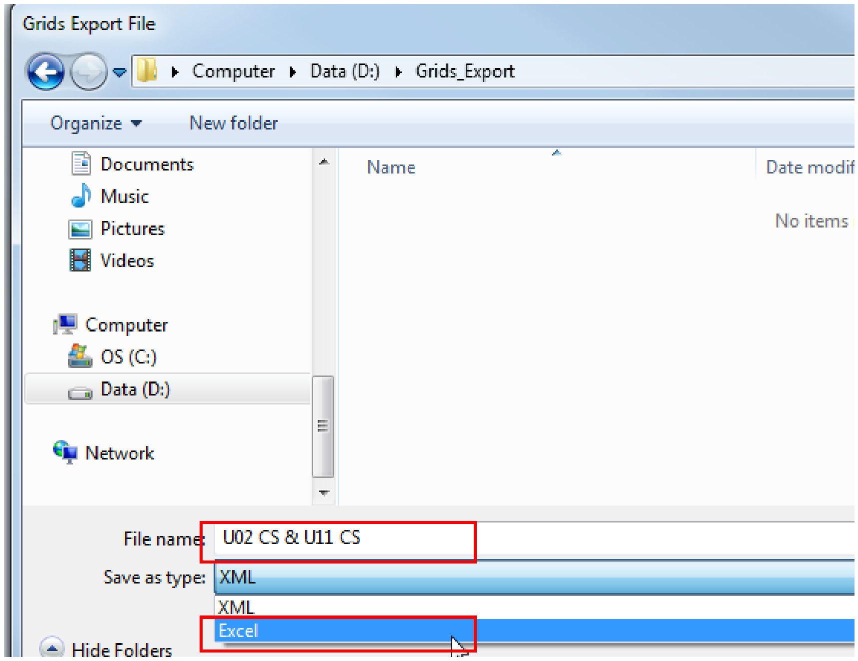
The File → Import → Grids command imports a coordinate system into the model. This command recognizes XML and XLS files that have been exported from a different model. The user can modify the distance and angles in the exported XML or XLS in any accepted Units of Measure formats, and then import. It is easier to modify the inputs in XLS than in XML. The user can utilize the AvailableUnitFormats sheet in the exported XLS file to understand the accepted Units of Measure formats.

1. Go to File →Export→Grids

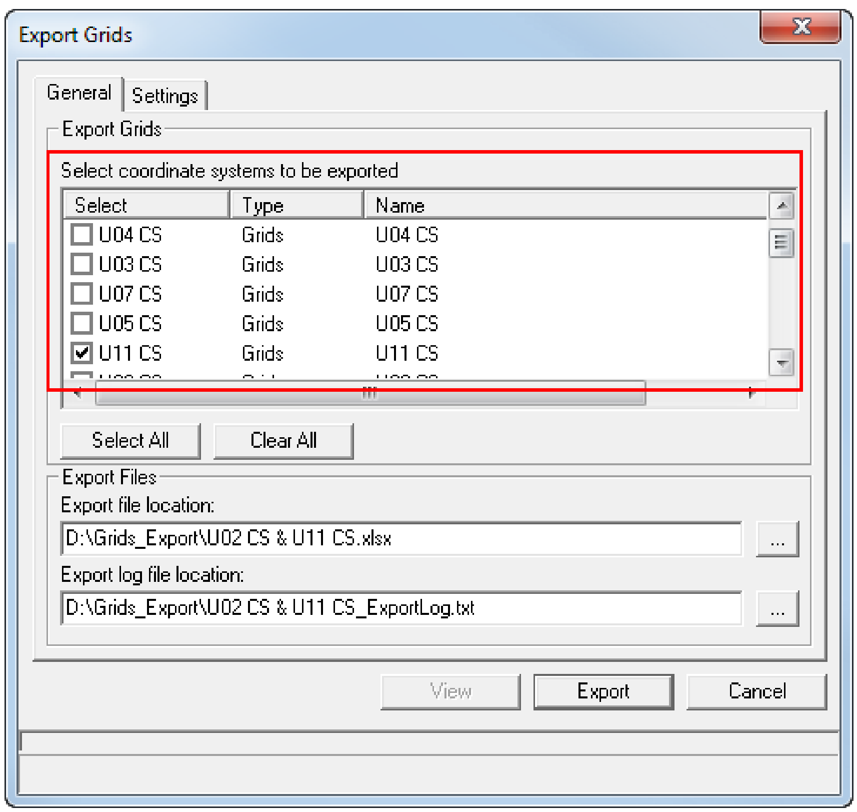


2. On the Export Grids window. Click on the browse button to specify the Export file location. Choose the “Save as type” option as Excel and provide the name as shown in the image below.

© Copyright 2014 Intergraph Corporation



3. Select U02 CS and U11 CS as shown in the image below.

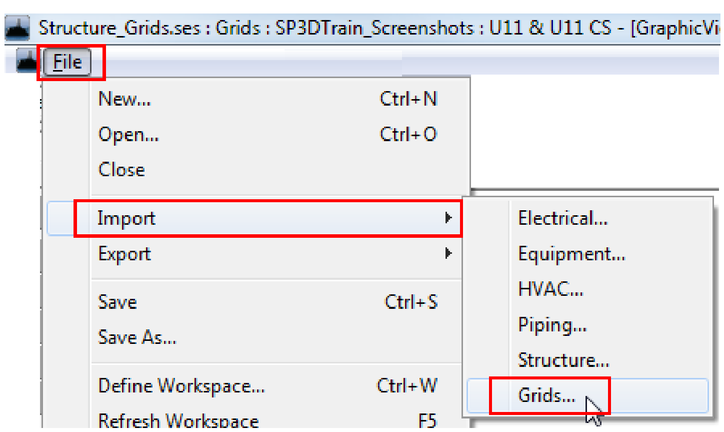


4. Your view should now represent as shown in the image below after opening the excel sheet.

© Copyright 2014 Intergraph Corporation

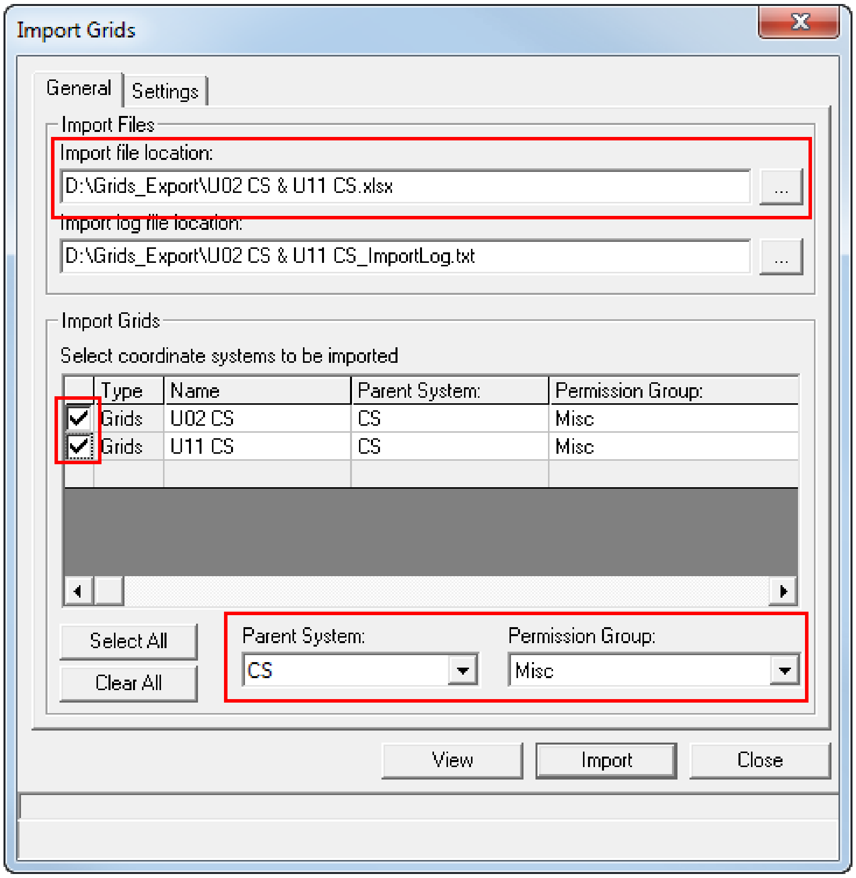
	A	B	C	D	E	F	G	H
1	Grid System	Name						
2		U02 CS						
3		Coordinate System						
4			System_Properties	Name	Type	System	Permission Group	GlobalX
5				U02 CS	Grids	CS	Misc	30 ft 0.00 in
6		Frames						
7			Elevation Planes	System_Properties	Name	Position	Axis Of Rotation	Name Rule
8					EL 0 ft 0.00 in (Grade)	0 ft 0.00 in		Imperial Position
9					EL 2 ft 1.00 in (BOBP)	2 ft 1.00 in		Imperial Position
10					EL 18 ft 0.00 in (TOS)	18 ft 0.00 in		Imperial Position
11					EL 30 ft 0.00 in (TOS)	30 ft 0.00 in		Imperial Position
12			X-Planes	System_Properties	Name	Position	Axis Of Rotation	Name Rule
13					A	0 ft 0.00 in	Y	Alphanumeric and Perce
14					B	20 ft 0.00 in	Y	Alphanumeric and Perce
15					C	40 ft 0.00 in	Y	Alphanumeric and Perce
16			Y-Planes	System_Properties	Name	Position	Axis Of Rotation	Name Rule
17					1	0 ft 0.00 in	X	Alphanumeric and Perce
18					2	15 ft 0.00 in	X	Alphanumeric and Percen
19			Radial Planes					
20			Radial Cylinders					
21								
22	Grid System	Name						
23		U11 CS						
24		Coordinate System						
25			System_Properties	Name	Type	System	Permission Group	GlobalX
26				U11 CS	Grids	CS	Misc	0 ft 0.00 in
27		Frames						
28			Elevation Planes	System_Properties	Name	Position	Axis Of Rotation	Name Rule
29					EL 0 ft 0.00 in (BOBP)	0 ft 0.00 in		Imperial Position
30					EL 15 ft 0.00 in (TOS)	15 ft 0.00 in		Imperial Position
31					EL 30 ft 0.00 in (TOS)	30 ft 0.00 in		Imperial Position
32					EL 45 ft 0.00 in (TOS)	45 ft 0.00 in		Imperial Position
33			X-Planes	System_Properties	Name	Position	Axis Of Rotation	Name Rule
34					A	0 ft 0.00 in	Y	Alphanumeric and Perc
35					B	10 ft 0.00 in	Y	Alphanumeric and Perce
36					C	20 ft 0.00 in	Y	Alphanumeric and Perce
37					D	30 ft 0.00 in	Y	Alphanumeric and Perce
38			Y-Planes	System_Properties	Name	Position	Axis Of Rotation	Name Rule
39					1	0 ft 0.00 in	X	Alphanumeric and Perce
40						5 ft 0.00 in	X	Alphanumeric and Per

5. Go to File →Import→Grids

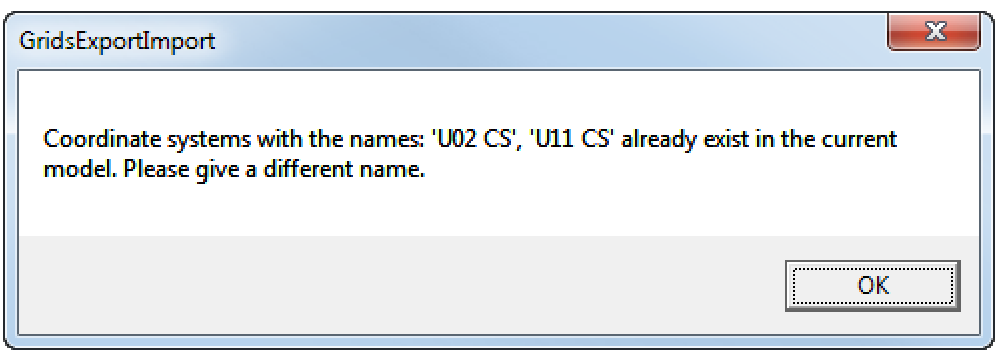


6. Select the grids to import by specifying the Import file location. For this exercise we will use the "U02 CS & U11 CS.xlsx" file that we exported in the steps above.

© Copyright 2014 Intergraph Corporation

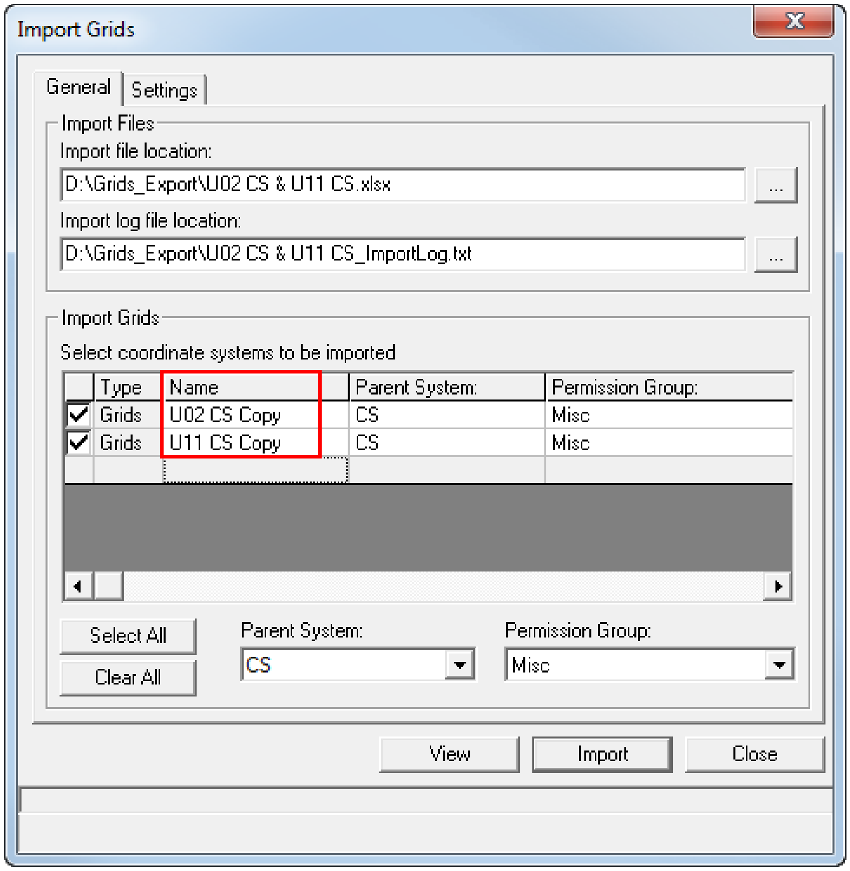


7. Click Import. The warning is displayed as shown below prompting to change the name. Click OK.

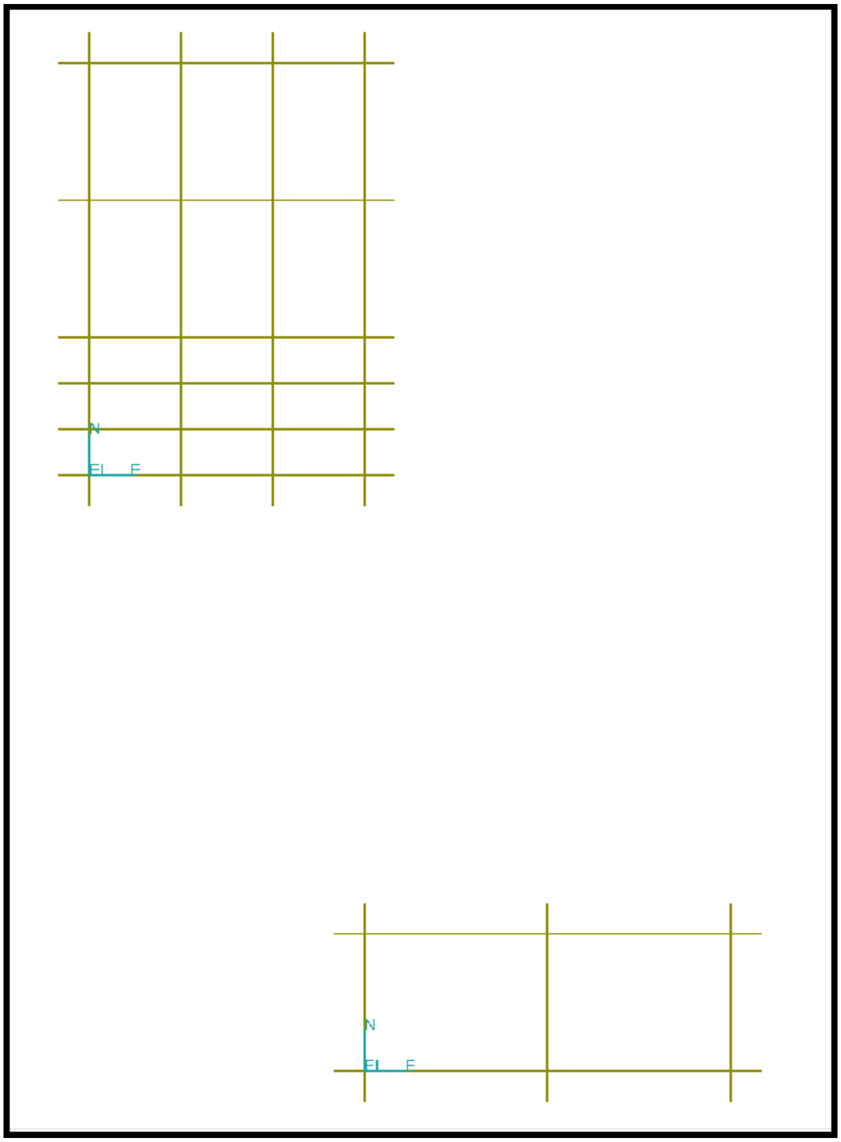


8. Change the name of “U02 CS” to “U02 CS Copy” & “U11 CS” to “U11 CS Copy”.

© Copyright 2014 Intergraph Corporation



9. Click Import.
10. View → Fit.
11. Select the “Looking Plan” view from the common view toolbar.
12. Your view should resemble the graphic below.



Structure: An Overview

Objective:

By the end of this session, you will be able to:

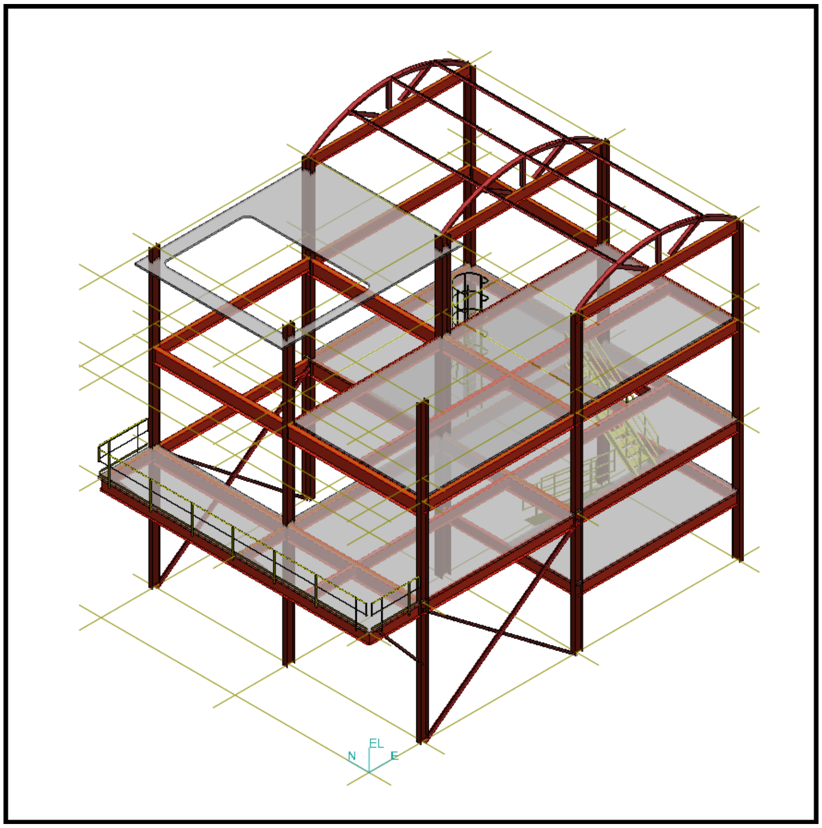
- Identify the tasks that can be performed in the **Structure** task.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Grids: An Overview

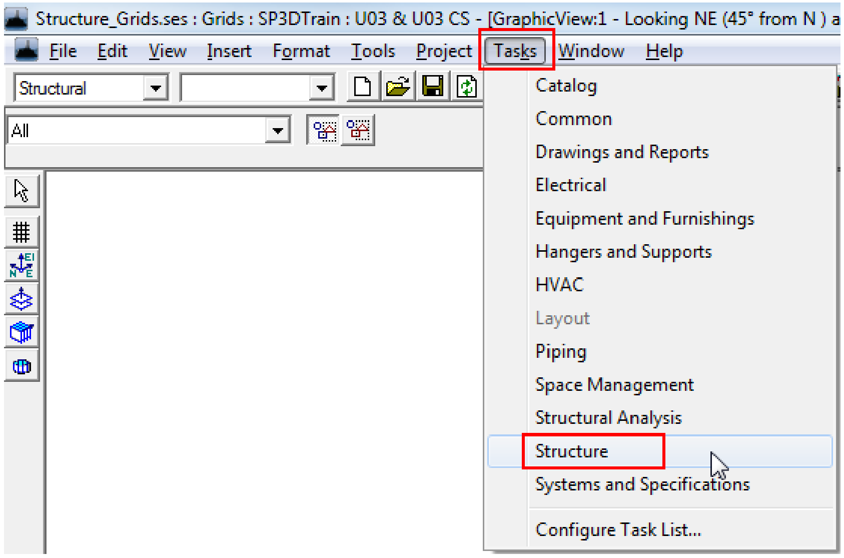
Overview:

The **Structure** task enables you to design structural steel framing and supports, foundations, floors, slabs, platforms, ladders, handrails, and simple industrial architecture with roofing, cladding, walls, doors, and windows.



To open the **Structure** task in SP3D, click the **Tasks** → **Structure** command.

© Copyright 2014 Intergraph Corporation



Common Structure Tasks:

Using the **Structure** task, you can place and modify the following design objects:

- **Member system and part**: Structural member design is divided into two design objects, the member system and the member part. A member system defines the logical axis of the member and the structural frame connectivity through frame connections. It is the system parent of a collinear set of one or more member parts. In other words, it forms the stick model of the structural frame. The member parts are created based on the logical axis of the parent member system and the user-defined location of splits. The command to create a member creates the member system and a member part at the same time. After a member system is split into multiple member parts, you can assign different materials, section sizes, and cardinal points to each member part. The available sections and materials are defined in the Catalog. The **Edit** ribbon of member parts allows you to edit both the member system and the member part. So, you normally consider the member as you design. The system/part model allows engineers to lock the frame design from changes without preventing the further detailing of the member parts for fabrication and construction purposes. You can trim the member part, as required, for the specific detailed connection design. You lock the member system by setting the **Status** property to **In Review** or **Approved**. You can separately lock the member part by setting its status.

- **Split in member system**: A split divides a member system into multiple parts. The splitting function is useful for designing intersecting grids of connected members. You select the member systems to split and the objects that define the split location.

- **Linear member**: A linear member is a member system consisting of a standard section from the Catalog projected along a line defined by two points.

- **Curved member**: A curved member is a member system consisting of a standard section from the Catalog projected along a curve. The curve can be defined by several sketching options including the most common, **Arc by 3 Points**.

© Copyright 2014 Intergraph Corporation

- **Frame connection**: A frame connection defines the connection and positioning of the member systems relative to each other, grid lines, and surfaces of other objects, or simply to a point in space. The ends of a member system can have different frame connection types. They are created as you place members. When placing a member, you select the type of frame connection that represents the intent of engineering positioning or you can let the system select the appropriate frame connection using the **By Rule** option. For example, the seated frame connection positions the supported member on top or bottom of the supporting member regardless of the cardinal points that were used to place the members.

- **Assembly connection**: An assembly connection adds or removes material from the member and creates the additional parts needed for the physical joint. It is a design object that controls the creation and placement of other design objects. The current default Catalog provides assembly connection types that simply approximate the final connection by creating trims, base plates, and gusset plates as necessary. You must do the detailed structural connection design external to SP3D. In the future, an assembly connection Catalog can be created that executes the full detailed structural connection design directly in the SP3D model. The assembly connection recalculates automatically when you edit the position and/or size of the members. You apply the assembly connections late in the design cycle to keep the model lighter during frequent editing of the frame design.

- **Slab**: A slab is used to represent any type of constant-thickness solid such as actual floor slabs, steel grating, steel plate, and roofing. The model geometry is a constant-thickness solid, but properties can define layers of different materials for reports. The default structure Catalog defines several different types. You place a slab by selecting or defining a plane and a boundary. Several methods are provided for defining the plane and the boundary.

- **Wall**: A wall is a cross-section projected along a planar path and optionally bounded or trimmed at the top by selected surfaces. The model geometry is a solid, but properties can define layers of different materials for reports. The cross-section can be rectangular for typical constant thickness walls or can have different shapes as defined in the Catalog. For example, retaining walls can have nonrectangular cross-section. You can define the path in the 2D sketching environment or by drawing directly in the 3D model. You choose how the cross-section is positioned relative to the path by cardinal points defined on the cross-section shape.

- **Opening**: An opening can be made in a slab, wall, or member. You define the shape of the opening by selecting boundaries, sketching the opening outline, or placing a predefined 2D shape from the Catalog. You can control the depth of the opening to create a fully penetrating hole or a recessed opening.

- **Door and window**: The default equipment Catalog provides several types of doors and windows under the **Architectural** node. When you place a door or window using the **Place Equipment** command, the required rough opening is also automatically created in the wall or slab.

- **Footing**: A footing generates geometry with single identity that is sufficiently detailed for general arrangement drawings and for participation in interference detection. It does not create multiple separate parts with fabrication identities and properties. You can position a footing relative to existing columns or by point before the columns exist. You define the height by selecting a supporting surface/plane or by entering the footing dimensions depending on the footing type selected from the Catalog.

- **Equipment foundation**: An equipment foundation can be composed of actual structural members with fabrication identities or just geometry sufficiently detailed for general arrangement drawings. The behavior depends on the type of equipment foundation chosen from the Catalog. You place a foundation relative to existing equipment. The height of the foundation can be defined by selecting a supporting surface/plane or by entering the foundation dimensions.

- **Stair and ladder**: Stair and ladder generate geometry sufficiently detailed for general arrangement drawings and for participation in interference detection. They do not create separate parts with fabrication identities and properties. However, the parameters of the design object can be used along with standards to drive useful material reports. You place stairs and ladders relative to a bottom plane, top edge, and an offset from a reference edge or surface.

- **Handrail**: A handrail generates geometry sufficiently detailed for general arrangement drawings and for participation in interference detection. It does not create separate parts with fabrication identities and properties. However, the parameters of the design objects can be used along with standards to drive useful material reports. You can place a handrail by defining a path by points or by selecting members.

- **Miscellaneous type constructions**: The **Place Equipment** command is available in the Structure task. This command allows you to place predefined constructions from the Catalog. You can place any standard equipment from the Catalog using this command. For example, you can place predefined manholes and catch basins while in the Structure task. However, if you need to model the unique geometry, then you must currently do this work from the Equipment task using the **Place Designed Equipment** command and combining shapes for modeling.

All structure objects are created using information defined in the Catalog. Your Catalog administrator can customize the content of your Catalog to represent your design practices.

You can import or export structure using CIMsteel Integration Standard file format (CIS/2) from the Structure task. The **File > Import > Structure** command 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. The **File > Export > Structure** command exports the structural physical model to the 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.

Typical Workflow:

The typical workflow for modeling in structure, civil, and architecture application area should focus on establishing the appropriate associative relationships for your design intent. These relationships can greatly enhance the productivity of any changes you may have to make in the design.

The tasks that constitute the workflow of structure are as follows:

1. <u>Defining a grid system:</u>

   Before placing structure in a model, it is advisable to place reference grids and elevations using the Grids task. The grids provide a frame of reference to place structural as well as other application design objects. Instead of placing columns in free space, for example, you can place columns with respect to a particular elevation and east/north grid intersections. Later, if you need to adjust the column spacing, you can move a grid and all associated design objects will automatically adjust to the new position.

2. <u>Modeling members in order of construction using frame connections:</u>

   Members are positioned relative to grid lines, existing members, or surfaces of other geometry by the frame connections. It means that the member you are placing needs a supporting member

© Copyright 2014 Intergraph Corporation

or other object to define the final position. This allows you to edit the member sizing and positioning and have all the details automatically adjust.

3. Fireproofing:

   Places fireproofing on selected members. Fireproofing is placed in the Insulation aspect, which you must select to display using the Format → View command to see fireproofing in the model. You can define fireproofing setback distances from each member end, apply multi-segmented fireproofing along a single member, and apply fireproofing to the entire model in one operation while still maintaining placement rules for different member types. The material, grade, thickness, and rating of the fireproofing is controlled by the fireproofing specifications defined in the catalog. The standard encasement shapes (block, block-top exposed, contour, contour- top exposed, and round) are provided. Encasements can be placed on members using the encasement rules or can be selected manually for placement on a member.

4. Executing structural analysis workflow:

   Member design and analysis design need to be cycled until frame structure is appropriate. For more information related to the structural analysis workflow, refer to the Structural Analysis sessions.

5. Placing slabs or gratings relative to elevation planes:

   Slabs and platform grating levels need to be positioned relative to elevation planes rather than member surfaces. You may decide to later delete a particular member when you change the model. While placing a platform, for which no elevation plane exists, use **Offset from Plane** option and reference the ground elevation plane or create another elevation plane. Do not use the **3 Points Plane** option of setting the elevation for slabs and gratings. You may pick associative points for initial positioning that you do not intended to later drive the elevation or some of which were deleted during detailed modifications. In other words, elevations should be managed, whenever possible, using your well-controlled elevation planes.
   Optionally, if you do not want to have designs that you can drive by editing the elevation planes or offsets from your standard elevation planes, then use the **3 Points Plane** option for positioning the slab. Make sure when you key in the three points, they do not reference geometry. This gives you a surface in space with no relationships.

6. Placing arbitrarily oriented plates by the **3 Points Plane** option using the **Place Slab** command:
   When you create the plane for the plate position, make sure you locate associative points (SmartSketch glyphs) to keep the plate in the right position relative to the supporting geometry.

7. Placing stairs, ladders, and handrails:

   These design objects should typically be positioned relative to the design objects they are physically attached to in actual construction. Again, the exception is to use elevation planes where possible since they represent the most stable design information.

8. Placing footings and equipment foundations:

   Footings are usually placed relative to the members they support. Equipment foundations are designed and positioned based on the equipment location and the supporting surface.

For more information related to structure tasks and concepts, refer to following topics in the user guide *StructureUsersGuide.pdf*:

- *Structure: An Overview*
- *Understanding the Structure Workflow: An Overview*
- *Working with Members: An Overview*
- *Working with Walls and Slabs: An Overview*
- *Placing Equipment from the Catalog: An Overview*

© Copyright 2014 Intergraph Corporation

- *Managing Equipment Foundations: An Overview*
- *Implementing Footings: An Overview*

You can access *StructureUsersGuide.pdf* from *StructurePrintGuide.htm*.

Quiz:

1. What is a member system?
2. Why is it advisable to place grids before placing structure objects?
3. What is the purpose of a frame connection?
4. What is the purpose of an assembly connection?
5. Which design objects can be cut by placing an opening?

© Copyright 2014 Intergraph Corporation

LAB-1A: Linear Member System

Objectives

After completing this lab, you will be able to:

- Understand the structure entities and relationships
- Use Place Linear Member System command
- How to use SmartSketch service to find placement points for the members

Note: Use the grid system to help you place the structural members. Though not absolutely necessary, grids help in the modeling process by providing SmartSketch intersection points at which structural members can be placed. You may also want to make a relationship with gridlines so that if a gridline is moved, everything associated with that gridline will move also.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Grids: An Overview
- Structure: An Overview

Overview:

The concepts of member systems and member parts were introduced in the Structure Overview session. After a member system is placed, you can split it into multiple parts. The procedure for placing splits is described in another session.

A linear member system consisting of single part can be placed by using the **Place Member System** command. The command allows you to position the ends of the member using frame connections. Frame connections define the connection and positioning of the member systems relative to each other, to the grid lines, to the surfaces of other objects, or simply to a point in space. The ends of a member system can have different types of frame connections, which are created as you place the member. The frame connections support effective initial modeling and define a structural model that will adjust appropriately as you move the members and change member sizes.

When placing a member, select the type of frame connection that represents your engineering design intent. Alternatively, you can let the software select the appropriate frame connection **By Rule**. For example, the seated frame connection positions the supported member on the top or bottom of the supporting member, regardless of the cardinal points used to place the members.

You can place a curved member system by using the **Place Curve Member System** command. A curved member is defined by placing a curve of one or more linear or arc segments. The ends can be positioned only by the Axis-Along or Unsupported frame connections.

© Copyright 2014 Intergraph Corporation

Assembly connections are used to trim the connected member parts to the correct length. When a member is placed using frame connections, no interference is reported between the brace and the supporting members. The steel-detailing design phase assures that the members have the required end-cuts. Steel detailing can be accomplished within SP3D by using customized assembly connections or executed using third-party programs outside SP3D. Assembly connections are discussed in another session.

Note:

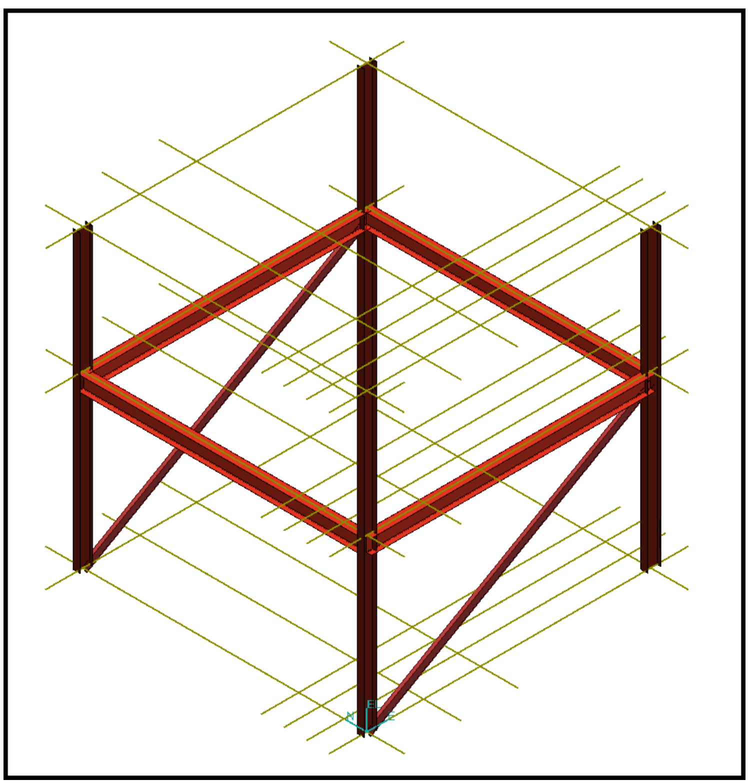
- *Use the grid system to position the structural members. Although grids are not absolutely necessary, they help in the modeling process by providing a reference location for placing columns, instead of placing them in free space. When a grid line is moved by changing the location of the grid plane, all objects associated with that grid line move along with it.*

Placing Columns / Beams / Vertical Braces

1. Open or create a session file and define an "U04 & U04 CS" filter for your workspace. Your workspace should include A2 → U04 and CS → U04 CS systems.

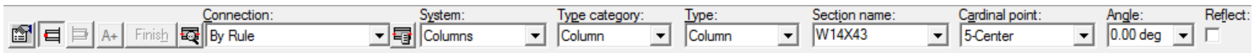
   Note: Make sure your SmartSketch dwell time is set to 0.1 sec. The dwell time specifies the time that you must pause the cursor over the object to add it to the SmartSketch list. SmartSketch service will locate precision points on geometric objects located in the SmartSketch list.

2. Go to the Structure Task environment.
3. Make sure the Active Permission Group is set to *Structural.*
4. At the end of this lab exercise your view will resemble to the following image.



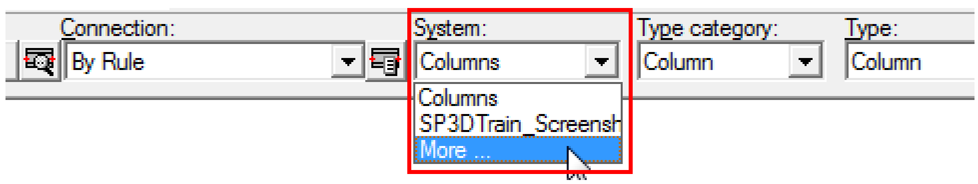
5. Select Place Linear Member System command.
6. System displays the Place Linear Member System smart step ribbon bar.

© Copyright 2014 Intergraph Corporation



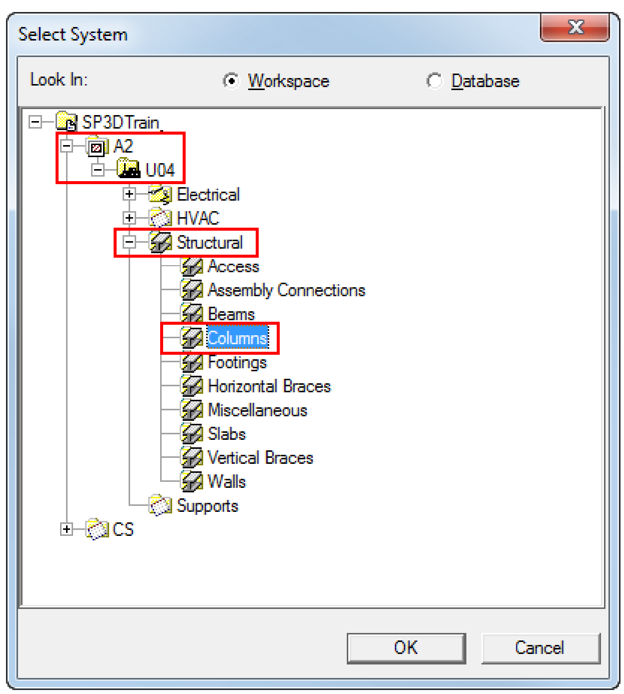
Notes:

- *In the **Connection** box, select a frame connection type to be used for positioning the member. The last frame connection you used will be selected by default. The option **By Rule** allows the software to automatically select a frame connection type based on the type of member being placed and the start and end points selected. You can manually select the frame connection from the Catalog using the **More...** option in the **Connection** drop-down list.*
- *In this workflow, you will locate the intersection of grid lines for the two points that define the member location. The frame connection selected by rule while locating the intersecting grid lines is Unsupported, but has associative relationships to the intersection point. If you modify the position of the grid plane, the position of the members will be updated.*
- *The **System** option identifies the parent system for the member.*



Notes:

- *In the **System** field, the software defaults to the system you used last.*
- *The drop-down list for the control shows the systems you last selected for you to use.*
- *If you want to use a system other than the last-used systems, select the **More...** option to browse for the required system. The **Select System** dialog box opens, as shown in the image below.*
- *In the **Select System** dialog box, expand **A2** → **U04** → **Structural** and select the **Columns** system to indicate the location where the columns will be placed, as shown in Figure 7.*

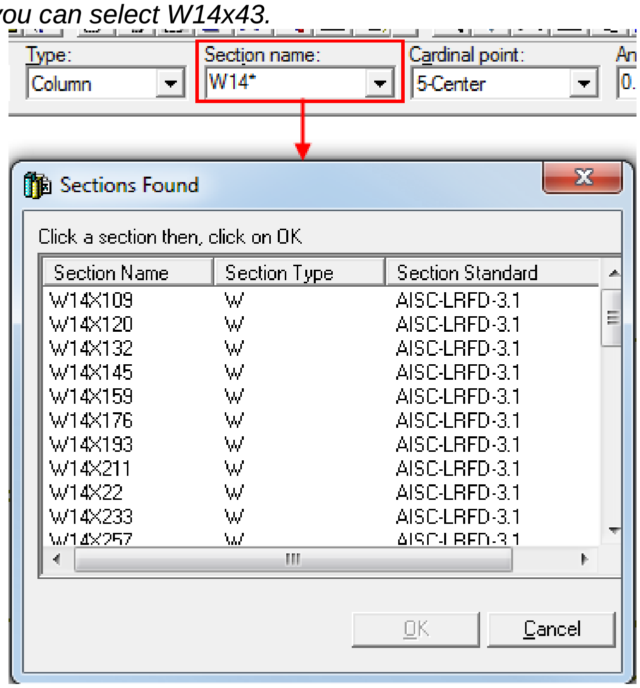


© Copyright 2014 Intergraph Corporation

5. Use the ribbon bar and set the active member parameters as follows:

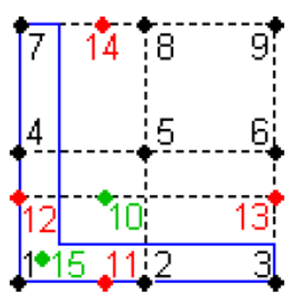
    a. Connection = “By Rule”
    b. System = A2 → U04 → Structural → Columns
    c. Type Category = Column
    d. Type = Column
    e. Section Name = W14x43

Note: Use the cross section lookup service to find the appropriate section. Key in W14 to see another dialog where you can select W14x43.*



    f. Cardinal Pt. = 5

Note: *There are 15 cardinal positions available. The location of the cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the shape of the section. For example, the cardinal point 5 is the center-center and the cardinal point 8 is the top-center.*



    g. Angle = 0

Note: *The **Angle** option defines the angle by which the cross-section is rotated about the logical axis of the column.*

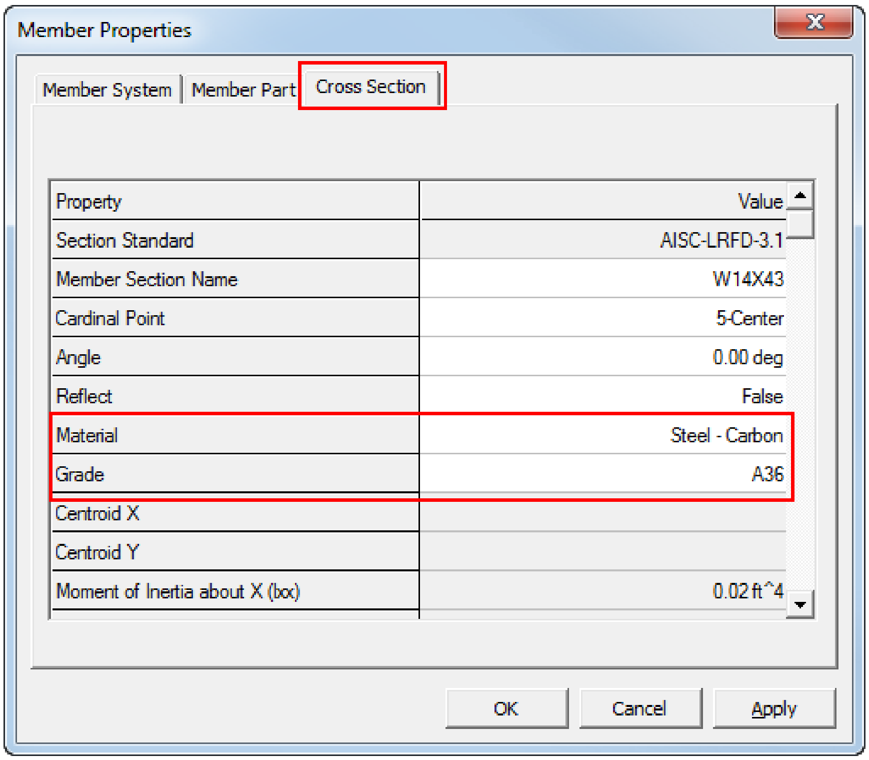
© Copyright 2014 Intergraph Corporation

h. Reflect = Off

Note: *The* ***Reflect*** *option mirrors the cross-section about the local z-axis of the member. This option affects both symmetric and asymmetric sections. You can use this option in situations such as when you want the flanges of a channel section to point in opposite directions*.

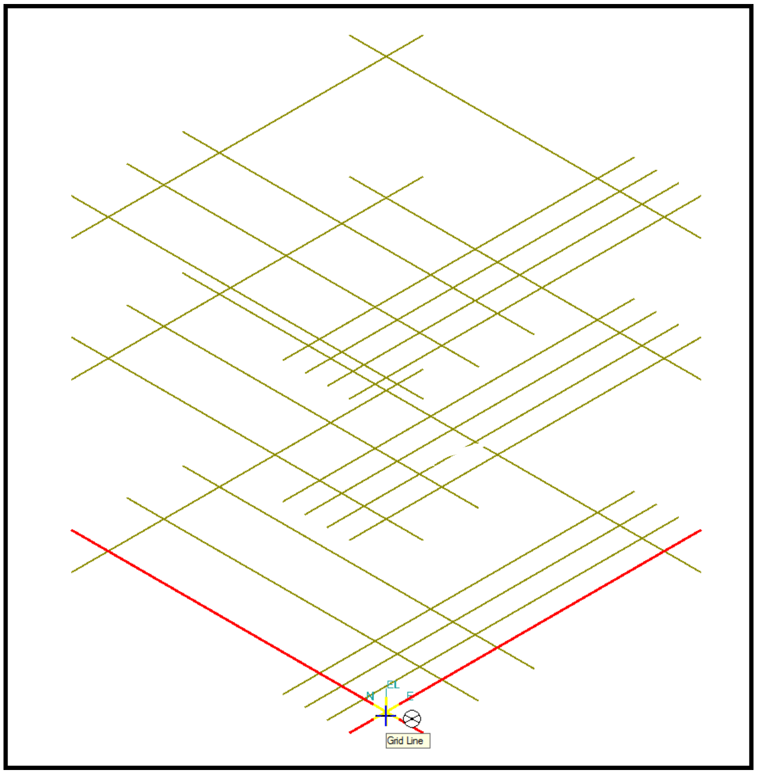
Note: Use the ***properties icon*** *in the ribbon bar to open the properties page. Select the Cross section Tab.*

i. Material: Steel- Carbon
j. Grade A36

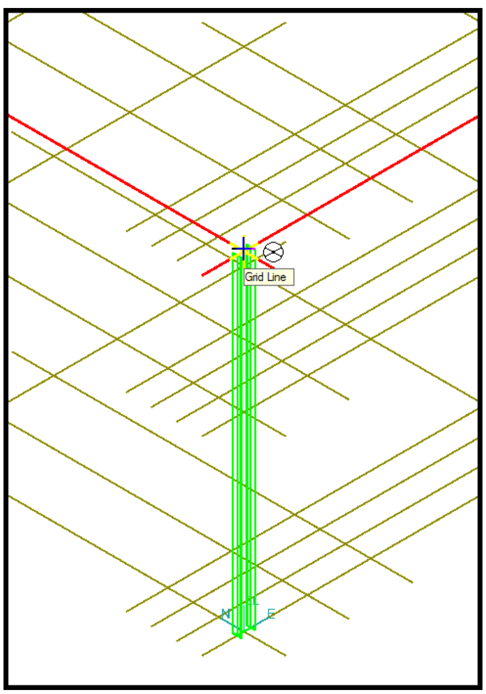


7. For “End 1” of the column, move your cursor over the intersection of the two grid lines. A glyph will appear that indicates the intersection of two lines. See figure below. Then give a Left Mouse click.

© Copyright 2014 Intergraph Corporation

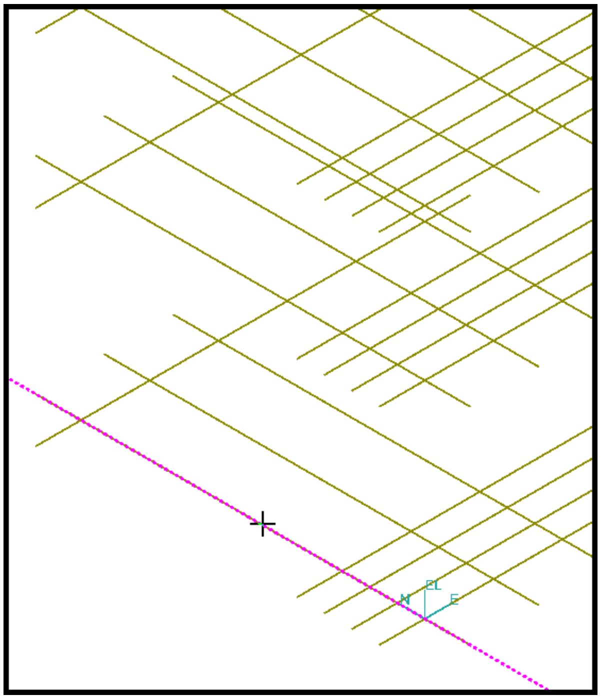


8. Do the same workflow for the other end of the column.



Note: An <u>easier</u> method of selecting an intersection is to use your middle mouse button to select one of the grids. When you do this a Purple infinite line appears telling you that you are locked onto the grid line. You can then use the perpendicular grid to indicate the intersection. See figure below.

© Copyright 2014 Intergraph Corporation

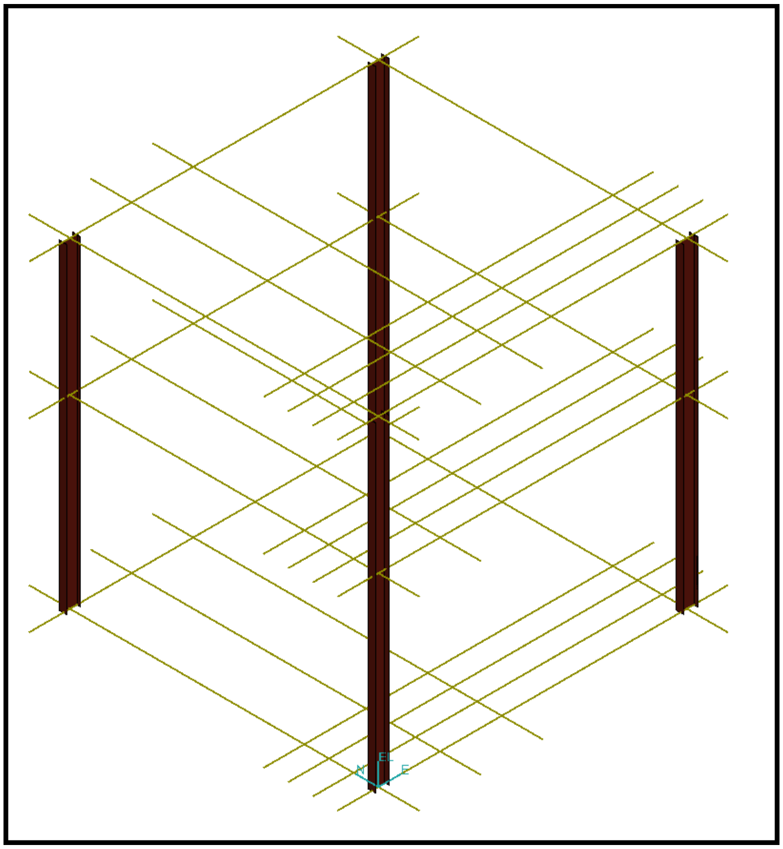


Note: On the Place Member ribbon bar the two <u>smartsteps</u> that indicate End 1 and End 2 of a member also indicated whether members will be placed in an interval fashion "End 1 - End 2" (like columns) or a continual fashion "End1 – End 2 – End 2 …" (like beams).

Notes:

- *After you enter the second point, the command cycles back and you are prompted to enter the first point of the next member.*
- *If you want to use the end point of the previous member as the first point of the next member, click the second point smartstep. You are prompted to enter the second point of the next member. The command will then continue to cycle, repeatedly prompting you to enter the second point of the next member and using the value of the first point that you last entered as the default. This action is repeated until you click on the first point smartstep to indicate you want to identify a different first point. The command cycle to prompt only for the second point sis useful for productively creating, for example, the outer frame of a platform. This is called the Contiguous Placement method.*

9. Place the rest of the columns as shown below:



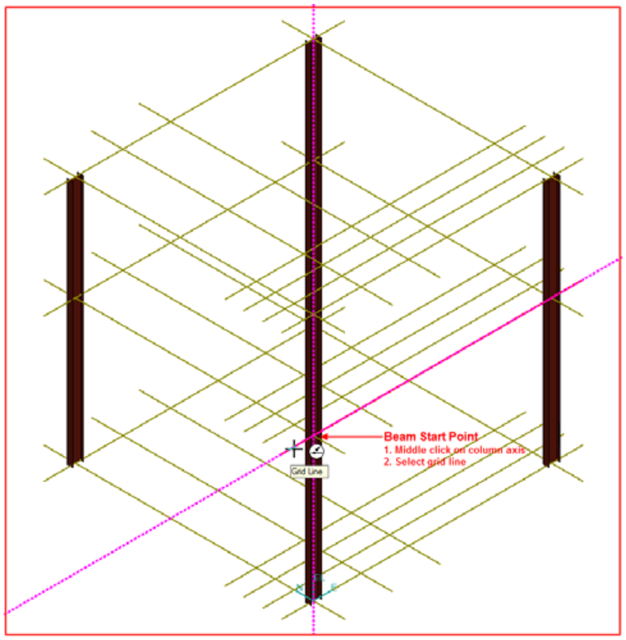
10. While still in the Place Linear Member System Command change the ribbon bar controls to:

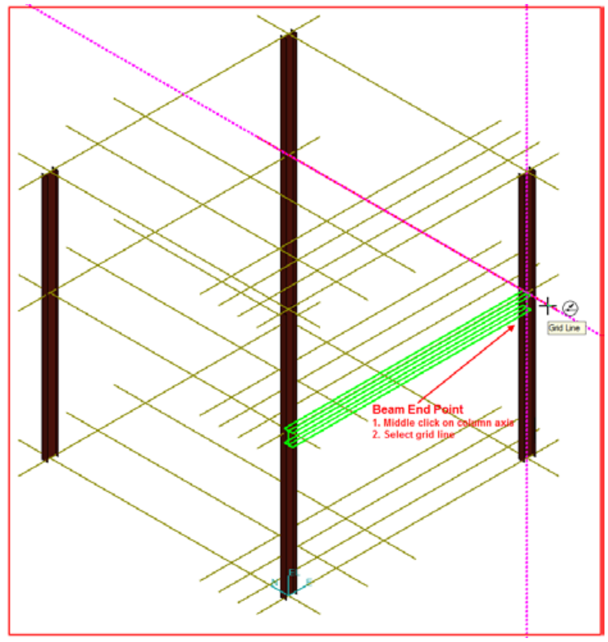
    a. System = A2 -> U04 -> Structural -> Beams
    b. Type Category = Beam
    c. Type = Beam
    d. Section Name = W16x67
    e. Cardinal Point = 8

    Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

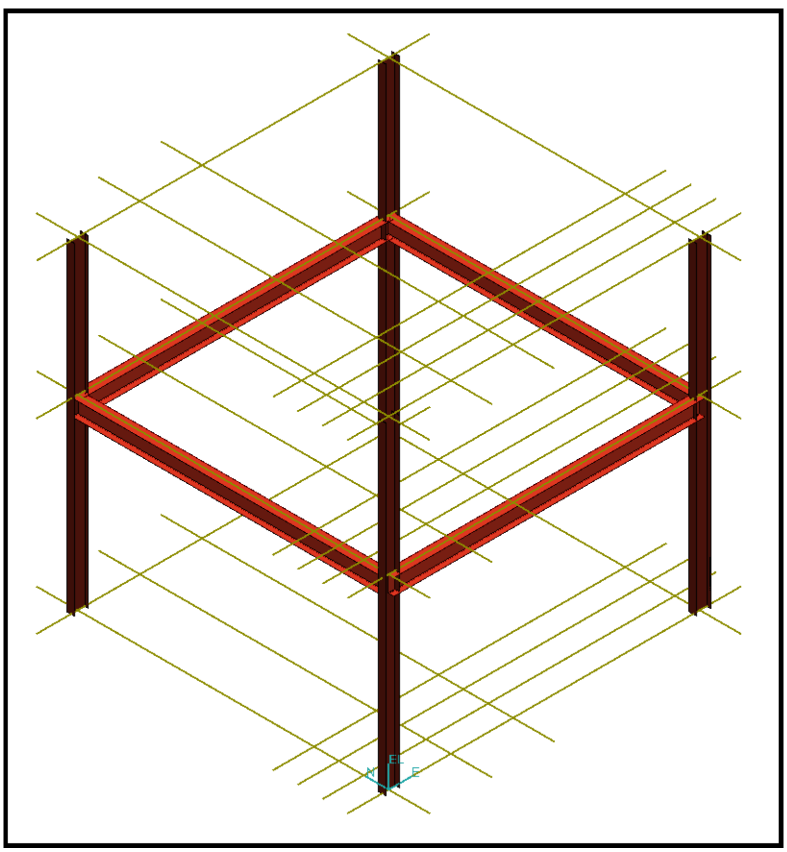
    f. Material: Steel- Carbon
    g. Grade A36

11. Place the beam using the Middle Mouse on the columns first, then pick the intersection grid line to specify the beam location. See figure below:





12. Place the rest of the beams as shown below.



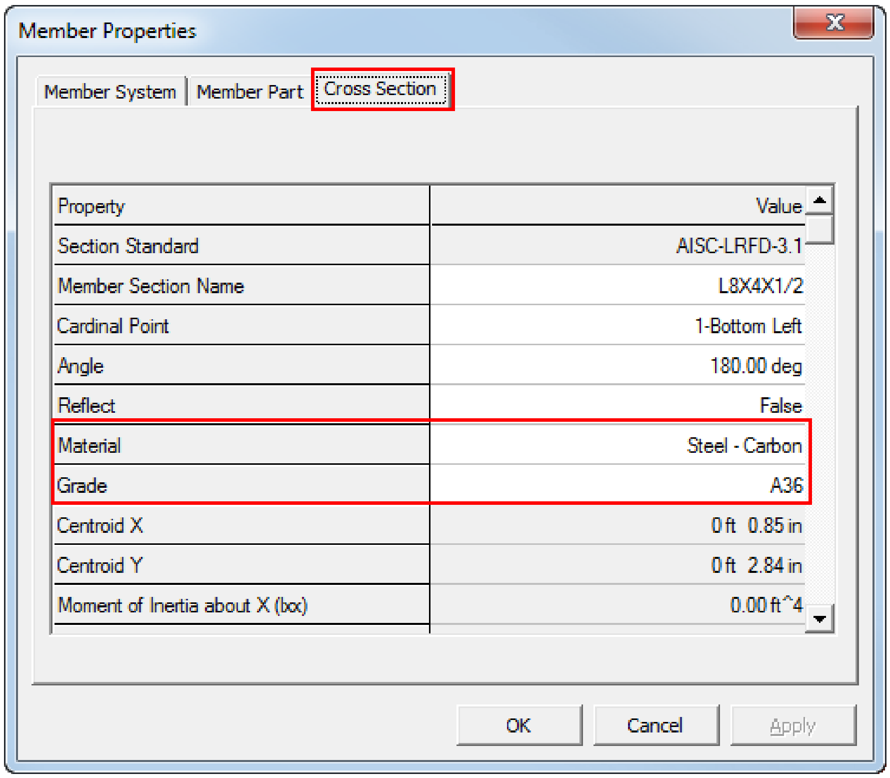
13. While still in the Place Linear Member System command change the ribbon bar controls to:

    h. System = A2 -> U04 -> Structural -> Vertical Braces

© Copyright 2014 Intergraph Corporation

i. Type Category = Brace
j. Type = Vertical Brace
k. Section Name = L8x4x1/2
l. Cardinal Pt. = 1
m. Angle = 180 degrees

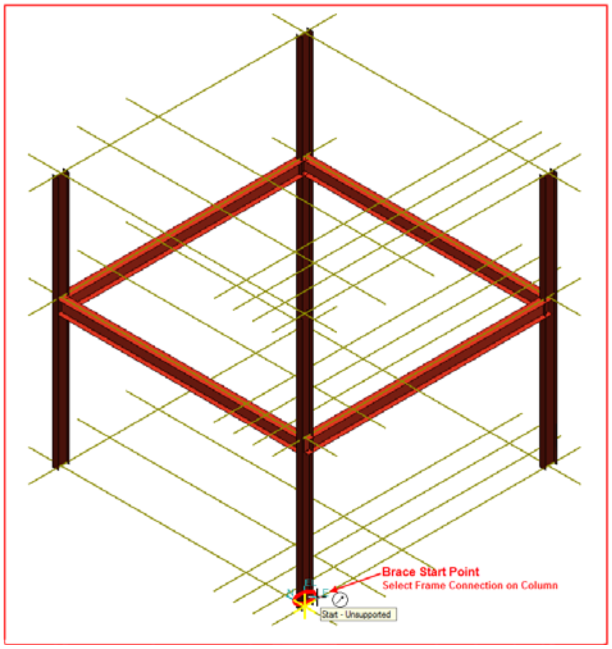
Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

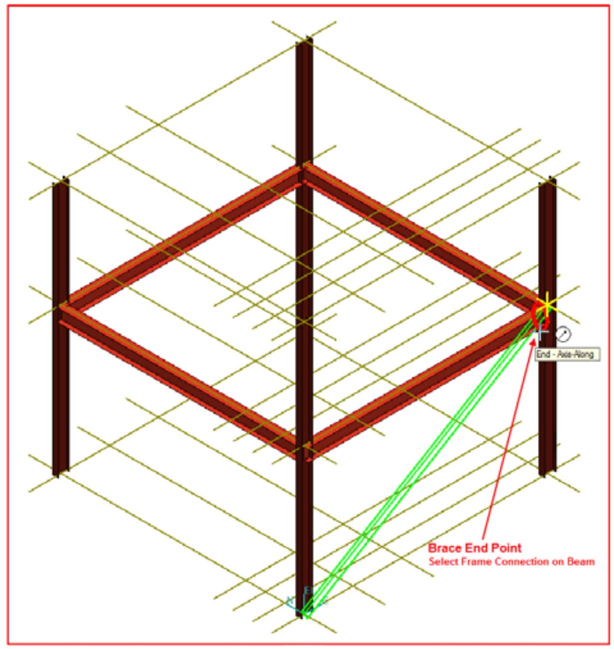


n. Material: Steel- Carbon
o. Grade A36

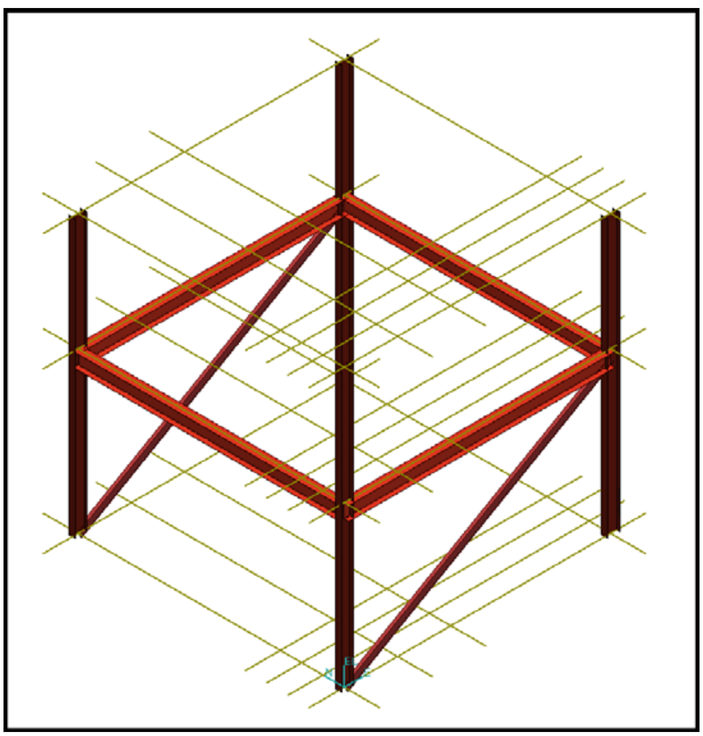
14. Place a vertical brace from the lower end of the first column (at the Frame Connection) to the east end of the first beam. See figure. To identify this location use the Frame Connection (an object used to connect the Suppor<u>ted</u> and Suppor<u>ting</u> member), then a Left Mouse click.

© Copyright 2014 Intergraph Corporation





15. Do the same workflow for the other side of the structure. Your View should now resemble the following graphic.



12. Save your session. Select File -> Save.

© Copyright 2014 Intergraph Corporation

LAB-1B: Copy/Paste Members

Objectives

After completing this lab, you will be able to:

- Understand the structure entities and relationships
- Use Place Linear Member System command
- How to use SmartSketch service to find placement points for the members
- Use Copy / Paste functionality

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview
- Placing Members in a Structure

Overview:

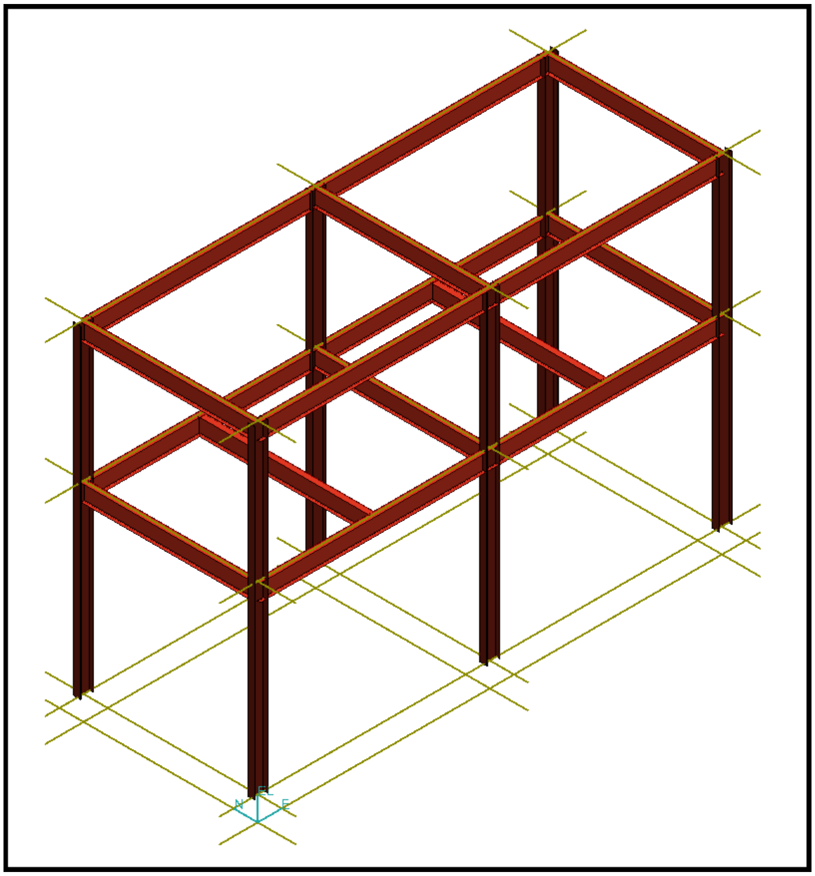
You now know how to place a member. While placing a member, you may have created frame connections to other members, surfaces, or grid lines. These connections influence the behavior of members when you edit, move, copy, or paste them. This session helps you understand how to recognize and use these relationships to get the required editing results.

You can see the connections a member system has to other objects by setting the **Select** command filter to **Member Systems** and then selecting a member system. The member system you select will appear in the chosen color and all the objects connected to it by frame connections will be highlighted in yellow.

You can also select a member system and click the **Edit → Properties** command to view the **Properties** dialog box**.** Click the **Relationships** tab in the **Properties** dialog box to view the relationships that the selected member system has with the other objects.

You will build a small steel-frame pipe rack as shown below using the Place Linear Member System Command and the Copy/Paste functionality.

© Copyright 2014 Intergraph Corporation



Placing Columns and Beams

1. Open or create a session file and define an “U02 & U02 CS” filter for your workspace. Your workspace should include the A2 → U02 and CS → U02 CS systems.
2. Go to the Structure Task environment.
3. Make sure the Active Permission Group is set to *Structural.*
4. Select Place Linear Member System command. System displays the smart step ribbon bar.
5. Use the ribbon bar to set the active member parameters as follows:

Connection:	By Rule
System:	A2 -> U02 -> Structural -> Columns
Type Category:	Column
Type:	Column
Section Name:	W14x53
Cardinal point:	5
Angle:	0 deg

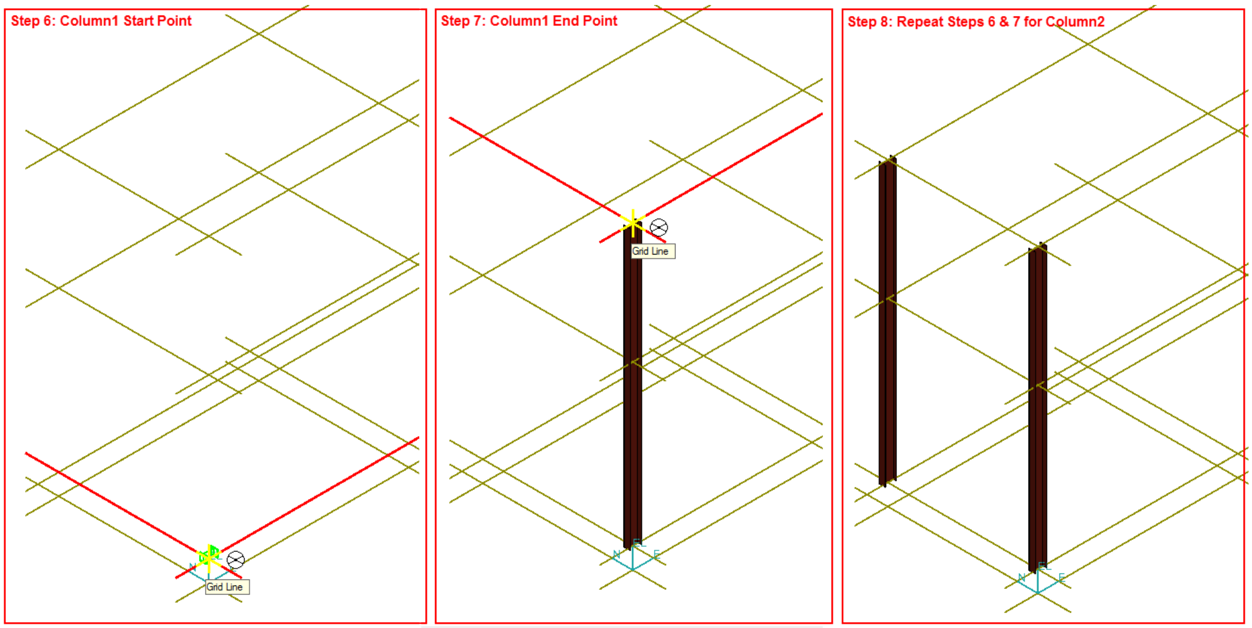
Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

Material:	Steel- Carbon
Grade:	A36

6. Place the first column end at the grid intersections Elevation 2’ 1”.

© Copyright 2014 Intergraph Corporation

7. Place the second column end at the grid intersections Elevation 30’ 0”.
8. Repeat the above steps to place the other supporting column.



9. While still in the Place Linear Member System command change the ribbon bar controls to:

Connection:	By Rule
System:	A2 -> U02 -> Structural -> Beams
Type Category:	Beam
Type:	Beam
Section Name:	W18x40
Cardinal point:	8
Angle:	0 deg

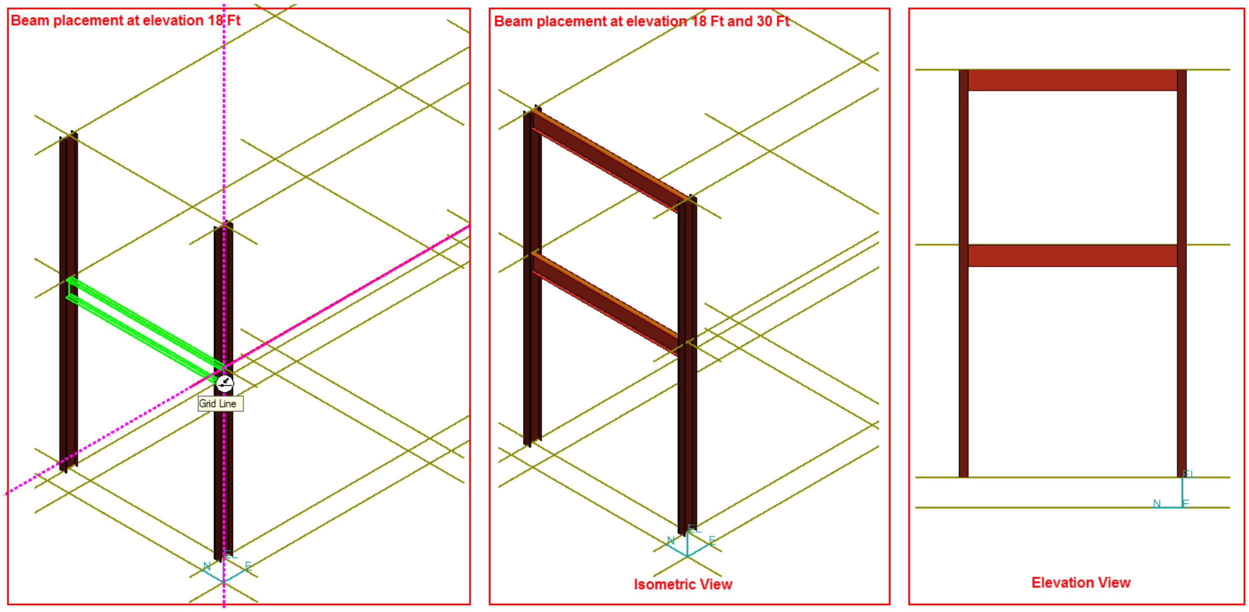
Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

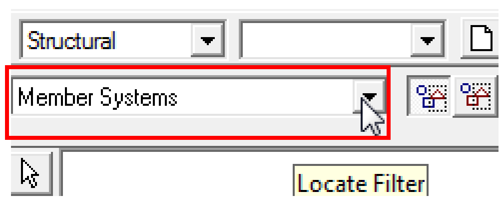
Material:	Steel- Carbon
Grade:	A36

Note: The next step is to place the first two beams at elevation 18 ft and elevation 30 ft. Beams are placed by selecting and lock constraint on to the column and then locate point-on any grid line passing through that column. Beams can also be placed by selecting Frame connections of Supporting members.

10. Place Beams at Elevation 18’ and 30’.

© Copyright 2014 Intergraph Corporation





11. Set the locate filter to Member Systems.
12. Select all the members using the fence method.
13. Go to the Main Menu and select Edit -> Copy.

Note: *You can also Click the **Copy** command on the **Common** toolbar. You can also press **CTRL+C** keys on the keyboard or select the **Copy** command on the right-click shortcut menu. Clicking **Copy** copies all the selected members to the clipboard.*

14. Select the end of the column as the reference point.
15. The system will prompt you to select a reference point for the objects being copied. When you paste objects at a point, the copied objects are positioned relative to this point, but the final position of the member is then computed based on the positioning relationships that control the member position. This example will illustrate this idea. In this case, click and select the end of the column as the reference point.
16. Go to the Main Menu and Select Edit -> Paste **OR** press the **CTRL+V** keys on the keyboard.
17. The Paste dialog box appears. The Paste dialog box shows relationships that can be established between the objects you are pasting and the objects that exist in the model. These are the relationships that existed between the objects you copied and design objects that were not in your copy set. There are two categories of such relationships, those required by the objects you are pasting and those that are optional. The system parent is an example of a required relationship. All design objects must have a system parent. The frame connections are an example of optional relationships. A member can exist without a frame connection relationship. The behavior of the paste dialog box is explained below.

Note:

Behavior of the Options in the Paste Dialog Box:

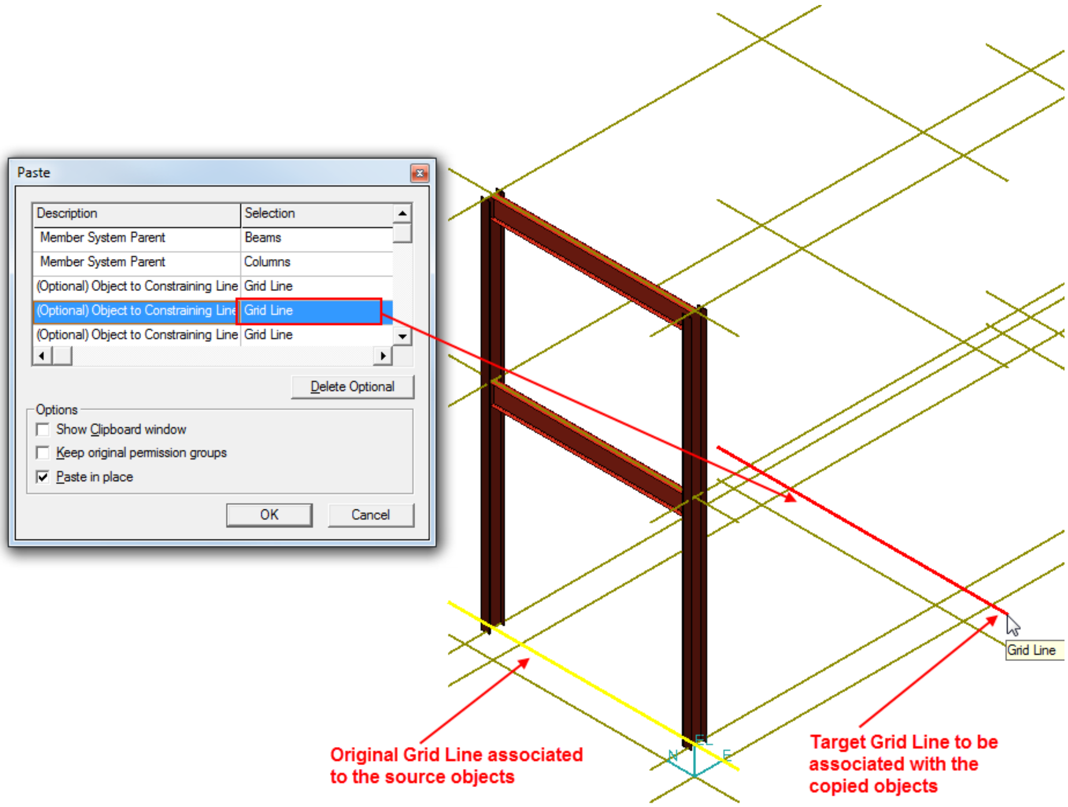
- *If all design objects that you copied had relationships to one object that you did not copy, then the **Paste** dialog box will list this relationship only once. All pasted objects will have relationships to the same object as they originally did.*
- *If you paste the objects into the same model they were copied from, the **Paste** dialog box will offer the original objects as the default inputs for the relationships created on pasting the objects. You can keep the default setting or select a row and identify a different object. When you select a row, the original parent object is highlighted so that you can graphically see what type of input is needed in the context of the objects you copied.*
- *Selecting the **Show Clipboard window** option will display a window that shows the copied graphics and the related objects that were not copied. This will help you understand the graphic object that is needed when you paste the copied objects into a different model. In this example, you will not need the clipboard window because you will see the original related objects highlight when you pick a row in the **Paste** dialog box.*
- *Selecting the **Keep original permission groups** option will assign objects created by the paste function to the permission group of the corresponding original object. This is called mapping permission group by name. However, if the person pasting the objects does not have Write access to that permission group, then the object will be assigned to the active permission group. If the **Keep original permission groups** option is not checked, all newly created objects will be assigned to the active permission group.*
- *Selecting the **Paste in place** option will paste the copied objects at exactly the same position as the original objects. This option is used most often when users paste objects in a different model from the original.*
- *If you are not using the **Paste in place** option, the system will prompt you to enter the **Paste to** point.*
- *The **Paste** command places objects at their original position or at the points you indicate. Then, the system recalculates the position of the pasted objects using the relationships you have established. This recalculation can change the size and position of the newly created objects based on the data from related objects you selected in the **Paste** dialog box. In this example, it does not matter where you click for the **Paste from** and **Paste to** points because the position of the members are fully controlled by relationships to the grid lines.*

In this example, some of the copied members have frame connections involving Grid Lines that were not copied. Use the **Paste** dialog box to select new Grid Lines to define the new position of the members.

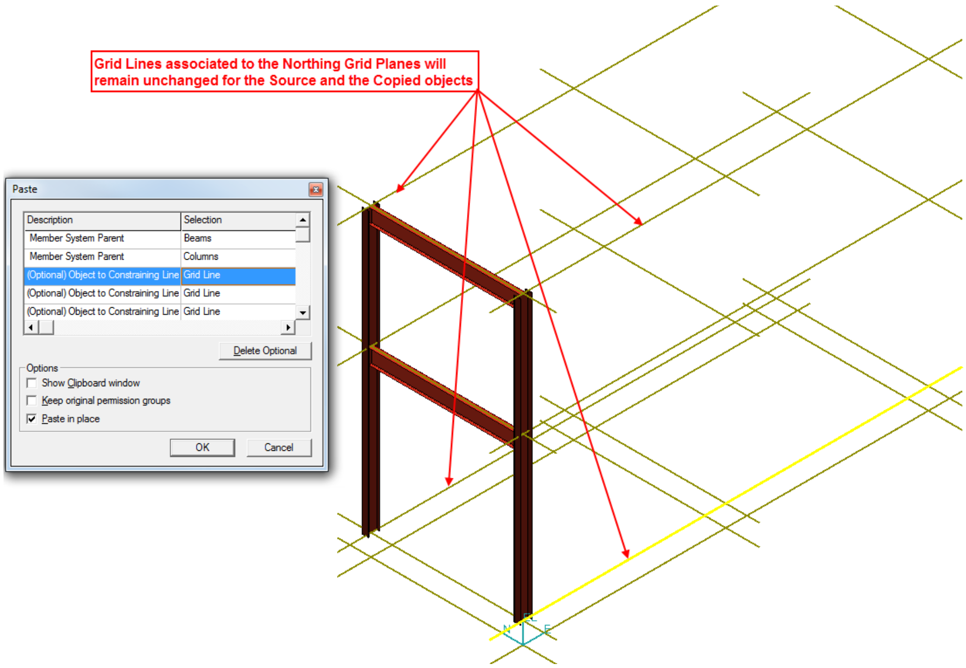
18. Use the arrow keys to move the selection in the **Paste** dialog to the first grid line input. You can also click on the **Description** field to select the row. You see that the grid line located at **–5 ft** Easting and **2 ft 1in** elevation is highlighted in yellow (refer to Figure below).

© Copyright 2014 Intergraph Corporation

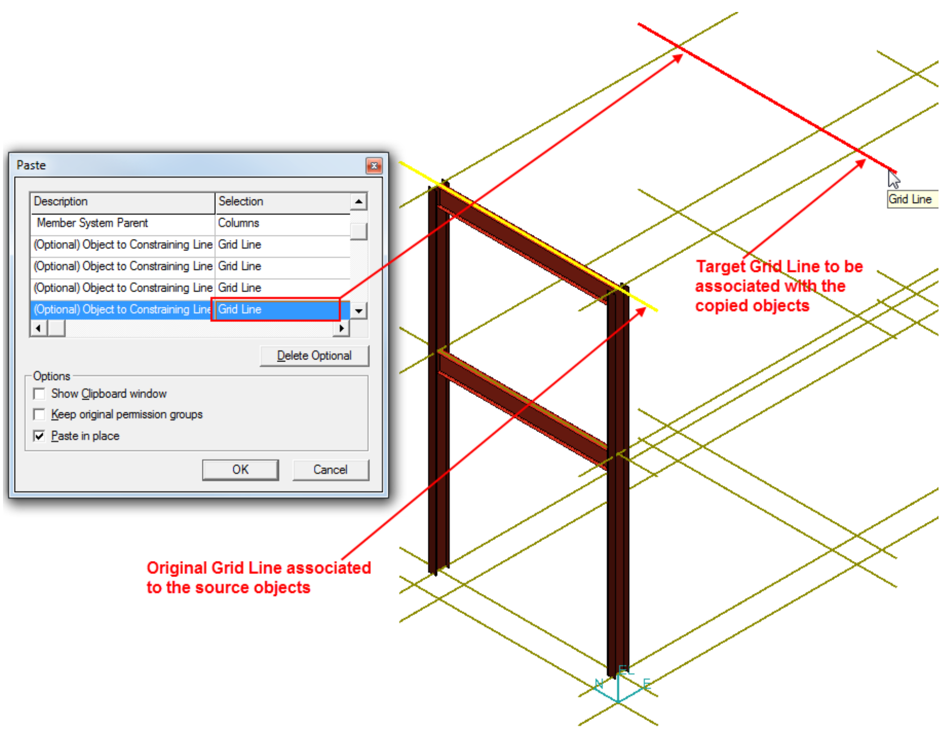
19. Select the new input grid line located at Easting of **20 ft** and Elevation of **2 ft 1in** in the graphic view. This grid line is highlighted in red in Figure below.



20. The grid line inputs associated with the Northing grid planes will remain unchanged. An example is shown in Figure below.



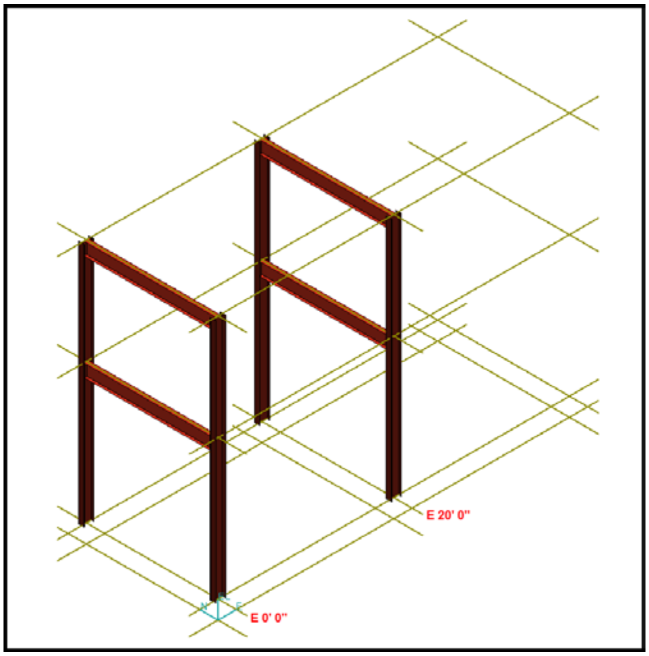
21. Use the arrow keys to move down and highlight the input grid line indicated in yellow. Select the new input located at Easting of **20 ft** and Elevation of **30 ft**. This grid line is highlighted in red in Figure below.



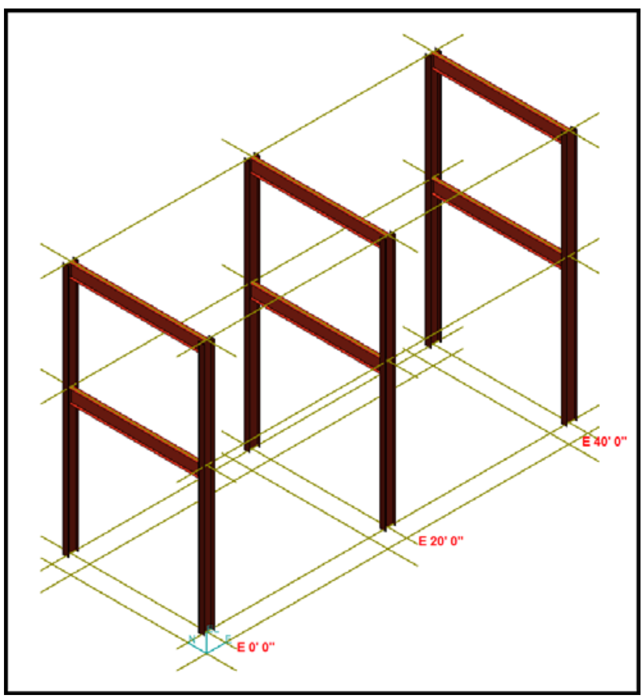
22. Keep all other default inputs.
23. Click "OK" button to commit the transaction.

© Copyright 2014 Intergraph Corporation

Your view should resemble the following graphic



24. Repeat the Paste step to place the next frame at East Plane location 40' 0".



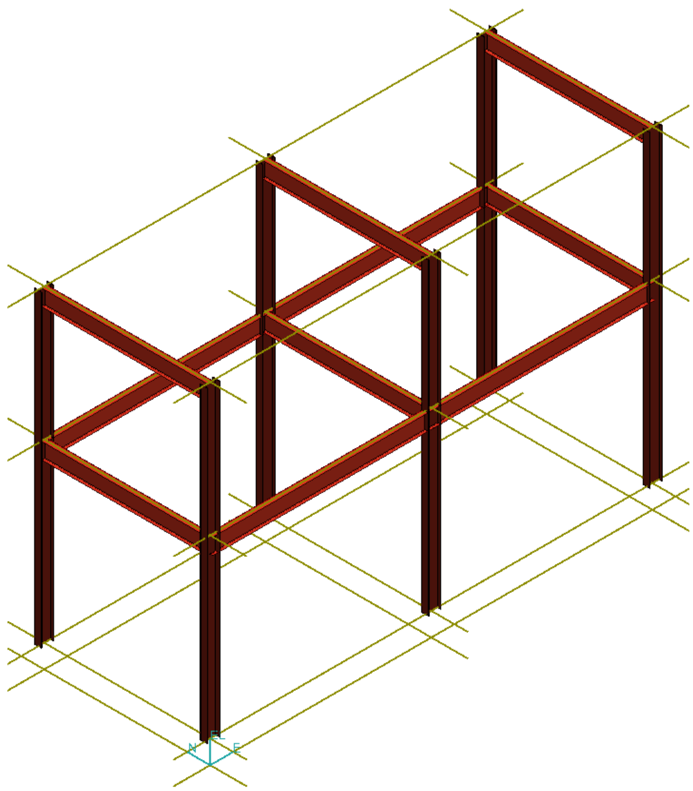
<u>Placing Perimeter Beams:</u>

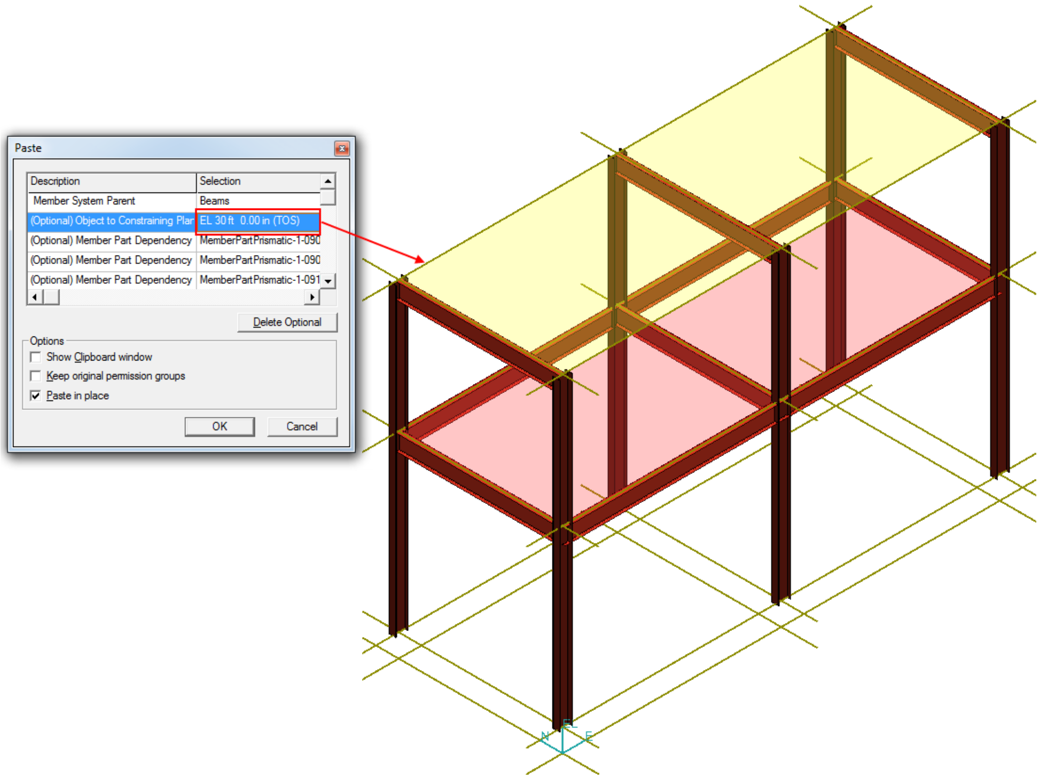
25. Select Place Linear Member System command to place beams as shown in figures below. Use following properties for perimeter beams:

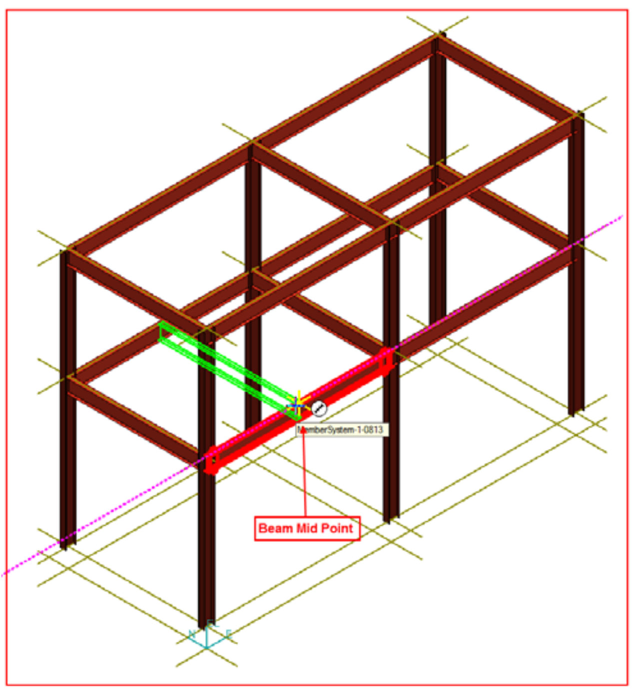
Connection:	By Rule
System:	A2->U02 -> Structural -> Beams
Type Category:	Beam
Type:	Beam
Section Name:	W18x40
Cardinal point:	8
Angle:	0 deg

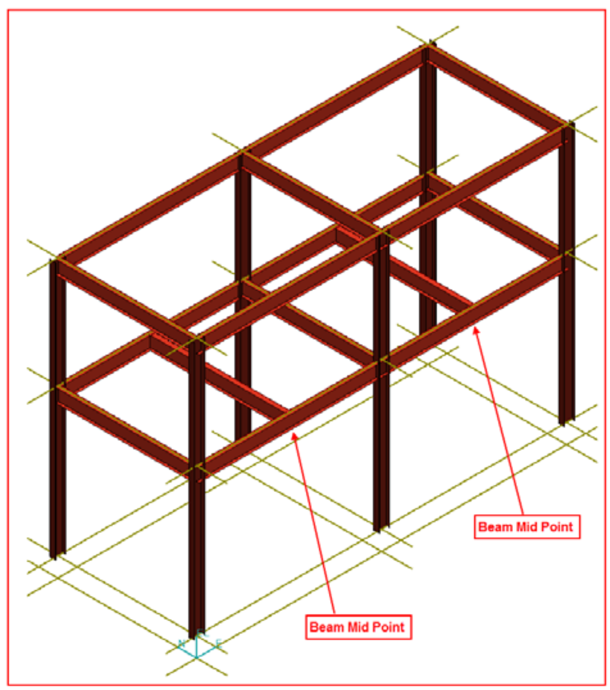
Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

Material:	Steel- Carbon
Grade:	A36




Paste
Description
Selection
Member System Parent
Beams
(Optional) Object to Constraining Plar
EL 30 ft 0.00 in (TOS)
(Optional) Member Part Dependency
MemberPartPrismatic-1-090
Delete Optional
Options
Show Clipboard window
Keep original permission groups
Paste in place
OK
Cancel


Beam Mid Point


Beam Mid Point
Beam Mid Point

LAB-1C: Productivity Commands

Objectives

After completing this lab, you will be able to:

- Place multiple members simultaneously.
- Understand the structure entities and relationships.
- Use Place Framing Members and Place Bracing Commands.
- Use Copy / Paste functionality.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common sessions
- Structure: An Overview
- Lab1A & Lab1B

Overview:

In addition to the **Place Linear Member System** and **Place Curve Member System** commands, the **Structure** task provides commands that you can use to place multiple members in a single operation.

These commands are:

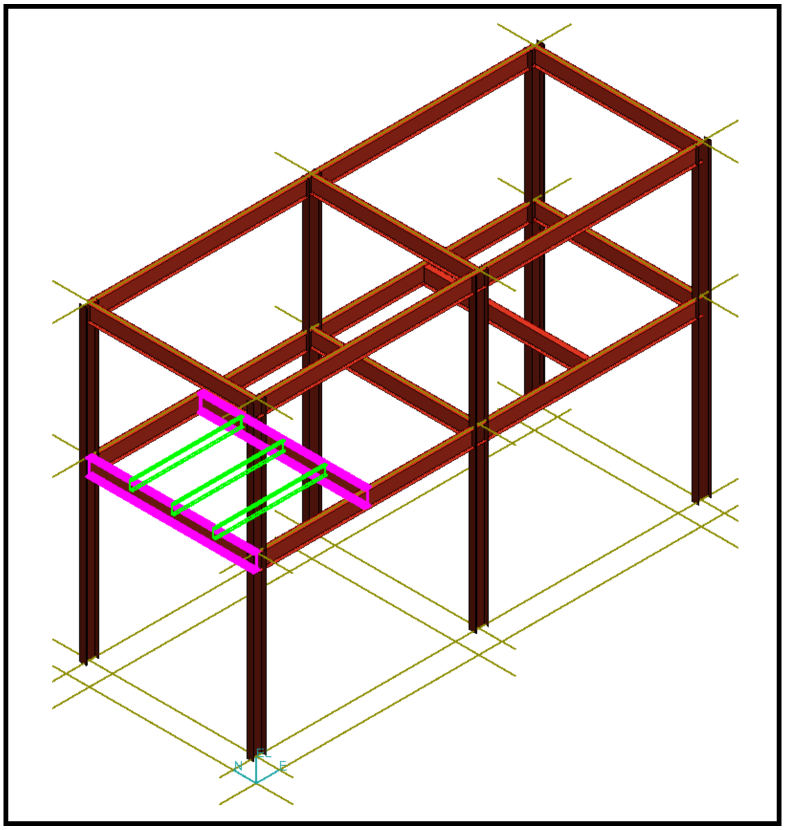
- **Place Columns at Grid Intersections** command: Places columns at each grid intersection within a fence-selected area in a single operation.
- **Place Bracing** command: Places cross braces and chevron braces between supporting members by selecting only the supporting members.
- **Place Framing Members** command: Places multiple framing members between two beams.
- **Place Vessel Supports** command: Places four support members around a vertical cylindrical vessel with a user-defined offset from the surface of the vessel.

You will add framing members and vertical braces to the steel-frame done in lab 1B.

Placing Intermediate Beams for the first floor frame

1. Select Place Framing Members command. Use the view shown in Figure below to place the structural members.

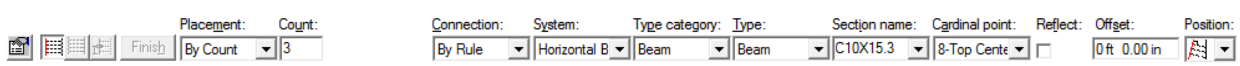
© Copyright 2014 Intergraph Corporation

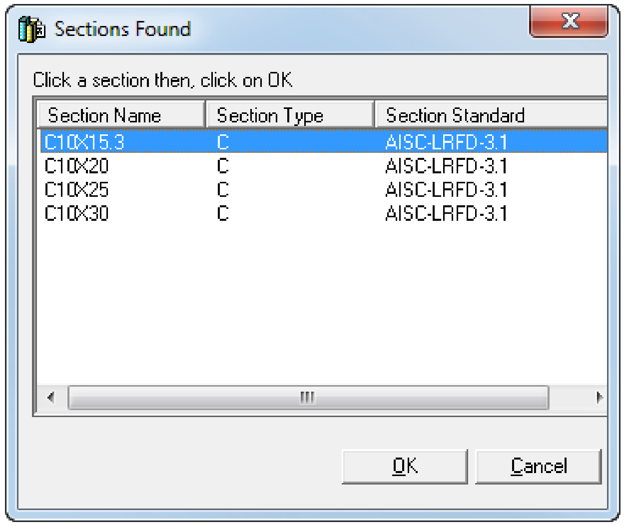


2. Use the Place Framing Member ribbon bar to set the active member parameters as follows:

Fit Mode:	By Count
Count:	3
Connection:	By Rule
System:	A2→U02 → Structural → Horizontal Braces
Type Category:	Beam
Type:	Beam
Section Name:	C10x15.3

Note: Use the cross section lookup service to find the appropriate section. Key in C10 to see another dialog where you can select C10x15.3.*





Cardinal point:	8
Angle:	0 deg
Reflect:	off
Offset:	0’ 0”
Position:	Perpendicular

Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

Material:	Steel- Carbon
Grade:	A36

*Note: On the **Place Framing Members** ribbon, the **Placement** drop-down list provides various methods of positioning framing members:*

- ***By Count:** Places the selected number of members equally spaced along the length of the first selected member.*
- ***Equal Spacing:** Places the required number of framing members to yield spacing equal to or less than (closest to) the maximum spacing value.*
- ***Best Fit:** Places multiple framing members based on the length of the supporting members and the value specified in the **Maximum Spacing** box. The software places all framing members with spacing equal to the Maximum Spacing value specified, except for the first and the last The spacing for the first and last framing member is automatically determined by the software.*
- ***Number and Spacing:** Places the selected number of members at the defined spacing, starting at the start point of the first supporting member.*

*In all the above placement options, you can offset the framing members normal to the axis of the supporting members by entering an offset value in the **Offset** field. This offset value applies to the frame connection.*

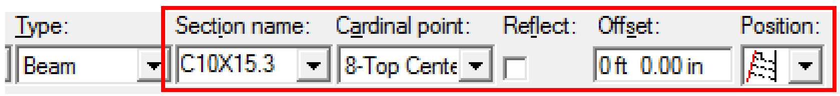
3. In the **Placement** drop-down list on the **Place Framing Members** ribbon, select the **By Count** option.

© Copyright 2014 Intergraph Corporation

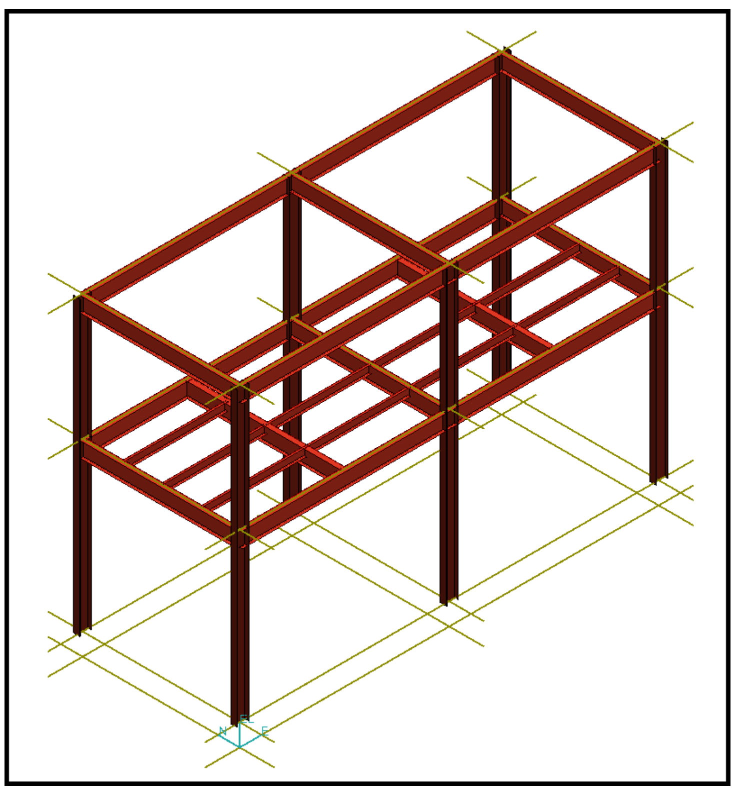
4. In the **Count** box, type **3**, the number of framing members to be placed.

5. In the **Type category** drop-down list, select the **Beam** option.
6. In the **Type** drop-down list, select the **Beam** option.

7. Set the **Section Name**, **Cardinal Point**, and **Offset** options to **C10X15.3**, **8,** and **0**, respectively.

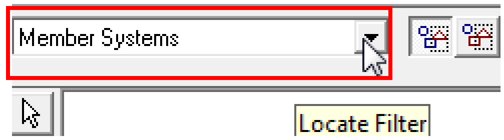


8. Continue using Place Framing Members command by selecting Supporting Member2 to place framing members as shown in Figure below.

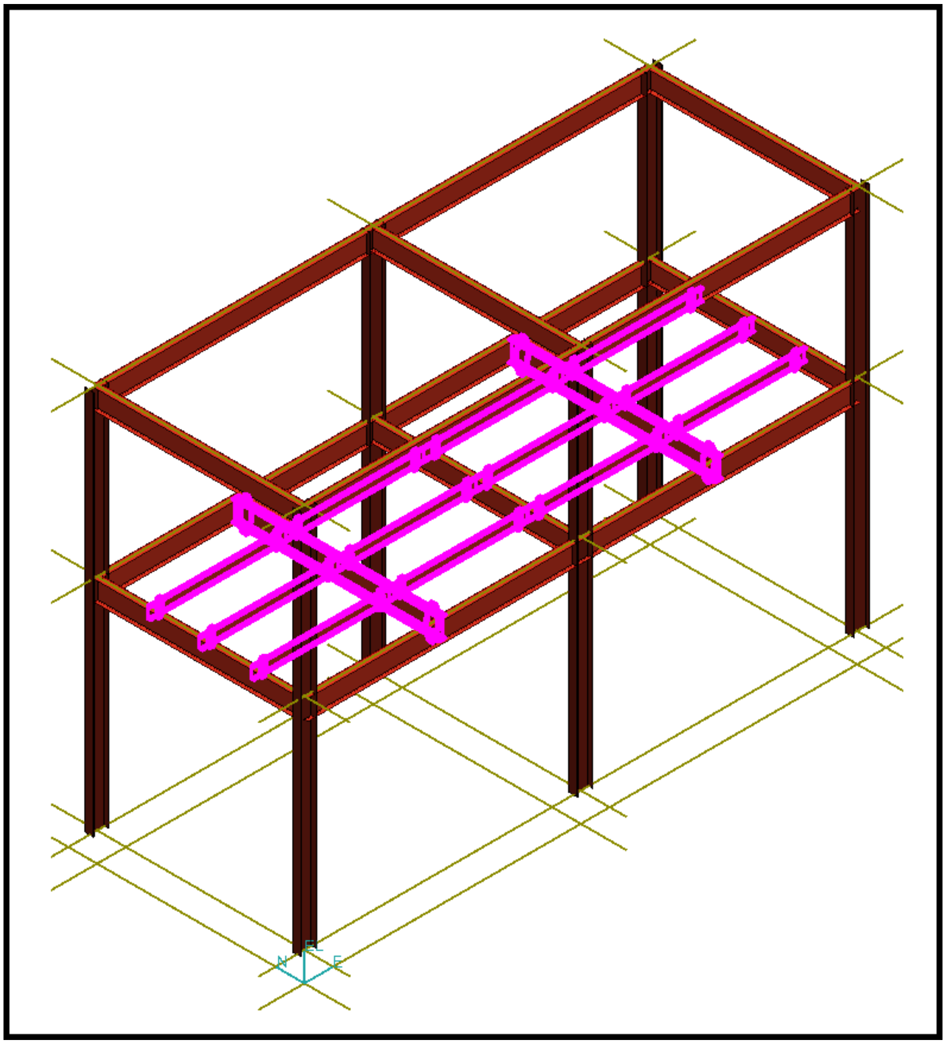


Intermediate Beams for the second floor frame

To place the intermediate beams for the second floor frame at elevation 30 ft, Use the Copy/Paste commands.



8. Set the locate filter to Member Systems.
9. Select 14 members using the Fence method or Multi-select method. See figure below for details:

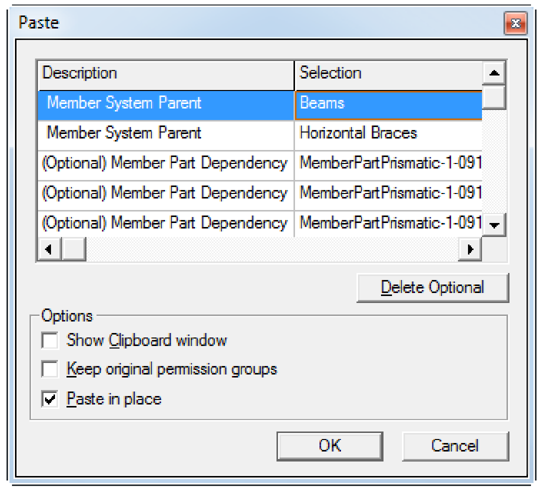


Isometric View of the Structure U02 System

10. Go to the Main Menu and select Edit -> Copy.
11. Select the end of a beam (we just need a point on a object within the select set) as the reference point.
12. Go to the Main Menu and Select Edit -> Paste.
13. Make sure to select the appropriate objects to re-establish the connection in the Paste Dialog box.

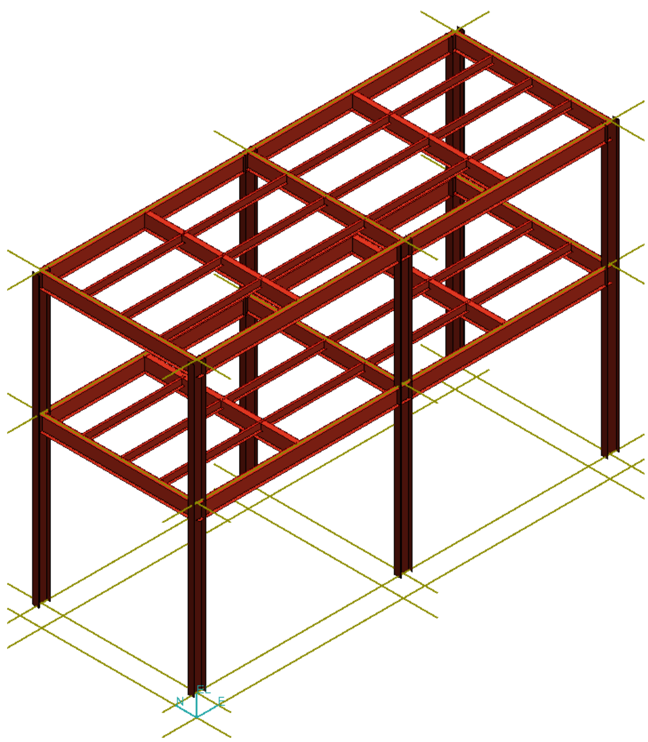
Hint: Only 7 related object need to re-establish the connections. Use the Paste command once.

© Copyright 2014 Intergraph Corporation



14. Hit “OK” button to commit the transaction

    Your View should now resemble the following graphic:



Placing Vertical Braces

15. Select Place Bracing command.
16. Use the Place Bracing ribbon bar to set the active member parameters as follows:

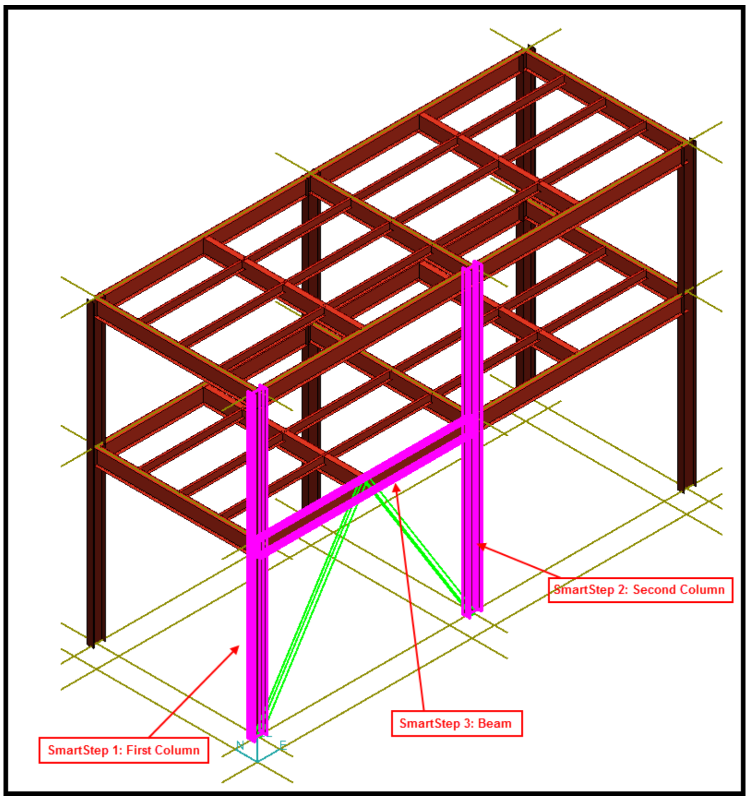
Bracing Type:	Chevron
Connection:	By Rule
System:	A2->U02 -> Structural ->Vertical Braces

Type Category:	Brace
Type:	Vertical Braces
Section Name:	L4x4x1/4
Cardinal point	5
Angle:	0 deg
Reflect:	off

Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

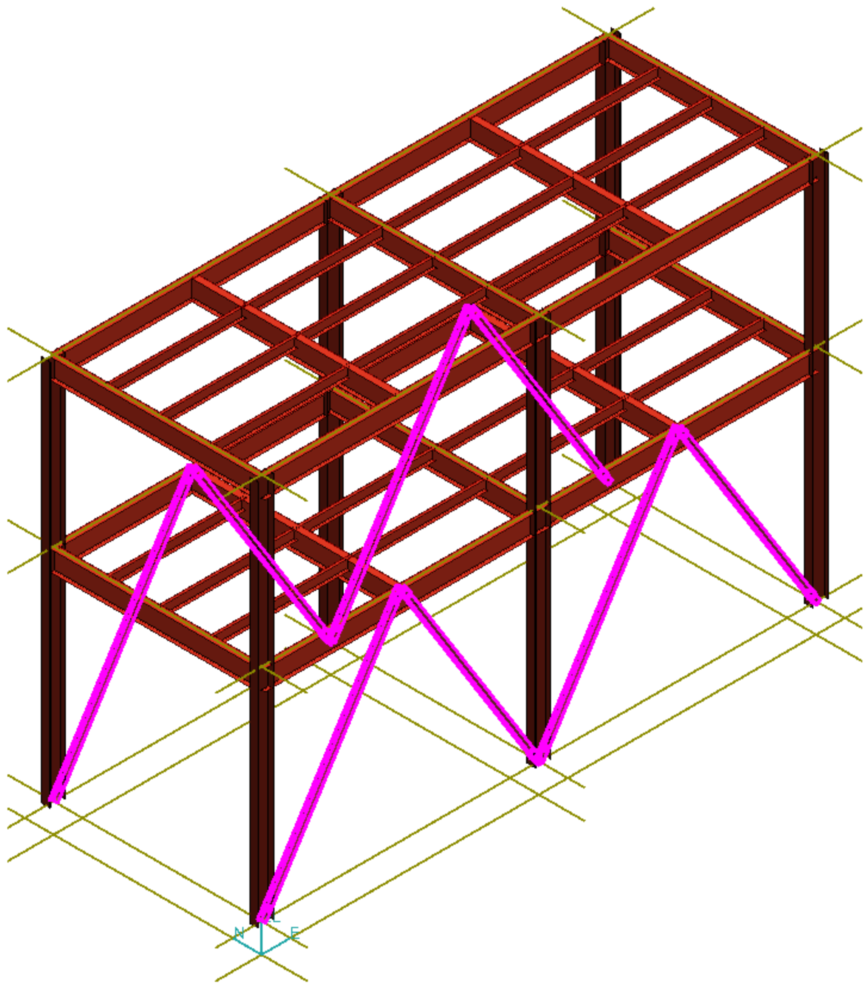
Material:	Steel- Carbon
Grade:	A36

17. Select the first column for the first SmartStep (See figure below).
18. Select the second column for the second SmartStep.
19. Select the Beam for the third SmartStep.
20. Select "Finish" button to commit the transaction.



21. Repeat the above workflow to place the other vertical braces as shown in figure below.

© Copyright 2014 Intergraph Corporation



22. Save your session. Select File -> Save.

© Copyright 2014 Intergraph Corporation

LAB-2: Modifying Member Properties

Objectives

After completing this lab, you will be able to:

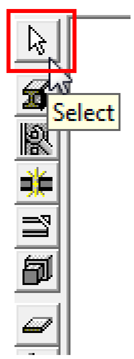
- Understand the properties tab of the member system.
- Change the name by using name rule / material / grade / priority of the member using properties tab.
- Understand the use of locate filter during the member modification.

Prerequisites:

- Lab1A & LAB1B

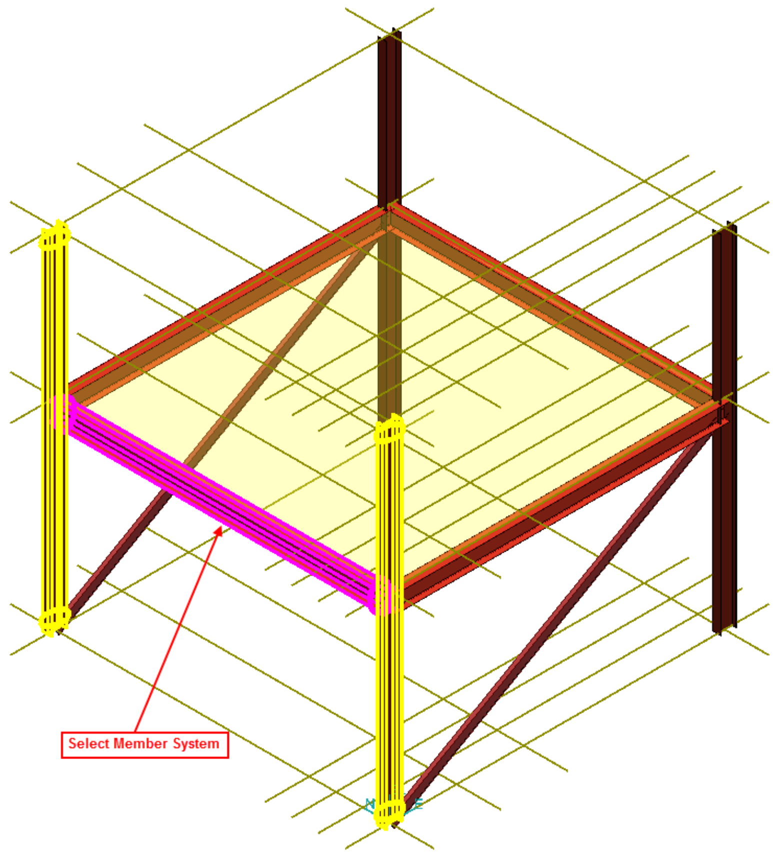
You can modify the properties of member systems or member parts, as required. Define your workspace that includes A2 → U04 and CS → U04 CS systems or select "U04 & U04 CS" filter using File → Define Workspace. To modify the properties of a member system, perform the following steps:

1. Click the **Select** button on the vertical toolbar.

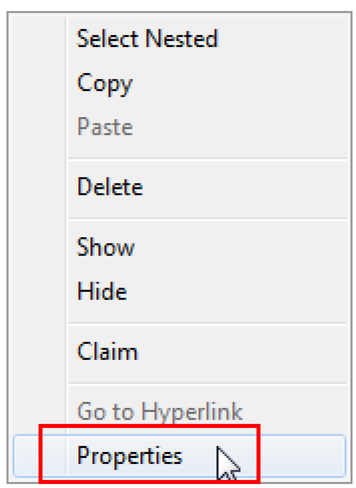


2. Select the **Member Systems** option in the **Locate Filter** drop-down list.
3. Select the member system as shown in the figure below,

© Copyright 2014 Intergraph Corporation



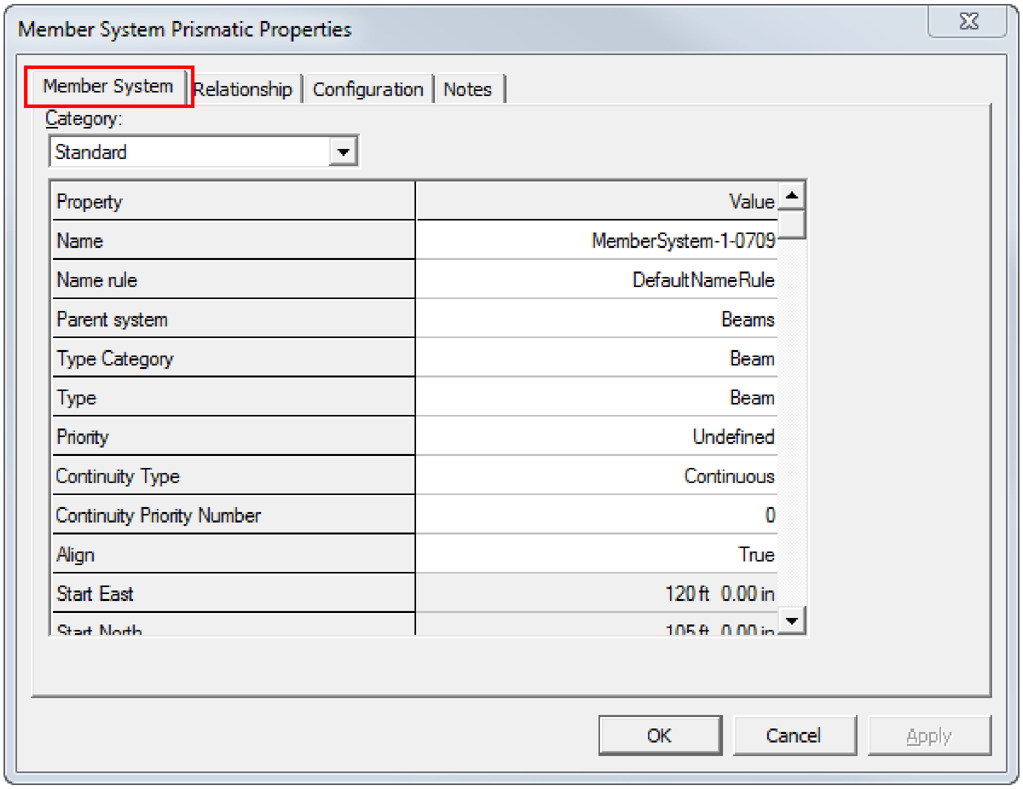
4. Right-click the member system to change the properties and then select the **Properties** command.



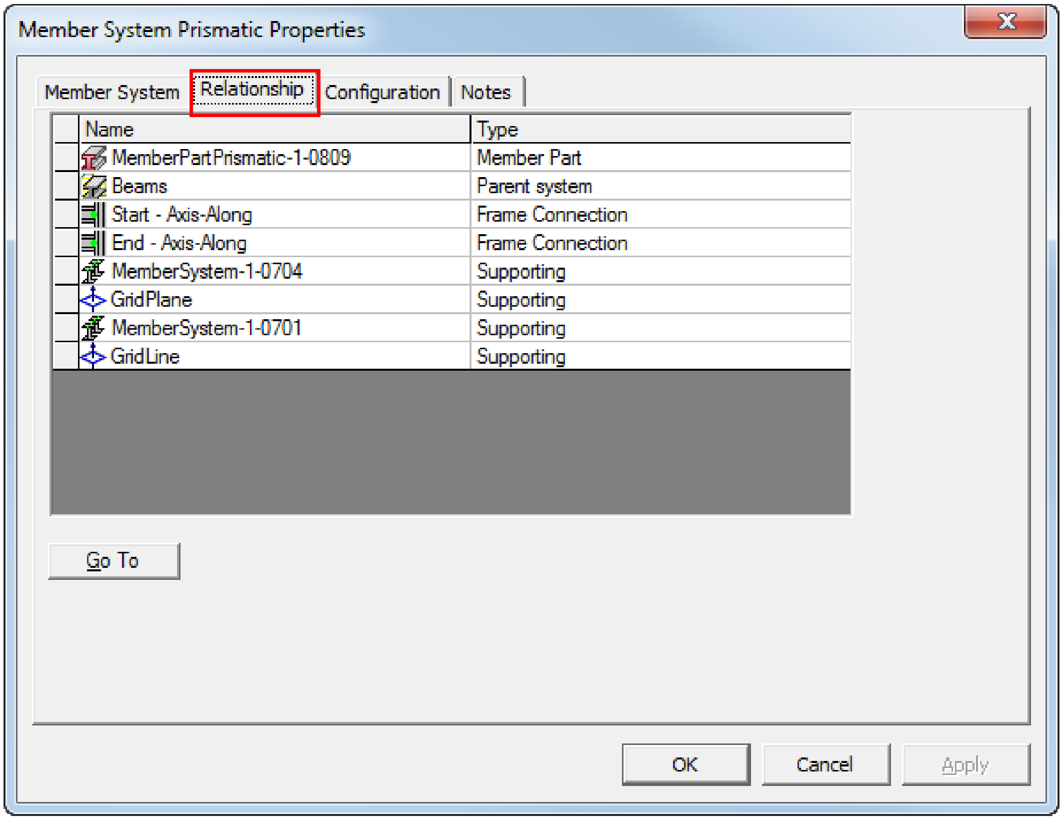
You can also click the  **Properties** button on the **Common** toolbar or click the **Edit → Properties** command to view the properties.

The **Member System Prismatic Properties** dialog box has the following tabs:

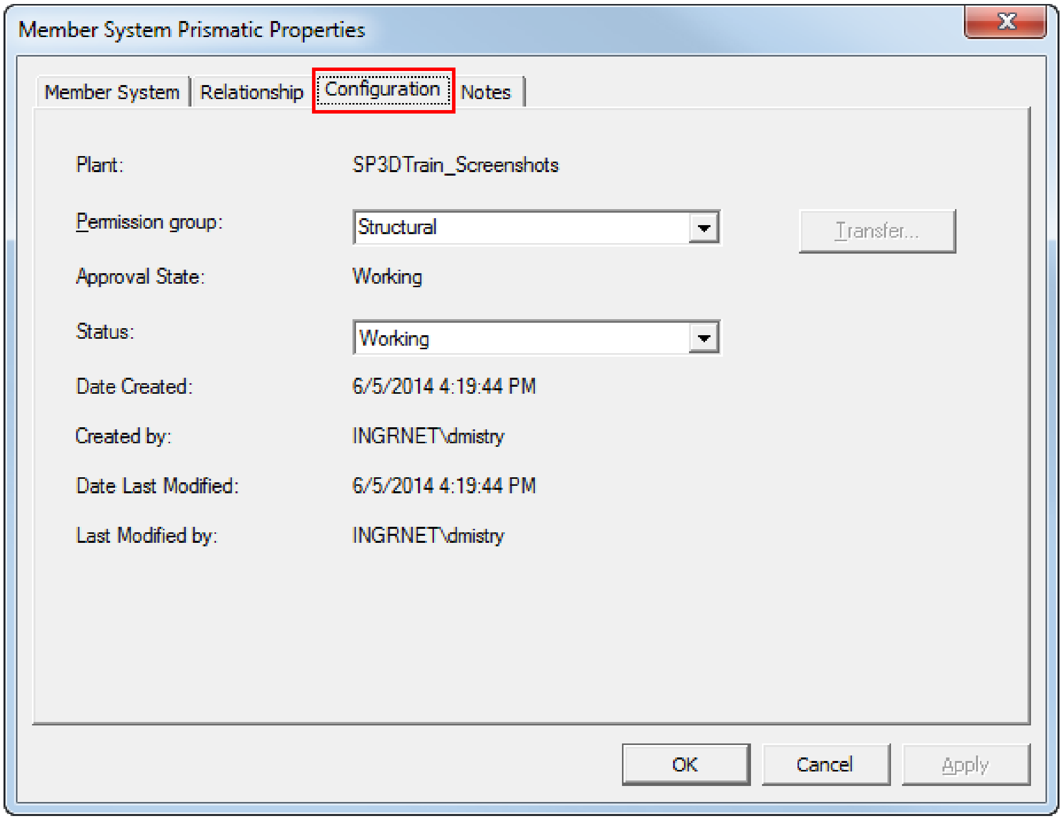
- **Member System**: The **Member System** tab displays the properties of the member system. The property name is displayed on the left and the value of the property on the right side.



- **Relationship**: The **Relationship** tab displays the objects related to the object for which you are viewing the properties. The dialog box indicates the type of relationship the object has, for example, the system parent or a supporting member. You can click the **Go To** button on the **Relationship** tab to select the related object. After the related object is selected, its properties are displayed in the **Property** dialog box. You can also graphically select a different object while the **Property** dialog box is displayed to see the properties of that object.

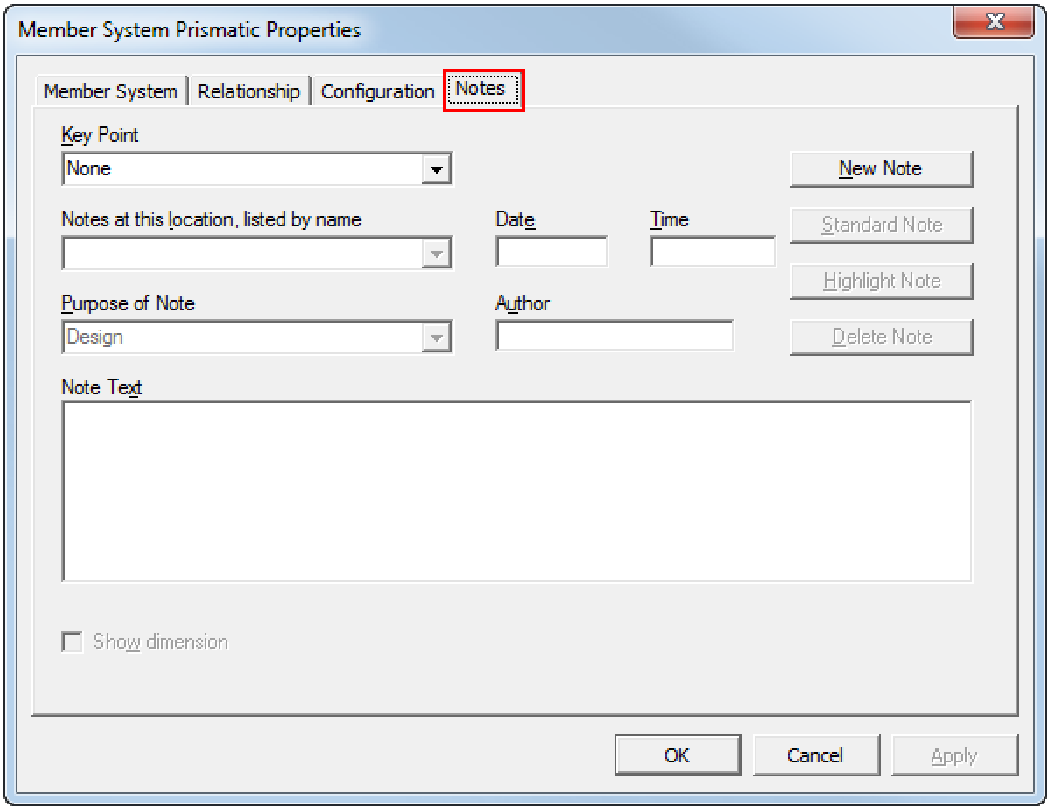


- **Configuration**: The **Configuration** tab displays information about the creation, modification, and status of an object.



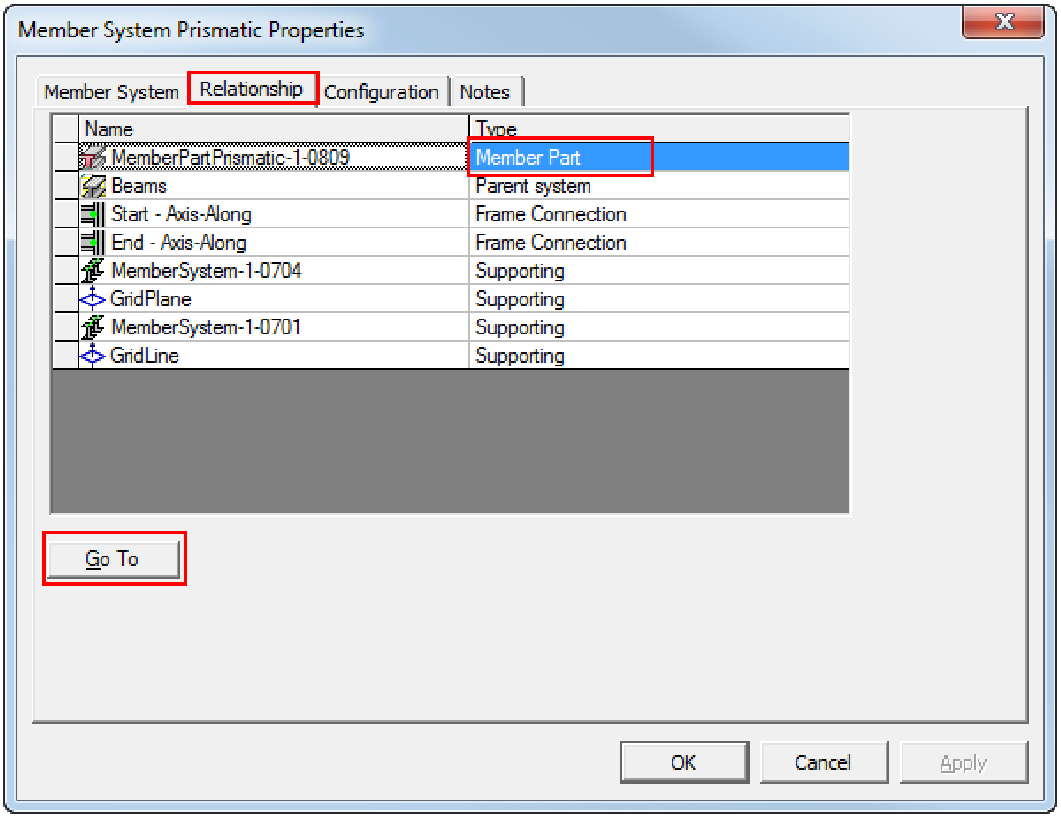
- **Notes**: You can use the **Notes** tab to create and edit special instructions about the design object. Your administrator can configure Drawings to automatically create labels with the text from a note.

© Copyright 2014 Intergraph Corporation



6. Switch to the Relationship tab and select the "Member Part". As soon as the user selects the Member Part in the Relationship tab, the member part is highlighted in the workspace.

© Copyright 2014 Intergraph Corporation



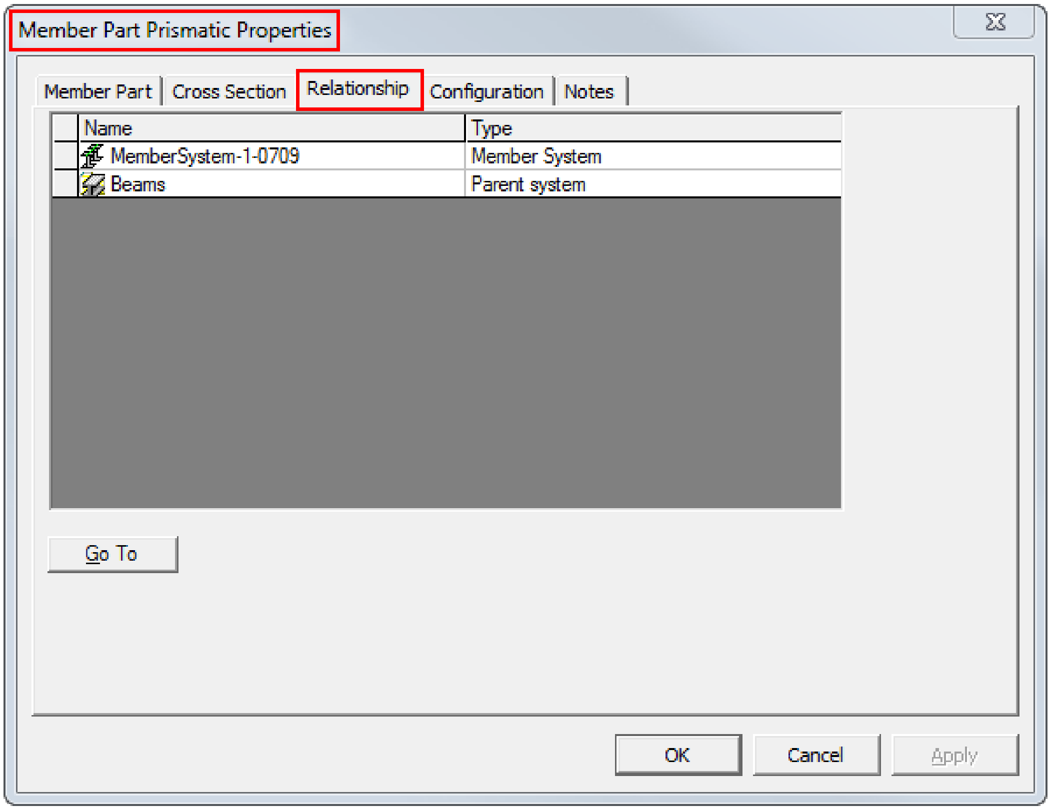
Note:

Relationship tab displays all objects related to the object for which you are viewing the properties. For example, you are viewing the properties of a member system, the related member part, parent system, frame connections, supporting members, grid line, grid plane will display on this tab. If the item is associated to WBS, project relationships will also appear on this tab.

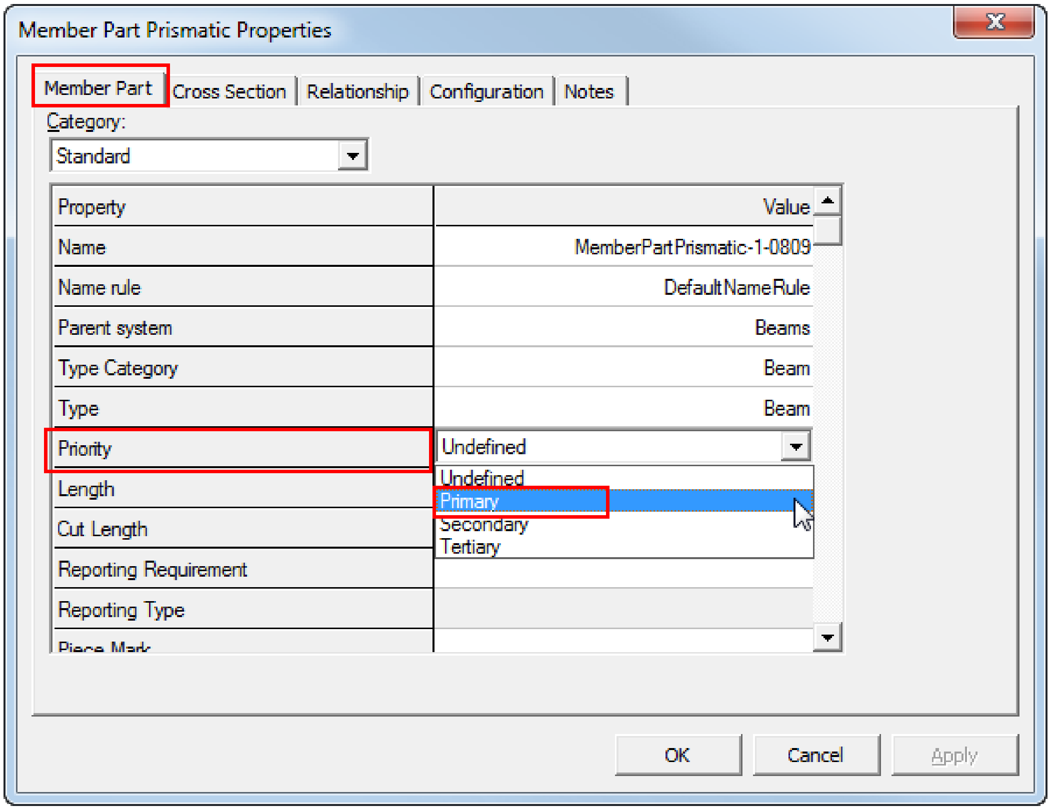
Name: Displays the name of the related object.
Type: Displays the type of related object.
Go To: Displays the properties of the selected object.

7. Click on the “Go To” option. There is a change in the properties window. “The Member System Prismatic Properties” has changed to “Member Part Prismatic Properties”.

Note: You may also double-click the row to invoke Go To option.

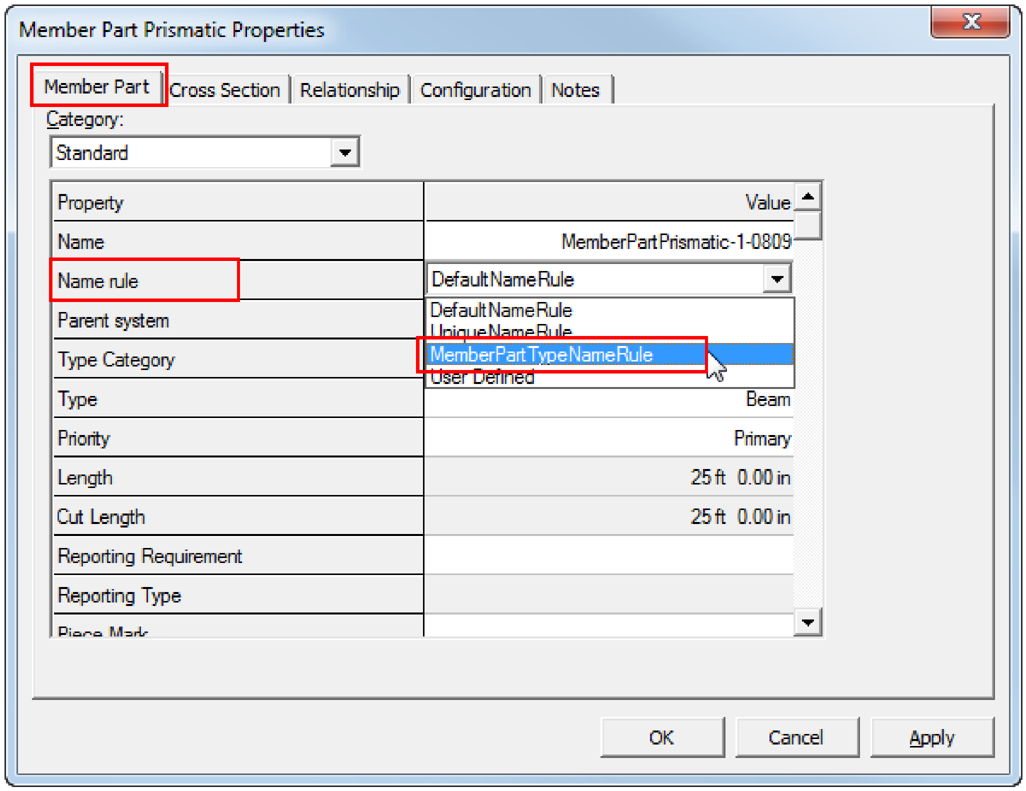


8. Switch to the “Member Part” tab and change the priority from Undefined to Primary.

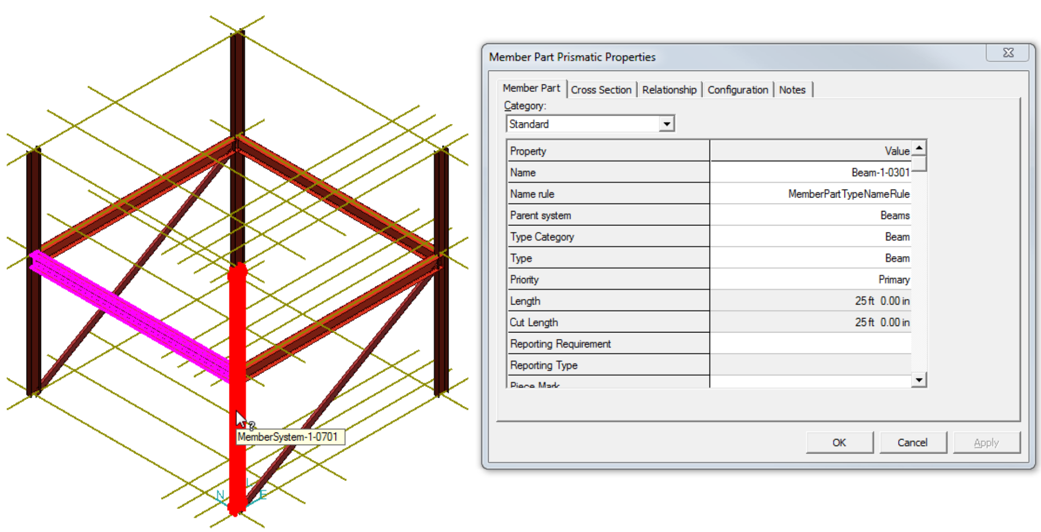


9. Click Apply.

10. Click on the “DefaultNameRule” as shown in the image above. A pull down menu will appear. Change the name rule to “MemberPartTypeNameRule” and click Apply.



11. The name changes to “Beam-X-XXXX”. Please refer to the note provided below for name rule functionality.



- **DON’T** CLICK ON OK.

Note:

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. There are different types of name rules.

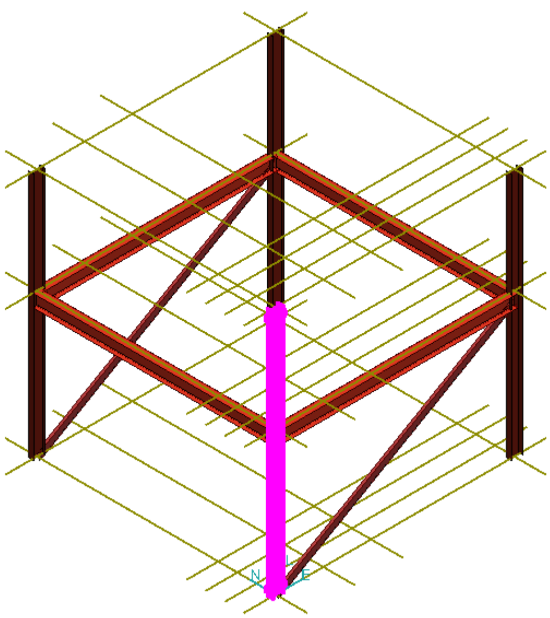
- ***Member Part Type Name Rule** - Names the member part using this method: <member type>-<location>-<index number> where <member type> is the selected Type property,<location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Beam-1-0003.*

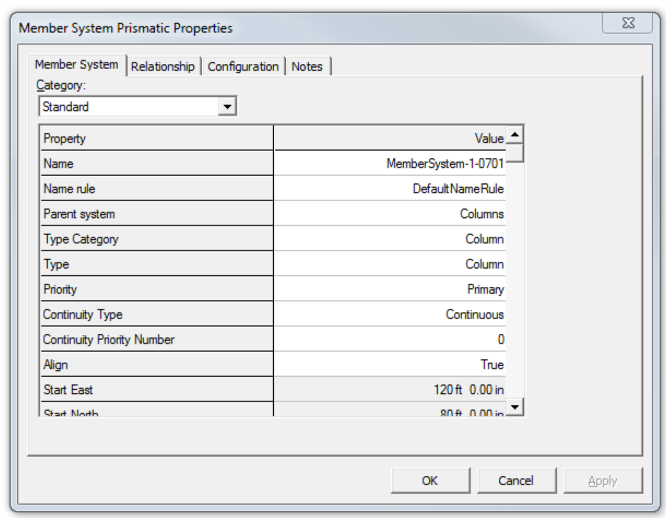
- ***Default Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.*

- ***Unique Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.*

- ***User Defined** - Select this name rule to name the member system yourself using the Name box.*

12. Move your mouse over the column as shown in the workspace.





13. Click on the column and observe the changes. The beam “Member Part Prismatic Properties” has changed to Column “Member System Prismatic Properties”. Please see the image below,

Note:

- *The locate filters was set to “Member Systems” initially. Hence, “Member System Prismatic Properties” will be shown. If the locate filter was set to “Member Part”, then “Member Part Prismatic Properties” will be shown.*
- *This way the user can view the properties of different members without closing and reopening the properties dialog box.*
- ***Set the locate filter to “All”. The user can view the properties of any object that is available in the workspace.***

14. Click Ok.

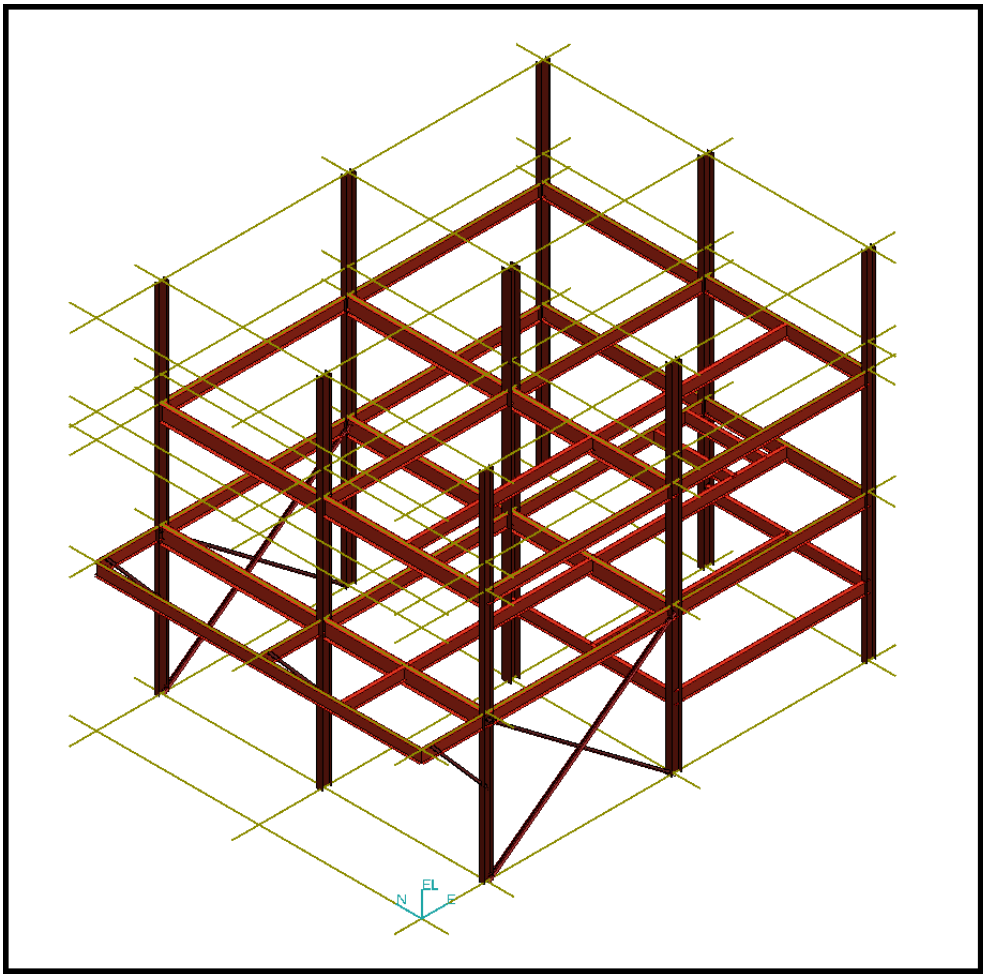
LAB-3: Structure Modeling

Objectives

After completing this lab, you will be able to:

- Understand the structure entities and relationships
- Use Place Column at Grid Intersection Command
- Use Place Split Command
- Use Place Linear Member System command using SmartSketch service and PinPoint tool

You will build simple steel-frame two-story building as shown below:



Part I: Columns

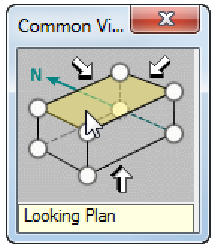
1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include the A2 -> U03 and CS -> U03 CS systems.

© Copyright 2014 Intergraph Corporation

2. Go to the Structure Task environment.

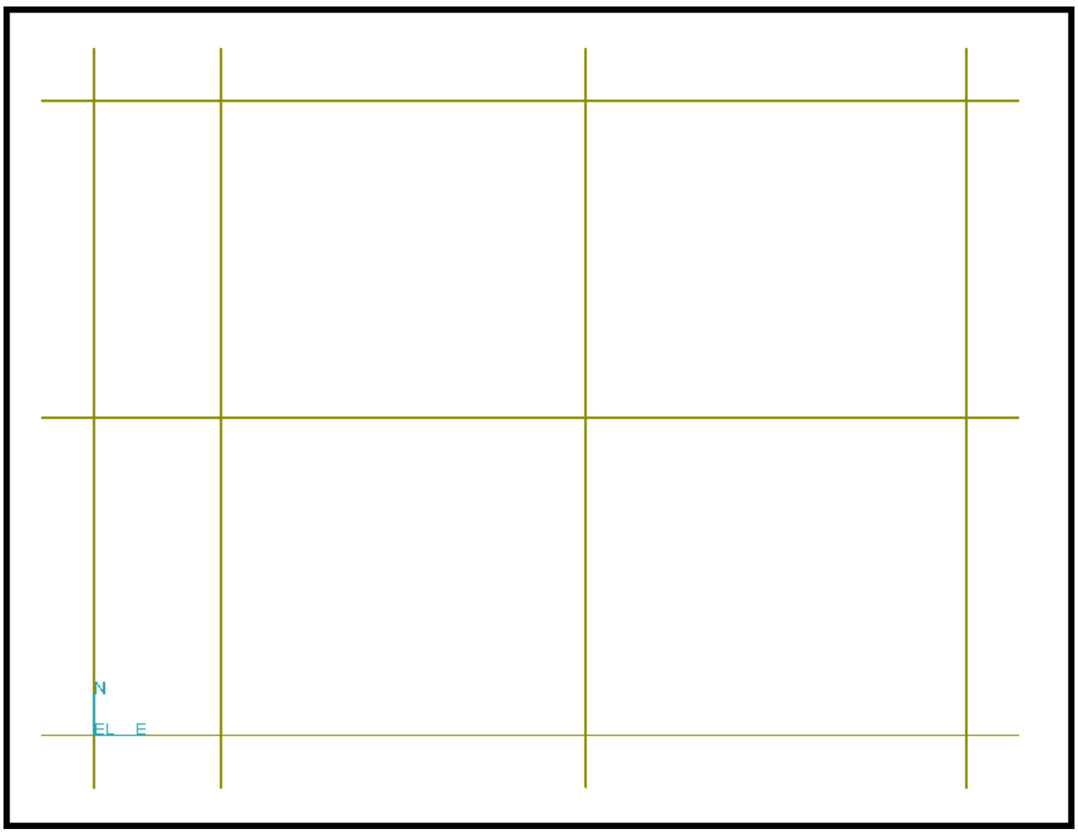
   Note: Notice that the grids you created in lab 1C are in this model. Though not absolutely necessary, grids are extremely useful as references for drawing dimensions. They also help in the modeling process by providing reference points at which you can place structural members.

3. Use the Common View control to look at Plan view. Select the top plane as indicated below.



4. Use the Fit command from the main toolbar to fit all graphics into the view.

   Your View should now resemble the following graphic.



5. Select Place Columns at Grid Intersections command. System displays the smart step ribbon bar.
6. Set the following parameters:

   System: A2 -> U03 -> Structural -> Columns
   Type Category: Column
   Type: Column
   Section Name: W14x53
   Cardinal point: 5
   Angle: 0 deg

© Copyright 2014 Intergraph Corporation

Reflect: off
Material: Steel- Carbon
Grade: A36

7. Use Workspace Explorer and select Elevation 0'-0" object for the first SmartStep.
8. Use Workspace Explorer and select Elevation 44'-0" object for the second SmartStep.
9. Place a fence around the floor grid intersections as shown in figure 1.
10. Click "Finish" button to place the columns.

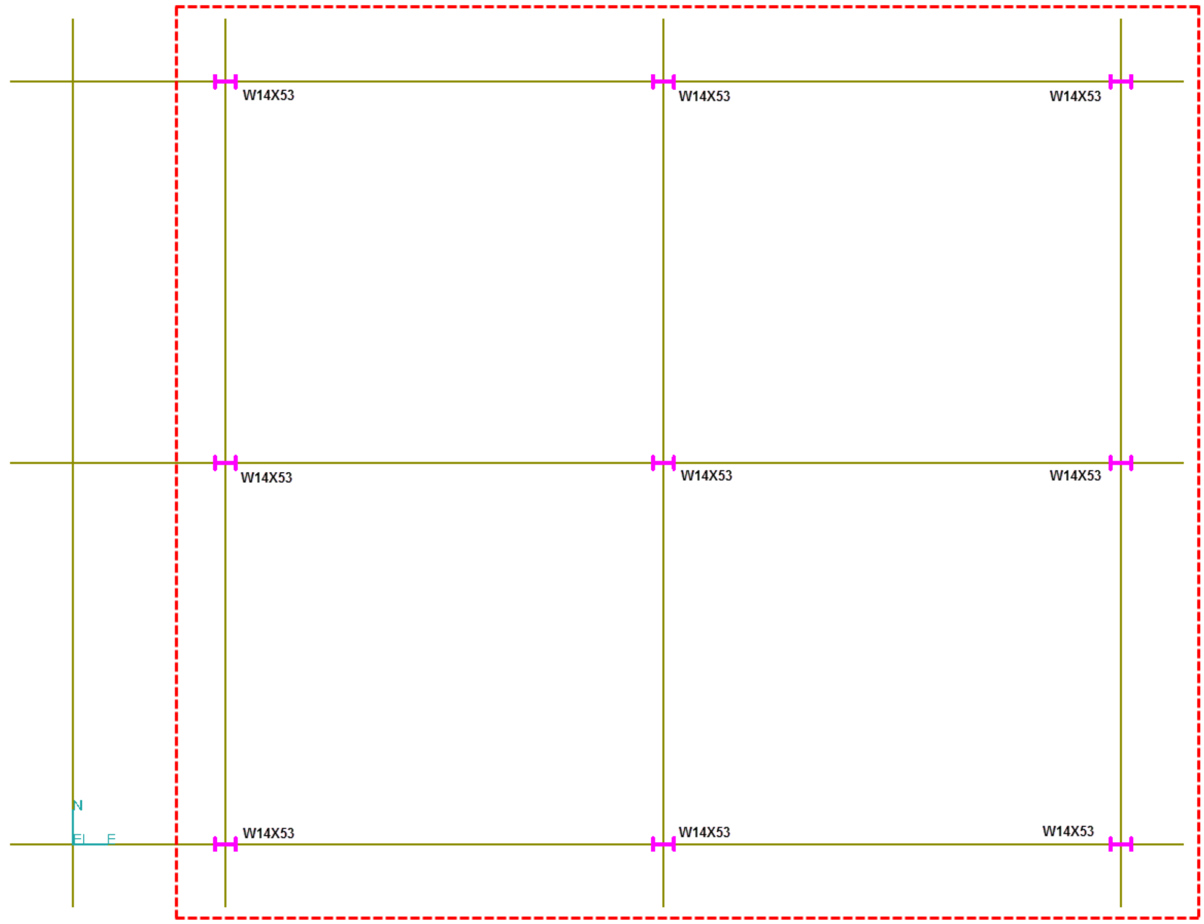
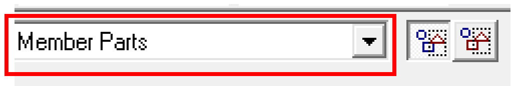


Figure 1 - Floor Plan

Change the column section size and orientation



11. Set the locate filter to Member Parts.
12. Multi-select the appropriate columns and use the SmartStep ribbon bar to change section size and the orientation as shown in figure 2.

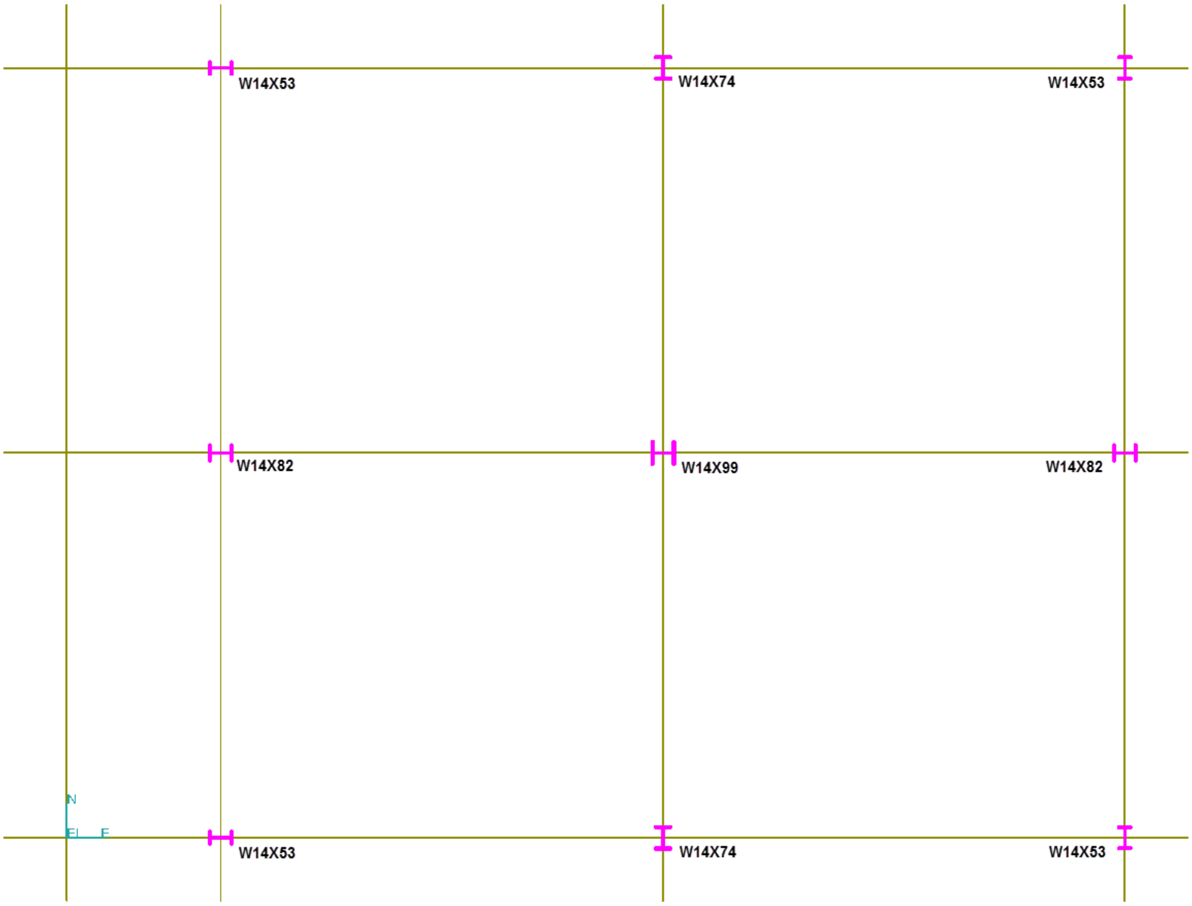


Figure 2 - Floor Plan

<u>Place split connections at elevation 34’-0” using the Place Split command</u>

13. Select Place Split command.
14. Make sure the placement type is set to Place: By Object Split Status is set By Rule.
15. Select 34’-0” Elevation Plane object in the Workspace Explorer or the Ruler.
16. Select all columns to split and click “Finish” button. Figure 3 shows Split connections highlighted.

© Copyright 2014 Intergraph Corporation

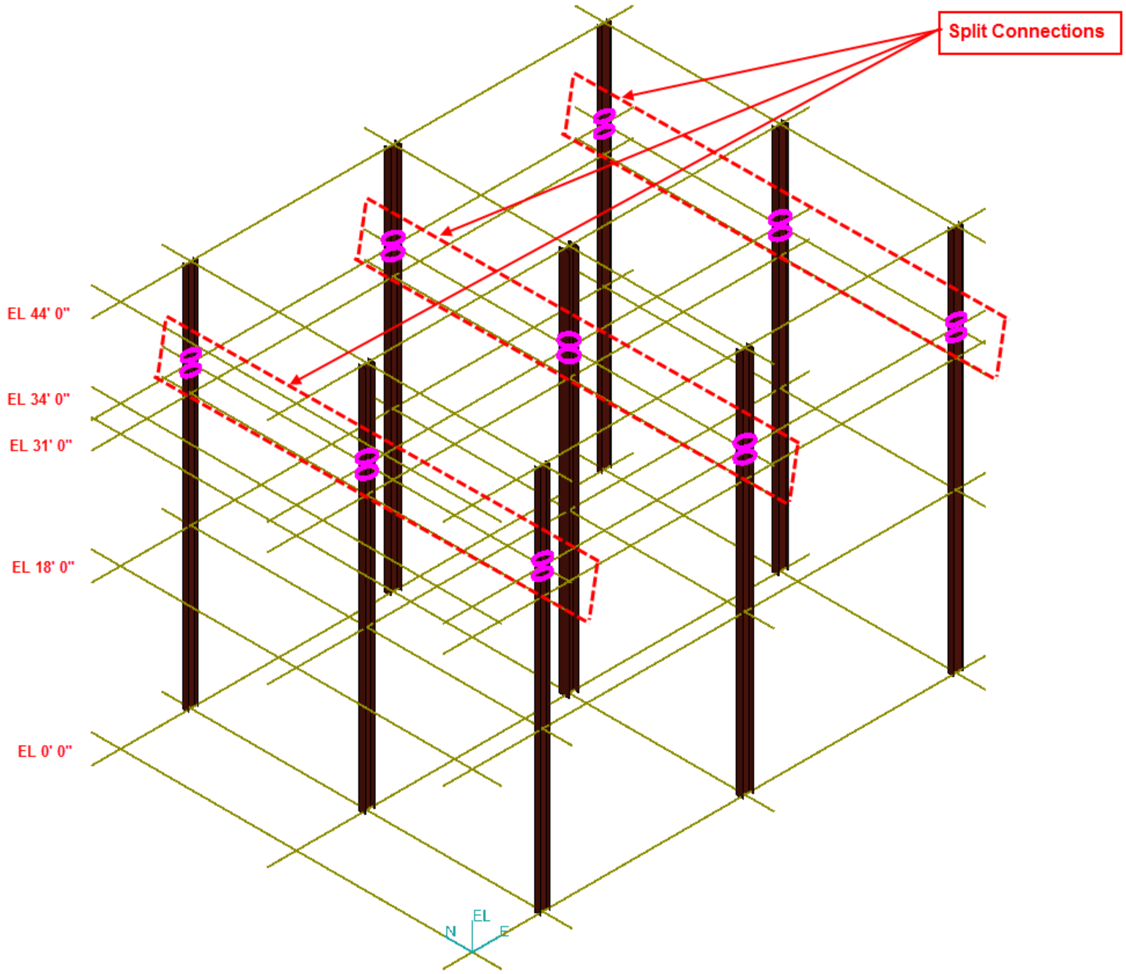


Figure 3 – Isometric View of the Structure U03

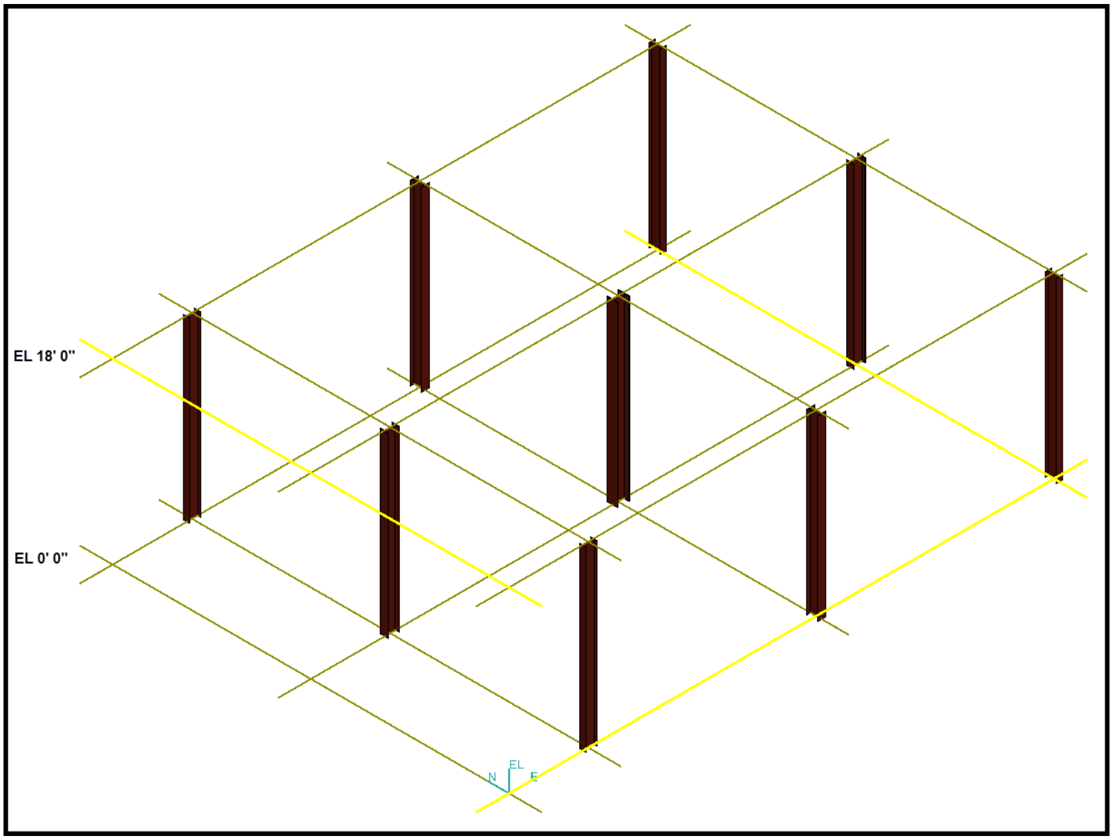
17. Save your session. Select File -> Save.

© Copyright 2014 Intergraph Corporation

Part II: First Floor Frame

18. Use Clip by Object command to isolate the gridlines and the columns in the first floor as shown below:

    Hint: Use the appropriate gridlines (EL 0'-0" and EL 18'-0") to define the clipping volume.



19. Select Place Linear Member System command to place the beams as shown in figure 4. Use following properties:

Connection:	By Rule
Type Category:	Beam
Type:	Beam
Cardinal point:	8
Angle:	0 deg
Material:	Steel- Carbon
Grade:	A36

Place all perimeter members in
System: A2 -> U03 -> Structural -> Beams.

Place all intermediate frame members in

System: A2 -> U03 -> Structural -> Horizontal Braces.

© Copyright 2014 Intergraph Corporation

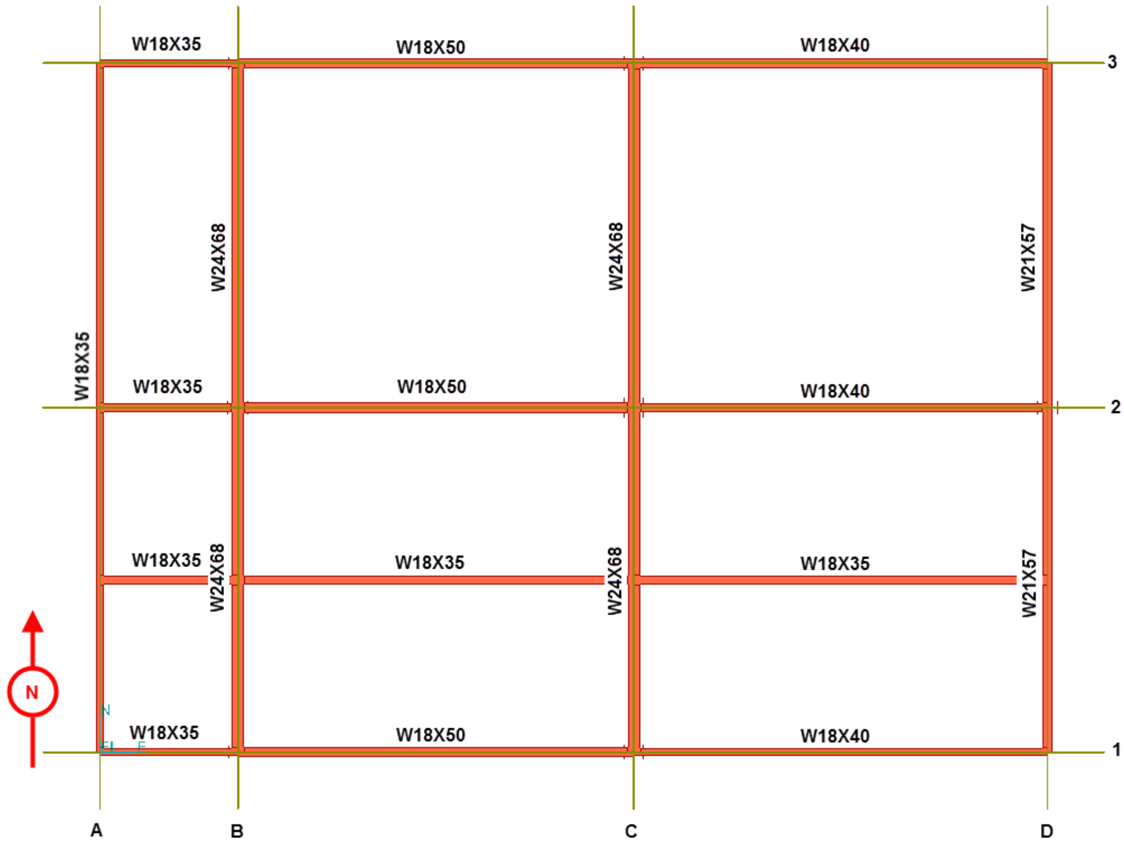
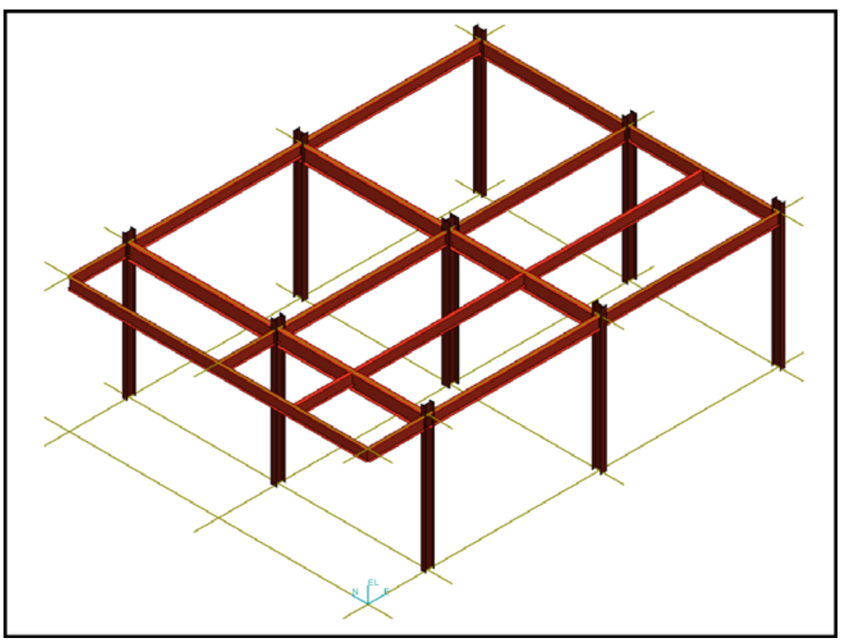


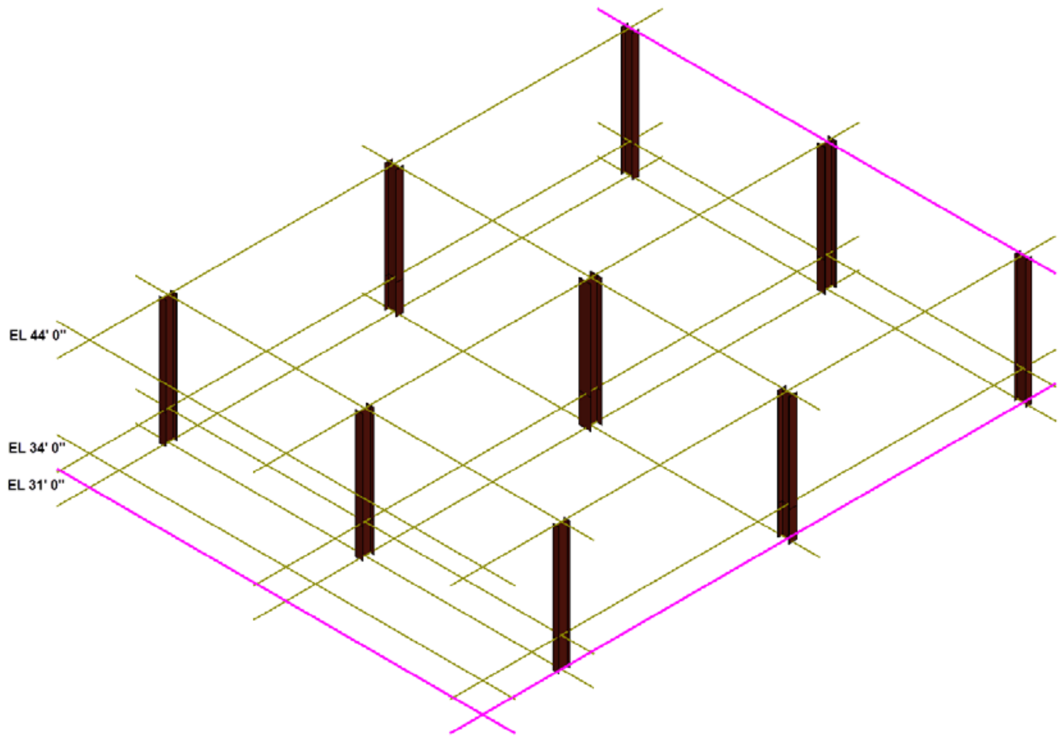
Figure 4 – First Floor Plan



© Copyright 2014 Intergraph Corporation

Part III: Second Floor Frame

20. Select View-> Clear Clipping to remove the clipping volume.
21. Use Clip by Object command to isolate the gridlines and the columns in the second floor as shown below



22. Select Place Linear Member System command.
23. Use the Second Floor Plan as shown in Figure 5 to place the beams at elevation 31'- 0".

    Use following properties:

Connection:	By Rule
Type Category:	Beam
Type:	Beam
Cardinal point:	8
Angle:	0 deg
Material:	Steel- Carbon
Grade:	A36

Place all perimeter beam in

A2 -> U03 -> Structural -> Beams.

Place all intermediate frame members in

A2 -> U03 -> Structural -> Horizontal Braces.

© Copyright 2014 Intergraph Corporation

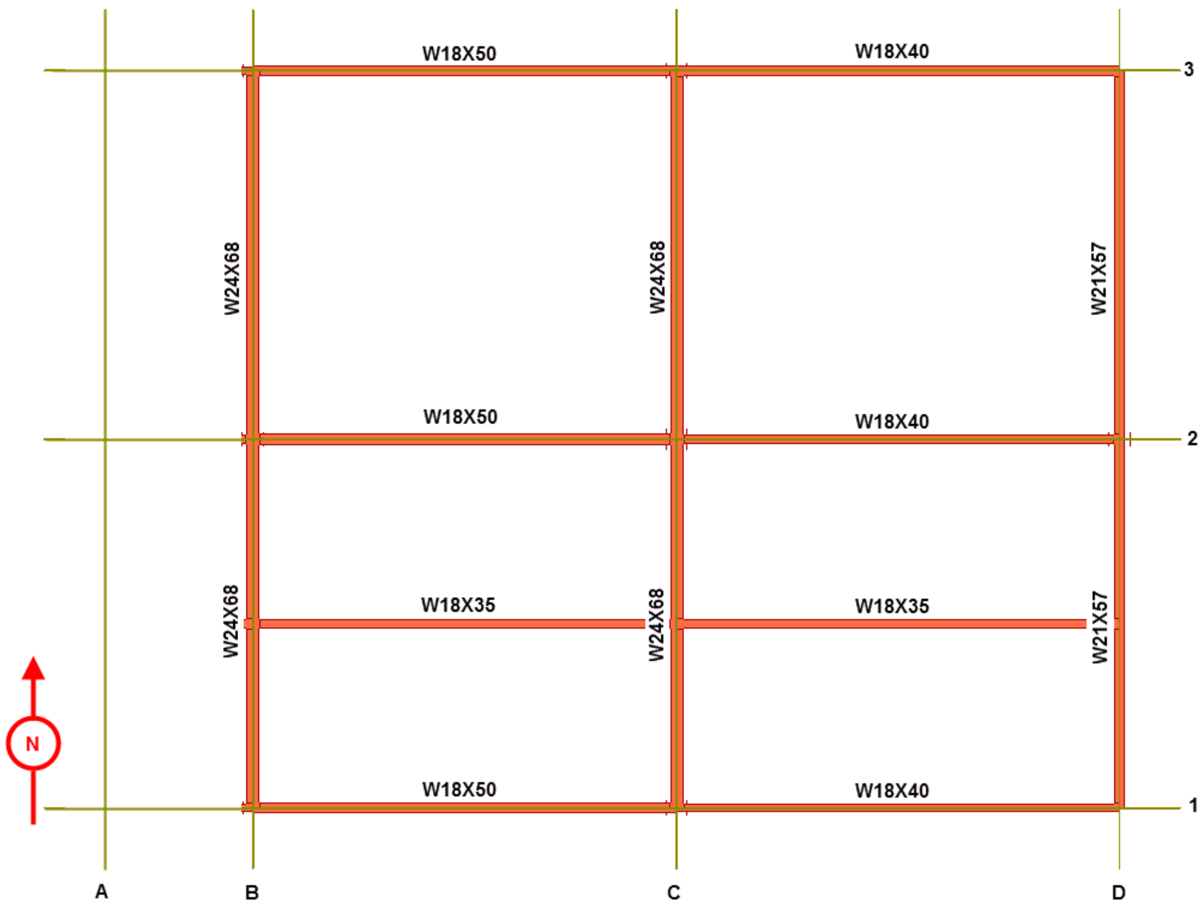


Figure 5 – Second Floor Plan

24. Select View -> Clear Clipping to remove the clipping volume.

    Your View should now resemble the following graphic:

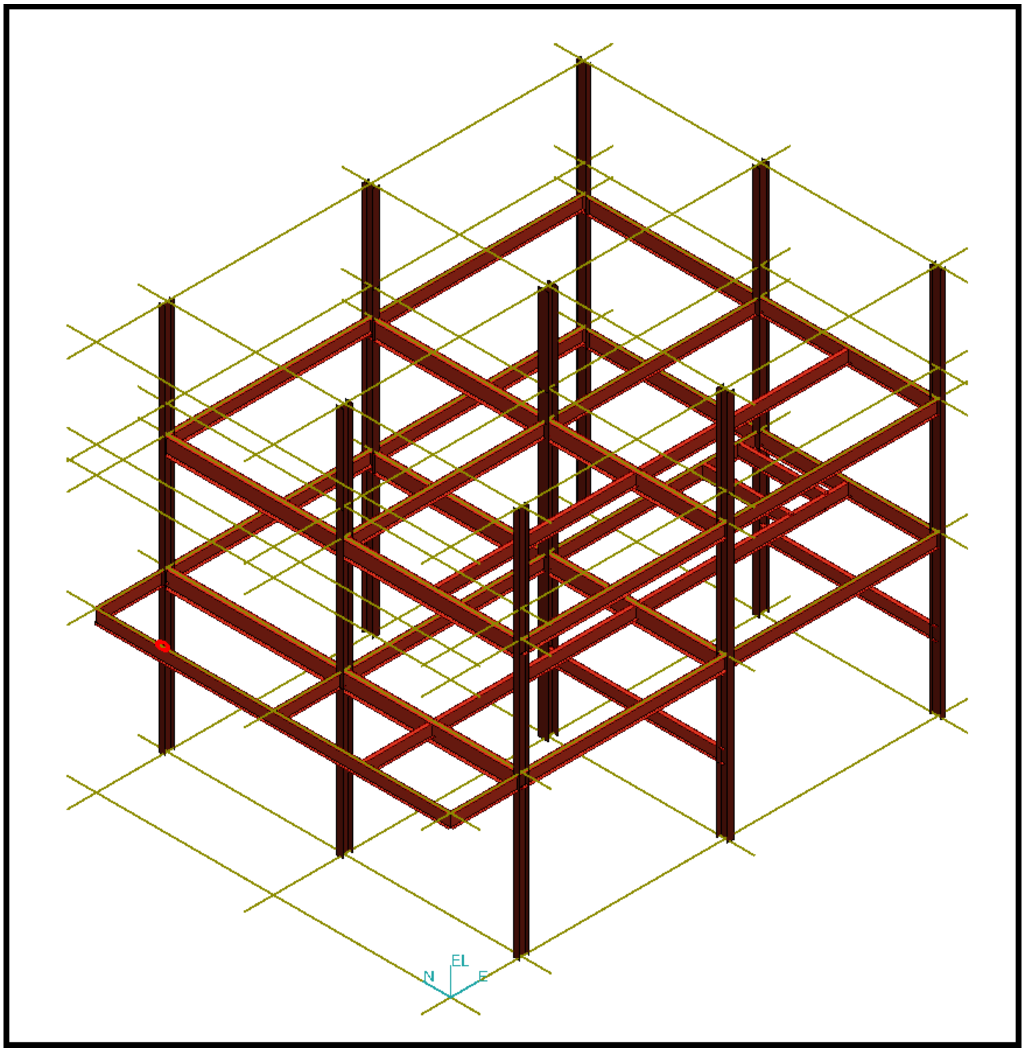


Figure 6 – Isometric View of the Structure U03

25. Save your session. Select File -> Save.

© Copyright 2014 Intergraph Corporation

Part IV: Vertical Braces

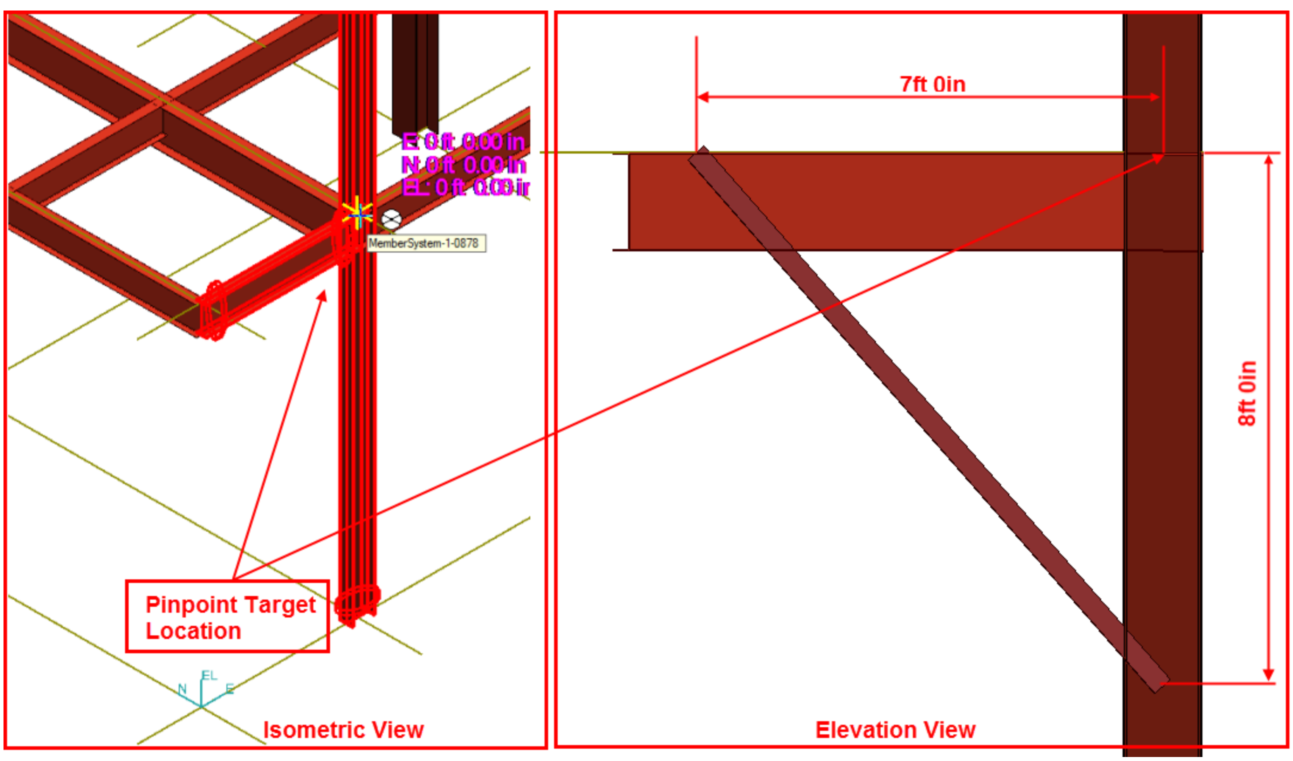
1. Activate the PinPoint ribbon bar by Selecting Tools → PinPoint (make sure Active Coordinate System is set to Rectangular Coordinate mode and Global)

2. Select Place Linear Member System Command to place the supported brace.

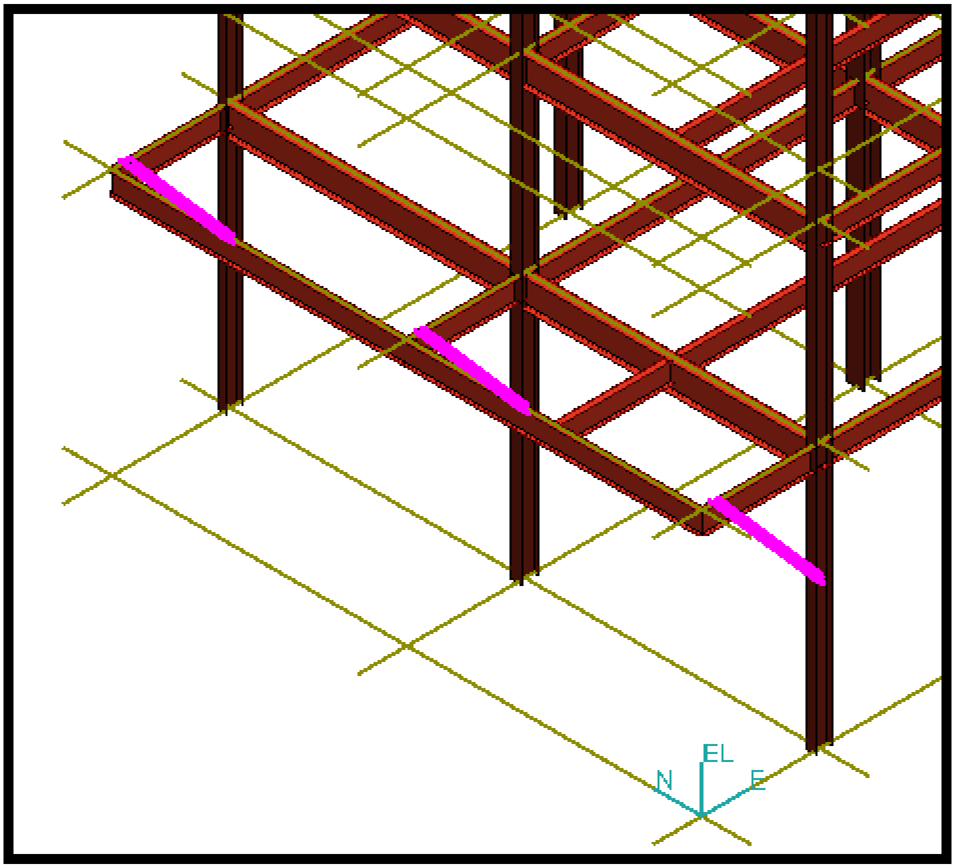
   Note: Use Pinpoint tool to place supported bracing at a given distance. Place the PinPoint target at the intersection of the column and the beam.

3. Set the parameters as

Connection:	By Rule
System:	A2 → U03 → Structural → Vertical Braces.
Type Category:	Brace
Type:	Vertical Brace
Section Name:	2L4X4X1/4X3/8
Cardinal point	5
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36



4. Use the Copy/Paste functionality to place the other two supported braces as shown below:

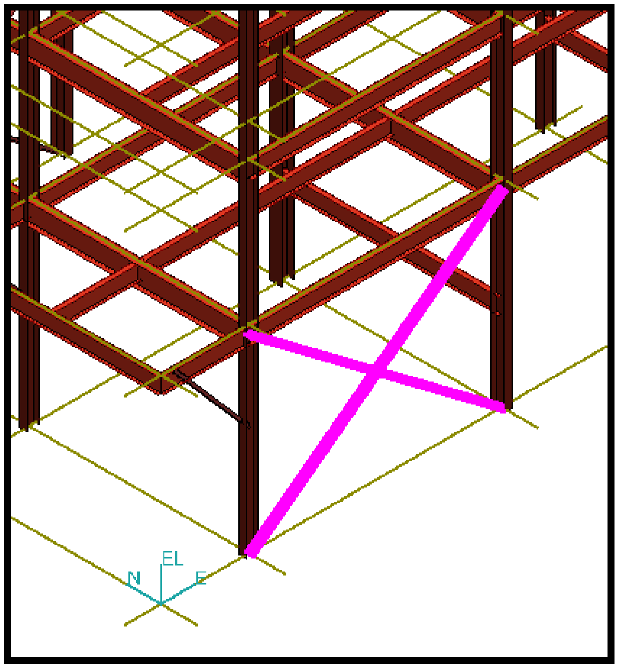


5. Use Place Bracing Command to place cross bracings on the Structure U03.
6. Use Place Bracing ribbon bar to set the active member parameters as follows:

Bracing Type:	Cross
Connection:	By Rule
System:	A2→U03 → Structural →Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	2L4X4X1/4X3/8
Cardinal point	5
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36

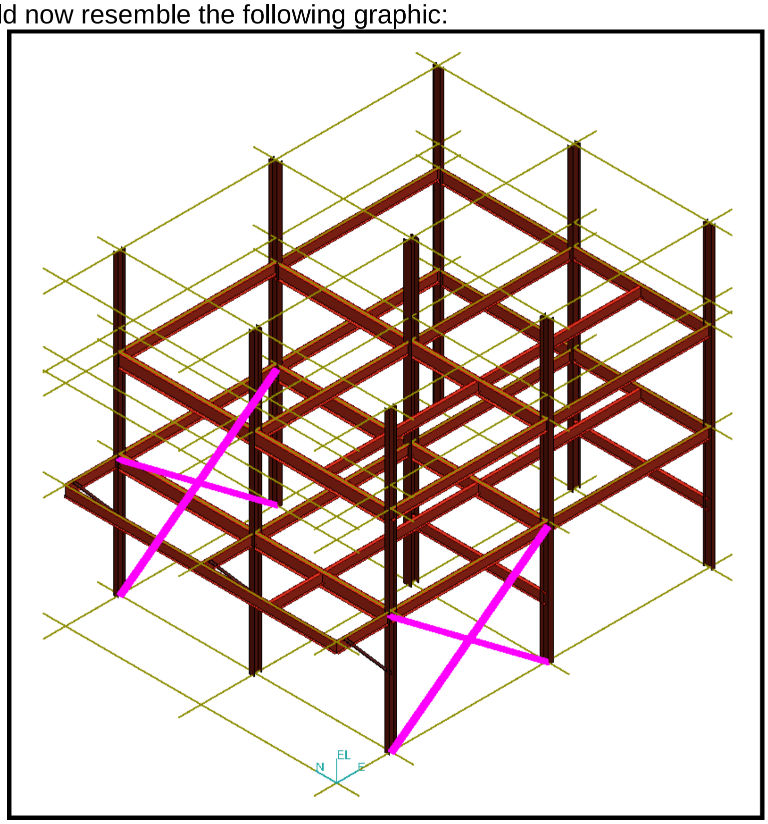
7. Select first column for the first SmartStep (See figure 7)**.**
8. Select second column for the second SmartStep.
9. Select Beam for the third SmartStep.
10. Select “Finish” button to commit the transaction.

© Copyright 2014 Intergraph Corporation



11. Repeat the above steps to place the other cross bracing.

    Your View should now resemble the following graphic:



Part V: Horizontal Braces

Place two horizontal beams on the first floor frame as shown in Figure 9.

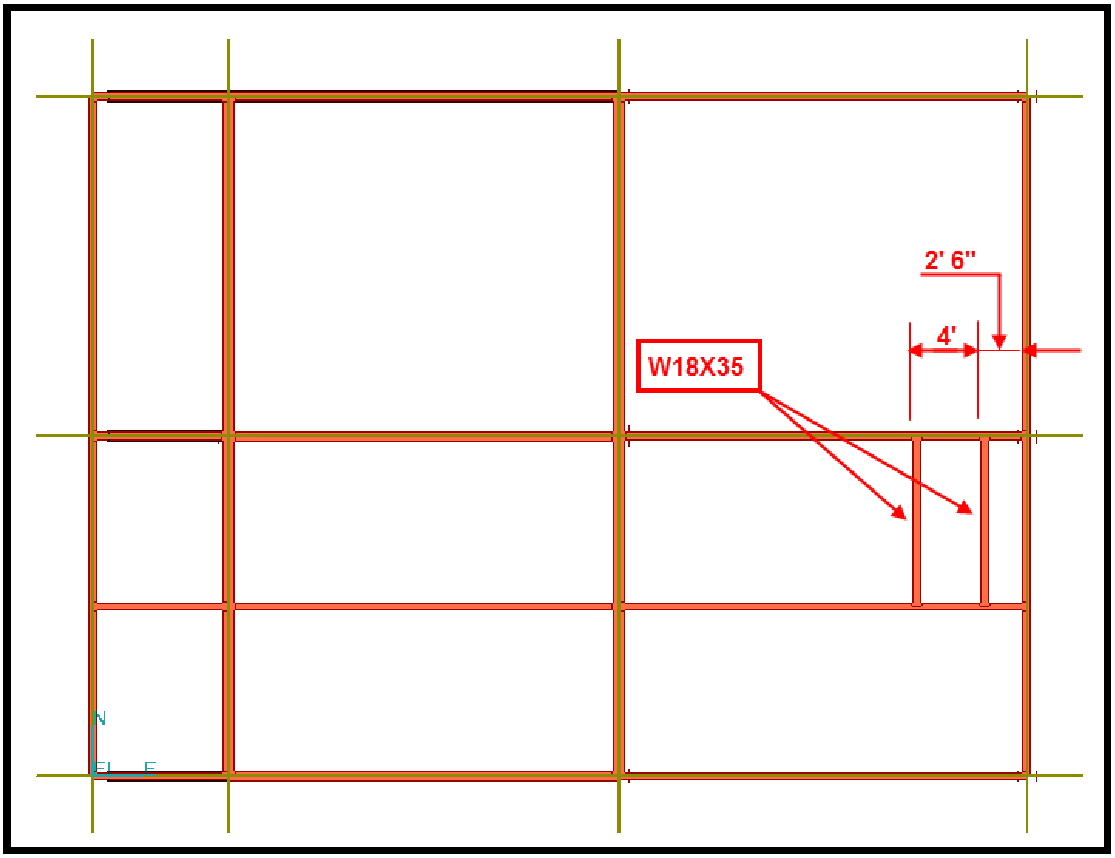
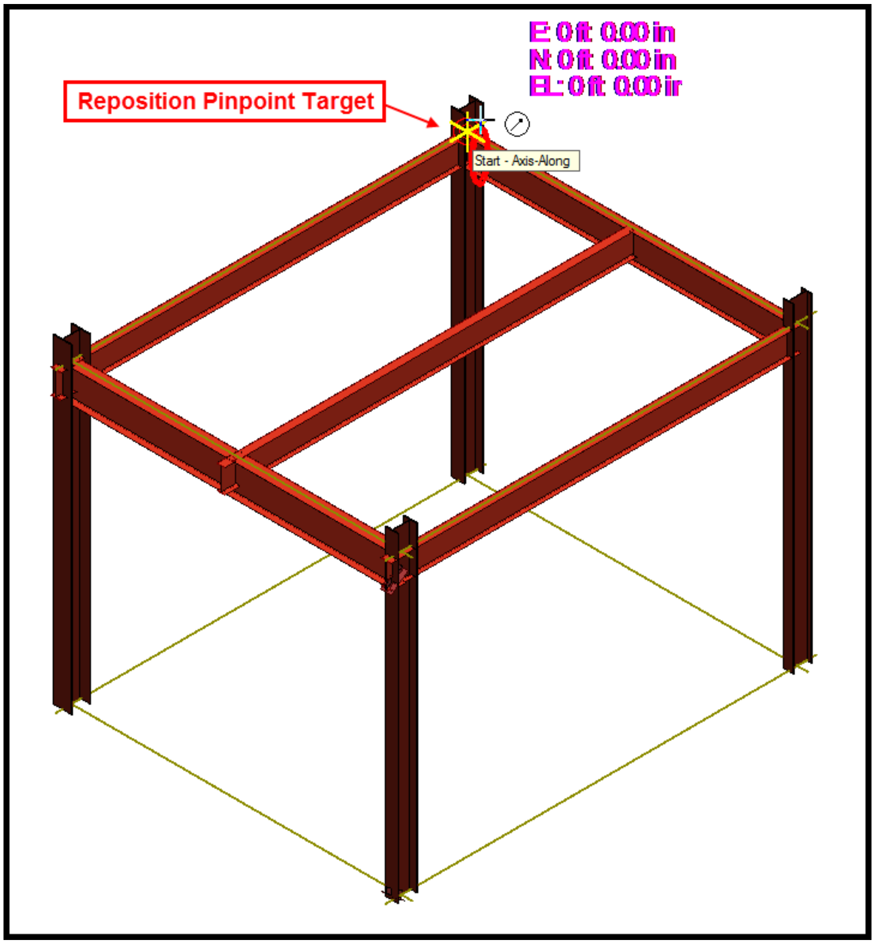


Figure 9 – First Floor Frame El –18' 0"

Note: Use Pinpoint tool to place framing members at a given distance. Place the PinPoint target at the intersection of the column and the beam.

1. Activate the PinPoint ribbon bar by Selecting Tools > PinPoint (make sure Active Coordinate System is set to Rectangular Coordinate mode).
2. Place the PinPoint target at the intersection of the column and the beam.
3. Use Clip by Object command to isolate the beams and the columns in the first floor as shown below:

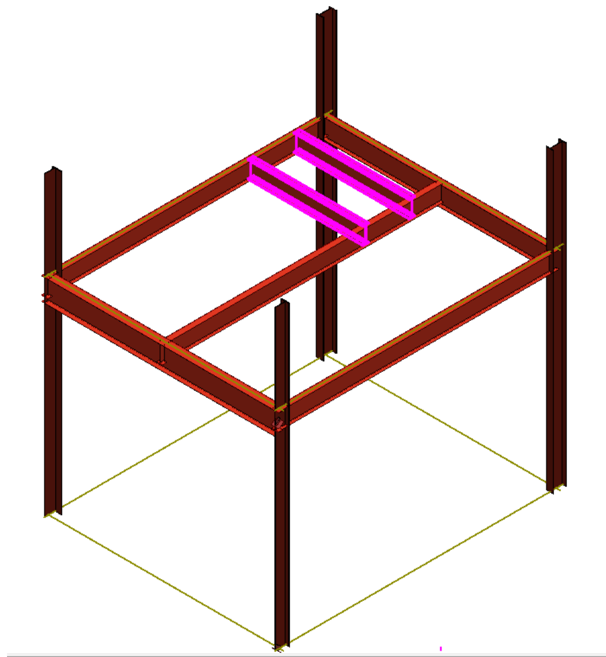


4. Select Place Linear Member System Command
5. Set the parameters as

Connection:	By Rule
Type Category:	Beam
Type:	Beam
Section Name:	W18X35
Cardinal point	8
System:	A2-> U03 -> Structural -> Horizontal Braces.
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36

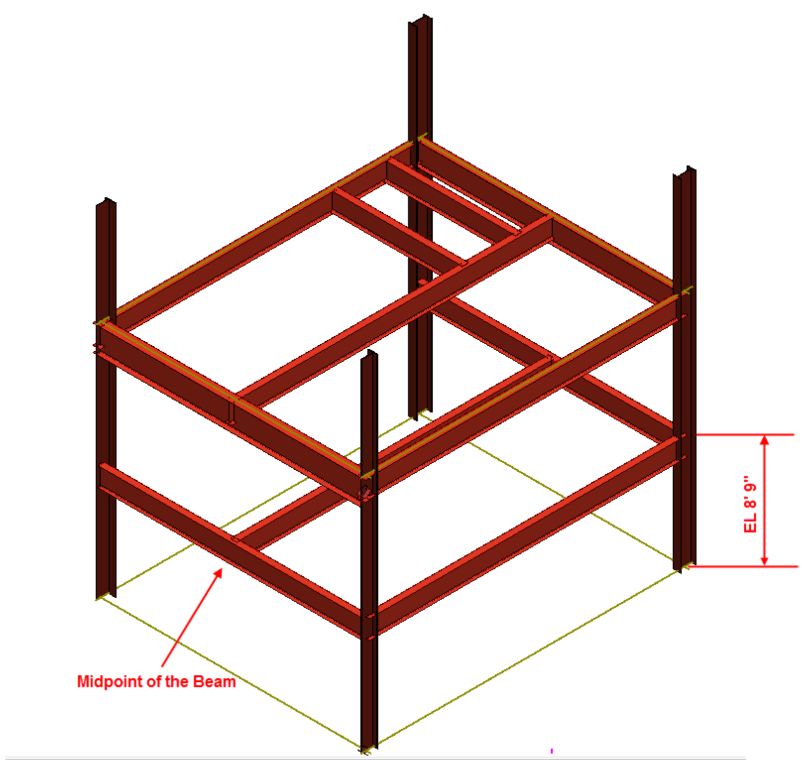
6. Use SmartSketch service to locate points along the geometric of the beam and the PinPoint key-ins to place End 1 and End 2 of the members at the given distance.

   Your View should now resemble the following graphic:



Part VI: Horizontal Beams on the Far-East Bay

Place three horizontal beams on the far-east bay of the building as shown in Figure 10.

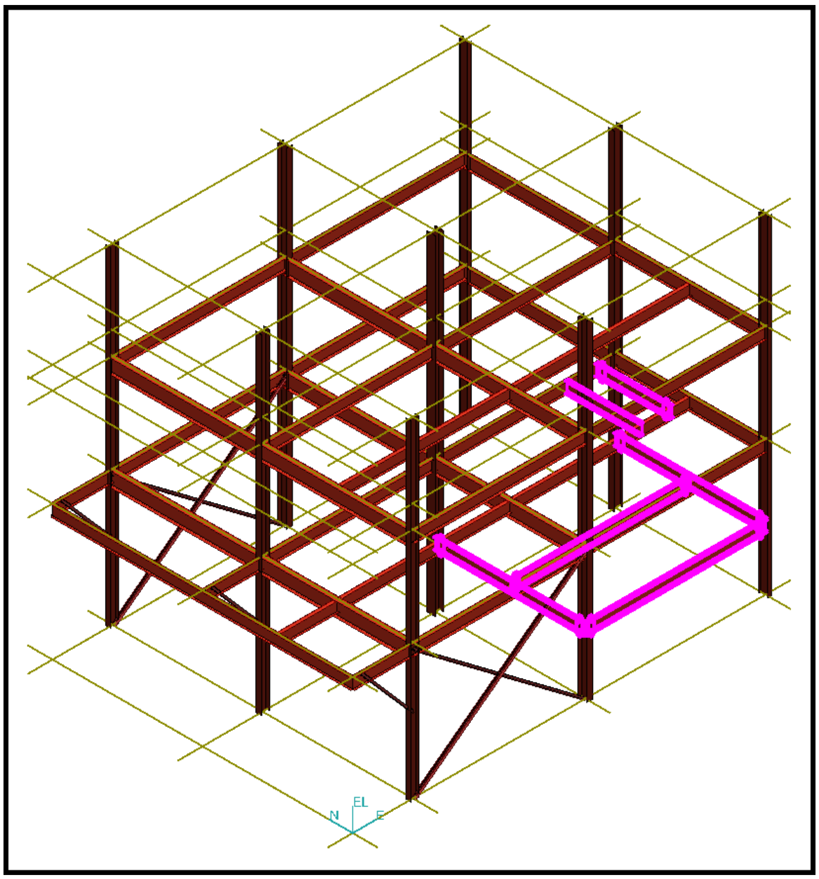


1. Place PinPoint target at the end of the column.

2. Select Place Linear Member Command.
3. Set the parameters as

Connection:	By Rule
Type Category:	Beam
Type:	Beam
Cardinal point	8
System:	A2 -> U03 -> Structural -> Horizontal Braces.
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36

4. Use SmartSketch service to locate points along the geometric of the member and the PinPoint key-ins to place End 1 and End 2 of the members at the given distance.
5. Select View-> Clear Clipping to remove the clipping volume.
Your View should now resemble the following graphic:



6. Save your session. Select File -> Save.

© Copyright 2014 Intergraph Corporation

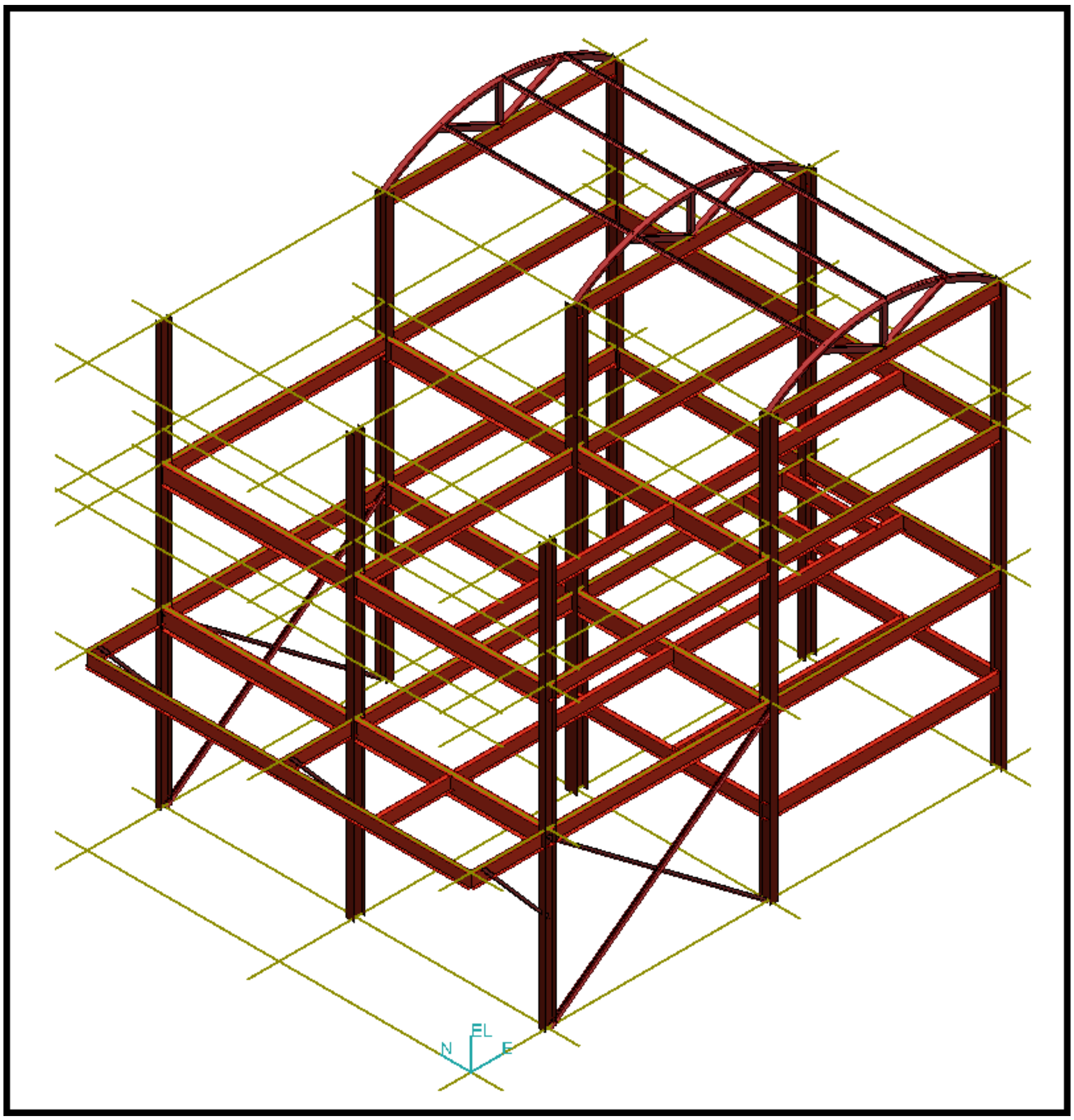
LAB-4: Curve Member System

Objectives

After completing this lab, you will be able to:

- Understand the structure entities and relationships
- Use Place Curve Member System Command
- How to use SmartSketch service to find the placement points for the members

You will add curve members and supported braces to build a rounded roof for the two-story building.



Placing Beams to support the trusses

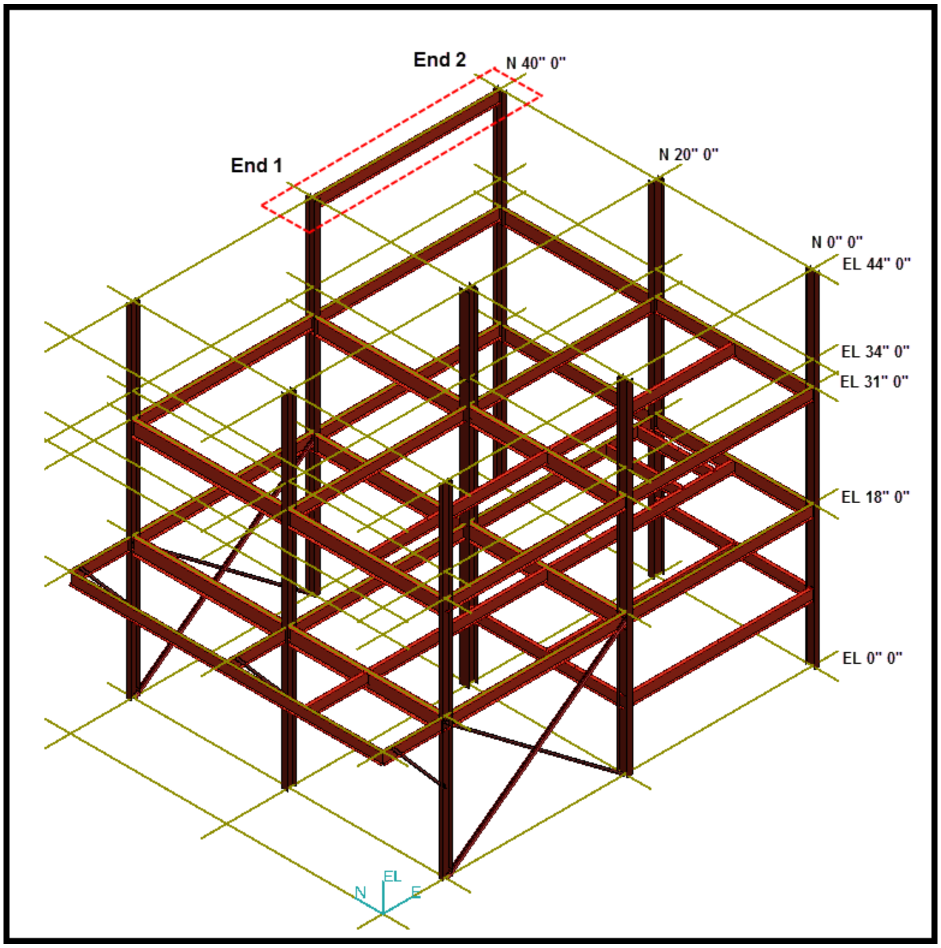
1. Select the Place Linear Member System Command.

© Copyright 2014 Intergraph Corporation

2. Set the parameters as

Connection:	By Rule
Type Category:	Beam
Type:	Beam
Section Name:	W18X35
Cardinal point	8
System:	A2-> U03 -> Structural -> Horizontal Braces.
Angle:	0 deg
Reflect:	off
Priority:	Secondary
Material:	Steel- Carbon
Grade:	A36

3. Place Member End1 at the end of the column located at North Plane 40'- 0" and Elevation 44'- 0" intersection.
4. Place Member End2 at the end of the column located at the other North Plane 40'- 0" and Elevation 44'- 0" intersection.



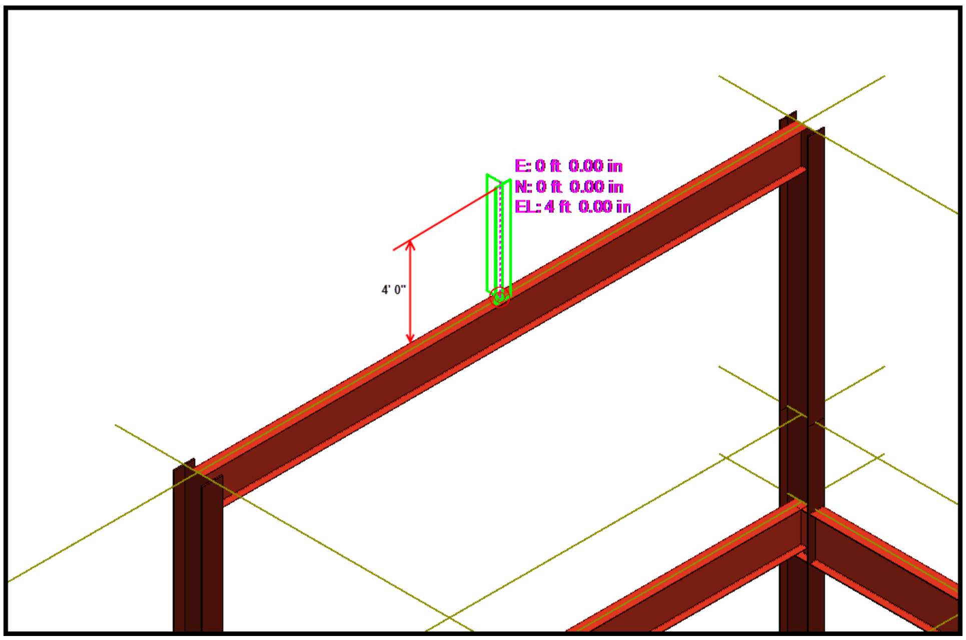
Next, you will be placing a vertical brace to support the actual curve member.

5. Select Place Linear Member System Command.
6. Set the parameters as

System:	A2->U03 -> Structural ->Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	WT8X22.5
Cardinal point	10 - Centroid
Angle:	-90 deg
Reflect:	off
Priority:	Secondary
Material:	Steel-Carbon
Material:	A36

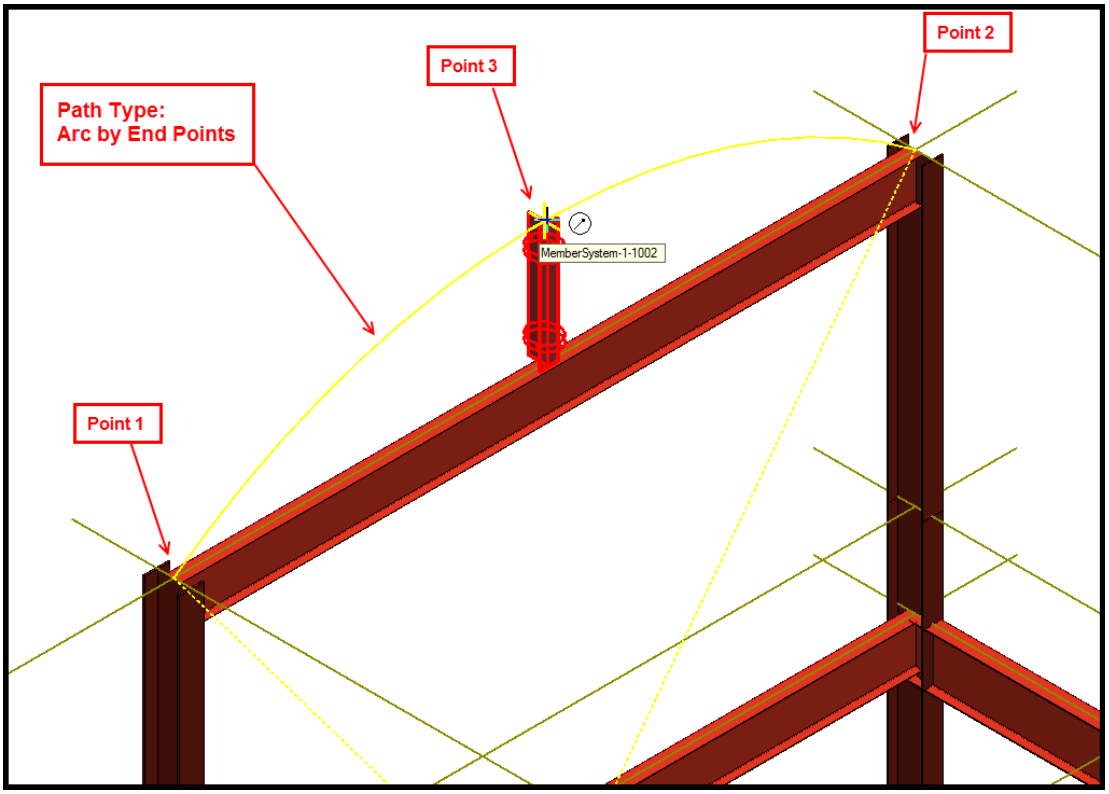
7. Use SmartSketch service to locate the mid-point geometric of the beam to place End1.
8. Use PinPoint key-ins to place End 2 at 4'- 0" up along the Z-axis.

   Your View should now resemble the following graphic:



9. Select Place Curve Member System Command. System displays the 3D Sketch Path ribbon bar.
10. Place Point 1 at the end of the column located at North Plane 40'- 0" and Elevation 44'- 0" intersection.
11. Select Path Type as Arc by End Points.
12. Place Point 2 at the end of the column located at the other North Plane 40'- 0" and Elevation 44'- 0" intersection.
13. Place Point 3 at the end of the vertical brace that you have just placed.

© Copyright 2014 Intergraph Corporation



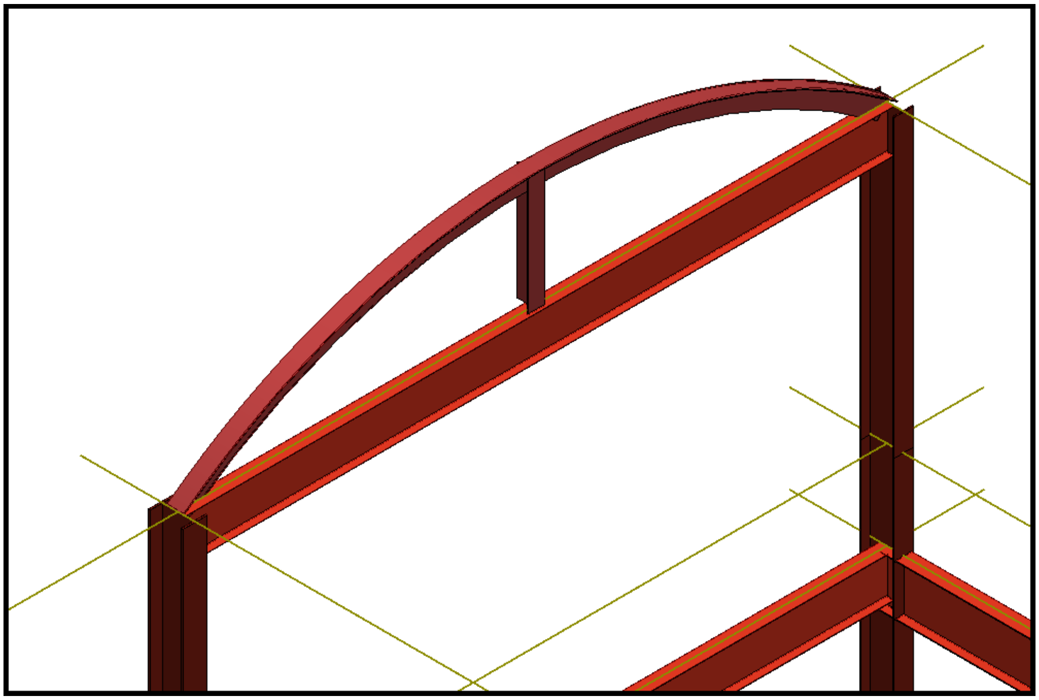
14. Hit the “Finish” Path button. System returns to the SmartStep ribbon bar.
15. Set the parameters as

System:	A2->U03 -> Structural ->Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	WT8X22.5
Cardinal point	10 - Centroid
Angle:	0 deg
Reflect:	off
Priority:	Secondary
Material:	Steel- Carbon
Grade:	A36

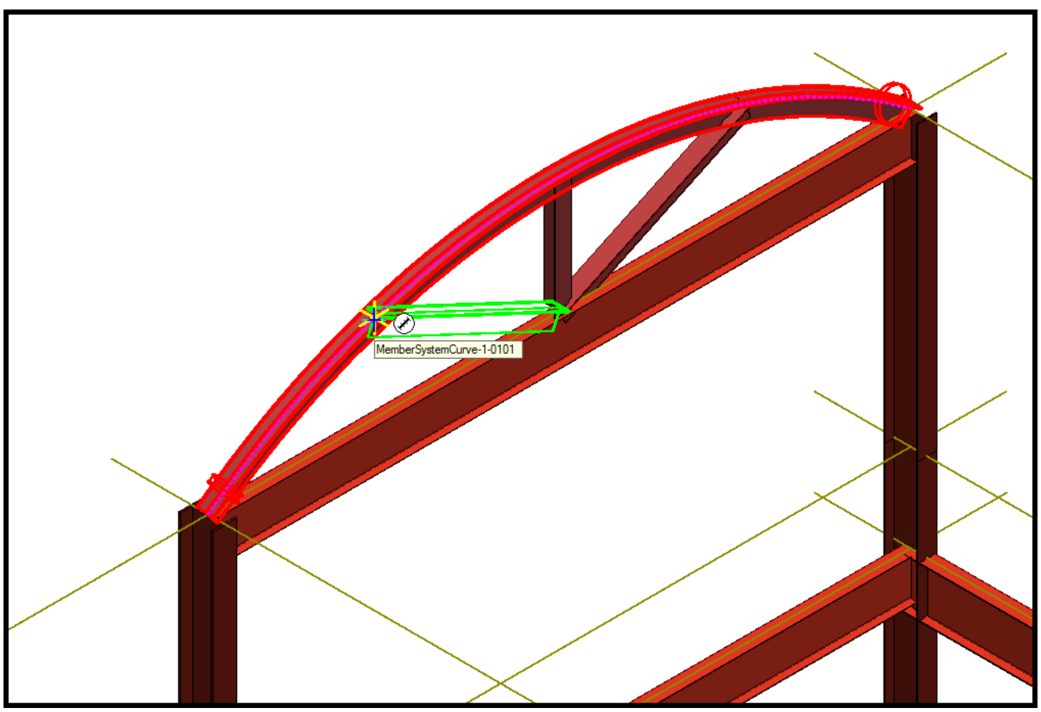
16. Hit “Finish” button to commit the transaction.

Your View should now resemble the following graphic:

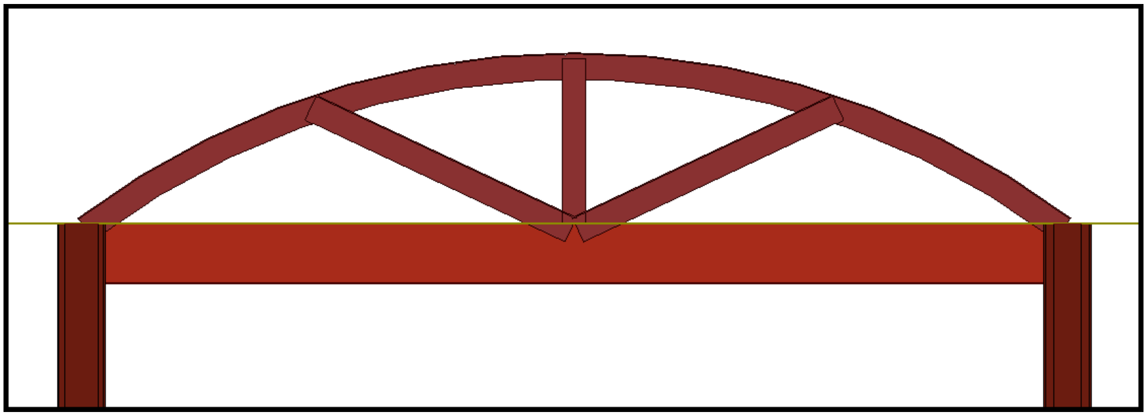
© Copyright 2014 Intergraph Corporation

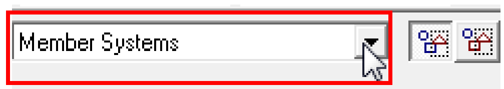


17. Place two more vertical braces from the center beam to the quarterpoints along the curve member. Use SmartSketch divisor (4) to get the correct quarter-points. See figure below:



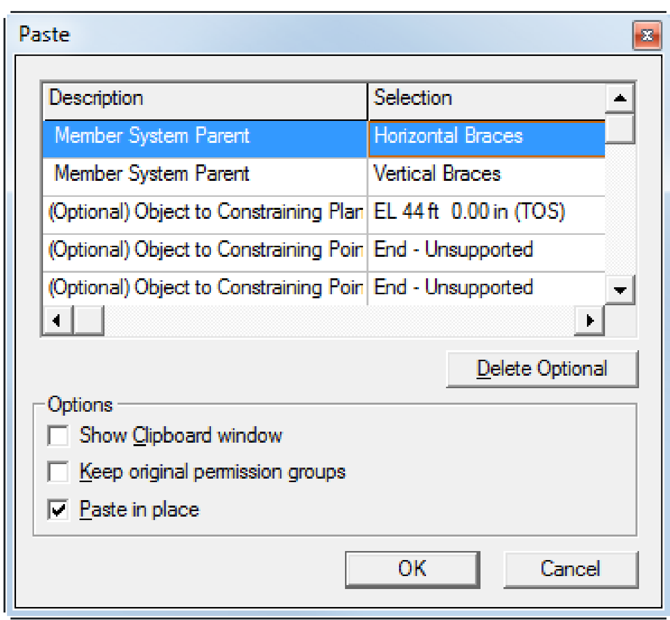
Your View should now resemble the following graphic:





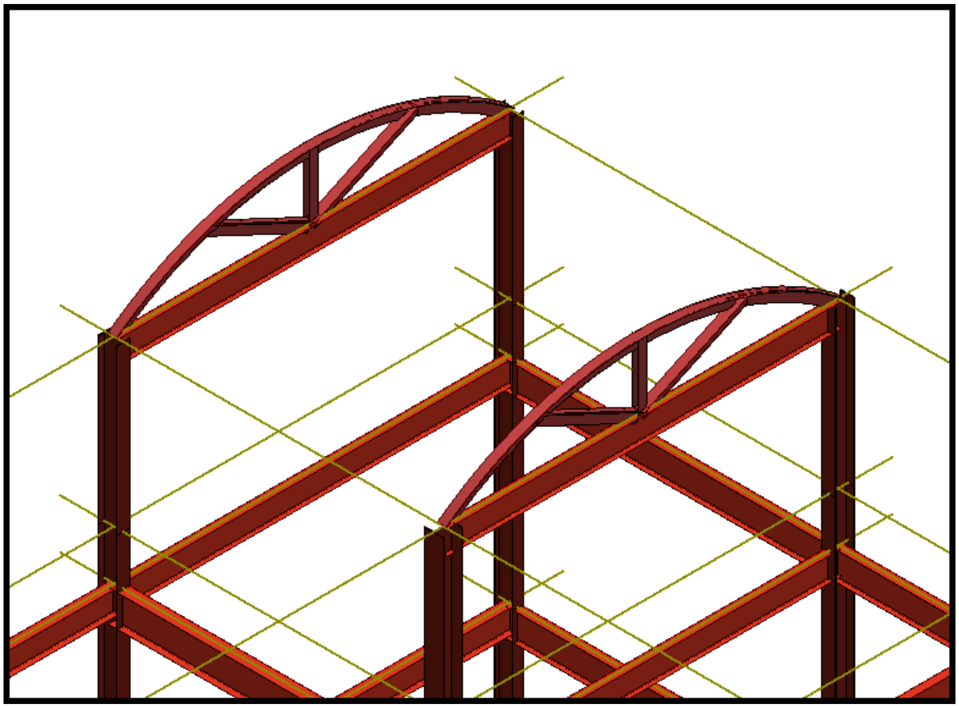
18. Set the locate filter to Member Systems.
19. Select the curve member, the support beam and the three vertical braces using the fence method or the Multi-Select method.
20. Go to the Main Menu and select Edit -> Copy.
21. Select the end of the column as the reference point.
22. Go to the Main Menu and Select Edit -> Paste.
23. Re-establish all the relationships with the columns located in North Plane 20'- 0" using the Paste dialog. See figure below.

    Note: Make sure to select the appropriate objects to re-establish the connection in the Paste Dialog box. In this case, you just need to re-establish the relationship of two frame connections and the two columns.



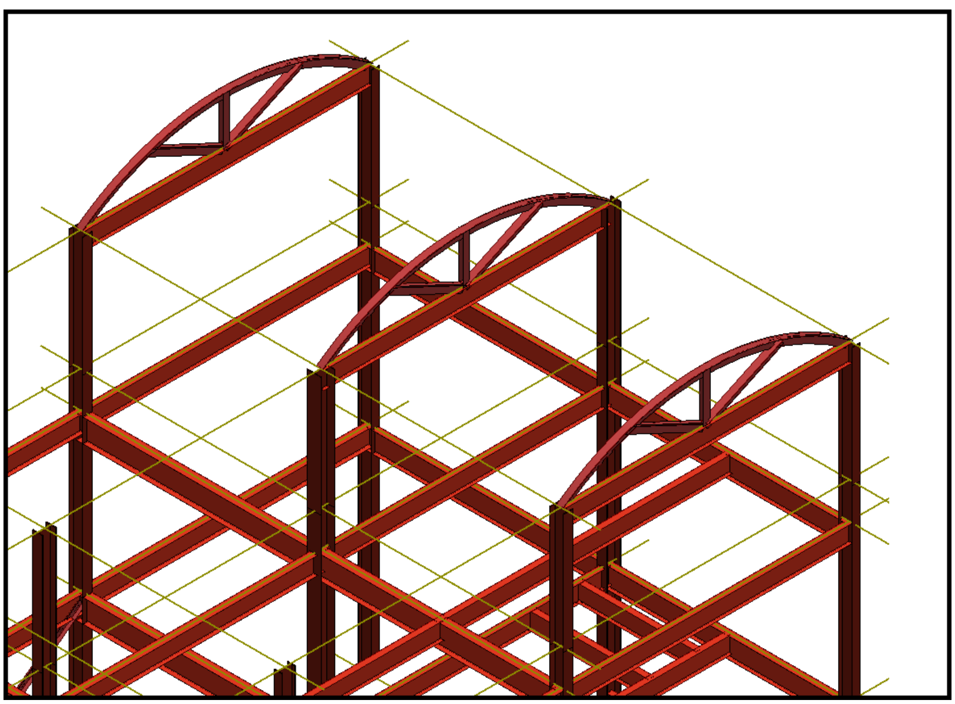
24. Hit "OK" button to commit the transaction.

© Copyright 2014 Intergraph Corporation



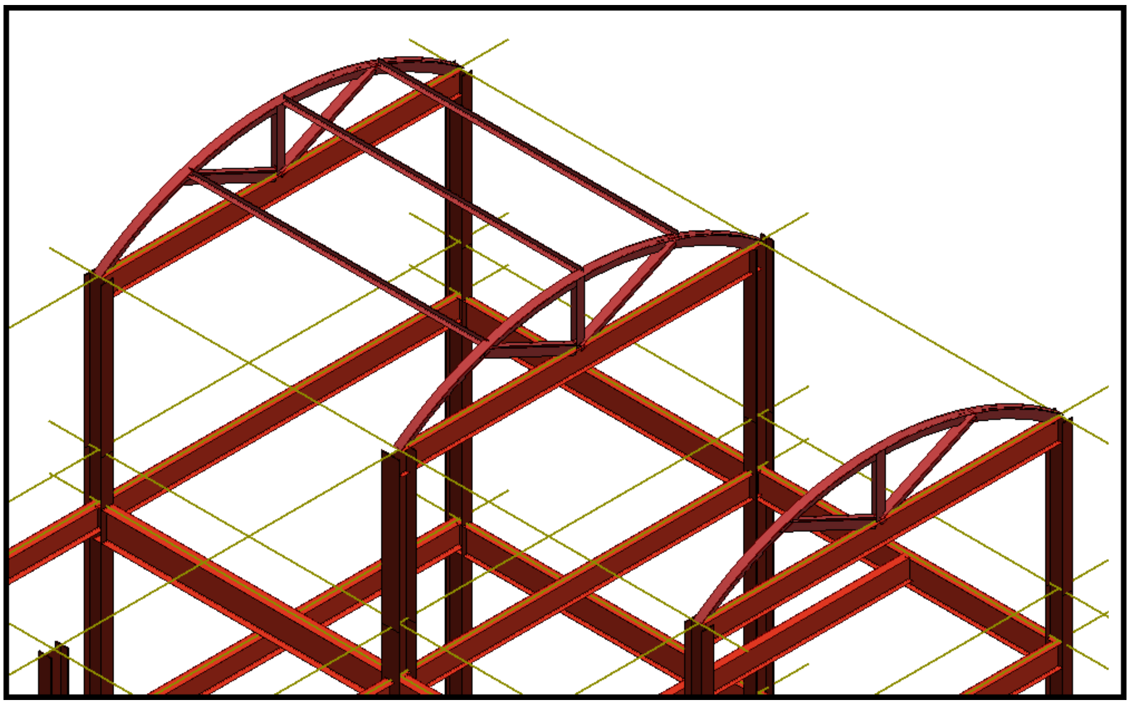
25. Repeat the Paste step to place the next truss at North Plane 0'- 0".

    Your View should now resemble the following graphic:



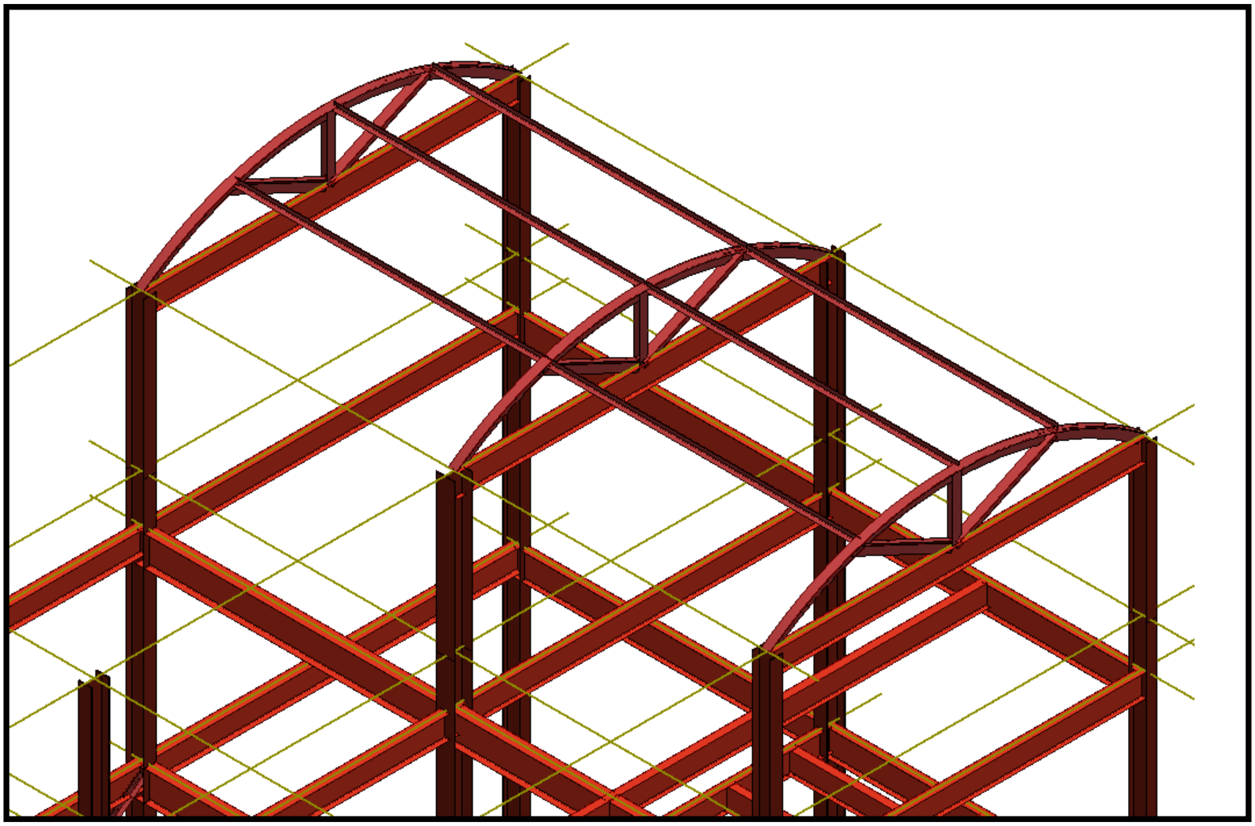
26. Select Place Framing Members command. Use the view shown in Figure 11 to place the structural members.
27. Set the active member parameters as follows:

Fit Mode:	By Count
Count:	3
Connection:	By Rule
System:	A2->U03 -> Structural -> Horizontal Braces
Type Category:	Brace
Type:	Horizontal Brace
Section Name:	L4X4X1/2
Cardinal point:	3
Reflect:	off
Offset:	0' 0"
Position:	Toggle to Skewed
Angle:	0 deg
Priority:	Secondary
Material:	Steel- Carbon
Grade:	A36



Continue using Place Framing Members command by selecting Supporting Member2 to place framing members as shown in Figure.

© Copyright 2014 Intergraph Corporation



28. Save your session. Select File → Save.

LAB 5: Centerline Aspect for Members

Objective:

By the end of this session, you will be able to:

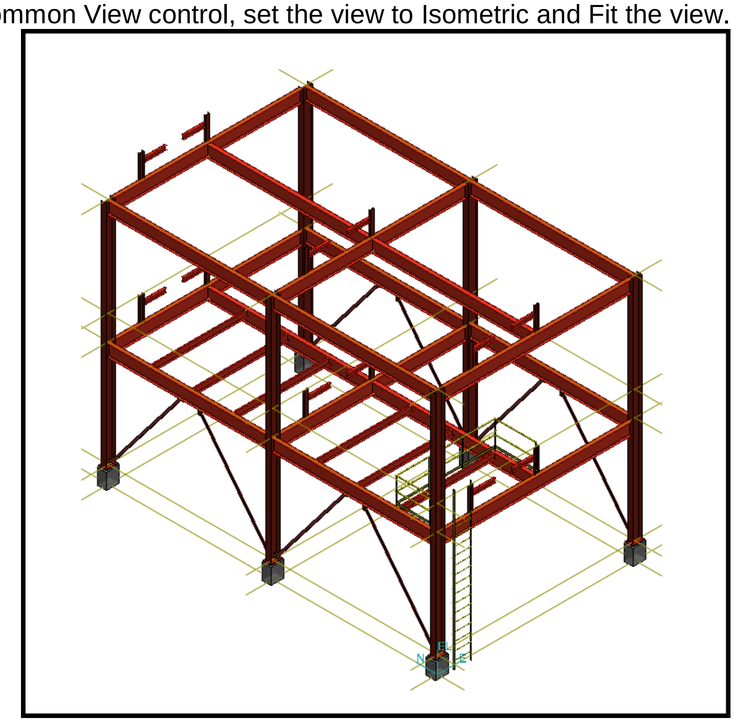
- Review the new centerline aspect for members.

Overview:

Centerlines aspect can be applied for both design (built-up) and standard members (linear and curved) for the purpose of generating plan and elevation view drawings with appropriate dimensions for any given project. Also, it can be used for designing other structural objects. For members that have non-symmetrical cross sections the location of the centerline must be CP5.

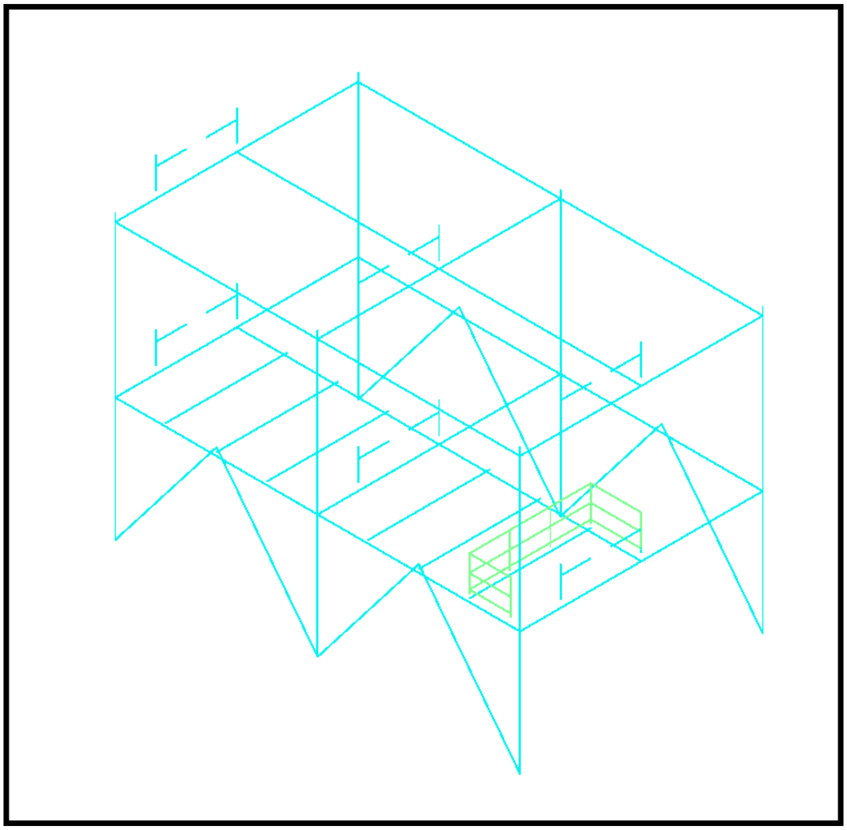
For drawings, members can be re-symbolized as a line. If the members support centerline aspect, drawing creators can use this graphic for line re-symbolization. Centerline is different than the logical axis of the member system.

1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include the A2 → U01 and CS → U01 CS systems.
2. Select Tools>Show All to display the hidden systems and objects.
3. Hide the Slabs structural system folder.
4. Using the Common View control, set the view to Isometric and Fit the view.



5. Select the Format →View command. Select the Centerline aspect and unselect the Simple physical aspect. Click OK to dismiss the form and review the centerline graphics. Note the cardinal points used in the display.

© Copyright 2014 Intergraph Corporation



6. Select the Format →View command. Select the Simple physical aspect and unselect the Centerline aspect. Click OK to dismiss the form.

© Copyright 2014 Intergraph Corporation

LAB 6: Placing Horizontal Cross Bracing

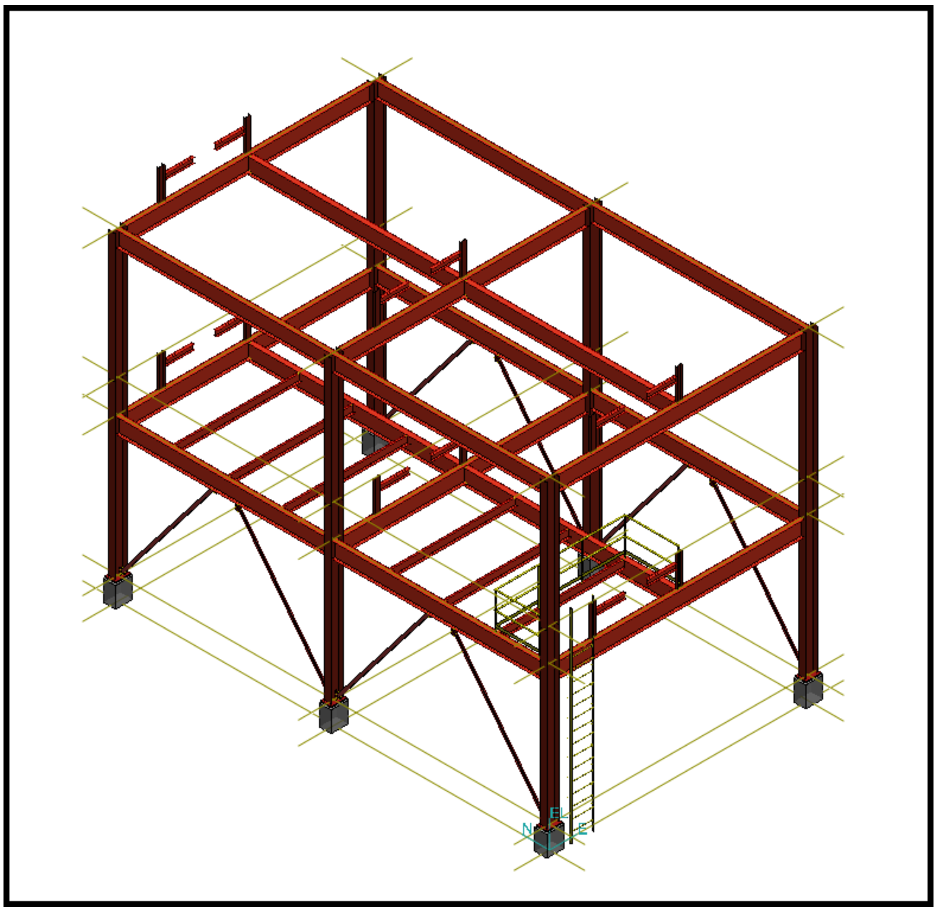
Objective:

By the end of this session, you will be able to:

- Place cross bracing in horizontal design scenarios.

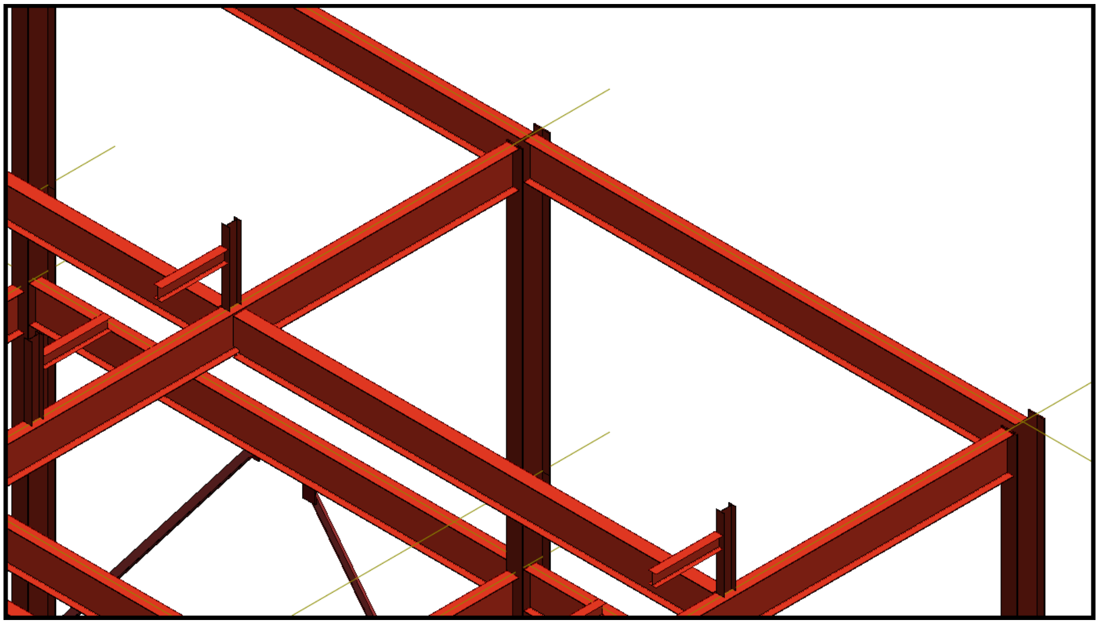
Placing Horizontal Cross Bracing:

1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include the A2 → U01 and CS → U01 CS systems.



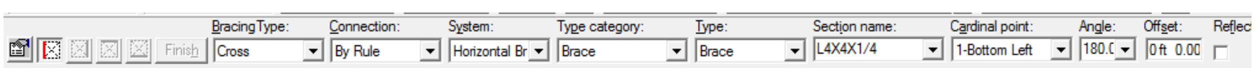
2. Zoom in on the top right side of the structure as shown below.

© Copyright 2014 Intergraph Corporation

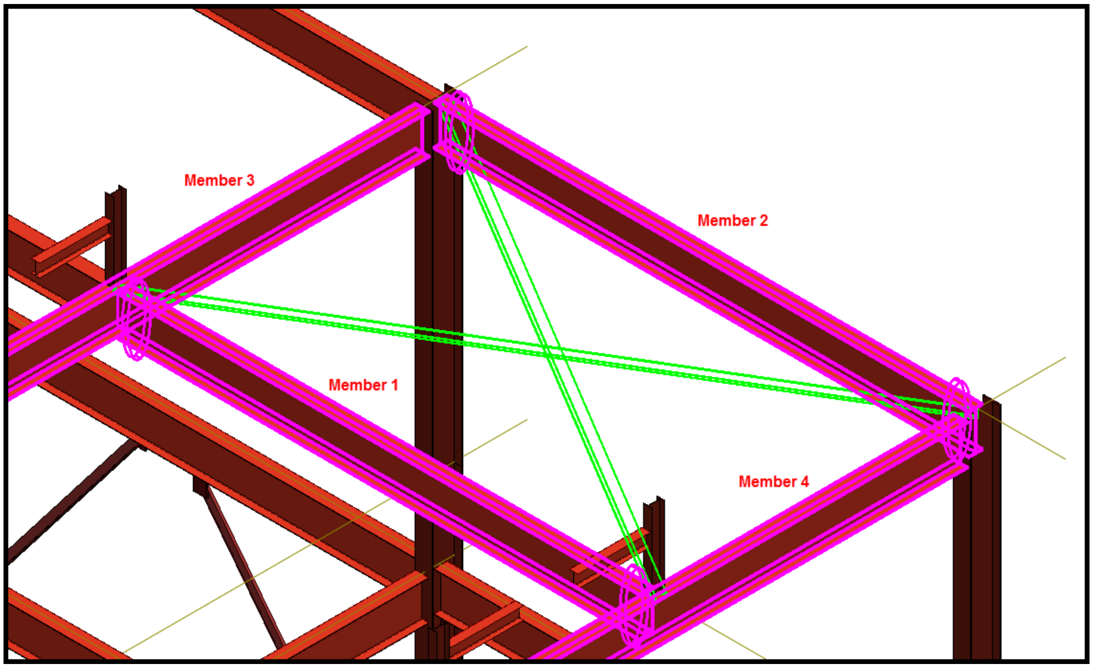


3. Select the Place Bracing command. Set the ribbon bar parameters as listed below.

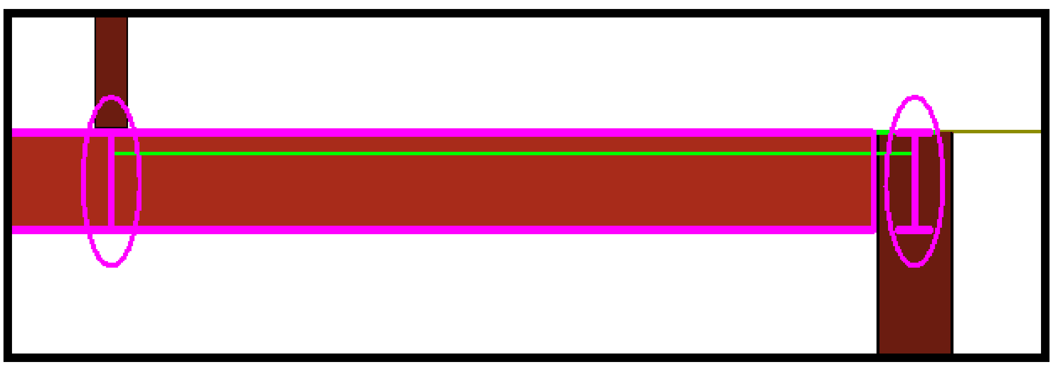
- Bracing Type: Cross
- Connection: By Rule
- System: A2→U01→Structural\Horizontal Braces
- Type category: Brace
- Type: Brace
- Section Name: L4x4x1/4
- Cardinal point: 1-Bottom Left
- Angle: 180 deg



4. Select the four beams in order as shown below and click Finish to place the horizontal braces. In previous releases this command only worked using columns as the first two inputs, limiting the user to vertical placements only. Now the bracing wizard can now be used in horizontal scenarios utilizing beams instead of columns.

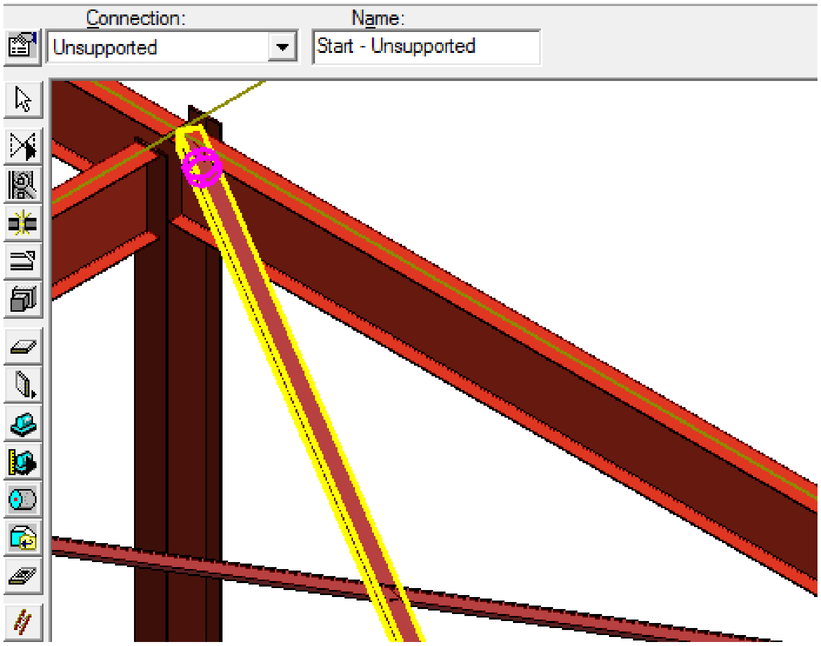


5. Select the two cross braces and using the Common View control, check the different side views to make sure the bracing is flush with the top of the other members.

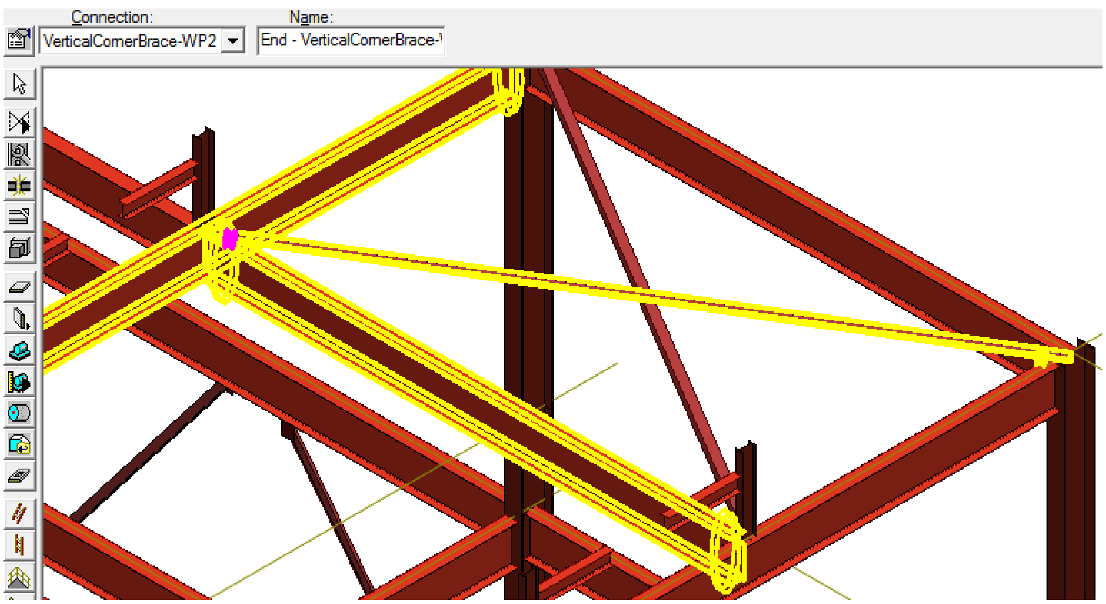


6. Set the locate filter to frame connections and select the top, right frame connection as shown below. Note that it is "Unsupported".

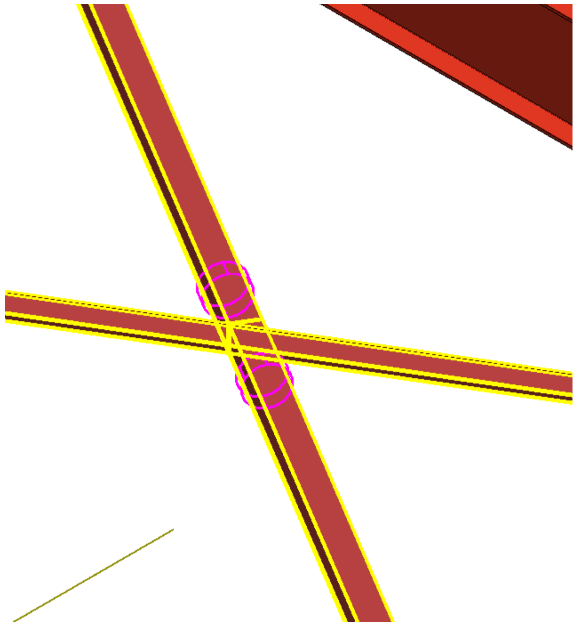
© Copyright 2014 Intergraph Corporation



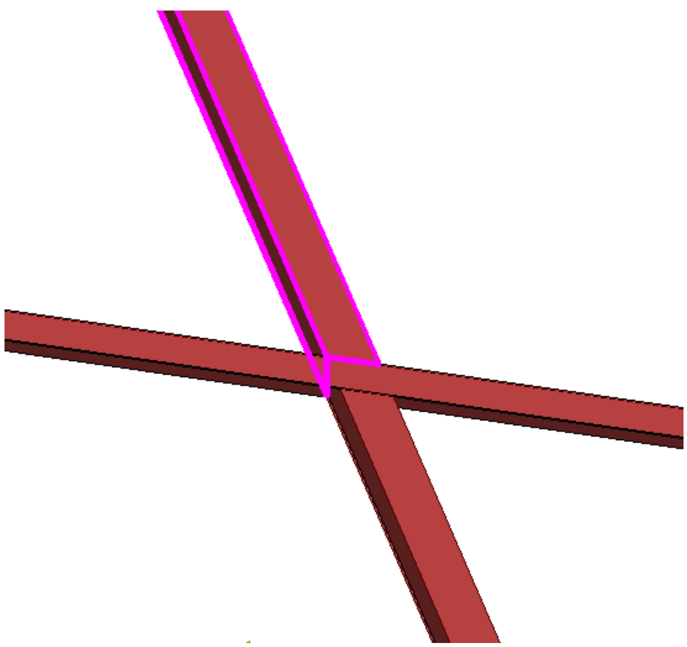
7. Select the top, left frame connection as shown. It is a vertical corner brace as expected.



8. Select the Place Split command. Set the Place option to “By Object” and the Split status to “Split First”. Select one of the cross braces and check to accept the selection. Select the intersecting brace and finish the command to place the split.



9. Set the locate filter to Split Connections.
10. Select the split connection placed above and then select the Place Assembly Connection command. Make sure the “By Rule” box is checked and the Condition is set to “Retain existing”. Set the System to A2→U01→Structural→Assembly Connections. Finish the command to place the assembly connection. The fitted connection works well in this scenario.



© Copyright 2014 Intergraph Corporation

LAB-7: Frame Connections

Objectives

After completing this lab, you will be able to:

- Understand the frame connection entities and relationships
- Edit the Member Part and Frame Connection properties

Overview:

Frame connections support effective initial modeling and define a structural model that will adjust appropriately as you move members and change member sizes. A frame connection automatically calculates the position of the end point of the member to achieve and maintain your positioning design intent. When you place a member, the frame connection used can be selected by rule or manually. Your company can customize the rule-based selection of the desired frame connection based on information about the objects involved with the connection.

Notes:

- *You can select the frame connection of another member as the start or end location of the member that you are placing if you intend the connection to be the logical end of the supporting member.*
- *The ends of the member can have different connection types, including one end as Unsupported.*
- *Set the Locate filter to the Member system and click a beam. The software will highlight all objects related to the frame connections. This will confirm that an accurate relation has been established between the columns and beams as well as between the beams and the elevation plane.*
- *The frame connection is an optional relationship. When you copy-paste, you can elect to re-establish the frame connection relationship to members outside of the copy set or delete the relationship and replace it with Unsupported relationship. See the session on copy-paste.*

You will need to experiment with different types of frame connections to fully understand their behaviors and the situations where they are most useful. The following gives a conceptual overview.

Types of Frame Connections:

There are three classes of frame connections:

- Connections that require no supporting members.

  These connections are of two types:

  1. Unsupported connection: Includes associative positioning relationship established to grid lines.
  2. General Surface connection: Includes associative positioning relationship established to the surface.

- Connections that require single supporting member as input.

  These connections are of three types:

© Copyright 2014 Intergraph Corporation

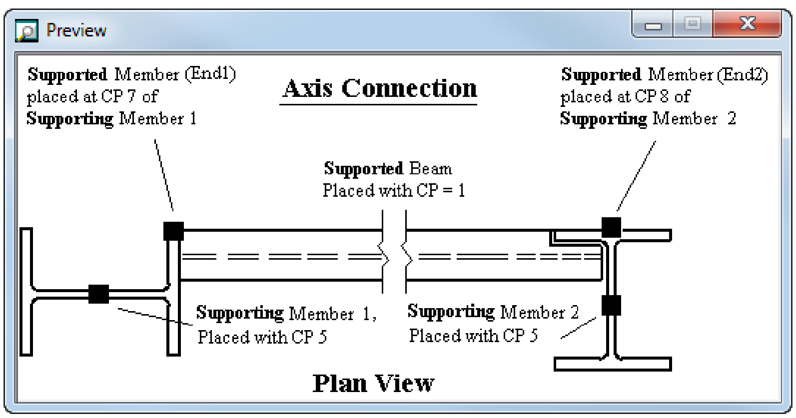
1. Seated connection: Includes member which lie on an exterior surface, above or below the surface.
2. Flush connection: Includes member with coincident surfaces that lie within the depth of the member.
3. Axis Along, Axis End, and Centerline connections: Includes member placed relative to a selected cardinal point axis of the supporting member.

- Connections that require two or more supporting member as input.

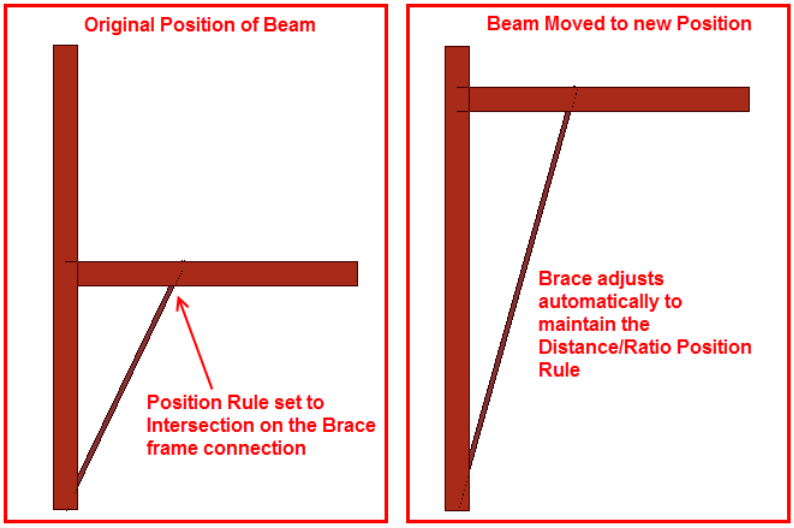
1. Vertical Corner Brace connections require two or more supporting members as input. Multiple inputs can be found by the connection by selecting the frame connection between two members.

Frame Connections Available in the Catalog:

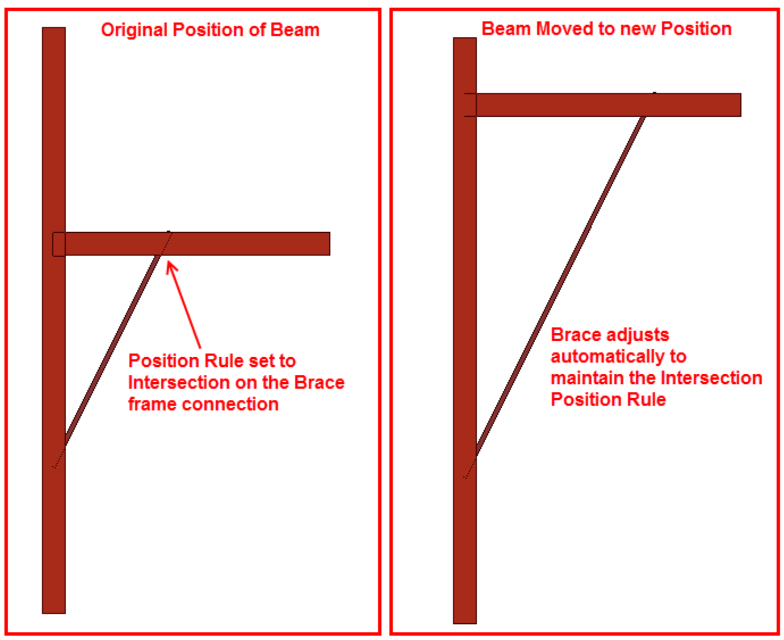
- **Axis-Along:** An Axis-Along frame connection positions the end point of the supported member axis at a point you define along the supporting member (with the option to have offsets from that point). It does not matter what cardinal point is used for the logical axis of the supporting member. You can choose to maintain this position after the connected structure members are modified, by using three different behaviors: Intersection, Distance, and Ratio. The same Axis-Along positioning behaviors are available for the Centerline, Flush, and Seated frame connections.



o Intersection: An Intersection frame connection maintains the connection by extending the supported member in the same direction until it intersects with the moved supported member.



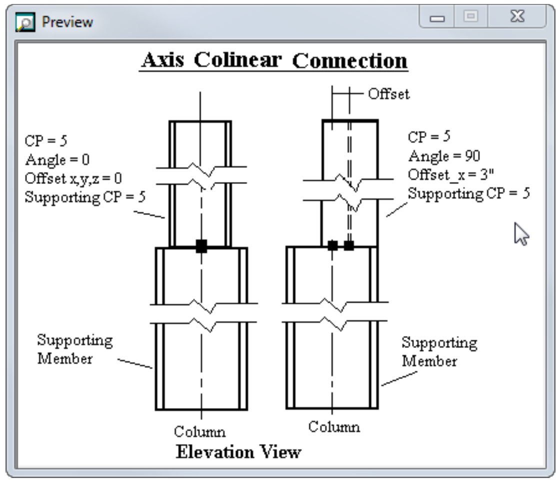
- Distance: A Distance frame connection maintains the connection at the original distance from the start of the supporting member.



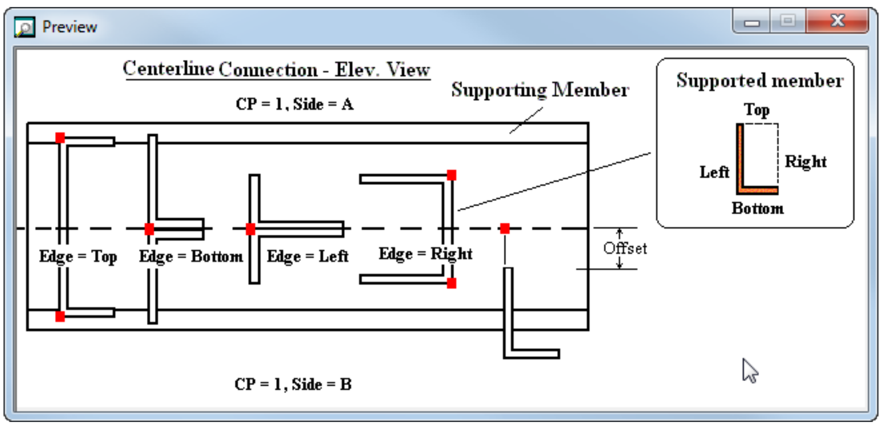
- Ratio: A Ratio frame connection maintains the connection at the original ratio of the supporting member length.

- **Axis-End:** An Axis-End frame connection is the same as Axis Along, except that it positions the supported member at the end of the supporting member.
- **Axis-Collinear:** An Axis-Collinear frame connection shows exactly the same behavior as Axis-End in the current version. The type should be removed from the Catalog.

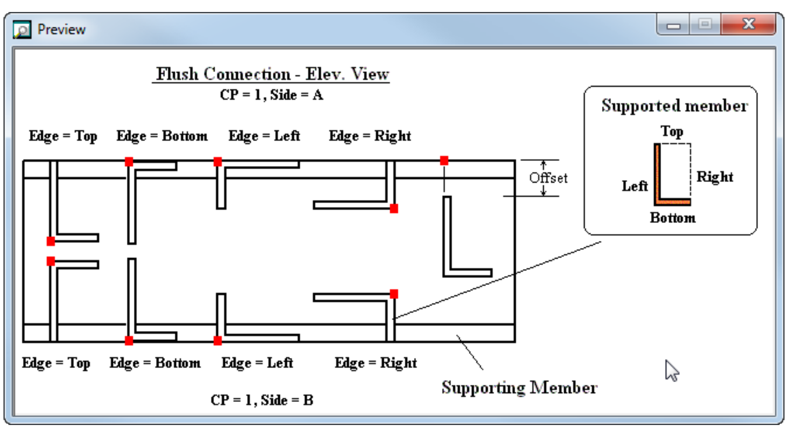
© Copyright 2014 Intergraph Corporation



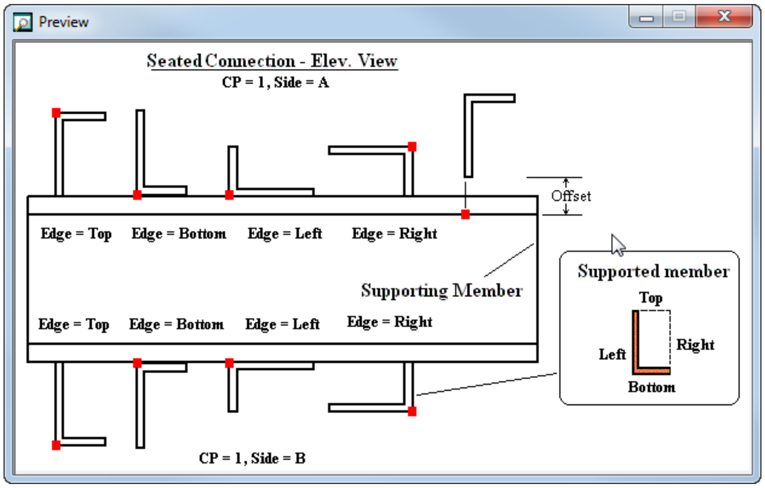
- **Centerline:** A Centerline frame connection uses the centerline of the supporting member to position the supported member, as shown in Figure 25, with the optional offset from the center.



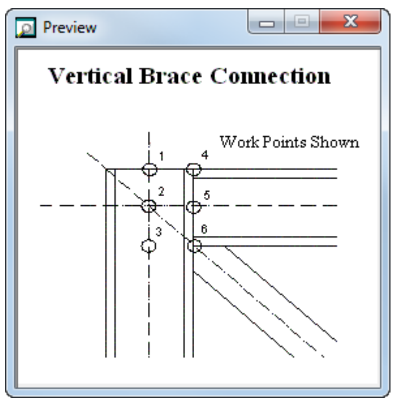
- **Flush:** A Flush frame connection uses the top and bottom extent of the supporting member to position the supported member. The supported member typically lies within the body of the supporting member, as shown in Figure 26.



- **Seated:** A Seated frame connection uses the top or bottom extent of the supporting member to position the supported member. The supported member typically rests against the supporting member, but can be offset as displayed in Figure 27.

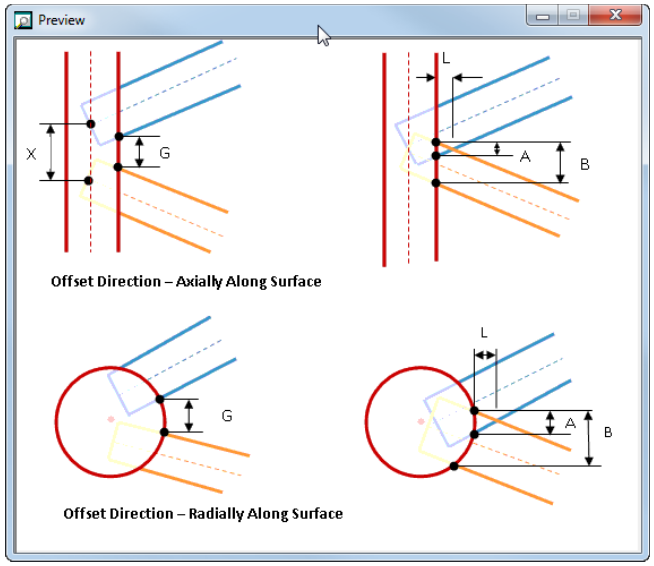


- **Vertical Corner Brace:** A Vertical Corner Brace connection positions a vertical brace that frames into a column-beam corner. You can define offsets in the X, Y, and Z directions. You select one of the six work points or the point through which the axis of the brace passes.

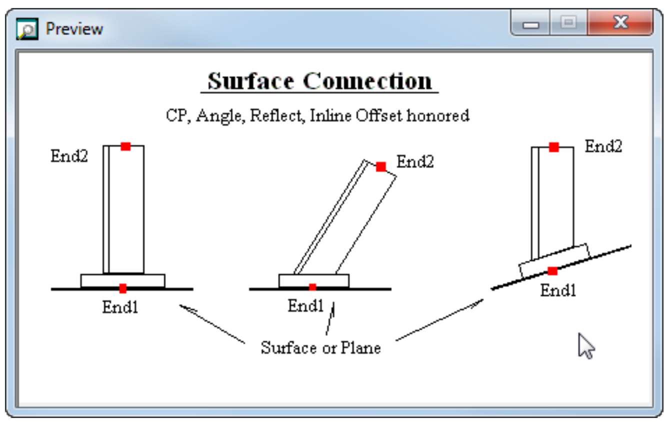


- **Gap:** A gap connection defines offsets between members to provide clearance for welding or simply as a work point adjustment. Smart 3D can calculate the offset either axially along or radially around the support member. You must use the **More...** option and select the gap frame connection that you want to use. The software will not pick a gap frame connection when using the **By Rule** connection option.

© Copyright 2014 Intergraph Corporation



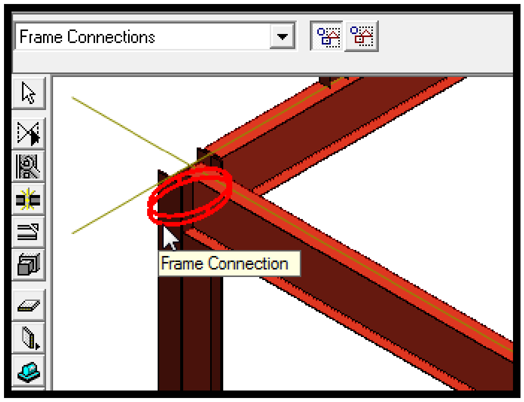
- **Surface:** A Surface frame connection positions the end of the supported member on a selected surface of any object type.



- **Unsupported:** An Unsupported frame connection positions the end of the supported member in space or relative to grid lines.

Selection of Frame Connections Existing in the Model:

You can graphically select the frame connections after they have been created by setting Locate Filter to Frame Connections or All and positioning the cursor near the ends of the member systems. The graphic handle for the selection is a hidden cylinder that highlights when you move the cursor over them, as shown in Figure below.



You can also select the frame connection by first selecting the member part and then selecting the end one smartstep on the edit ribbon bar for the member part. Finally, you can select the frame connection from the Workspace Explorer. First, select the member; the Workspace Explorer will automatically scroll to that member. You can then see the frame connections nested under the member system.

Selection of Frame Connections to Use During Member Placement:

1. **Selecting Frame Connections Using the By Rule Option:**

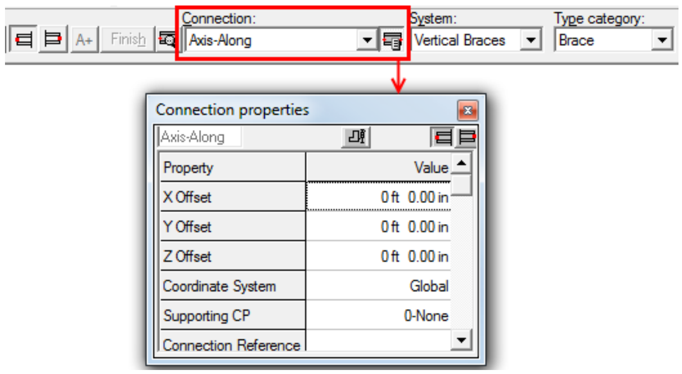
When placing linear members, if you select the By Rule option, the software selects the frame connections. Frame connections are selected as the software connects to the supporting member, based on the supporting and supported member type category, type, and geometry. These rules can be customized by your administrator:

- Surface-Default: This rule is selected when a member connects to a non-member object and the software selects the unsupported frame connection, unless the non-member object is a surface.
- Axis Along: This rule is selected when a member connects to another member.
- Axis-End: This rule is selected when:
  - Two members have the same type category, are parallel, end matched, and are in the same permission group.
  - Two members have the same type category and are end-matched.
  - The member being placed belongs to the Brace category and the two members are end-matched.
- Seated-Top: This rule is selected when the member being placed is of the type Girt or Purlin and the two members are not parallel.
- When placing a member, if you select the frame connection of another member as the end point, the software reads both the frame connection's member and its optional supporting member. If the member being placed is coplanar with those two members, the software selects Vertical Corner Brace-WP2.
- When placing a member, if 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 the two members, the software selects Vertical Corner Brace-WP2.

2. **Selecting Frame Connections Manually:**

© Copyright 2014 Intergraph Corporation

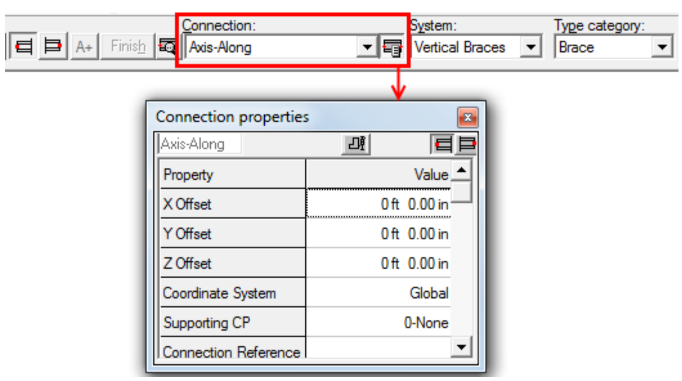
When placing a member, you can select the specific frame connection you want to use on the ribbon bar.



With the connection properties displayed, you can edit the Frame Connection positioning properties before you place the member.

Notes:

- *The Axis-Along frame connection created in the middle of the horizontal member is currently created with the position rule as Intersection. However, this may not be what you want for this type of connection. If the horizontal member moves up, you want the end point of the brace to maintain its position in the center of the beam. Select the frame connection and change the property position rule as Ratio to keep the brace in the middle of the horizontal member as it moves up or gets longer.*
- *When in the Finish state of member placement using the By Rule connection selection option, you can review and/or change the properties of either of the frame connections of the member being placed by activating the Connection properties dialog box. This dialog box will display whenever you are in the Place Member command, until you cancel it.*

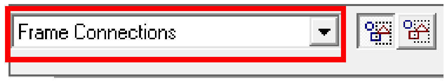


- *Once you change the properties of a frame connection type while using the placement command, the new property settings will become the default and will be used the next time you place the frame connection. This feature allows you to place several members with the same settings.*

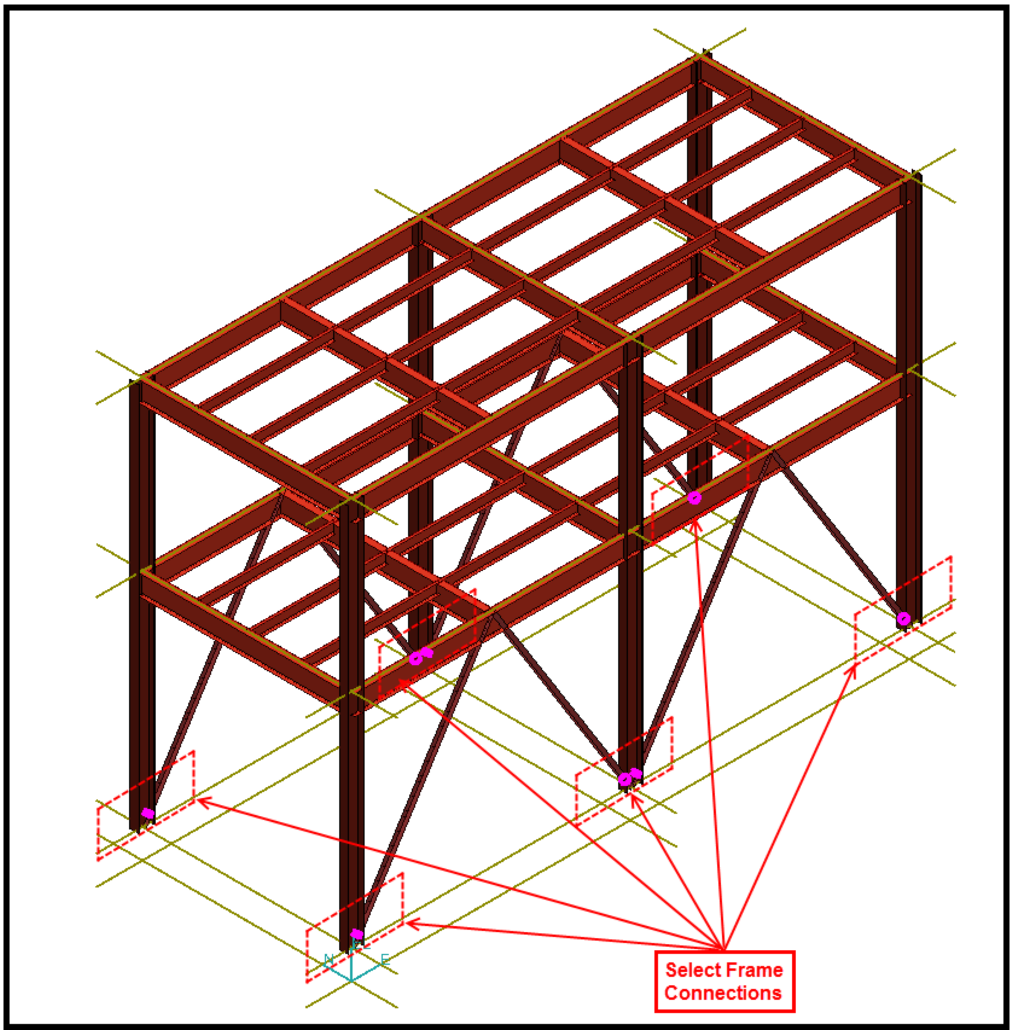
You will apply a working point offset for the vertical braces in the Structure U02.

© Copyright 2014 Intergraph Corporation

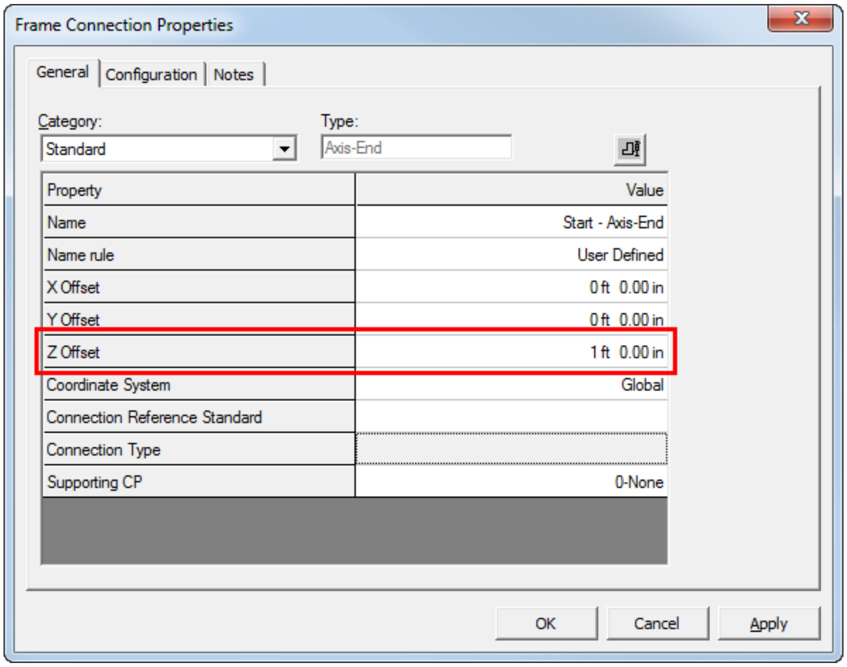
1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include the A2 -> U02 and CS -> U02 CS systems.
2. Go to the Structure Task environment.
3. Make sure the Active Permission Group is set to *Structural.*
4. Set the locate filter to Frame Connections.



5. Multi-select the Frame Connections located at then end of the vertical braces. See figure below:



6. Go to the Main Menu and select Edit -> Properties



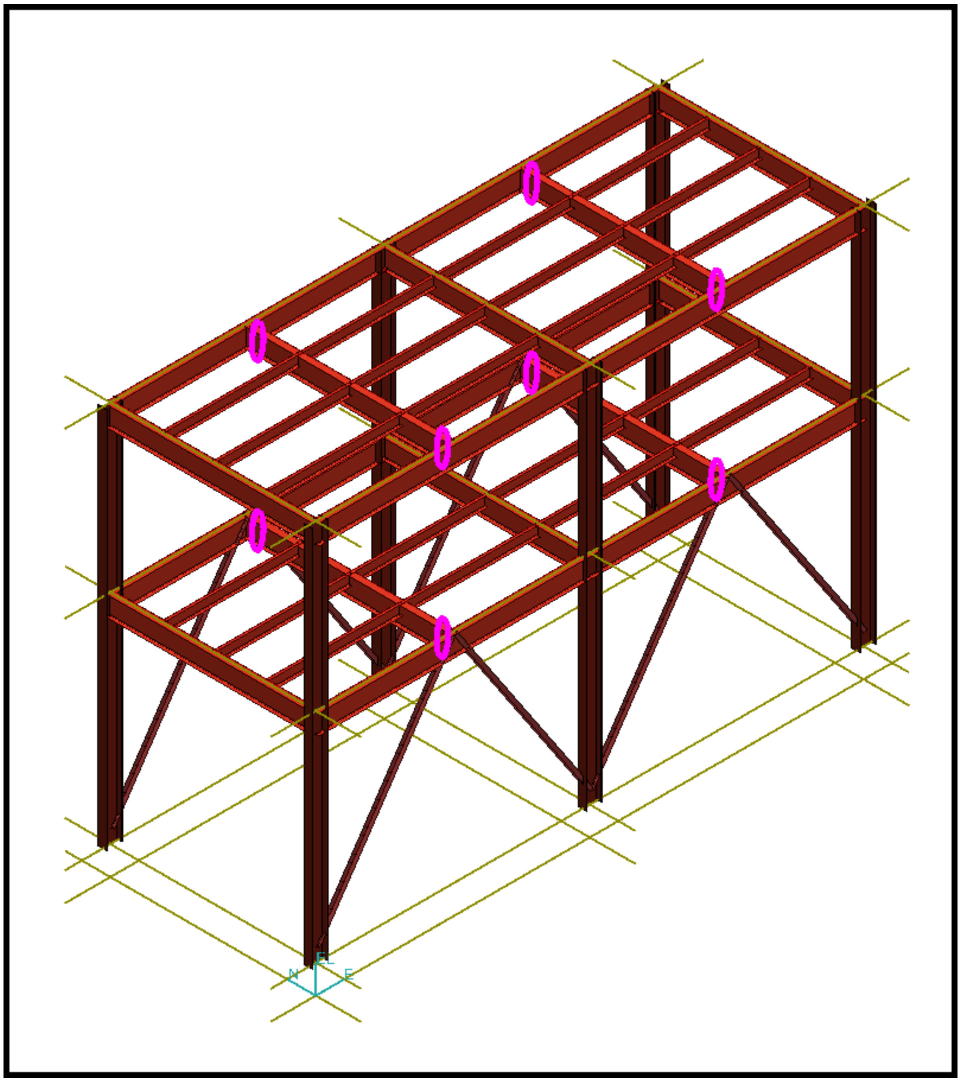
7. Keyin 1'- 0" for Z Offset. Hit "OK" button to commit the transaction.

8. Next, you will set the Position Rule from Intersection to Ratio for the intermediate beams in the Structure U02.

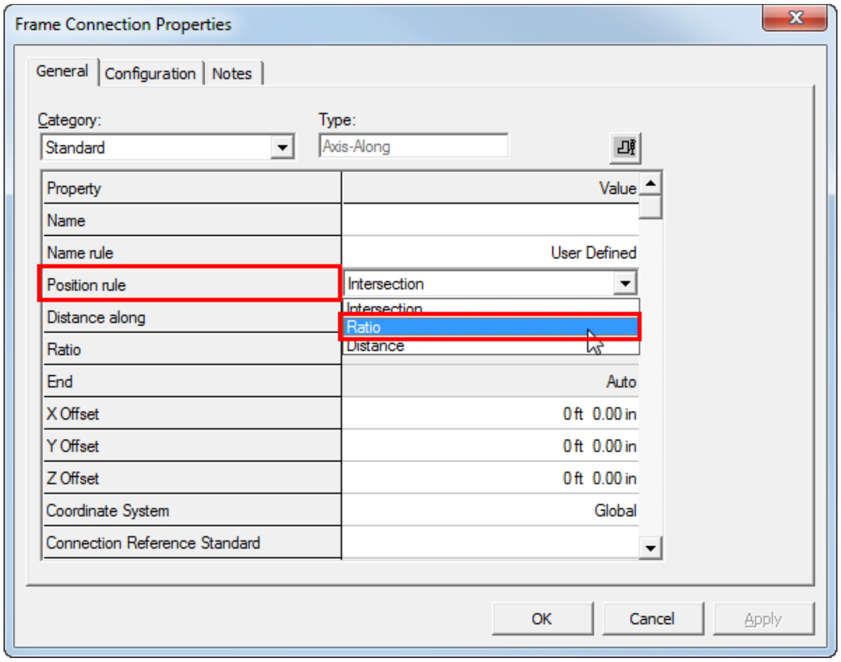
   Note: When Position Rule is set to Ratio, the system will maintain the frame connection's location the same distance (percent) along the supporting member's physical axis when the length of the supporting member is changed.

9. Multi-select the Frame Connections at the end of the intermediate beams. See figure below:

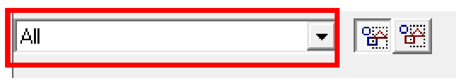
© Copyright 2014 Intergraph Corporation



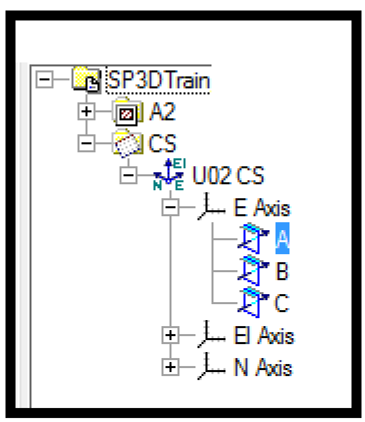
10. Go to the Main Menu and select Edit → Properties



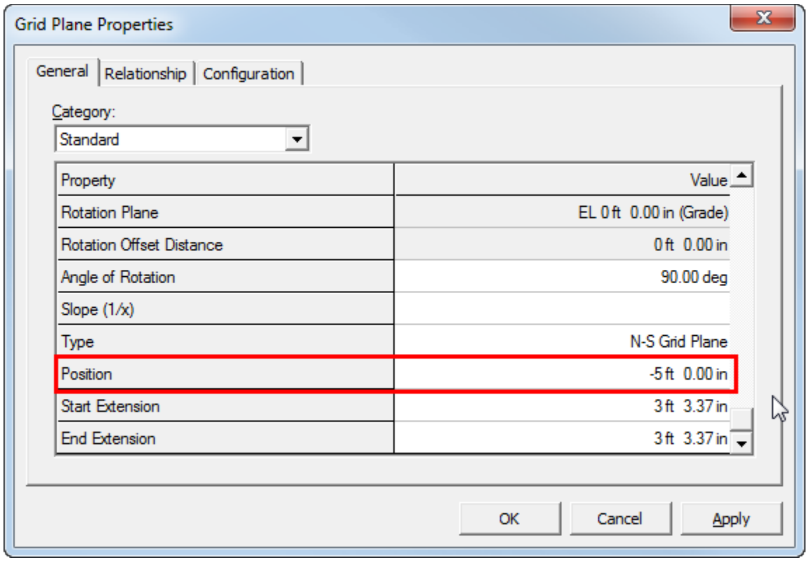
11. Change Position Rule from Intersection to Ratio. Hit “OK” button to commit the transaction.
12. Next, you will move the East Grid Plane 0’- 0” in the Structure U02.
13. Set the locate filter to All.



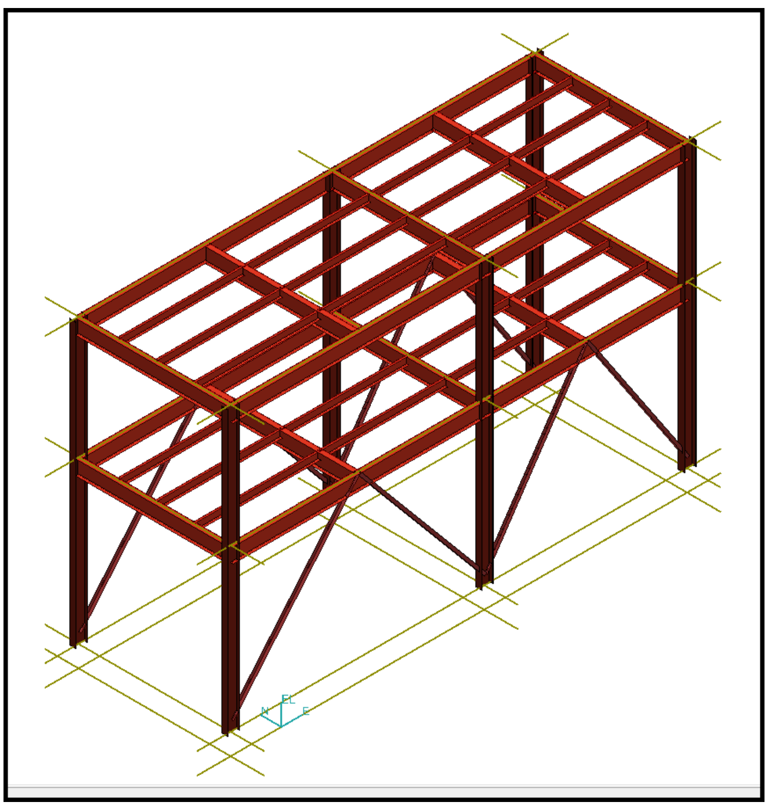
14. Go to the Workspace Explorer and select Grid Plane A



15. Go to the Main Menu and select Edit -> Properties



16. Keyin -5'- 0" for Position. Hit "OK" button to commit the transaction.
17. Verify that the Chevron vertical braces and all intermediate beams moved appropriately.



18. Save your session. Select File → Save.

LAB-8: Assembly Connections

Objectives

After completing this lab, you will be able to:

- Understand the assembly connection entities and relationships
- Use Place Assembly Connection command
- Use Trim Member Command
- Edit the assembly connection properties
- Place and manipulate assembly connections in a structure.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview
- Placing Members in a Structure

Overview:

You may have noticed that the members you connected using frame connections often have physical geometry that overlaps. Member geometry is a simple projection of the cross-section of a member along its logical axis. The geometry is the logical connection information needed during the frame design phase. The application of loads and connection end conditions allows structural analysis to be done.

Your company may subcontract the structural detailing of the physical connections between the members. However, it is often useful to trim the members to approximate their final length that is required to obtain a closer estimate of material requirements and a more finished model for customer design reviews. You may also wish to show base plates and large gusset plate geometry for the purpose of visualization, interference detection, and drawing generation.

Assembly connections create the next level of the design detail. An assembly connection can add or remove material from the member and create additional parts needed for the physical assembly at that joint. It is a design object that controls the creation and placement of other design objects. The connection recalculates automatically (subject to access permissions and status) when you edit the position and/or size of the members. You will typically want to apply the assembly connections later in the design cycle to keep the model lighter during heavy editing. However, assembly connections impact material cut lengths so you must apply them before generating material reports.

The default Catalog provides a small set of standard assembly connections for use during the light or approximate detailing stage of the design. The most basic type of assembly connection is the Fitted assembly connection that simply trims. The other types of available assembly connection are described in this session.

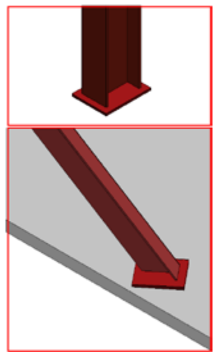
The assembly connection is designed so that it can be customized and integrated with third-party detailing or design products. As these additional libraries are developed, you will be able to extend the model to address the full structural detailing phase. It is anticipated that detailing can yield substantial improvement in the schedule and reduction in the errors introduced by import/export structural detailing workflows.

© Copyright 2014 Intergraph Corporation

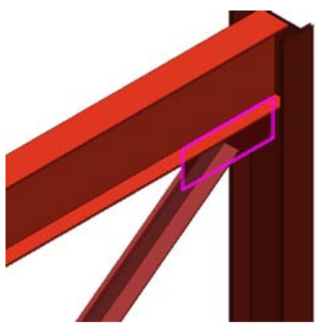
Types of Assembly Connections:

The following assembly connections are provided in the default Catalog and are used to support the level of design detail often generated prior to the final structural connection detailing activity:

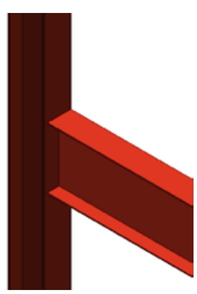
- **Base Plate**: This assembly connection places a plate at the end of a member. It trims the member to account for the thickness of the plate chosen and the angle of the member with the surface. The connection requires an unsupported or surface frame connection on one member. For example, you can place a base plate at the bottom of a column.



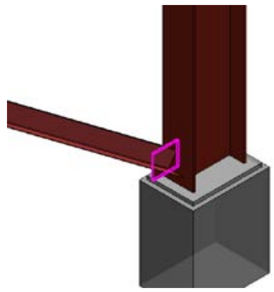
- **Corner Gusset Plate**: This assembly connection connects a vertical brace to a beam and a column intersection using a gusset plate. It trims the supported member (the brace) to the supporting members and places the gusset plate. The connection requires a frame connection with three members, such as a vertical corner brace.



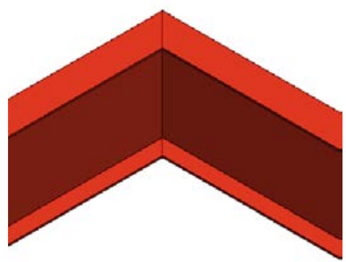
- **Fitted**: This assembly connection connects two members and trims the supported member to the supporting member. The connection requires a frame connection with two members, such as axis-along, seated, or flush.



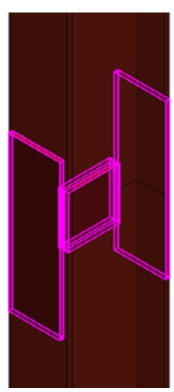
- **Gusset Plate:** This assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. The connection trims the supported member and requires a frame connection with two members, such as an Axis-Along.



- **Miter:** This assembly connection connects two members that are end-connected and co-planar. It trims both members with a single cutting plane.



- **Splice:** This assembly connection connects two members that are collinear and end- connected.



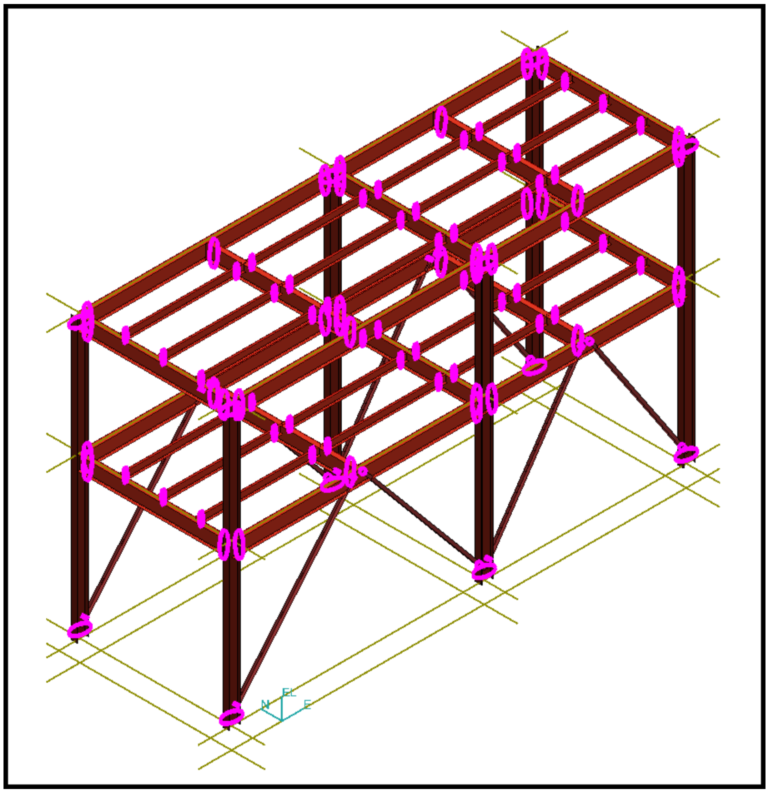
You will create planar cutbacks for all the members in the Structure U02.

1. Open or create a session file and define an “U02 & U02 CS” filter for your workspace. Your workspace should include the A2 -> U02 and CS -> U02 CS systems.
2. Go to the Structure Task environment.

© Copyright 2014 Intergraph Corporation

3. Make sure the Active Permission Group is set to *Structural.*

4. Set the locate filter to Frame Connections.
5. Select all the frame connections in Structure U02 using the fence method. See figure below:



6. Select Place Assembly Connection Command.
7. Set the parameters as

By Rule:	Check
Condition:	Retain existing
System:	A2 → U02 → Structural → Assembly Connections.

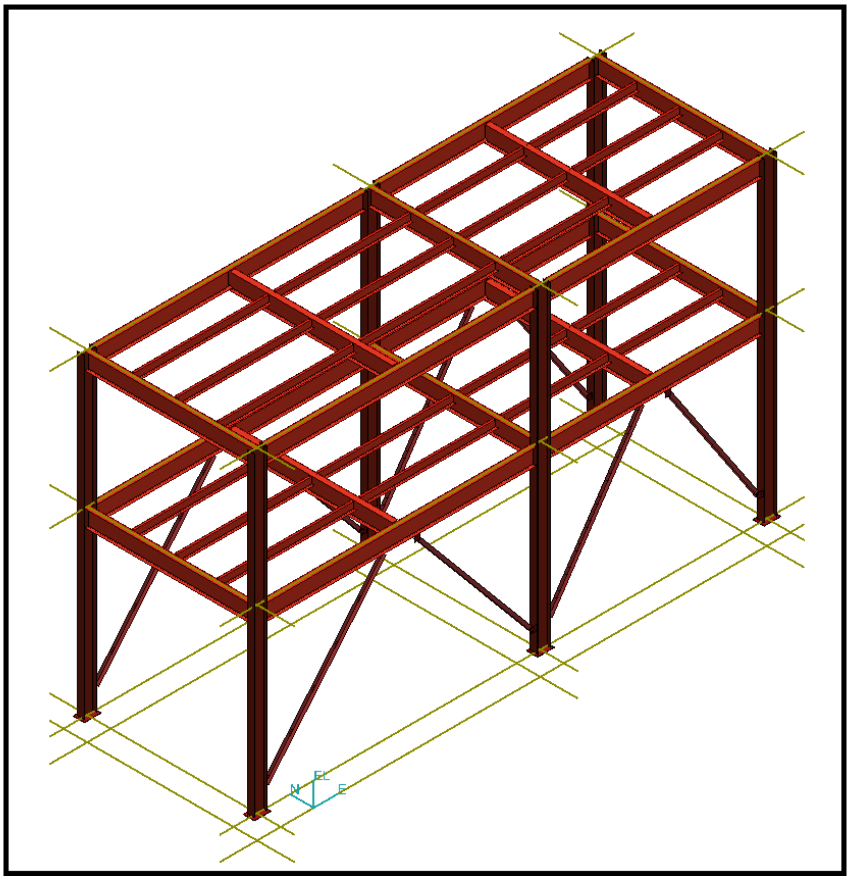
Notes:

- *The frame connections that are selected when you start the **Place Assembly Connection** command will remain selected for the **Select** smartstep of the **Place Assembly Connection** command. You can add or remove individual frame connections from the select set while in the **Select** smartstep of the command.*
- *If members are selected when you start the **Place Assembly Connection** command, the frame at the ends of the selected members will be automatically selected for the **Select** smartstep of the command. You can then individually deselect these frame connections.*

© Copyright 2014 Intergraph Corporation

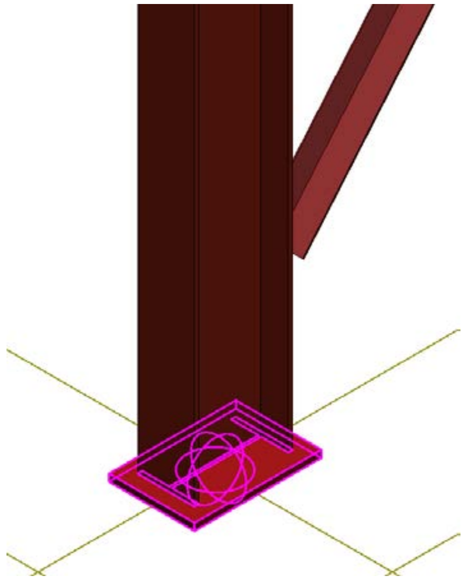
- *The **Select** smartstep of the **Place Assembly Connection** command will also allow you to select members. Selecting a member will automatically select the frame connections at each end of the member. You can deselect members that you select while in the select smartstep.*
- *Select the **By Rule** check box if you want the software to automatically select the type of assembly connection that needs to be placed. Clear the **By Rule** check box to manually select the type of assembly connection.*
- *To select a type of assembly connection, you need to click the **Type** drop-down list, click **More..**, and select the assembly connection.*
- *Select **Member Part Parent** to make the member part the parent object of the assembly connection. Clear this option to specify a generic system, using the **System** box, to be the assembly connection parent. Assembly connections created using member parts as their parents can be edited to have a system parent later if needed.*

8. Hit “Finish” button to commit the transaction.
9. Examine the model and verify that the planar cutbacks, base plates and gusset plates have been created.

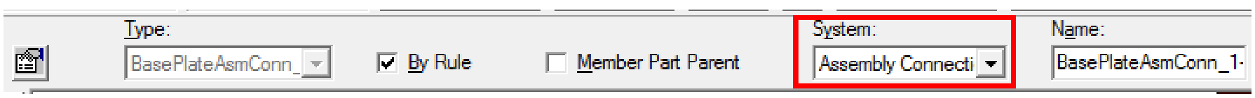


10. Set the Locate Filter to Member Assembly Connections and select the BasePlate assembly connection on the southwest corner of the structure as shown in the figure below;

© Copyright 2014 Intergraph Corporation

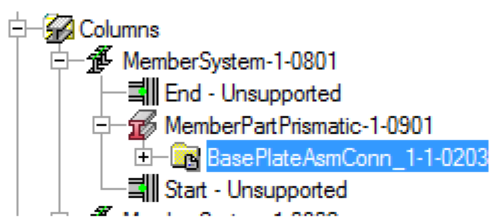


11. On the ribbon bar, note that the selected assembly connection was placed under the Assembly Connections System as expected.



12. Check the Member Part Parent box on the ribbon bar.

13. Note that the base plate assembly connection will now be a child of the Member Part Prismatic on which this connection was placed.



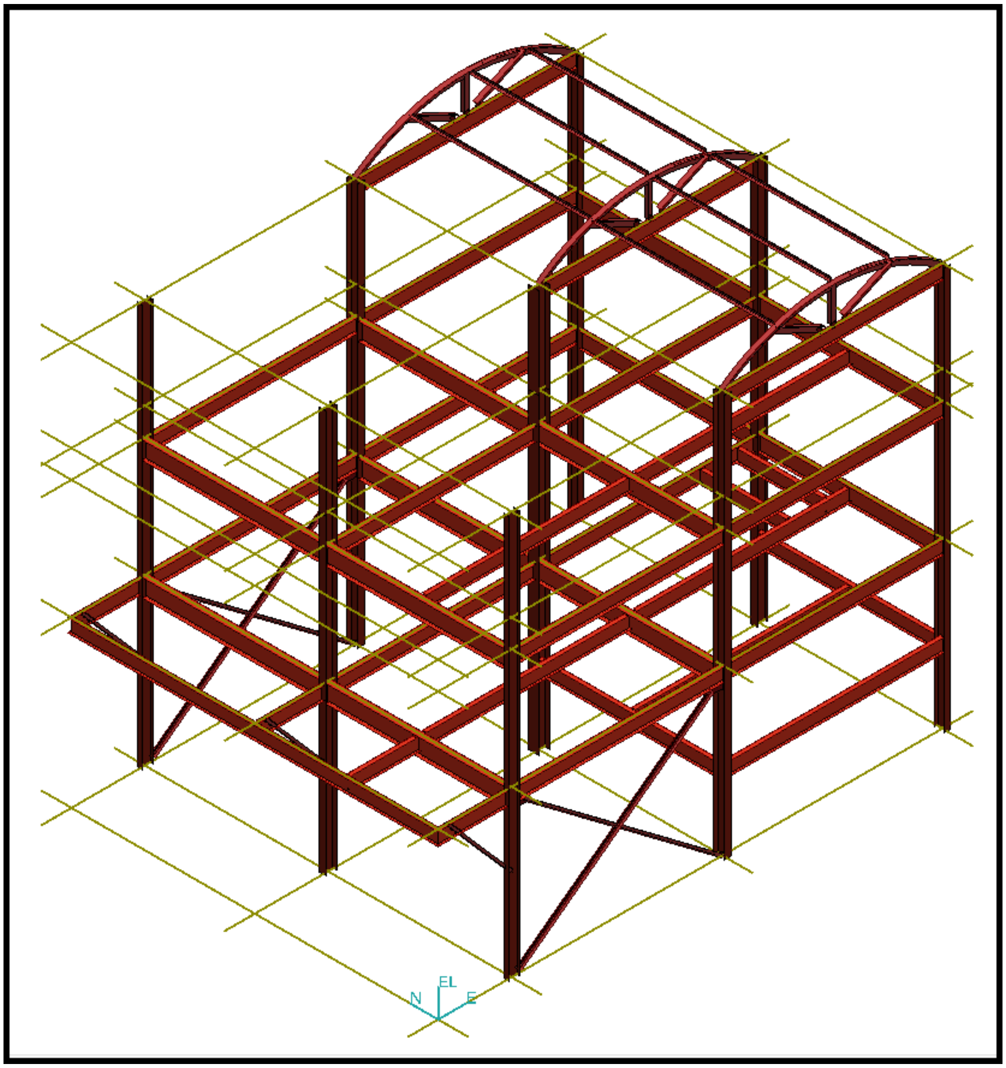
14. Save your session. Select File -> Save.
15. Re-define your workspace to include the A2 -> U03 and CS -> U03 CS systems.

    Repeat the above steps to create the assembly connections for structure U03.
16. Select Place Assembly Connection Command.
17. Set the parameters as

By Rule:	Check
Condition:	Retain existing
System:	A2-> U03 -> Structural -> Assembly Connections.

14. Hit “Finish” button to commit the transaction.

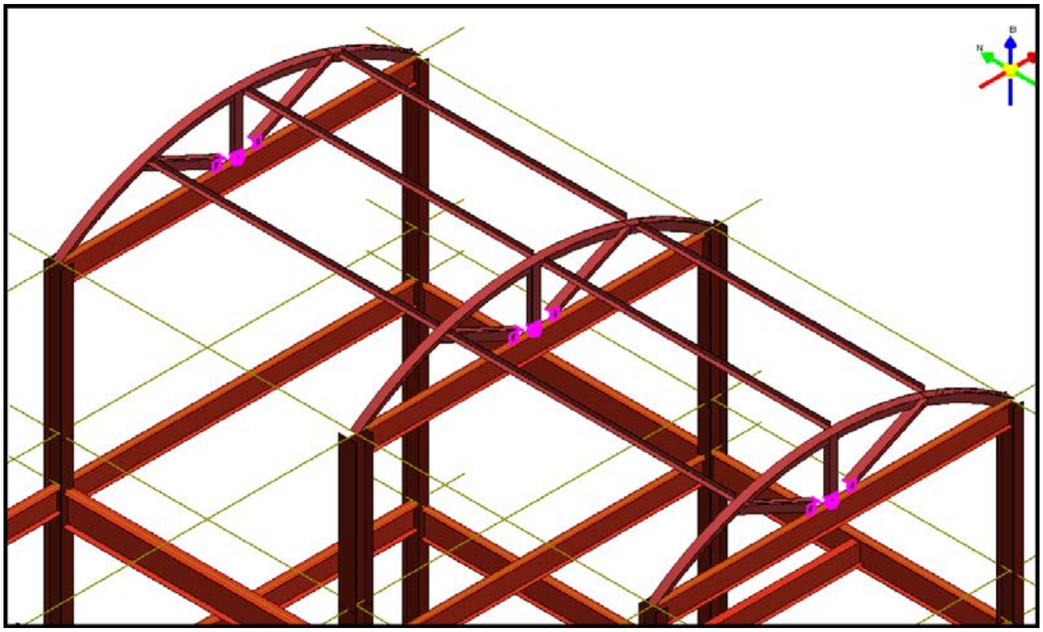
15. Examine the model and verify that the planar cutbacks, base plates and gusset plates have been created.



16. Use Zoom tool to zoom in around the rounded roof for the two-story building. 

17. Set the locate filter to Member Assembly Connections.

18. Multi-select the assembly connections located at the end of the braces. See figure below:



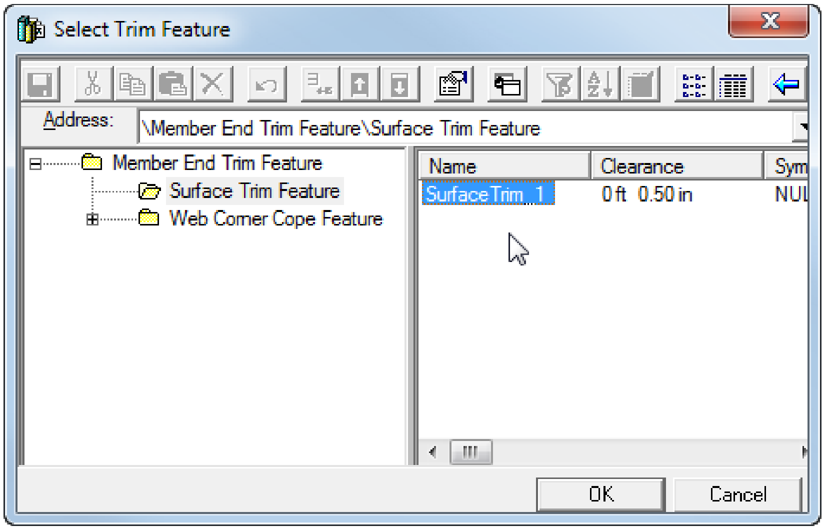
19. It should be nine objects in the select set. Click the Delete command to remove these objects.

Trimming a Member:

In some design situations, you may need to trim a member in a unique way that is not currently available with an assembly connection in your Catalog. In normal plant design applications, such occasions should be rare. The command must be used primarily in offshore and ship building applications. The **Trim Member** command enables you to manually trim member parts based on the surfaces you specify. You can apply the required number of trim operations to a member. You can place a trim and an assembly connection at the same member end. The results of the Trim features and the trimming applied by the assembly connection add and may overlap each other. Both the trim and the assembly connection are displayed in the **Workspace Explorer** under the member part.

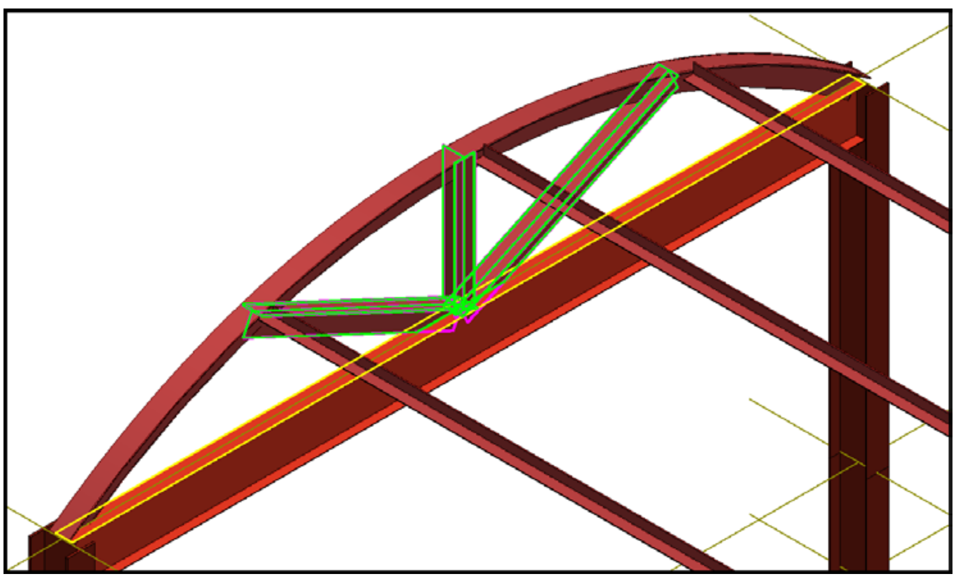
You can experiment with this command to see how it works. Pay attention to the **Squared End** property. If this property is set to **True**, then the member end is squared. If this property is set to **False**, then the member end trims exactly to your desired offset from the cutting surface.

20. Select Trim Member Command.
21. Expand the hierarchy and select the Surface Trim Feature folder.
22. Select Surface Trim 1.



23. Hit "OK" button. System returns to the Smartstep ribbon bar.

24. Select three vertical braces for the first SmartStep as shown below:



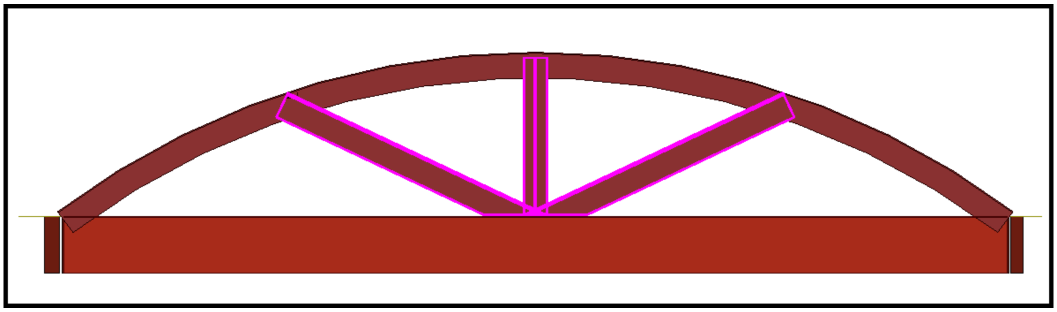
25. Select "Accept" button.

26. Select the top surface of the beam for the second SmartStep.

27. Select "Finish" button to commit the transaction.

    Repeat the above steps to create the assembly connections for the other braces on the trusses.

28. Verify the cutbacks have been created.



29. Save your session. Select File → Save.

LAB-9: Fireproofing

Objectives

After completing this lab, you will be able to:

- Place fireproofing on a pre-existing structure.
- Place fireproofing on entire structure.
- Modify the structure and observe the changes in fireproofing.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Grids: An Overview
- Structure: An Overview

Overview:

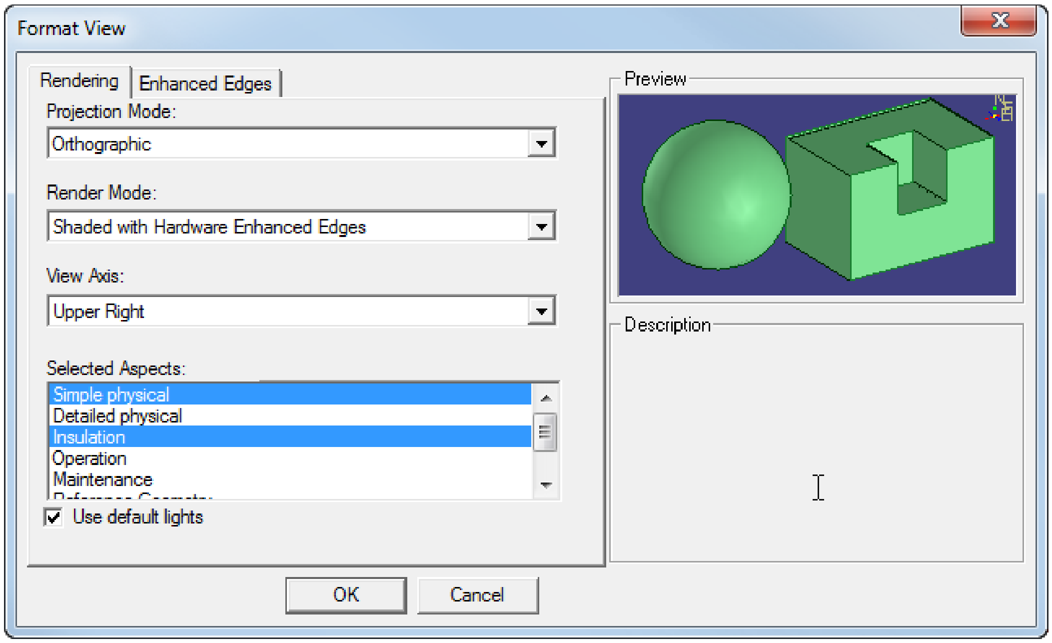
The concept of member systems and member parts were explained in the Structure Overview Session. These member systems can be selected and fireproofing can be placed on them. Fireproofing can be defined on setback distances from the member end. Multi-segmented fireproofing can be applied along side a single member or to an entire model in a single operation.

Fireproofing can be placed on a pre-existing structure using "By Rule" or through user defined encasement. The command allows fireproofing to be applied on member columns, beams, horizontal braces, vertical braces. The encasements are placed on the members using encasement rules. Fireproofing encasement rules are delivered in the catalog. Mainly two encasement rules, member concrete and member fibrous are delivered. Correct encasement will be applied even if the user defined fireproofing is used.

Retain fireproofing option allows the fireproofing to be placed on the new members and retains on the members that already have fireproofing placed on them. One or more of the selected members already has fireproofing on it.

1. Open a session file.
2. Define your workspace to include the A2 → U02 and CS → U02 CS system.
3. Set your active permissions to "*Structural*"
4. Go to *Format* → *View*. Select Insulation in the rendering tab.

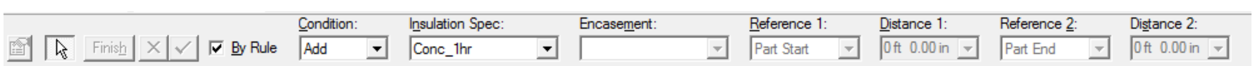
© Copyright 2014 Intergraph Corporation



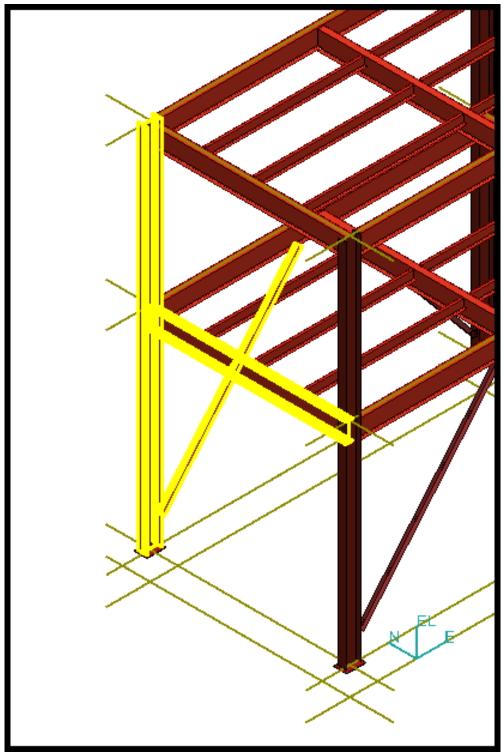
5. Your view should resemble the following graphic. Now, hit OK.
6. Go to File → Save.

Part A: Fireproofing on an existing structure

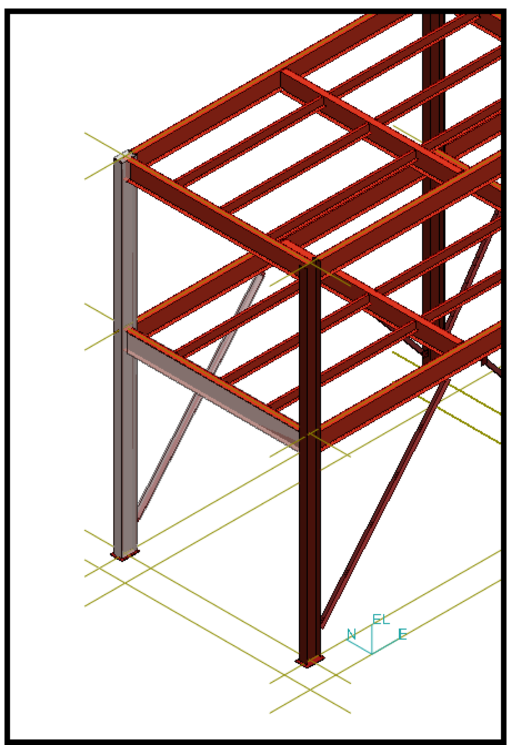
1. Select *Place Fireproofing* command. Insulation will be applied Using Place by rule. Make sure that the "*By Rule*" option is check marked.
2. Select "*Conc_1hr*" from the drop down list in the "*Insulation Spec*" menu.
3. Select "*Add*" from the drop down list in the "*Condition*" menu.



4. Select Column, Beam, Brace as shown in figure below,

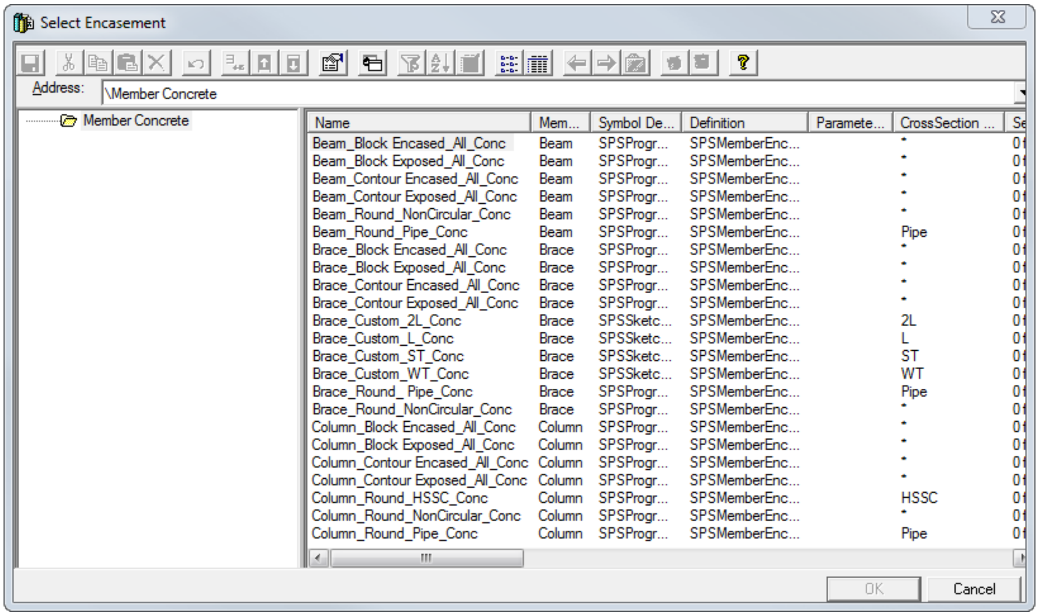


5. Hit the Accept button and then hit *Finish*. Your view should resemble the following graphic.



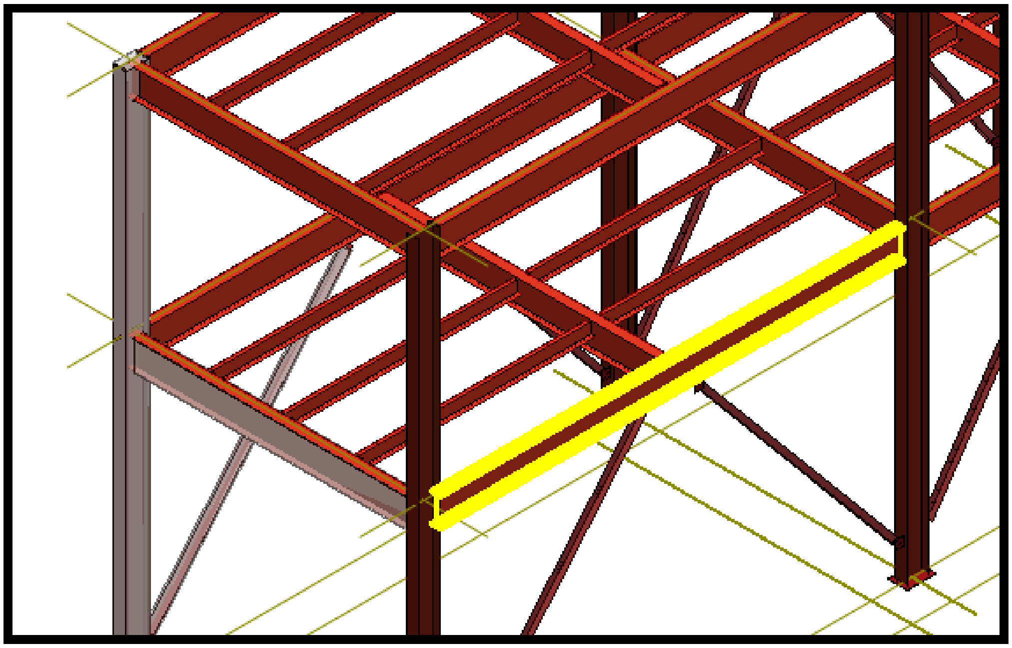
6. Select Place Fireproofing command. Insulation will be applied with the “User Defined” option. Make sure that “By Rule” option is <u>unchecked</u>.

7. Click on the drop down list in the "*Encasement*" menu. Go to More… Select Encasement window will appear. Your view should resemble the following graphic.

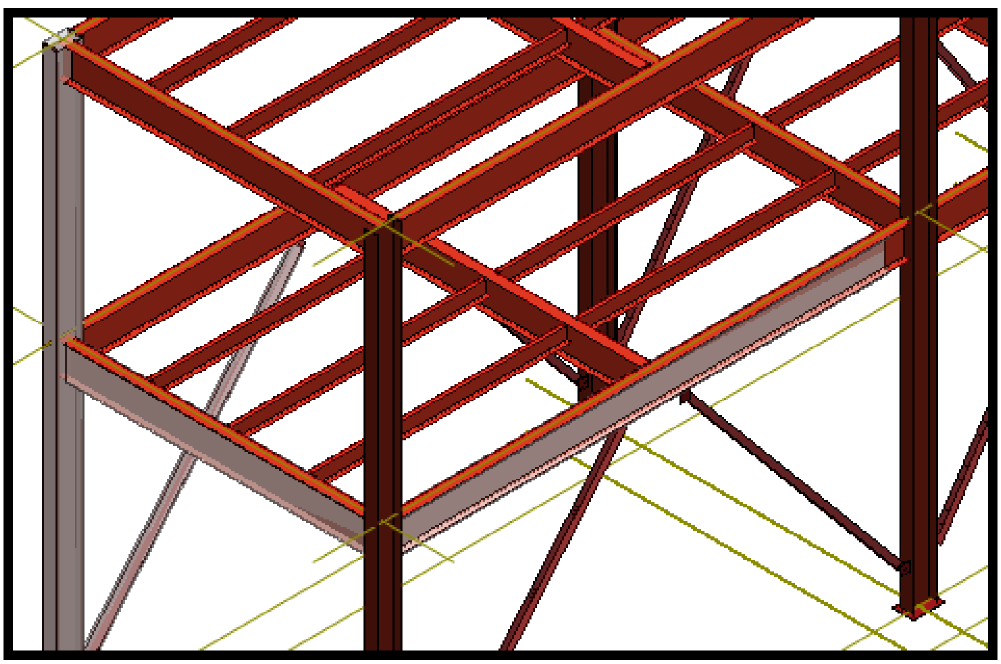


8. Select the "*Beam_Contour_Exposed_All_Conc*" Hit OK.
9. Select "*Part Start*" in Reference 1.
10. Enter "0" as Distance 1.
11. Select "*Part End*" in Reference 2.
12. Enter "1 ft" as Distance 2. This is the setback distance. This will not allow fireproofing to be applied on the beam at a distance of 1 ft from the Part End.
13. Select the Beam as shown in the figure below,

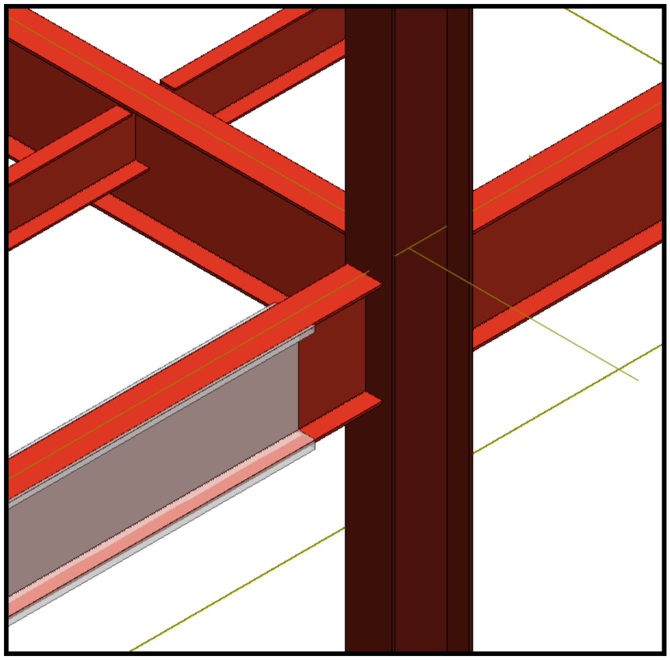
© Copyright 2014 Intergraph Corporation



14. Hit the Accept button and then hit *Finish*.
15. Hit the Cancel button. Your view should resemble the following graphic.

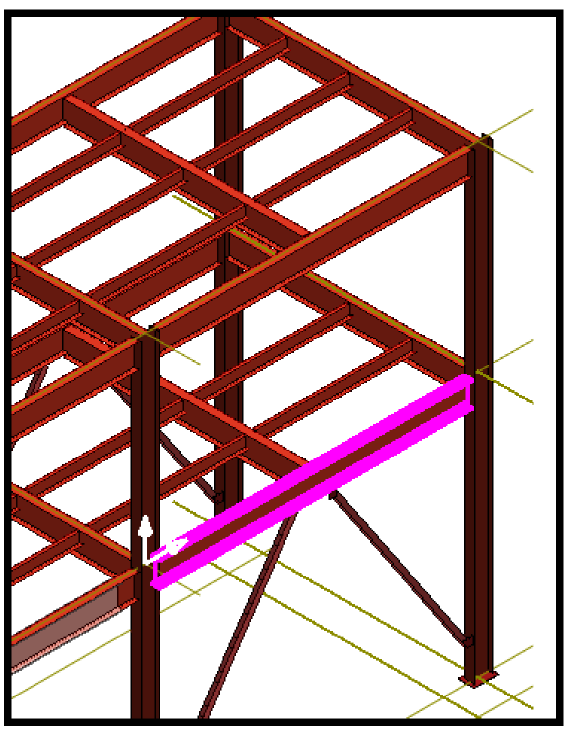


Please see below for close up image. This close up image shows the set back distance.



Part B: Correct encasement on an existing beam using "By Rule"

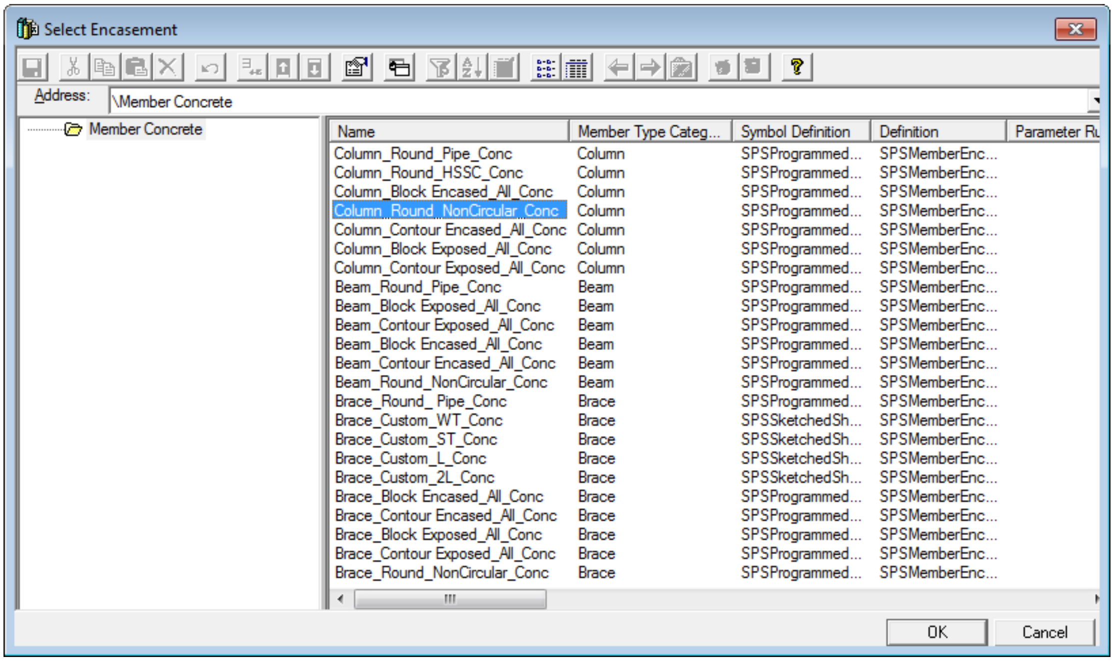
1. "*Place Fireproofing*" command will always apply column encasement to column and beam encasement to beam using "By Rule".
2. Select the beam as shown in the figure below,



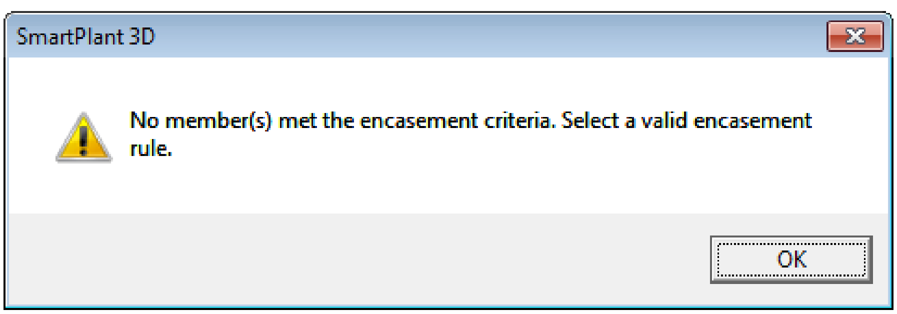
3. Insulation will be applied with the "User Defined" option. Make sure that "By Rule" option is <u>unchecked</u>.

© Copyright 2014 Intergraph Corporation

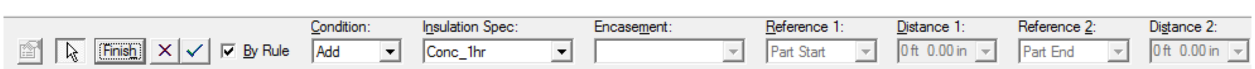
4. Click on the drop down list in the "*Encasement*" menu. Go to More… Select Encasement window will appear. Your view should resemble the following graphic.



5. Select the "*Column_Round_NonCircular_Conc*" Hit OK.
6. Select "*Part Start*" in Reference 1.
7. Enter "0" as Distance 1.
8. Select "*Part End*" in Reference 2.
9. Enter "0" as Distance 2.
10. Hit Accept. Warning message will be displayed. Your view should resemble the following graphic.

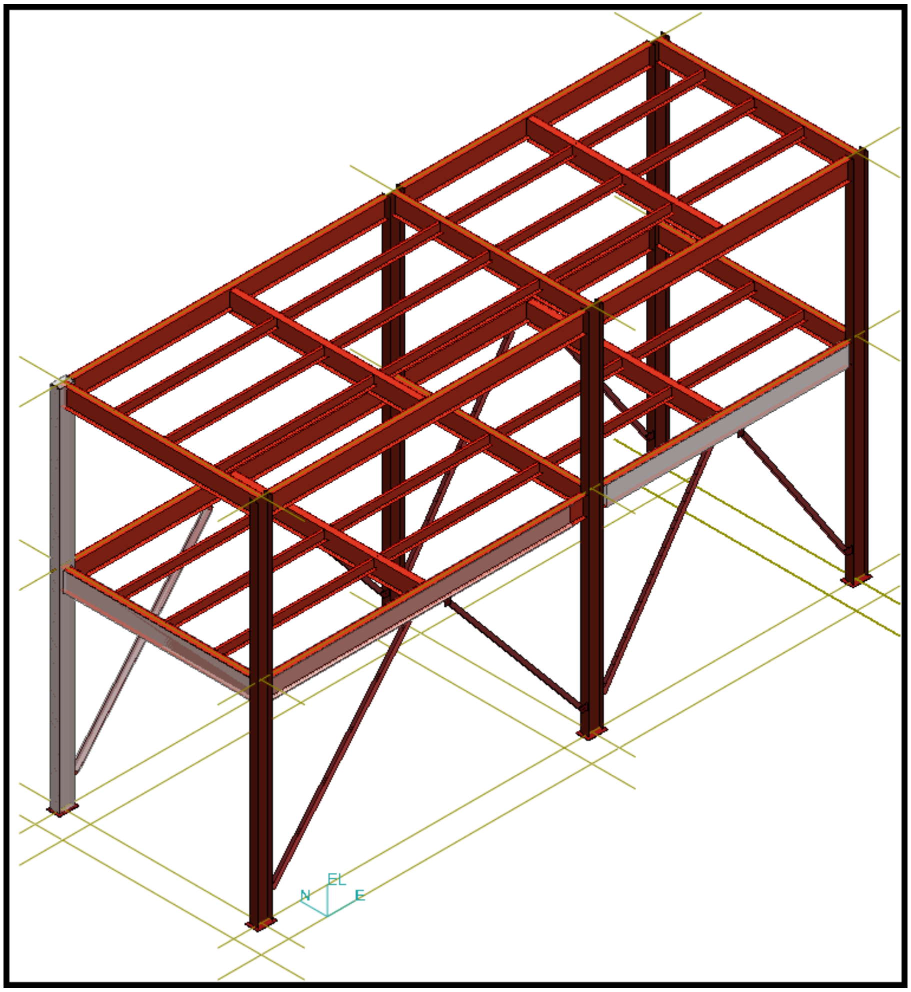


11. Hit OK.
12. Go to the toolbar and check mark the By Rule option and hit Accept.
13. Beam encasement fireproofing is applied to the beam. See the Encasement menu in the toolbar.



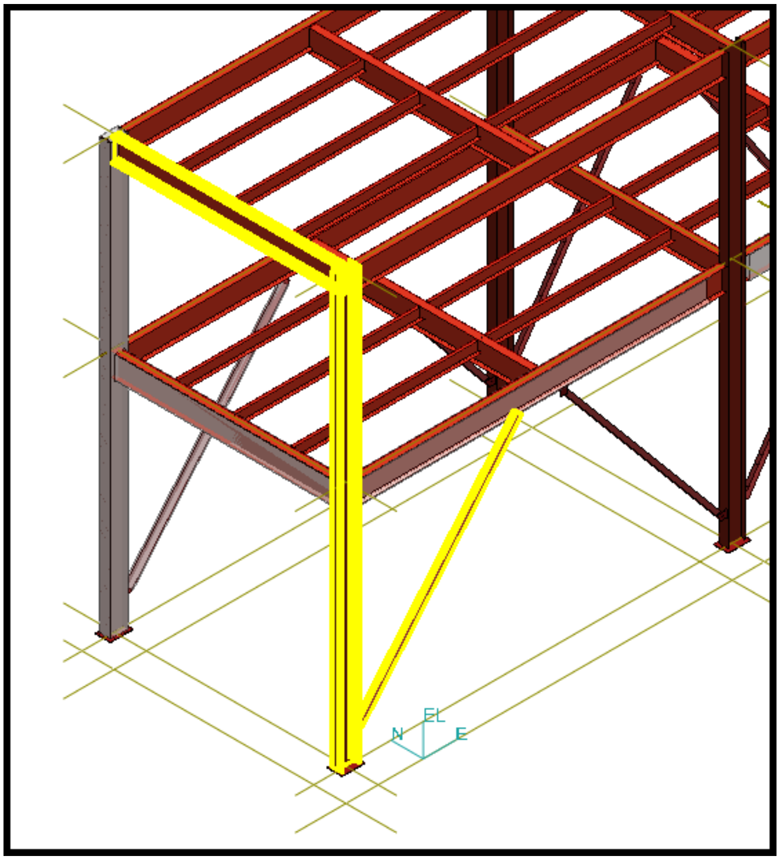
14. Hit Finish.
15. Hit Cancel. Your view should resemble the following graphic.

© Copyright 2014 Intergraph Corporation

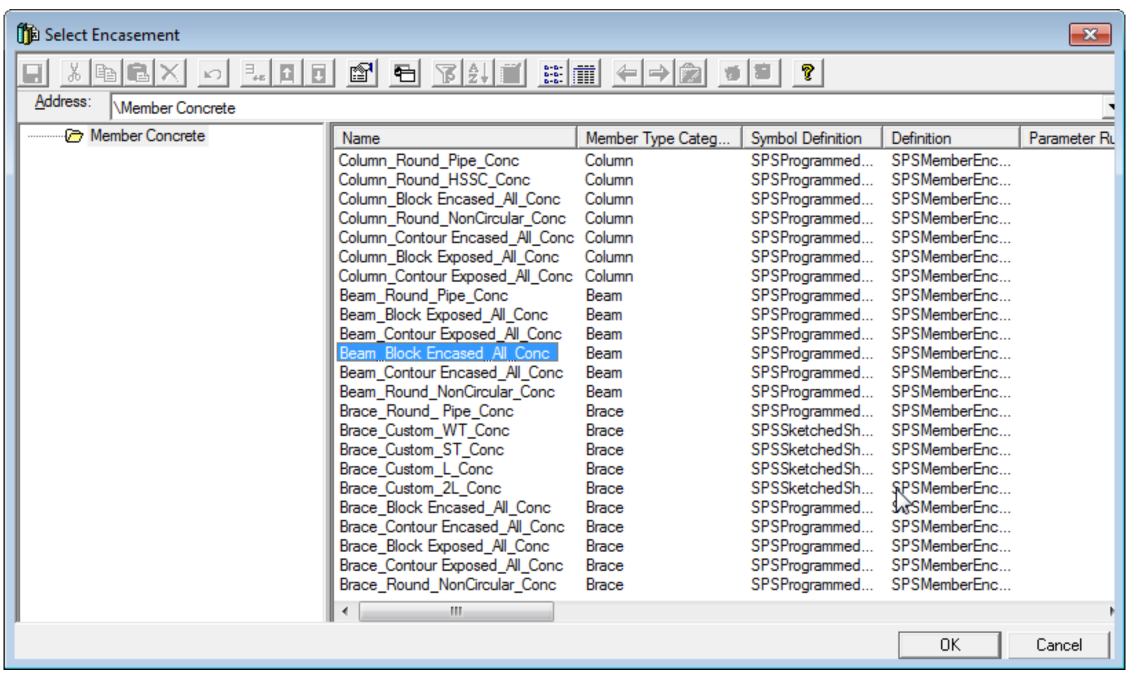


Part C: User defined correct encasement on existing members

1. "*Place Fireproofing*" command will always apply column encasement to column and beam encasement to beam by selecting the correct encasement without using By Rule.
2. Select the "*Place Fireproofing*" command. Make sure that By Rule is unchecked.
3. Select the column, beam and brace as shown in the figure below,

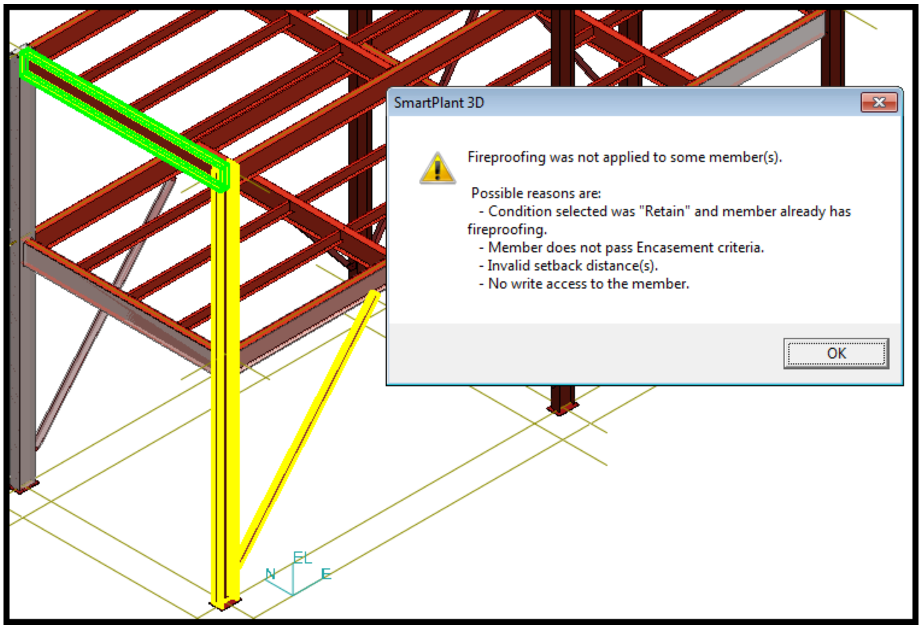


4. Go to more in the "*Encasement*" pull down list. Select Encasement window is shown.



5. Select "*Beam_Block_Encased_All_Conc*" and Hit OK.

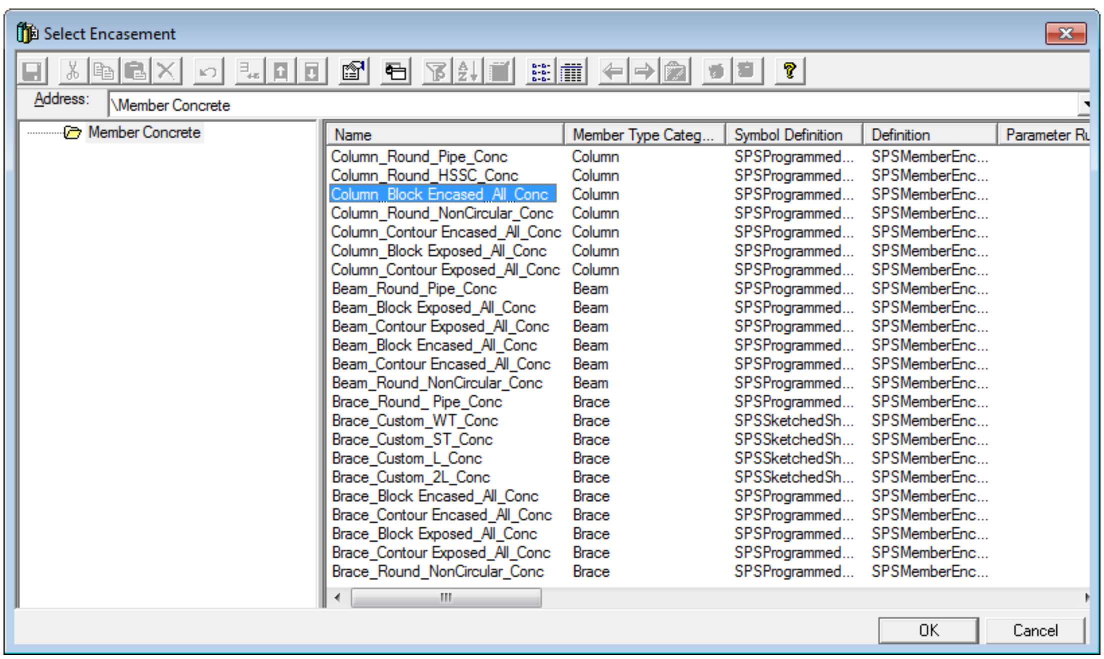
6. Hit Finish. Warning is shown.



7. Hit OK. Fireproofing is applied only to the beam.

Note: Selection is not lost.

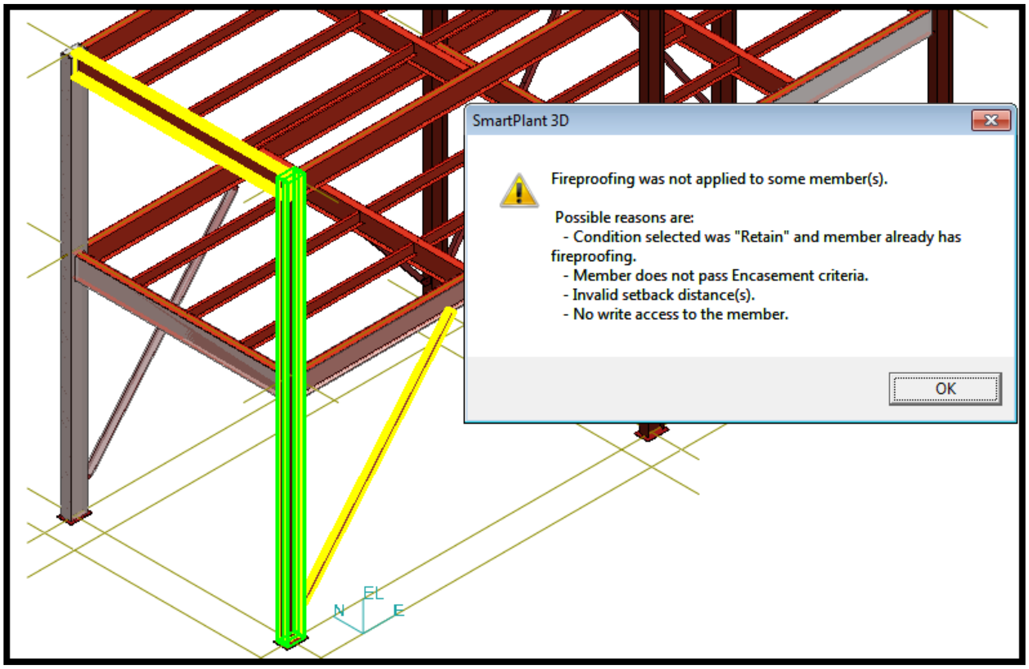
8. Go to more in the "*Encasement*" pull down list. Select Encasement window is shown.



9. Select "*Column_Block_Encased_All_Conc*" and Hit OK.

10. Hit Finish. Warning is shown.

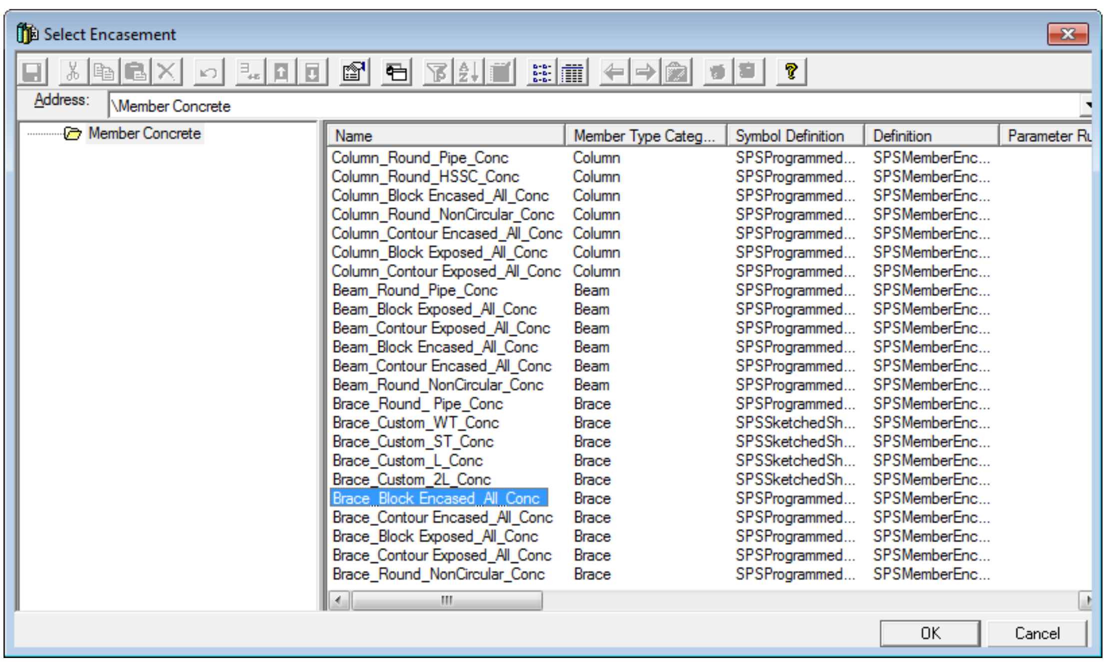
© Copyright 2014 Intergraph Corporation



11. Hit OK. Fireproofing is applied only to the Column.

Note: Selection is not lost.

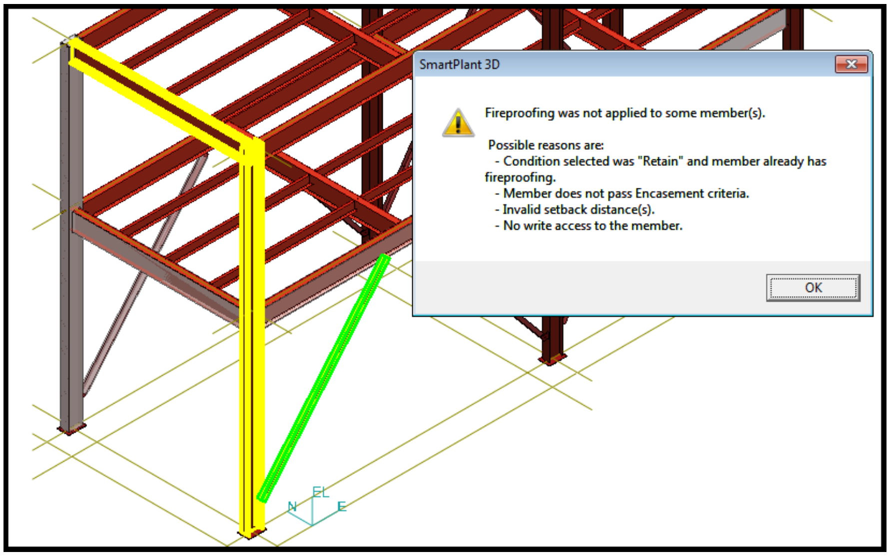
12. Go to more in the "*Encasement*" pull down list. Select Encasement window is shown.



13. Select "*Brace_Block_Encased_All_Conc*" and Hit OK.

14. Hit Finish. Warning is shown.

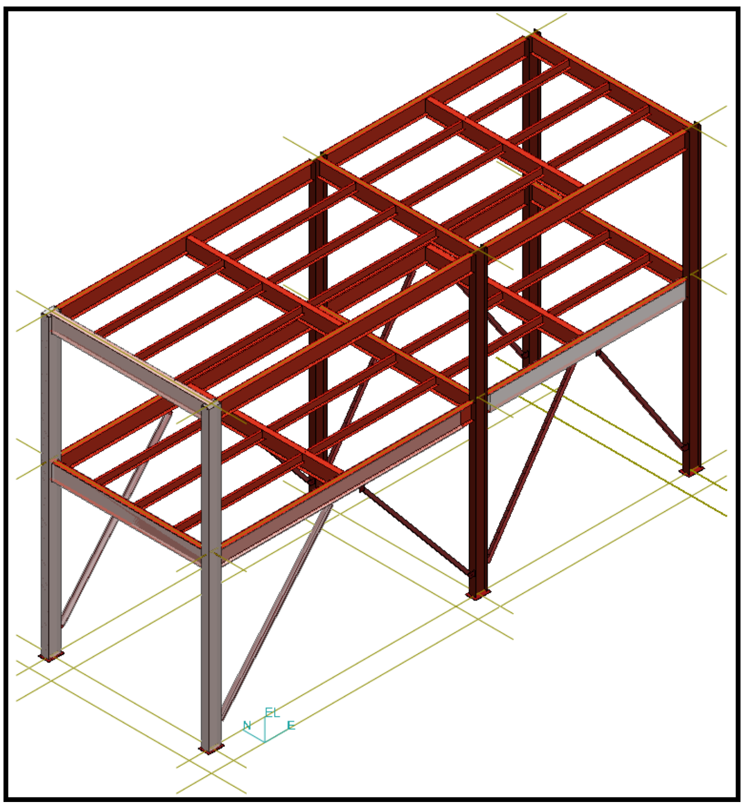
© Copyright 2014 Intergraph Corporation



15. Hit OK. Fireproofing is applied only to the Brace.

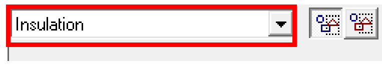
Note: Selection is not lost.

16. Hit Cancel. Your view should resemble the following graphic.

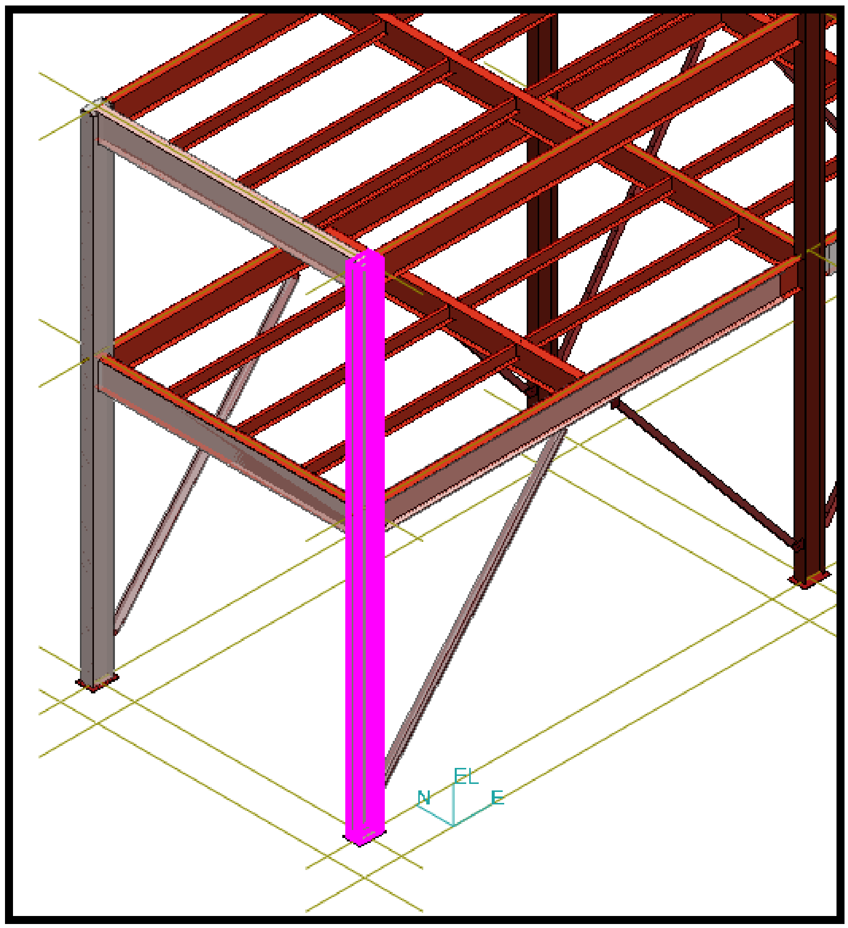


Part D: Add Fireproofing to a column that already has fireproofing.

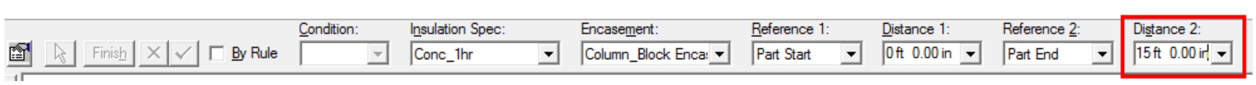
1. Set the Locate Filter to *Insulation.*



2. Select the insulation as shown in the image.

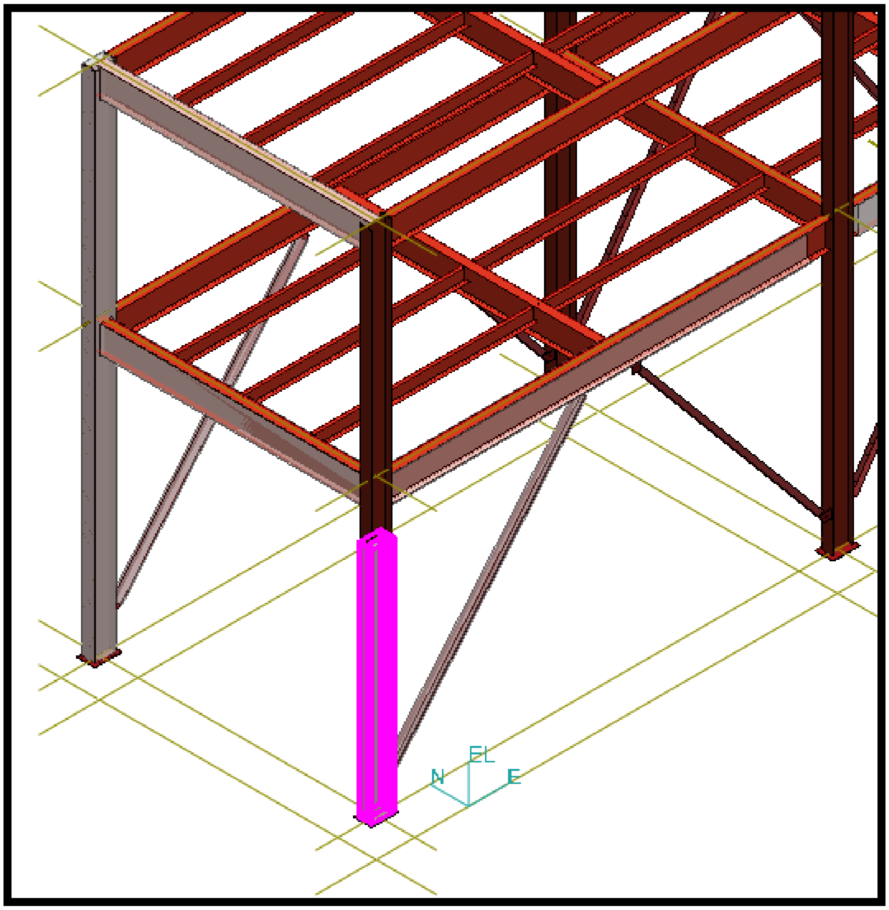


3. Change the setback distance2 to 15 ft and hit OK.

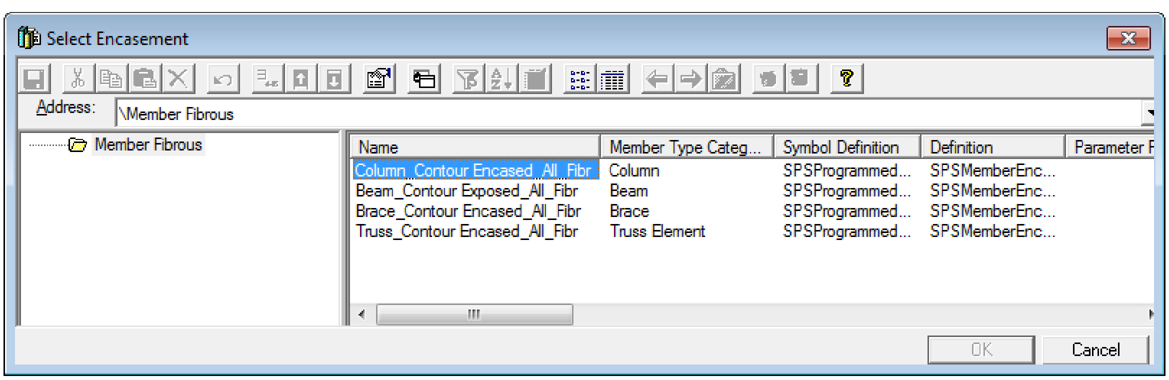


4. Right click to end the command. Your view should resemble the following graphic.

© Copyright 2014 Intergraph Corporation

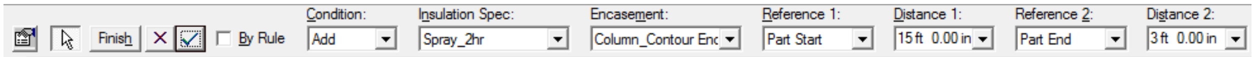


5. Start the *Place fireproofing* command and select the same column used in the previous step.
6. Change the Insulation Spec to *Spray_2hr.*
7. Go to More in the encasement menu. Select *Column_Contour_Encasement_All_Fibr.* Your view should resemble the following graphic.

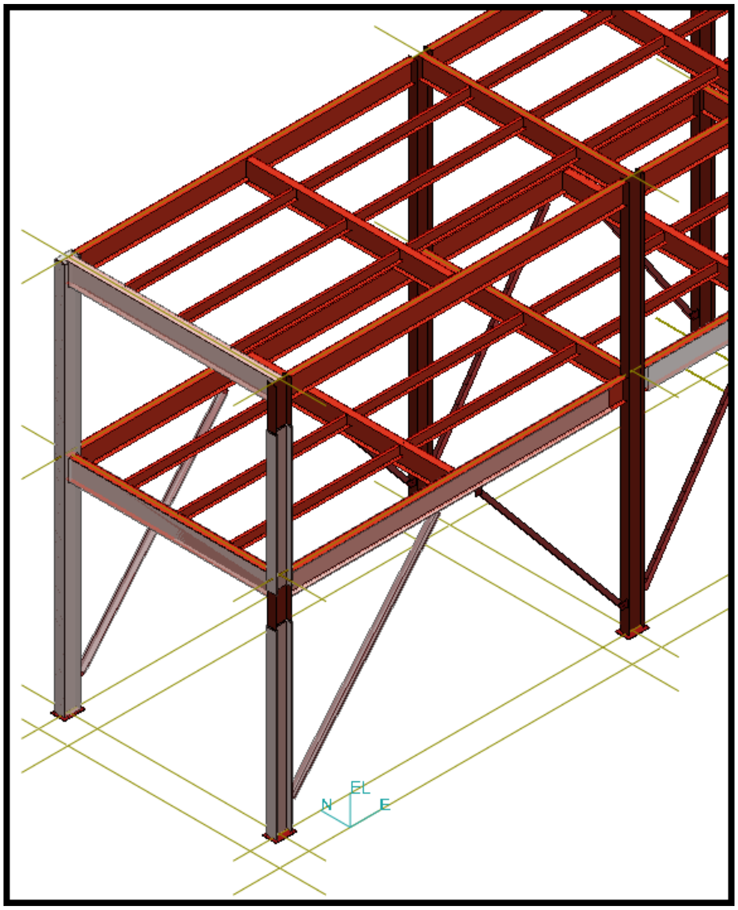


8. Hit OK.
9. Select "*Part Start*" in Reference 1.
10. Enter "15 ft" as Distance 1.
11. Select "*Part End*" in Reference 2.

12. Enter “3 ft” as Distance 2.



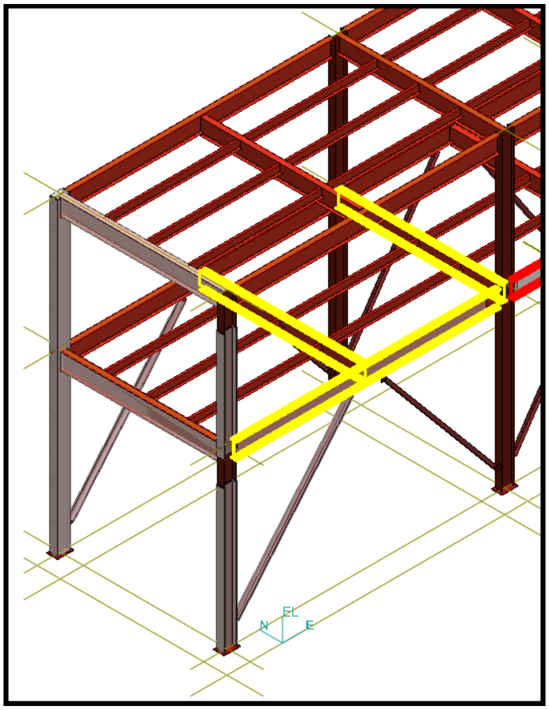
13. Hit the Accept button and hit Finish. Your view should resemble the following graphic.



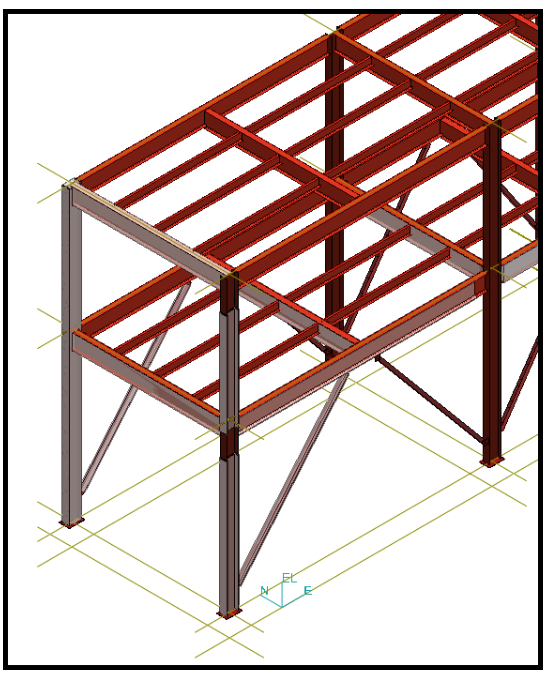
14. Hit Cancel.

Part E: Retain fireproofing.

1. Select the *Place Fireproofing* command.
2. Select beams (one beam on which fireproofing has already been applied and two other beams that have no fireproofing). Your view should resemble the following graphic.

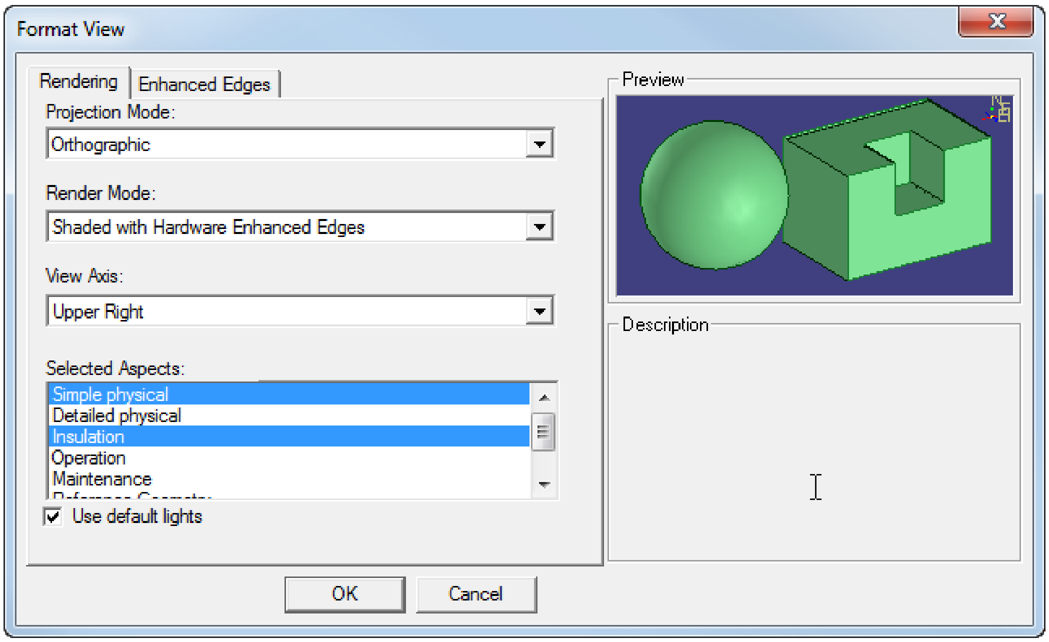


3. Change the *Condition* to “*Retain*” in the toolbar.
4. Select the *Insulation Spec* as Conc_1hr.
5. Select “By Rule”.
6. Hit Finish. Warning will be shown. Hit OK. Hit Cancel. Fireproofing is retained on the existing beam and fireproofing is applied on the other two beams. Your view should resemble the following graphic.



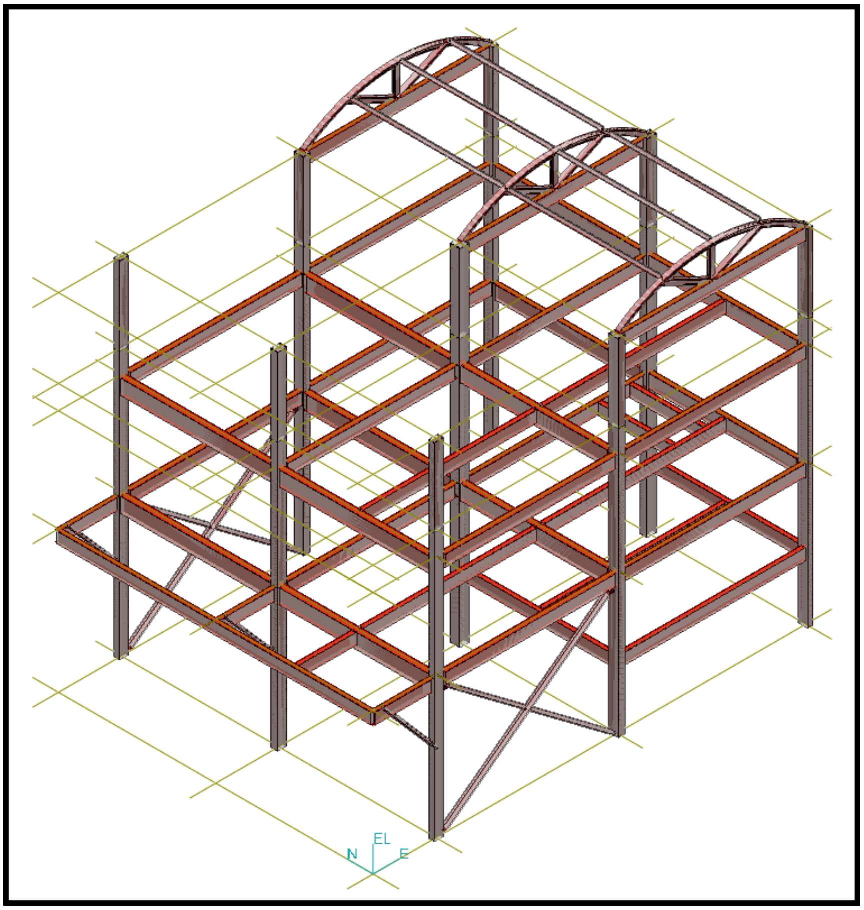
Part F: Place fireproofing for the entire structure.

1. Define your workspace to include the A2 → U03 and CS → U03 CS system.
2. Set your active permissions to "*Structural*".
3. Set the locate filter to Member Systems.
4. Go to *Format* → *View*. Select Insulation in the rendering tab.



5. Your view should resemble the following graphic. Now, hit OK.
6. Select "*Place Fireproofing*" command.
7. Fence select the entire structure.
7. Select the *Condition* as "*Add*" in the toolbar.
8. Select the *Insulation Spec* as Conc_1hr.
9. Select "By Rule".
8. Hit Finish. Hit OK. Hit Cancel. Fireproofing is applied to the entire structure. Your view should resemble the following graphic.

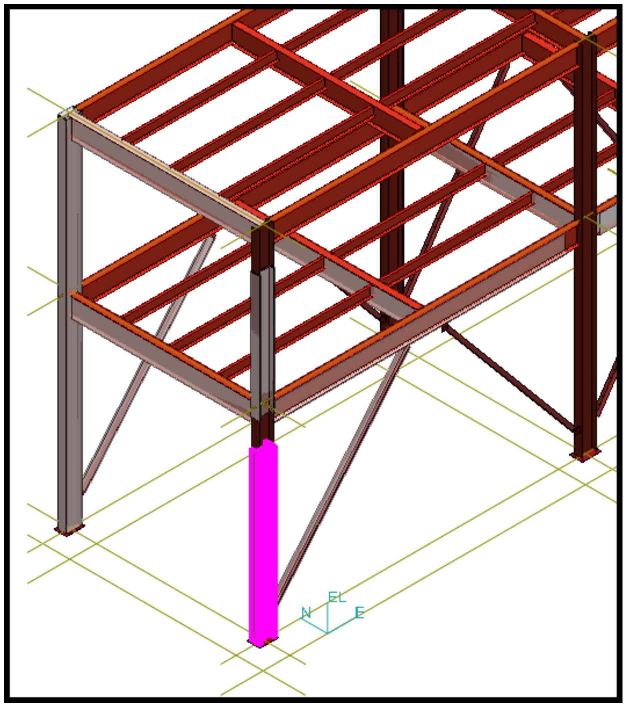
© Copyright 2014 Intergraph Corporation



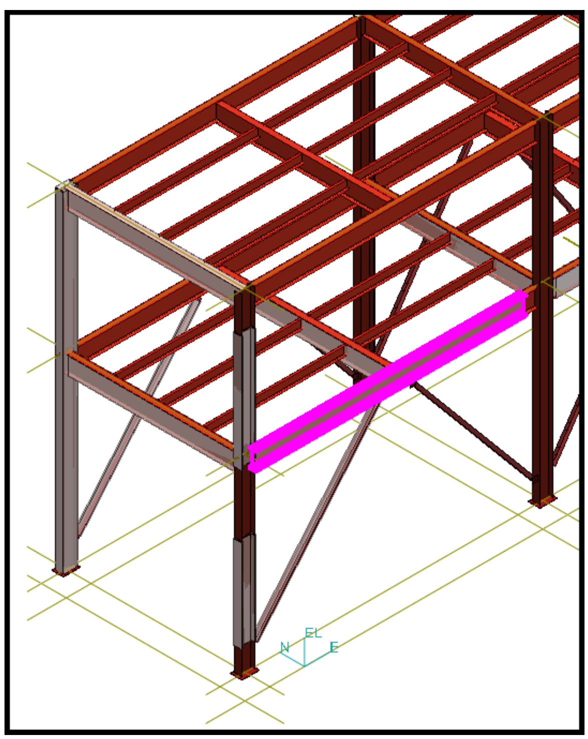
Part G: Modification.

1. Define your workspace to include the A2 → U02 and CS → U02 CS system.
2. Set the Locate Filter to *Insulation.* 

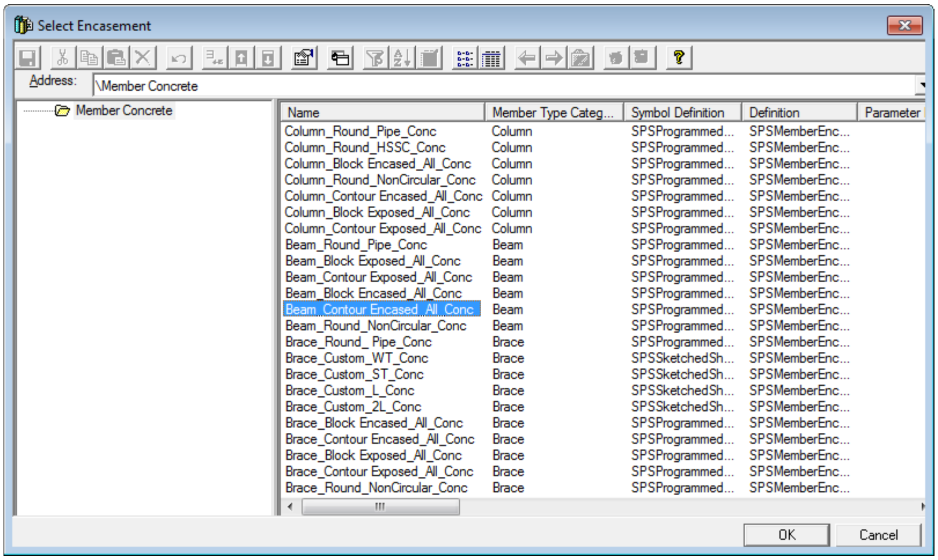
3. Select the insulation as shown in the image below,



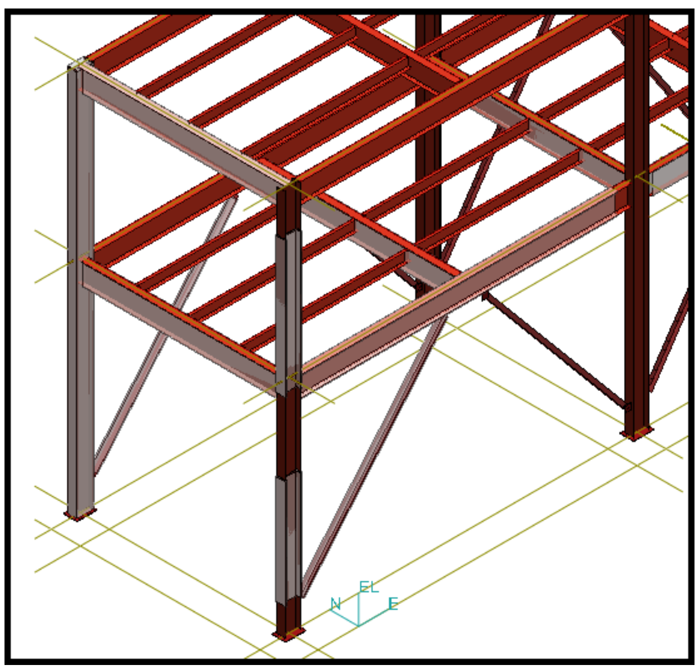
4. Change the *Part Start* distance from 0 ft to 2 ft.
5. Change the *Part End* distance from 15 ft to 18 ft.
6. Select the insulation for the beam as shown in the image below,



7. Go to More from the encasement pull down list. Your view should resemble the following graphic.

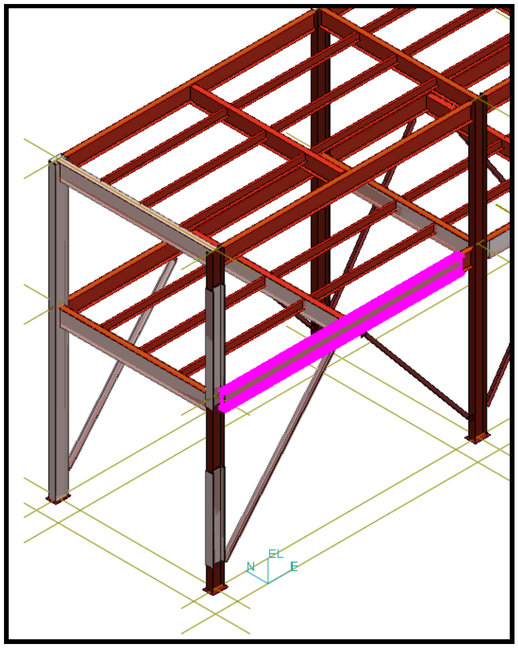


8. Change the encasement from "*Beam_Contour_Exposed_All_Conc*" to "*Beam_Contour_Encased_All_Conc*". Hit OK.
9. The encasement has changed. Your view should resemble the following graphic.

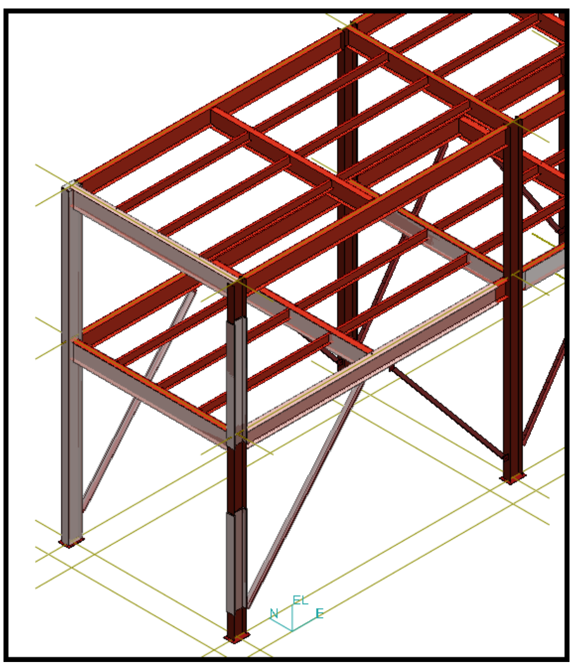


10. Set the Select filter to *Member Parts.*
11. Select the beam as shown in the image below,

© Copyright 2014 Intergraph Corporation



12. Change the section size to W16x36.
13. Insulation automatically adjusts to the new section size. Your view should resemble the following graphic.



14. Go to View → To Do List. There are no entries.
15. Go to Edit → Undo Modify Member or hit Ctrl + Z.
16. Go to *Format* → *View*. Deselect Insulation in the rendering tab and hit OK.

LAB-10: Slabs

Objectives

After completing this lab, you will be able to:

- Understand the slab entities and relationships
- Place Slabs using different options
- Place a slab by using the most common methods to define its plane and boundary.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

Overview:

The **Place Slab** command is really a misnomer. You can use the **Place Slab** command to model any type of planar, constant-thickness solid object such as actual floor slabs, steel grating, steel plate, and roofing. The slab properties define the type of solid object that you place by using the **Place Slab** command. The properties are used in your material reports. Your Catalog administrator can define the different types of slabs available for placement including specific properties associated with those types.

Placing a slab involves two steps:

1. Defining a plane for the slab relative to other objects in the model.
2. Defining the boundary of the slab with optional relationships to other objects in the model.

The methods of defining the location of the slab plane are:

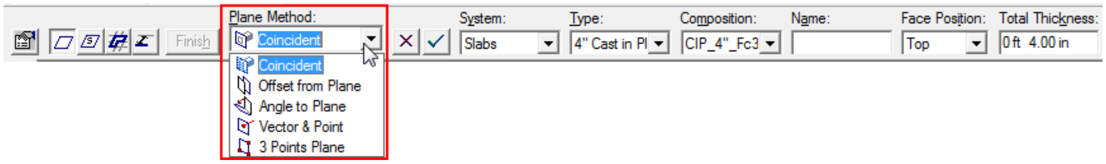
- **Coincident**: Defines a plane coincident to another plane. Use this method when you want the slab to remain coincident with another plane. For example, you can use this method to place a slab to remain coincident with an elevation plane of your grid system.

- **Offset from Plane**: Defines a plane at a specified offset distance from another plane. Use this method when you want the slab to remain at a given offset from another plane. For example, you can use this method to place a slab to remain 4 ft above the second storey elevation plane.

- **Angle to Plane**: Defines a plane at a specified angle or slope to another plane. Use this method when you need to place a sloped slab. For example, you can use this method to place a sloped slab with reference to the ground elevation plane and at any elevation.

- **Vector & Point**: Defines a plane by using two points to define the normal vector for the plane and a third point to define the plane position along the vector. Use this method when you need to orient the surface normal to a vector you know. This is useful in cases where you use the **Place Slab** command to model complex steel plate structures.

- **3 Points Plane**: Defines a plane using three points. Use this method when you just want to position the slab at a specific location in space or relative to specific points along existing design objects.



All of these plane creation methods create persistent relationships to the existing planes or key-points that you identify when you define the plane. If the related design object changes, your slab position will change. Or, if you attempt to move the slab, the movement will be restricted by the relationships you have defined. For example, if you have created a slab coincident with the ground elevation plane of your Grid system, you can move the slab in the horizontal plane but cannot change the slab elevation.

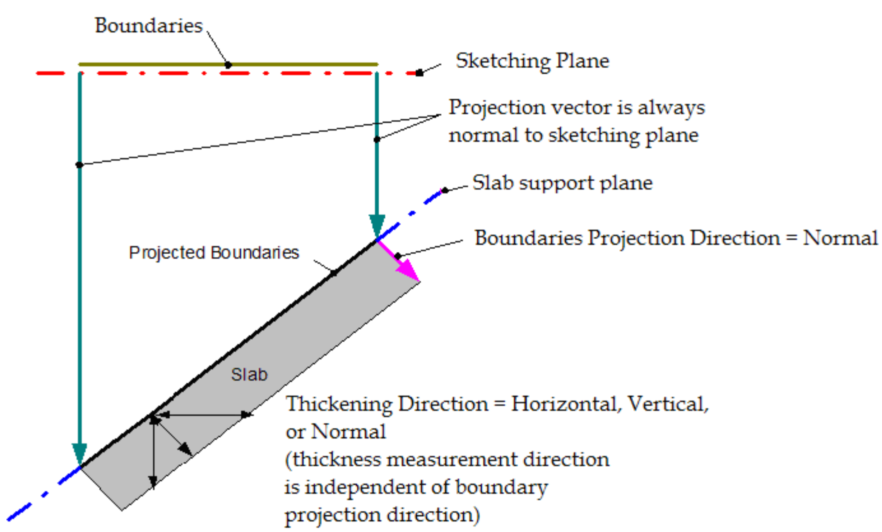
You can change the referenced parent plane by selecting the **Define Plane** smartstep on the slab **Edit** ribbon and selecting a different plane.

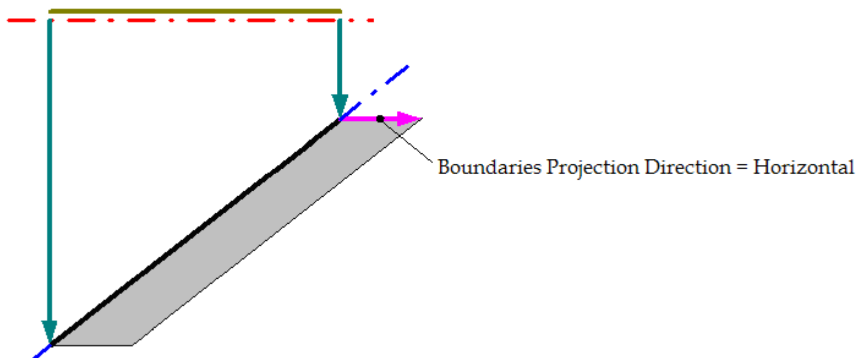
If no associative points were created when you defined the three-point plane, you can move the slab to exactly where you want it. If associative points were established, you cannot move the plane location. You can work around this problem by using the Copy and Paste options. Copy the slab and then paste it. In the **Paste** dialog box, select the **Delete Optional** option. This eliminates the associative point relationships and allows you to position the slab anywhere you want. You can then delete the original slab.

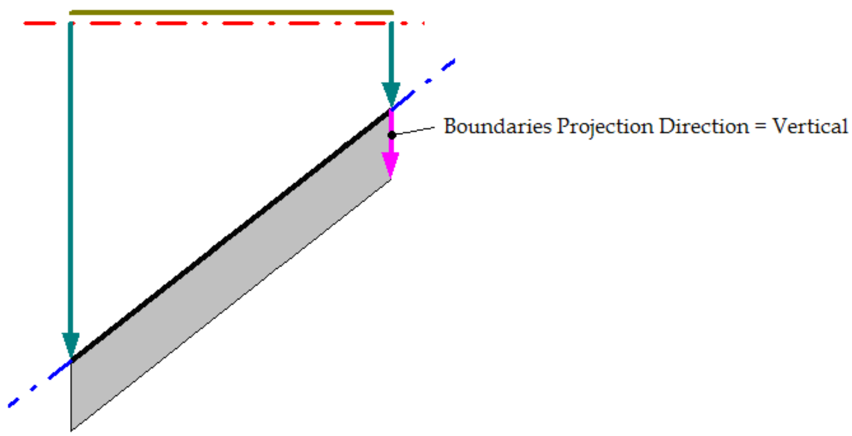
To define the boundaries of a slab, you can:

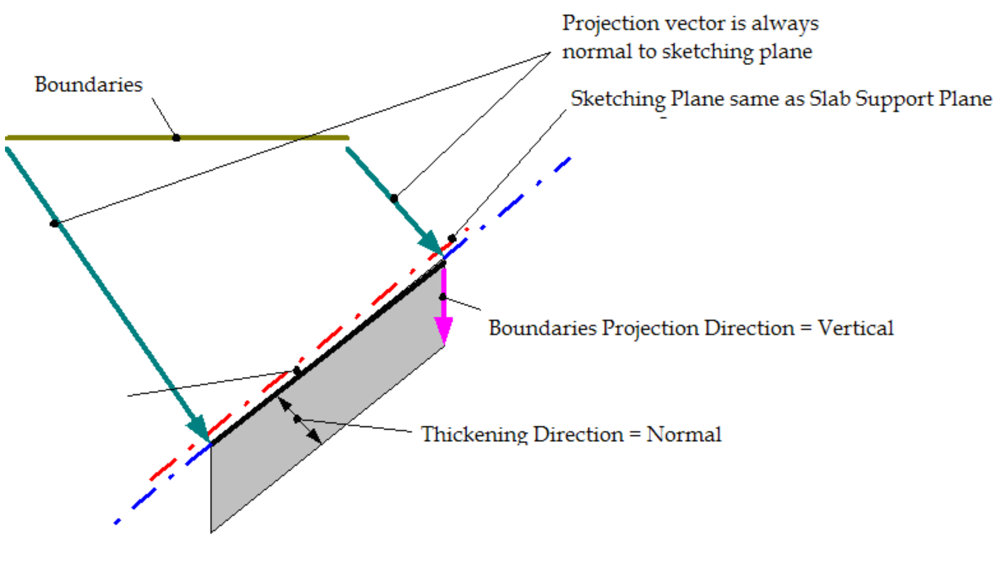
- Select boundary objects in the structure such as grid lines, members, the edges of members, the edges of equipment, the faces of other slabs, and other objects in a model.
- Sketch boundaries by using the 2D or 3D sketching options.

The following illustrates the effect of the indicated slab properties on the construction of the slab.





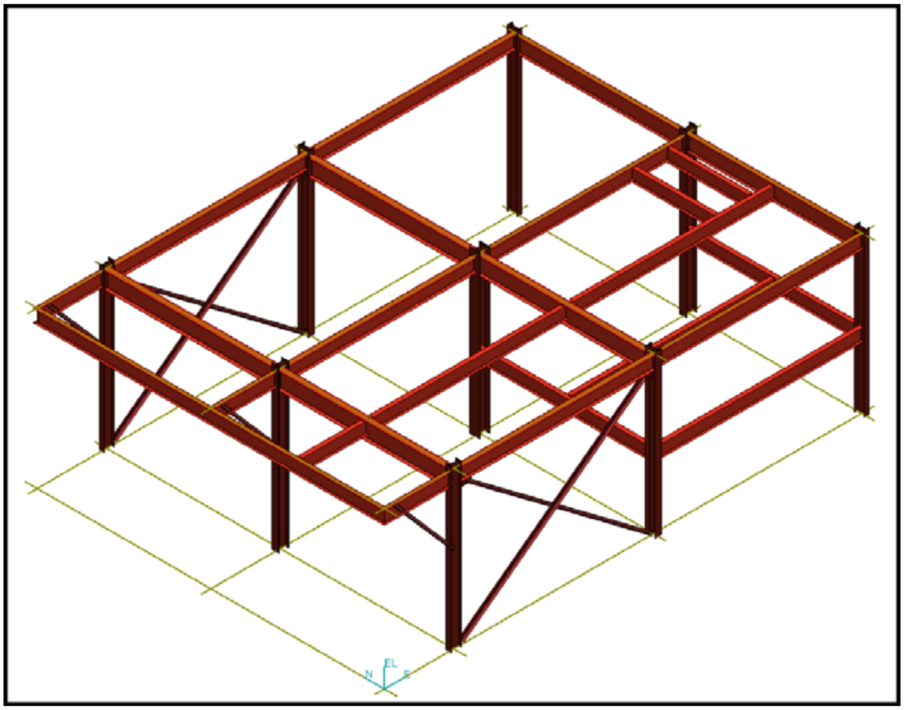




This session will cover the definition of the slab position by using the Coincident and 3 Points Plane methods. It will also cover the definition of boundaries by selecting boundary objects and by using the 2D and 3D sketching options.

Part I: Place Slab on First Floor:

1. Re-define your workspace to include the A2 -> U03 and CS -> U03 CS systems.
2. Make sure the Active Permission Group is set to *Structural.*
3. Set the locate filter to Member Systems.
4. Use Clip by Object command to isolate the beams and the columns in the first floor as shown below:



5. Select Place Slab command to place one slab on the first floor.
6. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U03 -> Structural -> Slabs
Slab Type:	4” Elevated slab - Composite
Composition:	4”_Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary

Boundary Offset Reference: Outer port-face of a bounding member

7. Select Elevation Plane at 18’- 0” for the support plane. Click “Accept” button.
8. Select the boundaries as shown in Figure 1. Click “Accept” button.

   Note: You don’t have to select every member to define the boundaries of the slab. When you select a member, the system will extend its axis until intersect to a neighbor boundary. The system switches to the solve ambiguity step automatically if ambiguous boundaries exist after clicking “Accept” button.

   If a boundary is in the select set and the system does not need that boundary to resolve the ambiguity, then, the boundary is displayed in red and all possible bounded areas appear in the graphic view outlined in green. You must move the mouse over a bounded area, and then left mouse *click to select that bounded area. The selected area highlights in yellow. Continue to select bounded areas until you have defined the entire bounded area that you want.*

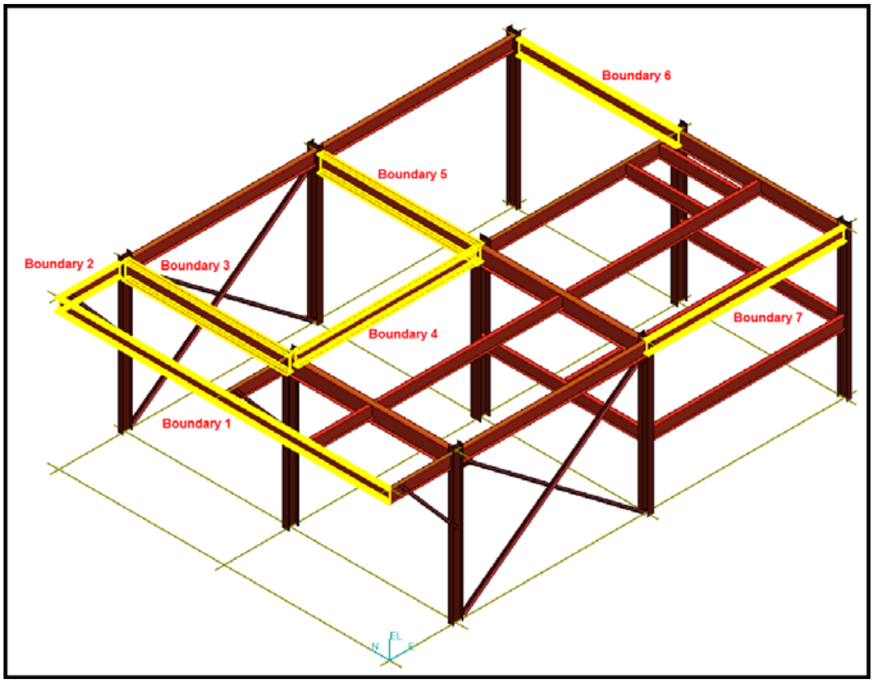
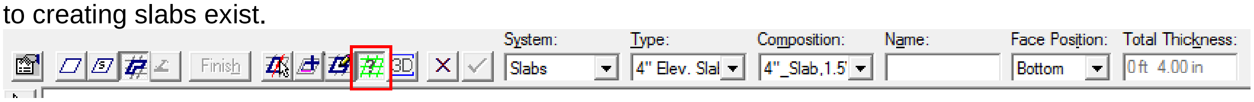


Figure 1 – ISO View of the First Floor Frame

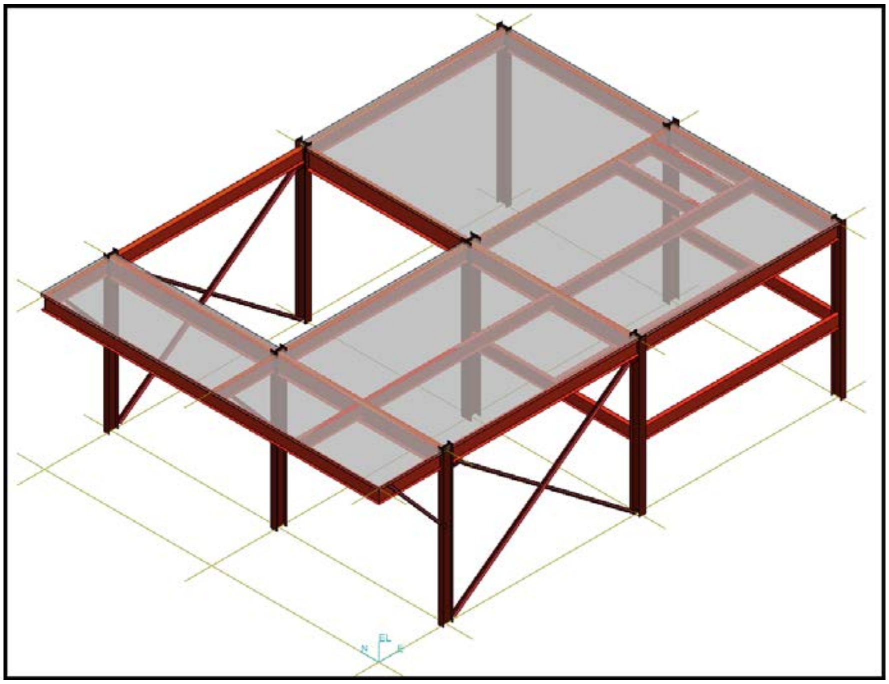
9. At this point, the system will note that there are ambiguous boundaries defined and multiple solutions to creating slabs exist.



9. Move the mouse over a bounded area, and then click to select that bounded area. Continue to select a bounded area until none of the boundary selected is not red.

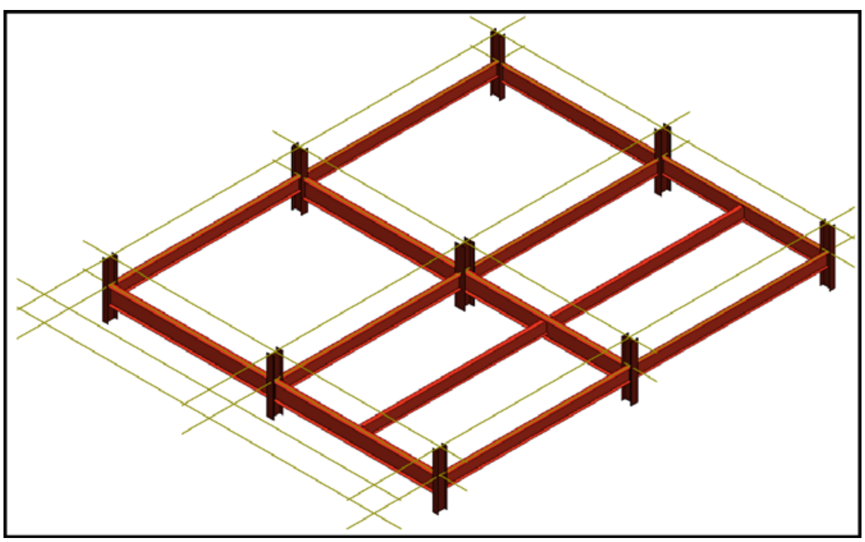
10. Skip Define the Boundaries offsets step

11. Hit “Finish” button to commit the transaction. Your View should now resemble the following graphic:



Part II: Place Slab on Second Floor:

1. Select View-> Clear Clipping to remove the clipping volume.
2. Use Clip by Object command to isolate the beams and the columns in the second floor as shown below:



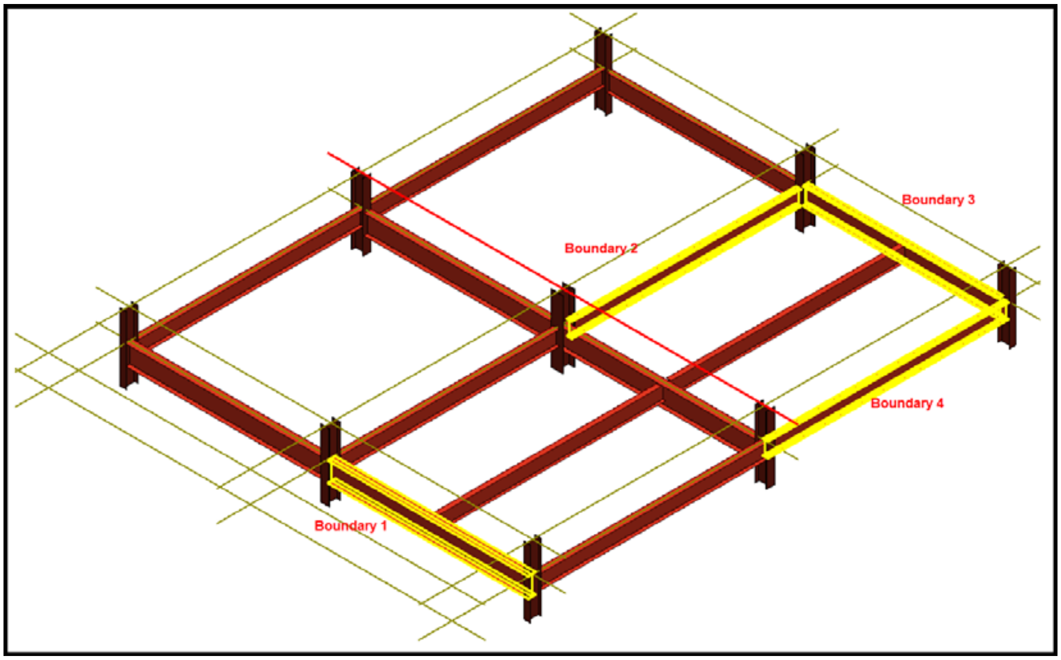
3. Select Place Slab command to place one slab on the second floor.
4. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U03 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite

© Copyright 2014 Intergraph Corporation

Composition: 4"_Slab, 1.5VL22
Face Position: Bottom
Priority: Primary
Boundary Offset Reference: Outer port-face of a bounding member

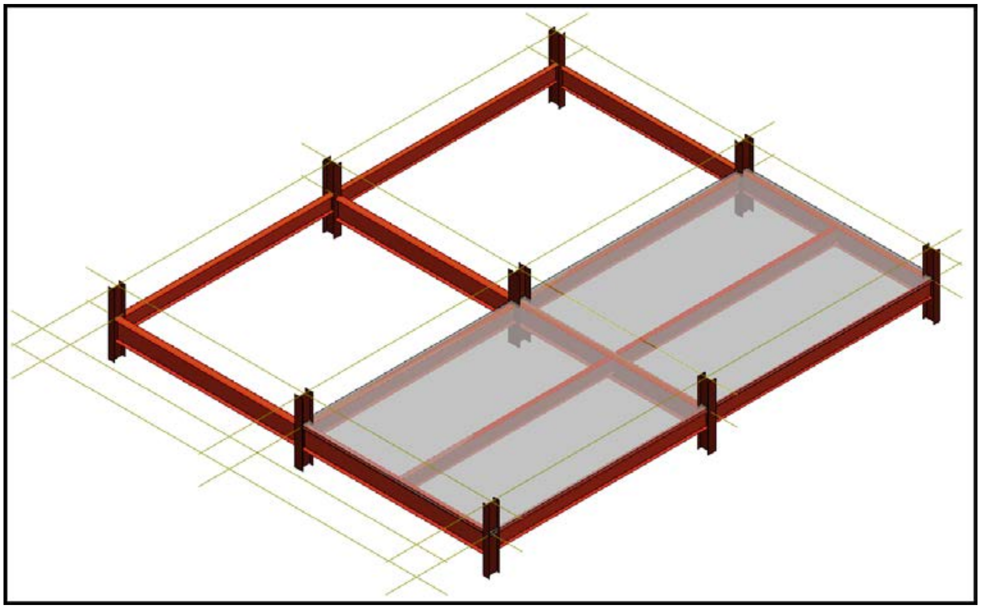
5. Select Elevation Plane at 31'- 0" for the support plane. Click "Accept" button.
6. Select the boundaries as shown in Figure below.



7. Skip Define the Boundaries offsets step.
8. Hit "Finish" button to commit the transaction.

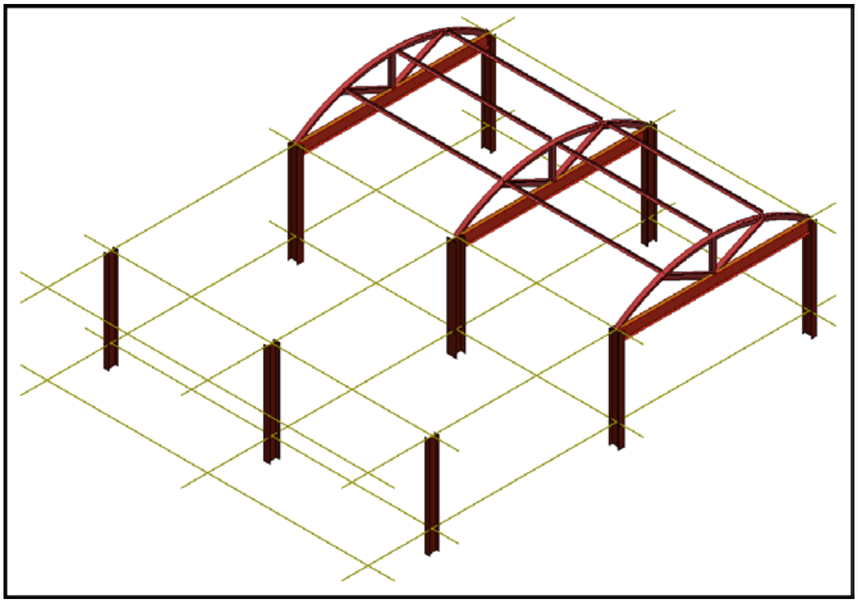
   Your View should now resemble the following graphic:

© Copyright 2014 Intergraph Corporation



Part III: Roof Deck

1. Select View-> Clear Clipping to remove the clipping volume.
2. Use Clip by Object command to isolate the beams and the columns in the roof deck as shown below:



3. Select Place Slab command.
4. Select Plane Method: Offset from a Plane.
5. Select Elevation Plane at 44'- 0" for the support plane using the Workspace Explorer.
6. Key in -2'-0" for offset.

7. Select “Accept” button.
8. Set the slab parameters as follows:

System:	A2 -> U03 -> Structural -> Slabs
Slab Type:	Roof Deck
Composition:	RD_1.5B24
Face Position:	Bottom
Priority:	Primary

9. Select the boundaries as shown in Figure 2

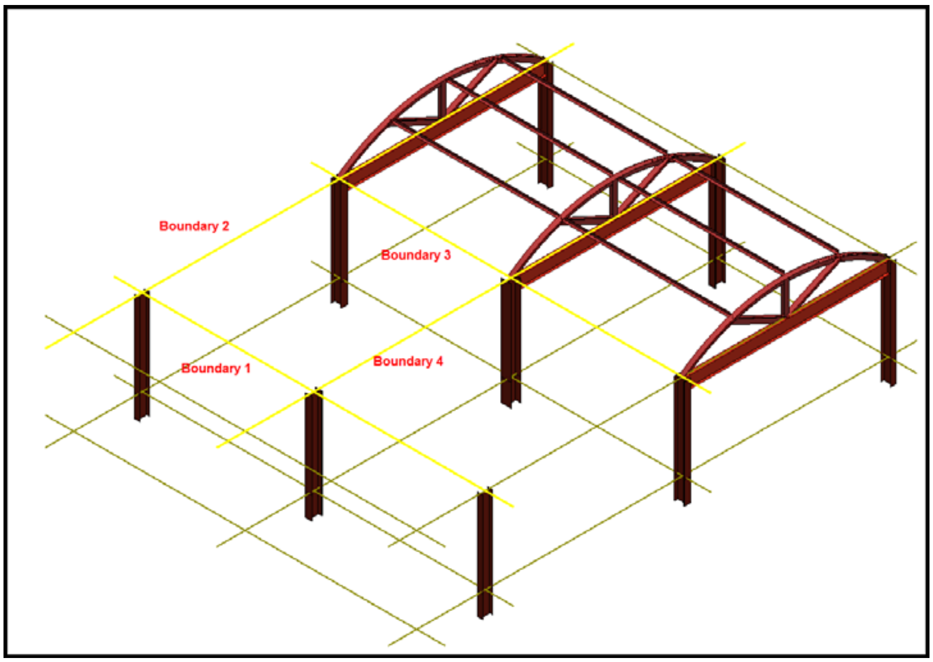


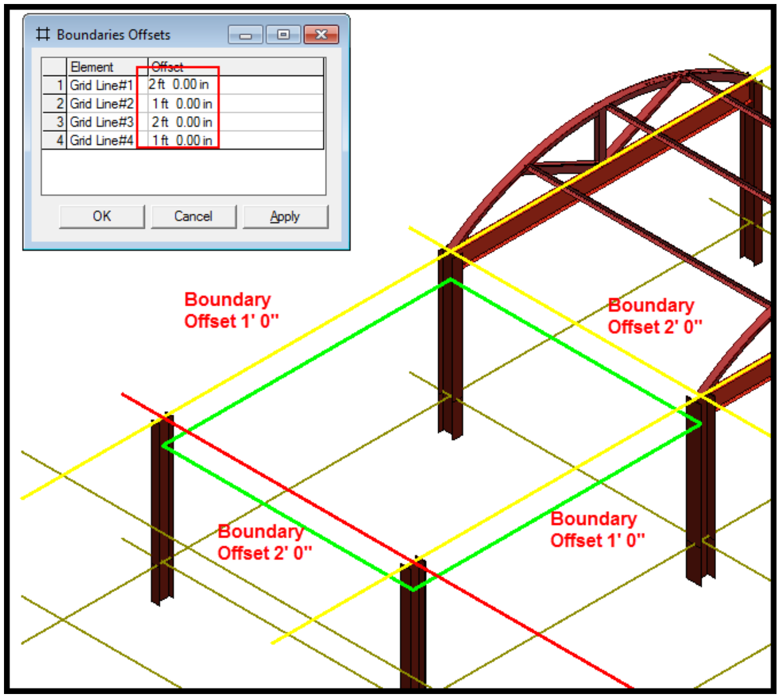
Figure 2 – ISO View of the Roof Deck

10. Select “Accept” button.

11. Click the **Define Boundaries Offsets** option on the ribbon bar.

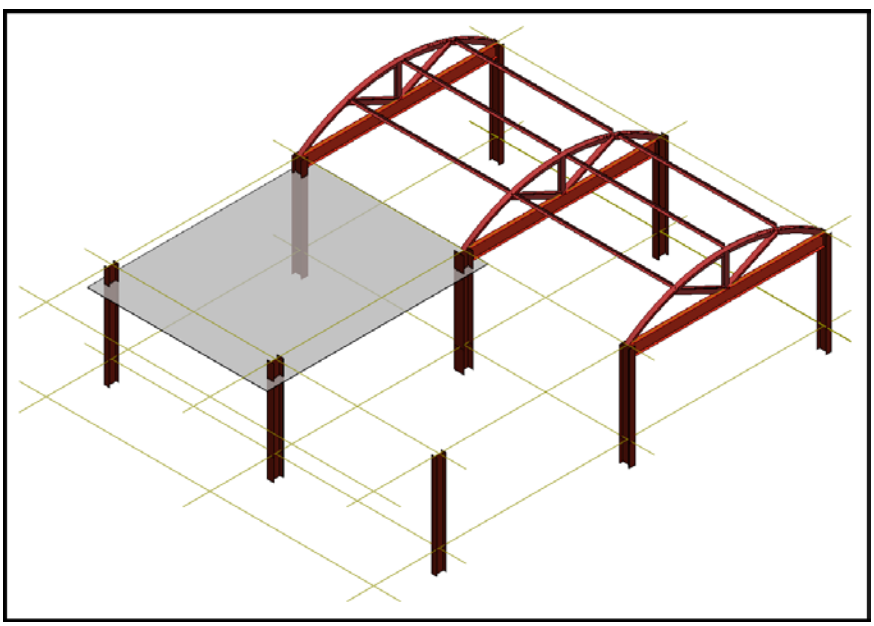
Define the Boundaries offsets step as shown below:

© Copyright 2014 Intergraph Corporation



12. Hit “Finish” button to commit the transaction.

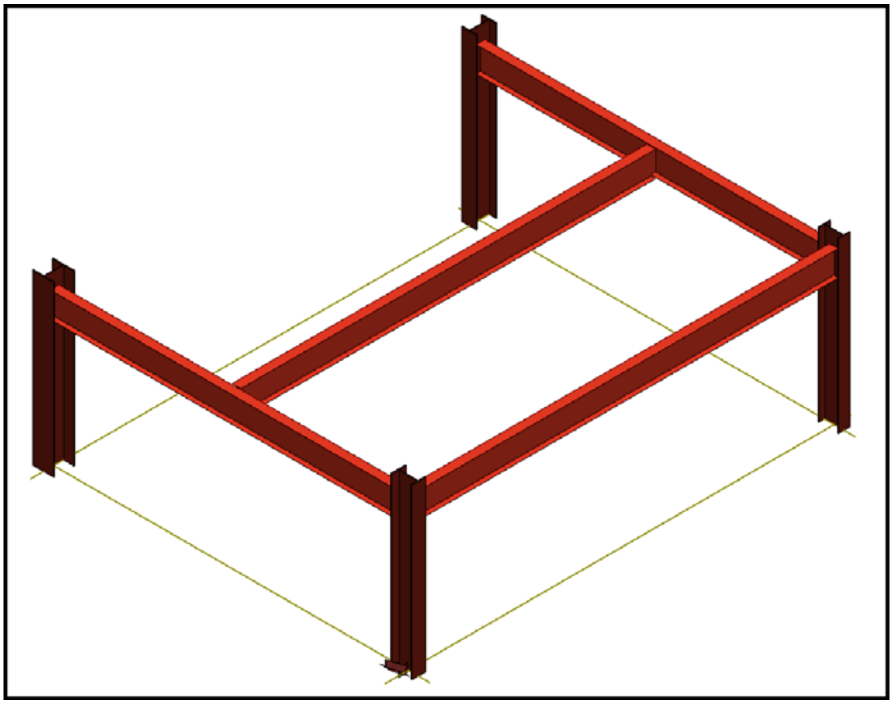
    Your View should now resemble the following graphic:



<u>Part IV - Place a Slab on the Far-East Bay:</u>

1. Select View-> Clear Clipping to remove the clipping volume.

2. Use Clip by Object command to isolate the beams and the columns in the second floor as shown below:



3. Select Place Slab command.
4. Select the Top surface of the beam located at Elevation 8'- 9" for the support plane.
5. Select "Accept" button.
6. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U03 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4"_Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Boundary Offset Reference:	Outer port-face of a bounding member

7. Select the boundaries as shown in figure 3.

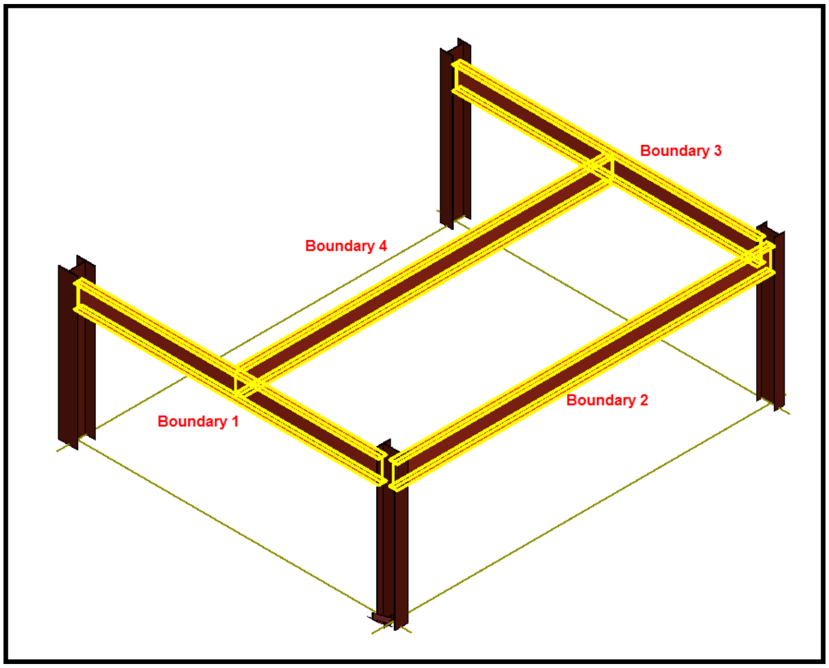
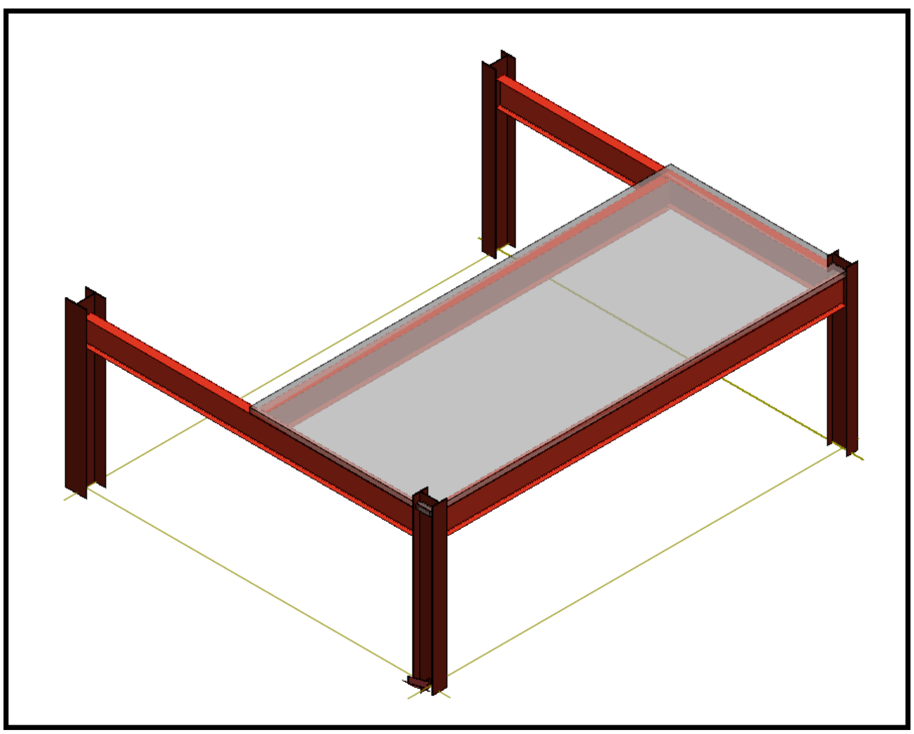


Figure 3 – ISO View of the Far-East Bay

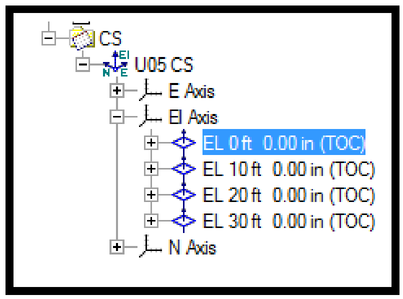
8. Select “Accept” button.
9. Hit “Finish” button to commit the transaction.

   Your View should now resemble the following graphic:



Part V - Sketch a Slab floor for a maintenance building:

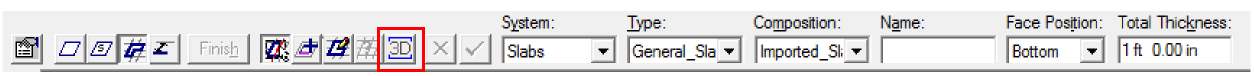
1. Select View-> Clear Clipping to remove the clipping volume.
2. Re-define your workspace to include the A2 -> U05 -> Structure and CS -> U05 CS systems.
3. Select Place Slab Command.
4. Select Elevation 0'- 0" for the support plane using the Workspace Explorer.



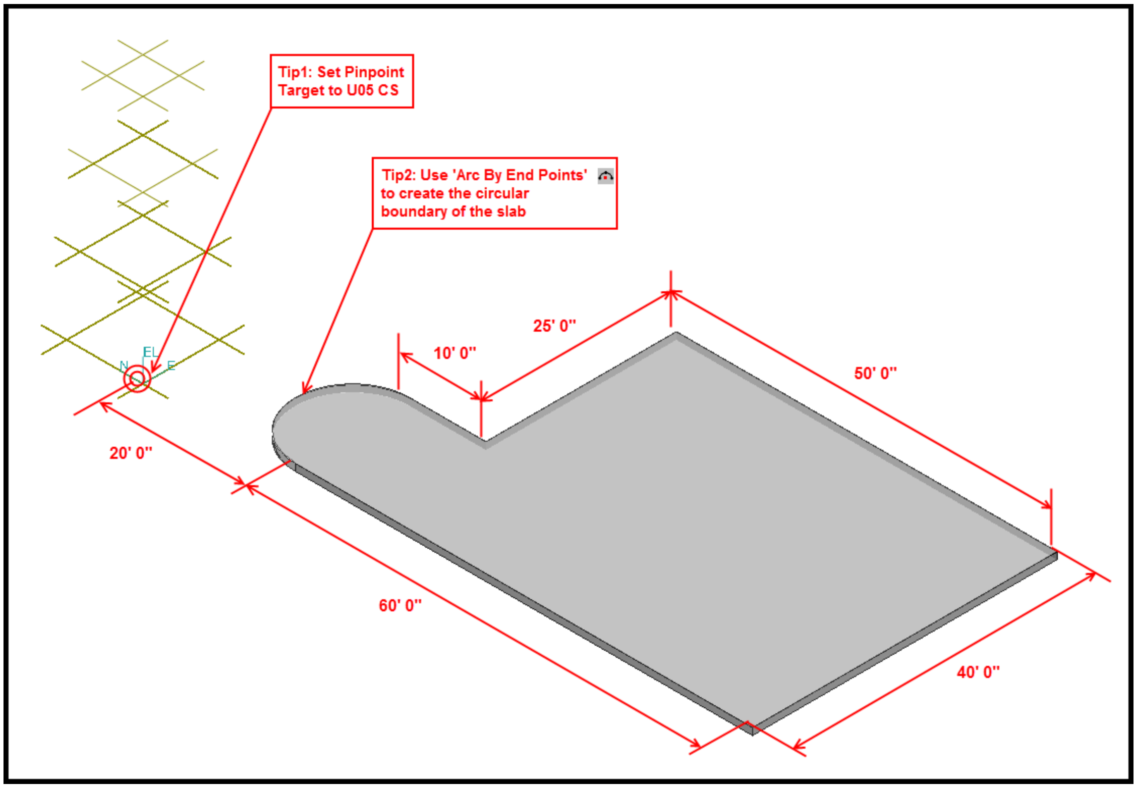
5. Select "Accept" button.
6. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U05 -> Structural -> Slabs
Slab Type:	General_Slab
Composition:	Imported_Slab
Face Position:	Bottom
Total Thickness:	1' - 0"

7. Select the Sketch 3D step.



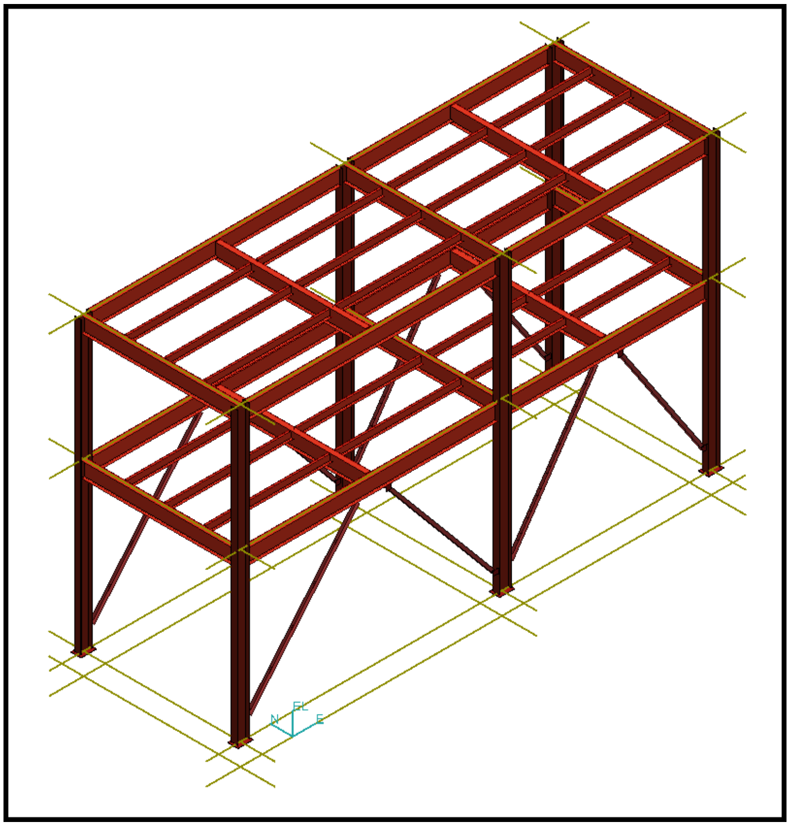
8. Sketch the boundaries of the slab as shown below:



9. Hit “Finish” button to commit the transaction.

<u>Part VI: Sketch Multiple Slabs</u>

1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include the A2 -> U02 and CS -> U02 CS systems and name the filter as U02 & U02 CS.
2. Go to the Structure Task environment.
3. Make sure the Active Permission Group is set to *Structural.*
4. Your view should resemble the following graphic.



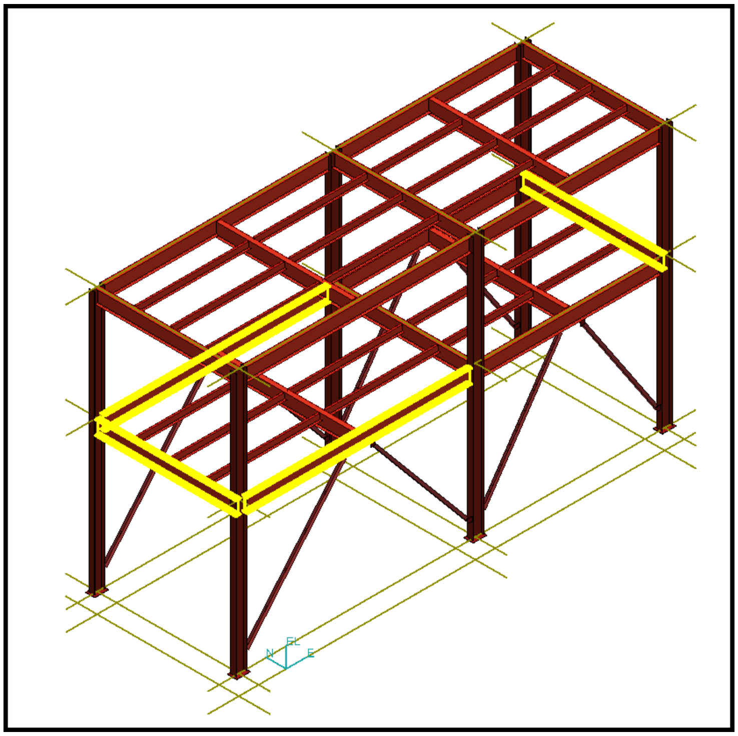
5. Select the *Slabs* command to place multiple slabs.

6. Set the slab placement parameters as follows:

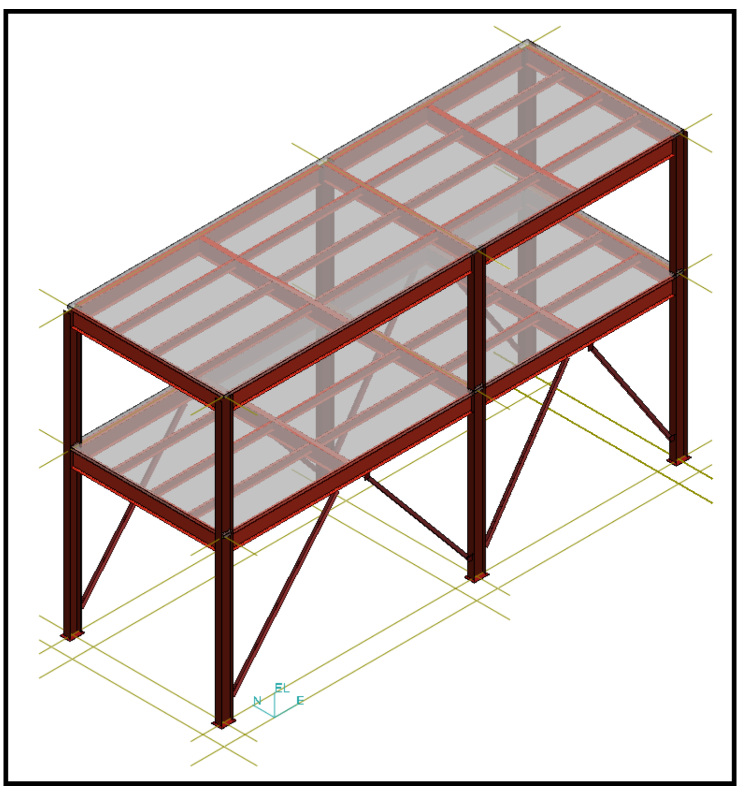
Plane Method:	Coincident
System:	A2-> U02 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4"_Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Boundary Offset Reference:	Inner port-face of a bounding member

7. Select Elevation Plane at 18'- 0" and Elevation Plane 30' – 0'' for the support plane. Click "Accept" button.

8. Select the boundaries as shown in the figure below. Click "Accept" button.



9. Hit Finish. Your view should resemble the following graphic.



LAB-11: Walls

Objectives

After completing this lab, you will be able to:

- Understand the wall entities and relationships
- Place Wall using different options
- Place Architectural objects like windows and doors in the model

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

Overview:

The solid 3D geometry of walls is modeled by defining a path on a plane, the cross-section size, and optionally, selecting the graphic boundaries to trim the top of the wall. You choose the positioning of the cross-section relative to the path by cardinal points defined on the cross- section shape. The wall properties define the layered composition of the wall.

A wall system consists of a path (called a wall run), a set of properties defining the wall construction, and a wall part for the entire wall run. The wall part has no current application. The design is done in preparation for the ability to split a wall into multiple parts for detailed panel modeling. This design is similar to the concept of splitting a member system. Since a wall system must have the same thickness along the wall run, you must create separate wall systems to have walls with different thickness properties. You can connect one wall system to another. At the connection point, the wall geometry is automatically trimmed to represent the corner or tee properties. You can also split a wall system into multiple wall systems, if you need to make changes to a portion of the original wall system after initial modeling.

© Copyright 2014 Intergraph Corporation

The Catalog contains the wall cross-section shapes, material, and construction parts that you can place in the structure. The Catalog administrator can customize the Catalog wall definitions.

The supporting plane of the wall can be an elevation plane, the surface of a slab, or any other planar surface in the model. You can define the path (or wall run) using the 3D Sketch or 2D Sketch methods.

The 3D Sketch method allows you to directly reference 3D model design objects while drawing the path. The 2D Sketch method requires you to specifically select the design objects you want to reference within the Sketch 2D drawing environment. The Sketch 2D drawing environment gives you more drawing functionalities including the ability to define dimensions that automatically adjust the wall size.

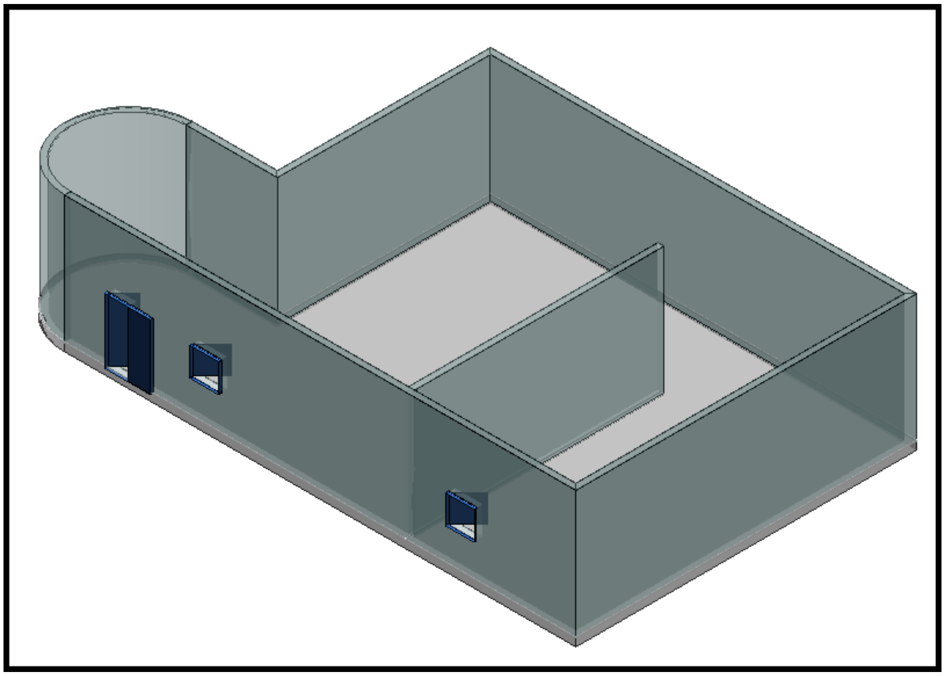
In both the sketching methods, you can efficiently create multiple wall systems as you define the wall run, without exiting the Place Wall command. You can then edit the thickness or other construction properties of the different wall systems you have defined.

You can later split these wall systems into multiple wall systems if you need to change the properties of a portion of the existing wall system. For example, if you want to edit the thickness of one of the walls in a wall system, you can split the wall system into multiple wall systems and then change the thickness of one of the walls.

You can place doors and windows in the wall and cut arbitrary openings by using the Place Opening command. You can get reports on the surface area with or without the openings.

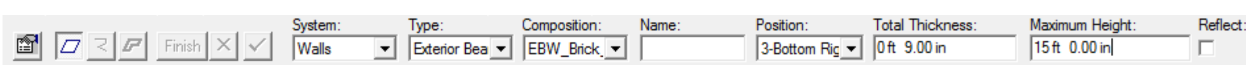
© Copyright 2014 Intergraph Corporation

You will add walls, windows and a door to the maintenance building as shown below:



<u>Part I - Place wall objects to build the maintenance building:</u>

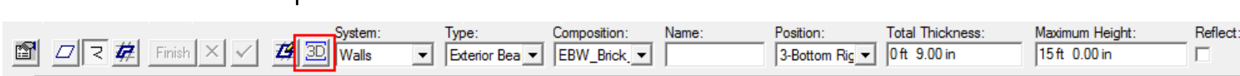
1. Define your workspace to include the A2 -> U05 -> Structure and CS -> U05 CS systems.
2. Activate the PinPoint ribbon bar by Selecting Tools → PinPoint. (Make sure Active Coordinate System is set to Rectangular Coordinate mode).
3. Set the Pinpoint target to U05 CS.
4. Select Place Wall Command to open the SmartStep ribbon bar.



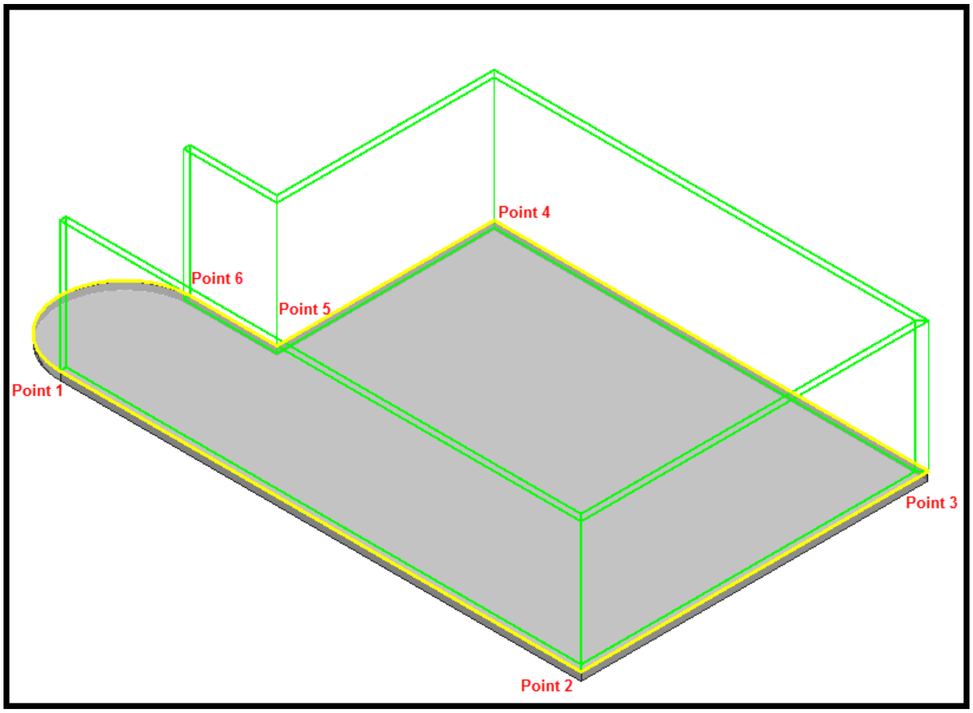
4. Set the wall parameters as follows:

System: A2 → U05 → Structural → Walls
Wall Type: Exterior Bearing Wall - Composite
Composition: EBW_Brick_5”_Air_4”_Bath&Roll_2”_Siding_Vinyl_0.046”
Face Position: 3 - Bottom_Right
Total Thickness: 0’-9”
Maximum Height: 15’-0”

5. Select the top surface of the slab for the sketching plane.
6. Select the Sketch 3D step.

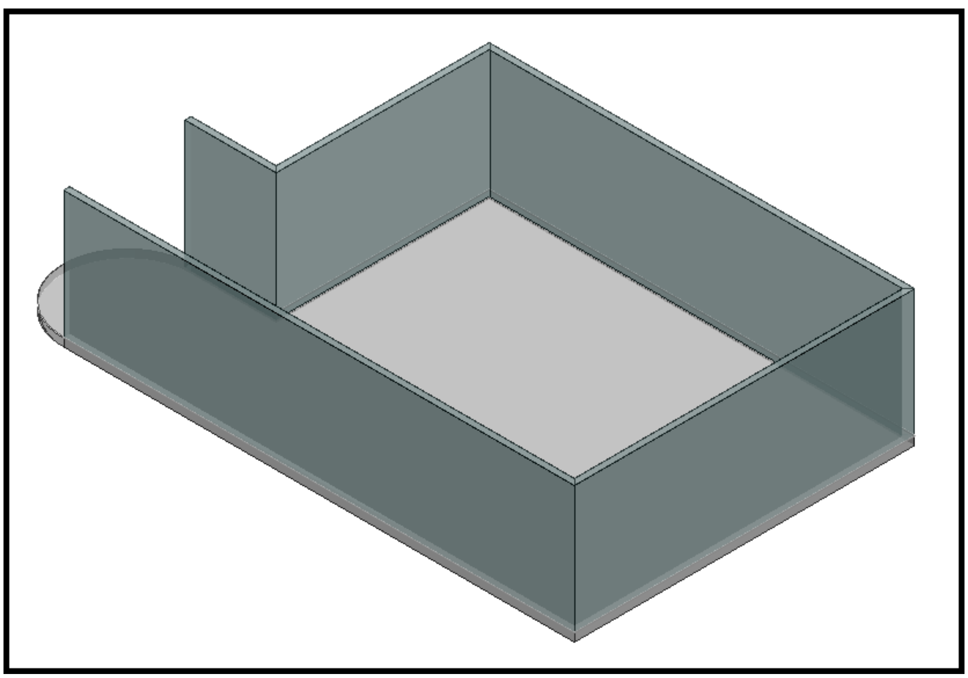


7. Sketch the boundaries of the wall as shown below:



Note: Make sure you get the end point using the Smartsketch service. If you don't select the end points, then the system will not create a relationship to the slab.

8. Hit "Finish Path" button to commit the wall run (path). System returns to the SmartStep ribbon bar of the Place Wall command.
9. Hit "Finish" button to persist the wall part in the model.

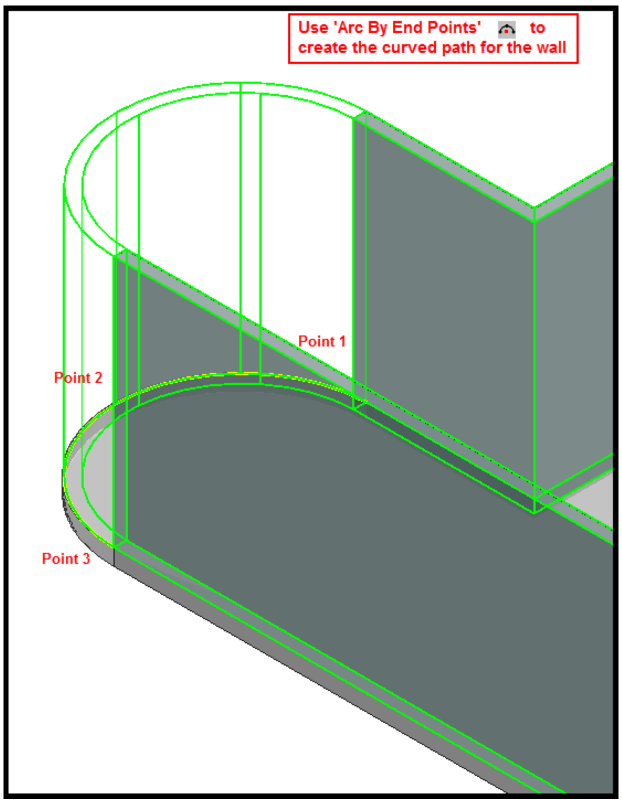


10. Repeat the above step and place a curve wall.
11. Set the wall parameters as follows:

System: A2 → U05 → Structural → Walls
Wall Type: Exterior Bearing Wall - Composite
Composition: EBW_Brick_5"_Air_4"_Batt&Roll_2"_Siding_Vinyl_0.046"
Face Position: 3 Bottom_Right
Total Thickness: 0'-9"
Maximum Height: 15'-0"

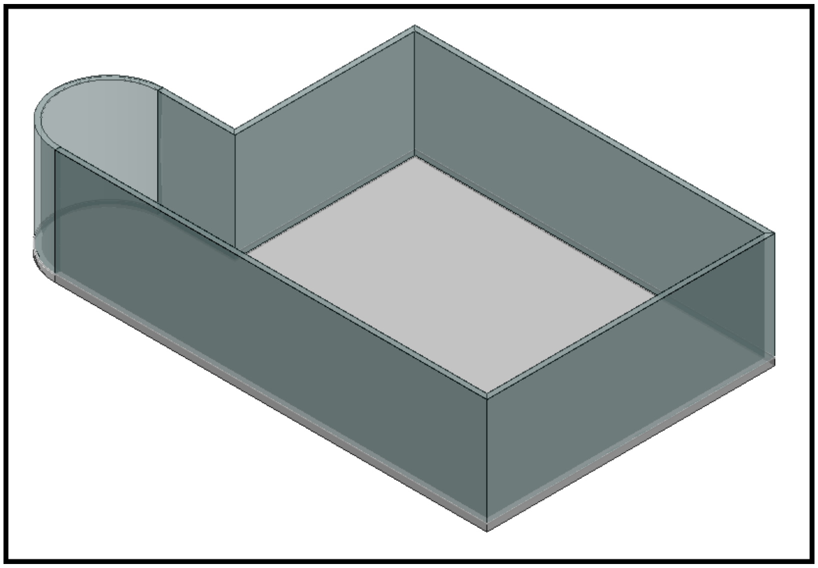
12. Select the top surface of the slab for the sketching plane.
13. During the sketch step, select the path type as Arc by End Points and define three points as shown below:

Note: Use the Function key <F3> to toggle on/off the SmartSketch point on surface when you select end of the wall run (path).



14. Select "Finish Path" button to commit the wall run (path). System returns to the SmartStep ribbon bar of the Place Wall command.
15. Select "Finish" button to persist the wall part in the model.

© Copyright 2014 Intergraph Corporation

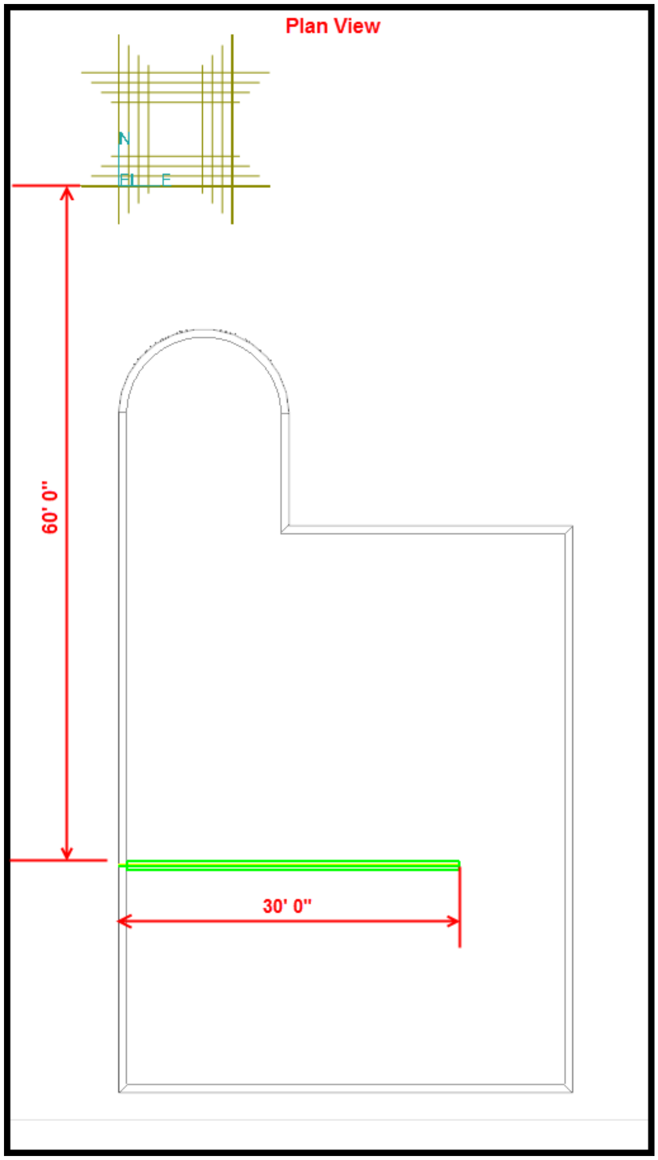


16. Repeat the above step to place a divider wall.
17. Set the wall parameters as follows:

System:	A2 → U05 → Structural → Walls
Wall Type:	Interior Bearing Wall – Non-Composite
Composition:	IBW_Stone_Granite_6”
Face Position:	2 Bottom Center
Total Thickness:	0’-9”
Maximum Height:	15’-0”

18. Select the top surface of the slab for the sketching plane.
19. During the sketch step, select the path type as Line and define the two points as shown below:

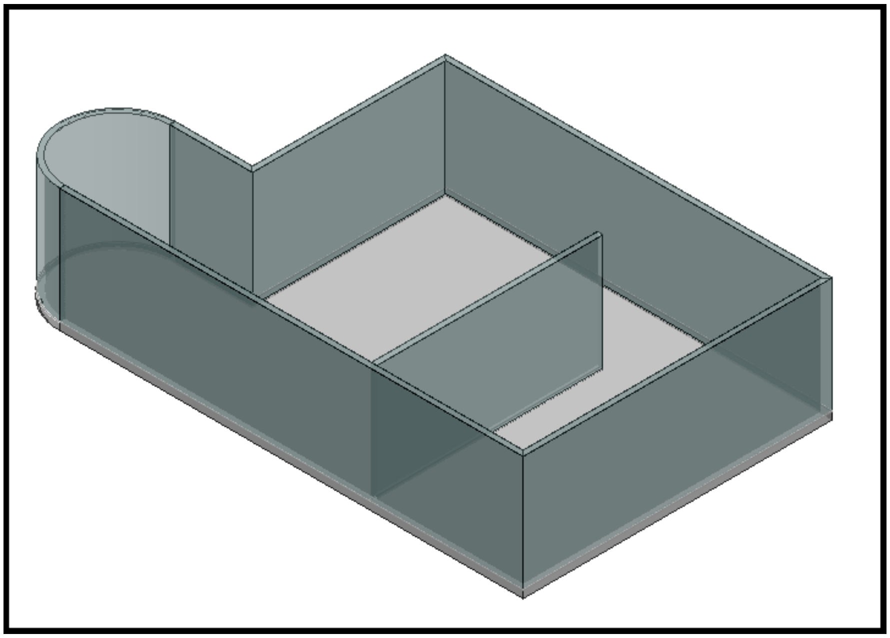
© Copyright 2014 Intergraph Corporation



20. Select “Finish Path” button to commit the wall run (path). System returns to the SmartStep ribbon bar of the Place Wall command.

21. Select “Finish” button to persist the wall part in the model.

    Your View should now resemble the following graphic:



Part II - Place Door and Windows to the maintenance building:

Overview:

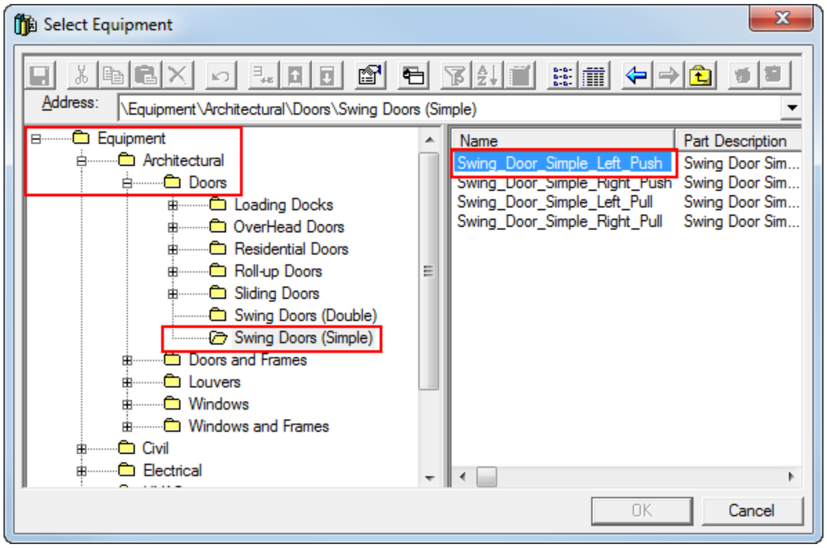
You can place doors, windows, and louvers in walls and slabs in a model by using the **Place Equipment** command and selecting the desired items from the architectural section of the equipment Catalog. Door and window placement generates geometry of the door/window and creates an appropriately sized rough opening in the wall. This equipment-based method of representing design objects permits the Catalog administrator to customize the available doors and windows. To do this, the administrator can use the same methods as used for defining other equipment in the Catalog.

When placing a door, a window, or a louver, identify the point on the surface of the wall where you want the design object to be placed. This point identifies the wall in which the opening will be cut and the position where the design object will be placed on the wall.

You can also add additional positioning relationships if you want to offset the door/window from another surface (refer to the sessions that cover equipment modeling design). It is also effective to simply use **PinPoint** or **Point Along** to define the final position along the wall.

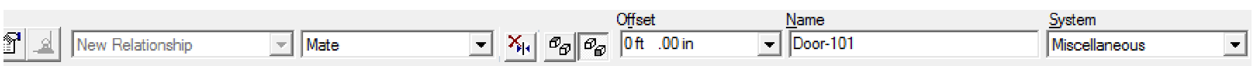
1. Make sure the Active Permission Group is set to *Structural*.
2. Select Place Equipment Command.
3. Expand the equipment hierarchy \Equipment\Architectual\Doors\Swing Doors (Simple) folder until you see the part Swing_Door_Simple_Left_Push. Select the part and click the “OK” button.

© Copyright 2014 Intergraph Corporation



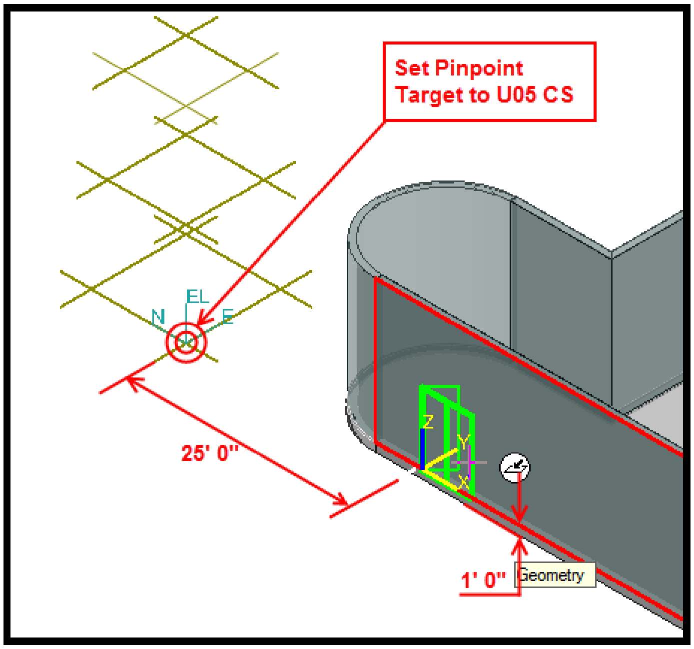
4. Equipment Property page is displayed on your screen. Hit “OK” button.
5. Go to the equipment ribbon bar and make sure the positional relationship is set to Mate.
6. Set the following parameters:

System: A2 -> U05 -> Structural -> Miscellaneous
Name: Door-101

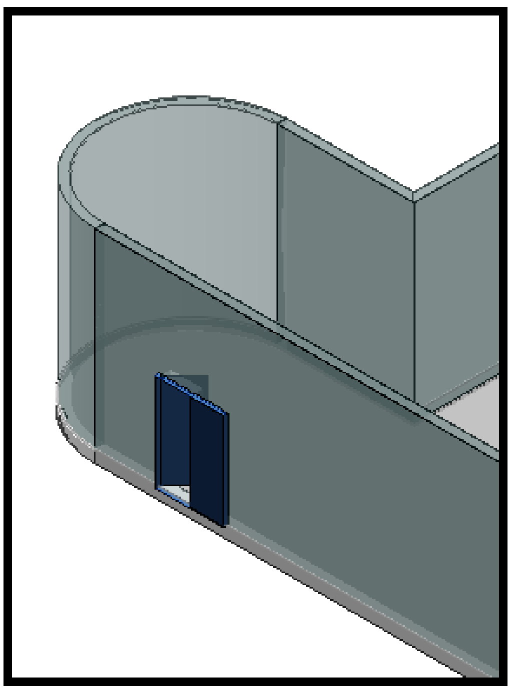


7. Go to the PinPoint ribbon bar and key in -25’ for North and 1’for Elevation. Don’t keyin anything or lock the Easting field.

8. Use the Left Arrow key to rotate the equipment 90 deg about the z-axis.
9. Move the cursor over the wall object as shown below:



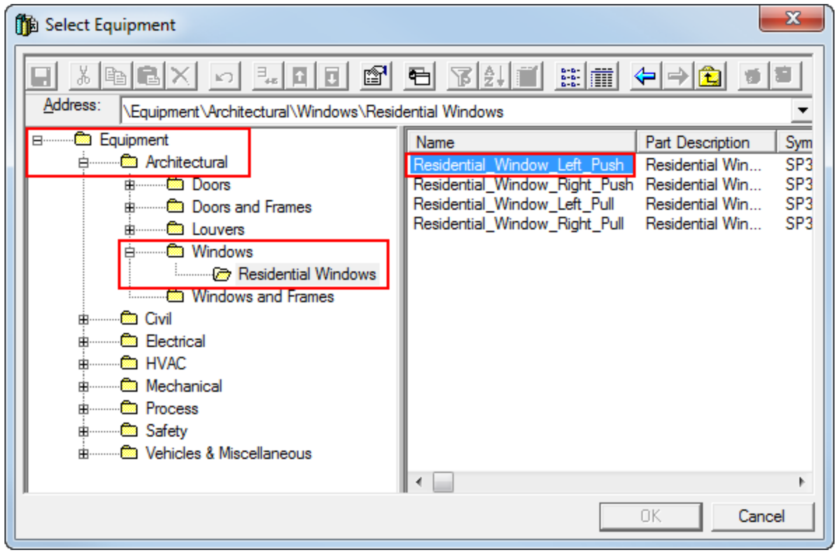
10. Left Mouse Click to place the door on the model.



Repeat the above workflow to place the first window.

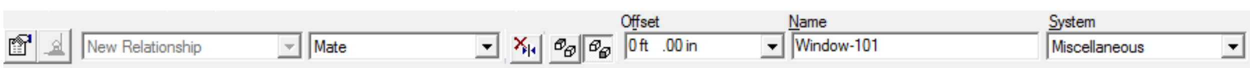
11. Select Place Equipment Command.

12. Expand the Equipment Hierarchy \Equipment\Architectural\Windows\Residential Windows folder until you see the part Residential_Windows_Left_Push. Select the part and click the “OK” button.



13. Equipment Property page is displayed on your screen. Hit “OK” button.
14. Go to the equipment ribbon bar and make sure the positional relationship is set to Mate
15. Set the following parameters:

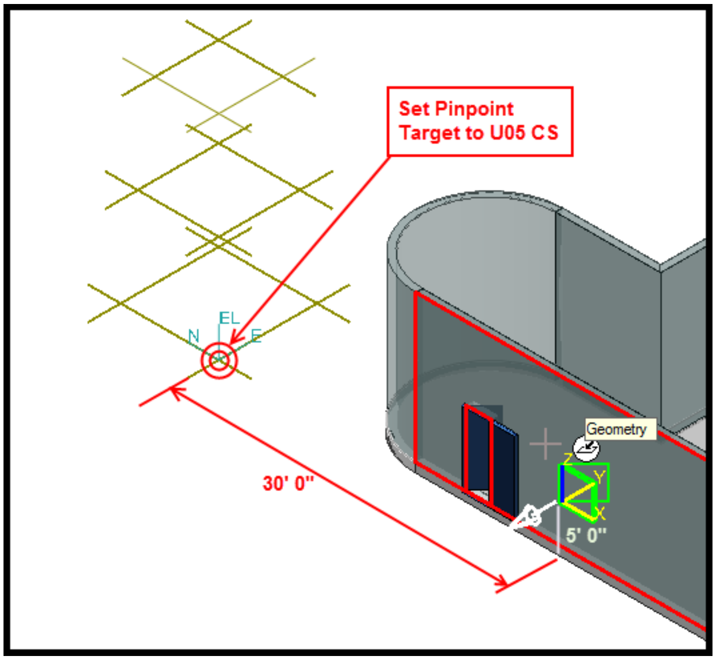
System: A2 -> U05 -> Structural -> Miscellaneous
Name: Window-101



16. Go to the PinPoint ribbon bar and key in -35’ for North and 5’for Elevation. Don’t keyin anything or lock the Easting field.

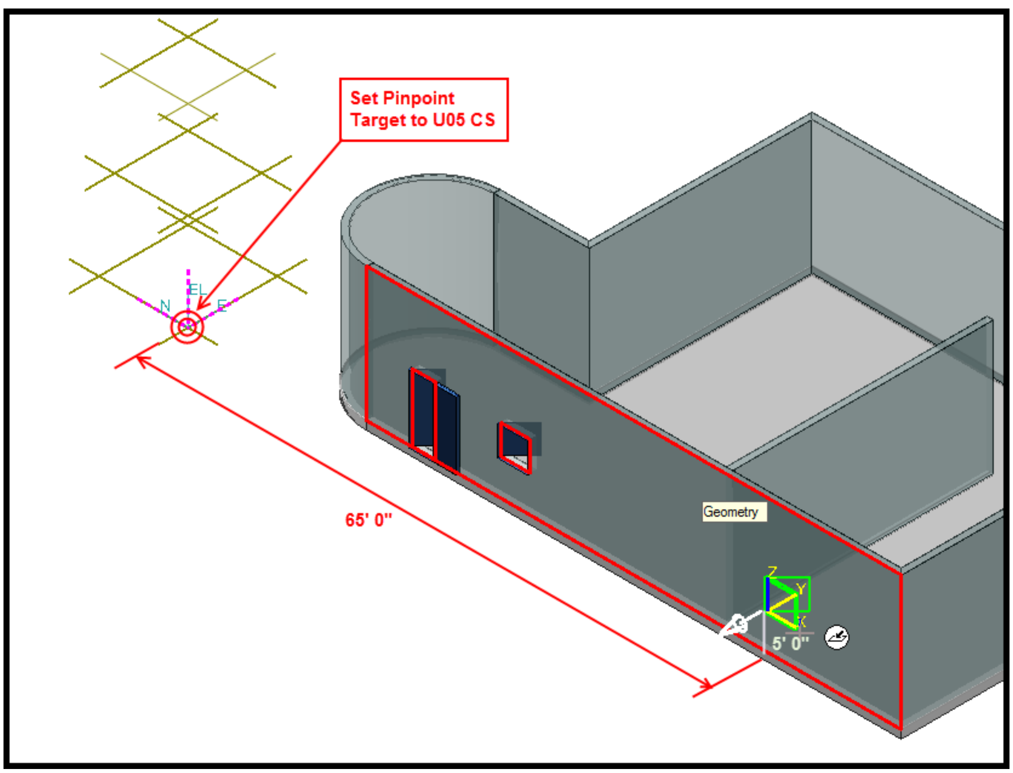
17. Use the Left Arrow key to rotate the equipment 90 deg about the z-axis.
18. Move the cursor over the wall object as shown below:

© Copyright 2014 Intergraph Corporation



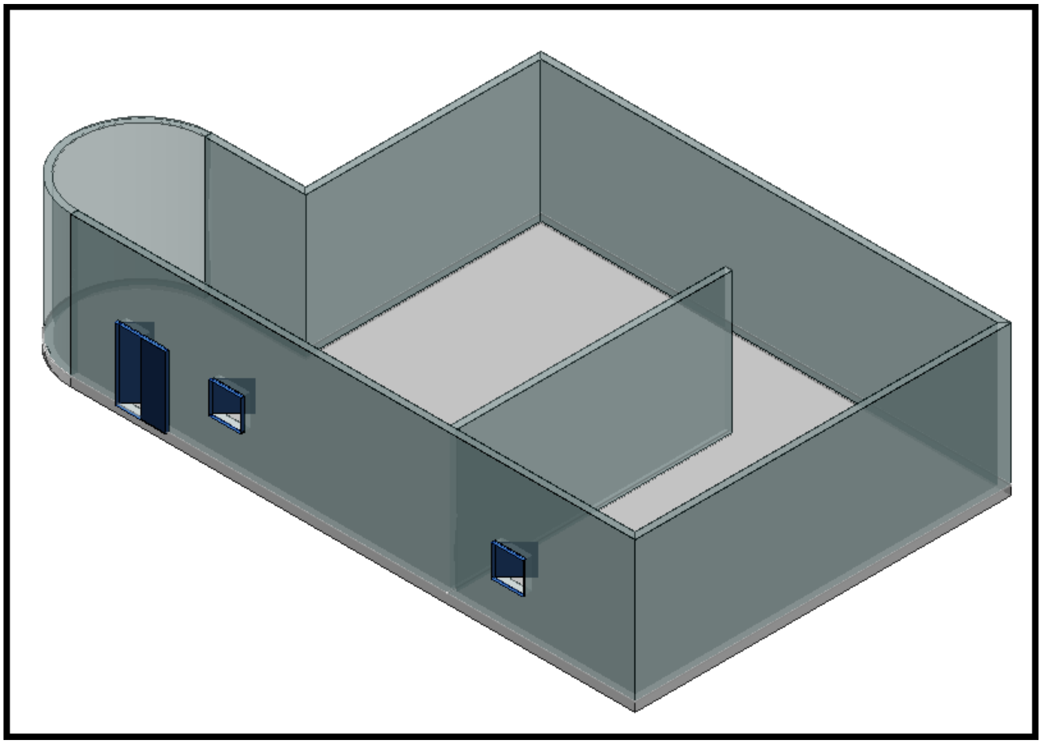
19. Left Mouse Click to place the door on the model.

20. Repeat the above workflow to place the second window as shown below. Name the second window as Window-102.



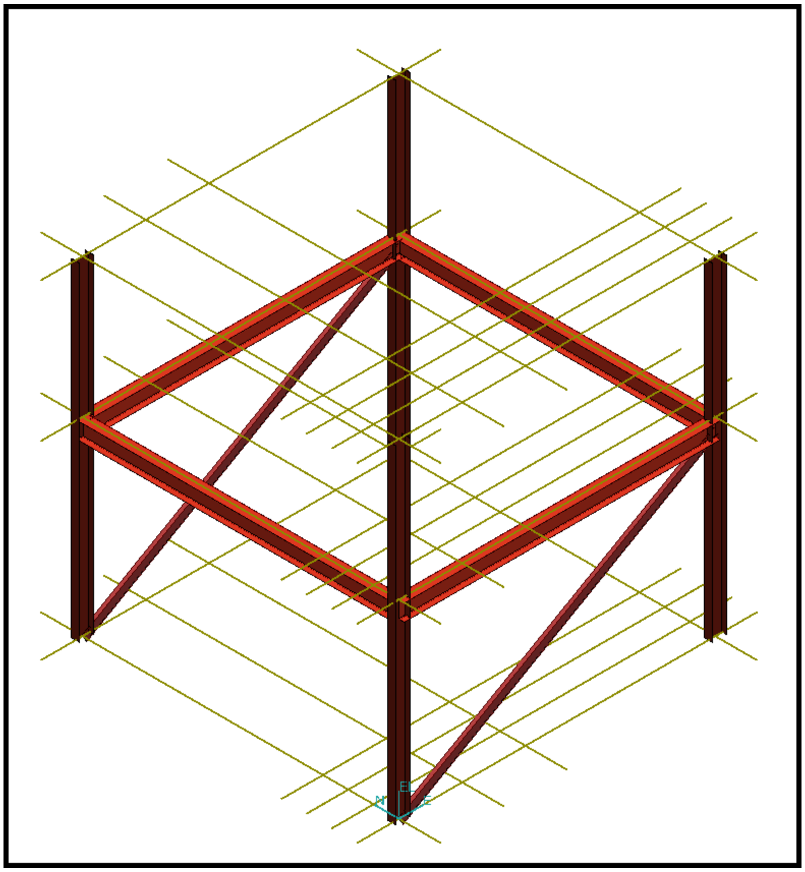
Your View should now resemble the following graphic:

© Copyright 2014 Intergraph Corporation



Part III - Place wall objects to Structure U04:

1. Re-define your workspace to include the A2 -> U04 and CS -> U04 CS systems. See figure below:

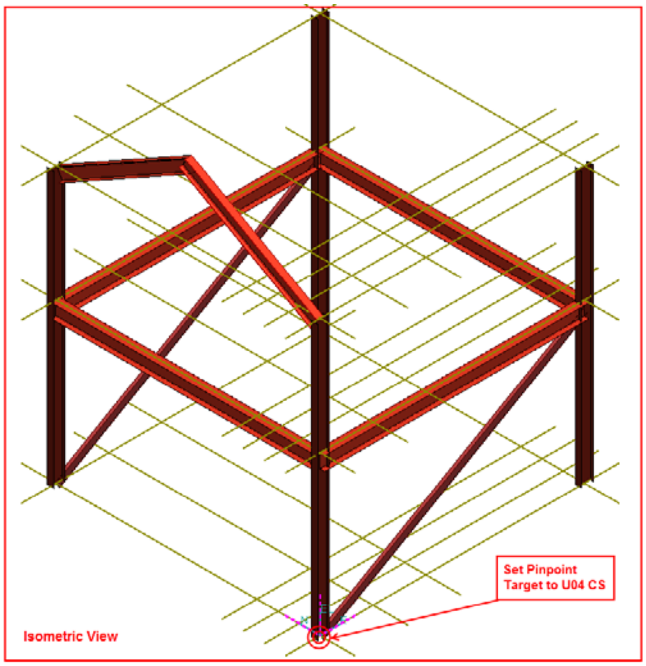


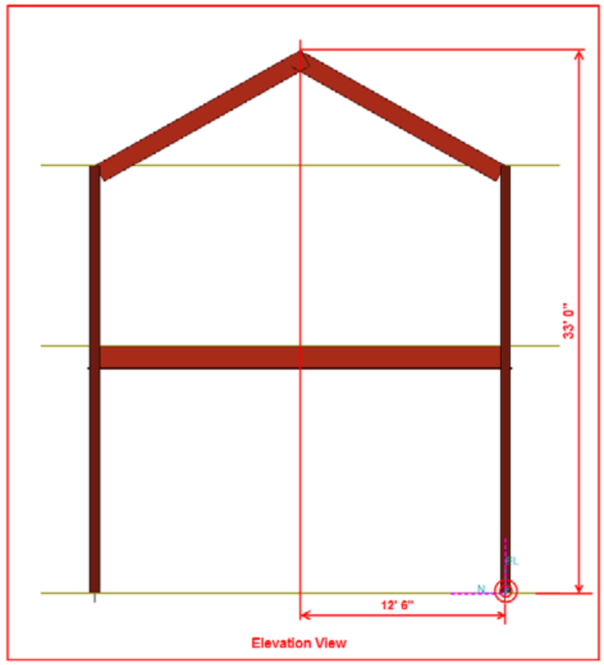
2. Select Place Linear Member System command. System displays the smart step ribbon bar.
3. Use the ribbon bar to set the active member parameters as follows:

Connection:	By Rule
System:	A4 -> U04 -> Structural -> Beams
Type Category:	Beam
Type:	Beam
Section Name:	W14x53
Cardinal point:	8
Angle:	0 deg
Material:	Steel- Carbon
Grade:	A36

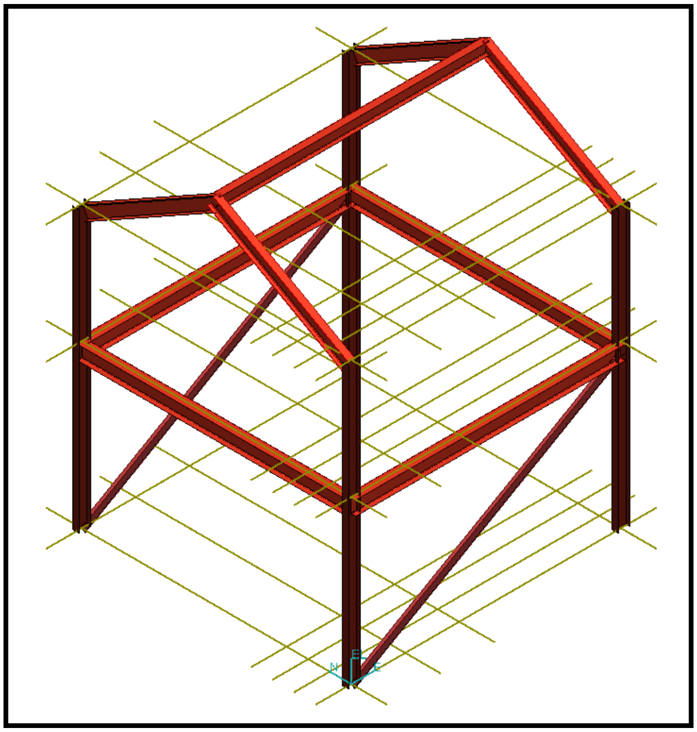
4. Place End 1 at the end of the column located at North Plane 25'-0" and Elevation 26'- 0" intersection.
5. Place End 2 at East 0'-0", North 12'- 6" and Elevation 33'- 0".
6. Continue to place the second beam (End 3) at the end of the column located at North Plane 0'- 0" and Elevation 26'- 0".

© Copyright 2014 Intergraph Corporation





7. Repeat the above workflow to finish building the roof as shown below:



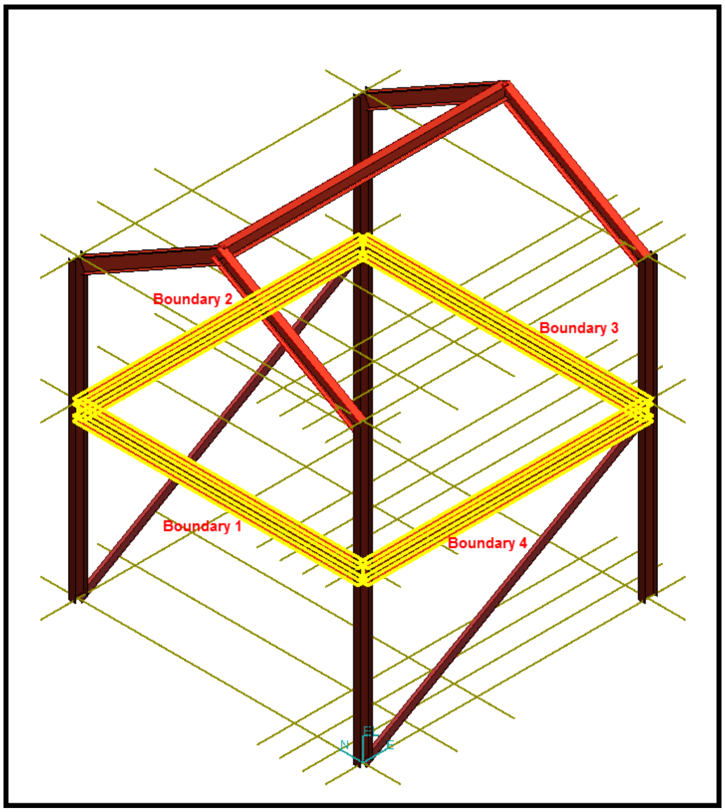
8. Select Place Slab command.
9. Select the Top surface of the beam located at Elevation 15'- 0" for the support plane.
10. Select "Accept" button.

© Copyright 2014 Intergraph Corporation

11. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U04 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4"_Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Thickness:	0'-4"
Boundary Offset Reference:	Outer port-face of a bounding member

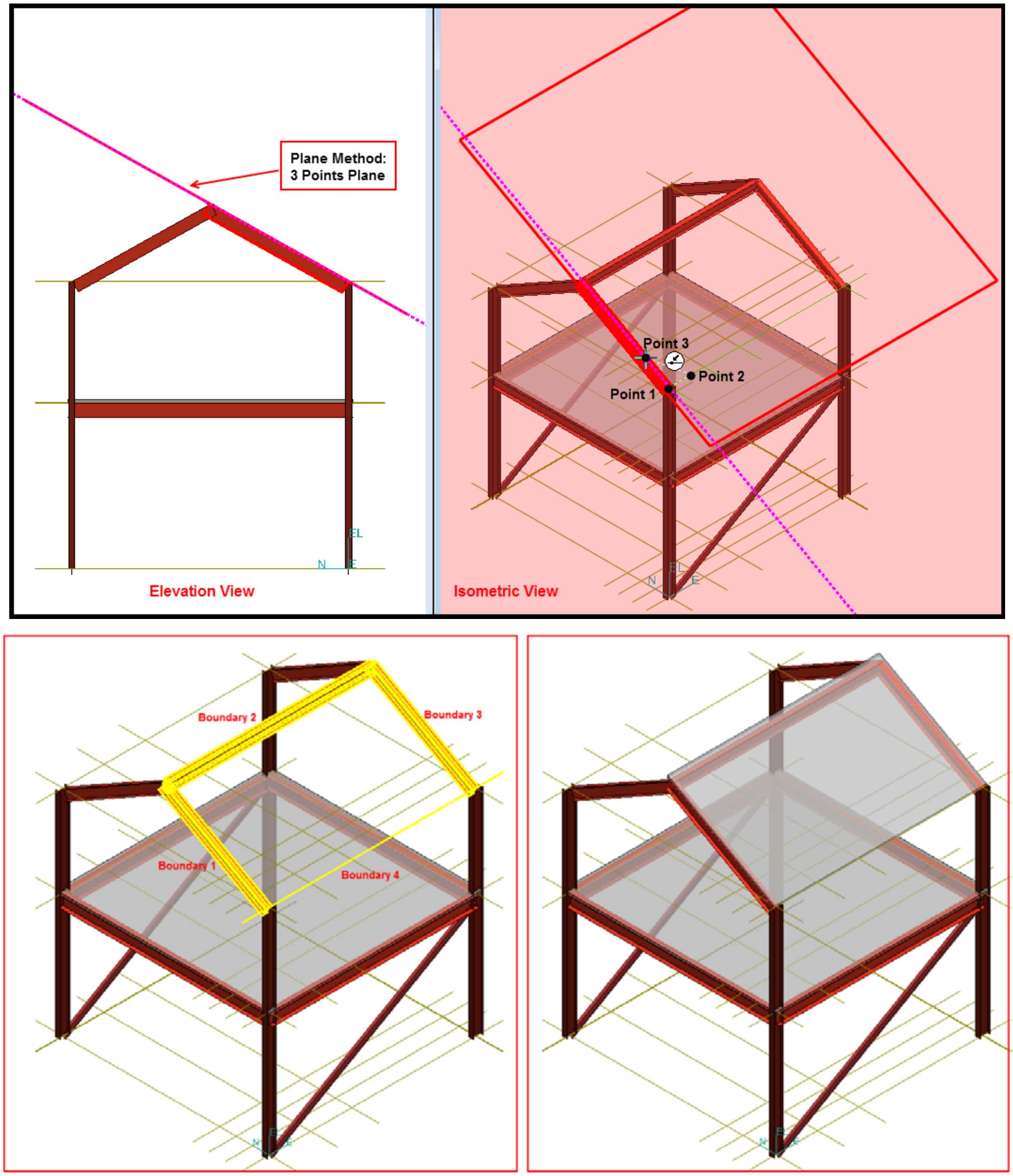
12. Select the boundaries as shown below:



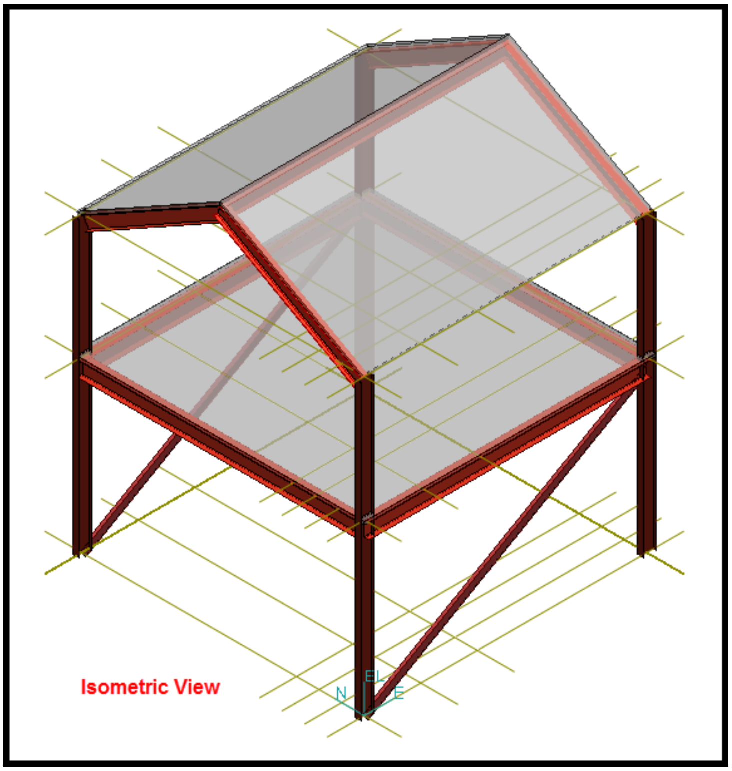
13. Select "Accept" button.

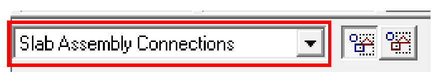
14. Hit "Finish" button to commit the transaction.

15. Repeat the Place Slab Command to place another slab as shown below:

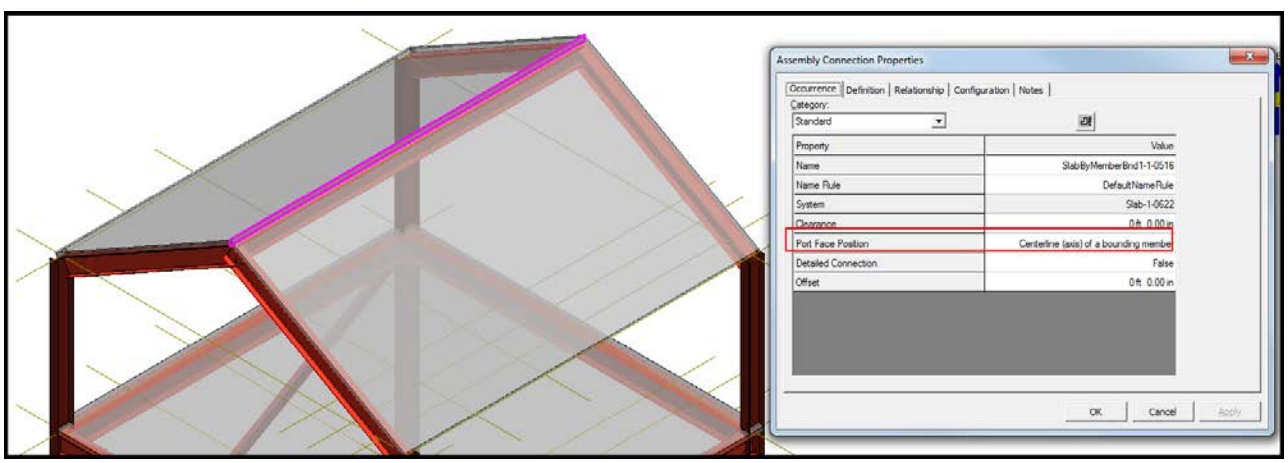


16. Repeat the Place Slab Command to place another slab to finish building the roof as shown below:

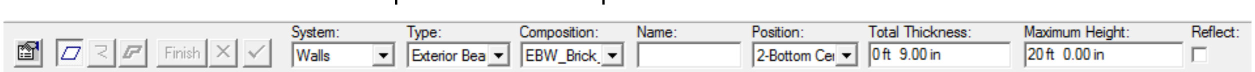


Tip: Set you locate filter to Slab Assembly Connections 

Select the slab assembly connection as shown below and go to its properties and set the Port face Position to 'Centerline (axis) of a bounding member.



17. Select Place Wall Command to open the SmartStep ribbon bar.



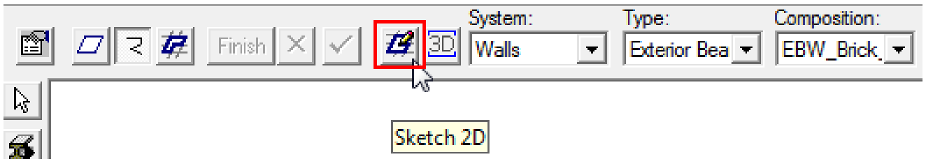
18. Set the wall parameters as follows:

System: A2 -> U05 -> Structural -> Walls

Wall Type: Exterior Bearing Wall - Composite
Composition: EBW_Brick_5"_Air_4"_Batt&Roll_2"_Siding_Vinyl_0.046"
Face Position: 2 - Bottom_Center
Total Thickness: 0'-9"
Maximum Height: 20'-0"

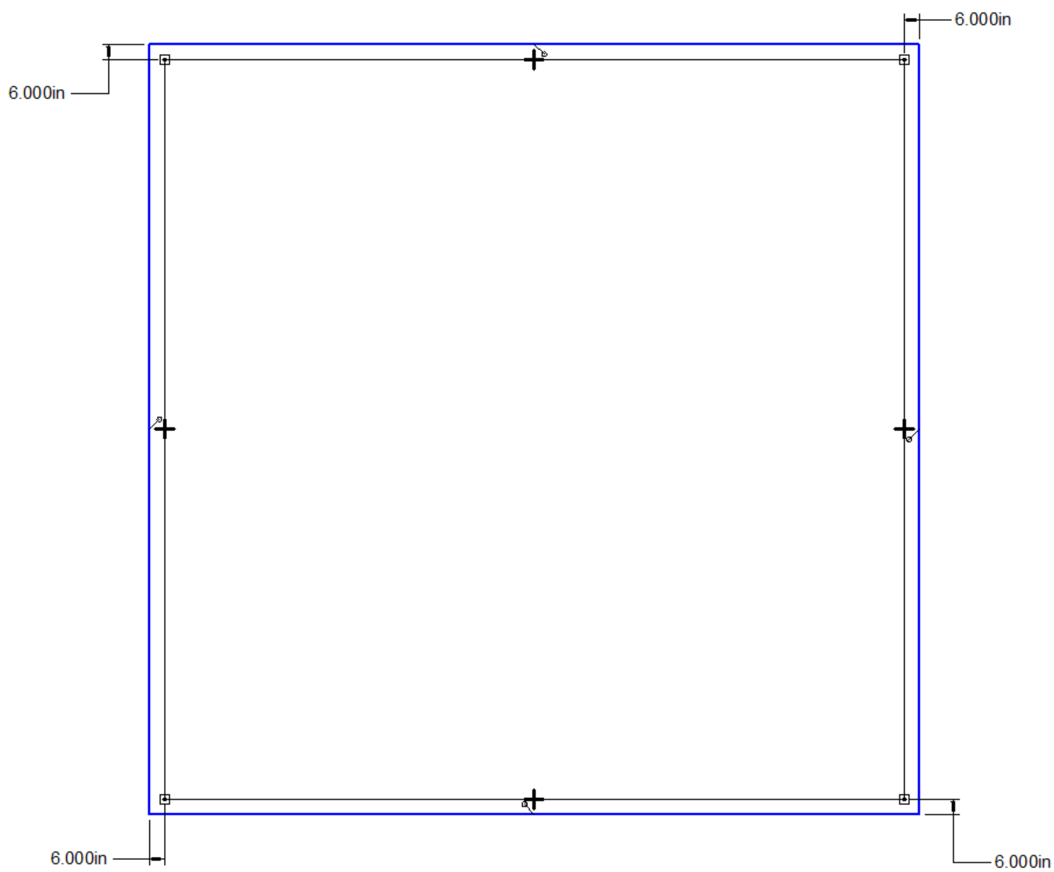
19. Select the top surface of the slab (EL 15'-0" for the sketching plane).

20. Select the Sketch 2D step.



21. Select "Draw" button to open the 2D Editor.

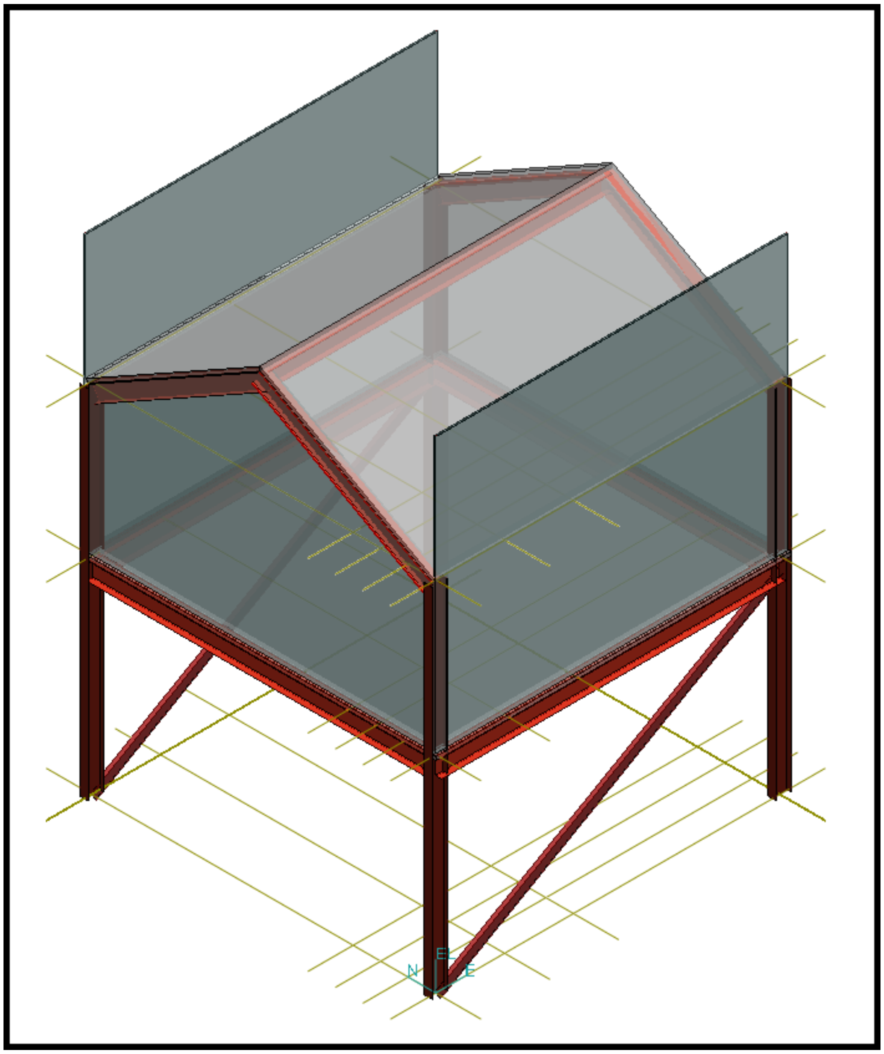
Hint: Use place line and dimension commands and sketch the boundaries as follows: OR fence select the blue lines and use offset command and key in 6in towards the inner side.

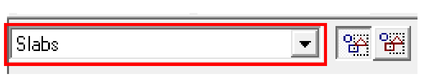


22. Hit "Close" button to return to the SmartStep ribbon bar.

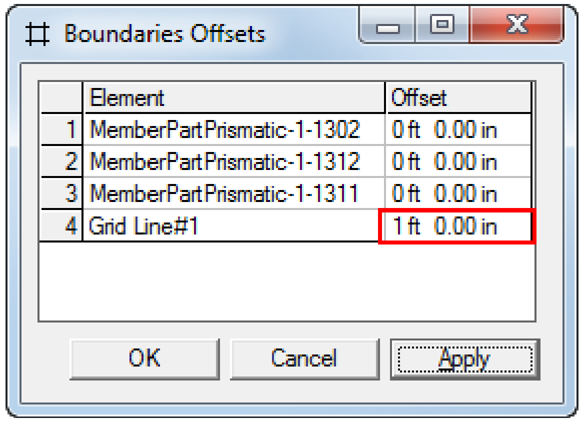
23. Select Finish button.
24. Select the boundary step and pick the two slabs located on the roof.
25. Hit “Finish” button.

Your View should now resemble the following graphic:

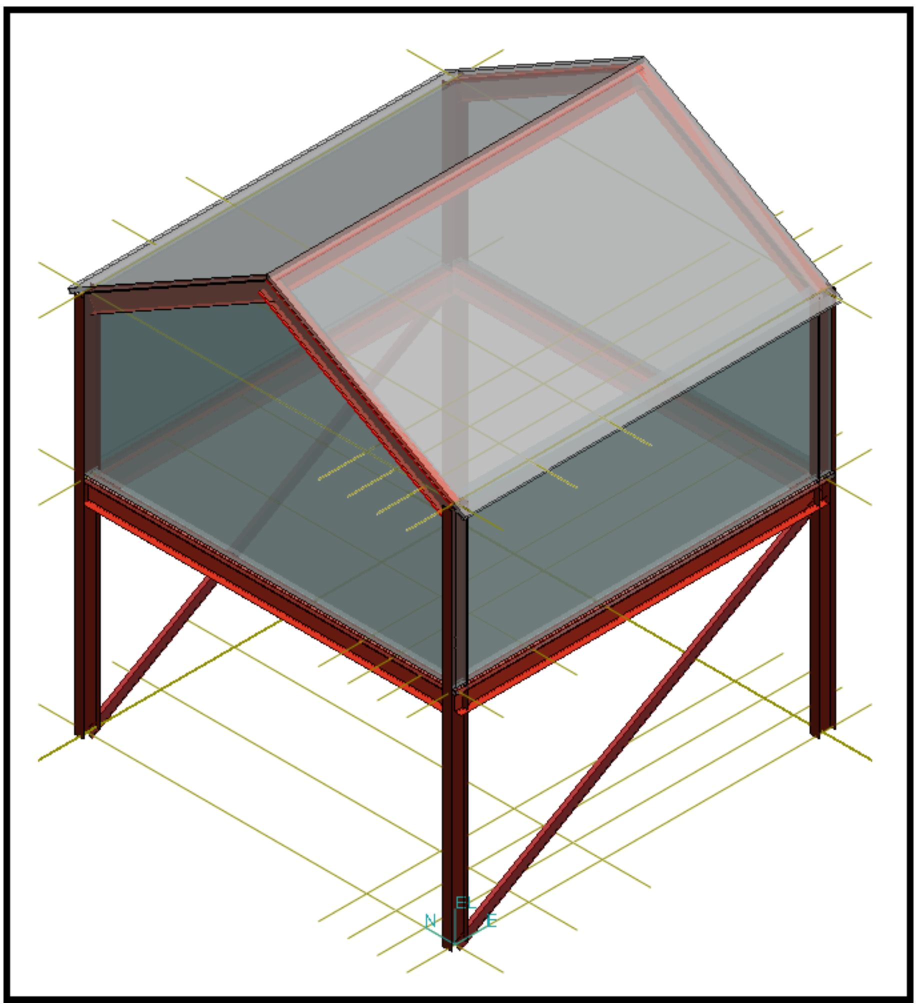


26. Set your locate filter to Slabs. 

27. Select one of the slabs on the roof and select ‘Define Boundaries Offsets’ on the slab ribbon bar.
28. Key in an offset value of 1 ft 0.00 in for the Grid Line boundary of the slab.



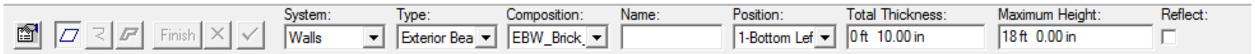
29. Repeat steps 27 and 28 for the other slab on the roof.
30. Your View should now resemble the following graphic



Part IV: Split Walls 3-D sketch

© Copyright 2014 Intergraph Corporation

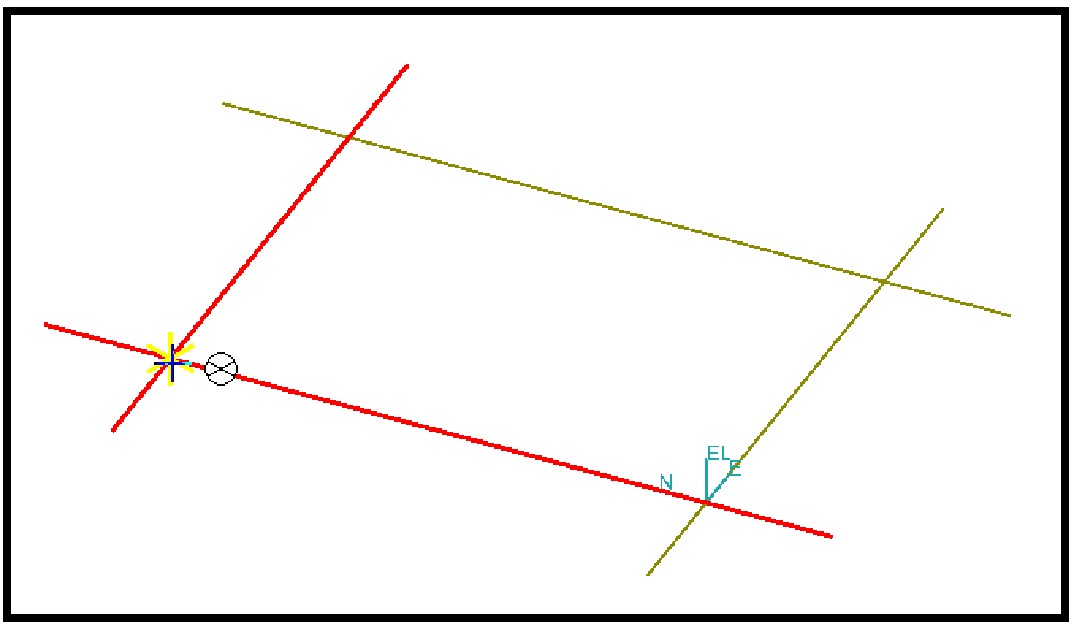
1. Re-define your workspace to include the A2 → U06 and CS → U06 CS system.
2. Select Place Wall Command to open the SmartStep ribbon bar.



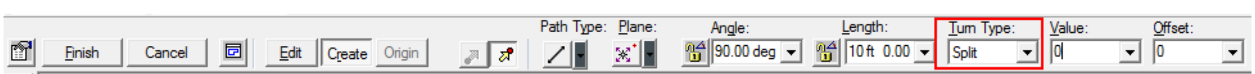
3. Set the wall parameters as follows:

   System: A2 -> U06 -> Structural -> Walls
   Wall Type: Exterior Bearing Wall - Composite
   Composition: EBW_Brick_5"_Air_4"_Bath&Roll_2"_Siding_Vinyl_0.046"
   Face Position: 1 - Bottom_Left
   Total Thickness: 0'-10"
   Maximum Height: 18'-0"

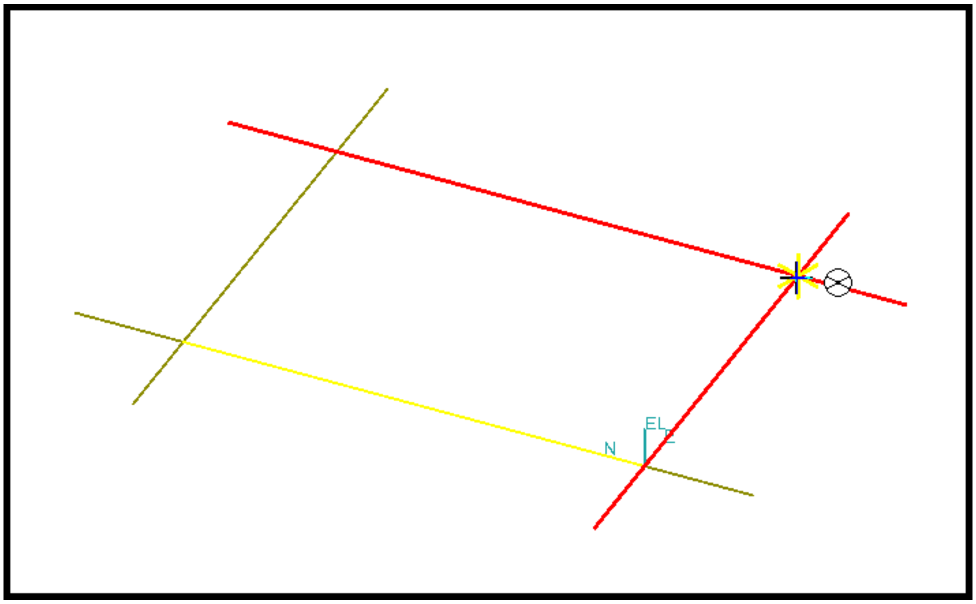
4. Select the Elevation plane, El 0ft as the sketching plane.
5. Select the Sketch 3D step.
6. Select the intersection of the grids as shown below.



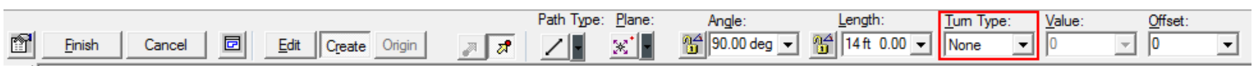
7. Select the next intersection along the North grid line and perform a left mouse click.
8. Go to the ribbon bar and change the Turn Type to "Split".

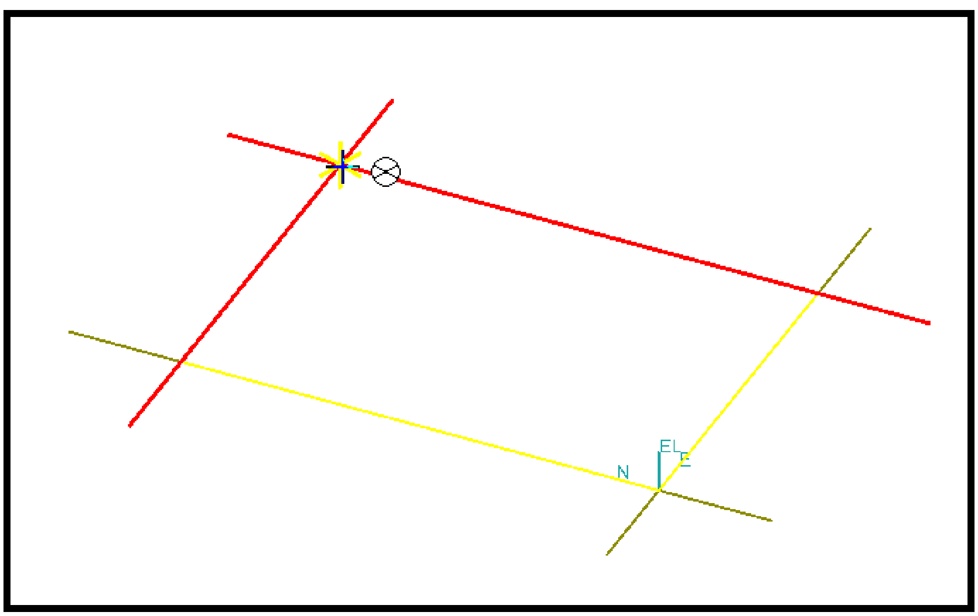


9. Now, select the intersection as shown below.

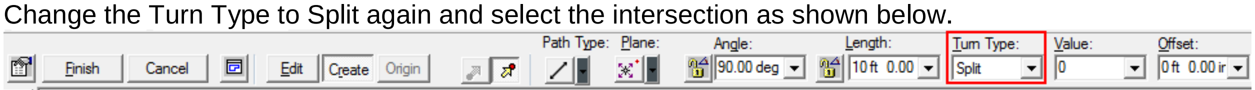


10. Change the Turn Type to “None” and then select the next intersection along the North and as shown below.

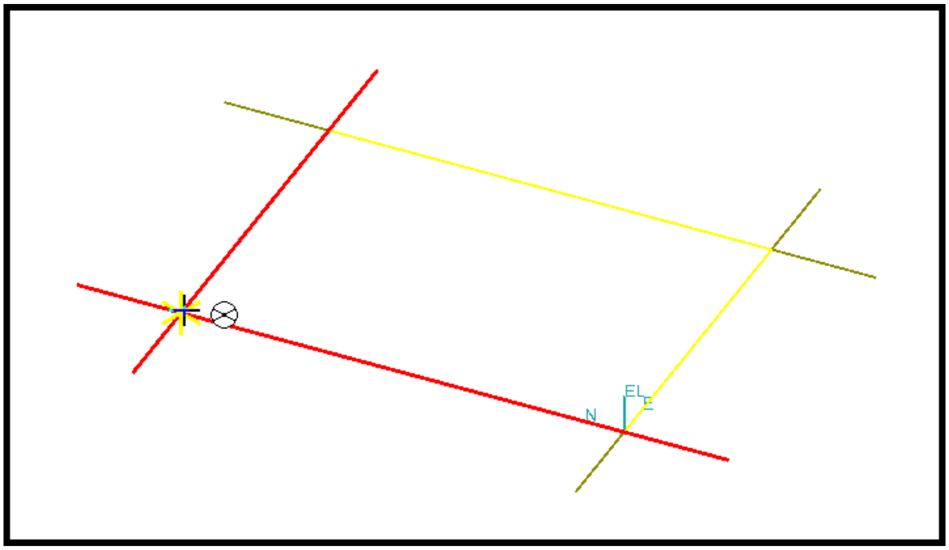




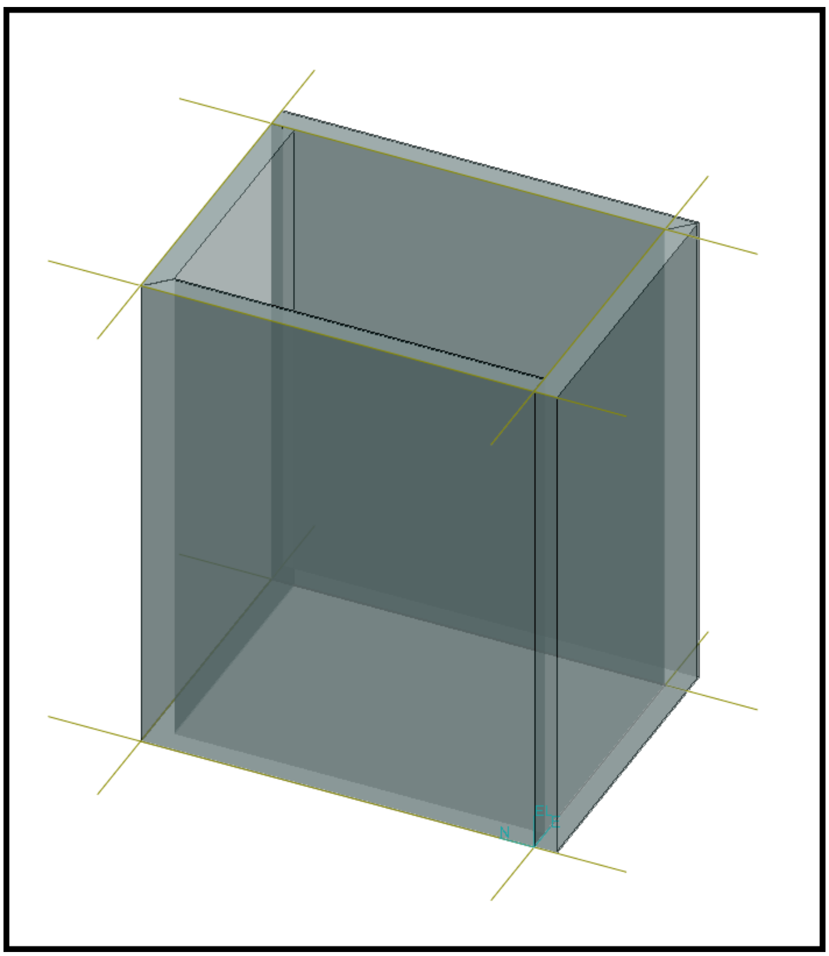
11. Change the Turn Type to Split again and select the intersection as shown below.



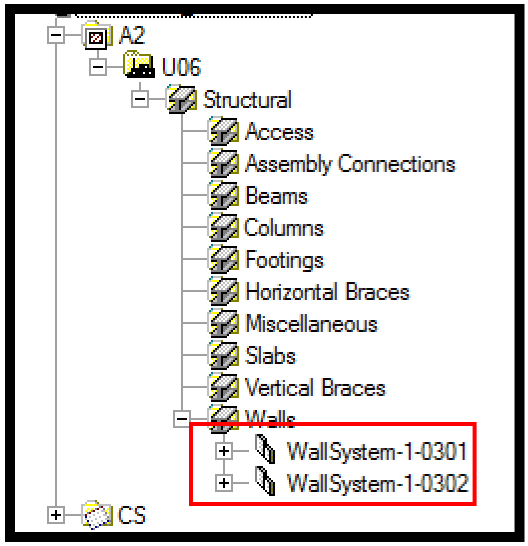
© Copyright 2014 Intergraph Corporation



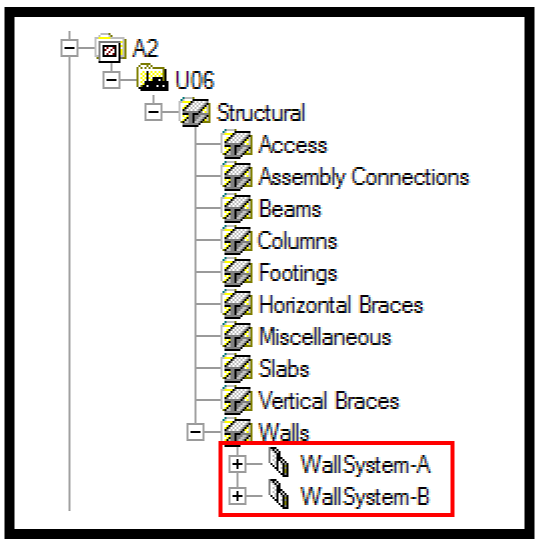
12. Hit Finish twice. Your view should resemble the following graphic.



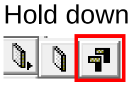
13. Two wall systems are created as shown in Workspace explorer window.



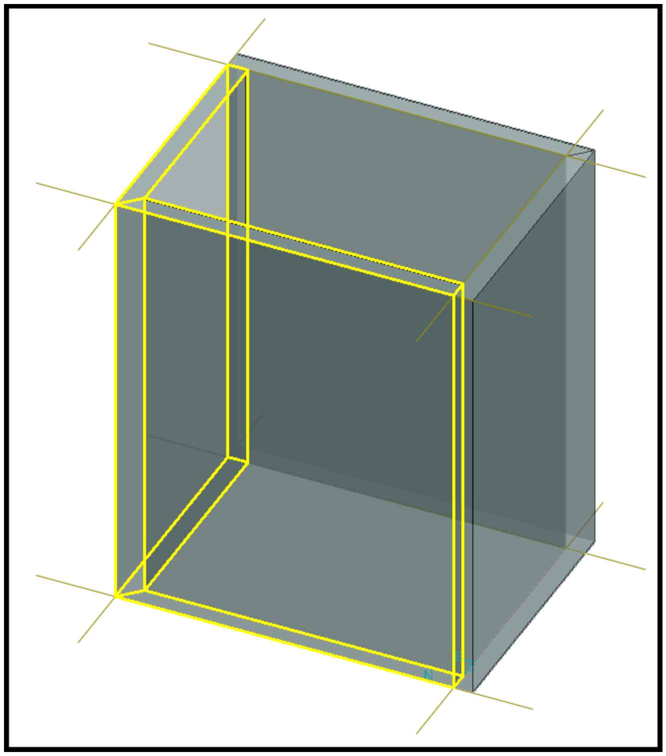
14. Rename the WallSystems as WallSystem-A and WallSystem-B. Your workspace explorer should view as shown below.



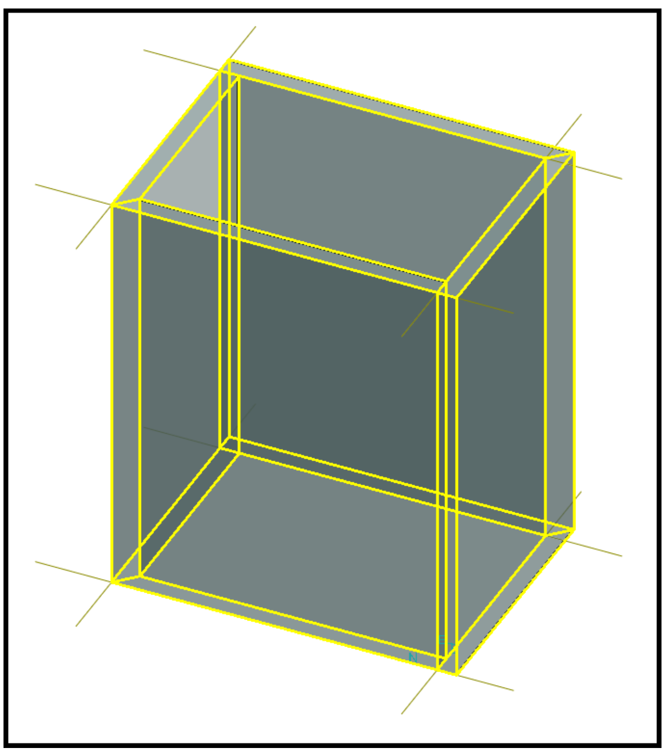
<u>Part V: Toggle Wall Corner</u>

15. Hold down the Place Wall command in the vertical toolbar until the Toggle Corner Wall Operators  dialog appears. Select the Toggle Corner Wall Operators command.
16. Select the WallSystem-A as shown below.

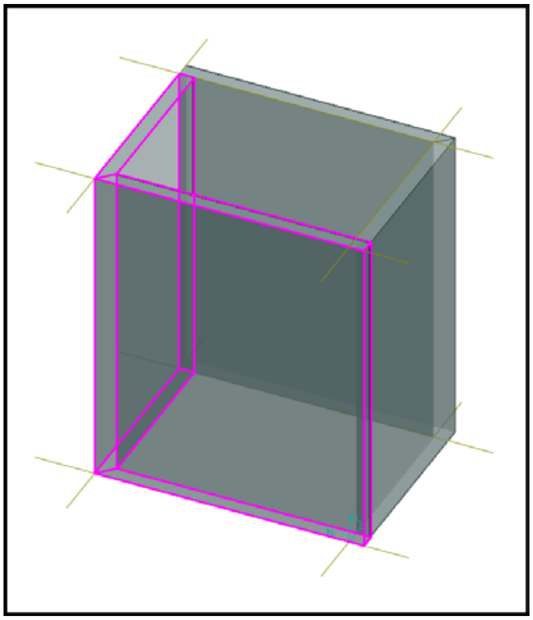
© Copyright 2014 Intergraph Corporation



17. Select the WallSystem-B as shown below.



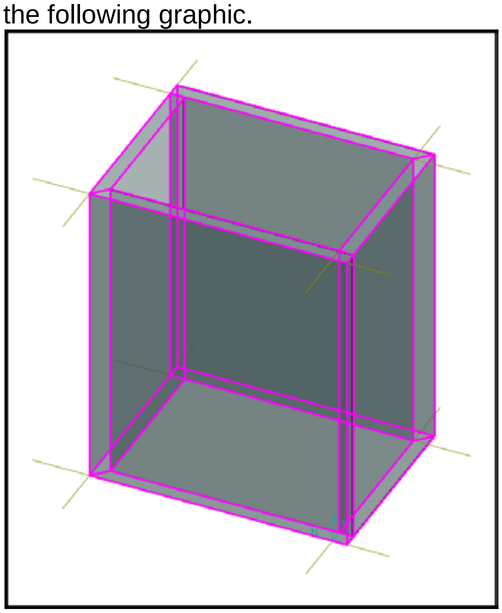
18. Hit Finish.
19. Select WallSystem-A as shown below and observe the changes.



20. When two separate walls join at a corner, one wall extends past the other wall to form the corner. This command is used to toggle which wall extends past the other.

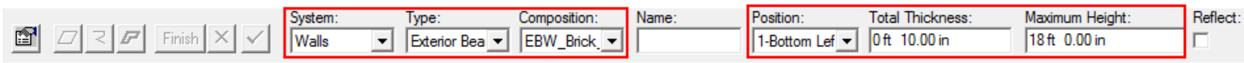
Part VI: Multiple Wall modification

21. Multiple walls can be selected for modification. Select WallSystem-A and WallSystem-B.
22. Your view should resemble the following graphic.

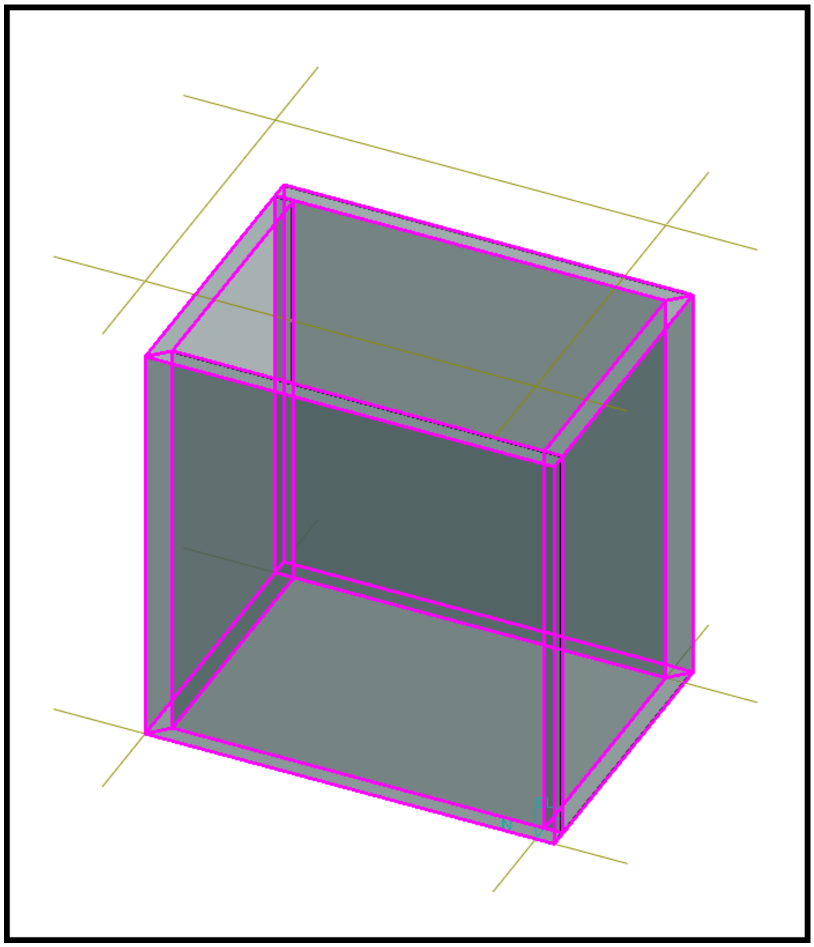


23. System, Type, Composition, Position, Total Thickness and Maximum height values can be changed in the ribbon bar.

© Copyright 2014 Intergraph Corporation



24. Change the Total Thickness to 0ft 8in and maximum height to 15ft. Hit Enter. Your view should resemble the following graphic.



LAB-12: Openings

Objectives

After completing this lab, you will be able to:

- Understand the opening entities and relationships
- Place Openings using different options

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

Overview:

You can place an opening in an existing slab, wall, or member. You can define the shape of an opening by selecting the boundaries, sketching the opening outline, or using a predefined shape from the Catalog. In addition, you can control the depth of an opening to create a fully penetrating hole or a recessed opening.

You can place an opening in an existing slab, wall, or member by using the following methods:

- **Placing an Opening by Boundaries**: You select the geometry of design objects to define the outline of an opening. You can define offsets for the actual opening from the boundaries. When the design objects you referenced are moved or modified, the opening will be automatically modified. For example, you can define an opening for a pipe that penetrates a grating. When the pipe moves or changes diameter, the opening will also move or change diameter.

- **Placing an Opening by Sketch**: You sketch the outline of an opening in the 2D environment. The sketch can consist of a closed set of lines, curves, arcs, fillets, and other drawing options and can reference other geometrical objects in the model.

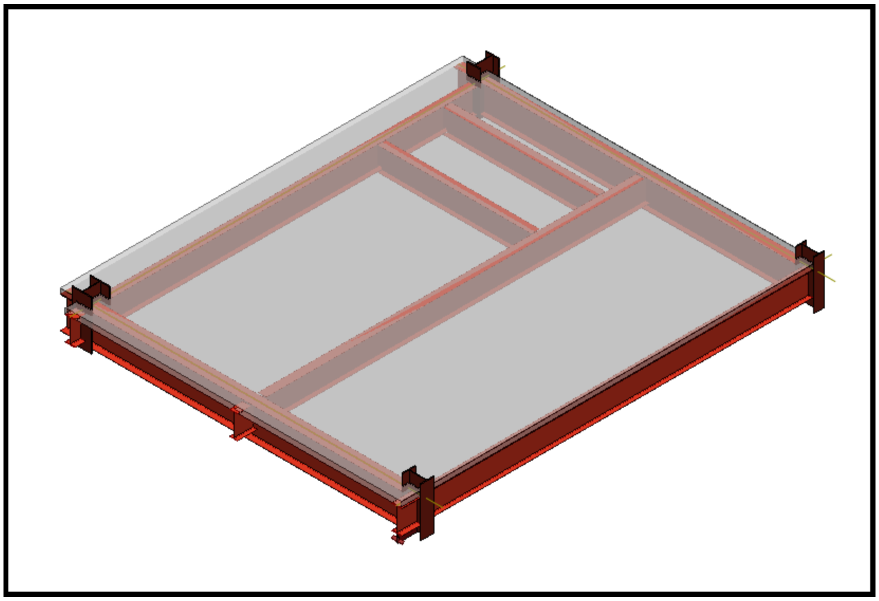
Placing an Opening by Shape: You select the shape of an opening from the Catalog. The default Catalog is delivered with several standard shapes. The Catalog administrator can add custom shapes with the required default sizes to your Catalog. After placing an opening, you can resize or edit the shapes by using the sketch environment editing commands.

<u>Part I -Place a Stairway Opening</u>

1 Re-define your workspace to include the A2 → U03 and CS → U03 CS systems.

2 Use Clip by Object command to isolate the beams and the columns for the stairway opening as shown below:

© Copyright 2014 Intergraph Corporation

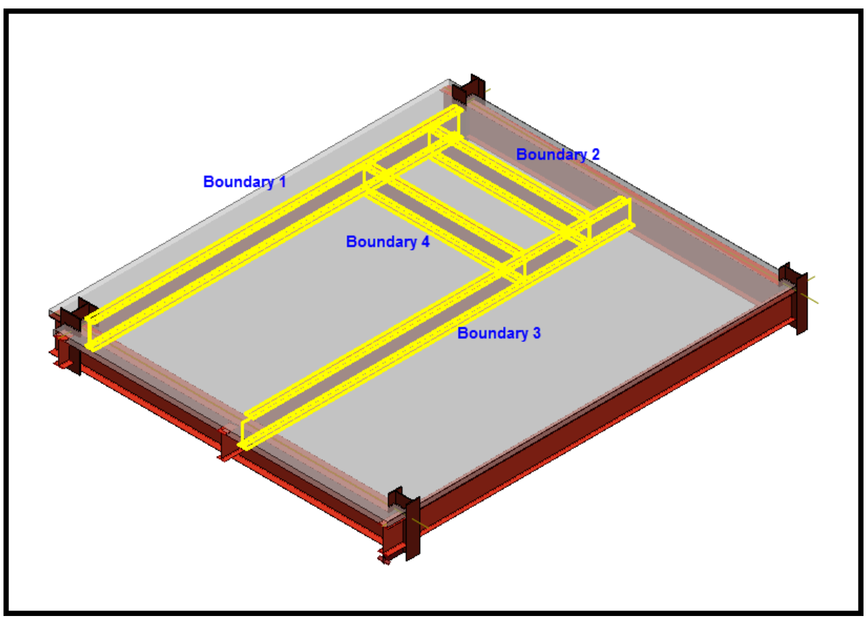


3 Select Place Opening Command.

4 Select the slab located in the first floor frame shown in clipped view above.

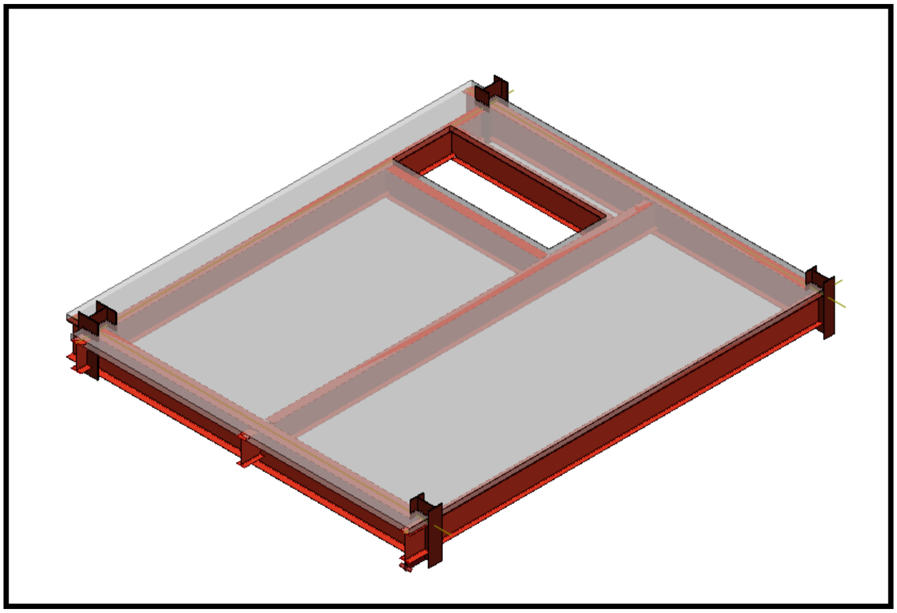
5 Keep the default parameter in the smartstep ribbon bar.

6 Select the members as shown in Figure below to define the boundaries of the opening. Use quick pick service to pick the members.



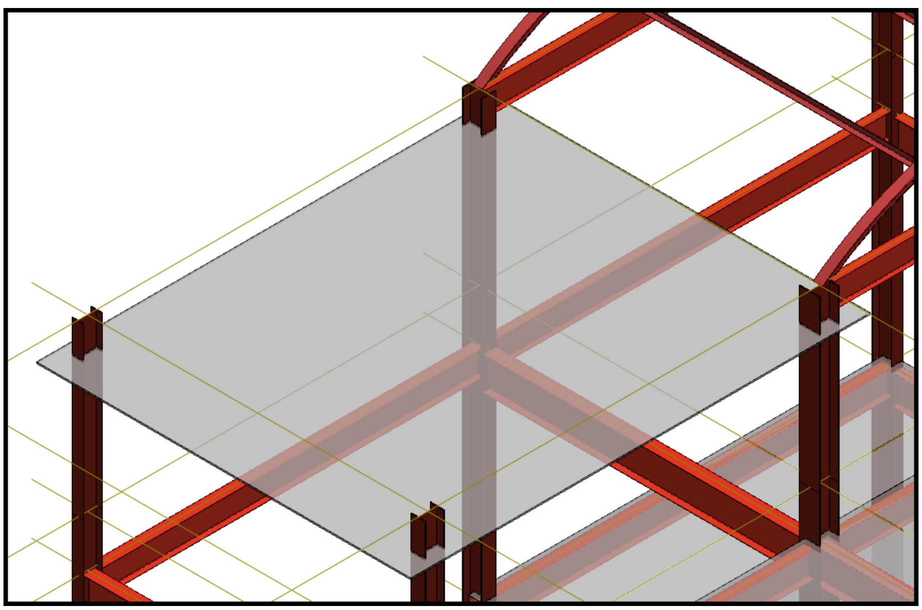
7 After selecting the members. Hit the “Finish” button.

Your View should now resemble the following graphic:



Part II -Place Opening on the Roof Deck

1. Select View-> Clear Clipping to remove the clipping volume.
2. Use Zoom Tool to window area the roof deck as shown below:



3. Select Place Opening Command.
4. Select the roof deck located at Elevation 42’- 0” for the Smartstep1.
5. The **Place Opening** ribbon appears. Click the **Draw** option to define the boundaries of the opening, as shown in Figure below.

© Copyright 2014 Intergraph Corporation

Note:

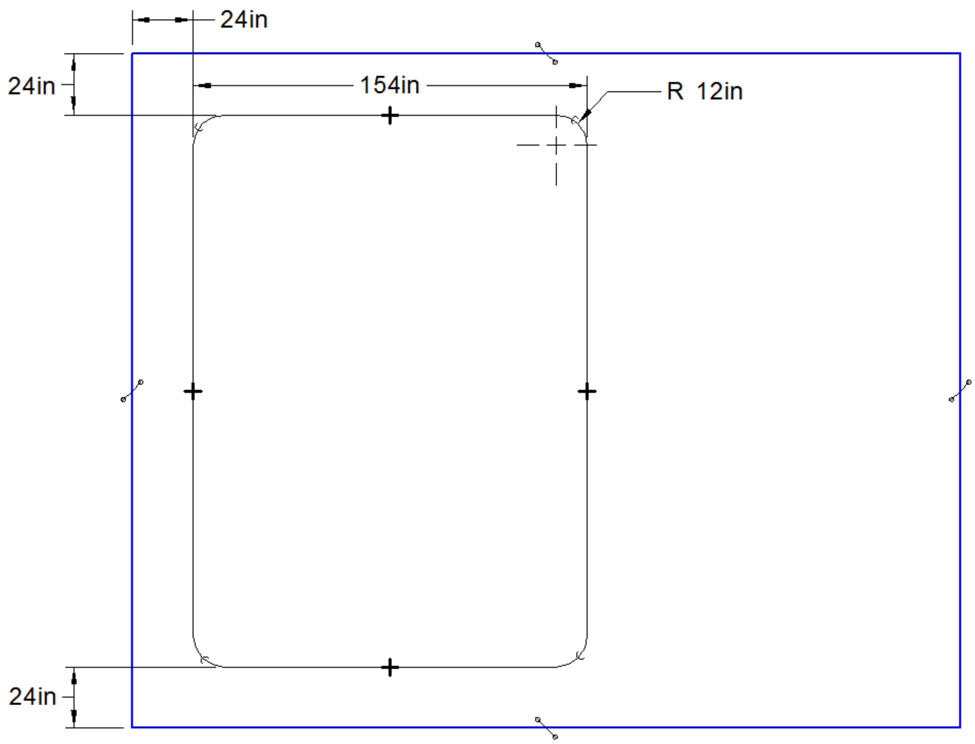
*The orientation you will see in the Sketch 2D editor is the default. If you want to define a different orientation, select the **Define Orientation** smartstep before you enter the Sketch 2D editor. This smartstep allows you to specify the orientation of the Sketch 2D view. You select an edge of the object you are cutting and a vertex of the edge to define the view x-axis in the Sketch 2D graphic view.*

6. The software automatically opens the **Draft 2D View** window. Draw the opening as shown in Figure 11, by using the drawing commands available in the 2D environment. Use the **Place Line**, **Fillet**, **Distance Equal Relationship**, and **Dimension** commands.

Notes:

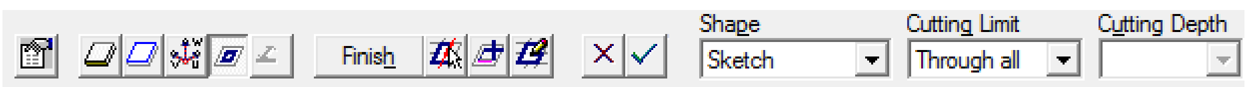
- *The **Sketch 2D** editor provides many commands. It is a 2D drawing application that has specific training in the Drawings sessions.*
- *The **Sketch 2D** editor will display the boundaries of the slab (or other object) in which you are placing the opening. If you want to sketch the opening relative to the other geometry that is in the 3D model, then click the **Add References to Sketch 2D** option on the ribbon. You can then select the geometry that will be projected to the sketch plane for reference as you draw the opening in the Sketch 2D editor. When you create relationships to this reference geometry, the opening is updated when the parent geometry changes. This allows you to define an opening, for example, which is positioned relative to a pipe going through the platform. When the pipe is moved, the opening will also move. You can also position the opening by defining the dimensions from the edge of the slab. If the slab edges are modified, the opening will also be modified, maintaining the distance that you defined from the edge of the slab.*
- *If you do not create relationships from the sketch geometry to the model geometry to control the position of the opening, then the opening will stay at the position in space where you defined it when you edit the boundaries of the slab in which the opening is defined. However, if you create the opening relative to the model geometry and move the object in which the opening is placed, the opening is automatically moved. The Sketch 2D editor has graphic indicators of the relationships and dimensions that you create. You can select these indicators and delete them to remove the relationships. You can also choose an option to reset the relationships by using the **Maintain Relationships** button on the **Dimension**/**Relationship** toolbar.*

© Copyright 2014 Intergraph Corporation



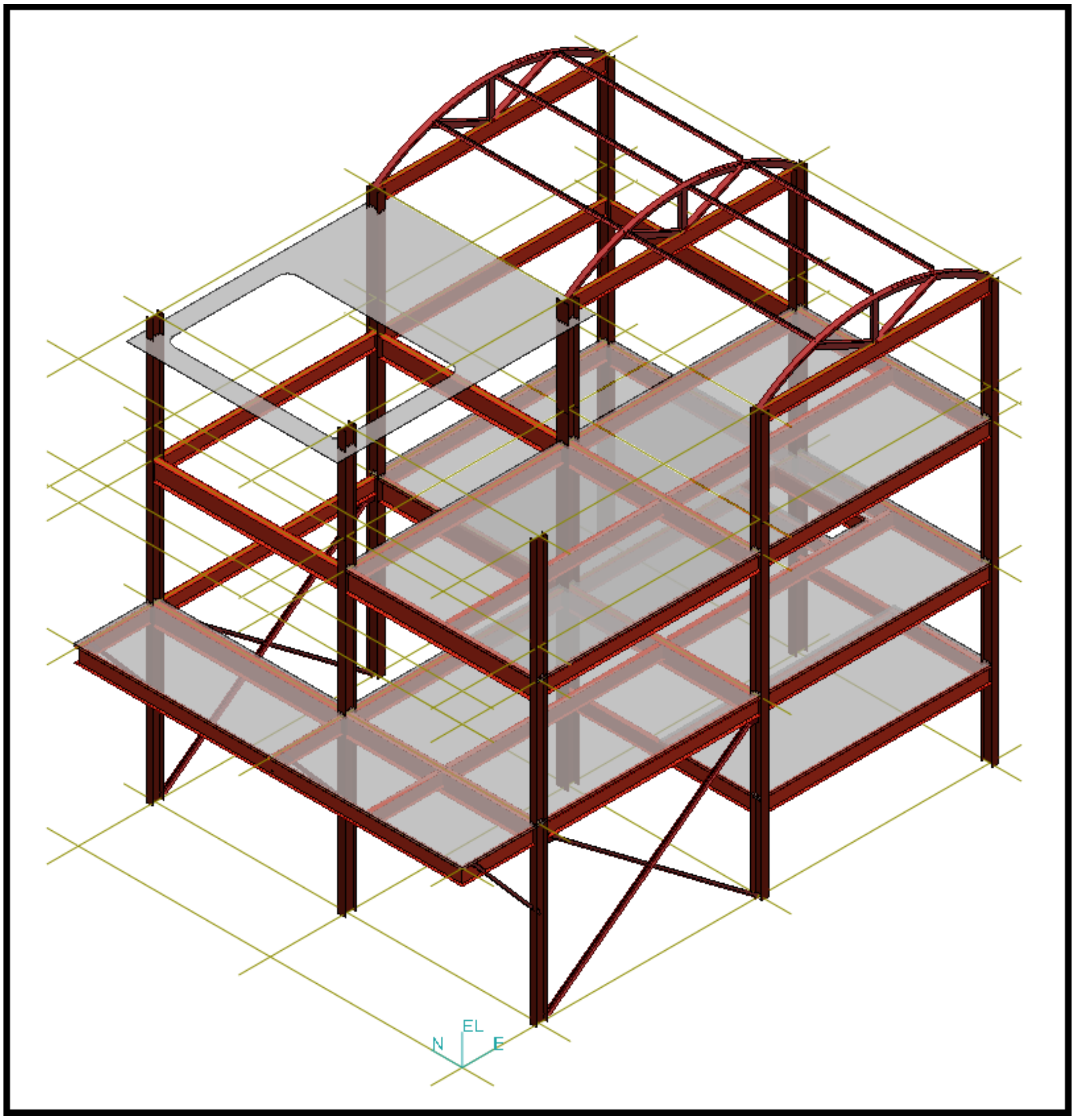
Hint: Use place line, fillet, distance equal relationship and dimension commands

5. Hit “Close” button to return to the SmartStep ribbon bar.
6. Keep the default parameters.



7. Hit “Finish” button.

Your View should now resemble the following graphic:


EL
N
E

LAB-13: Stairs / Ladders

Objectives

After completing this lab, you will be able to:

- Understand the stair and ladder entities and relationships
- Place stairs and ladders using different methods

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

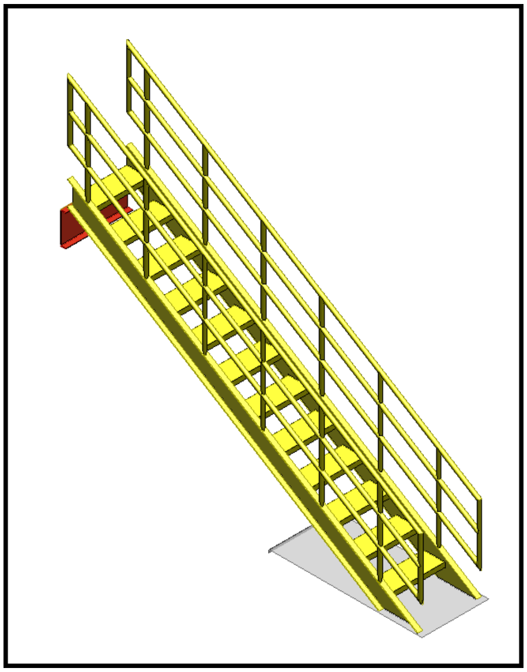
Overview (Stair/Ladder):

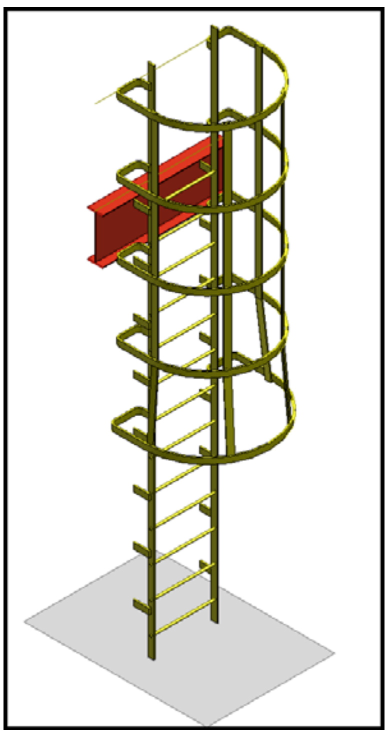
The stair and ladder design objects generate geometry sufficiently detailed for general arrangement drawings and for participation in interference detection, as shown in Figures 1 and 2. The 3D model does not contain all the details needed for fabrication of the stairs and ladders. The design objects do not create individual parts with fabrication identities and properties. This is similar to the function of the handrail and equipment design objects in the 3D model. The properties of the stairs and ladders can be used along with standards as specifications for the detailed design.

Different types of stairs and ladders are defined by the Catalog with associated graphic symbols. The default Catalog has several types of stairs and ladders with many different options. You will need to place the different types and experiment with the options to understand all the configurations that are available to you. Your company can develop custom stairs and ladders to serve your specific requirements. All stairs and ladders, however, use the same set of graphic inputs for the positioning and construction of the geometry.

The top of a stair/ladder is related to an edge of an object and the bottom of the stair is related to a plane to establish the height. The position of the stair/ladder along the top edge is defined by an offset from a reference edge. The offset value can be either keyed in or established by graphically indicating the point. The graphically defined height combined with the properties of the construction determines the number and position of the steps.

© Copyright 2014 Intergraph Corporation

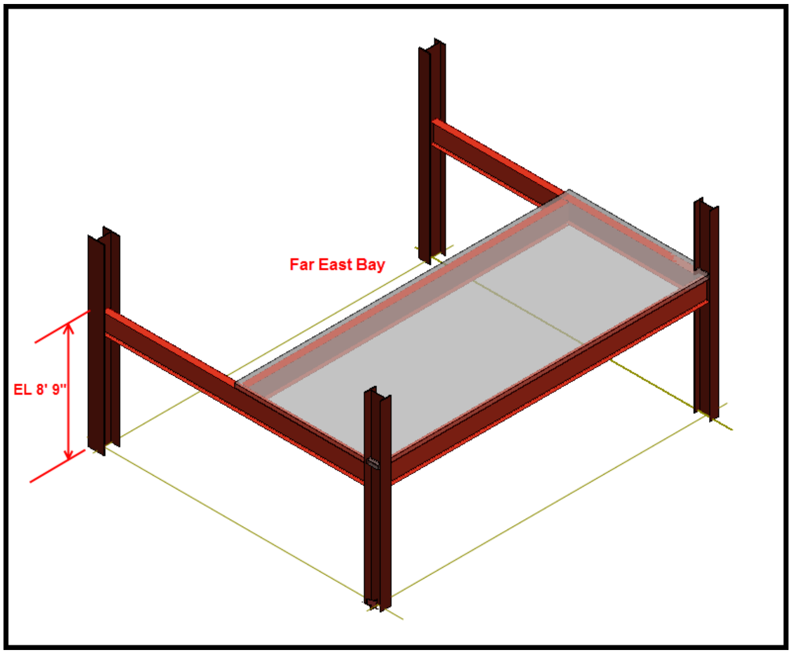




As the stair/ladder is connected to the top edge and the bottom plane, the interference detection service will not detect interference between any part of the object, such as the stringer, handrail, and tread of the stair with the edge or bottom surface.

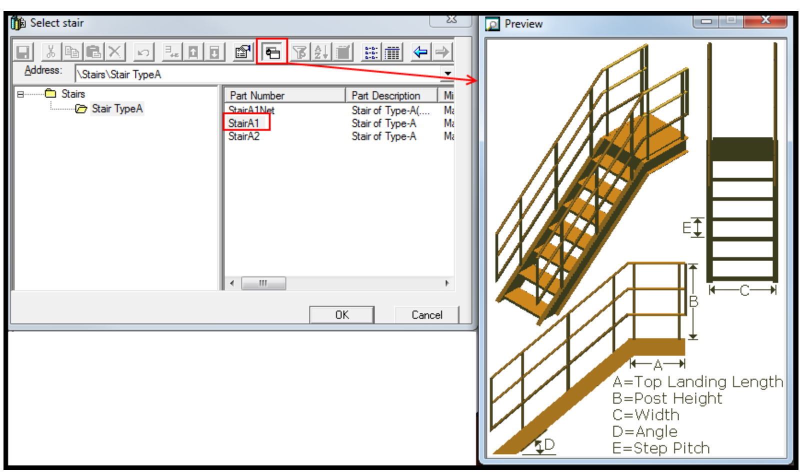
Part I –Place First Stair

1. Use Clip by Object command to isolate the beams and the columns for the Far-East Bay as shown below:

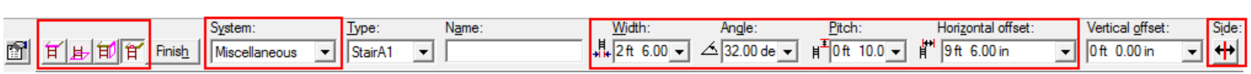


2. Select Place Stair Command on the vertical toolbar.

3. The **Select stair** dialog box appears. To select the type of stair in the **Select stair** dialog box, expand the folder **Stairs\Stair TypeA** until you see the part number **StairA1**. Select the part number and click **OK**. The selected part number becomes the default selection for the next time you place a stair. You can change the default option by using the **Type** option on the **Place Stair** ribbon. Your company can create standard sizes in your Catalog so you just have to select the desired standard and do not have to enter the data.

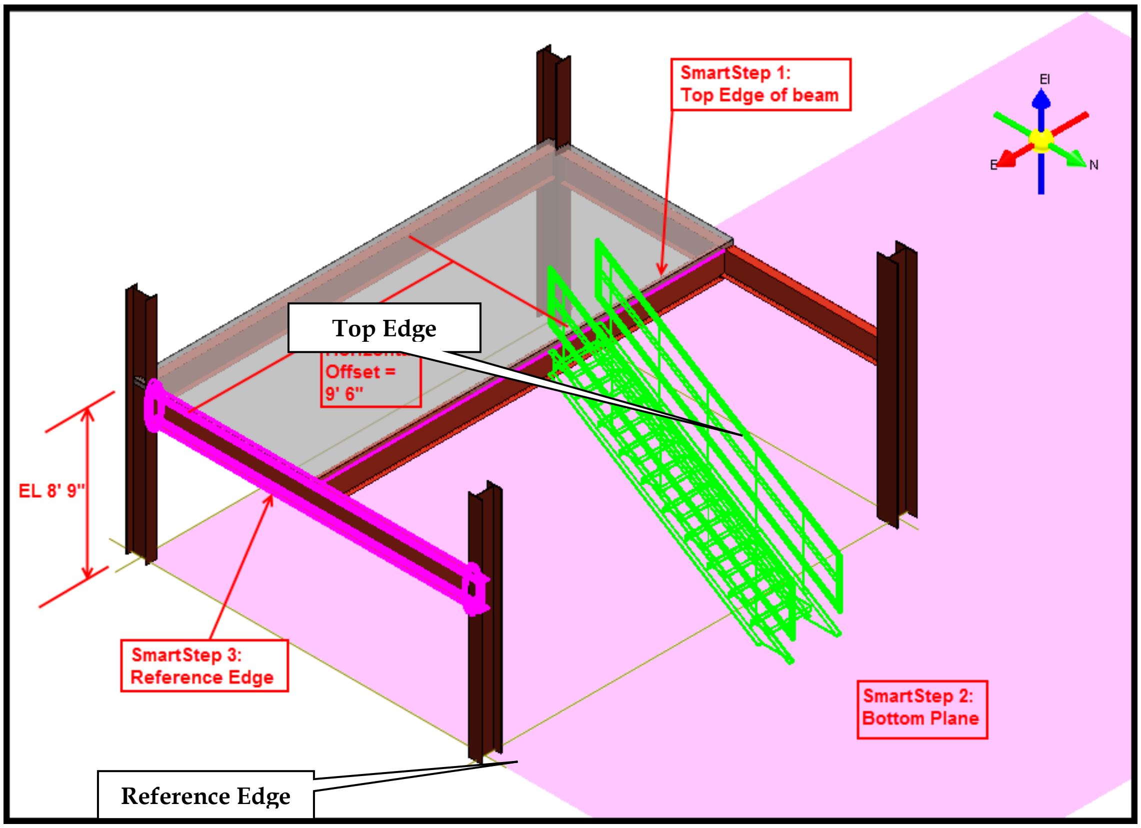


*Note: As you select the top edge, bottom plane, and reference edge in the model, the appropriate primary parameter options are displayed on the **Place Stair** ribbon.*



- You can specify the following settings on the **Place Stair** ribbon:
  - **System**: System to which a stair belongs
  - **Width**: Width of a stair
  - **Angle**: Slope of a stair in degrees
  - **Pitch**: Distance between the steps of a stair
  - **Horizontal offset**: Distance from the selected reference edge
  - **Vertical offset**: Distance from the selected top edge, if required

4. Select the top edge of the beam. This defines the top elevation of the stair. See Figure 1.



5. Select the elevation plane at 0'- 0" for the stair. This plane defines the bottom elevation of the stair.

6. Select the reference edge for the stair.

7. Set the following parameters:

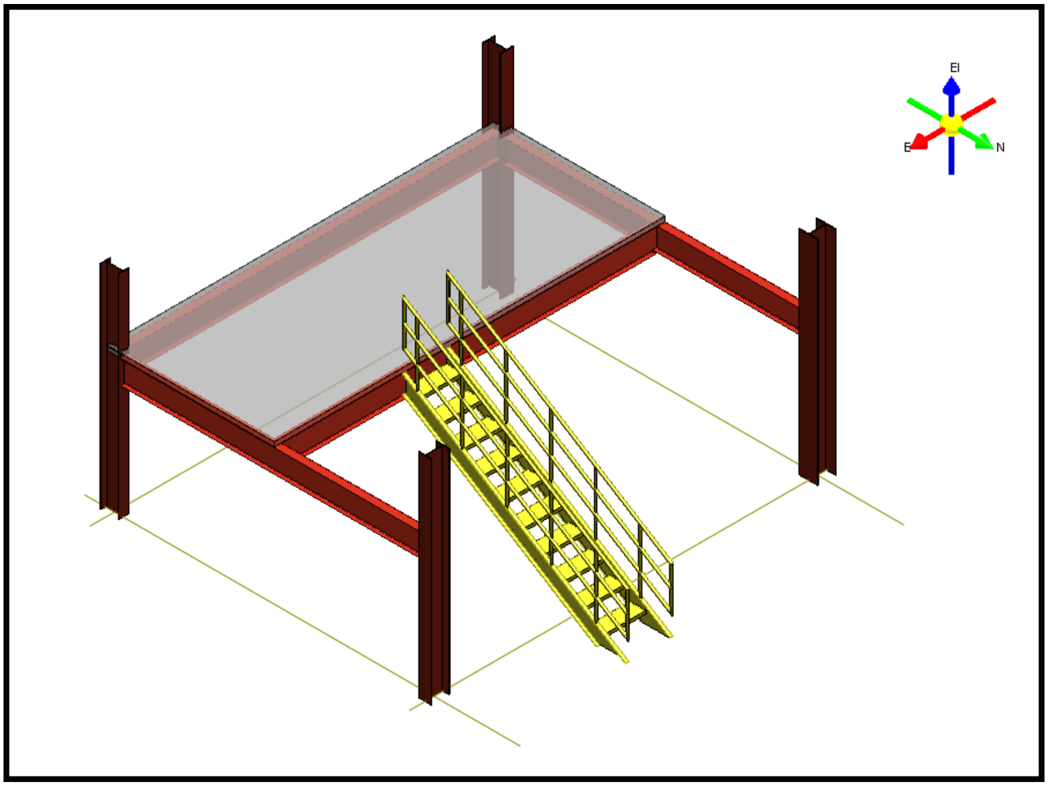
Width:	2' - 6"
Angle:	32 deg
Pitch:	0' -10"
Horizontal Offset:	9' - 6"
Vertical Offset:	0' - 0"
System:	A2-> U03 -> Structural -> Miscellaneous

8. Press the Enter key after keying in the **Horizontal Offset** value. You may have to click the **Side** button on **Place Stair** ribbon to orient the stair correctly.

© Copyright 2014 Intergraph Corporation

9. Click “Finish” Button.

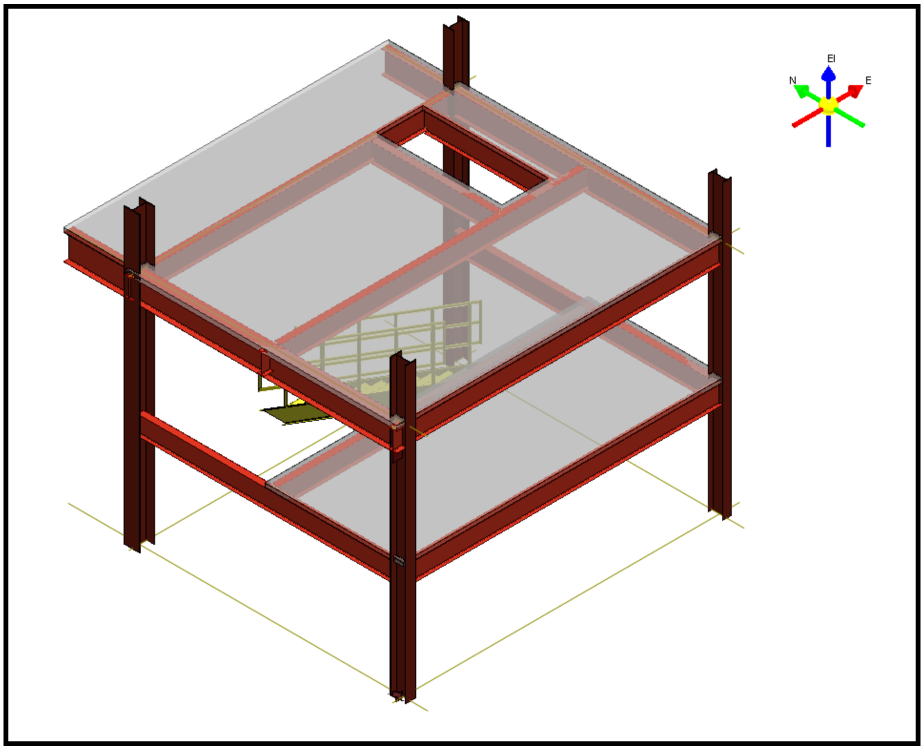
Your View should now resemble the following graphic:



© Copyright 2014 Intergraph Corporation

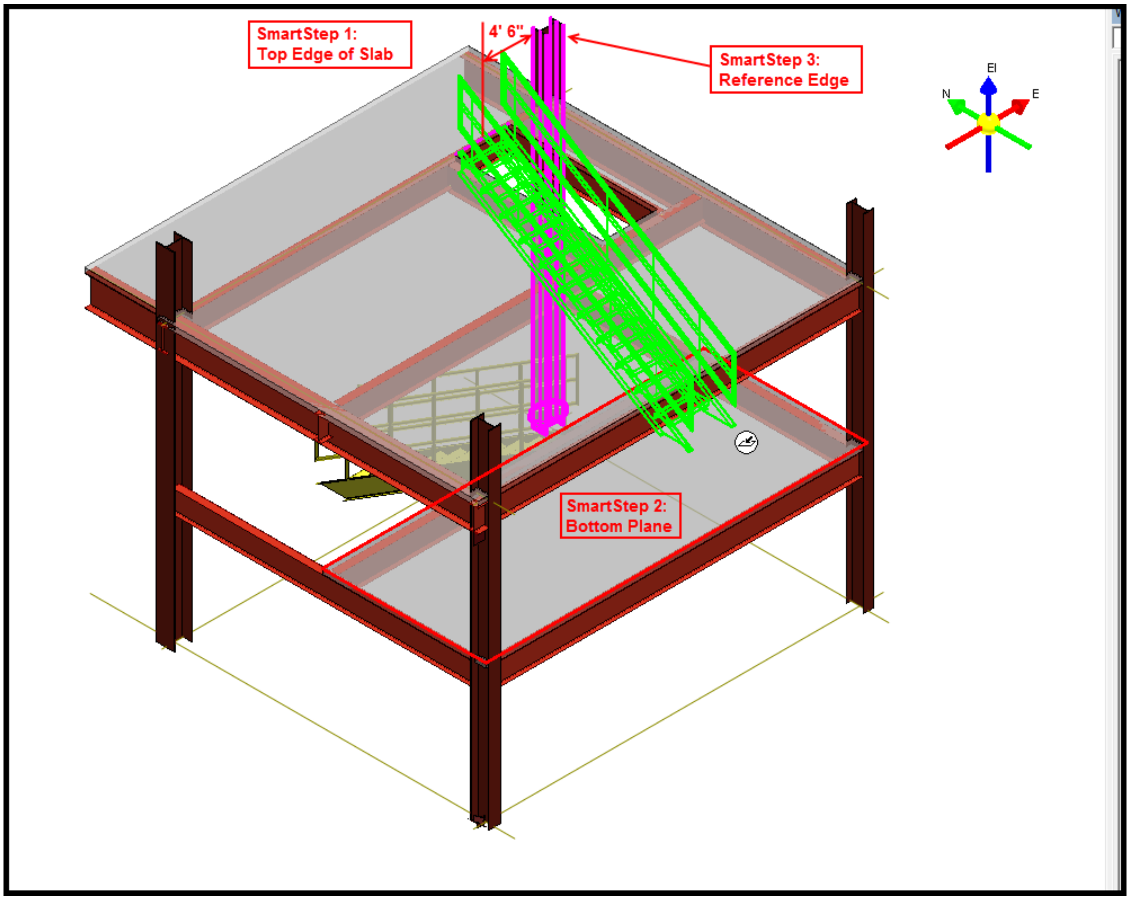
Part II -Place the Second Stair

10. Use Clip by Volume command to adjust the clipping volume as shown below:



11. Select Place Stair Command.
12. Select stair A1 from the catalog browser dialog box.
13. Select the top edge of the beam. This defines the top elevation of the stair. See Figure 2.

© Copyright 2014 Intergraph Corporation

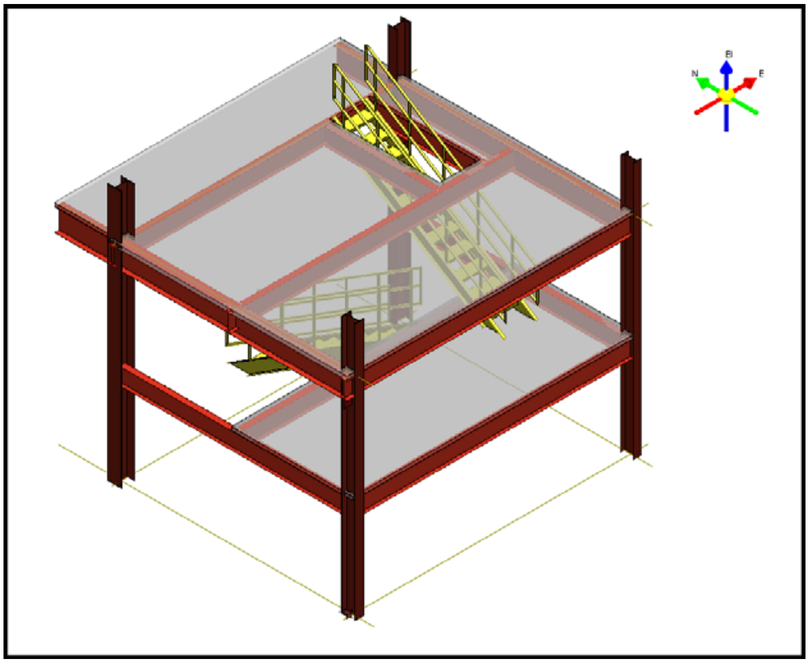


13. Select the top surface of the slab for the stair. This plane defines the bottom elevation of the stair.
14. Select the reference edge (column) for the stair.
15. Set the following parameters:

System:	A2 →U03 → Structural → Miscellaneous
Width:	2'- 6"
Angle:	32 deg.
Pitch:	0.83'- 0"
Horizontal Offset:	4'- 6"
Vertical Offset:	0'- 0"

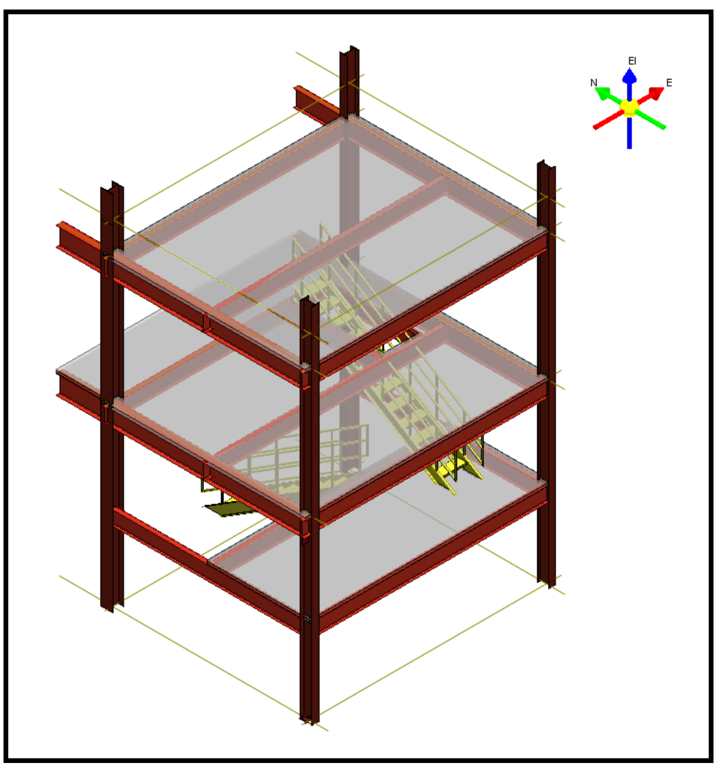
16. Click "Finish" button.

Your View should now resemble the following graphic:

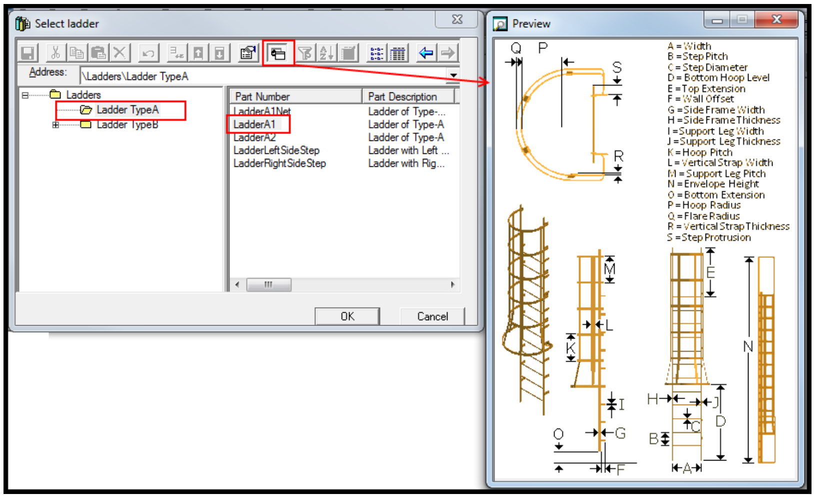


Part III – Place a Ladder

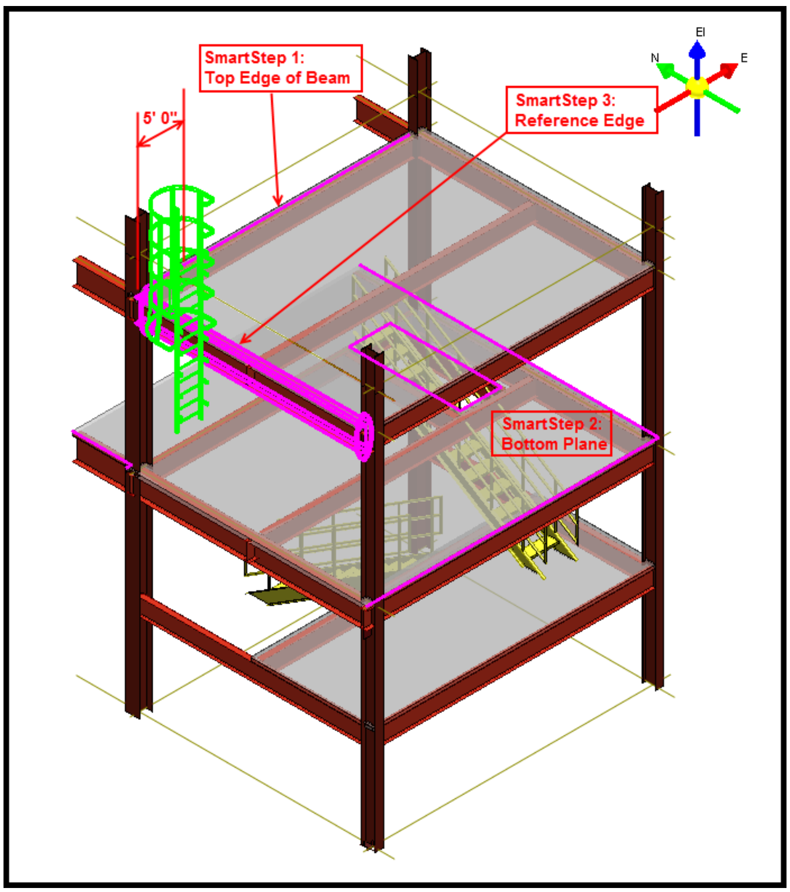
1. Use Clip by Volume command to adjust the clipping volume as shown below:



2. Select Place Ladder Command on the vertical toolbar.
3. Select ladder A1 from the catalog browser dialog box.



4. Select the top edge of the beam. This edge defines the top elevation of the ladder. See Figure 3 for detail.



5. Select the top surface of the slab located at Elevation 18'- 0".This plane defines the bottom elevation of the ladder.

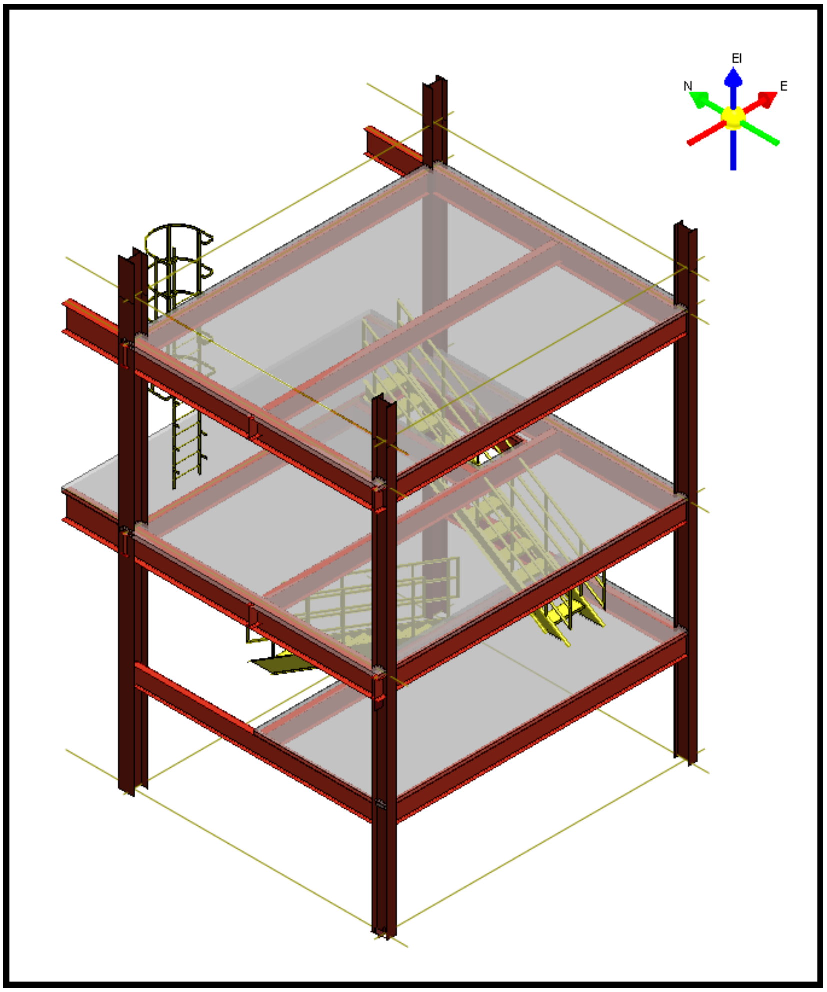
6. Select the reference edge for the ladder.

7. Set the following parameters as follows:

System:	A2 -> U03 -> Structural -> Miscellaneous
Width:	2' - 0"
Angle:	90.0 deg.
Pitch:	1'- 0"
Horizontal Offset:	5' - 0"
Vertical Offset:	0'- 0"

8. Click "Finish" button.

Your View should now resemble the following graphic:



LAB-14: Handrails

Objectives

After completing this lab, you will be able to:

- Understand the handrail entities and relationships
- Place handrails using different methods
- Convert handrails to Designed Handrails containing member systems.
- Modify components of the new Designed Handrails.
- Understand the common tool behavior of the new Designed Handrail.
- Utilize the new “By Point” option of the Place Split command.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

Overview

The handrail design object generates geometry sufficiently detailed for general arrangement drawings and for participation in interference detection. It does not create separate parts with fabrication identities and properties. The 3D model does not contain all the details needed for fabrication of the handrail. This is similar to the function of the ladders, stairs, and equipment design objects in the 3D model. The parameters of the handrail can be used along with standards as specifications for the detailed design.

Different types of handrails are defined in the Catalog with associated graphic symbols. The default Catalog has several handrails with different options. You will need to place a handrail and experiment with the options to understand all the configurations that are available to you. To serve any specific requirements, your company can develop their own handrails. All handrails, however, are positioned in the model by using the same methods — by path or by member.

You place a handrail along a 3D path by using the **Place Handrail** command. The handrail path can be straight, curved, or a combination of both. The points of the path can be on the same plane or can be at different elevations. The points you identify for the path will create the indicated relationships to the objects when the SmartSketch glyphs are displayed when you enter the point. When those objects move, the handrail will be modified.

You can place a handrail relative to one or more members by using the **Place Handrail by Member** command. This command defines the points of the handrail path by the end points of the selected members. It creates a separate handrail for every member. If the members are end-connected, the handrails will also appear end-connected. The end result is a handrail that could have been created by the **Place Handrail** command and selecting the ends of the member as the path points. You edit a handrail placed by the **Place Handrail by Member** command with the same modify ribbon as the handrail placed by the **Place Handrail** command.

© Copyright 2014 Intergraph Corporation

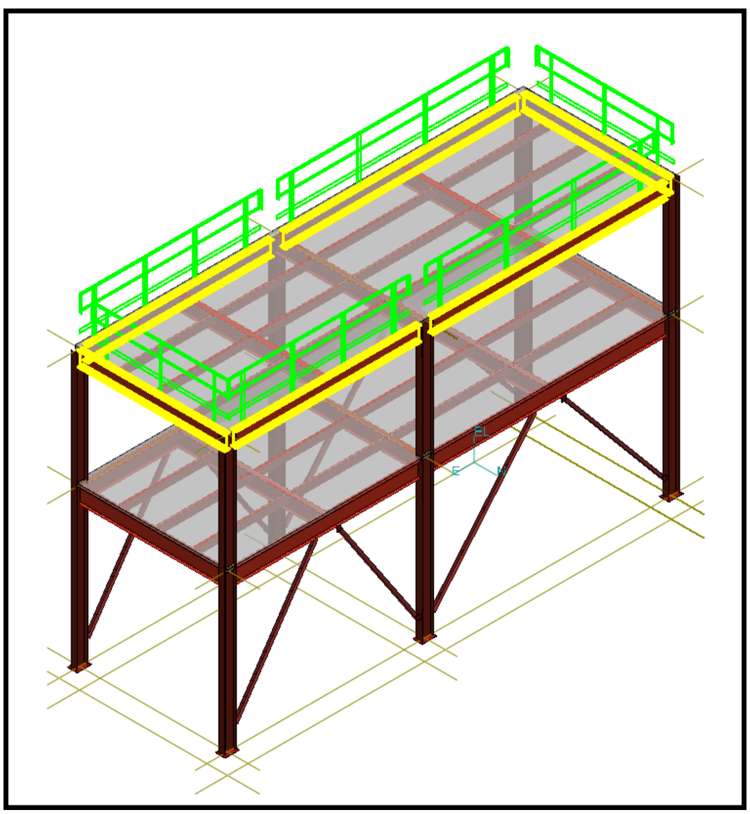
When a handrail is converted, it becomes a Designed Handrail system that contains members systems connected with standard frame connections. Split none connections are placed at the intersections of the posts and rails so the user can decide with member is continuous. New member types have been added to the catalog to support this process.

Part I – Placing Handrails on Unit U02

1. Re-define your workspace to include the A2 → U02 and CS → U02 CS systems.
2. Select Place Handrail by Member Command on the vertical toolbar.
3. Select Handrail TypeA Mounted To Member → SideMountedtoMember from the catalog browser dialog box.
4. Set the following parameters as follows:

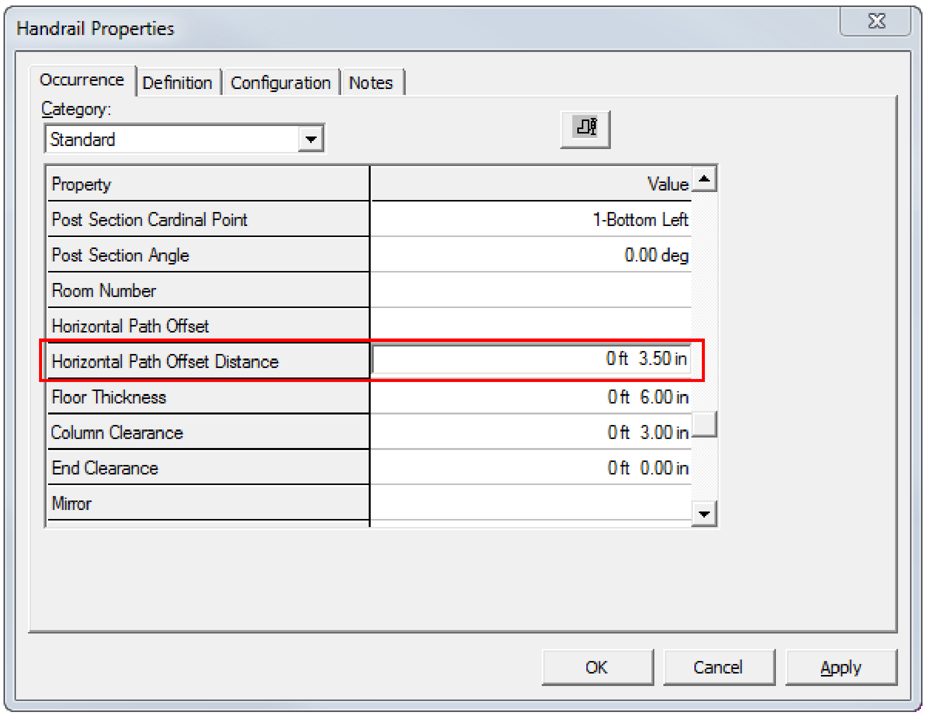
System:	A2→ U02 → Structural →Miscellaneous
Begin Treatment:	Rectangular
End Treatment:	Rectangular
Offset Reference:	Centerline
Column Clearance:	0'- 3"
End Clearance:	0'- 0"

5. Select six beams for the SmartStep1 and click "Accept" button.



6. Select an interior beam as the walking surface location for SmartStep2.

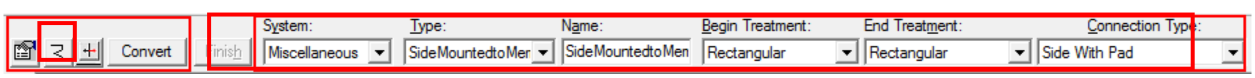
7. Click “Finish” button.
8. Click “Select command” on the vertical toolbar and select all the handrails you have just placed.
9. Open the properties page and change the Horizontal Path Offset Distance to 0’- 3.5”.



10. Click “OK” button.

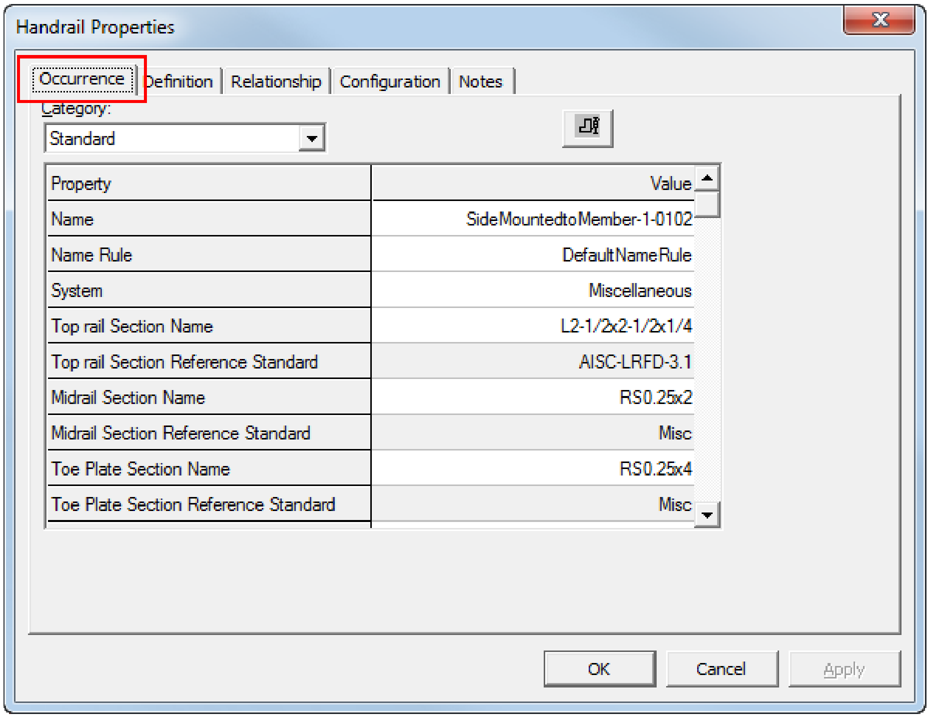
Handrail properties

After placing a handrail, you can edit the handrail properties on the **Edit** ribbon or click the **Properties** smart step on the **Edit** ribbon to view the **Handrail Properties** dialog box. You can use the **Handrail Properties** dialog box to view or edit all properties of the handrail.

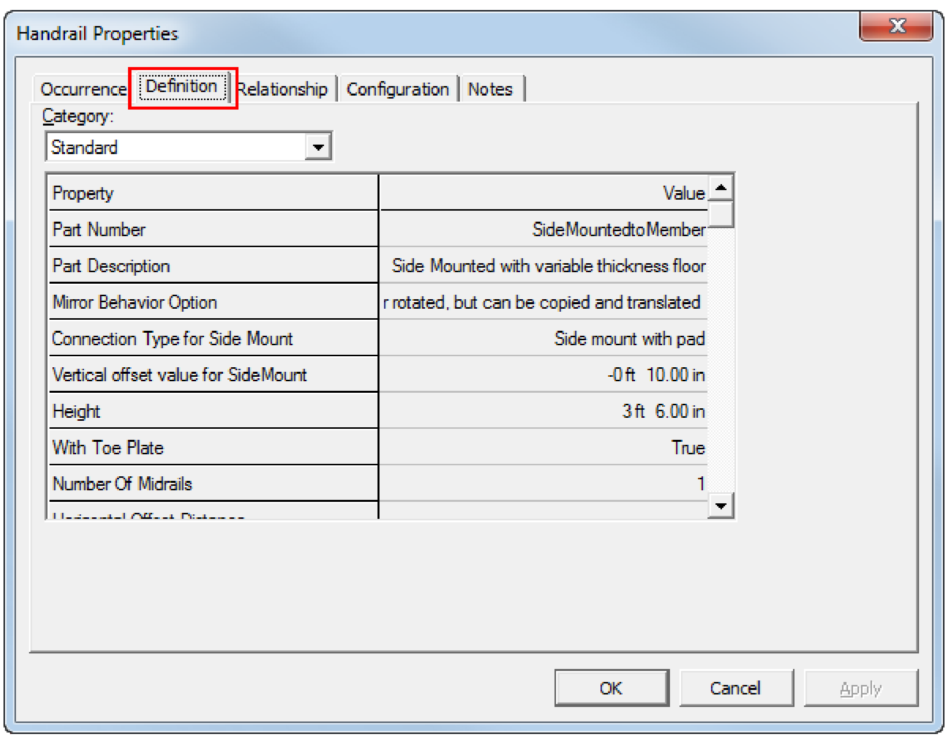


The **Handrail Properties** dialog box has the following tabs:

- **Occurrence**: Displays the properties specific to the handrail type that you can edit for the selected handrail occurrence. The properties are organized into categories. The default values for the occurrence properties are defined in the Catalog. The Catalog administrator can define a number of typical handrails with different default values for the occurrence properties. This gives you a quick way to place typical handrails, but still allows you to make changes to the property values after you have placed the handrail.

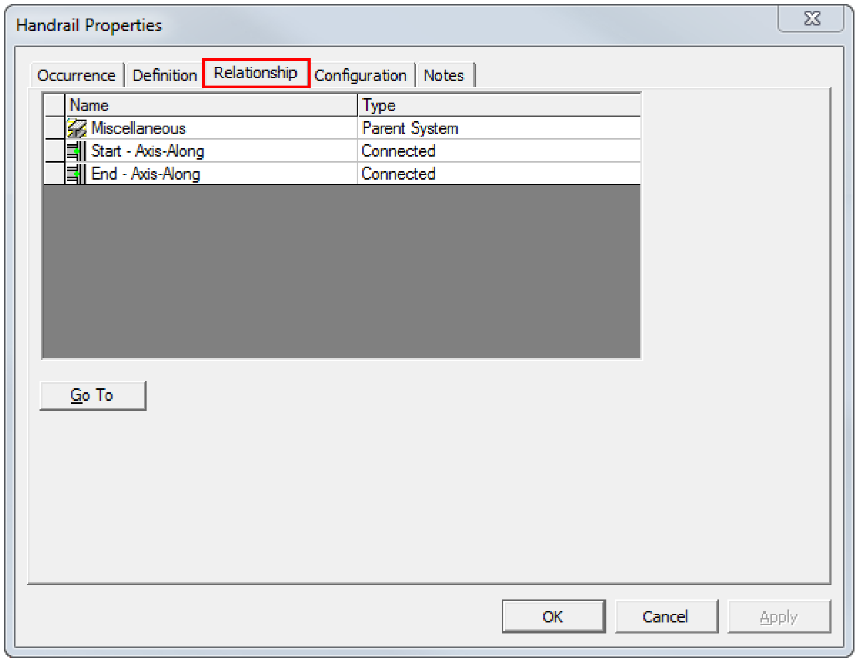


- **Definition**: Displays all properties for which values are defined in the Catalog. This includes the occurrence properties for which defaults are recorded in the reference data. You cannot edit properties on this page.

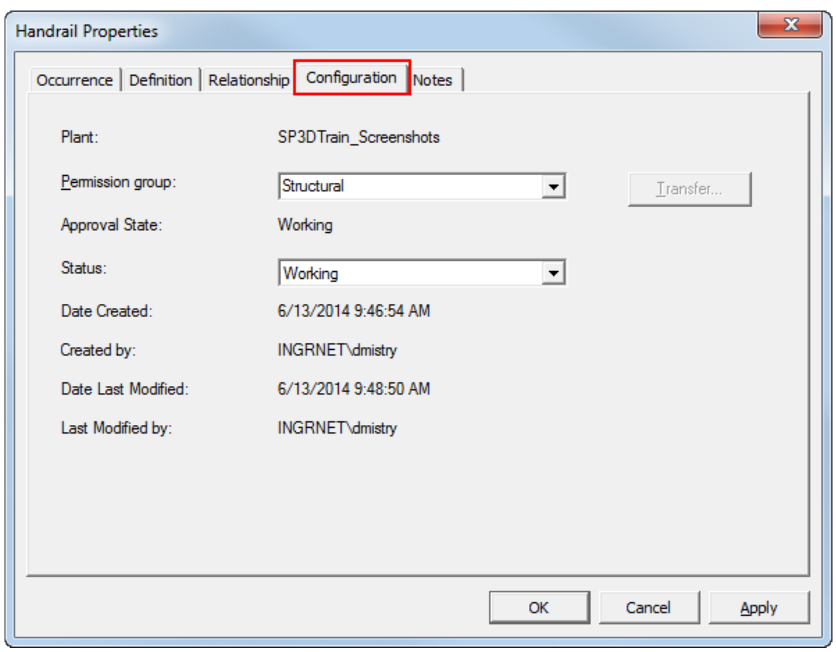


- **Relationship**: Displays the objects that are related to the handrail in the following ways:
  - o System parent

- o WBS parent (if any)
- o Associative point parents (if any)

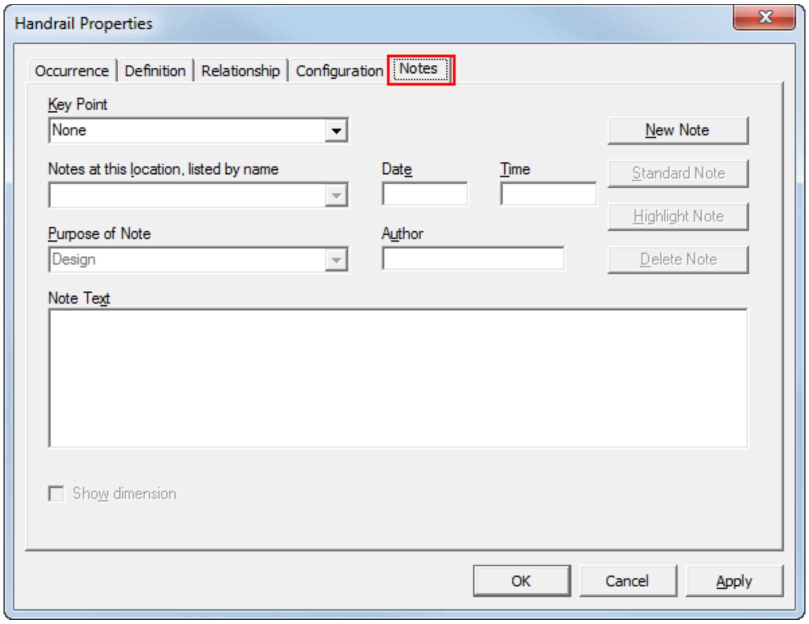


- **Configuration**: Displays the creation, modification, and status information. You can edit the permission group and the status on this page.

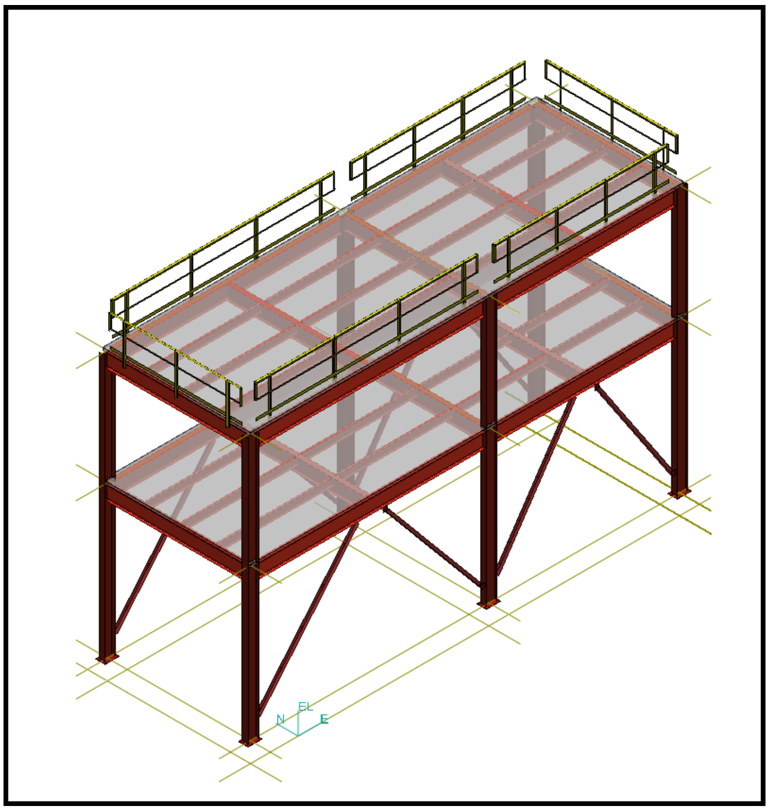


- **Notes**: Allows you to create and edit special instructions about the design object. The administrator can configure drawings to automatically create labels with the text from a note.

© Copyright 2014 Intergraph Corporation

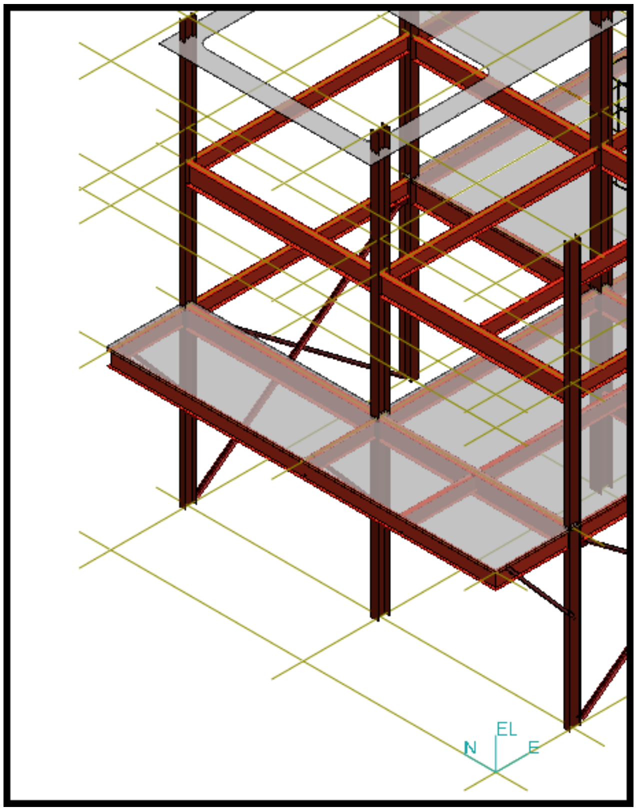


Your View should now resemble the following graphic:



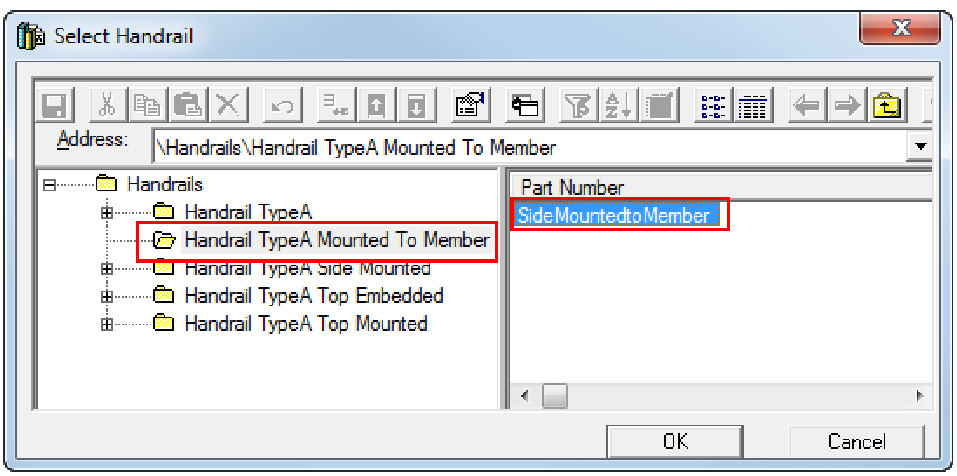
Part II – Placing handrails on Unit 03

1. Re-define your workspace to include the A2 → U03 and CS → U03 CS systems.
2. Use Zoom Tool to window area the West side of the building as shown below:



3. Select Place Handrail by Member Command on the vertical toolbar.

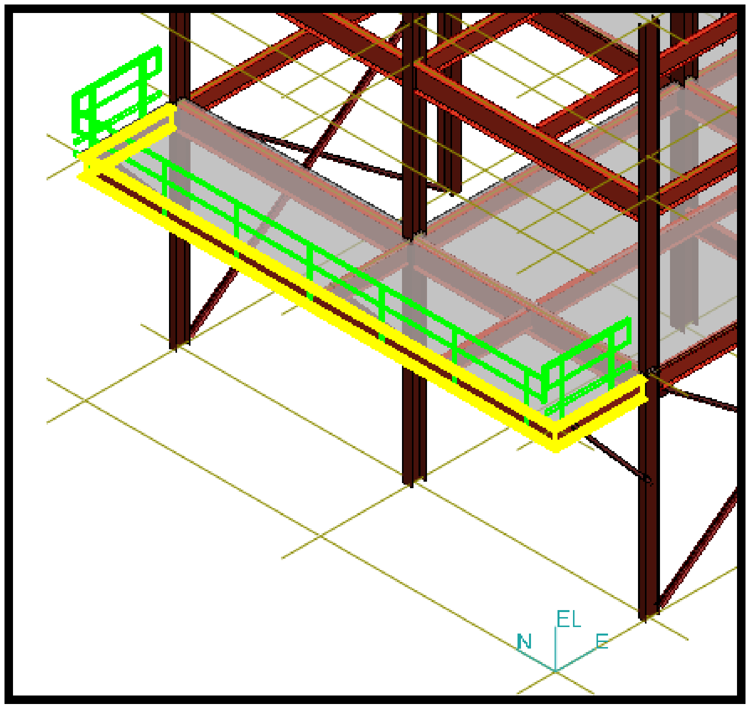
4. Select handrail SideMountedtoMember from the catalog browser dialog box.



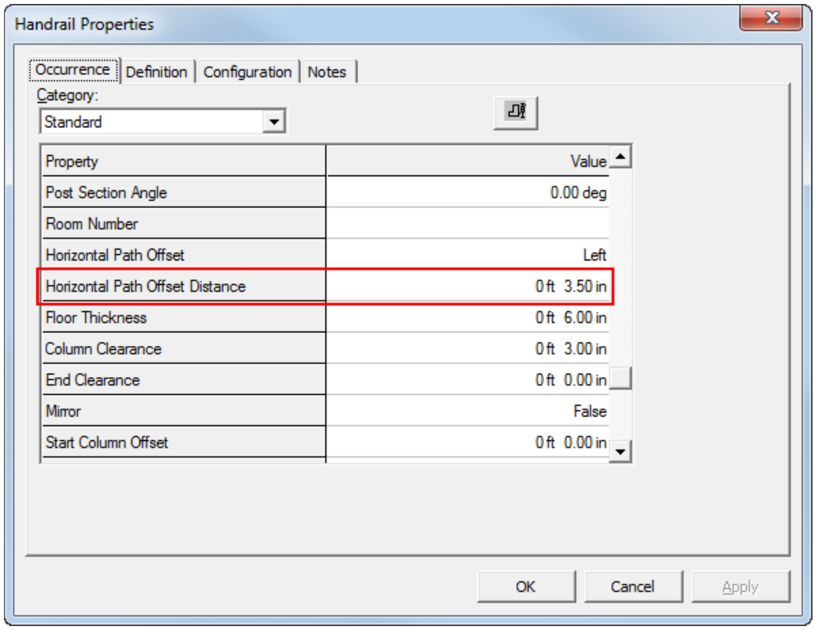
6. Set the following parameters as follows:

System:	A2 → U03 → Structural → Miscellaneous
Begin Treatment:	Rectangular
End Treatment:	Rectangular
Offset Reference:	Centerline
Column Clearance:	0’- 3”
End Clearance:	0’- 0”

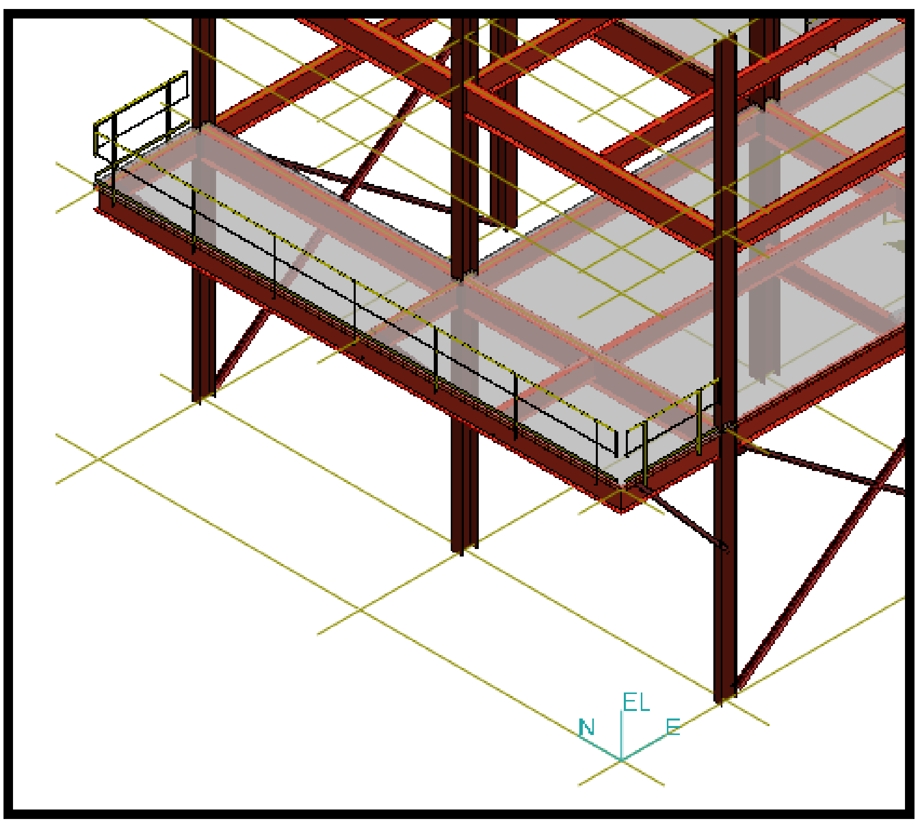
7. Select three beams for the SmartStep1 and click “Accept” button.



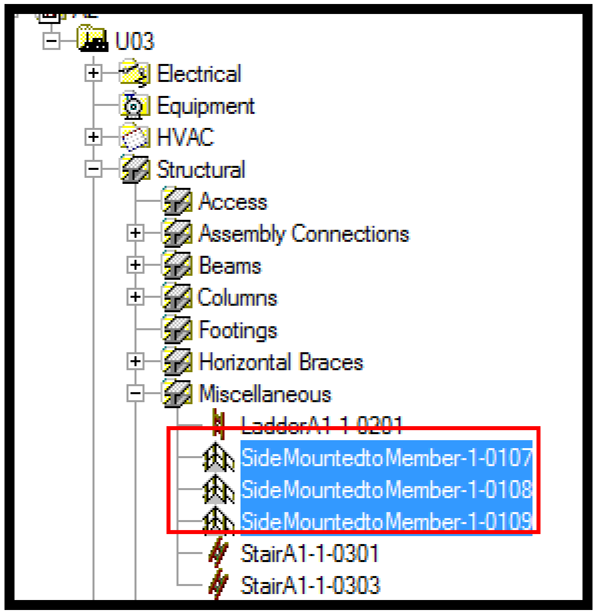
8. Select the slab as the walking surface location for SmartStep2.
9. Click “Finish” button.
10. Click “Select Command” on the vertical toolbar and select all the handrails you just have placed.
11. Open the properties page and change the Horizontal Path Offset Distance to 0’- 3.5”.



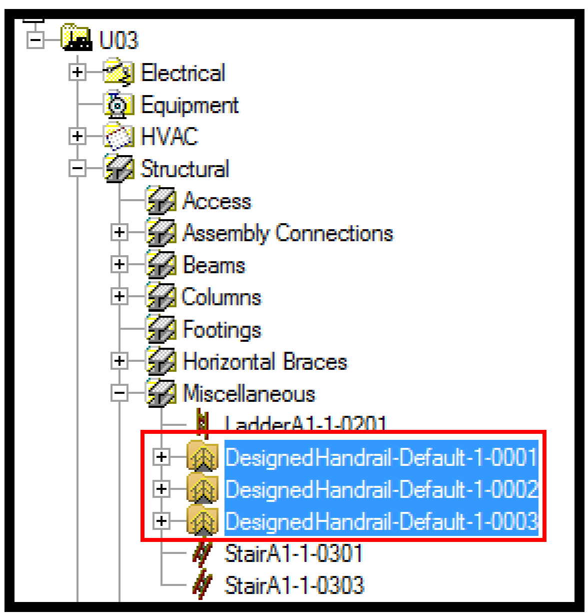
11. Click "OK" button.
    Your View should now resemble the following graphic:



12. Select the handrails that you just placed. Notice that these handrails are symbol based handrails and the each handrail corresponds to one object as can be seen in the workspace explorer.

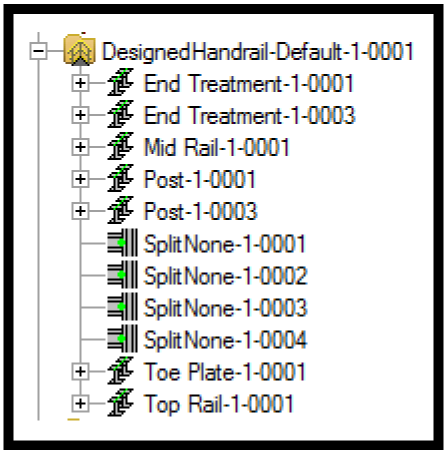


13. While the selection is still active, click on the Convert button on the Handrail ribbon bar. Click “Yes” in the warning dialogue that appears. Once the convert operation is complete note that the handrail is changed on the WSE to a Designed Handrail.



14. Expand one of the Designed Handrails on the WSE and review the contents. New member types specific to handrails have been added to the structural catalog. When new members are added to a Designed Handrail, these types should be used. Also notice the “split none” connections. These connections can be modified by the user to split the rails or posts, depending on which is continuous.

© Copyright 2014 Intergraph Corporation



Note:
At this time, the Designed Handrail is just a structural system folder containing all the handrail parts. Therefore it behaves like a system folder and not like a Designed Equipment. When the top level Designed Handrail is selected from the WSE, nothing will highlight in the graphic view. The common commands such as Fit, Clip by Object, and Move/Rotate will not work. To get these commands to work as expected, the user must right-click and use the “select nested” option.

© Copyright 2014 Intergraph Corporation

LAB-15: Footings

Objectives

After completing this lab, you will be able to:

- Understand the footing entities and relationships
- Able to Place different type of footings

Prerequisite Sessions:

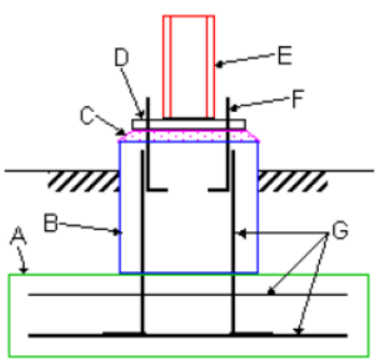
- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

Overview:

The footing generates geometry with a single identity that is sufficiently detailed for general arrangement drawings and for participation in interference detection. It does not create multiple separate parts with fabrication identities and properties.

A typical footing comprises the following parts as shown in Figure 1:

- **Base** - A base supports one or more piers. Some footings for light-weight structures such as a light pole do not have a base.
- **Pier** – A pier 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.
- **Grout** – A grout layer rests on the pier and supports the base plate.
- **Base Plate** - A base plate rests on the grout layer and is welded to the supported column.



A	Base
B	Pier
C	Grout
D	Base Plate
E	Column
F	Anchor Bolts
G	Reinforcing Bars

© Copyright 2014 Intergraph Corporation

The default structure catalog provides several types of footings. The footings in the default catalog have properties to size the geometry for all of the above components except the anchor bolts and reinforcing bars. No properties for these components of the footing are provided. The catalog administrator can add properties to the footing in the catalog to allow you to document these components if it is important to your workflow.

There are two types of footings that you can place:

- **Single Footing -** Single footing supports a single column.
- **Combined Footing** - Combined footing supports multiple columns.

  Notes:
  - *Combined footing requires a minimum of two columns for placement. Combined footing with a single column are put to the **To Do List.***
  - *When placing a combined footing with a merged pier, all columns must have the same bottom elevation. The software does not allow adding a column with a different bottom elevation to the merged pier.*

There are two methods to place footings:

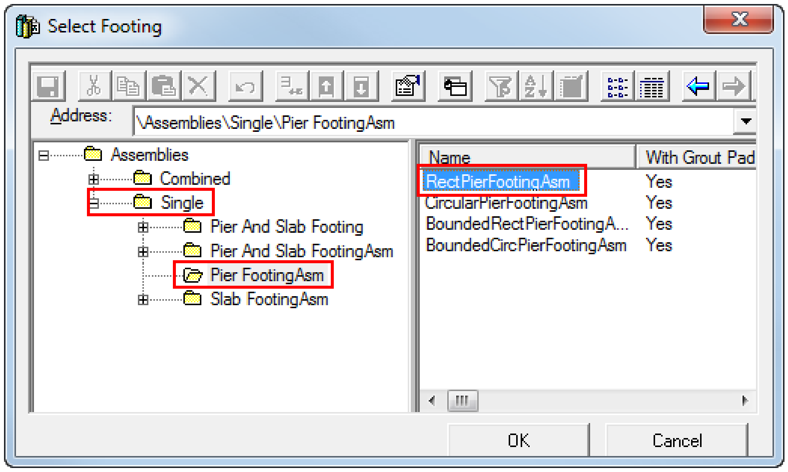
- **By Member** – You select the ends of members to position the footing.
- **By Point** – You place the footing at a specific point.

  Note:
  - *The method for placing footings by point is normally used to place footings relative to grid intersections - before the placement of members.*

In both cases, the height of the footing can be defined by the identification of a supporting surface or by entering the height of the footing, depending on the specific type of footing chosen from the catalog.

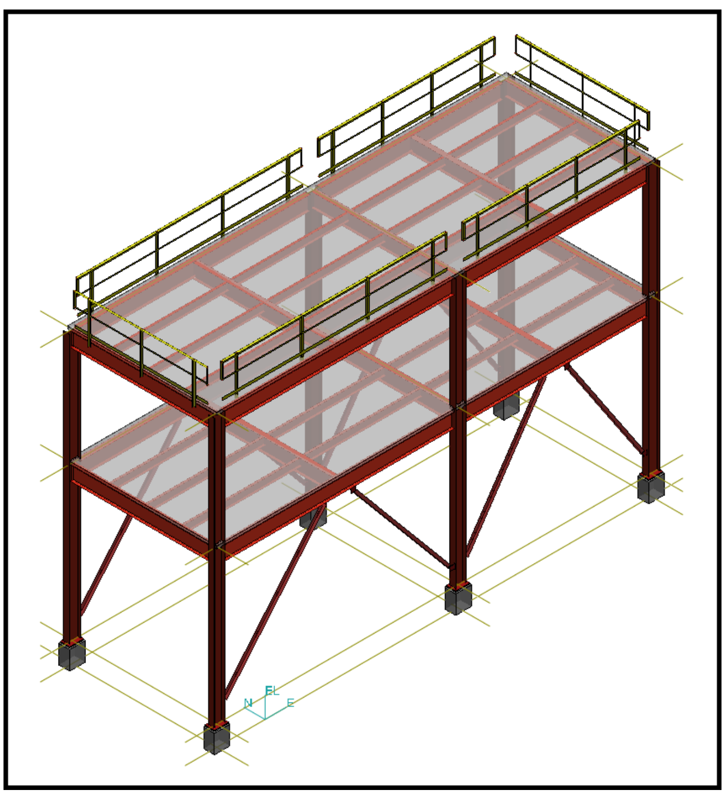
Steps to place the footings:

1. Re-define your workspace to include the A2 → U02 and CS → U02 CS systems.
2. Select Place Footing Command on the vertical toolbar.
3. Navigate the Assemblies Hierarchy and select RectPierFootingAsm from the catalog browser dialog box as shown below.



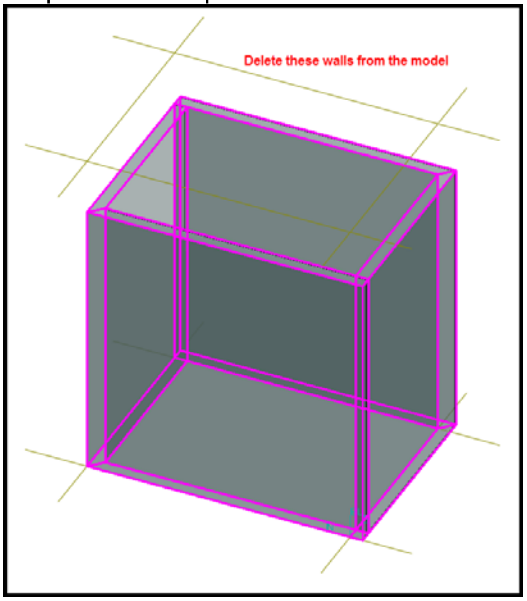
4. Click “OK” button to close the catalog browser dialog box.
5. Select all columns in U02 and then select “Accept” button.
6. Go to the System combo box and select A2 → U02 → Structural → Footings

7. Click the “Finish” button to place all footings in the model.

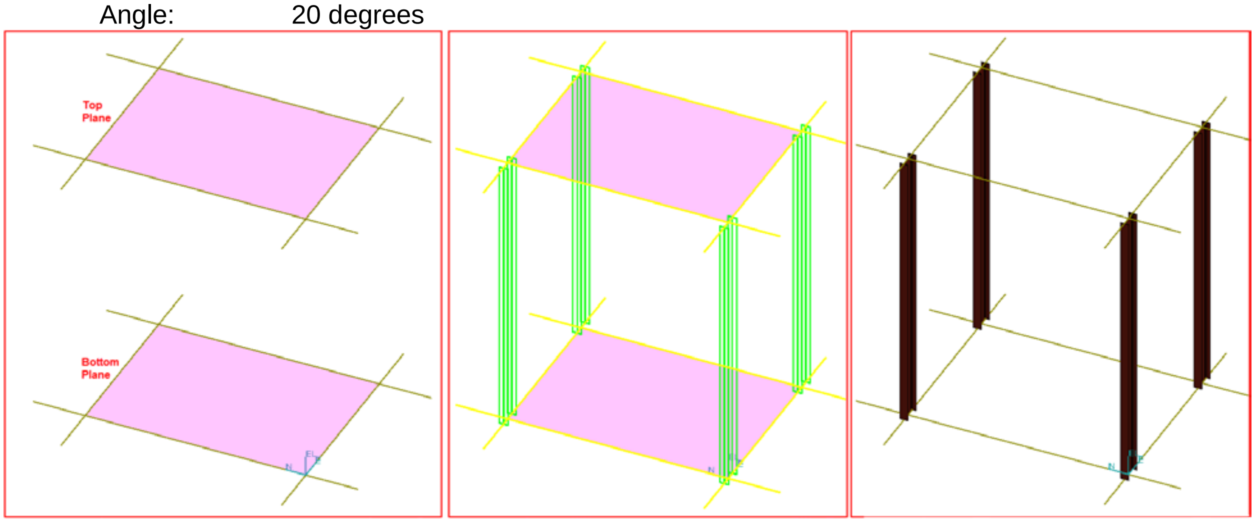


Placing and Modifying Combined Footings:

1. Define the workspace with only the U06 CS and the U06 Structure folder. Set the view to Plan and fit the view.
2. Select the two walls that were placed in the previous lab exercise and delete them.



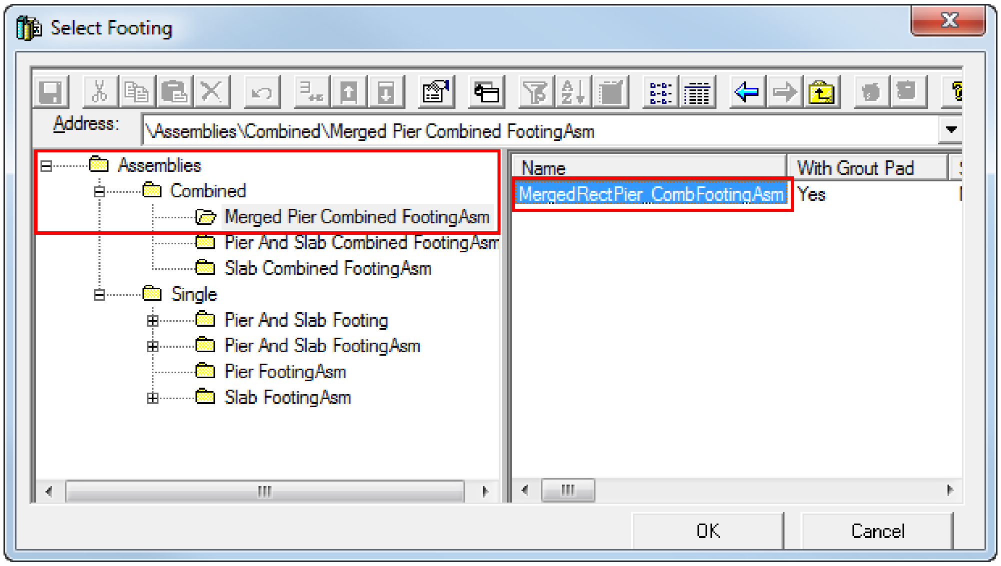
3. Now we will place some members using the U06 CS coordinate system.
4. Using the Place Columns at Grid Intersections command, place columns at the four grid intersections as shown below.
   System: A2→U06→Structural→Columns.
   Type Category: Column
   Type: Column.
   Section: AISC W14x30
   Cardinal Point: 5
   Angle: 20 degrees



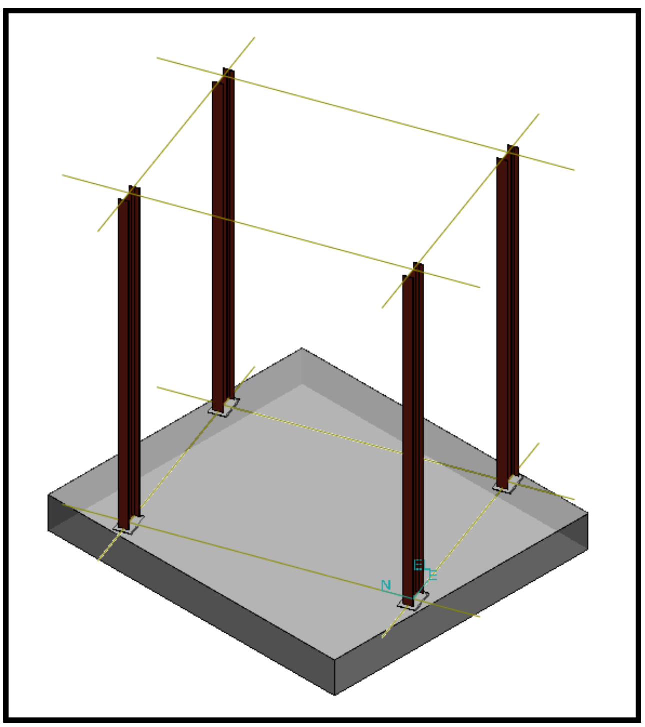
5. Using the Place Footing command, place a combined footing on the four columns.
   Placement: By Member
   System: A2→U06→Structural\Footings.

© Copyright 2014 Intergraph Corporation

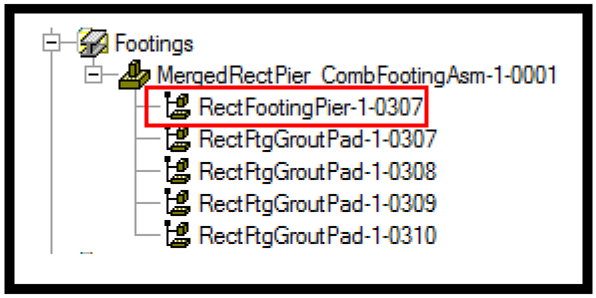
Type: Combined → \Merged Pier Combined FootingAsm → MergerdRectPier_CombFootingAsm



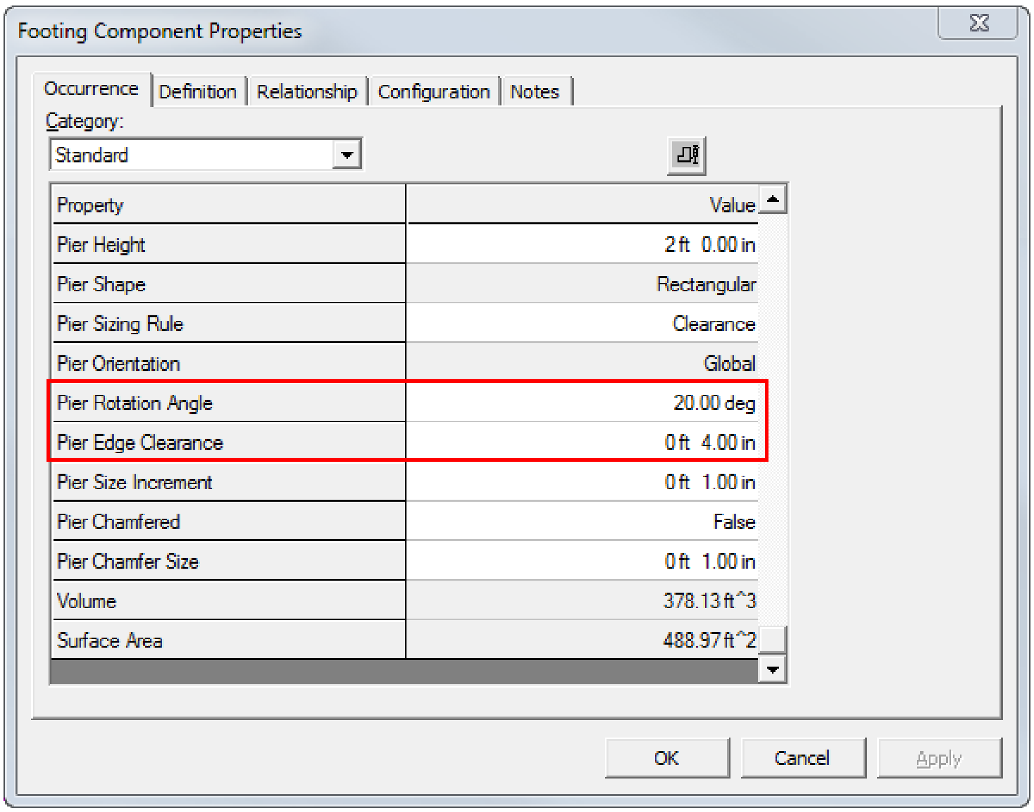
Click Finish. Note that the orientation is not correct.



6. Expand the combined footing on the Workspace Explorer and select the RectFootingPier component.

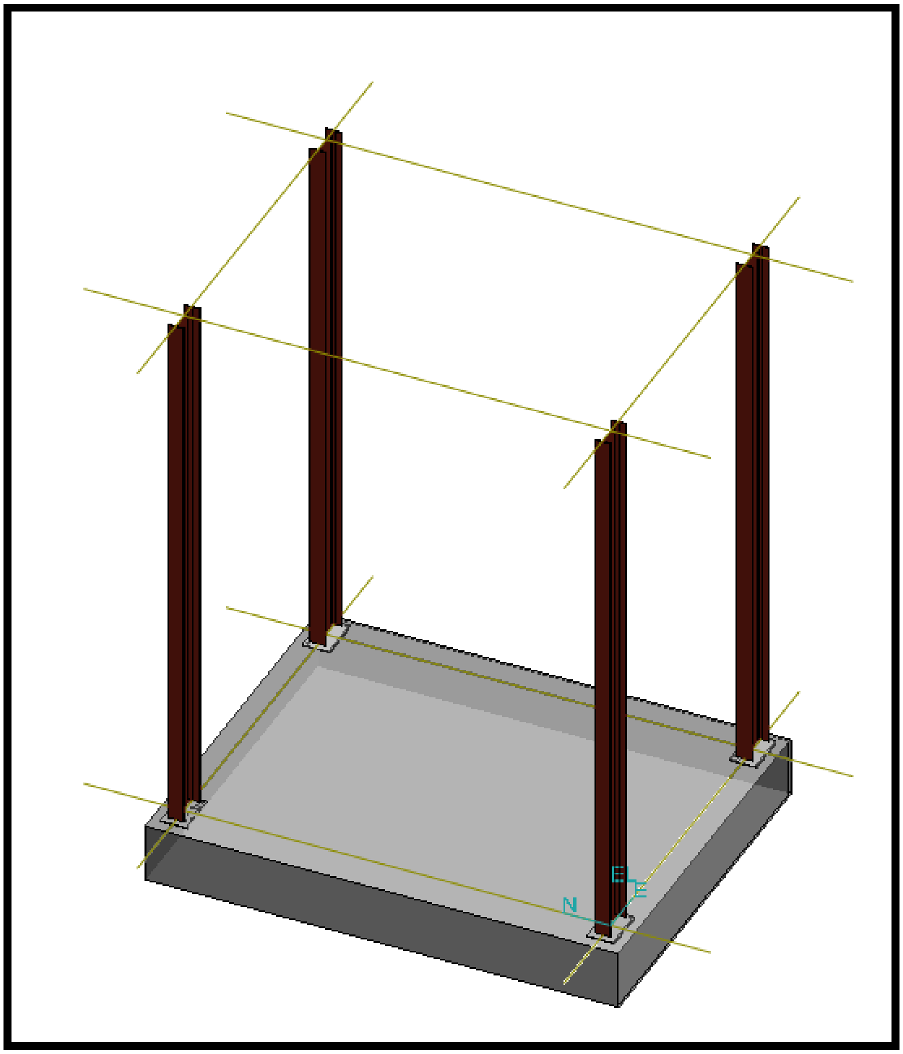


7. Select the Property page and set the Pier Rotation Angle to 20 degrees to match the column angle. Set the Pier Edge Clearance to 4 inches. Click OK to apply the angle and clearance and to exit the form.



8. The orientation of the combined footing is now correct as shown below.

© Copyright 2014 Intergraph Corporation



© Copyright 2014 Intergraph Corporation

LAB-16: Equipment Foundations

Objectives

After completing this lab, you will be able to:

- Understand the footing entities and relationships
- Able to Place different type of foundations

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview
- SP3D Equipment Sessions

Overview:

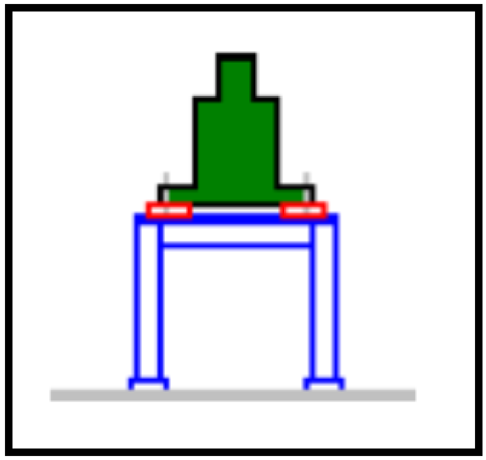
You can place equipment foundations by using the Place Equipment Foundation command. The equipment foundation design object creates standard foundations for equipment as defined in the Catalog. The equipment foundation types in the Catalog can either generate just the geometry for the foundation as a whole, which is similar to the concept of stairs, ladders, and handrails, or create and position other design objects automatically, such as structural members with their own identity and design information. The equipment foundation design object controls the properties that you can edit on the design objects it creates, such as members.

Figure below shows an equipment foundation in which:

- The green object indicates the equipment.
- The red highlight indicates the equipment pad.
- The blue object indicates the legs and support of the foundation.

The gray object indicates the floor, an input to the equipment foundation rather than a component of the foundation.

© Copyright 2014 Intergraph Corporation



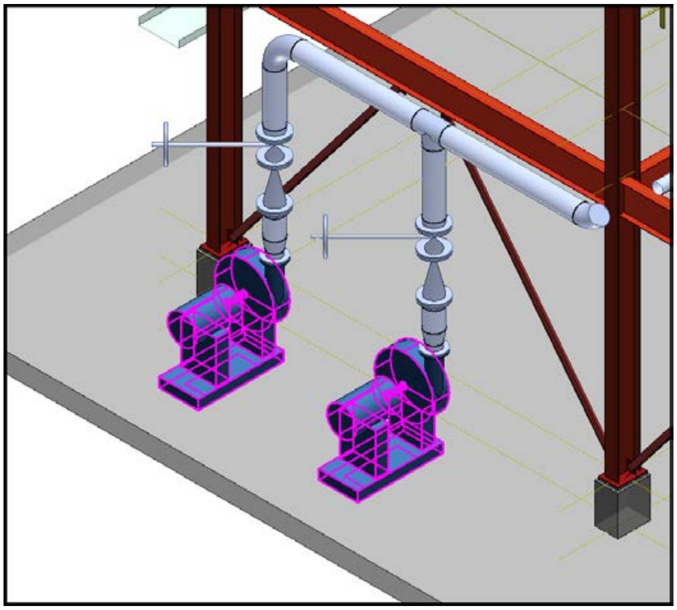
All equipment foundation types are placed in the same way. When equipment has single foundation port, you identify the equipment and the mounting surface for the foundation, such as a floor or a wall. However, if the equipment has more than one foundation port, then you identify the foundation port directly and the mounting surface for the foundation.

The foundation port provides the equipment mounting bolt hole pattern and position information. The equipment foundation type uses this information to size the foundation.

Procedure for placing the equipment foundation:

1. Define your workspace to display Unit **U01** and the coordinate system **U01 CS** and switch to the **Structure** task.
2. On the **Common** toolbar, click the **Fit** button to fit the view on the screen, and then use the **Common Views** command to orient the view such that pumps are visible.
3. Click the **Place Equipment Foundation** button on the vertical toolbar.
4. Select the pumps, **Pump-001** and **Pump-002** located in Unit **U01** to place the equipment foundation.

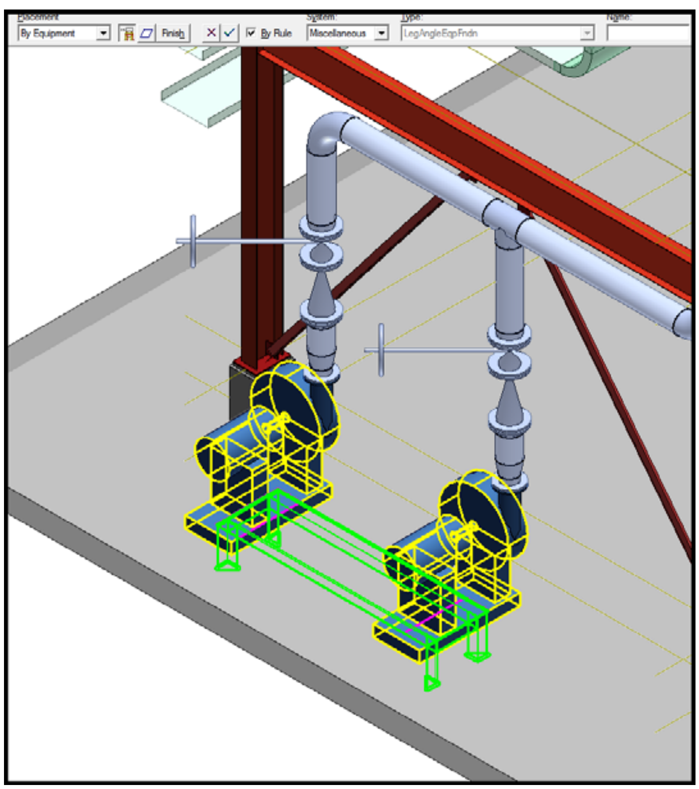
© Copyright 2014 Intergraph Corporation



Note:

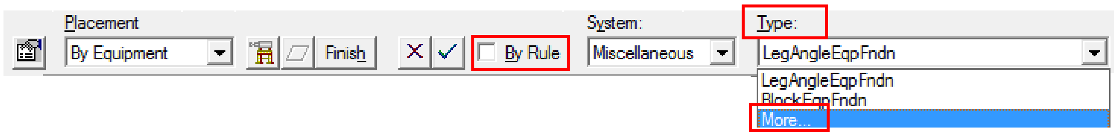
*In the **Type** option on the **Place Equipment Foundation** ribbon, the system automatically selects the default foundation defined in the reference data.*

5. Click the **Accept** button on the **Place Equipment** ribbon. The view of the model will resemble the view displayed in Figure below.



6. On the Place Equipment Foundation ribbon, click the More… option in the System drop-down list and select the A2 → U01 → Structural → Miscellaneous system.

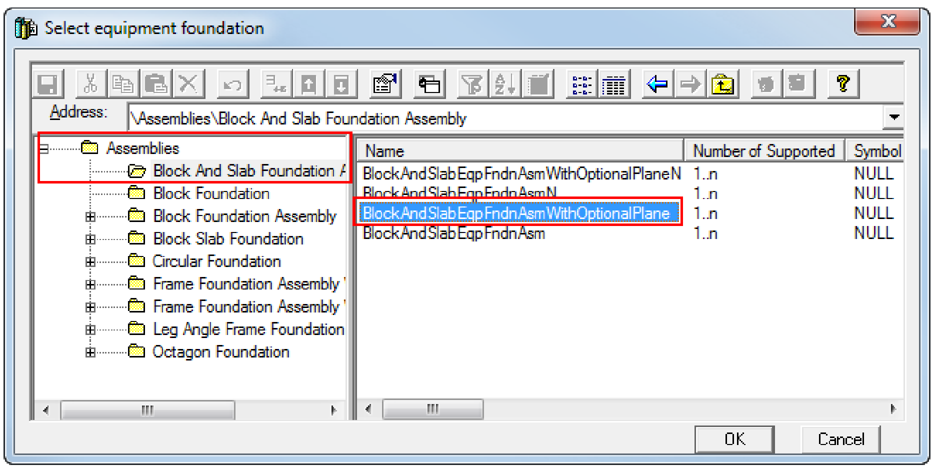
7. Deselect the By Rule check box.
8. In the Type drop-down list, click the More… option to select the equipment foundation



Note:

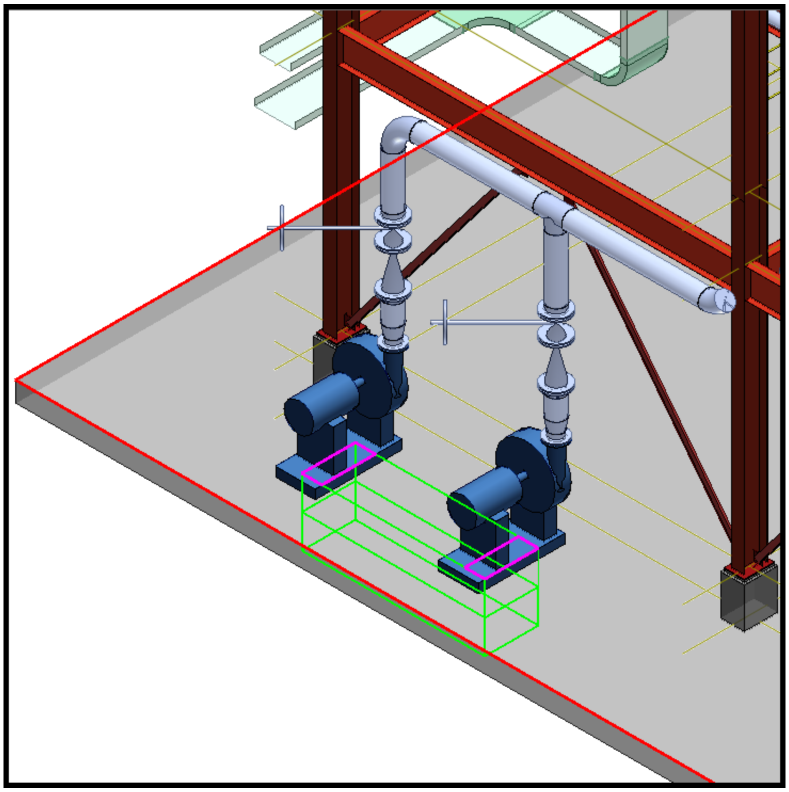
*If you want the software to select the equipment foundation based on the default foundation defined for the equipment part, select the **By Rule** option. The default foundation for the equipment appears in the **Type** box.*

9. In the Select equipment foundation dialog box, navigate to the Assemblies hierarchy, select the Block and Slab Foundation Assembly folder, click BlockAndSlabEqpAsmWithOptionalPlane, and then click OK, as shown in Figure below.



10. The next step is to specify the supporting plane for the equipment foundation. Some equipment foundations do not require you to locate the supporting plane. The foundation automatically finds the surface that is closest under the equipment foundation. Other equipment foundations do not locate a supporting surface at all, but require you to enter the height manually. The command will prompt you for the surface if the selected foundation type requires it. You will see on the **Create** ribbon that the surface selection smartstep is active.
11. Select the slab on the grade as the supporting plane for the equipment foundation, as shown in Figure below.

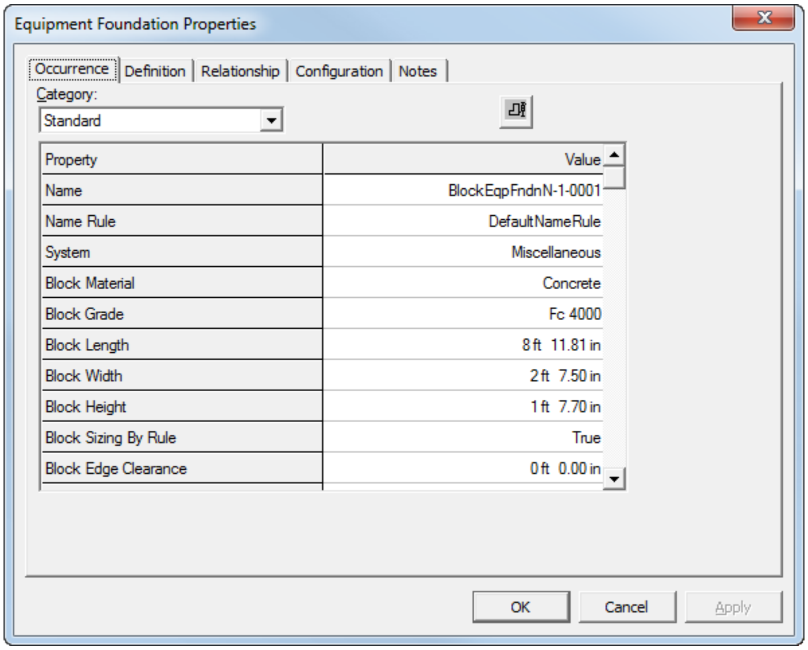
© Copyright 2014 Intergraph Corporation



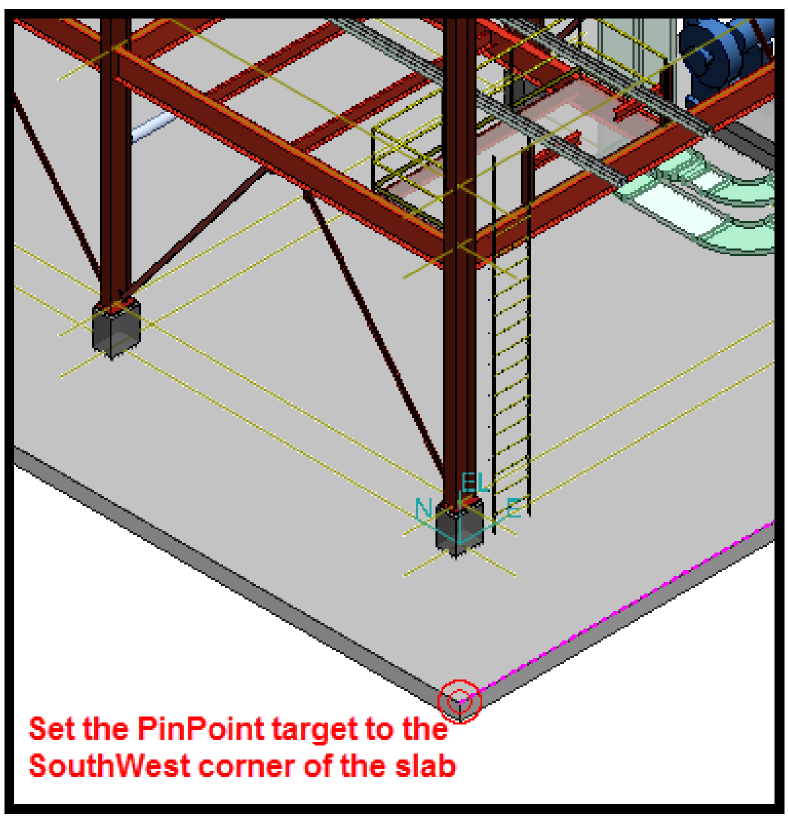
12. Click the **Finish** button on the **Place Equipment Foundation** ribbon to place the equipment foundation

Note:

a. *You can edit the properties of the equipment foundation by activating the **Equipment Foundation Properties** dialog box.*

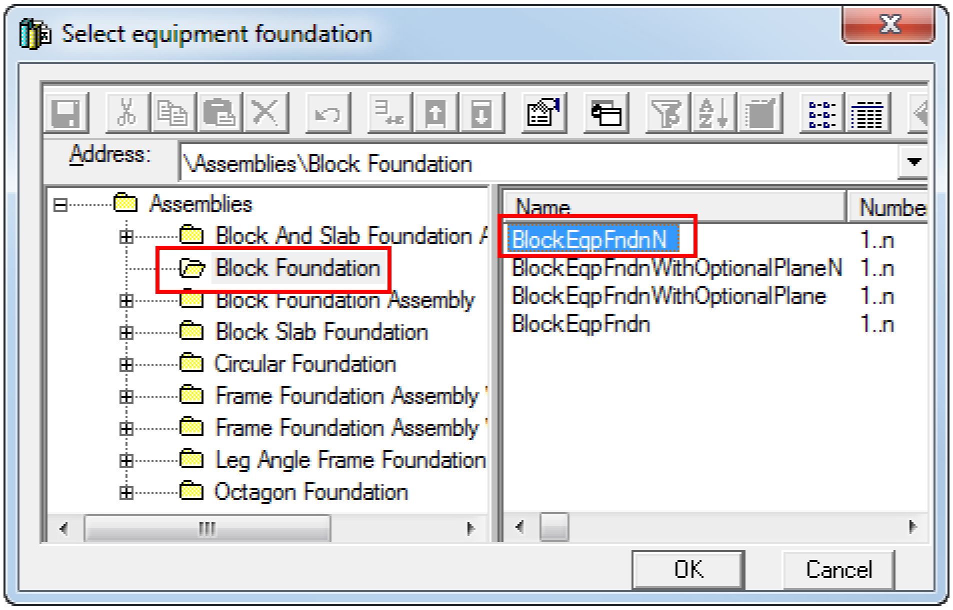


13. Start the Pinpoint command and set target to the southwest corner of the slab as shown in the figure below;



14. Select the Place Equipment Foundation command.
15. Set the Placement option to By Point.

16. Set the System to U01→Structural→Miscellaneous.
17. From the catalog browser Select the Type Block Foundation→BlockEqpFndnN.

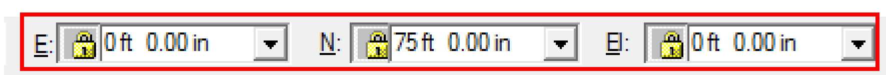


18. On the pinpoint ribbon, key in the coordinates for East, North and Elevation as follows;

    East: 0 ft 0.00 in

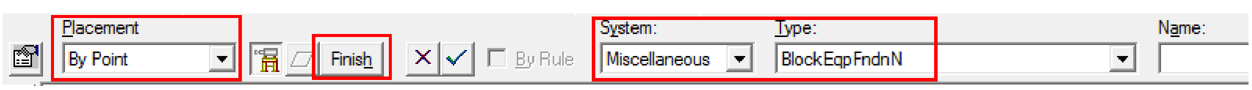
    North: 75 ft 0.00 in

    Elevation: 0 ft 0.00 in



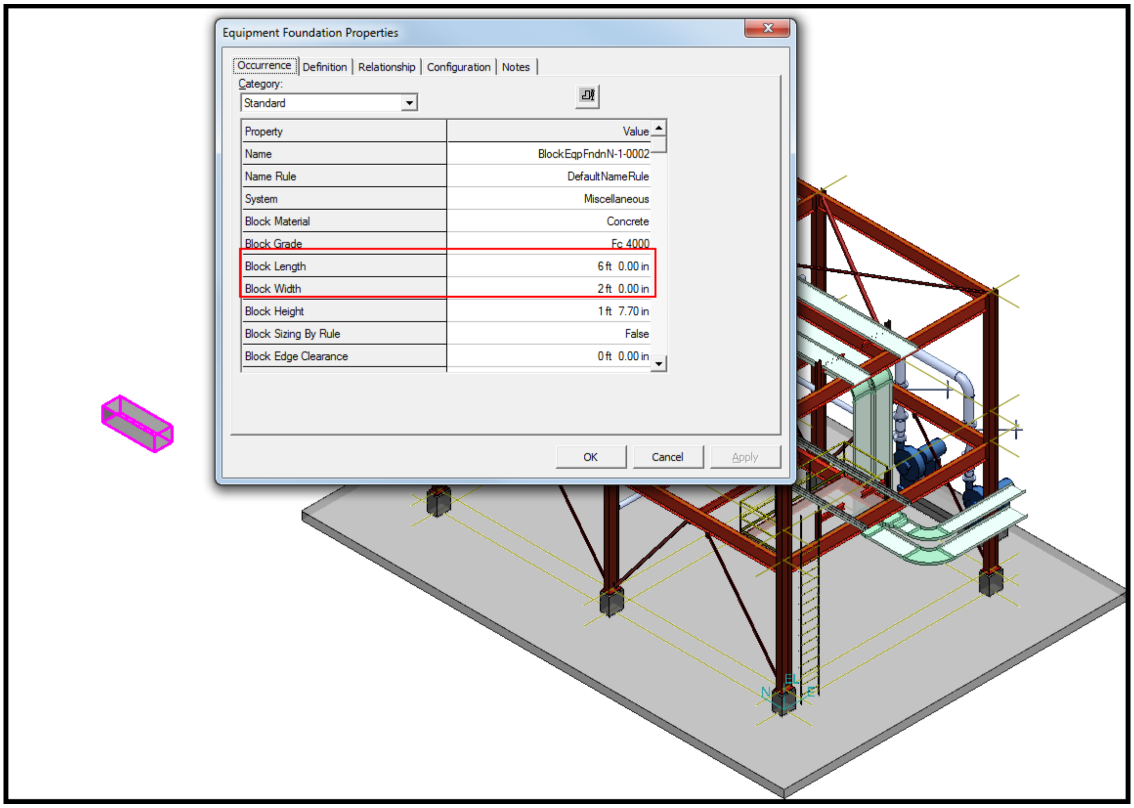
    Click in the graphic view to select the point where the foundation will be placed.

19. Hit finish on the Place Equipment foundation ribbon bar to place the foundation by point.



20. Select the foundation and go to its properties. Modify the Block foundation Length and Width to be 6ft and 2ft respectively. You view should resemble to the graphic below.

© Copyright 2014 Intergraph Corporation


Equipment Foundation Properties
Occurrence
Definition
Relationship
Configuration
Notes
Category:
Standard
Property
Value
Name
BlockEqpFndnN-1-0002
Name Rule
DefaultNameRule
System
Miscellaneous
Block Material
Concrete
Block Grade
Fc 4000
Block Length
6 ft 0.00 in
Block Width
2 ft 0.00 in
Block Height
1 ft 7.70 in
Block Sizing By Rule
False
Block Edge Clearance
0 ft 0.00 in
OK
Cancel
Apply

LAB-17: Place Piles Custom Command

Objective:

By the end of this session, you will be able to:

- Set up the new Place Piles custom command.
- Place and modify piles using the new Place Piles custom command.

Prerequisite Steps:

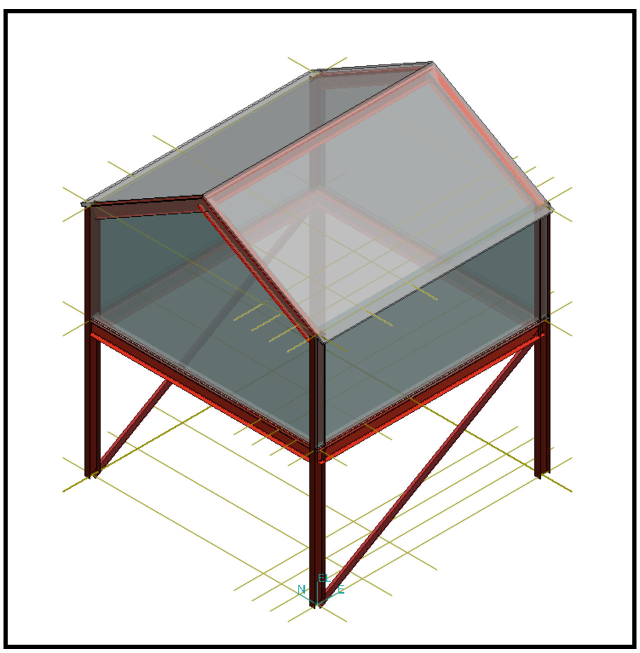
- Place Piles ProgID: PileFoundation,PileFoundation.PileFoundation

Overview:

The Place Piles custom command provides the user a placement wizard to facilitate the modeling of pile members. Through the Place Pile command, the user can specify the number of piles in each direction as well as edge clearances and depth dimensions. The user can also specify the member cross section and material for the piles.

Placing Piles:

1. Open a session file and define a workspace containing the objects in Area A2→U04→Structural and the U04 CS.

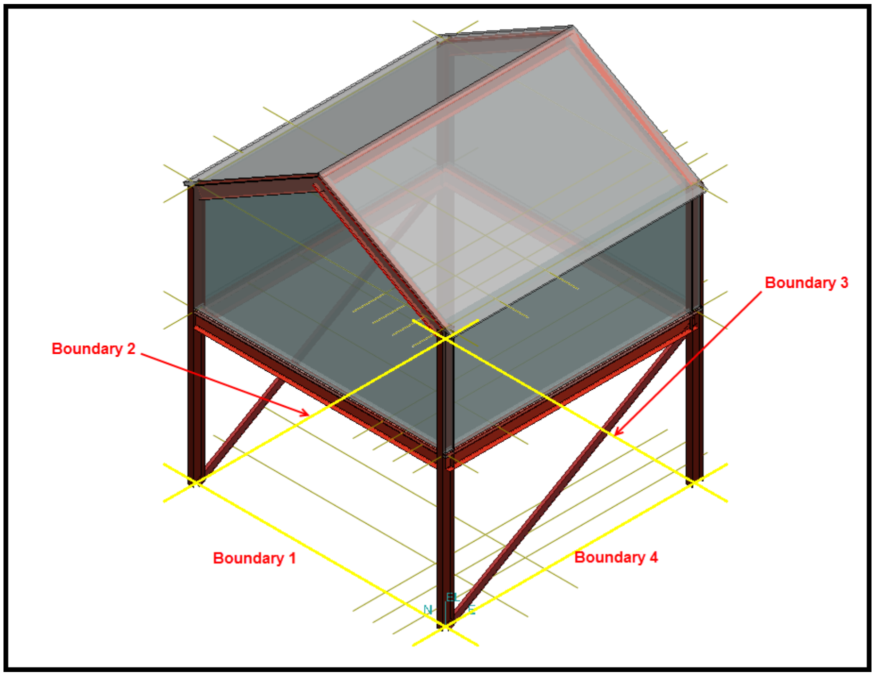


2. Make sure the Active Permission Group is set to *Structural.*

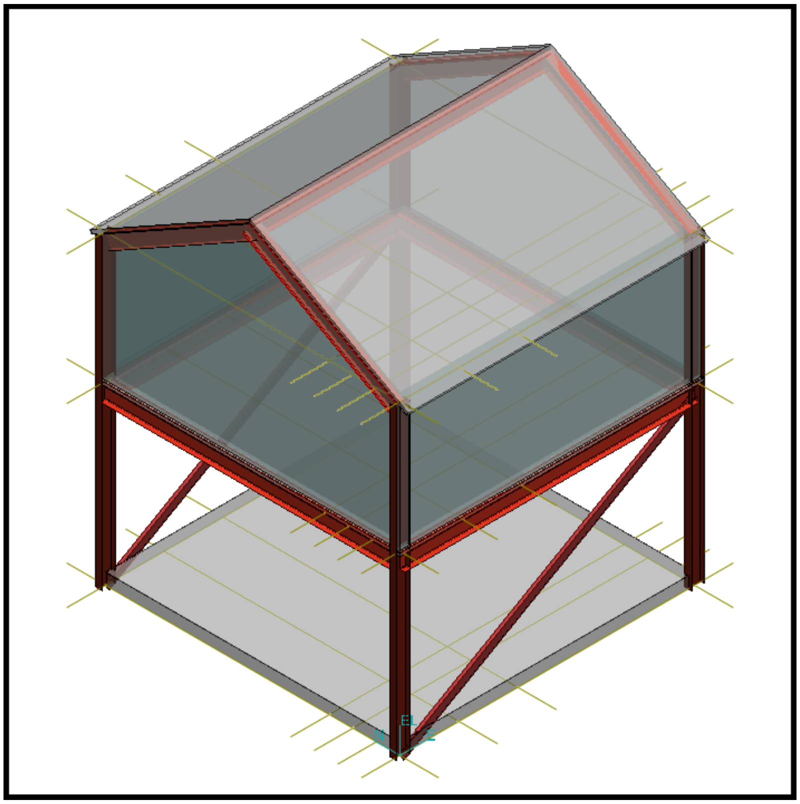
3. Select Place Slab command to place one slab on Elevation plane at 0' 0".
4. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U04 -> Structural -> Slabs
Slab Type:	General Slab
Composition:	Imported Slab
Face Position:	Bottom
Priority:	Primary
Total Thickness:	1 ft 0.00 in

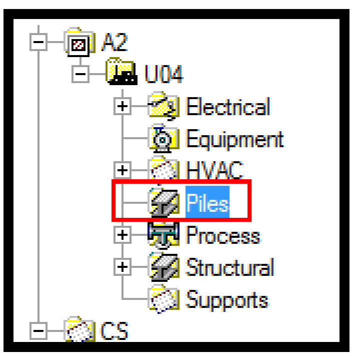
5. Select Elevation Plane at 0'- 0" for the support plane. Click "Accept" button.
6. Select the boundaries as shown in Figure. Click "Accept" button.



7. Your view should resemble the following graphic;

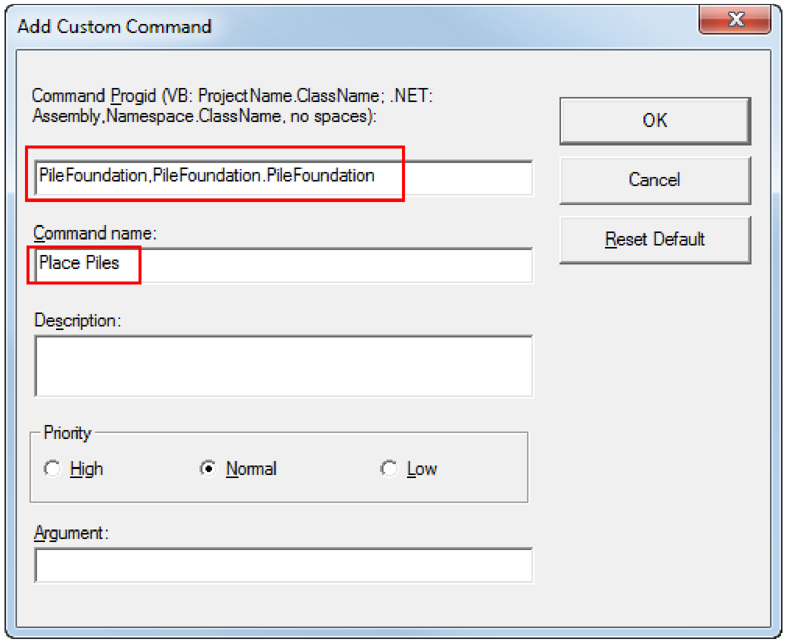


8. Set the locate filter to All. Select the U04 Structural system folder, right-click and select New System. Create a new Structural system named Piles.

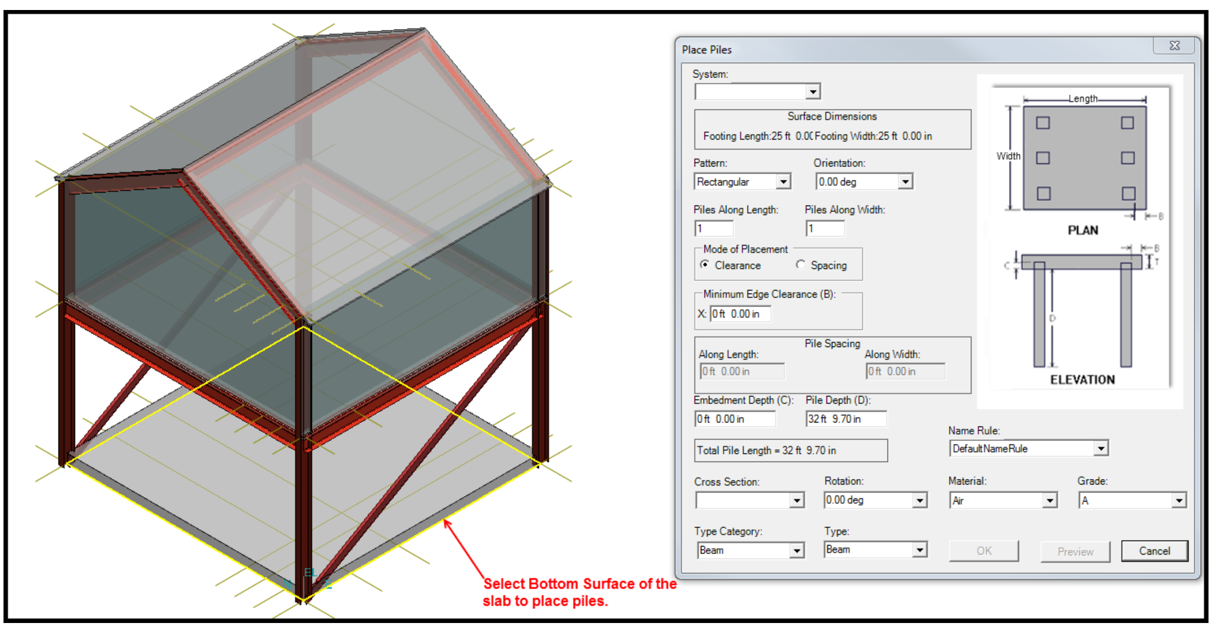


9. Select Tools>Custom Commands. Click the Add button on the right side of the form. Enter the Progid "***PileFoundation,PileFoundation.PileFoundation***", enter a command name and a description if so desired. Click OK to add the Place Pile command to the custom command list.

© Copyright 2014 Intergraph Corporation



10. Select the Place Piles command from the custom command list and click the Run button to start the command.
11. The Place Pile command form is displayed and the user is prompted to select a surface. Select the bottom surface of the slab that you placed in the previous steps (use quick pick).

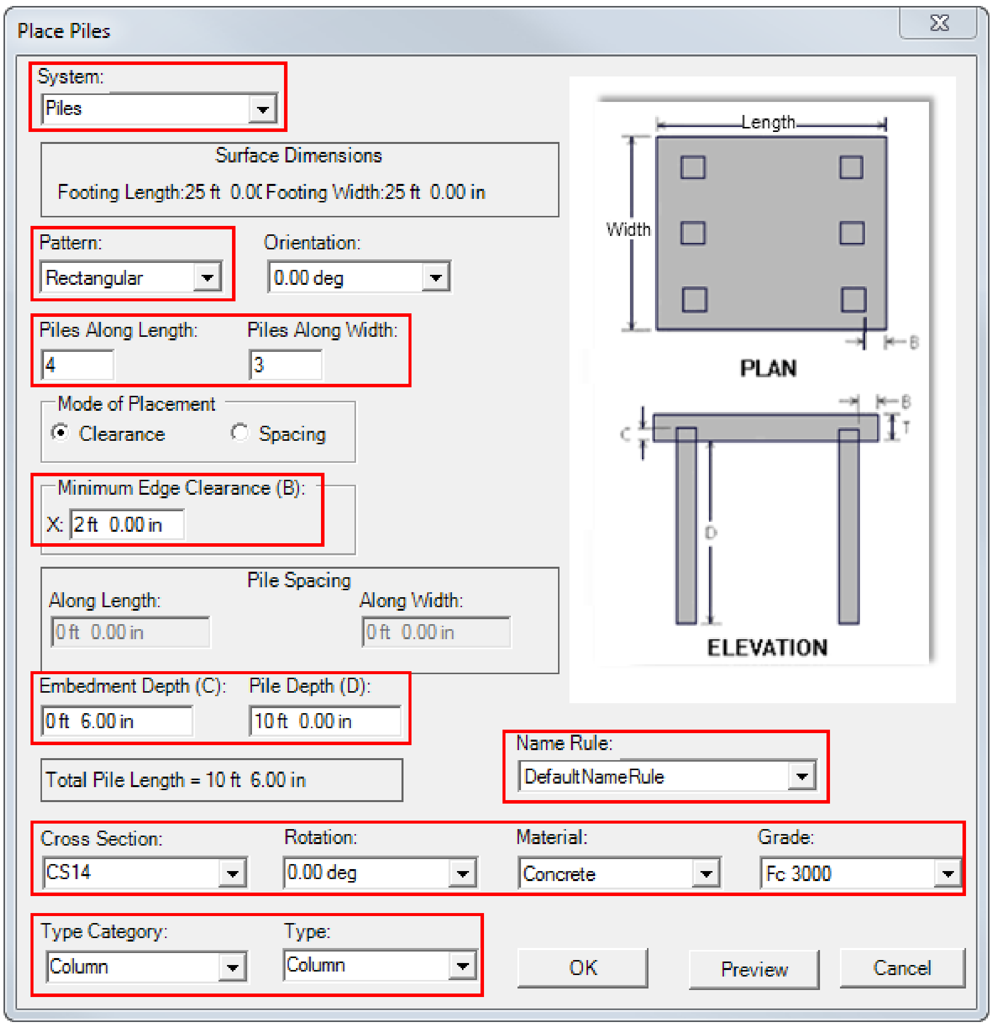


12. From the System pull down, select the new Piles system. Complete the settings below:

- Piles along Length: 4
- Piles along Width: 3
- Mode of Placement: Clearance

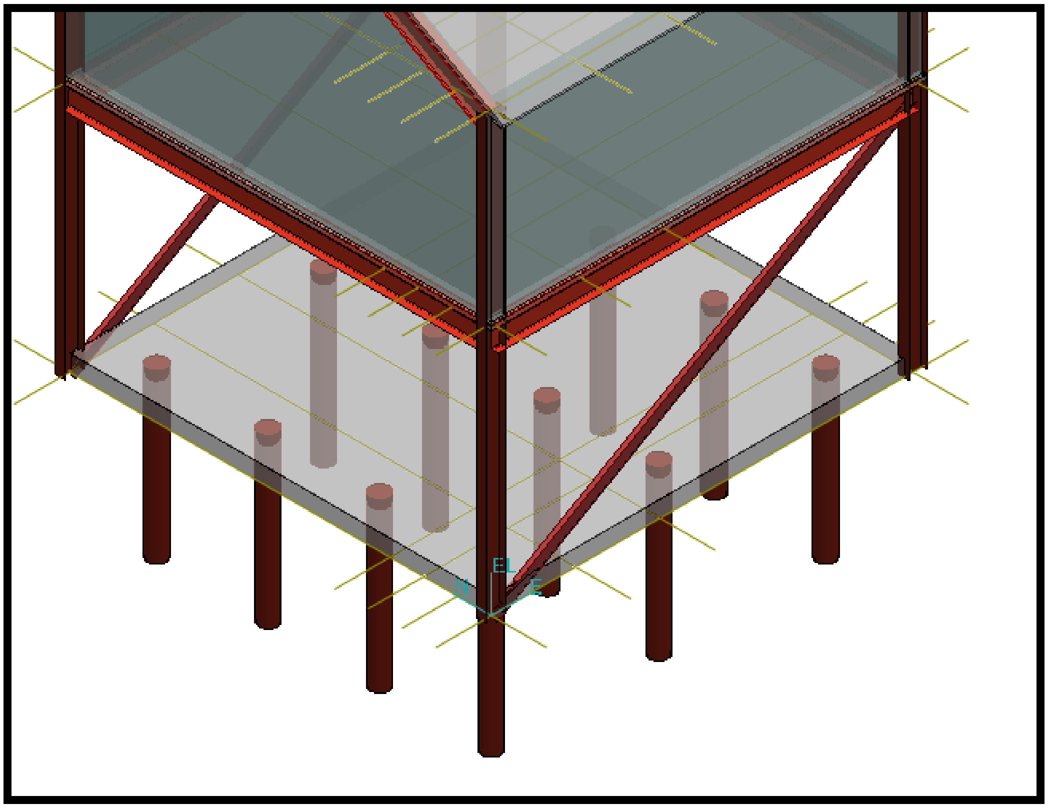
© Copyright 2014 Intergraph Corporation

- Minimum edge clearance: 2 ft
- Embedment depth: 6 in
- Pile depth: 10 ft
- Name Rule: Default NameRule
- Cross Section: CS14 (from Shapes\Misc\CS)
- Material: Concrete
- Grade: Fc3000
- Type Category: Column
- Type: Column

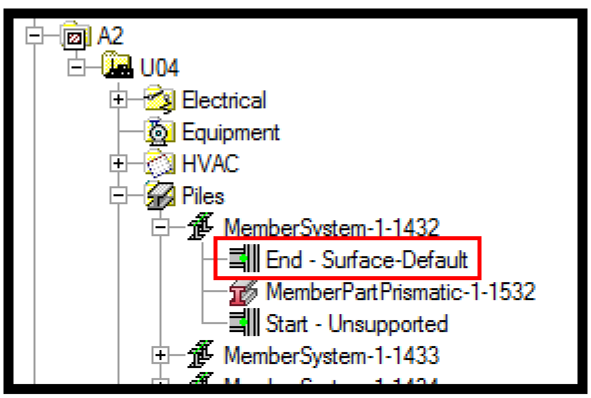


13. Click the Preview button to review the Piles prior to placement.
14. Click OK to place the Pile member systems in the Piles system folder. Review the piles in the Plan view and other orthographic views.

© Copyright 2014 Intergraph Corporation



15. Expand a Pile member system to review the frame connections. Note that Surface-Default connections are used to connect the pile members to the slab surface.



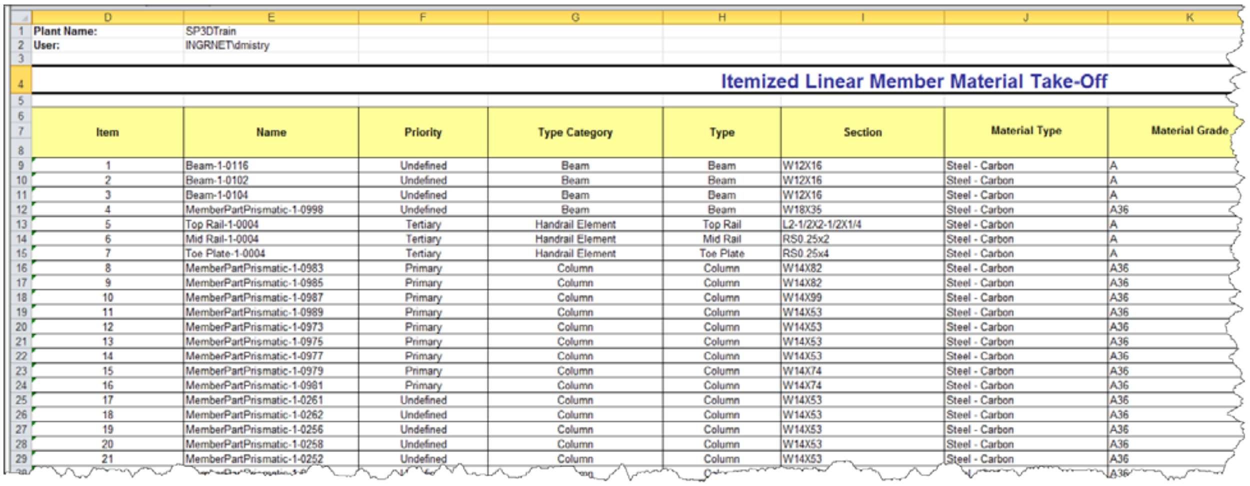
Note:

This custom command is simply a placement wizard to help the user place the pile members quickly based on the specified array. Piles can be placed using the surfaces of Slabs, Footings and Foundations.

© Copyright 2014 Intergraph Corporation

LAB-18: Reports

Extract a Linear Member material list from the model.

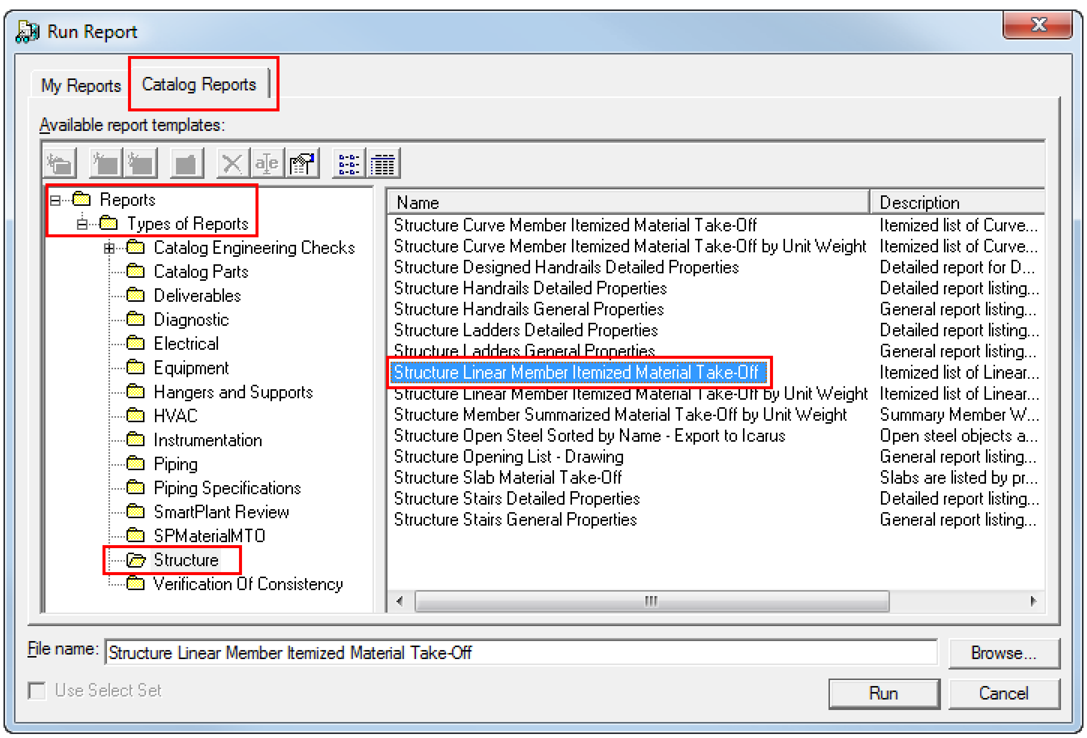


Plant Name: SP3DTrain
User: INGRNET\dmistry

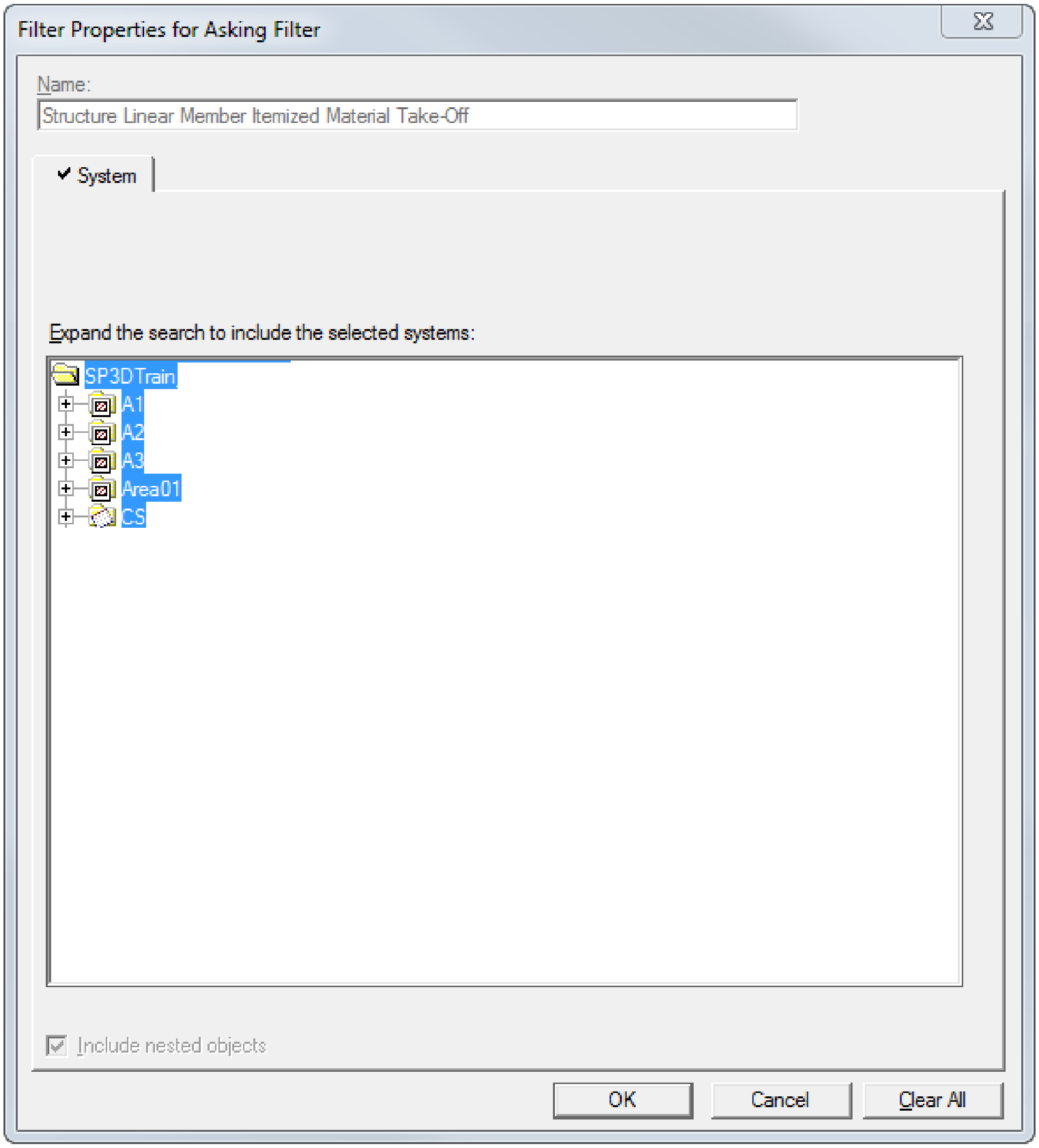
Itemized Linear Member Material Take-Off

Item	Name	Priority	Type Category	Type	Section	Material Type	Material Grade
1	Beam-1-0116	Undefined	Beam	Beam	W12X16	Steel - Carbon	A
2	Beam-1-0102	Undefined	Beam	Beam	W12X16	Steel - Carbon	A
3	Beam-1-0104	Undefined	Beam	Beam	W12X16	Steel - Carbon	A
4	MemberPartPrismatic-1-0998	Undefined	Beam	Beam	W18X35	Steel - Carbon	A36
5	Top Rail-1-0004	Tertiary	Handrail Element	Top Rail	L2-1/2X2-1/2X1/4	Steel - Carbon	A
6	Mid Rail-1-0004	Tertiary	Handrail Element	Mid Rail	RS0.25x2	Steel - Carbon	A
7	Toe Plate-1-0004	Tertiary	Handrail Element	Toe Plate	RS0.25x4	Steel - Carbon	A
8	MemberPartPrismatic-1-0983	Primary	Column	Column	W14X82	Steel - Carbon	A36
9	MemberPartPrismatic-1-0985	Primary	Column	Column	W14X82	Steel - Carbon	A36
10	MemberPartPrismatic-1-0987	Primary	Column	Column	W14X99	Steel - Carbon	A36
11	MemberPartPrismatic-1-0989	Primary	Column	Column	W14X53	Steel - Carbon	A36
12	MemberPartPrismatic-1-0973	Primary	Column	Column	W14X53	Steel - Carbon	A36
13	MemberPartPrismatic-1-0975	Primary	Column	Column	W14X53	Steel - Carbon	A36
14	MemberPartPrismatic-1-0977	Primary	Column	Column	W14X53	Steel - Carbon	A36
15	MemberPartPrismatic-1-0979	Primary	Column	Column	W14X74	Steel - Carbon	A36
16	MemberPartPrismatic-1-0981	Primary	Column	Column	W14X74	Steel - Carbon	A36
17	MemberPartPrismatic-1-0261	Undefined	Column	Column	W14X53	Steel - Carbon	A36
18	MemberPartPrismatic-1-0262	Undefined	Column	Column	W14X53	Steel - Carbon	A36
19	MemberPartPrismatic-1-0256	Undefined	Column	Column	W14X53	Steel - Carbon	A36
20	MemberPartPrismatic-1-0258	Undefined	Column	Column	W14X53	Steel - Carbon	A36
21	MemberPartPrismatic-1-0252	Undefined	Column	Column	W14X53	Steel - Carbon	A36

1. In Structure Task, select Tools →Run Report to extract reports.
   From the Run Report interface, select the Catalog Reports interface and navigate to the Structure report types to use standard delivered report formats.



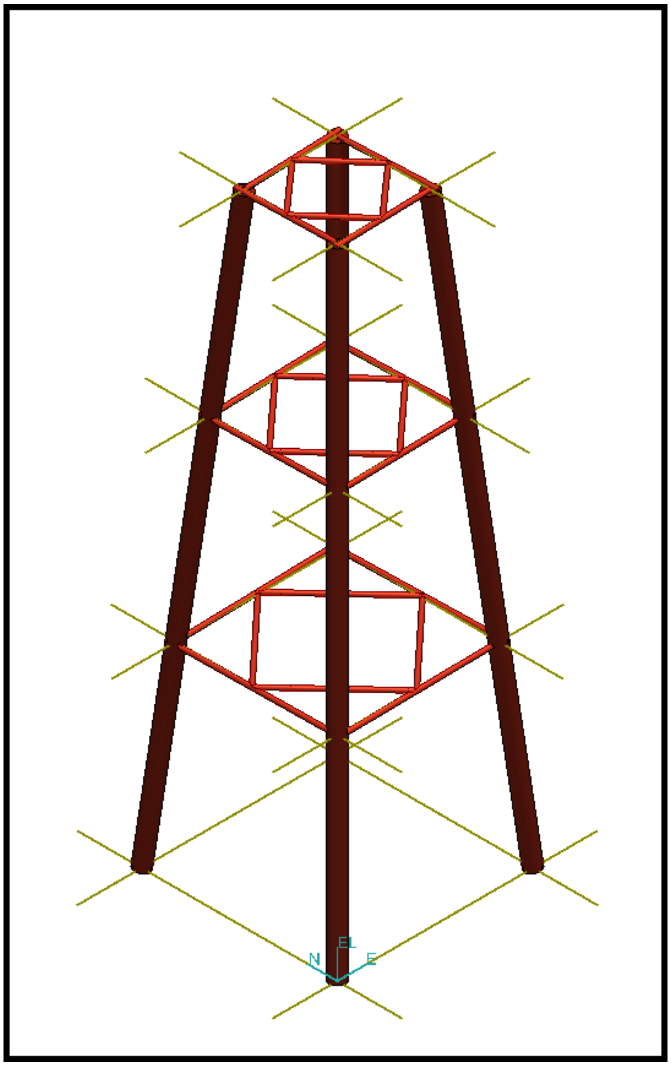
2. Select "Run" button.
3. When prompted. Select the SP3DTrain on the system tab of the asking filter.



4. Click Finish. The report will open after the execution is complete.

© Copyright 2014 Intergraph Corporation

LAB-19: Structural Modeling (offshore Jacket) - Optional



Part I: Jacket Pipes

1 Define your workspace to display Unit **U05** and the coordinate system **U05 CS**.

2 Go to the Structure Task environment. Make sure the Active Permission Group is set to *Structural.*

3 Select Place Linear Member System Command. Use the views shown in Figure to place the appropriate support columns and beams.

4 Set the active member parameters as follows:

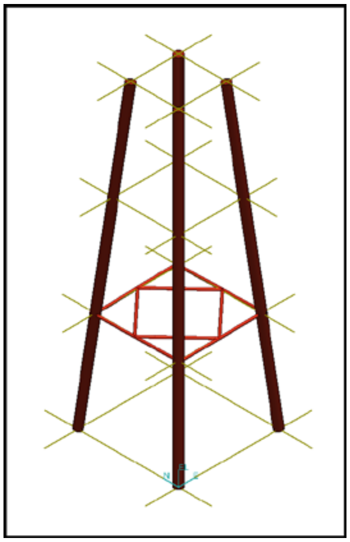
© Copyright 2014 Intergraph Corporation

Connection:	By Rule
System:	A2 → U05 → Structural → Column
Type Category:	Column
Type:	Column
Section Name:	CS10
Cardinal point:	5
Angle:	0 deg

5. Place the first point for column end at the grid intersections Elevation 0'- 0".
6. Place the second column end at the grid intersections Elevation 30'- 0".
7. Repeat the above steps to place the other support columns. Toggle the start/end icons to set the discrete/contiguous placement method.
8. Place the Beams at elevation 10'- 0".
9. Set the active member parameters as follows:

Frame Connection:	Flush-Right for the perimeter pipe and Flush-Top for the intermediate pipe
System:	A2 → U05 → Structural → Beams
Type Category:	Beam
Type:	Beam
Section Name:	CS3
Cardinal point:	5
Angle:	0 deg

Your View should now resemble the following graphic:

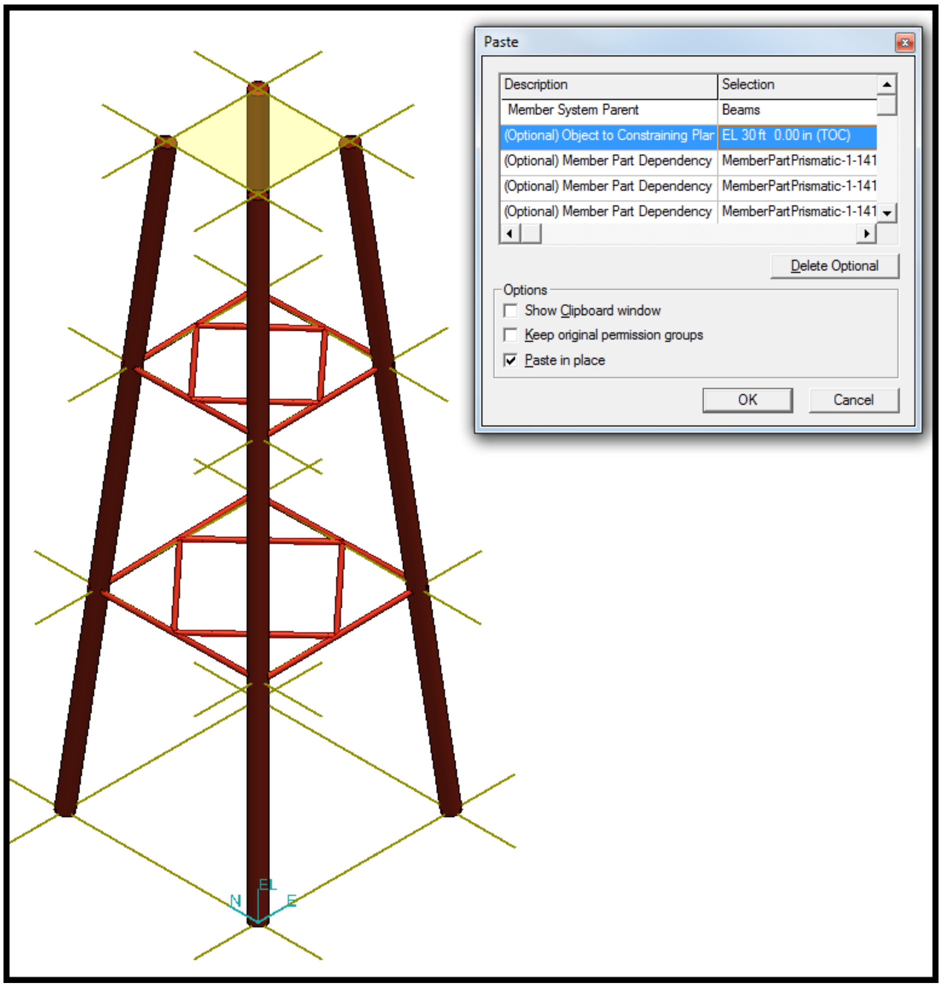


10. Select the Frame connections of the intermediate beams and set the Position Rule to Ratio.
11. Use Copy/Paste functionality to place the other steel frames on the other EL 20' and 30'.

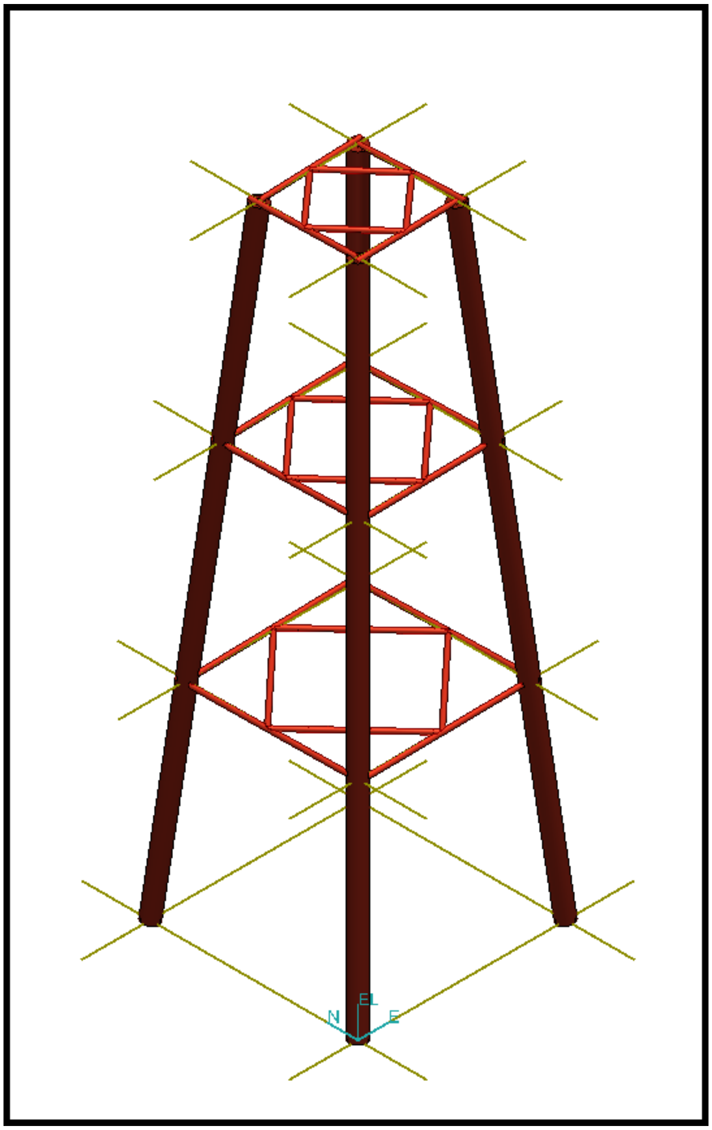
Note: Make sure to select the appropriate objects to re-establish the connection in the Paste Dialog box.

Hint: *Set the select filter to Member Systems to select 8 members.*
Only 1 related object is needed to re-establish the connection (EL plane).

12. Use the Paste command three times.

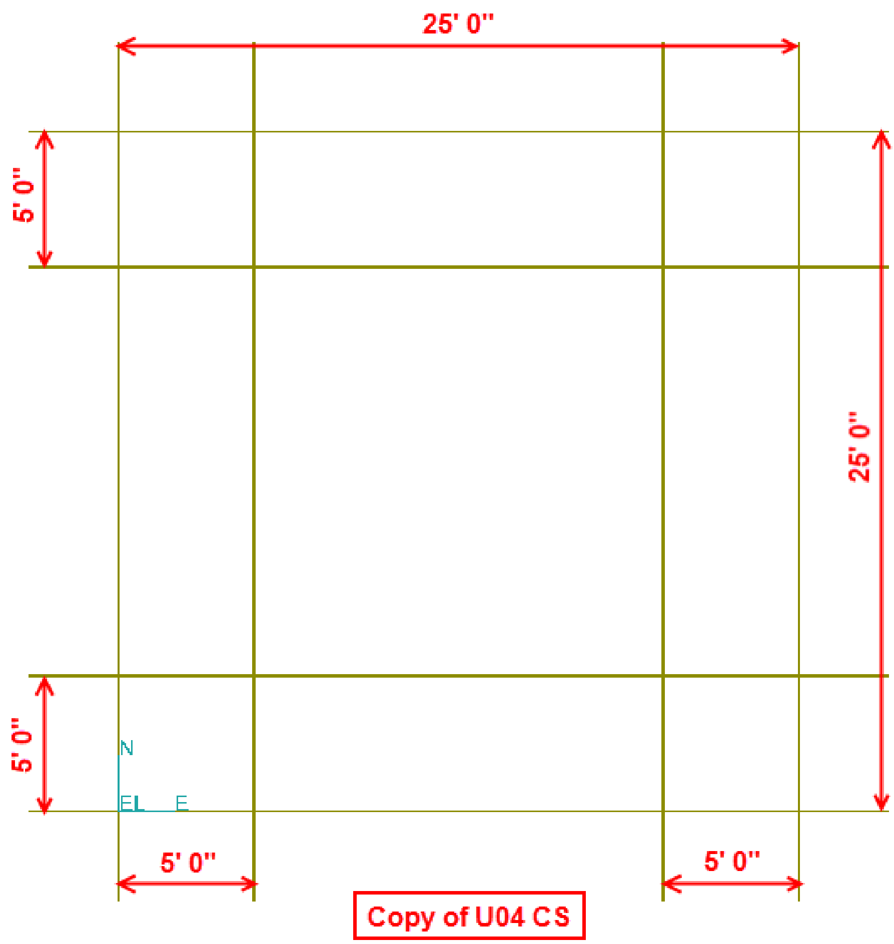


Your View should now resemble the following graphic:

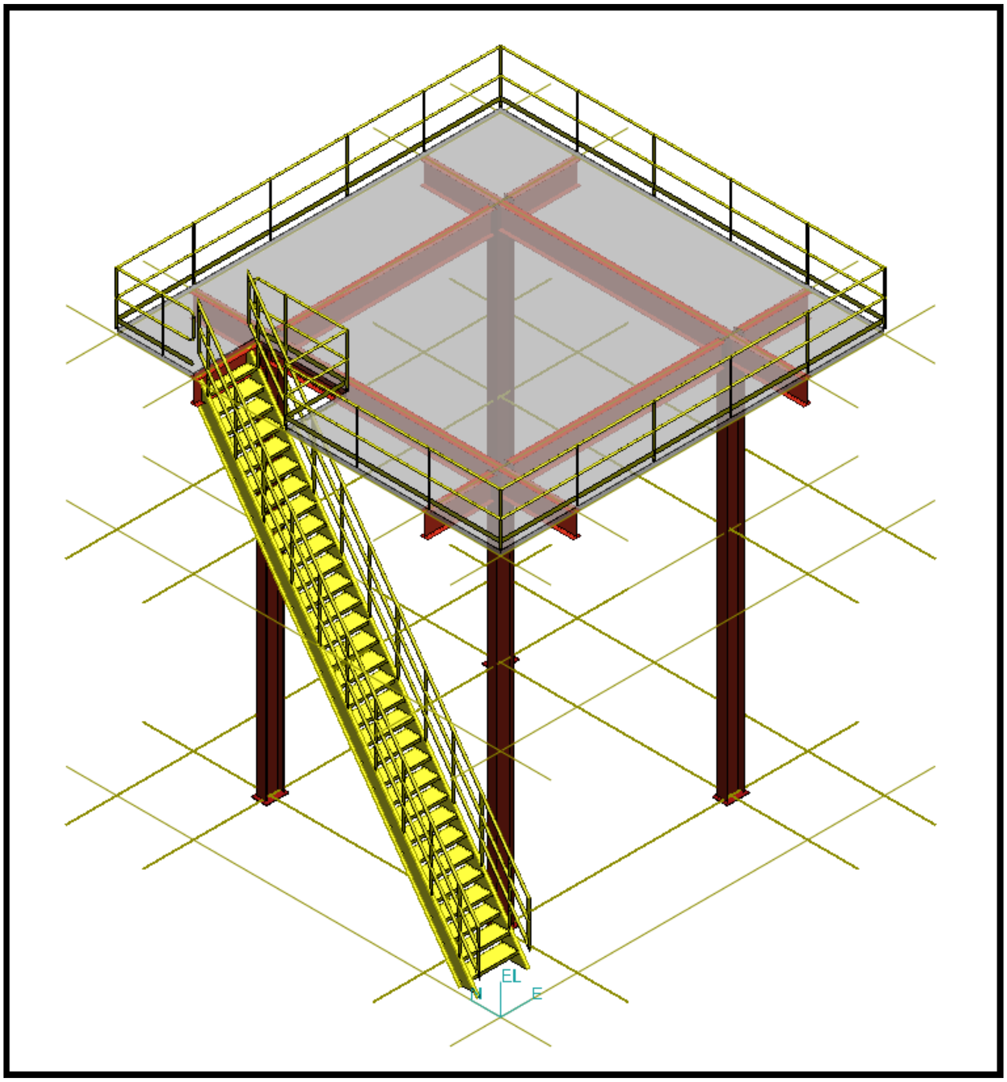

EL
N
E

Part II: Rotated Coordinate System – Optional

- Use the commands you have learned to model the following:
- Use Copy/Paste functionality to make a copy of the U04 Coordinate system and place it at Easting 120'-0", Northing 150'-0" Elevation 0'-0".
- Modify/delete the grid planes under the Copy of U04 CS to achieve the results shown below;



- Place members, slab and stair as shown in the figure below using the following configurations;



Columns:

Connection:	By Rule
Type Category:	Column
Type:	Column
Section Name:	W14x53
Cardinal point:	5
Angle:	20
System:	A2 → U04 → Structural → Columns

Beams:

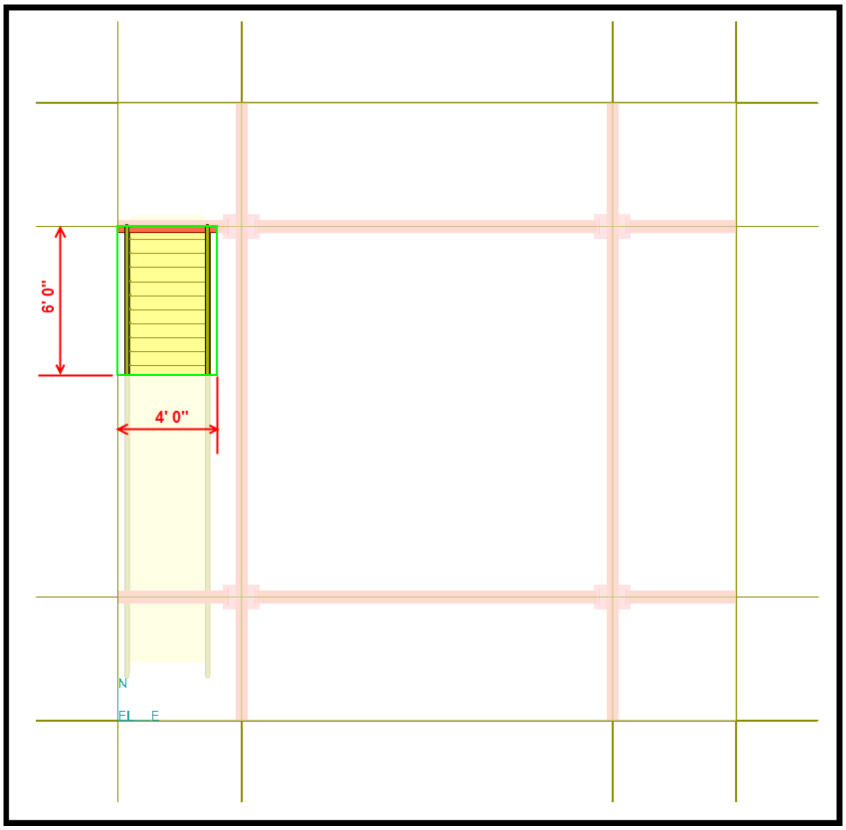
Connection:	By Rule
Type Category:	Beam
Type:	Beam
Section Name:	W18X40
Cardinal point:	8
Angle:	0
System:	A2 → U04 → Structural → Beams

© Copyright 2014 Intergraph Corporation

Beam Extension Length: 5’ (TYP)

Stair:

Type:	StairA1
Width:	3’
Angle:	55
Horizontal Offset	3’
System:	A2 → U04 → Structural → Miscellaneous



© Copyright 2014 Intergraph Corporation

LAB-20: Using Building Wizard (Optional)

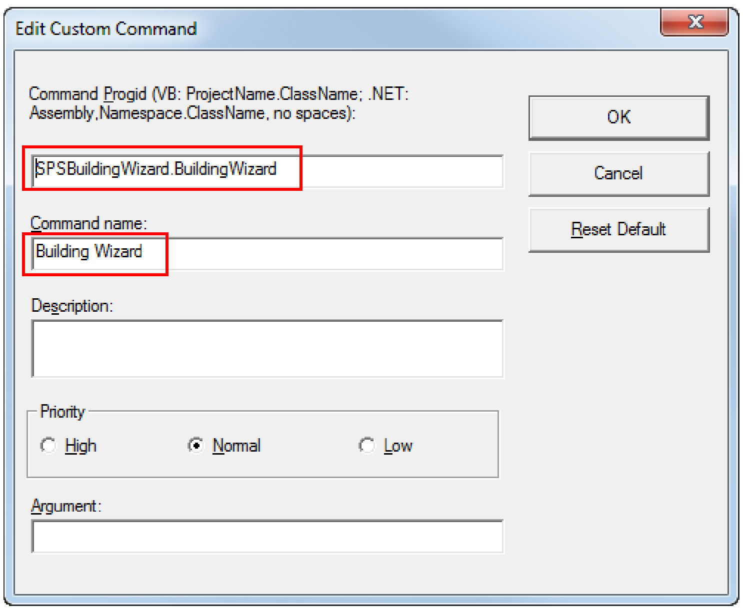
Objective:

- Create Building_2 at -500,10, 0 using the building wizard

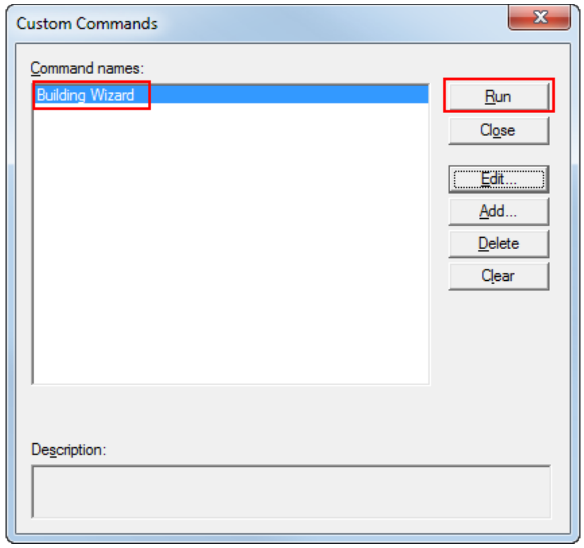
Add custom command

Add a custom command for the building wizard as follows:

1. Select Tools -> Custom Commands
2. Click “Add” button
3. Enter values as shown:
   Command ProgID: ***SPSBuildingWizard.BuildingWizard***
   Command Name: Building Wizard



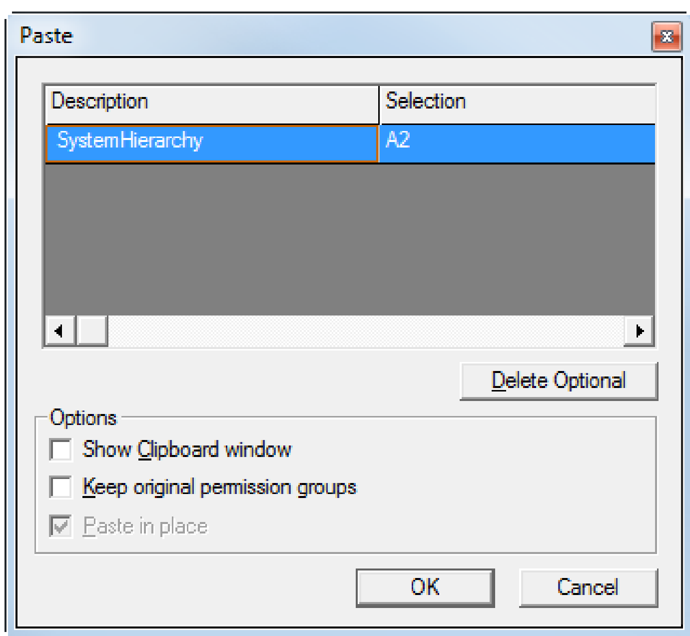
4. Click “OK” Button.



5. Click “Close” Button.

<u>Create the systems required</u>

1. Switch to Systems and Specifications Task
2. Expand A2 in the system hierarchy.
3. Right click U10 to open the popup menu. Click on the select nested option.
4. Select Edit -> Copy.
5. Select Edit -> Paste and click “OK” Button.



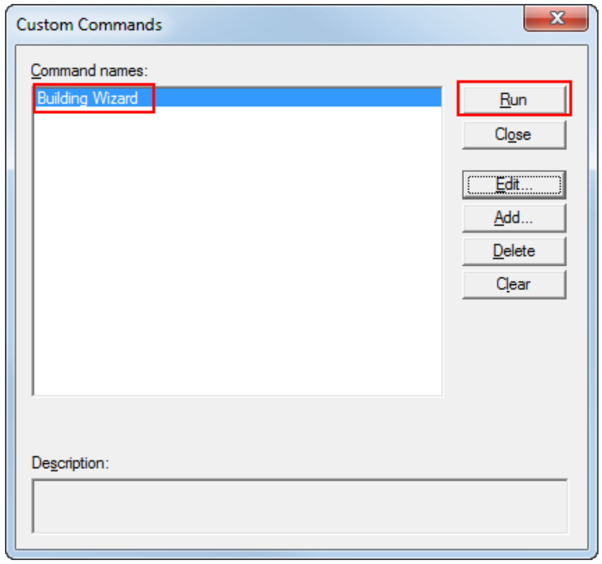
7. Select copied system “U10(2)”. Go to the properties and rename the system to U20.

<u>Run the building wizard</u>

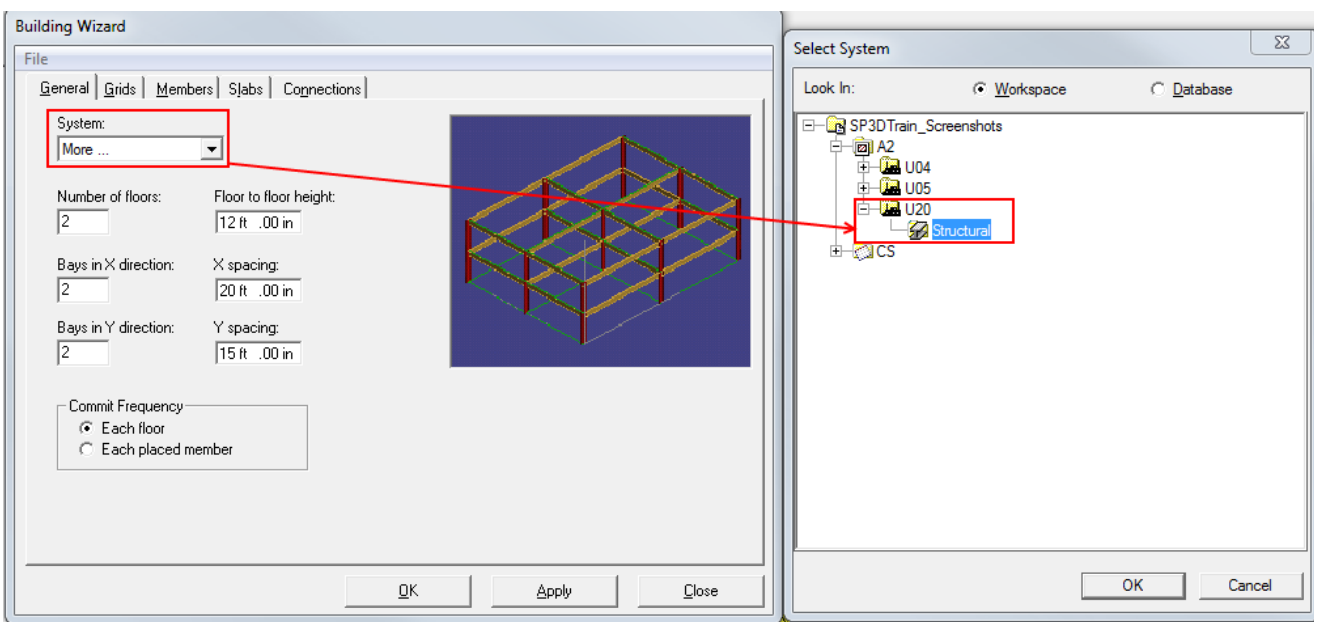
1. Switch to the Structure Task.

© Copyright 2014 Intergraph Corporation

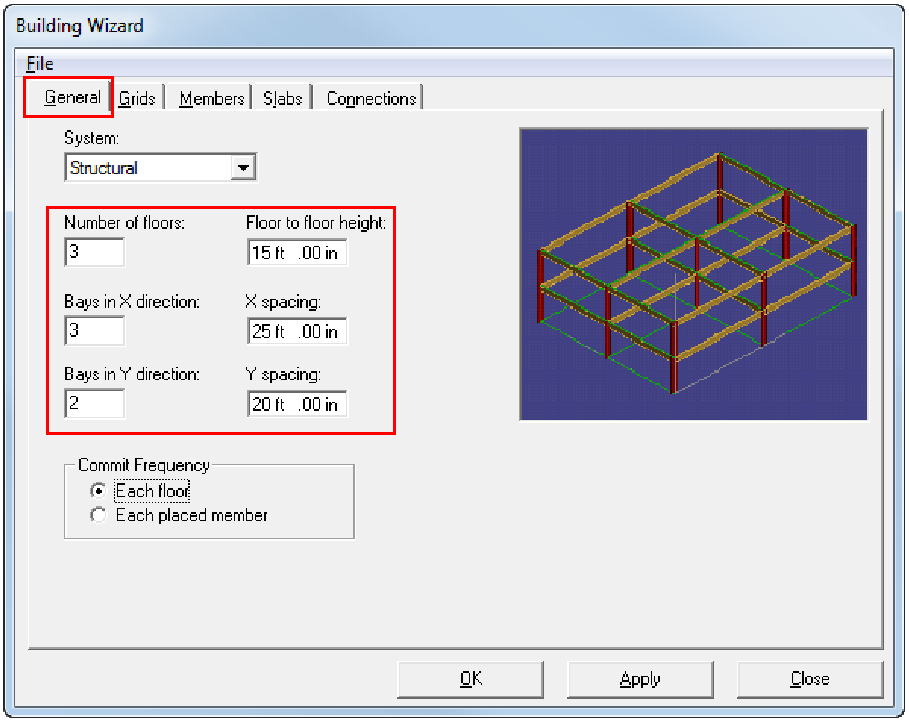
2. Select Tools ->Custom Commands.



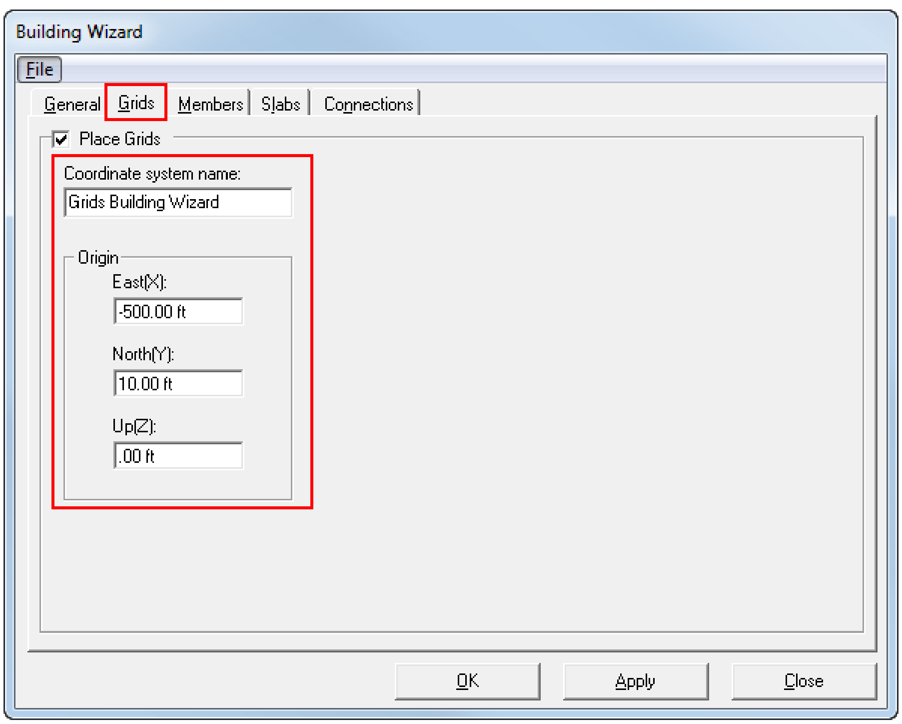
3. Select the Building Wizard.
4. Click "Run" button.
   The building wizard opens up.
5. In the System field, select More…, then pick the A2 -> U20 -> Structural



6. On the General Tab, key-ins the following:

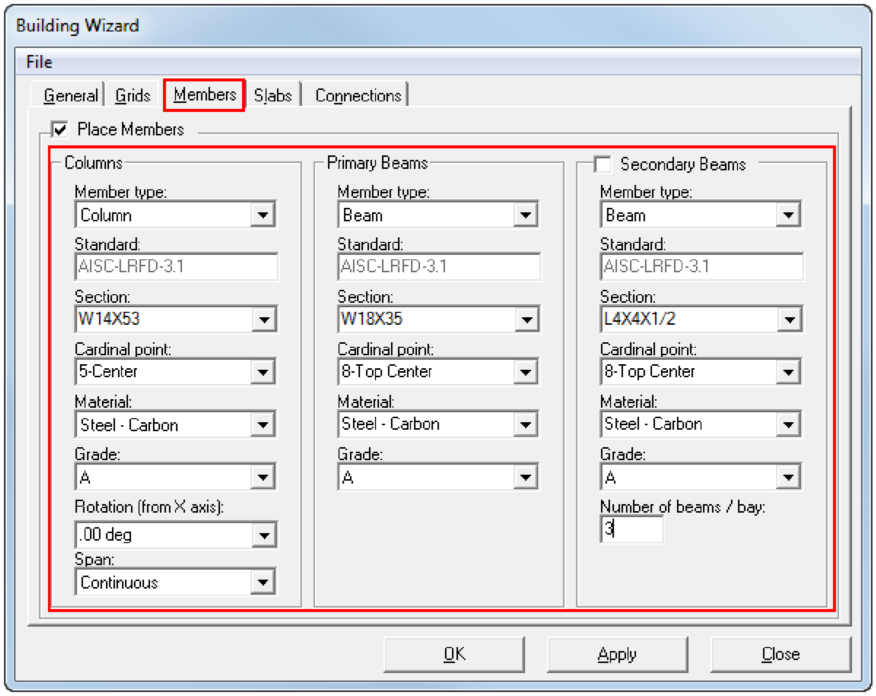


7. On the Grids tab, key-ins the followings:

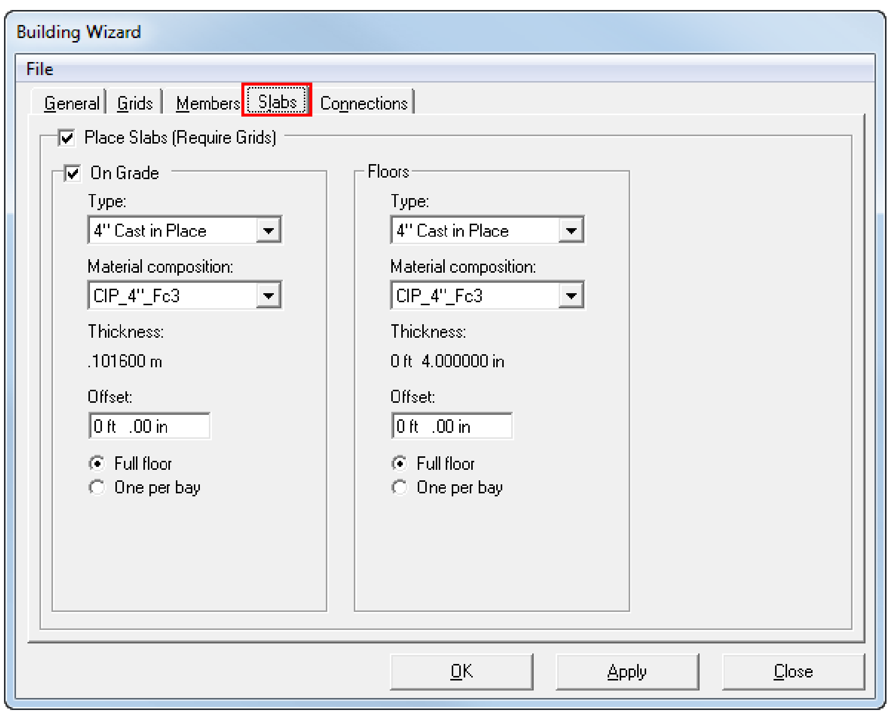


8. On the Members tab, key-ins the followings:

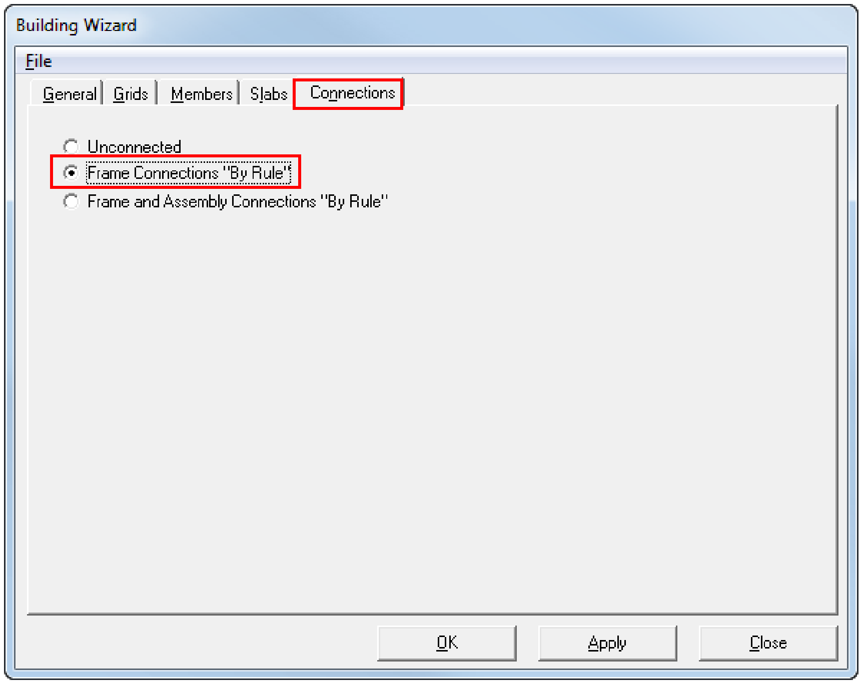
© Copyright 2014 Intergraph Corporation



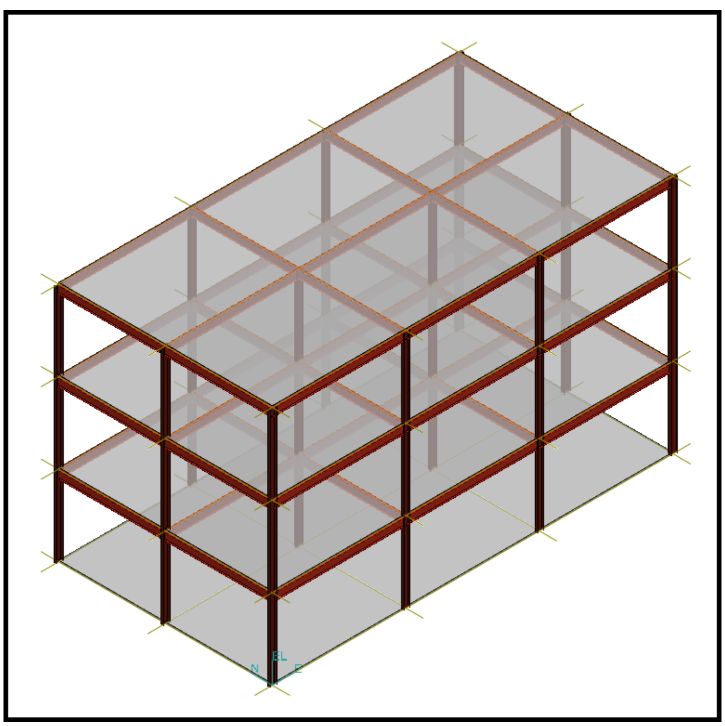
9. On the Slabs tab. Keep the defaults.



10. On the Connections tab, select 'Frame connections by rule'.



11. Click "OK" Button.
    Your View should now resemble the following graphic:



Designed Solids

Prerequisite Sessions

- SP3D Overview
- SP3D Common Sessions

Overview:

The place designed solids command is used to create customized solids for concrete in structure modeling and for equipment modeling.

Solids will enable the user to use Add, Subtract and Suppress Boolean operations that will assist in creating complex shapes. Designed equipment command should be used to place the designed solid in the model. Multiple designed solids can be placed under single designed equipment. Multiple designed equipments also can be placed under single designed equipment.

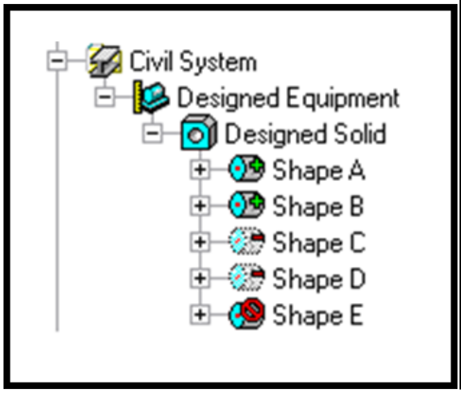
A shape must be a child of the designed solid in order for the software to compute the weight and center of gravity. Weight and center of gravity of the designed solid is the sum of all the shapes underneath the designed solid. Order Shapes controls the order in which the designed solid's shapes are processed by the software, which can be very important when a shape that cuts material from the designed solid overlaps a shape that adds material to the designed solid. The designed solid could look very different depending on which shape, the cut or the add, the software processes last. This dialog box is activated by Operators List  on the Modify Designed Solid ribbon

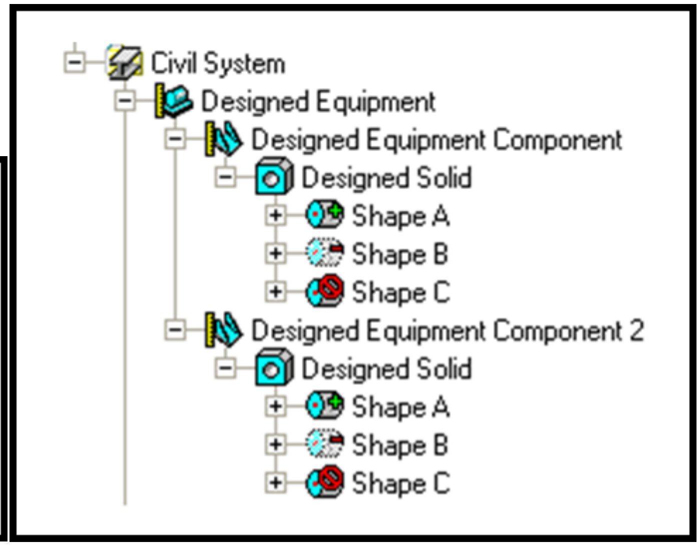
Theory

Solids are basically a container for a collection of Shapes. Solids have Surface Area and Volume properties, which will allow users to compute Weight and Center of Gravity (CG). However, in order to compute the Weight and CG, a material must be defined on the Occurrence tab of the Solid. Material density is **required** to compute the weight of the Solid. The density value for a material can be extracted from the catalog when the user supplies a Material Type and a Material Grade. (Note: Material properties only apply to *Designed Solids* placed in the “Simple physical” or “Detailed physical” aspects.)

To place a Solid, a Designed Equipment or a Designed Equipment Component must first be placed in the model. Then, the Solid can be placed as a child of the Designed Equipment or the Designed Equipment Component. At this point, Shapes can be added underneath the Solid. (Note: You can have multiple Solids under one Designed Equipment or Designed Equipment Component. You can also have multiple Designed Equipment Components under one Designed Equipment).

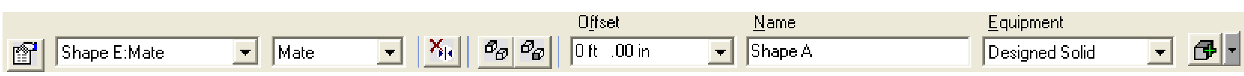
© Copyright 2014 Intergraph Corporation

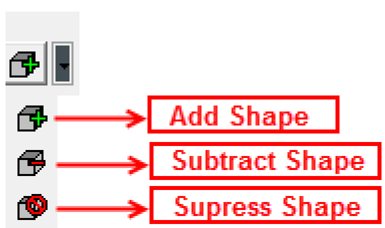




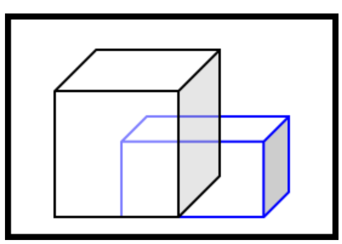
A Shape **must** be a child of a Solid in order to compute Weight and CG. The Weight and CG properties will be summed up for all Shapes underneath a Solid for a total Weight and CG of the Solid itself. For example, *Shape A* + *Shape B* + *Shape C* + *Shape D* + *Shape E* = Total Surface Area, Volume, Weight and CG of the *Designed Solid*. If there is more than one Solid underneath a Designed Equipment parent, these Solids can also be summed up to give a Total Weight and CG for the Designed Equipment. For example, *Solid A* + *Solid B* + … + *Solid X* = Total Weight and CG of the *Designed Equipment*.

The icon shown on the Shape in Workspace Explorer will represent an Add, Subtract or Suppress operation. By default, the operation performed on a Shape at placement time is Add. During placement time or after placement, the user can change the operation to Subtract Shape or Suppress Shape using the horizontal ribbon bar.

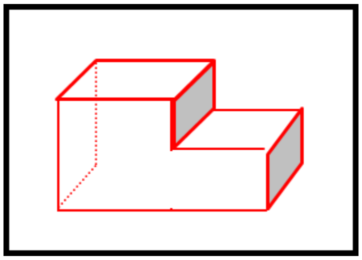




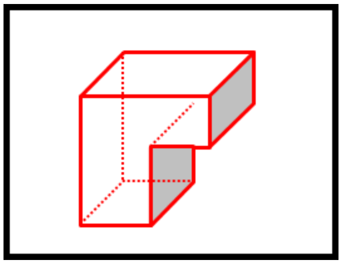
You can think of the *Add Shape* and *Subtract Shape* operations as Boolean operations. For example, see *A* and *B* below:



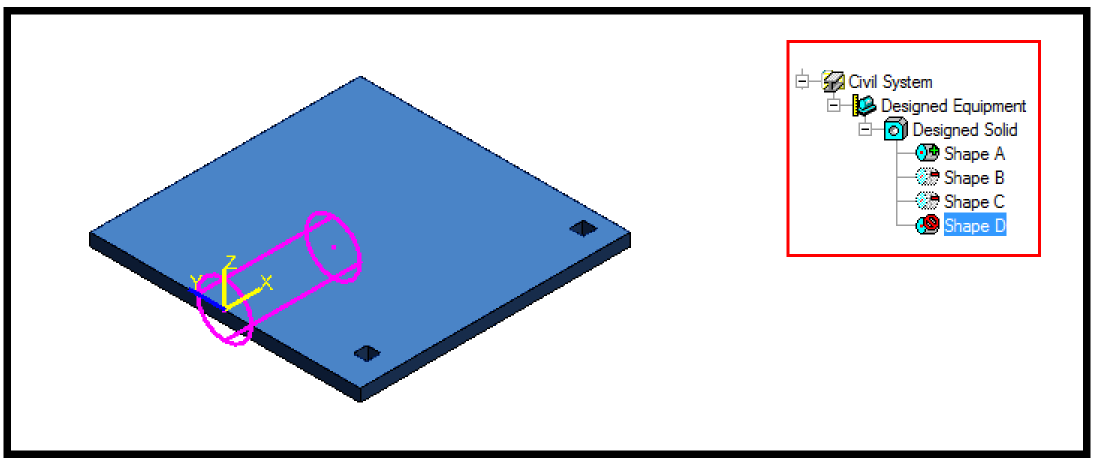
If we were to Add Shape *B* to Shape *A*, the result would be merging the two shapes into one as shown below:



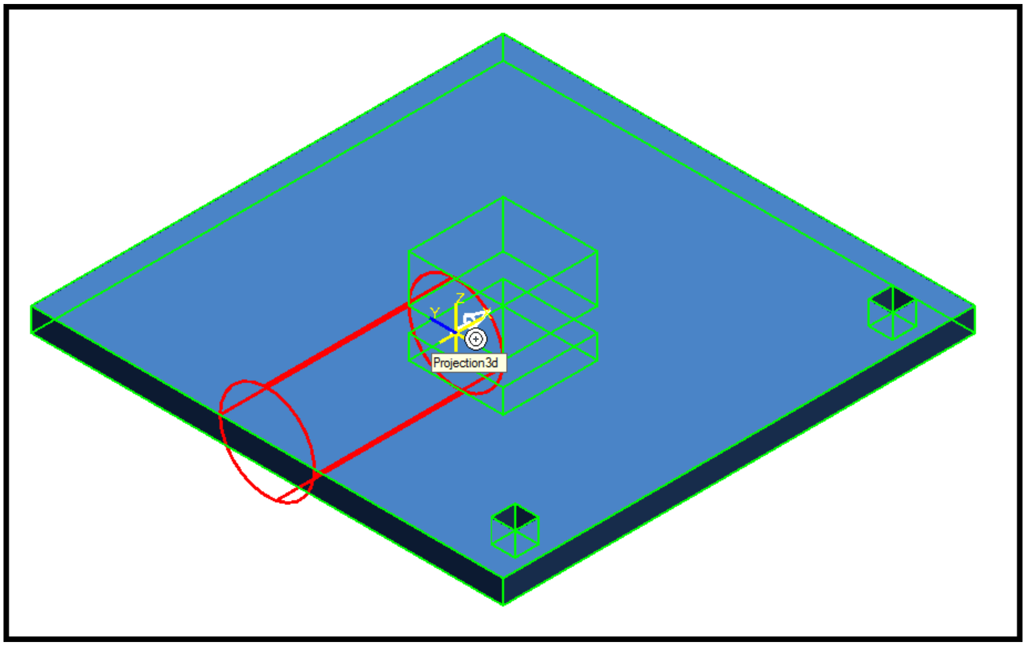
Similarly, if we were to Subtract Shape *B* from Shape *A*, the result would be removing the cross-section of Shape *B* from Shape *A* as shown below:



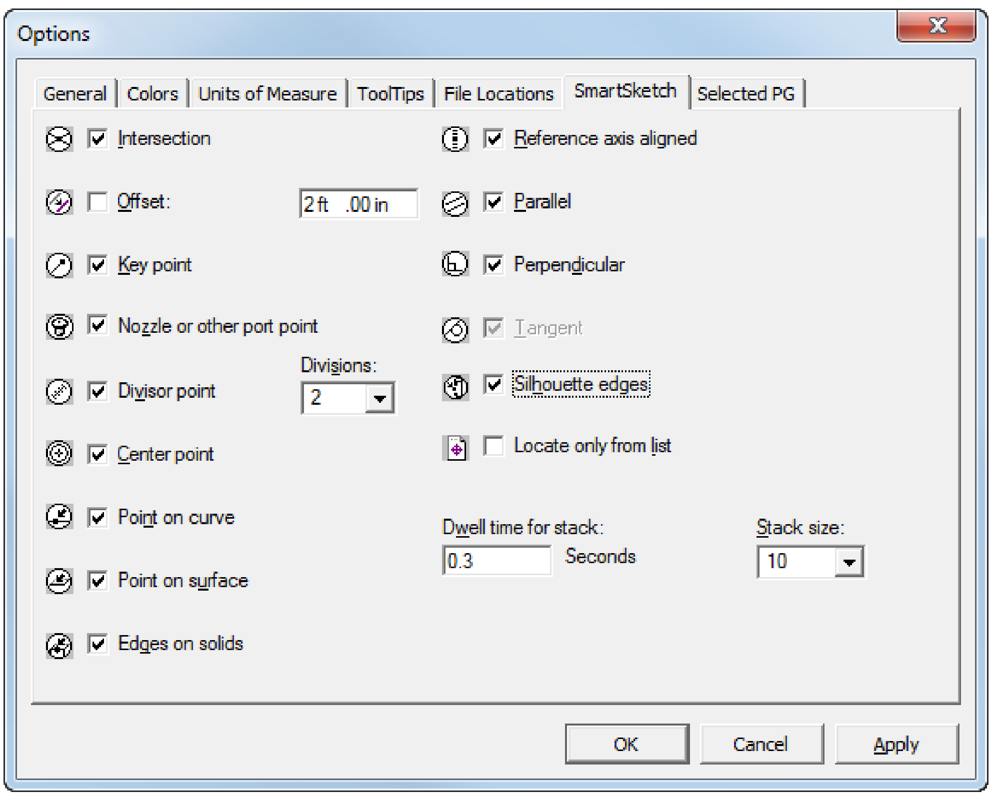
The *Suppress Shape* operation is unique from the *Add Shape* and *Subtract Shape* operations. This operation is used for creating construction geometries for reference when placing other Shapes. The Shape itself is not included in the Solid. Therefore, it will not affect Weight and CG calculations. In the screenshot below, the cylinder is suppressed, and it can only be seen when it is selected.



In the next screenshot, the center point of the cylinder is used as a reference for placing the cube.



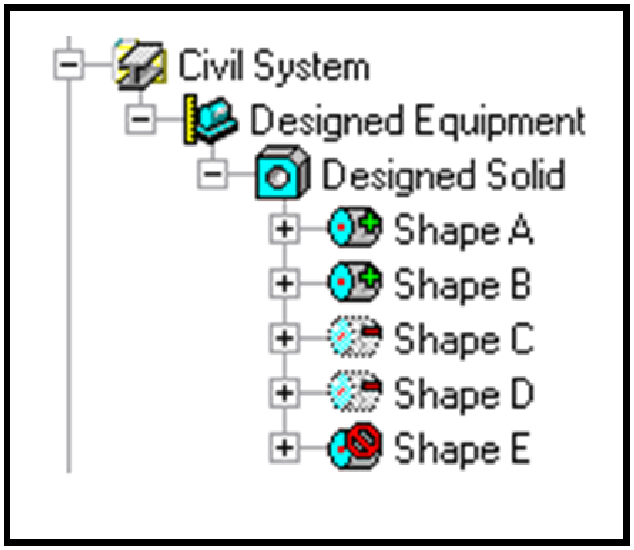
The  glyph that you see is the SmartSketch Center point glyph. There are many more glyphs like this one that are very useful when positioning Shapes. These glyphs can be turned on and off through SmartSketch tab on the Tools -> Options dialog.



Therefore, a Shape that is a child of a Solid is unique. *Add Shape*, *Subtract Shape* and *Suppress Shape* operations can be performed on them, and they are used to calculate Surface Area, Volume, Weight and

© Copyright 2014 Intergraph Corporation

CG of a Solid. In the image shown below *Shape A* + *Shape B* – *Shape C* – *Shape D* = Total Surface Area, Volume, Weight and CG of *Designed Solid.*



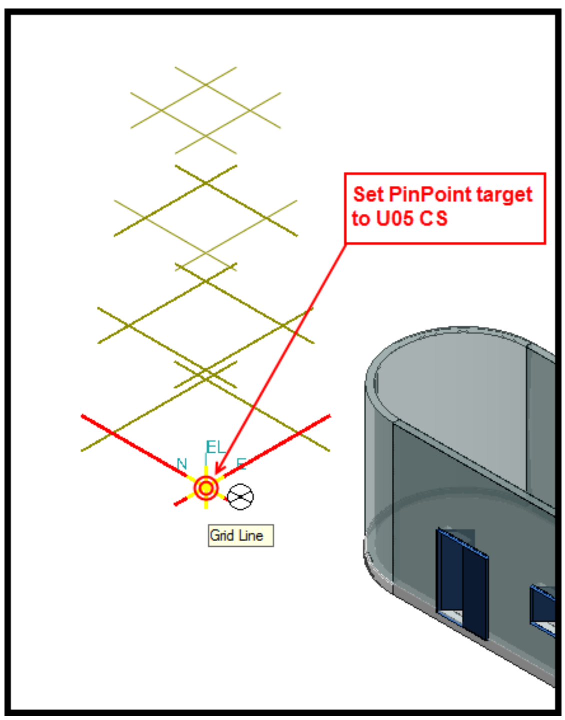
LAB-21: Designed Solids – Stack Tower

Objective

After completing this lab, you will be able to:

- Use Connect relationships to position Shapes, rotate Shapes with the Arrow keys on the keyboard. For this lab, we will create a hierarchy using Designed Equipment Components. Understand and apply relationship. Use Boolean operations (Add, Subtract and Suppress) effectively.

1. Open a session file.
2. Define your workspace to include the A2 → U05 and CS → U05 CS system.
3. Activate the PinPoint toolbar by selecting Tools -> PinPoint or by pressing the PinPoint command in the common toolbar and change the coordinate system from Global to U05 CS in the ribbon bar. If the U05 CS is not shown in the pull down list. Then go to More and select U05CS by expanding the CS. Select the reposition target and select the highlighted U05 CS. Your view should resemble the following graphic.

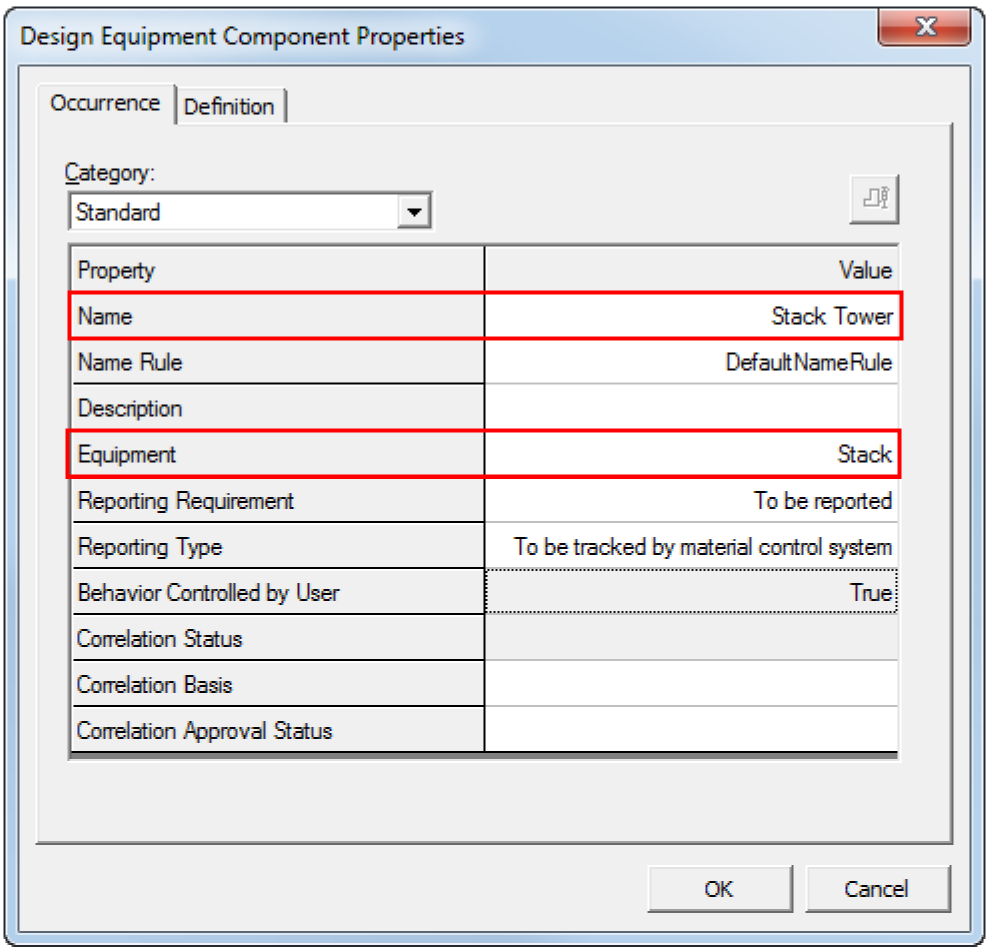


4. Expand the fly-out toolbar and click the  icon on the vertical toolbar to start the Place Designed Equipment command . Expand *Equipment* -> *Civil* and select *Miscellaneous*. Click OK.
5. In the Design Equipment Properties dialog, choose A2 → U05 →Structural →*Solids* as the System and give a User Defined name of *Stack*. Press OK.
6. Place the Designed Equipment at E: -50 ft, N: -90 ft and EL: 0 ft.

© Copyright 2014 Intergraph Corporation

7. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Equipment Component  icon.
8. Select *Stack* as the parent.
9. In the Select Equipment Component type dialog, choose Civil Components → Miscellaneous.

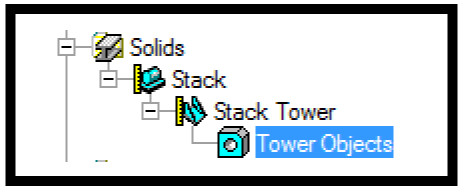
   The type of the Component does not matter. This dialog can be customized through the catalog to suit your needs. For example, expand \Equipment Components\Civil equipment and click OK.
10. In the Designed Equipment Component Properties dialog, make sure the Equipment property is set to *Stack*. Name the Component *Stack Tower* and click OK. See image below.



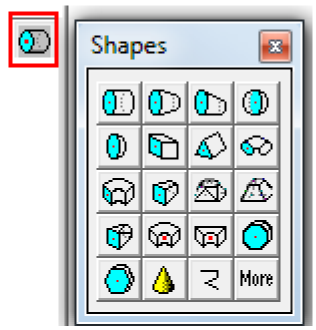
11. Place *Stack Tower* at E: -50 ft, N: -90 ft and EL: 0 ft.
12. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Solid  icon.
13. Choose *Stack Tower* as the parent. Rename the Solid to *Tower Objects*.

© Copyright 2014 Intergraph Corporation

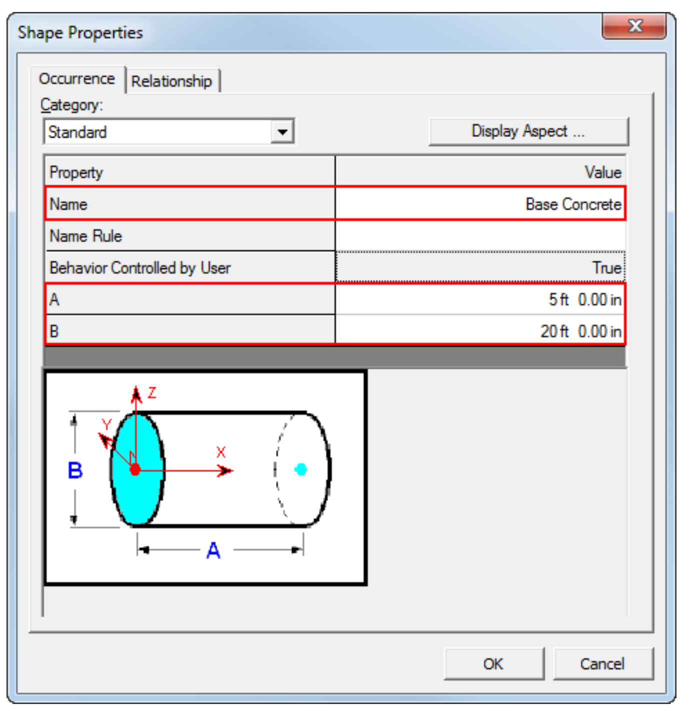
14. Go to Edit → Properties. Properties window will show up. Set the Material Type and Grade to Concrete and Fc 3000 for Solids, *Tower Objects*. Your workspace explorer should resemble the following graphic.



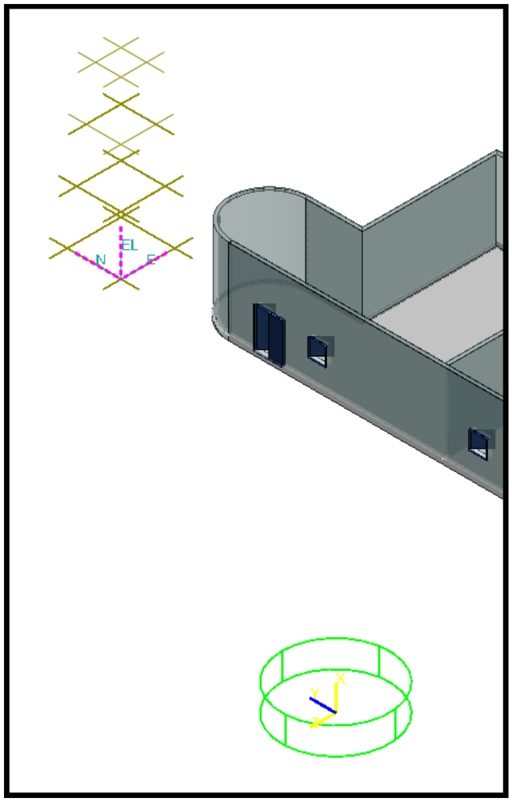
15. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears.



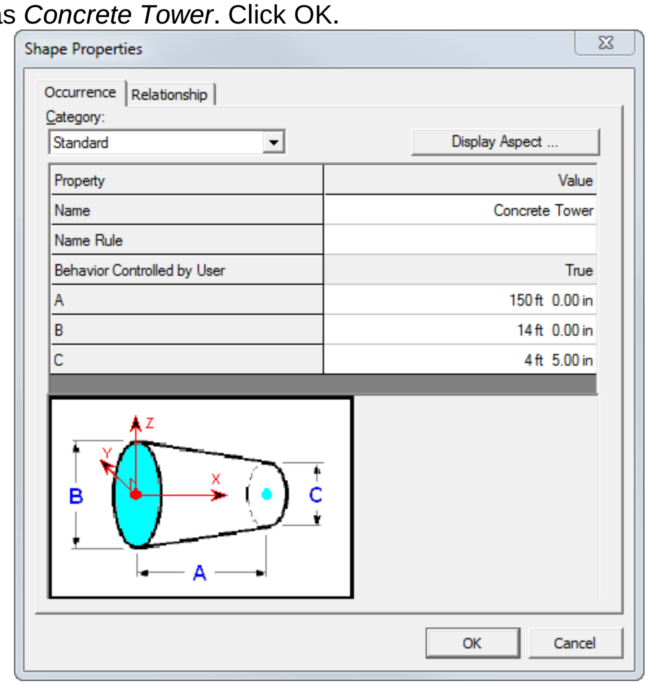
16. Select *RtCircularCylinder 001* from the Shapes dialog. Choose *Tower Objects* from the workspace explorer as the parent of the Shape.
17. In the Shape Properties dialog, set A to 5 ft, B to 20 ft. Name the *RtCircularCylinder 001* as *Base Concrete*. Click OK.



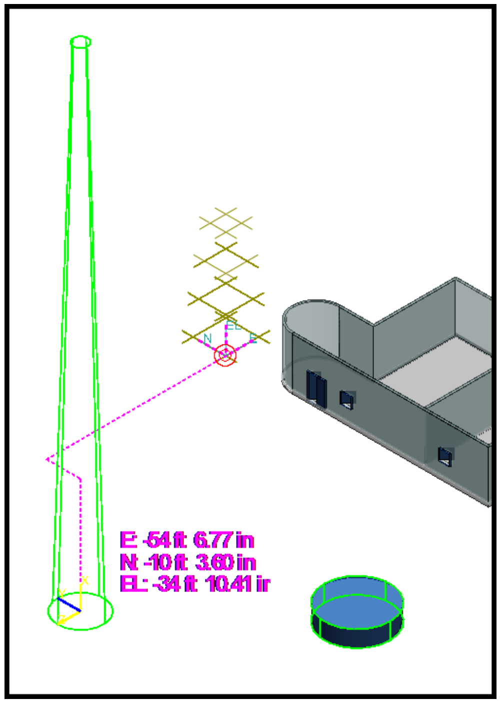
18. Use the Up arrow key to change the axis of rotation and the left/right arrow keys to rotate the shape. Your view should resemble the following graphic.



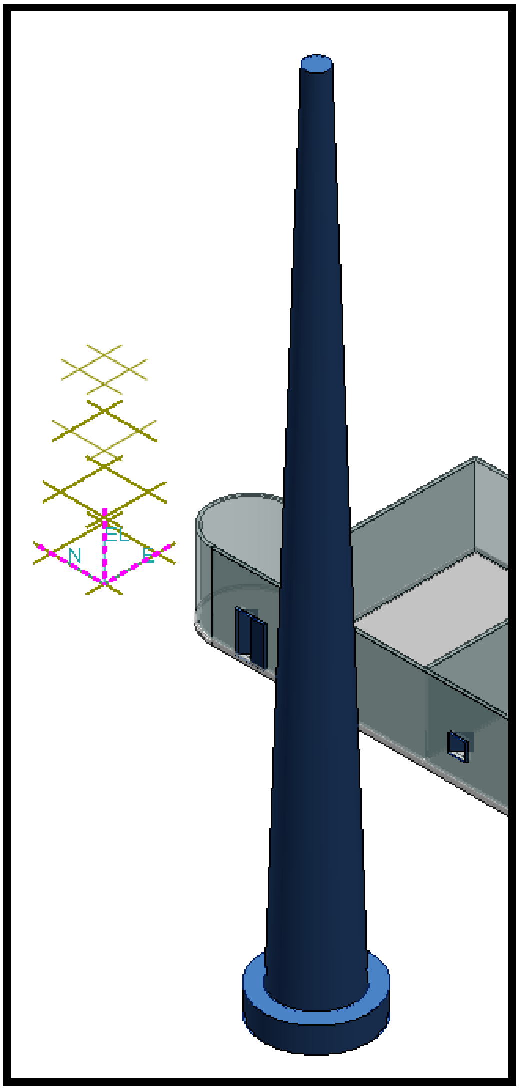
19. Place the Shape at E: -50 ft, N: -90 ft and EL: 0 ft.
20. Select *RtCircularCone001* from the Shapes dialog. Choose *Tower Objects* as the parent of the Shape.
21. In the Shape Properties dialog, set A to 150 ft, B to 14 ft and C to 4ft 5in. Name the *RtCircularCone001* as *Concrete Tower*. Click OK.



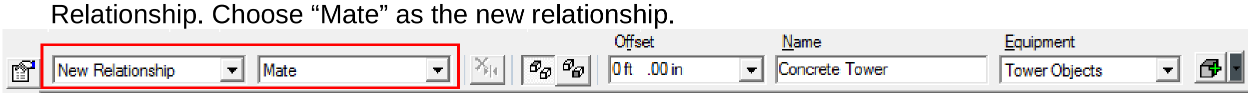
22. Use the Up arrow key to change the axis of rotation and the left/right arrow keys to rotate the shape. Your view should resemble the following graphic.



23. Place the Shape at E: -50 ft, N: -90 ft and EL: 5 ft.
24. Your view should resemble the following graphic.

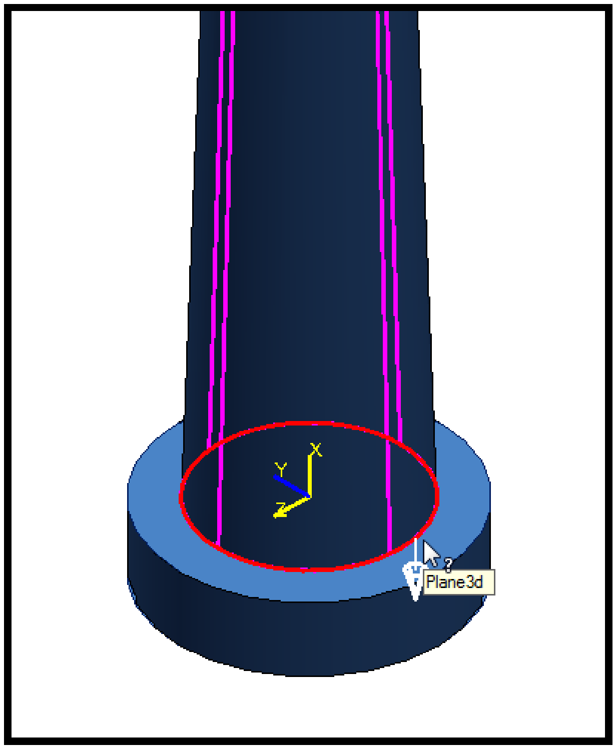


25. Select *Concrete Tower* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Mate” as the new relationship.

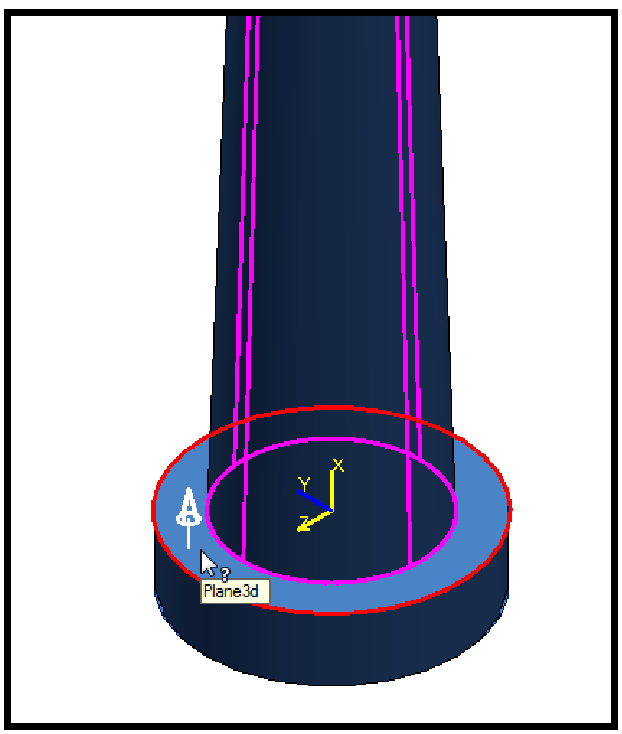


26. Move the mouse over the *Concrete Tower.* Your view should now resemble the following graphic.

© Copyright 2014 Intergraph Corporation

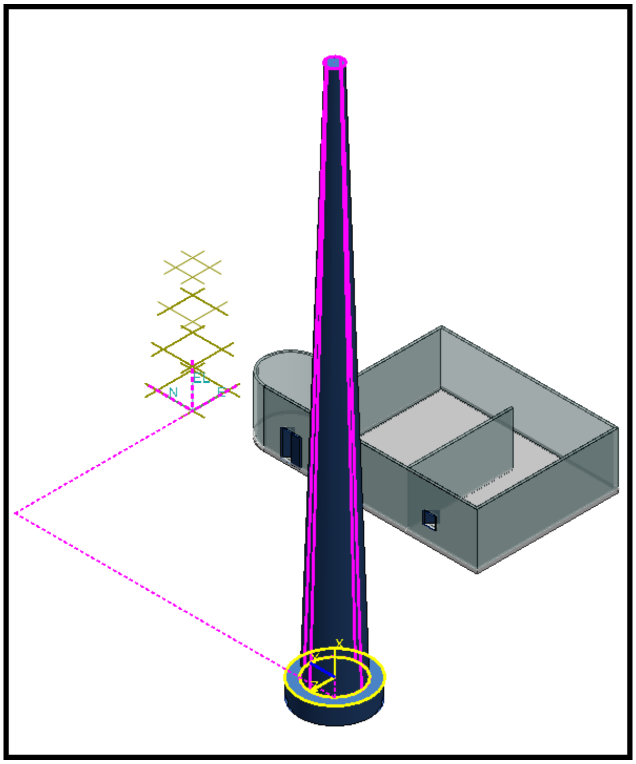


27. Move your mouse over the *Tower Objects* and it should highlight *Base Concrete* and click the left mouse button once at the location as shown in the figure. Your view should now resemble the following graphic.

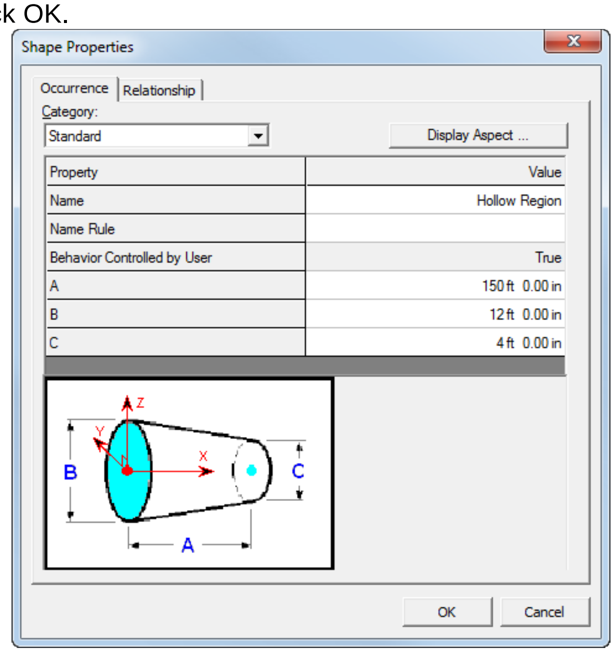


28. After selecting the highlighted plane, Mate Relationship will apply to the *Base Concrete* and *Concrete Tower.* Your view should resemble the following graphic.

© Copyright 2014 Intergraph Corporation

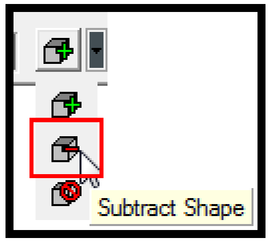


29. Select *RtCircularCone001* from the Shapes dialog. Choose *Tower Objects* as the parent of the Shape.
30. In the Shape Properties dialog, set A to 150 ft, B to 12 ft and C to 4ft. Name the *RtCircularCone001* as *Hollow Region*. Click OK.

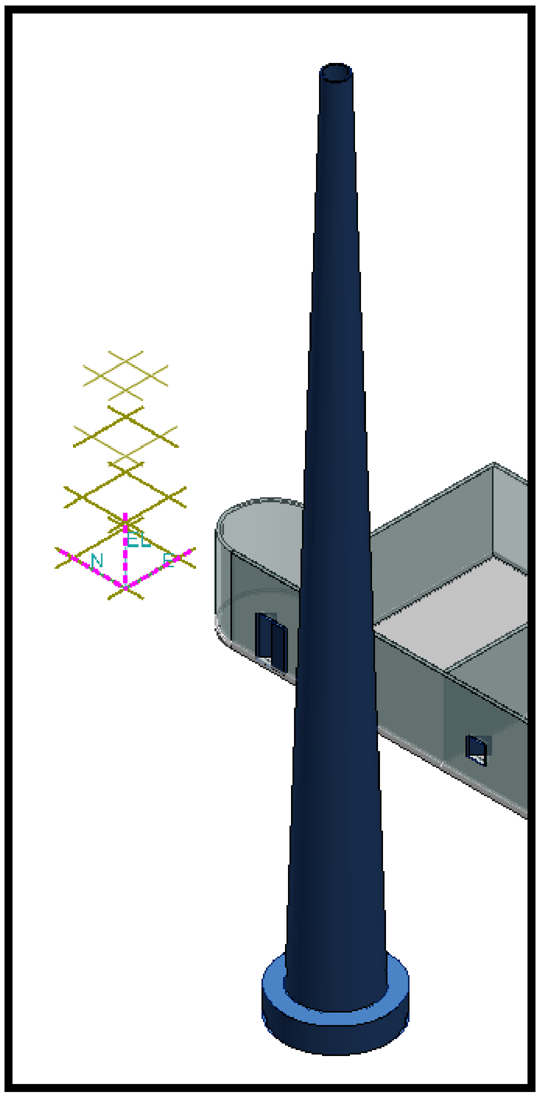


31. Use the Up arrow key to change the axis of rotation and the left/right arrow keys to rotate the shape.
32. Place the Shape at E: -50 ft, N: -90 ft and EL: 5ft.

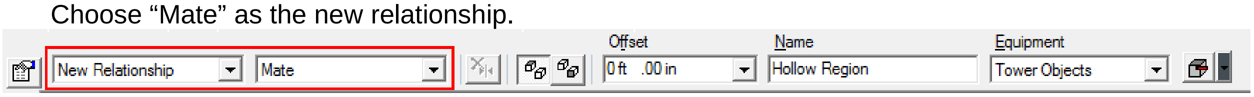
33. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.



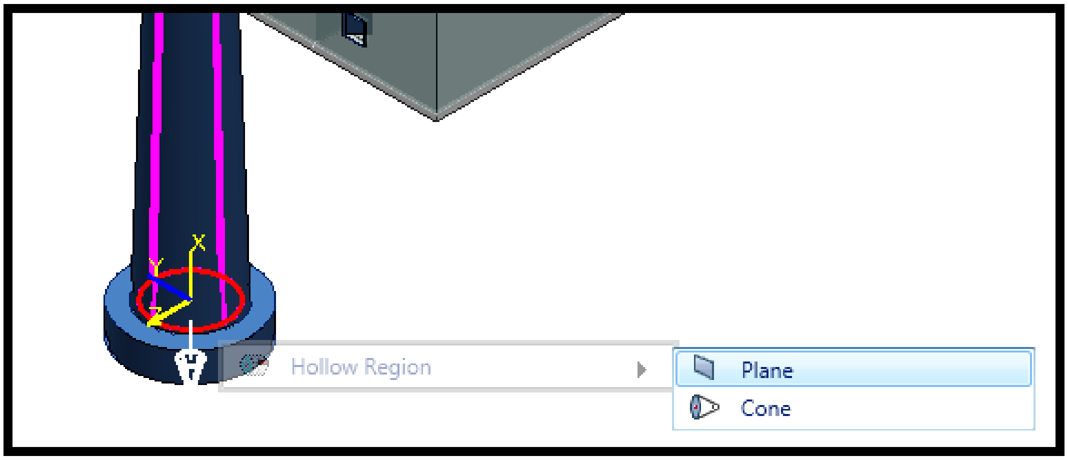
34. Your view should resemble the following graphic.



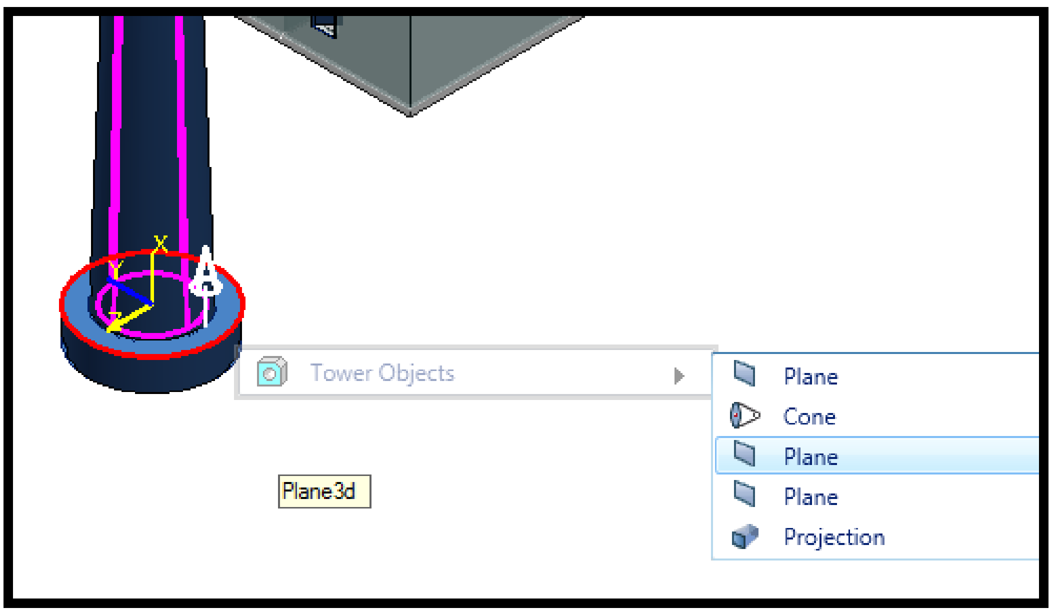
35. Select *Hollow Region* from the workspace explorer. From the smart toolbar choose New Relationship. Choose "Mate" as the new relationship.



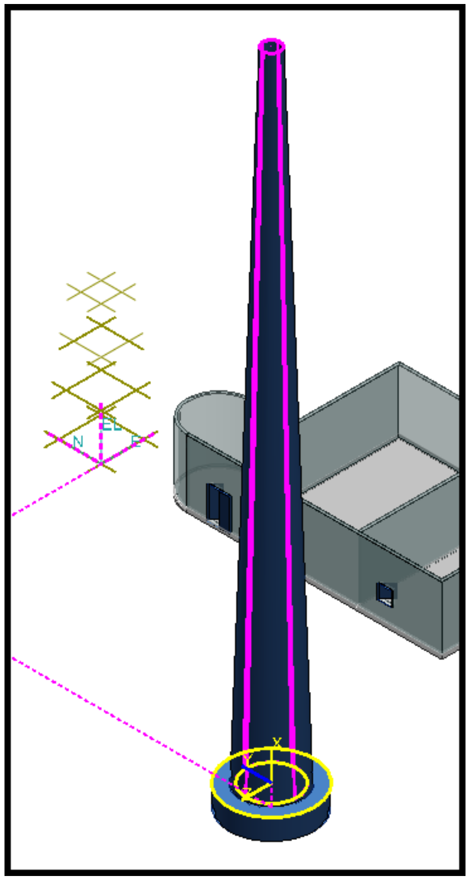
36. Move the mouse over the *Hollow Region.* Your view should now resemble the following graphic.



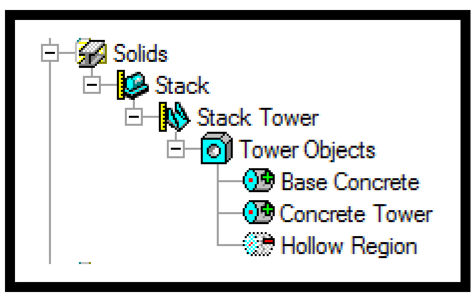
37. Move your mouse over the *Tower Objects* and it should highlight *Base Concrete* and click the left mouse button once at the location as shown in the figure. Your view should now resemble the following graphic.



38. After selecting the highlighted plane, Mate Relationship will apply to the *Base Concrete* and *Hollow Region.* Your view should resemble the following graphic.



39. Your workspace hierarchy should resemble the following graphic.



40. File Save.

© Copyright 2014 Intergraph Corporation

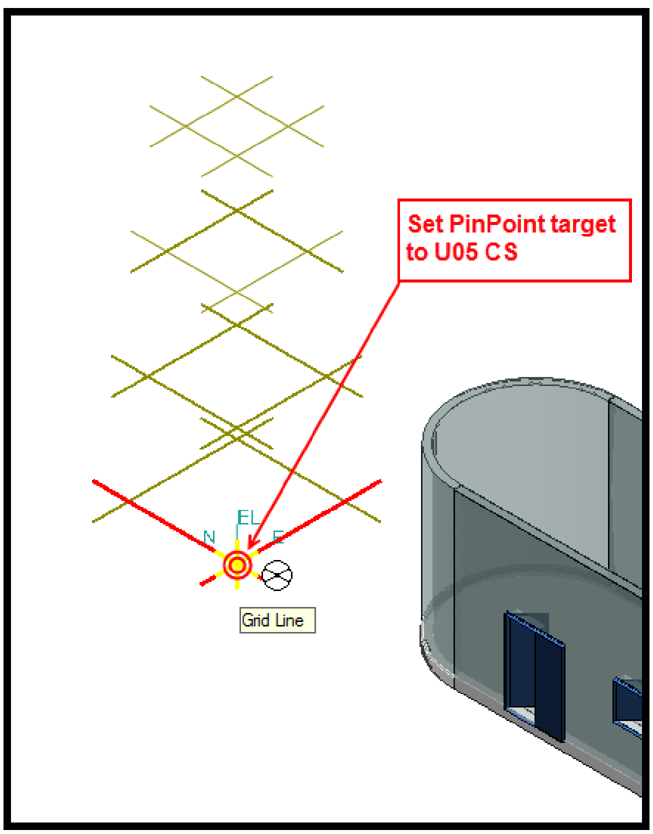
LAB-22: Designed Solids - Concrete Trench

Objective

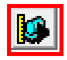
After completing this lab, you will be able to:

- Use Connect relationships to position Shapes, rotate Shapes with the Arrow keys on the keyboard. For this lab, we will create a hierarchy using Designed Equipment Components. Understand and apply relationship. Use Boolean operations (Add, Subtract and Suppress).

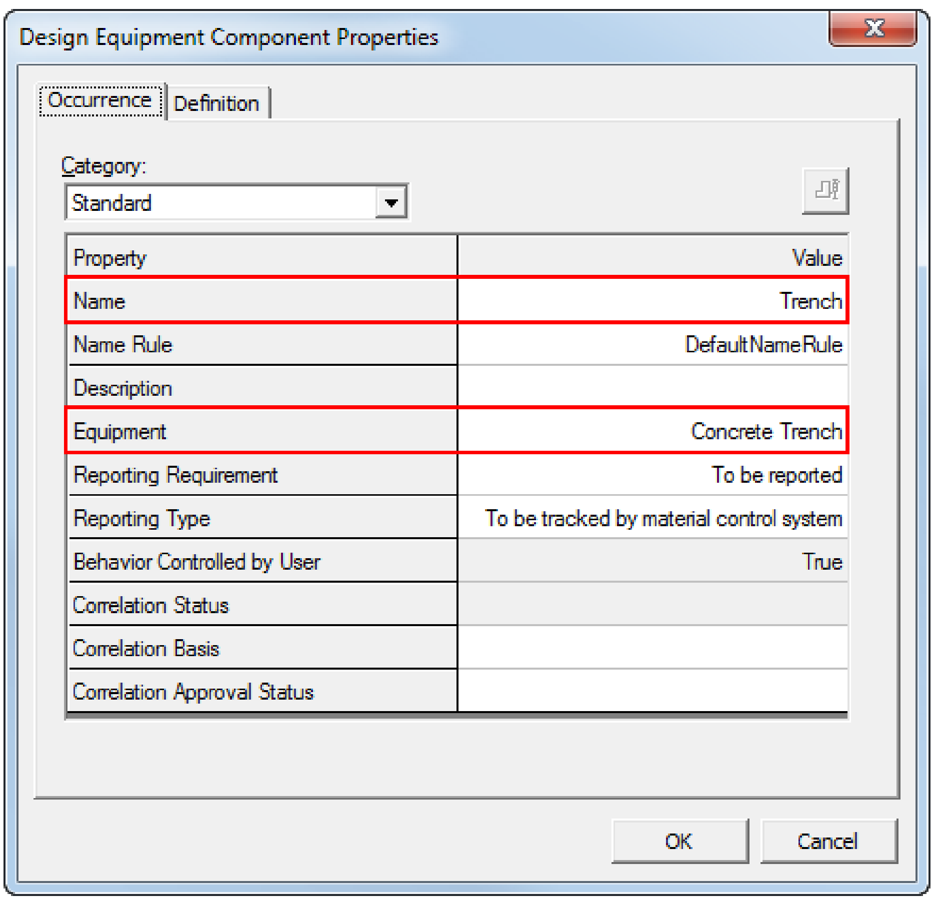
1. Open a session file.
2. Define your workspace to include the A2 → U05 and CS → U05 CS system.
3. Activate the PinPoint toolbar by selecting Tools -> PinPoint or by pressing the PinPoint command in the common toolbar and change the coordinate system from Global to U05 CS in the ribbon bar. If the U05 CS is not shown in the pull down list. Then go to More and select U05CS by expanding the CS. Select the reposition target and select the highlighted U05 CS. Your view should resemble the following graphic.



© Copyright 2014 Intergraph Corporation

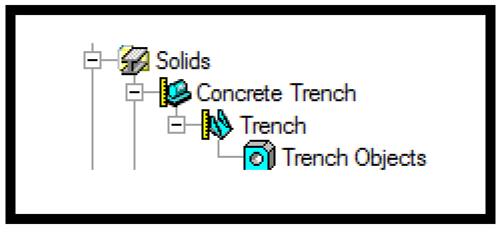
4. Expand the fly-out toolbar and press the  icon on the vertical toolbar to start the Place Designed Equipment  command. Expand *Equipment -> Civil* and select *Miscellaneous*. Click OK.
5. In the Design Equipment Properties dialog, choose A2 → U05 →Structural →*Solids* as the System and give a User Defined name of *Concrete Trench*. Click OK.
6. Place the Designed Equipment at E: -100 ft, N: -20 ft and EL: 0 ft.
7. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Equipment Component  icon.
8. Select *Concrete Trench* as the parent.
9. In the Select Equipment Component type dialog, choose Civil Components → Miscellaneous.

   The type of the Component does not matter. This dialog can be customized through the catalog to suit your needs. For example, expand \Equipment Components\Civil Equipment and Click OK.
10. In the Designed Equipment Component Properties dialog, make sure the Equipment property is set to *Concrete Trench*. Name the Component *Trench* and click OK. See image below.

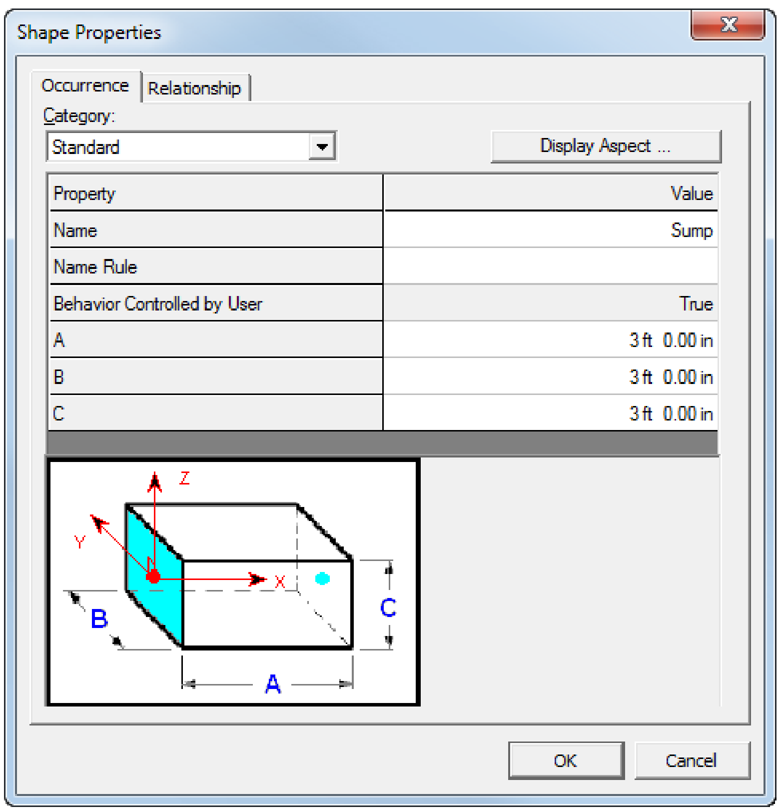


11. Place *Trench* at E: -100 ft, N: -20 ft and EL: 0 ft.

12. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Solid  icon.
13. Choose *Trench* as the parent. Rename the Solid to *Trench Objects*.
14. Go to Edit → Properties. Properties window will show up. Set the Material Type and Grade to Concrete and Fc 3000 for Solids, *Trench Objects*. Your workspace explorer should resemble the following graphic.

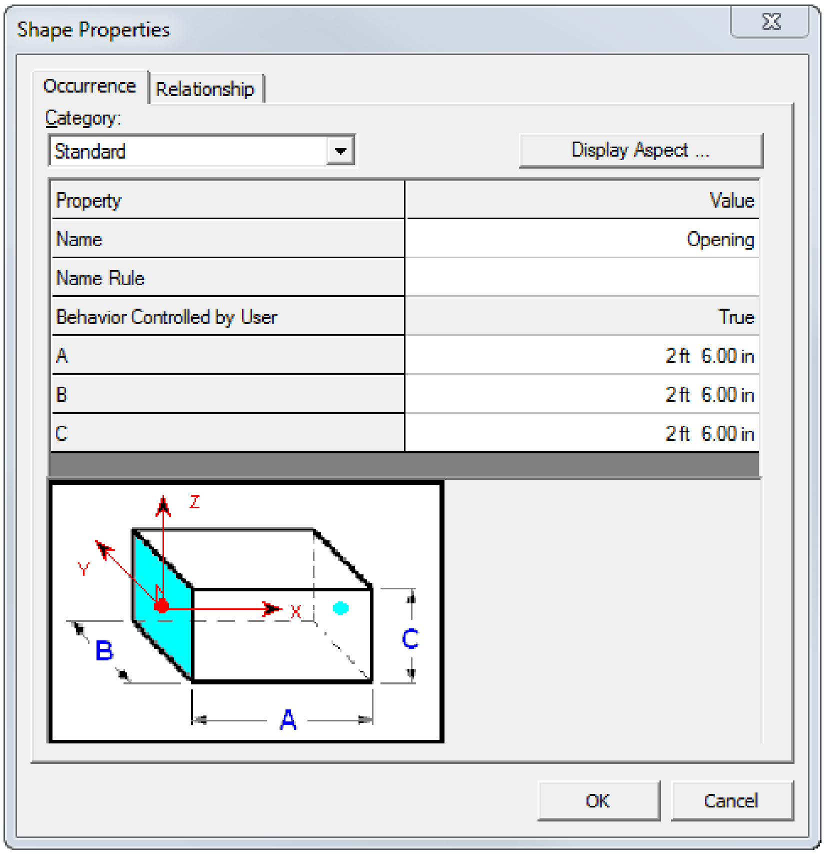


15. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid 001* from the Shapes dialog. Choose *Trench Objects* from the workspace explorer as the parent of the Shape.

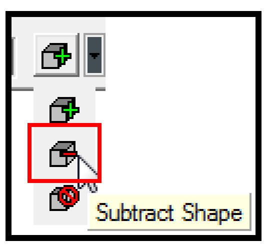


16. In the Shape Properties dialog, set A to 3 ft, B to 3 ft and C to 3ft. Name the *RectangularSolid 001* as *Sump*. Click OK.
17. Place the Shape at E: -100 ft, N: -20 ft and EL: 0 ft.

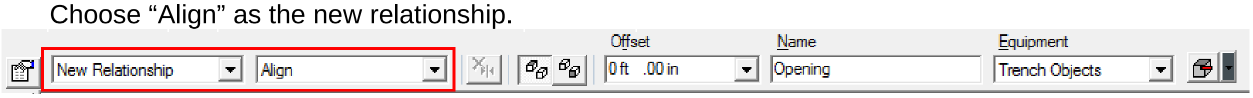
18. Select *RectangularSolid 001* from the Shapes dialog. Choose *Trench Objects* as the parent of the Shape.
19. In the Shape Properties dialog, set A to 2 ft 6in, B to 2 ft 6in and C to 2 ft 6in. Name the *RectangularSolid 001* as *Opening*. Click OK.



20. Place the Shape at E: -99 ft 9 in, N: -20 ft and EL: 0 ft.
21. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.

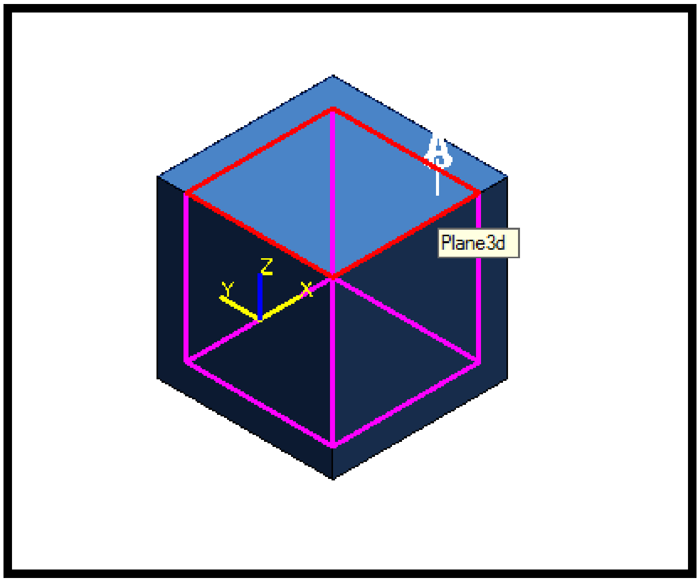


22. Select *Opening* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Align” as the new relationship.

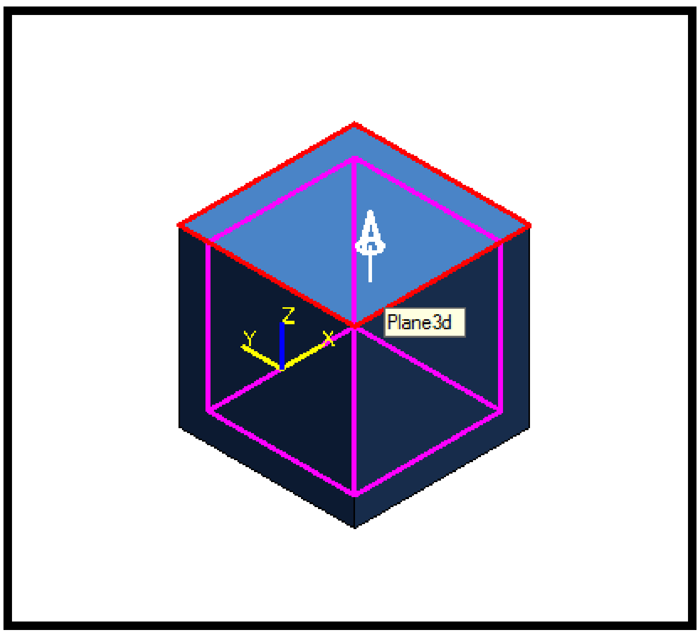


23. Move the mouse over the *Opening.* Select the highlighted region as shown below. Your view should now resemble the following graphic.

© Copyright 2014 Intergraph Corporation

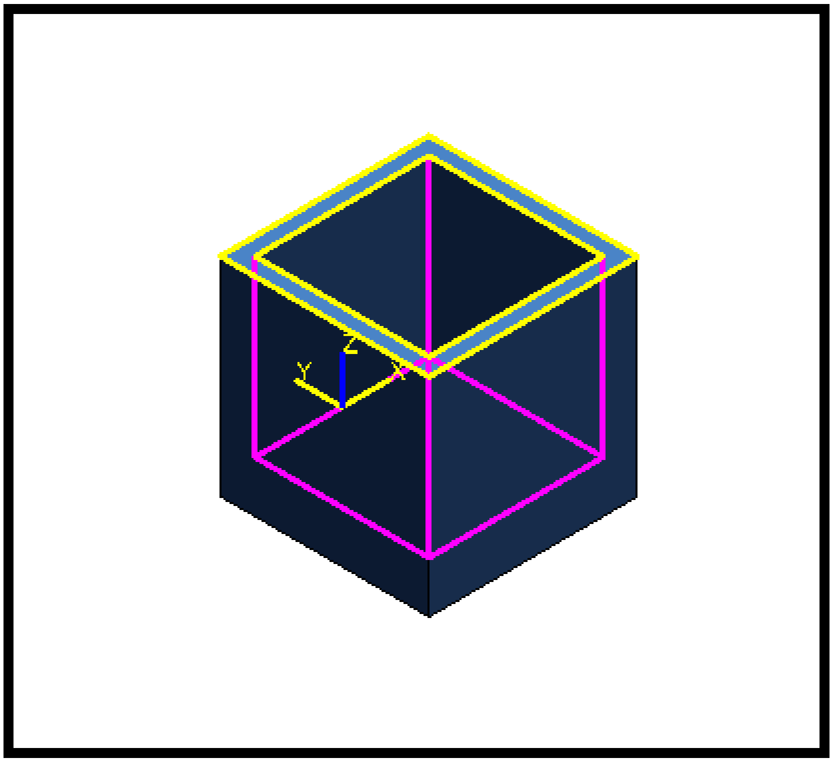


24. Move the mouse over the *Sump.* Select the highlighted region as shown below. Your view should now resemble the following graphic.

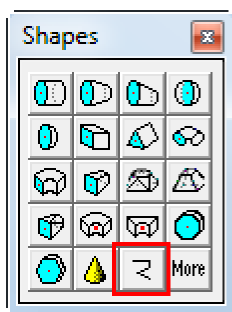


25. Your view should resemble the following graphic.

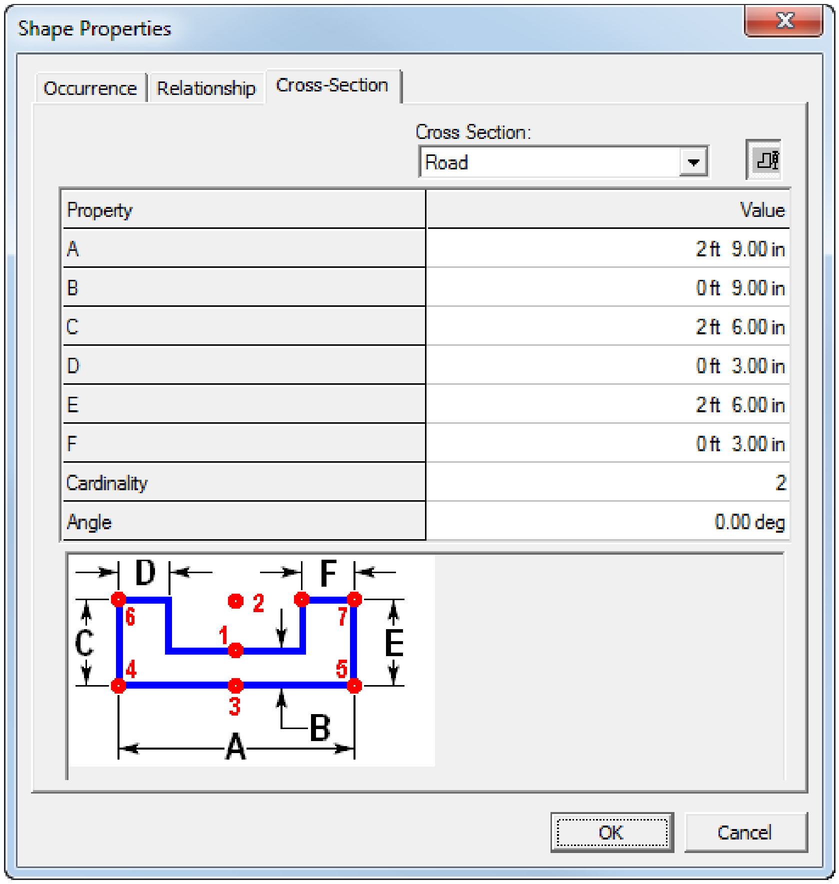
© Copyright 2014 Intergraph Corporation



26. Click the Prismatic Shape option from the shape window.



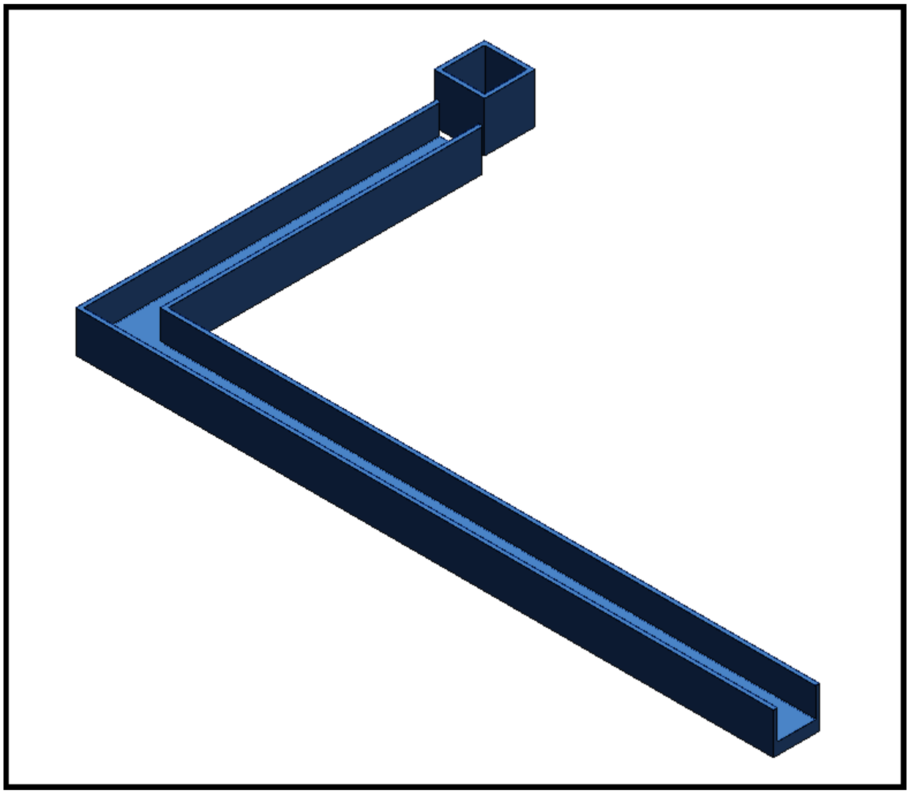
27. Choose *Trench Objects* as the parent of the Shape from the workspace explorer. Switch to the Cross-Section tab on the Shape properties dialog and choose “Road” as the cross-section. In the Shape Properties dialog, set A to 2 ft 9in, B to 0 ft 9in, C to 2 ft 6in, D to 0ft 3in, E to 2ft 6in and F to 0ft 3in and click OK.



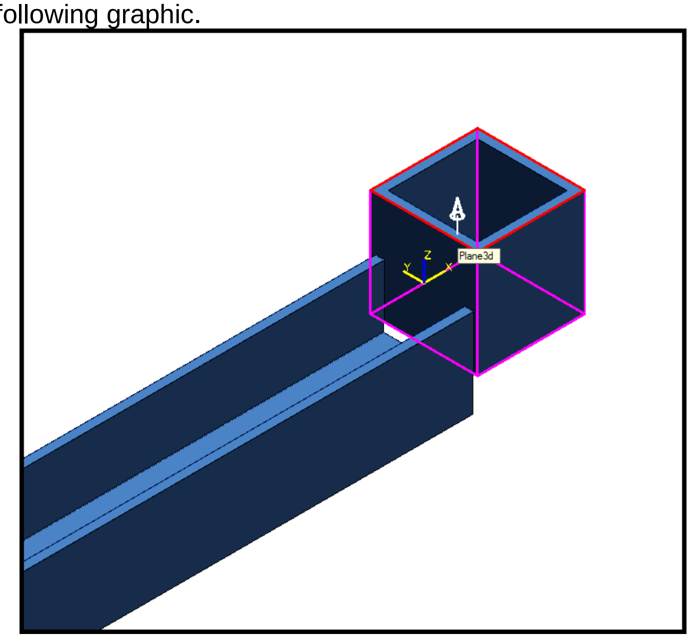
28. Key in E: -100 ft, N: -20 ft and EL: 0 ft as the start point.
29. Key in E: -120 ft, N: -20 ft and EL: 0 ft as the second point.
30. Key in E: -120 ft, N: -60 ft and EL: 0 ft as the end point.
31. Click Finish to complete the path.
32. Click Finish to complete shape placement.

Your view should resemble the following graphic.

© Copyright 2014 Intergraph Corporation

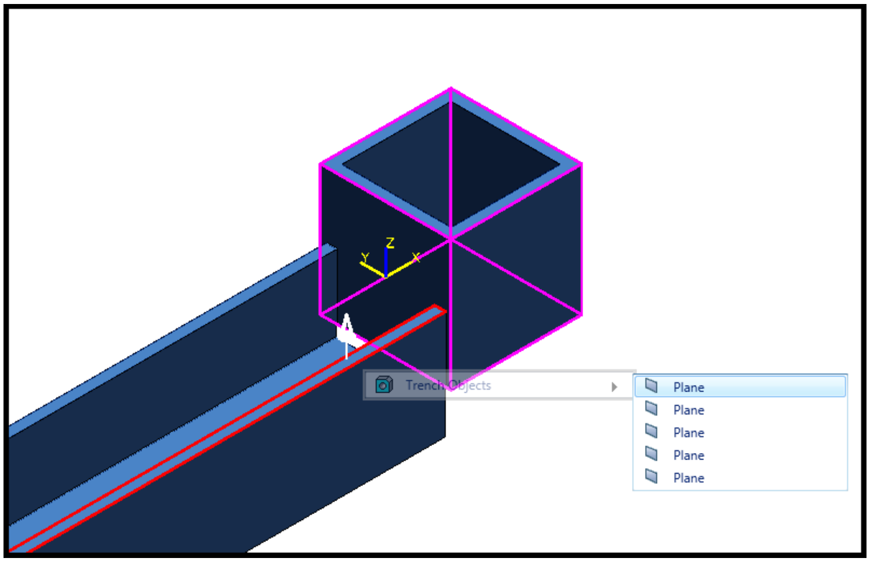


33. Rename the prismatic shape as "*Path*".
34. Select *Sump* from the workspace explorer. From the smart toolbar choose New Relationship. Choose "Align" as the new relationship.
35. Move the mouse over the *Sump.* Select the highlighted region as shown below. Your view should now resemble the following graphic.

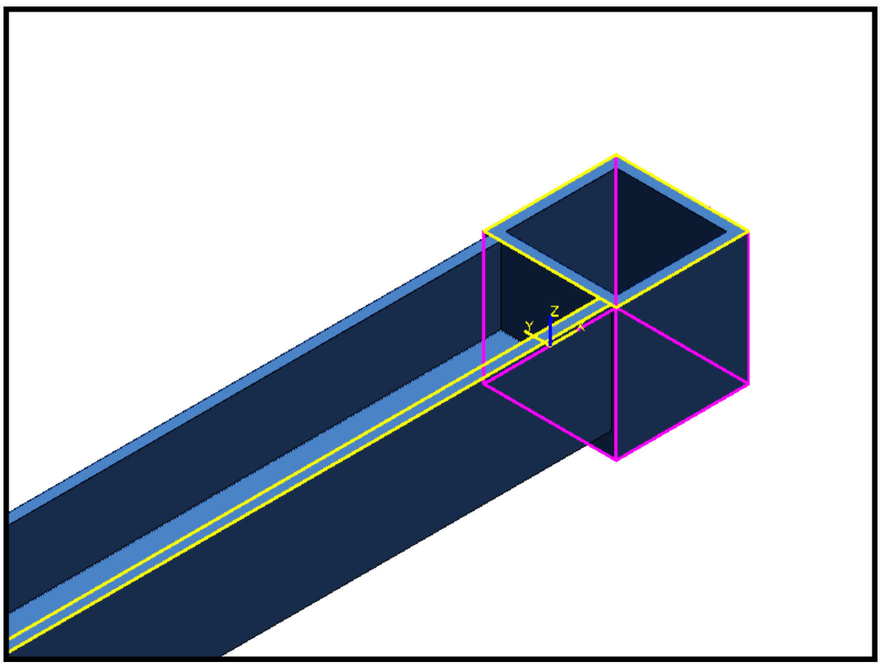


36. Move the mouse over the *Path.* Select the highlighted region as shown below. Your view should now resemble the following graphic.

© Copyright 2014 Intergraph Corporation

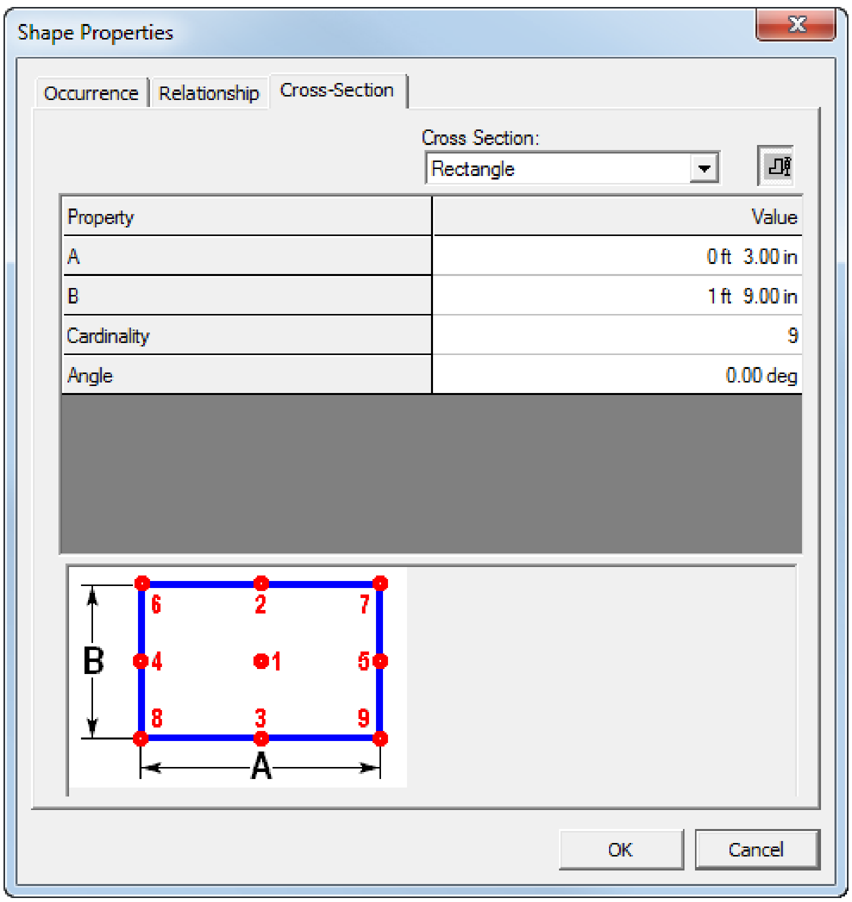


37. Your view should resemble the following graphic.

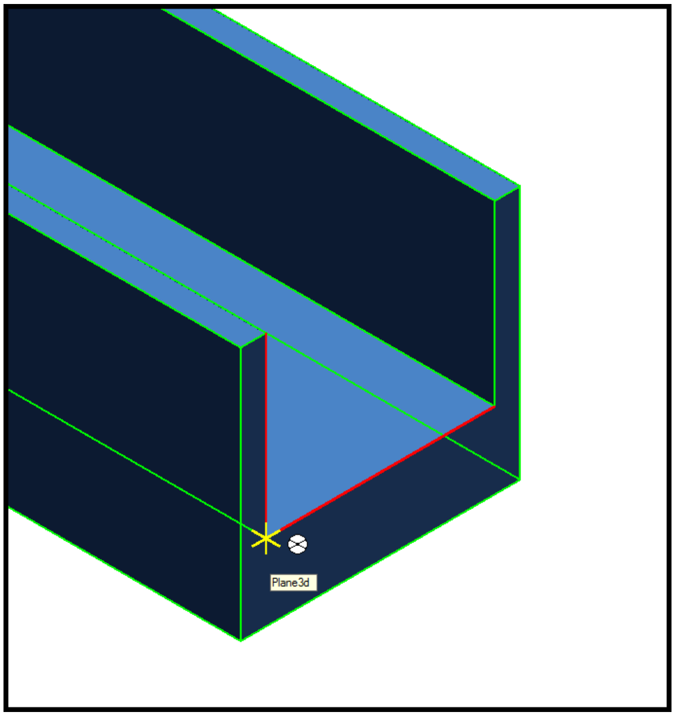


38. Click the Prismatic Shape option from the shape window.

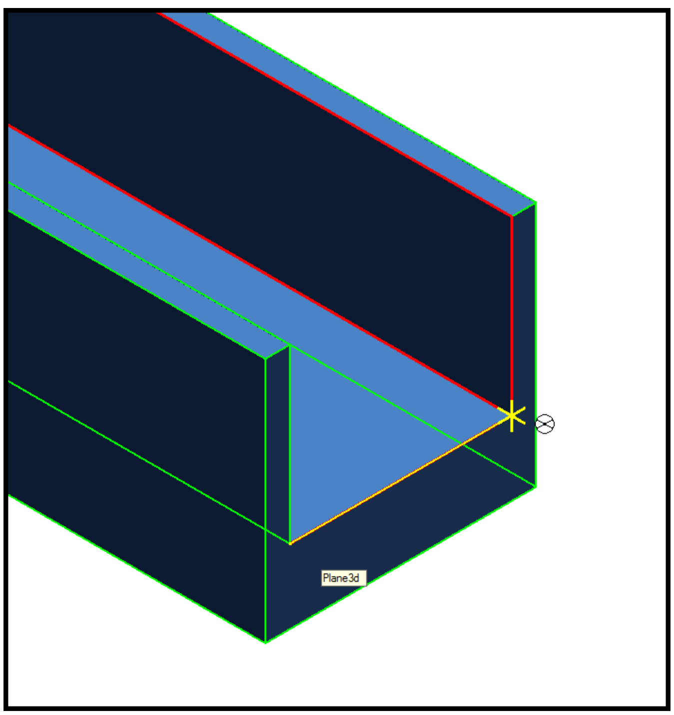
39. Choose *Trench Objects* as the parent of the Shape from the workspace explorer. Switch to the Cross-Section tab on the Shape proeprties dialog and choose “Rectangle” as the cross-section. In the Shape Properties dialog, set A to 0 ft 3in, B to 1ft 9in and cardinality to 9. Click OK.



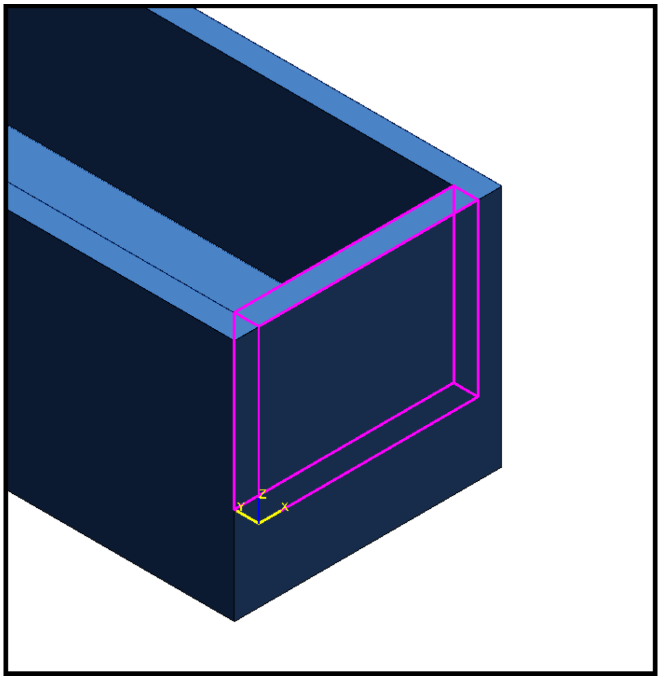
40. Select the point as shown in the image below.



41. Select the end point as shown in the image below.



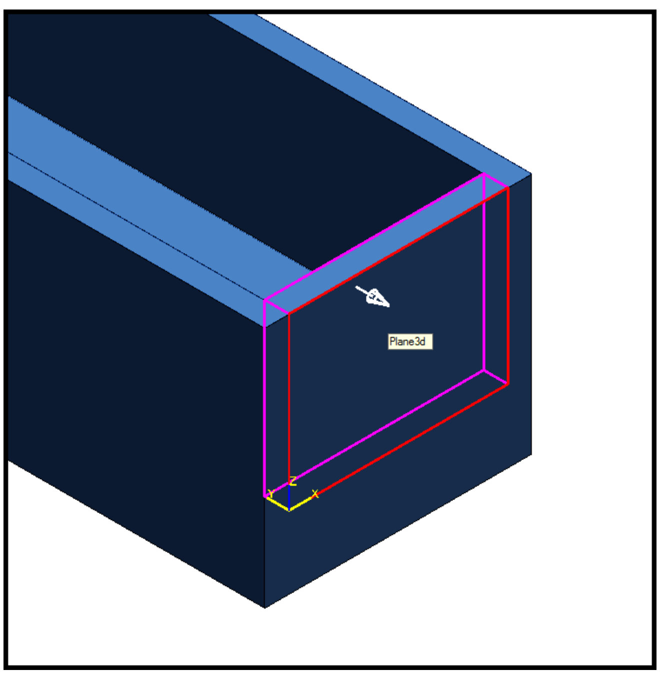
42. Click Finish. Your view should resemble the following graphic. Click Finish to complete the placement.



43. Rename the prismatic shape as “*End Part*”.

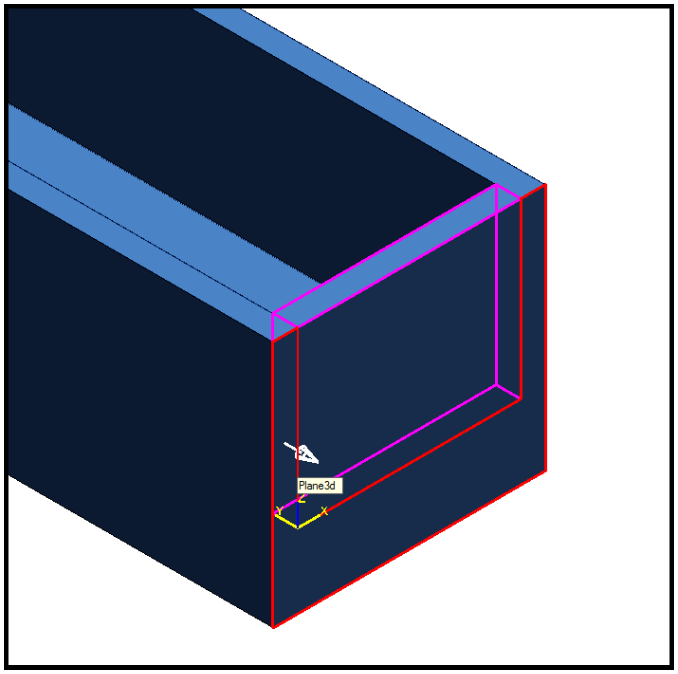
44. Select *End Part* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Align” as the new relationship.

45. Move the mouse over the *End Part.* Select the highlighted region as shown below. Your view should now resemble the following graphic.

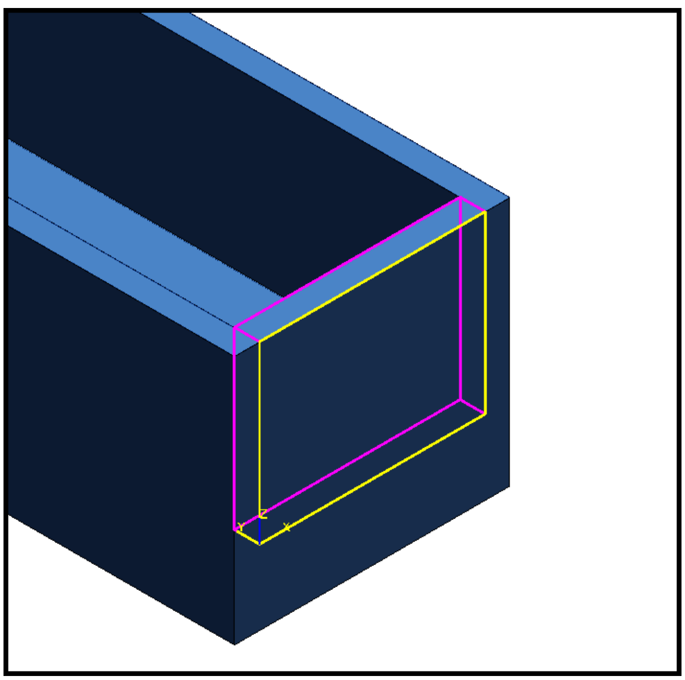


46. Move the mouse over the *Path.* Select the highlighted region as shown below. Your view should now resemble the following graphic.

© Copyright 2014 Intergraph Corporation

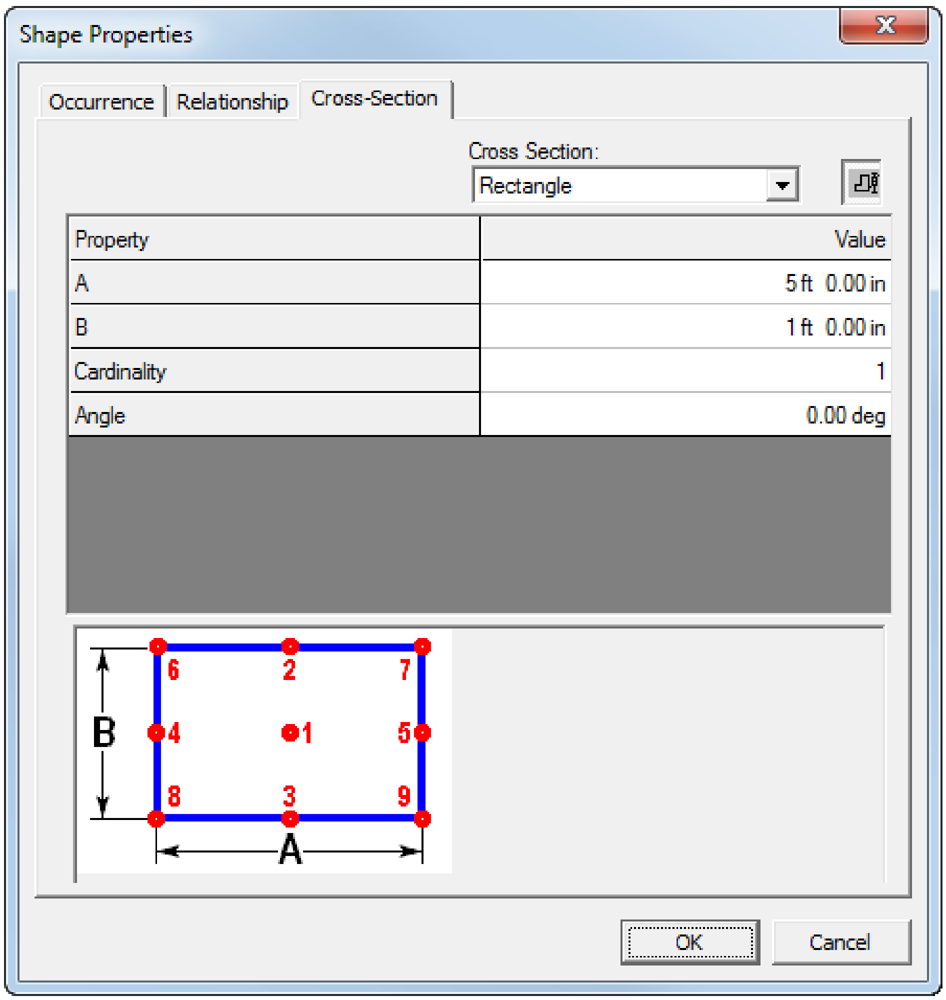


47. Your view should resemble the following graphic.



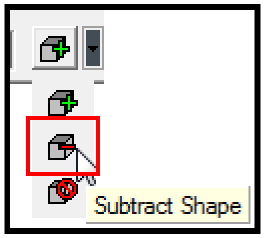
48. Click the Prismatic Shape option from the shape window.

49. Choose *Trench Objects* as the parent of the Shape from the workspace explorer. Choose “Rectangle” as the cross-section. In the Shape Properties dialog, set A to 5 ft 0in, B to 1ft 0in. Click OK.

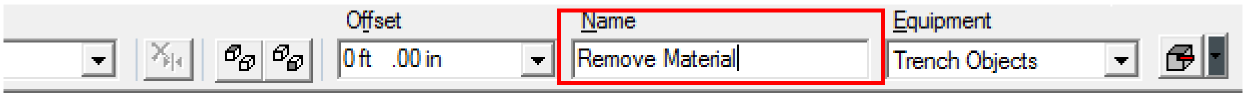


50. Key in E: -100 ft, N: -20 ft and EL: -1 ft as the start point.
51. Key in E: -120 ft, N: -20 ft and EL: -1 ft as the second point.
52. Key in E: -120 ft, N: -65 ft and EL: -0 ft 9in as the end point.
53. Click Finish.
54. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.

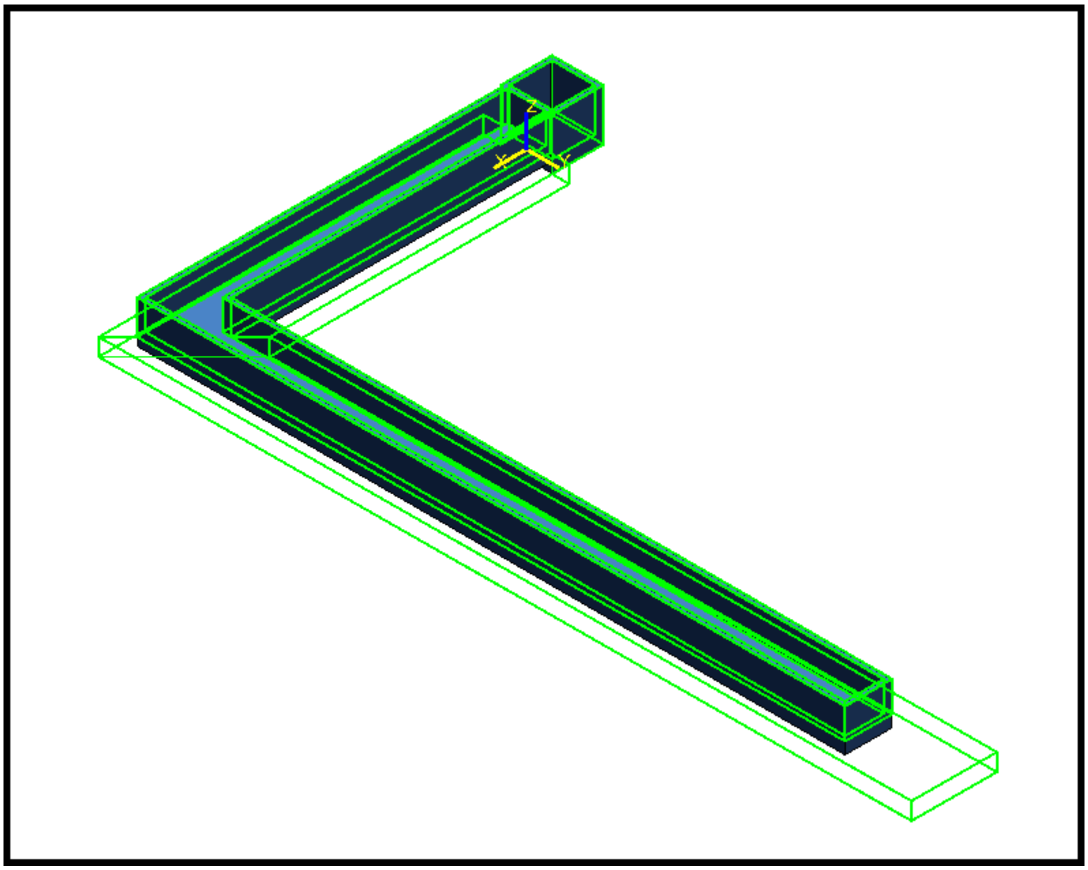
© Copyright 2014 Intergraph Corporation



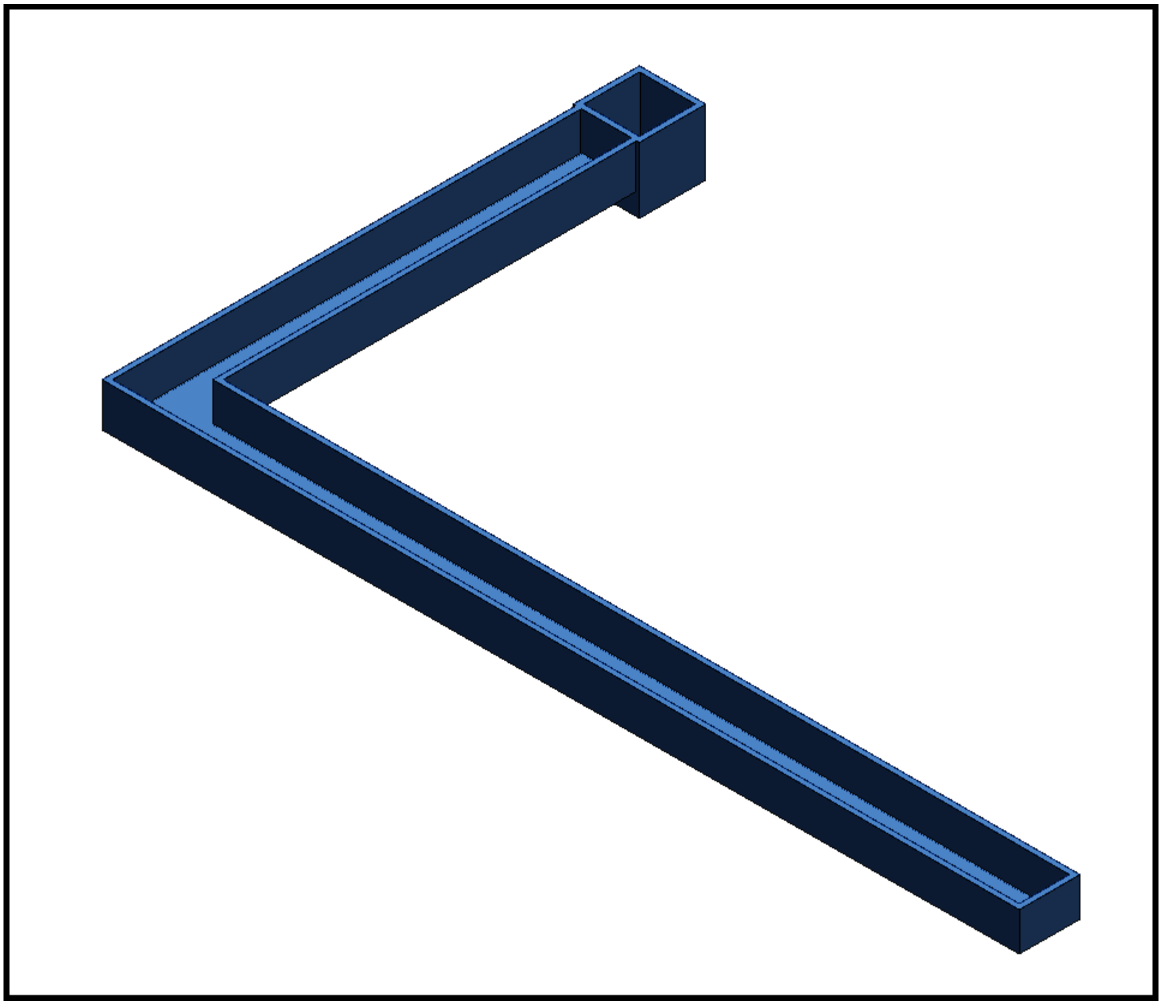
55. Name this object as "*Remove Material*" in the toolbar.



56. Click Finish. Your view should resemble the following graphic.



57. Click the select button. Your view should resemble the following graphic.



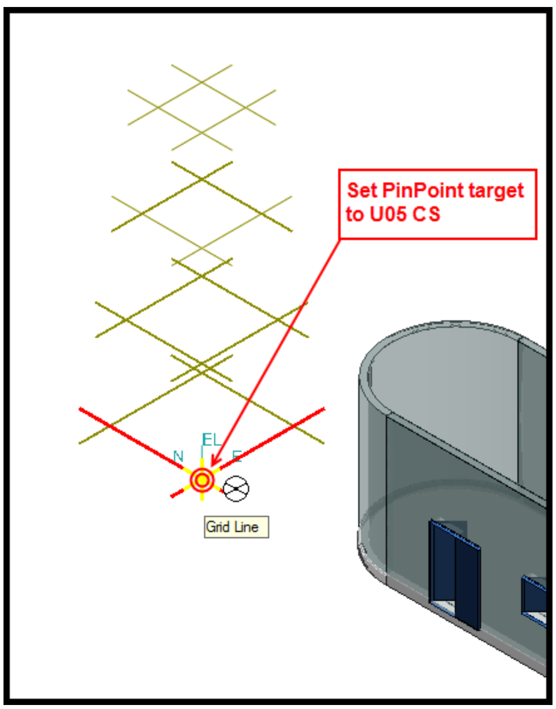
LAB-23: Designed Solids – Sloped Slab

Objective

After completing this lab, you will be able to:

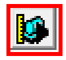
- Use Connect relationships to position Shapes, rotate Shapes with the Arrow keys on the keyboard. For this lab, we will create a hierarchy using Designed Equipment Components. Understand and apply relationship. Use Boolean operations (Add, Subtract and Suppress) effectively and understand Order Shapes option.

1. Open a session file.
2. Define your workspace to include the A2 → U05 and CS → U05 CS system.
3. Activate the PinPoint toolbar by selecting Tools -> PinPoint or by pressing the PinPoint command in the common toolbar and change the coordinate system from Global to U05 CS in the ribbon bar. If the U05 CS is not shown in the pull down list. Then go to More and select U05CS by expanding the CS. Select the reposition target and select the highlighted U05 CS. Your view should resemble the following graphic.

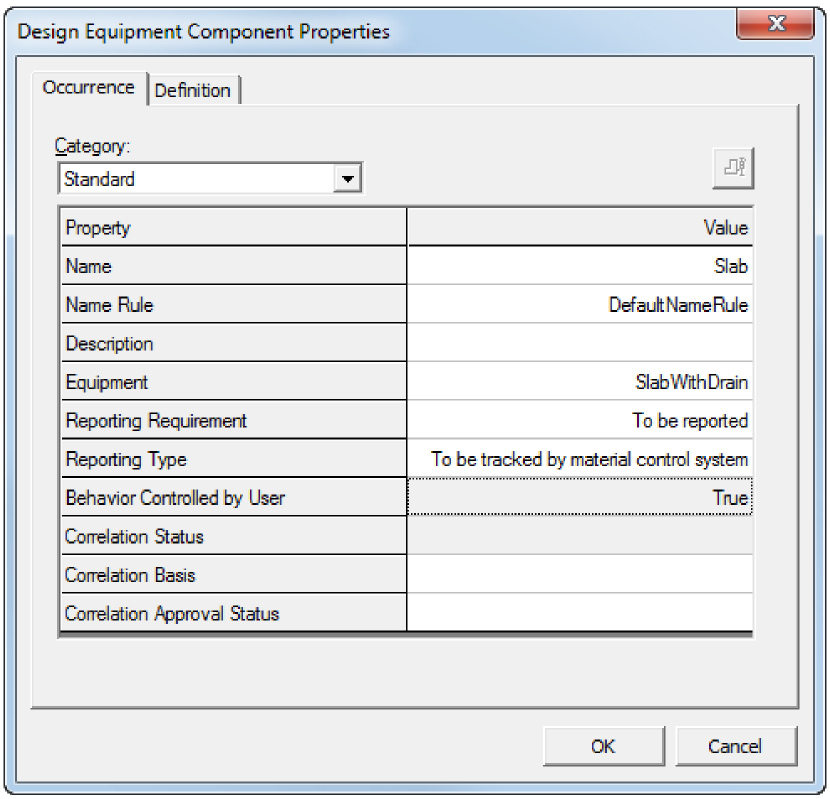


4. Expand the fly-out toolbar and press the  icon on the vertical toolbar to start the Place Designed Equipment  command. Expand *Equipment -> Civil* and select *Miscellaneous*. Click OK.
5. In the Design Equipment Properties dialog, choose A2 → U05 →Structural →*Solids* as the System and give a User Defined name of *SlabWithDrain*. Click OK.
6. Place the Designed Equipment at E: -60 ft, N: -20 ft and EL: 0 ft.

© Copyright 2014 Intergraph Corporation

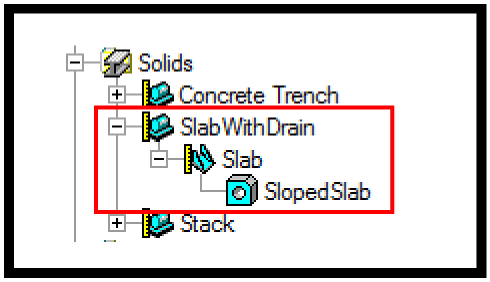
7. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Equipment Component  icon.
8. Select *SlabWithDrain* as the parent.
9. In the Select Equipment Component type dialog, choose Civil Components → Miscellaneous.

   The type of the Component does not matter. This dialog can be customized through the catalog to suit your needs. For example, expand \Equipment Components\Civil Equipment and click OK.
10. In the Designed Equipment Component Properties dialog, make sure the Equipment property is set to *SlabWithDrain*. Name the Component *Slab* and click OK. See image below

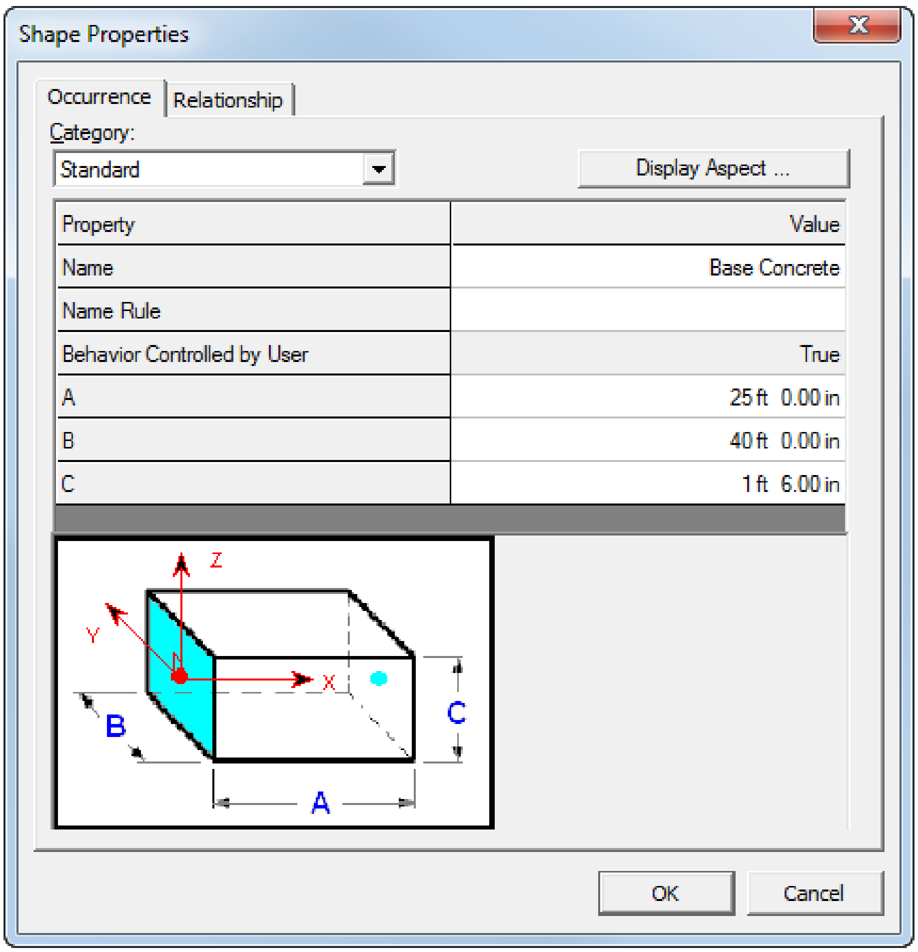


11. Place *Slab* at E: -60 ft, N: -20 ft and EL: 0 ft.
12. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Solid  icon.
13. Choose *Slab* as the parent. Rename the Solid to *SlopedSlab*.
14. Go to Edit → Properties. Properties window will show up. Set the Material Type and Grade to Concrete and Fc 3000 for Solids, *SlopedSlab*. Your workspace explorer should resemble the following graphic.

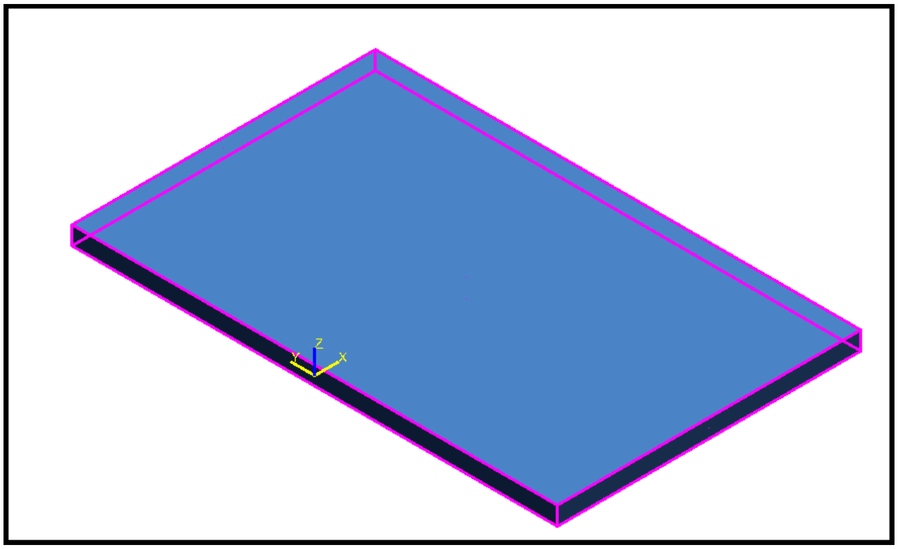
© Copyright 2014 Intergraph Corporation



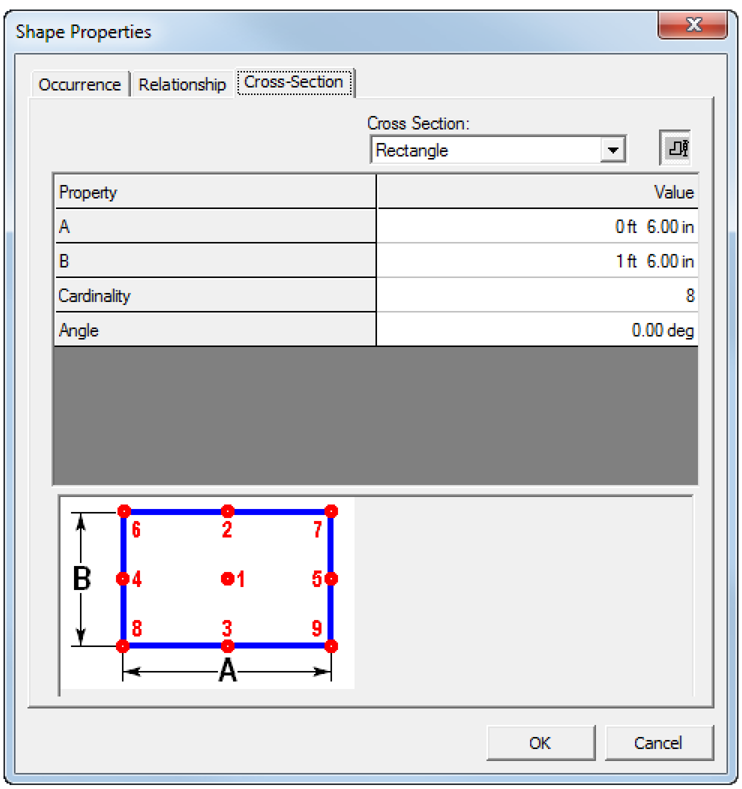
15. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid* from the Shapes dialog. Choose *SlopedSlab* as the parent of the Shape.
16. In the Shape Properties dialog, set A to 25 ft, B to 40 ft, and C to 1 ft 6in. Name the *RectangularSolid* as *Base Concrete*. Click OK.



17. Place the Shape at E: -60 ft, N: -40 ft and EL: 0 ft.
18. Your view should resemble the following graphic.

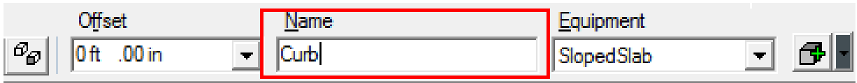


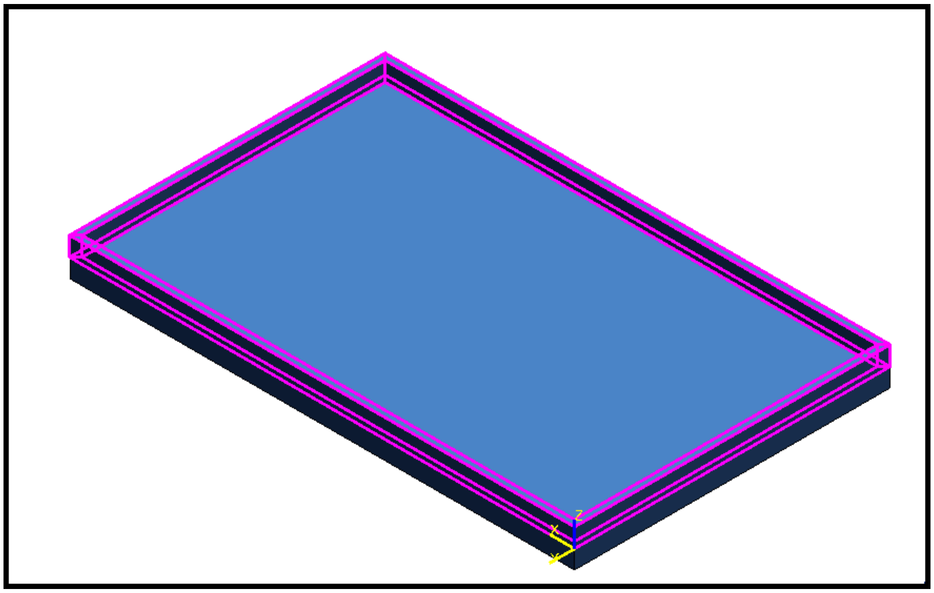
19. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *Prismatic Shape* from the Shapes dialog. Choose *SlopedSlab* as the parent of the Shape.
20. Choose *SlopedSlab*. Shape properties window will show up. Select the Cross-Section tab.
21. Choose Rectangle from the Cross Section pull down menu and enter the value for A = 6 in and B = 1 ft 6 in. Enter cardinality = 8. You view should resemble the following graphic.



22. Sketch a path along the top edges of the *Base Concrete.*

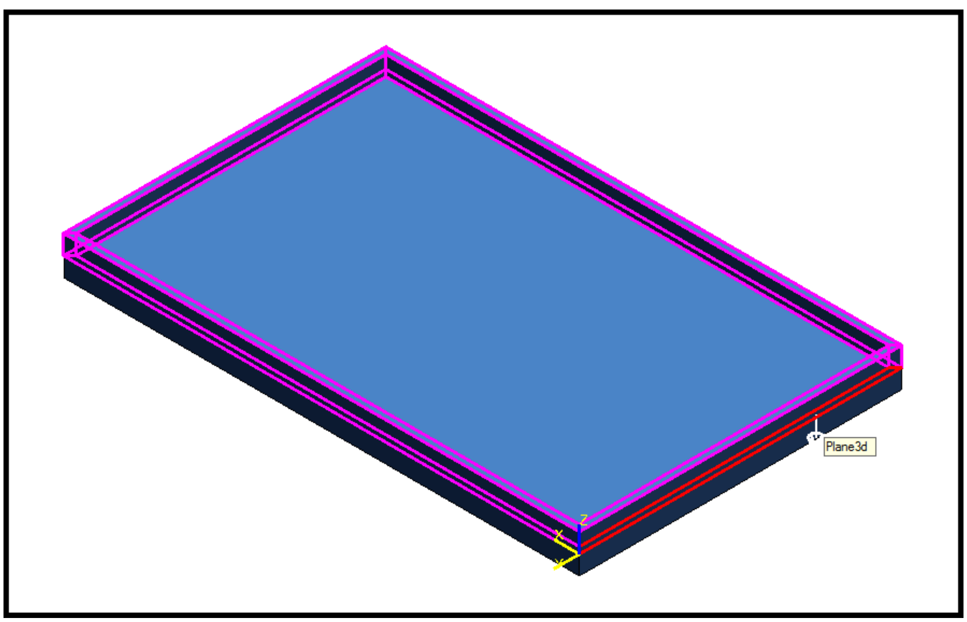
23. Name the *Prismatic Shape* as "*Curb"* and hit the finish. Your view should now resemble the following graphic. Click Finish.



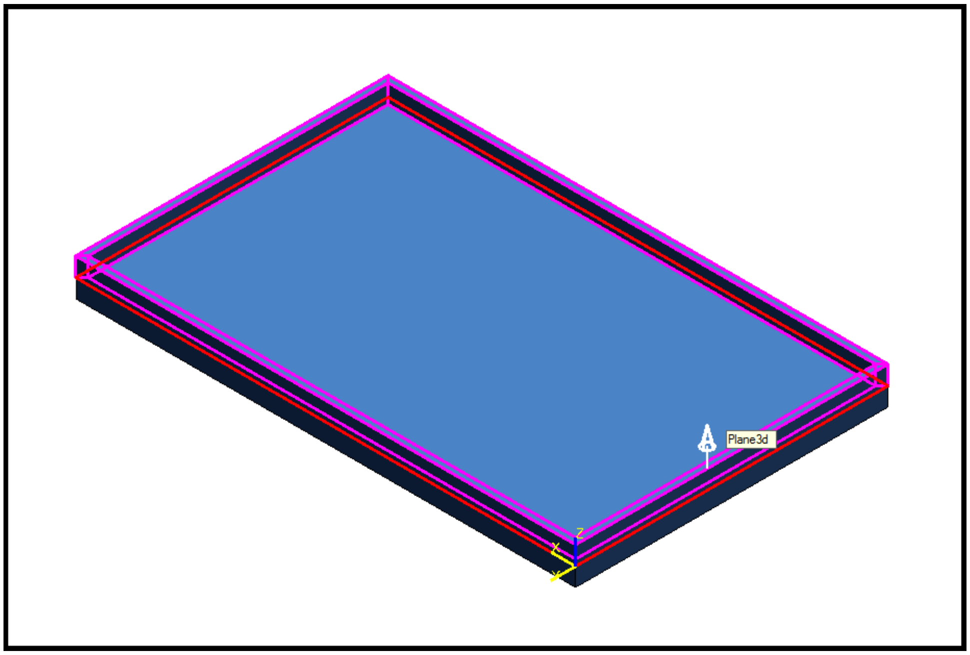


24. Select *Curb* from the workspace explorer. From the smart toolbar choose New Relationship. Choose "Mate" as the new relationship.

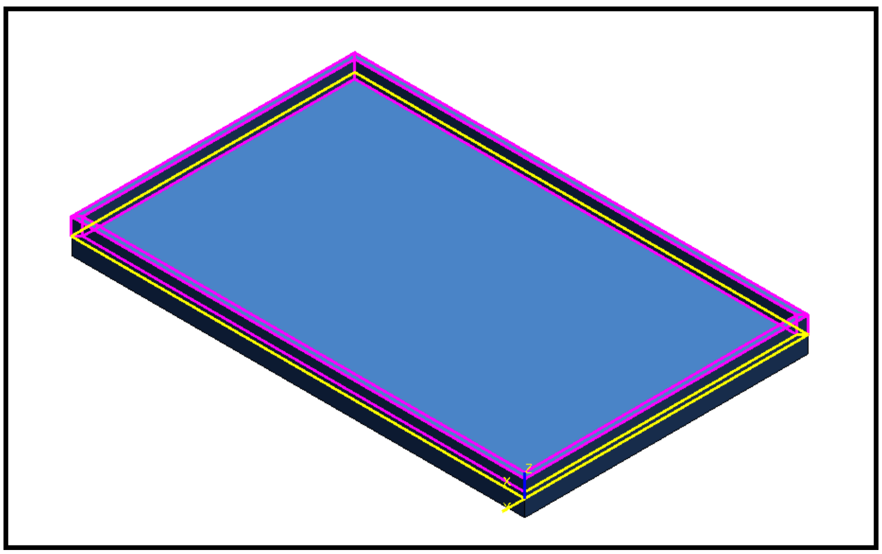
25. Select the plane as shown in the following graphic.



26. Select the top face of *Base Concrete* and your view should resemble as shown in the graphic below,



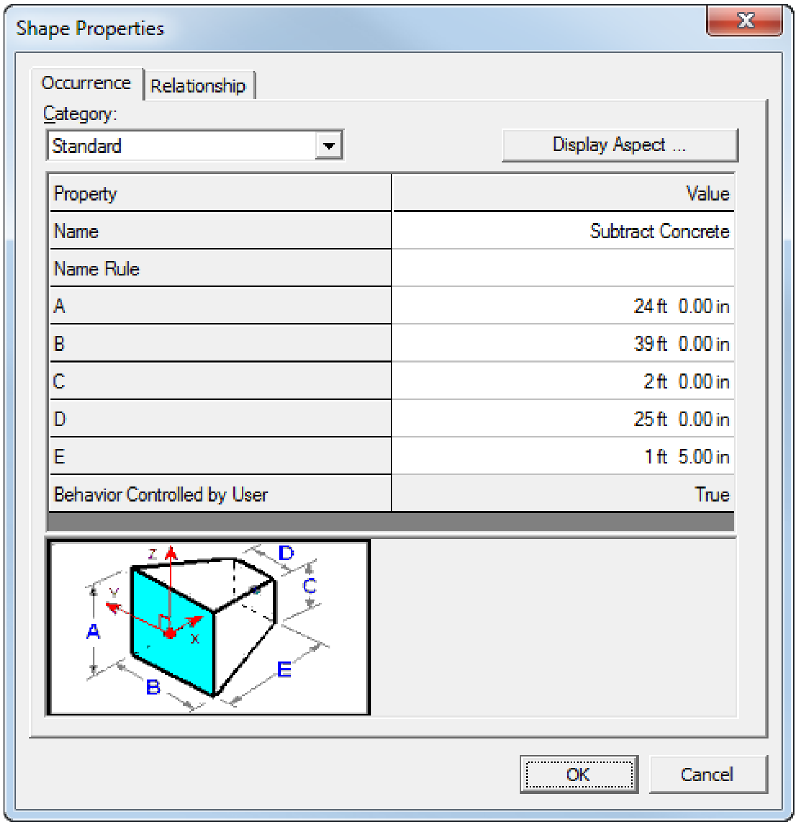
27. Your graphic should resemble as shown below.



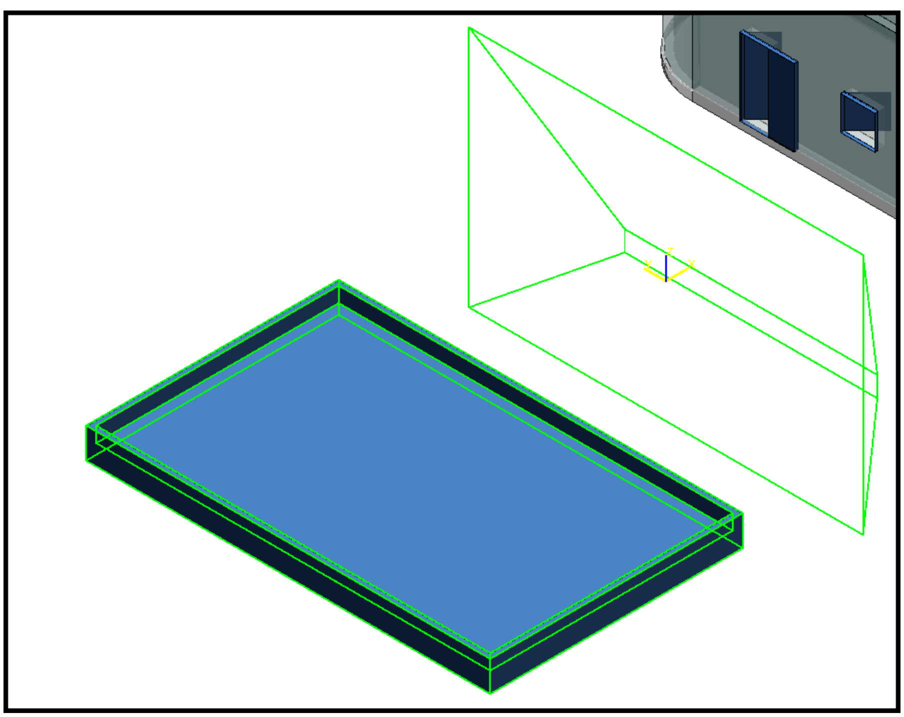
28. Select the EccentricRectangularPrism 001 from the shapes dialog box as shown below in the image.

29. Choose *SlopedSlab* as the parent of the Shape.

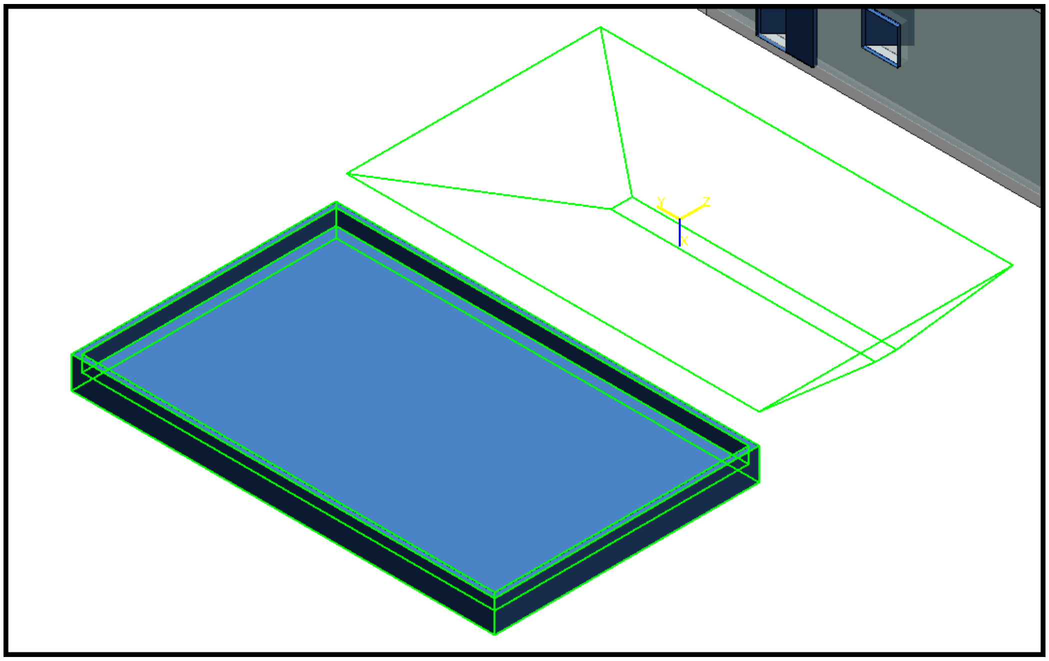
30. In the Shape Properties dialog, set A to 24 ft, B to 39 ft, C to 2 ft, and D to 25 ft and E to 1 ft 5 in. Name the EccentricRectangularPrism 001 as "*Subtract Concrete*". Click OK.



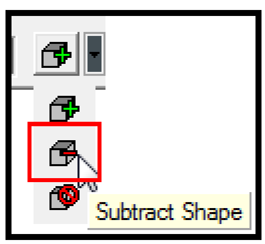
31. Your view should now resemble the following graphic



32. We need to rotate the Shape to place it correctly. This can be done before placement time using the Arrow keys on the keyboard.
33. Use the Up arrow key to change the axis of rotation and use the left/right arrow keys to rotate the shape. Your view should resemble the following graphic.

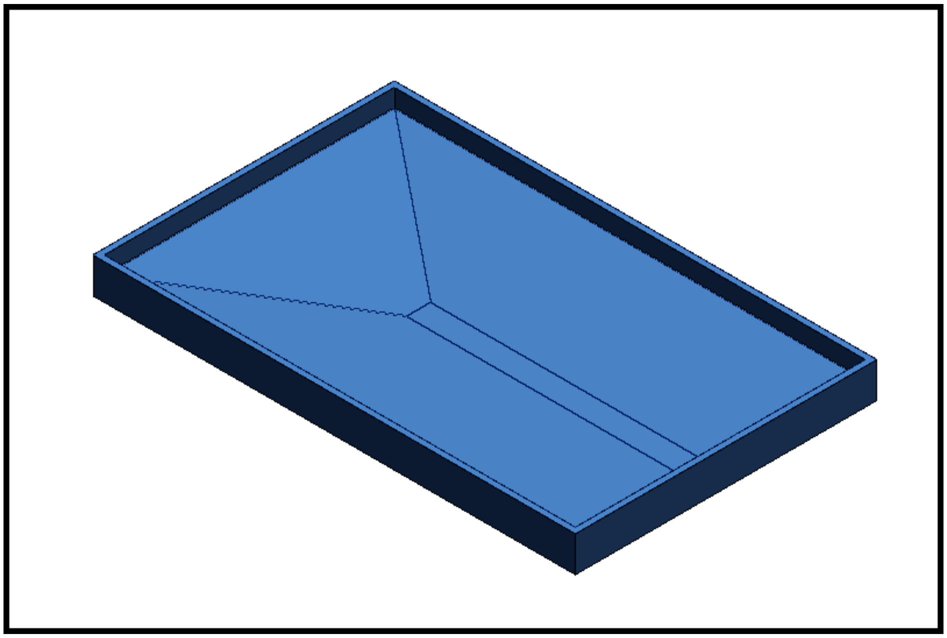


34. Place the Shape at E: -47 ft 6in, N: -40 ft and EL: 0ft 9 in.
35. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.

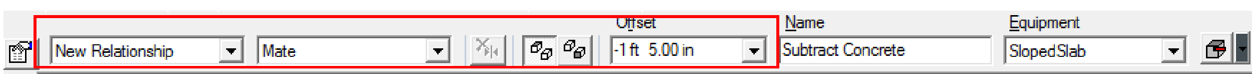


36. Your view should resemble the following graphic.

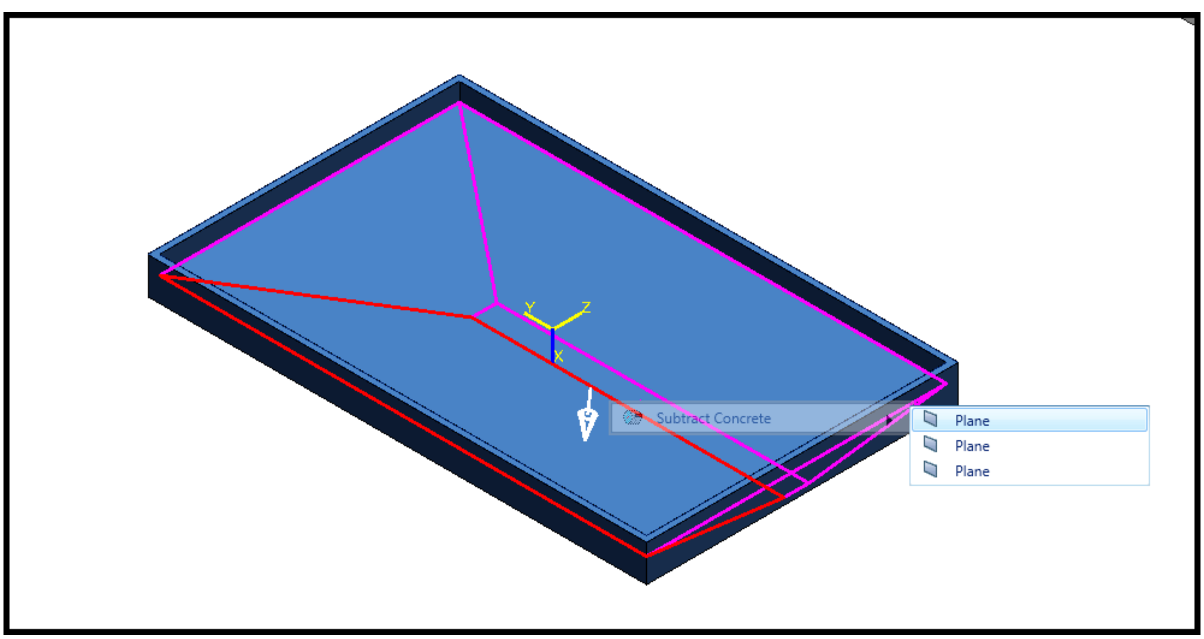
© Copyright 2014 Intergraph Corporation



37. Select *Subtract Concrete* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Mate” as the new relationship and key in -1 ft 5 in as the offset value.



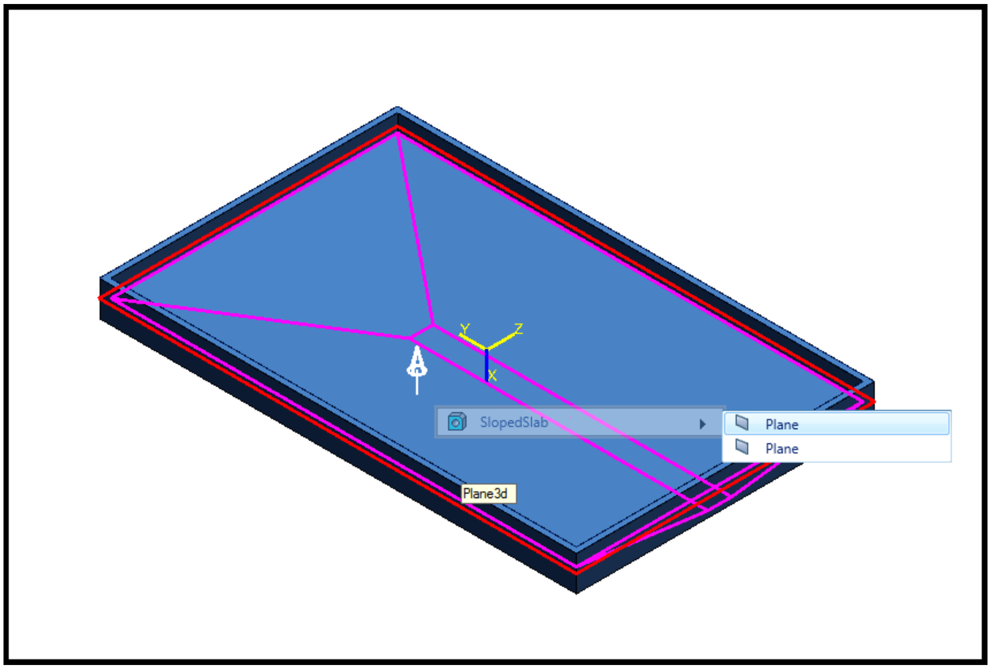
38. Move your mouse over the SlopedSlab and it should highlight *Subtract Concrete* and click the left mouse button once at the location as shown in the figure. Your view should now resemble the following graphic.



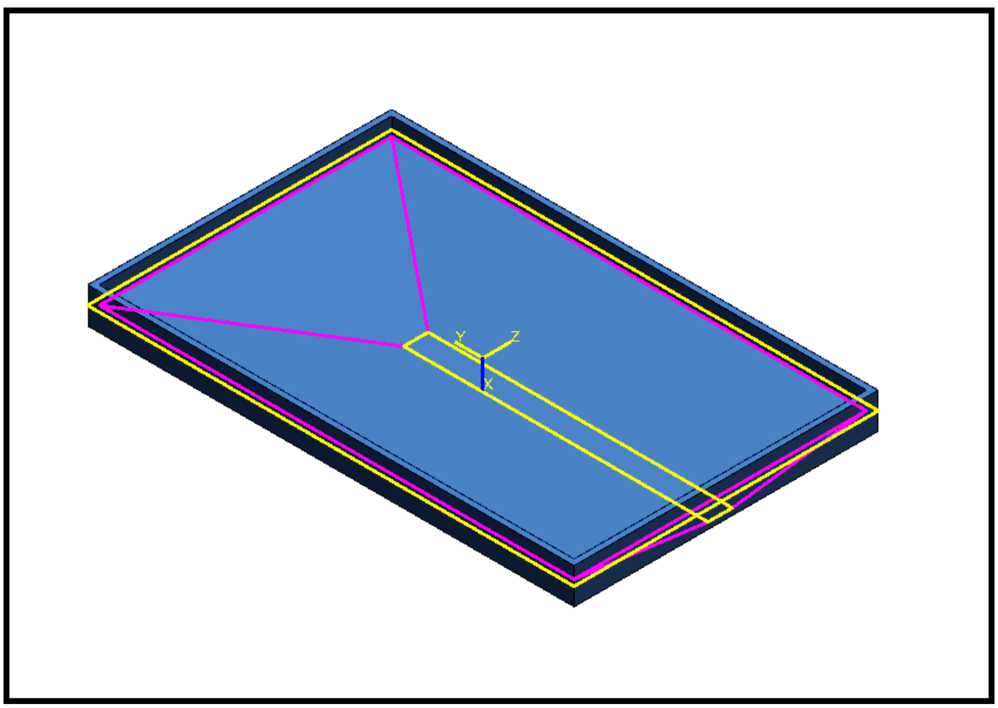
39. Select the correct plane from the quick pick list.

© Copyright 2014 Intergraph Corporation

40. Select the *Base Concrete.* Your view should now resemble the following graphic.



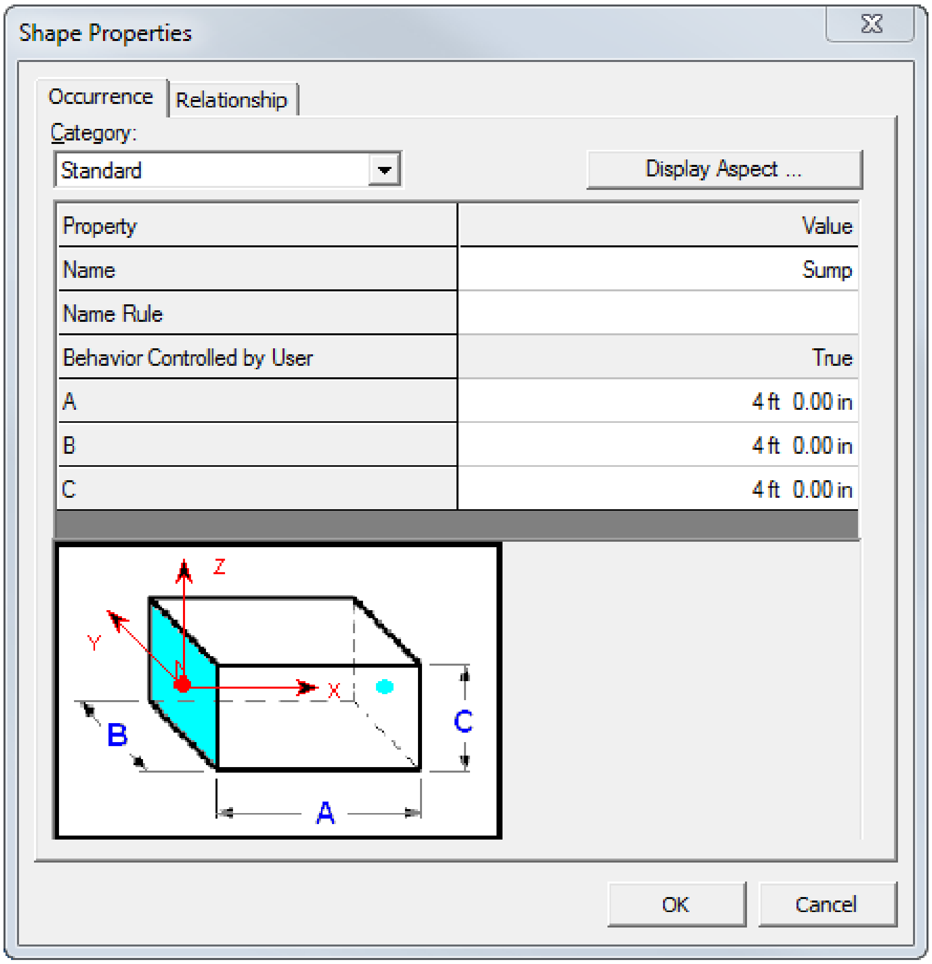
41. After selecting the highlighted plane, Mate Relationship will apply to the *Base Concrete* and *Subtract Concrete.* Your view should resemble the following graphic.



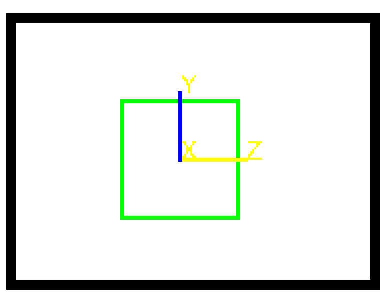
42. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid* from the Shapes dialog.

43. Choose *SlopedSlab* as the parent of the Shape.

44. In the Shape Properties dialog, set A to 4 ft, B to 4 ft, and C to 4 ft and enter the name as "*Sump*". Click OK

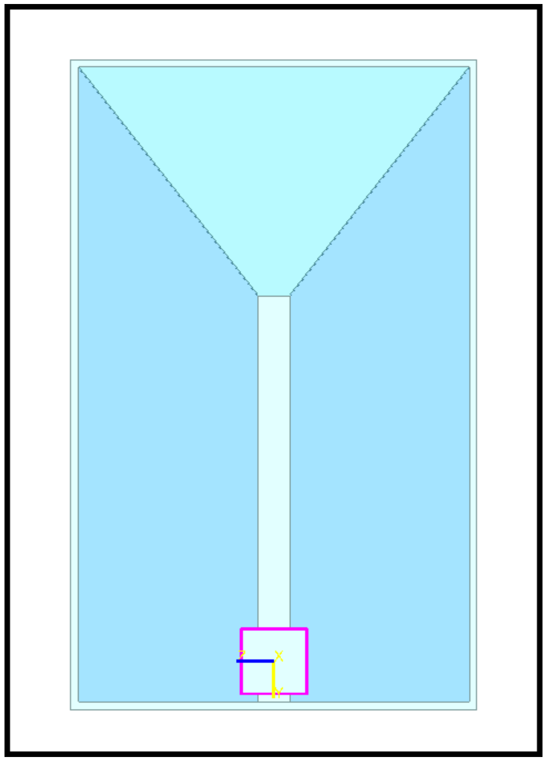


45. While looking in the Plan Plane, use the Up arrow keys (if needed) to change the axis of rotation and use the Left or Right arrow keys to rotate the shape until the point of placement is the center of the rectangle.

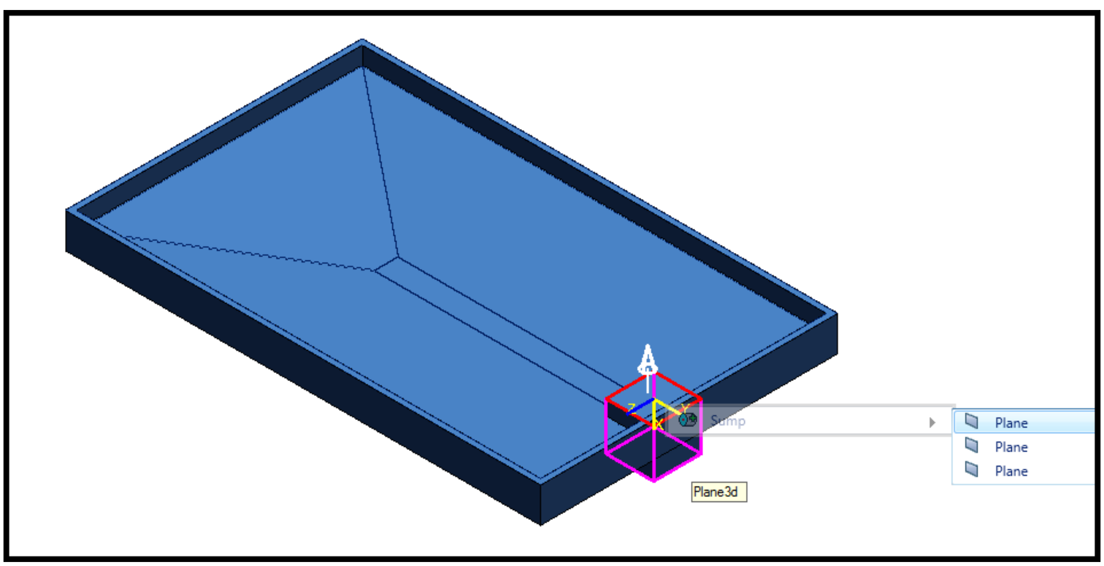


46. Place the Shape at E: -47 ft 6 in, N: -57 ft and EL: 2 ft 2 in.

47. Change back to the Looking Plan view. Create a Connect Relationship at the point shown below and click to place the Shape. Your view should resemble the following graphic.

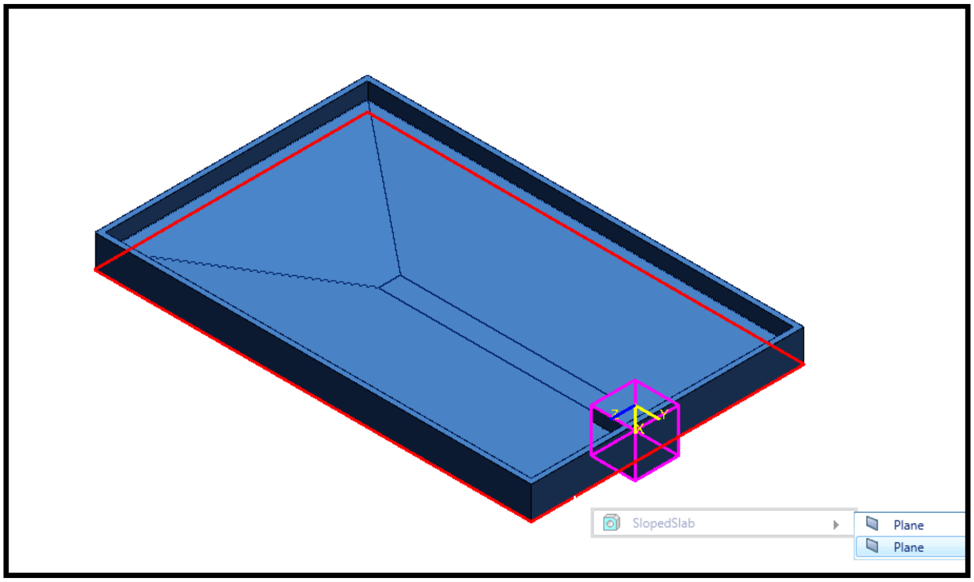


48. Change the view back to isometric. Select the S*ump* and add mate as the new relationship. Select the top face of the S*ump* and your view should resemble the following graphic.

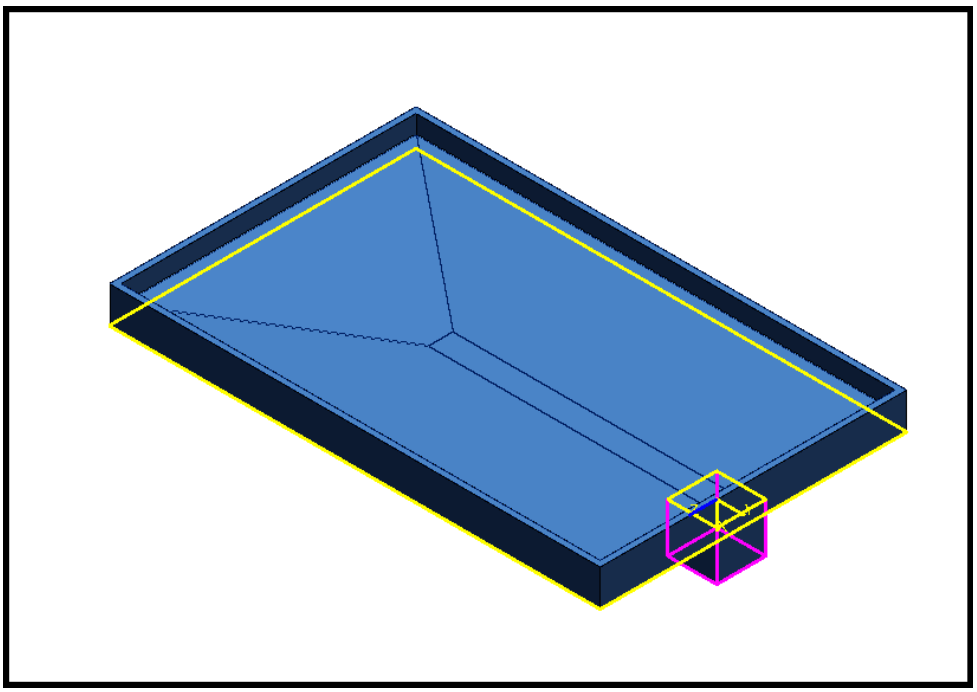


49. Select the bottom face of the *Base Concrete.* Your view should resemble the following graphic.

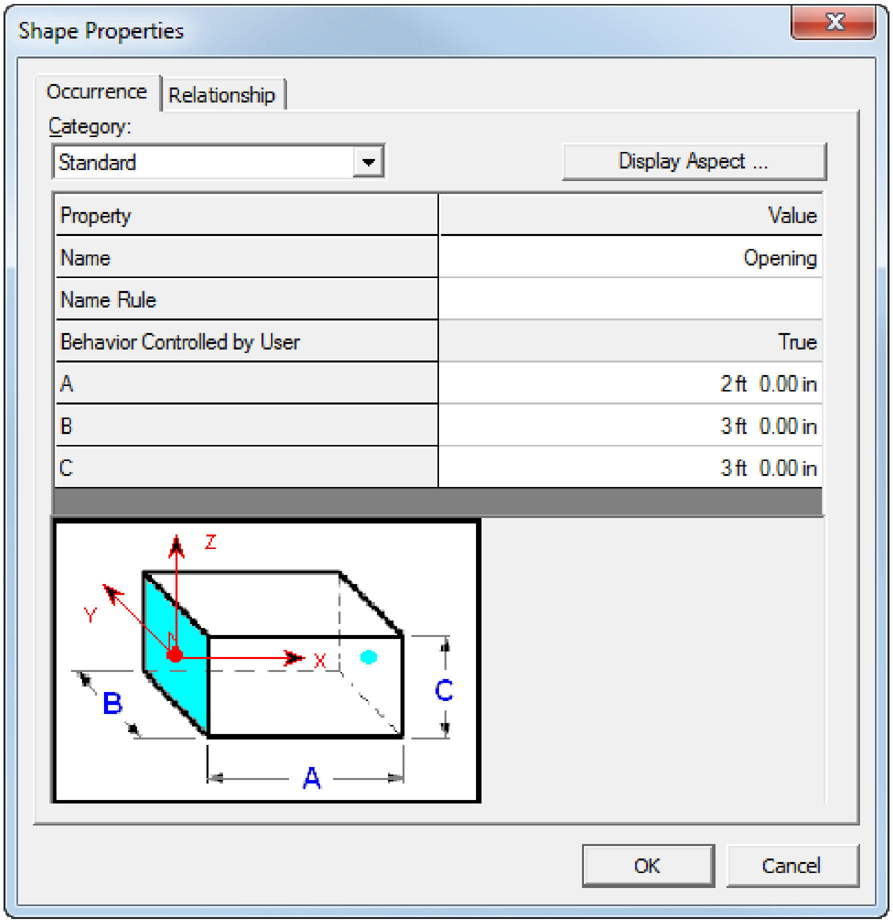
© Copyright 2014 Intergraph Corporation



50. Once the mate relationship has been applied, your view should resemble the following graphic.



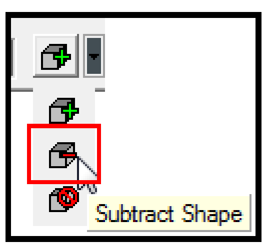
51. Change back to the Plan Plane view and start the Place Shape command again.
52. Select *RectangularSolid* from the Shapes dialog
53. Choose *SlopedSlab* as the parent of the Shape and make the following additions as shown in the image below.



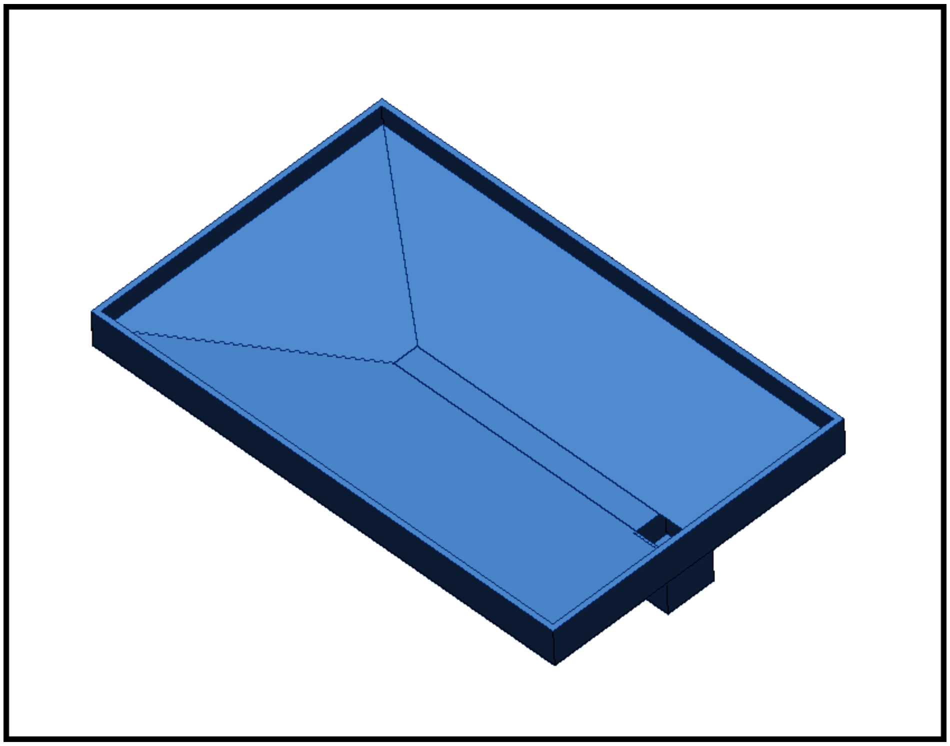
54. While looking in the Plan Plane, use the Up arrow keys (if needed) to change the axis of rotation and use the Left or Right arrow keys to rotate the shape until the point of placement is the center of the rectangle.

55. Place the Shape at E: -47 ft 6 in, N: -57 ft and EL: -2 ft 8 in.

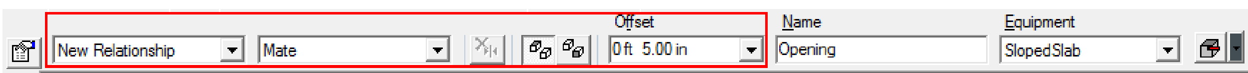
56. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.



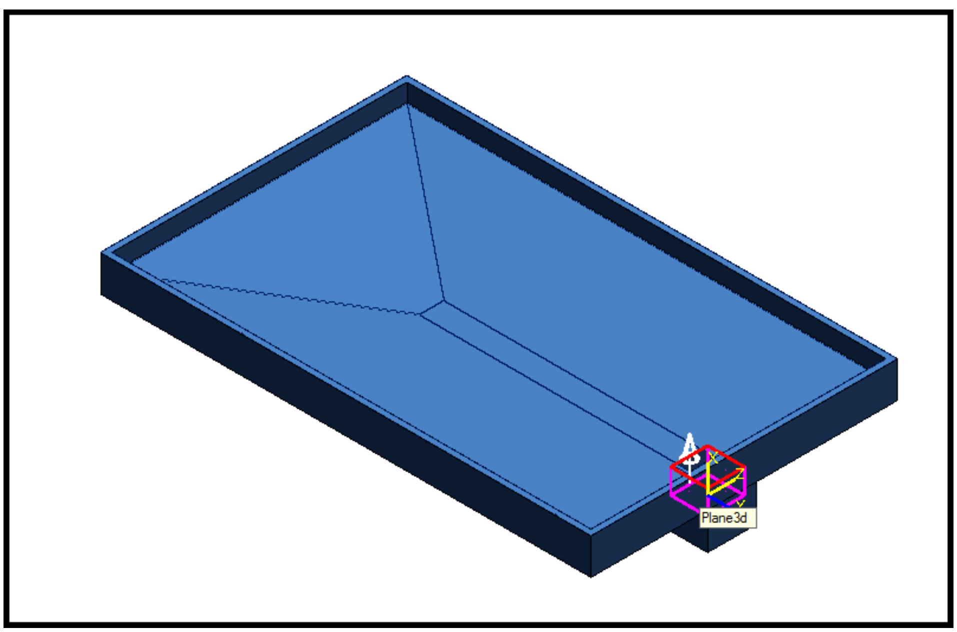
57. Change back to the Looking isometric view. Create a Connect Relationship at the point shown below and click to place the Shape. Your view should resemble the following graphic.



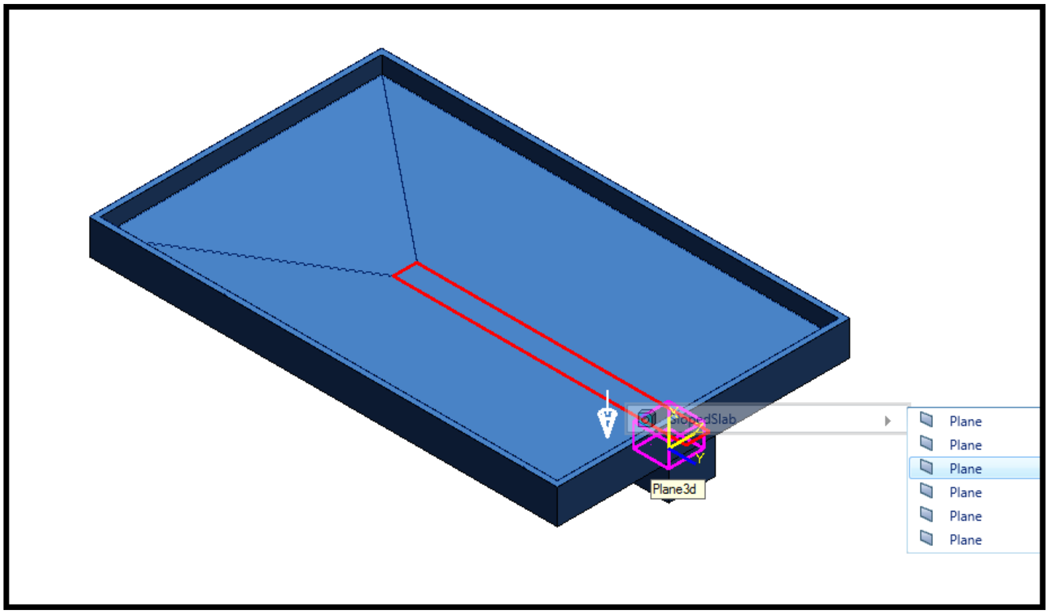
58. Select the opening and in the ribbon bar enter offset value as -0 ft 5 in.



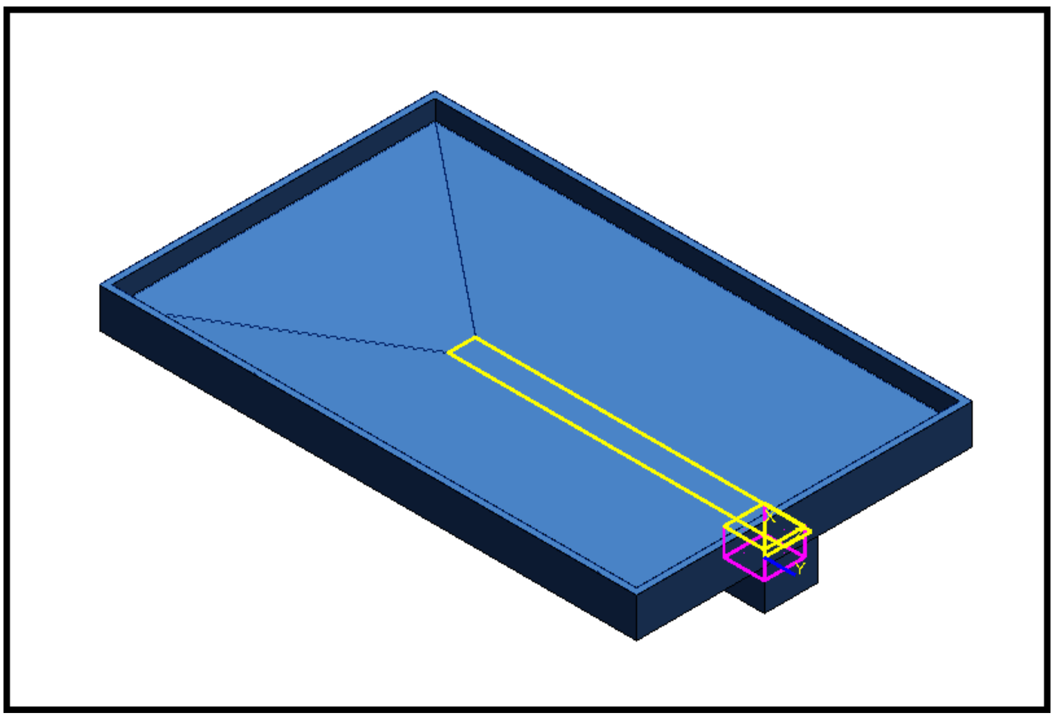
59. Select the top face of the *Opening,* select 2 from the quick pick. Your view should resemble the following graphic.



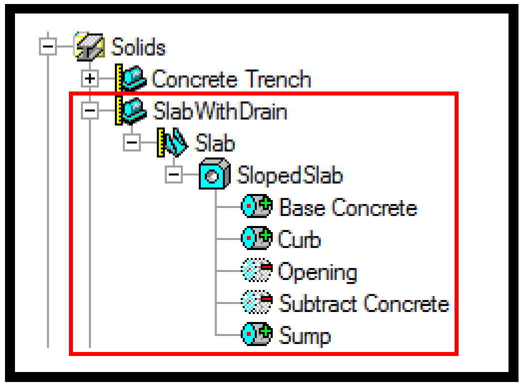
60. Select the surface of the *Subtract Concrete*. Your view should resemble the following graphic.



61. Once the mate relationship has been applied, your view should resemble the following graphic.



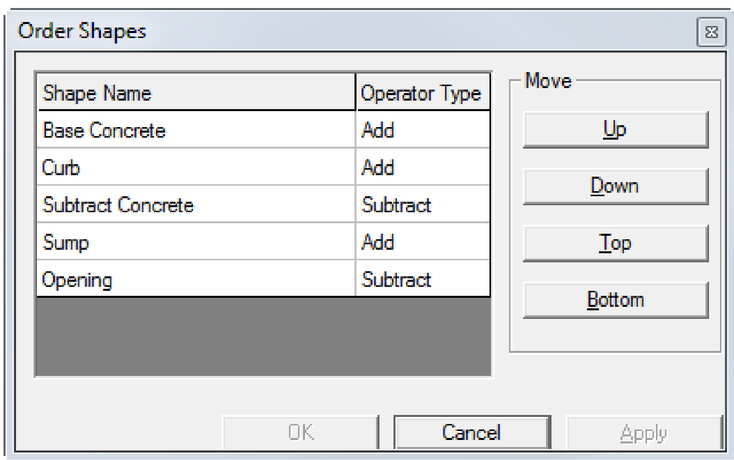
62. The finished hierarchy in Workspace Explorer should look like the following with the exception of the Shape Names.



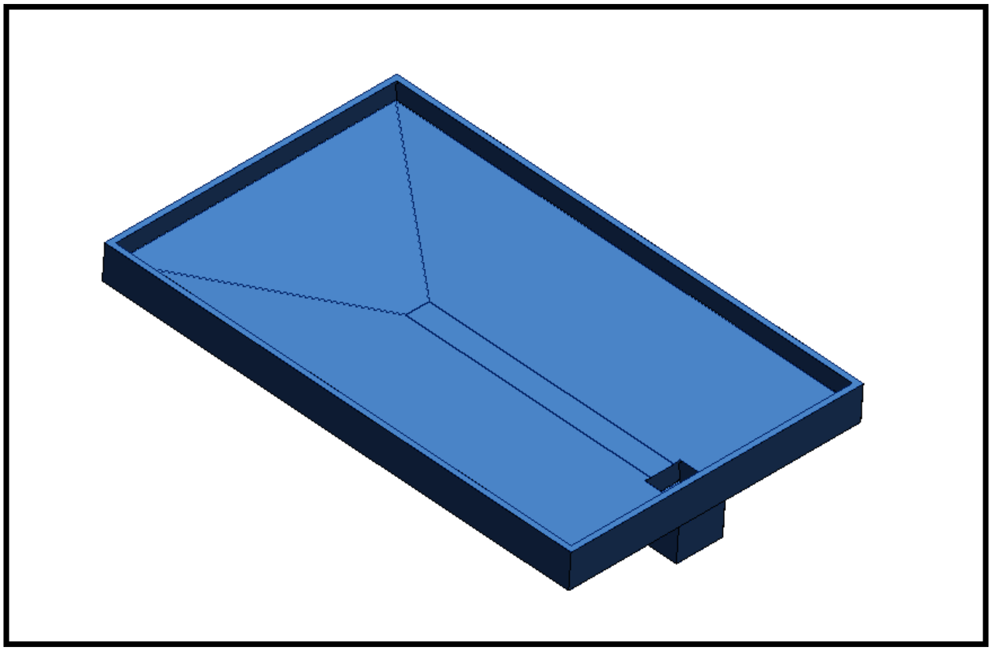
Order Shapes

63. Order Shapes controls the order in which the designed solid's shapes are processed by the software, which can be very important when a shape that cuts material from the designed solid overlaps a shape that adds material to the designed solid. The designed solid could look very different depending on which shape, the cut or the add, the software processes last. This dialog box is activated by Operators List  on the Modify Designed Solid ribbon.

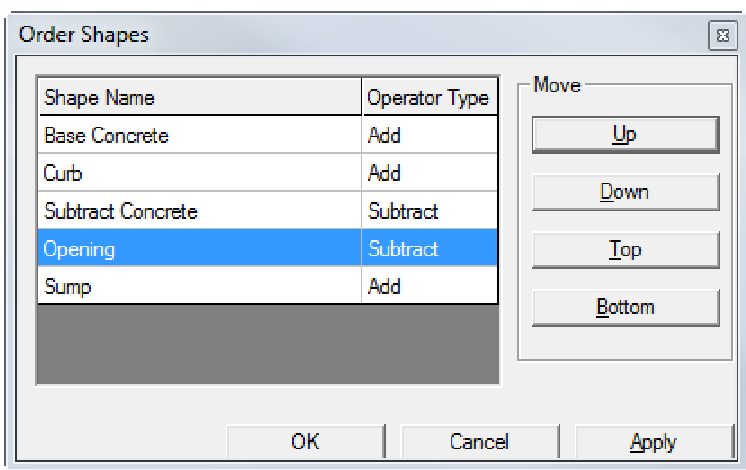
64. Select the Sloped Slab in the workspace hierarchy. Click on the Operators List and Order Shapes dialog box should appear. Your view should resemble the following graphic.



65. Your view should resemble the following graphic for the sloped slab.

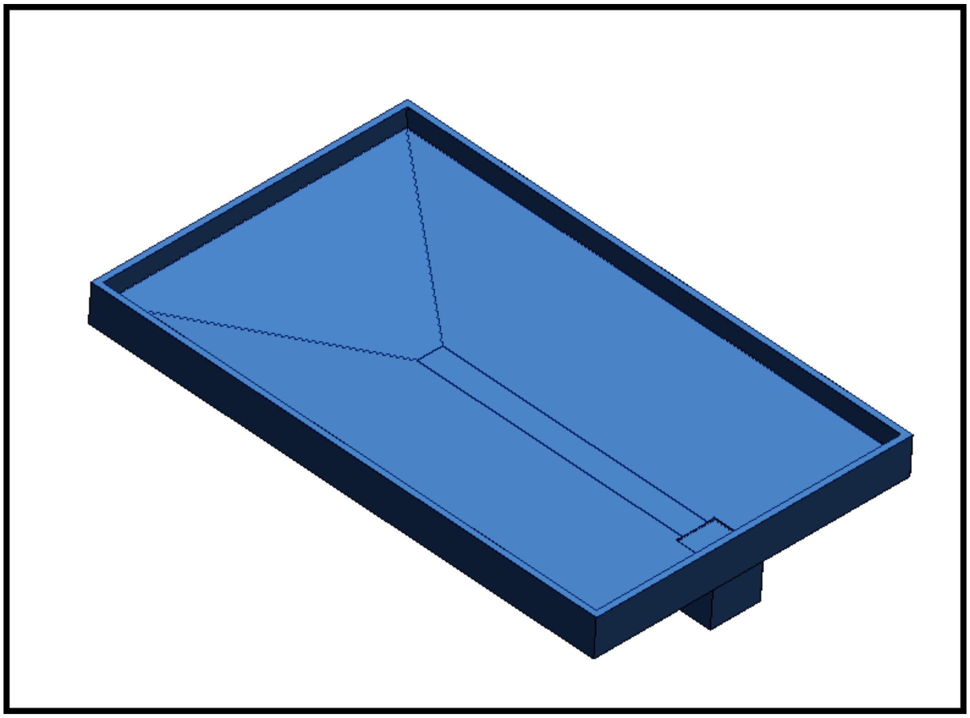


66. Select the Opening in the dialog box and hit the “Up”. Your view should resemble the following graphic.

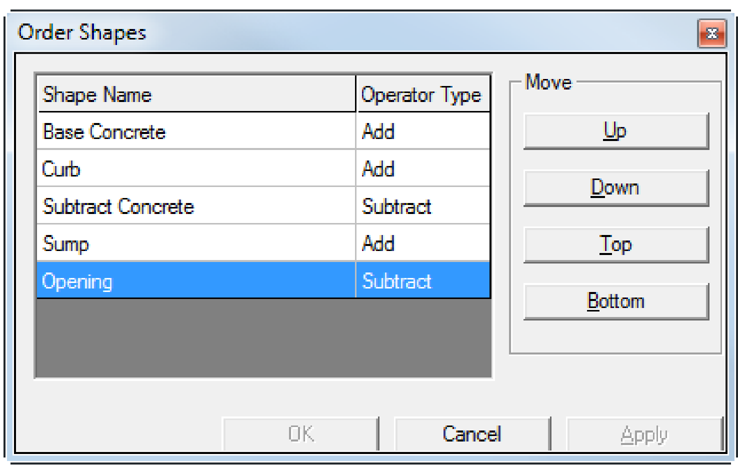


67. Click Apply. Click Cancel.

68. Your view should resemble the following graphic for the sloped slab.



69. Select the Opening in the dialog box and hit the “Down”. Your view should resemble the following graphic.



70. Click Apply. Click Cancel.

<u>Properties</u>

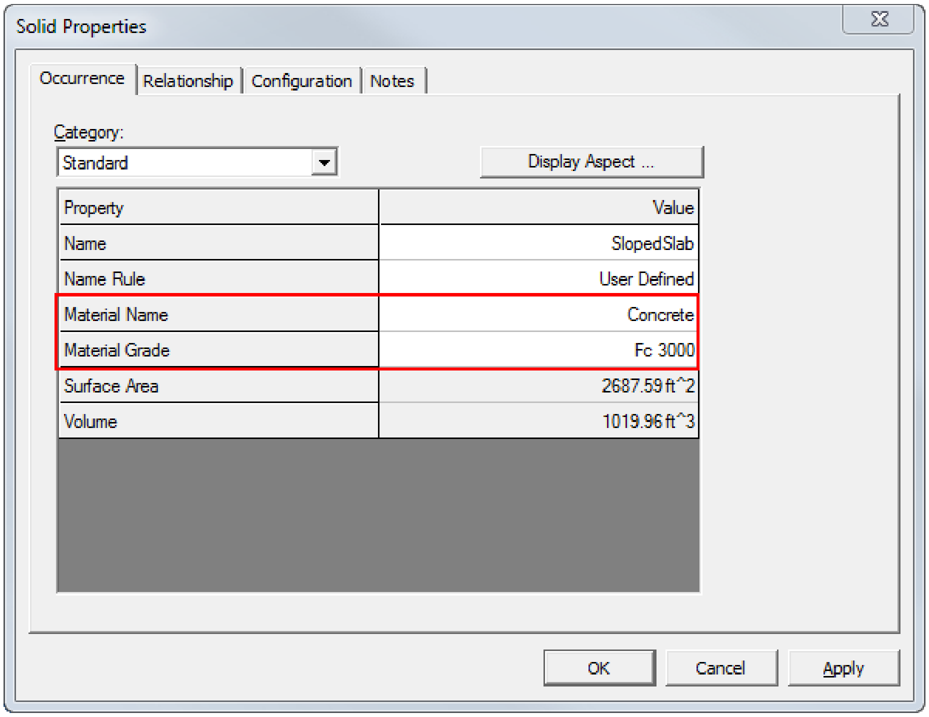
71. Now we can see the Surface Area, Volume, Weight and CG values for the *SlopedSlab*..

72. *SlopedSlab* with the *Sump* and without the *Sump* weight can be computed using suppress option.

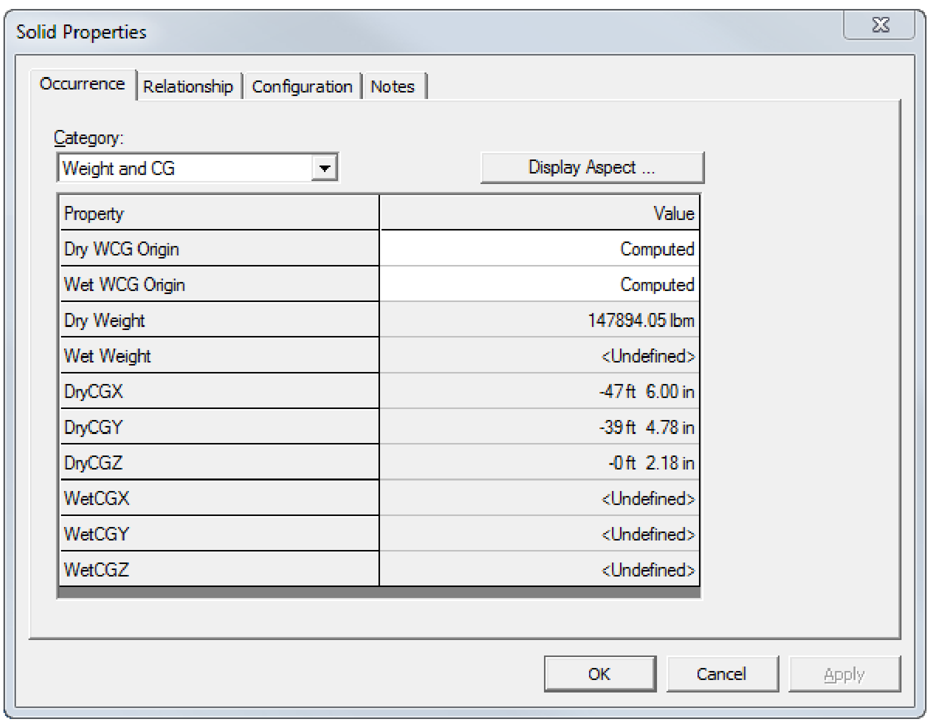
73. Select *SlopedSlab* in Workspace Explorer and go to the Properties page.

74. Change the Material name to Concrete and Material Grade to Fc 3000. Click Apply.

© Copyright 2014 Intergraph Corporation



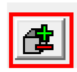
75. Change the Category dropdown to Weight and CG and set Dry WCG Origin to Computed. Click Apply.



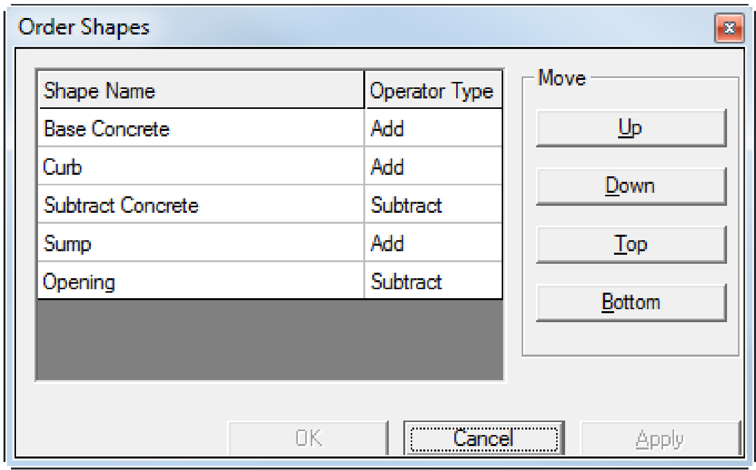
Suppress

76. Select *SlopedSlab* in Workspace Explorer.

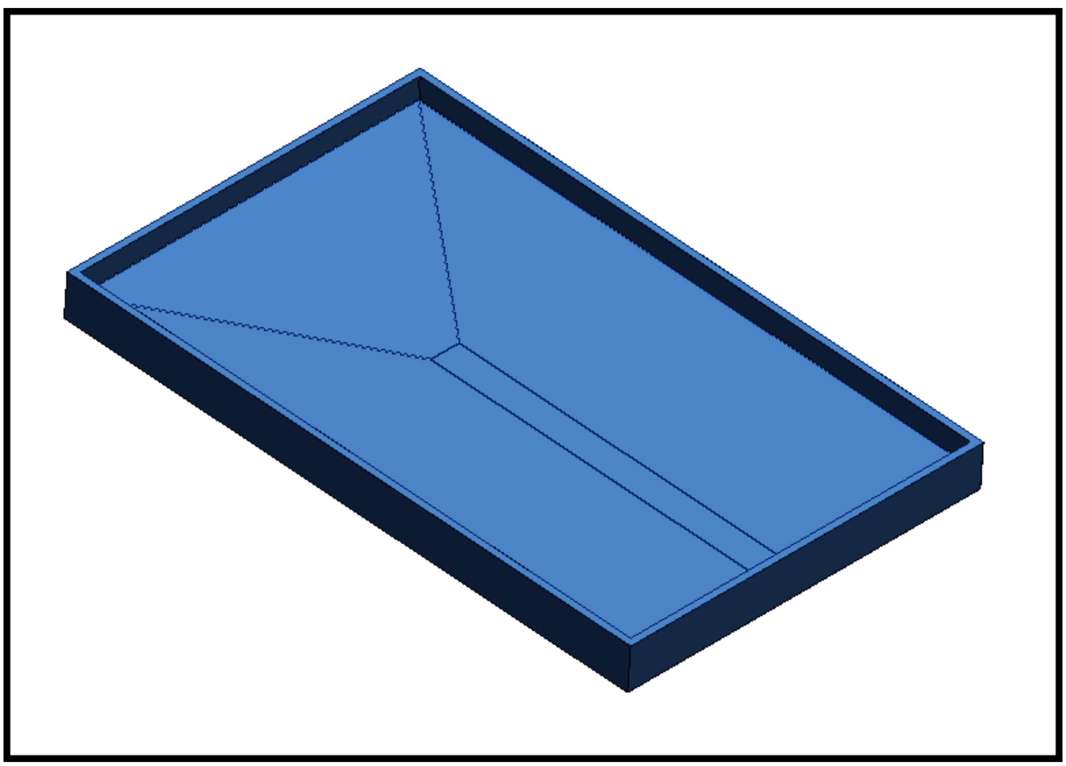
© Copyright 2014 Intergraph Corporation

77. In the ribbon bar, select the Operators List  icon.

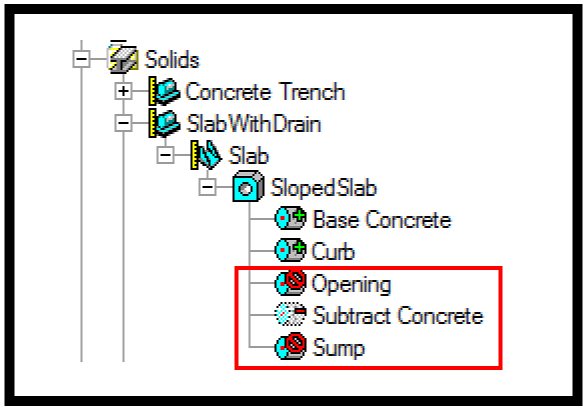
78. The Order Shapes dialog will appear. Notice that the order of the Shapes in the dialog is the same as the order in which the Shapes were placed.



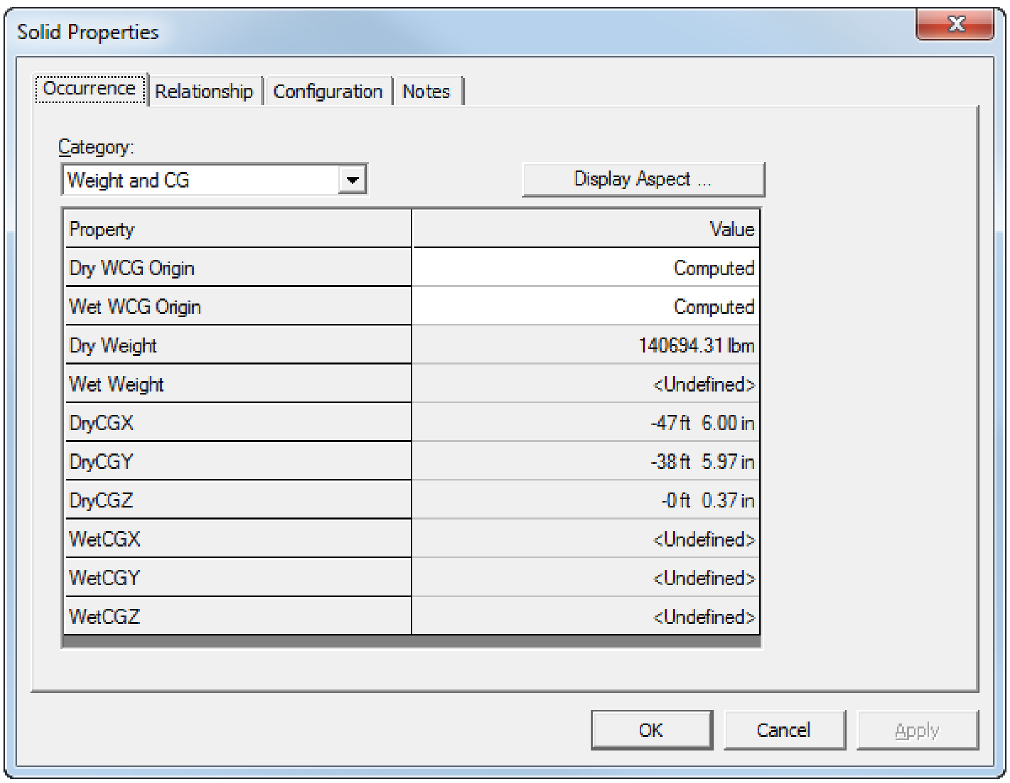
79. Change the operator type to Suppress for *Sump* and *Opening.* Click OK. Your view should resemble the following graphic.



80. Workspace hierarchy should resemble the following graphic.

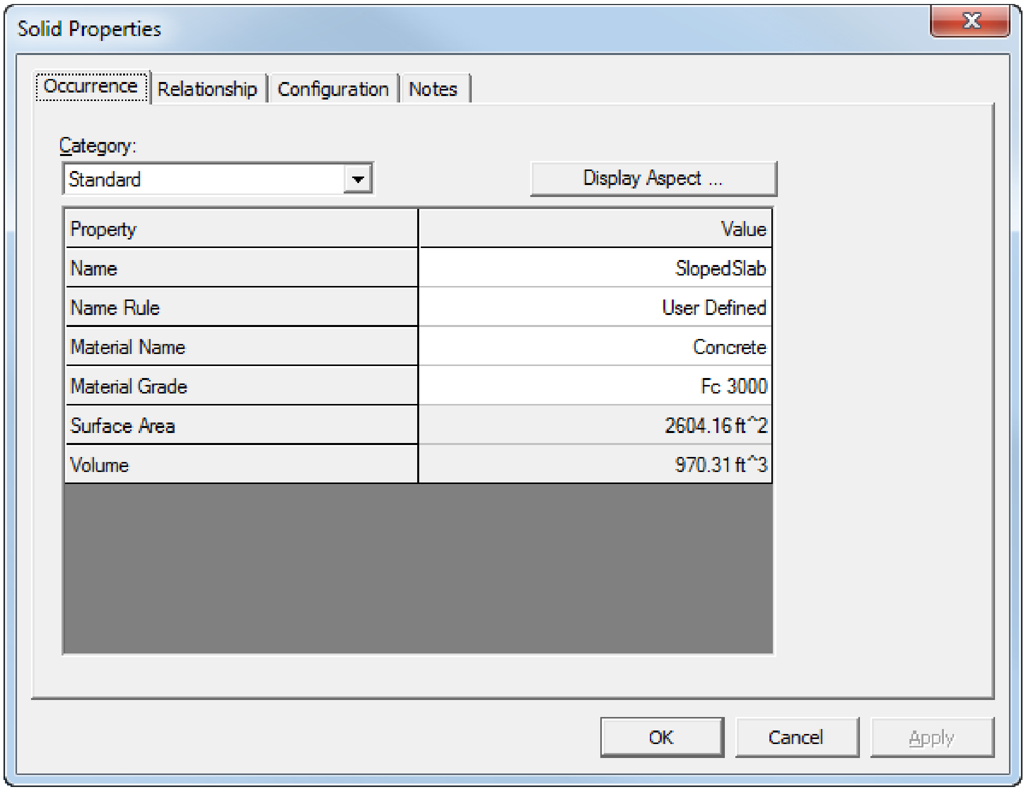


81. Select *SlopedSlab* in Workspace Explorer and go to the Properties page.
82. Change the Category dropdown to Weight and CG and set Dry WCG Origin to Computed. Click Apply.



83. Change the Category dropdown to Standard.

© Copyright 2014 Intergraph Corporation


Solid Properties
Occurrence
Relationship
Configuration
Notes
Category:
Standard
Display Aspect ...
Property
Value
Name
SlopedSlab
Name Rule
User Defined
Material Name
Concrete
Material Grade
Fc 3000
Surface Area
2604.16 ft^2
Volume
970.31 ft^3
OK
Cancel
Apply

LAB-24: Export Structure Model from SP3D in CIS/2 format

Objective

- After completing this lab you will be able to
- Export a physical structural model from SP3D in CIS/2 format.

Prerequisite Sessions

- SP3D Overview
- SP3D Common Sessions
- Structure Modeling: An Overview

Overview

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*

The STandard for the Exchange of Product model data (STEP - ISO 10303) provides a neutral computer-interpretable representation of product data throughout the life cycle of a product, independent of any particular system.

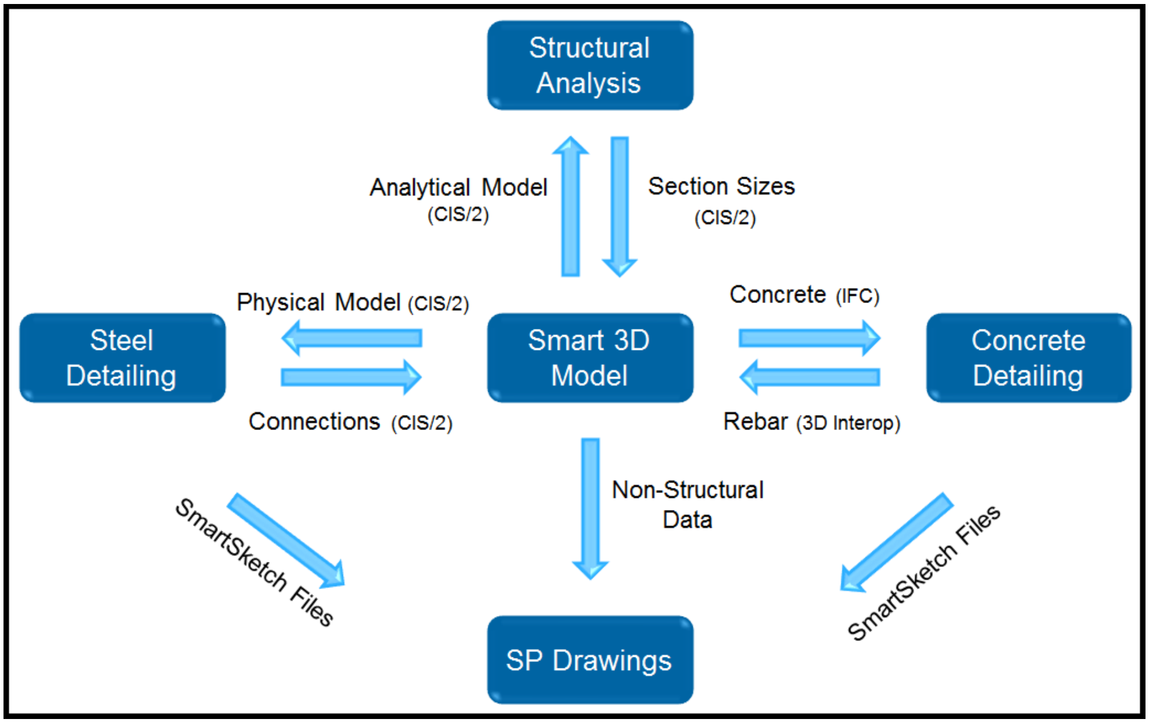
The CIMSteel Integration Standards (CIS/2) is the product model and electronic data exchange file format for structural steel project information. CIS/2 is intended to create a seamless and integrated flow of information among all parties of the steel supply chain involved in the construction of steel framed structures. It has been adopted by the American Institute of Steel Construction as their format for Electronic Data Interchange (EDI). CIMsteel stands for the Computer Integrated Manufacturing of Constructional Steelwork. The technical basis for CIS/2 is STEP (ISO 10301).

CIS/2 has been implemented as a file import or export capability by many steel design, analysis, engineering, fabrication, and construction software packages. A CIS/2 file exported by an analysis or design program could be imported into a detailing program to detail the connections.

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

© Copyright 2014 Intergraph Corporation



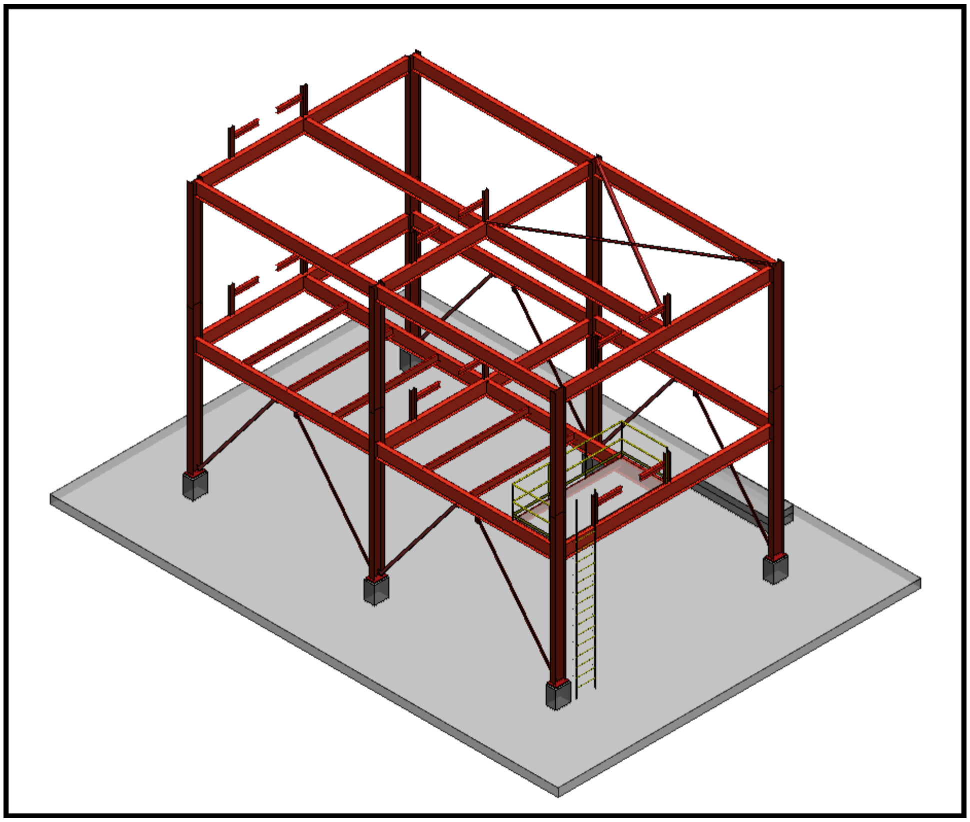
Define workspace in SP3D

1. Open a new session file and define your workspace by using the “U01_Structure” filter located under the Plant Filters →Training Filters node. Your workspace should include A2 →U01 → Structural systems.
2. Go to the Structure Task environment. Click on the Fit button on the common toolbar in order to fit all the objects into the graphic view.
3. Make sure the Active Permission Group is set to *Structural.*

Delete existing assembly connections

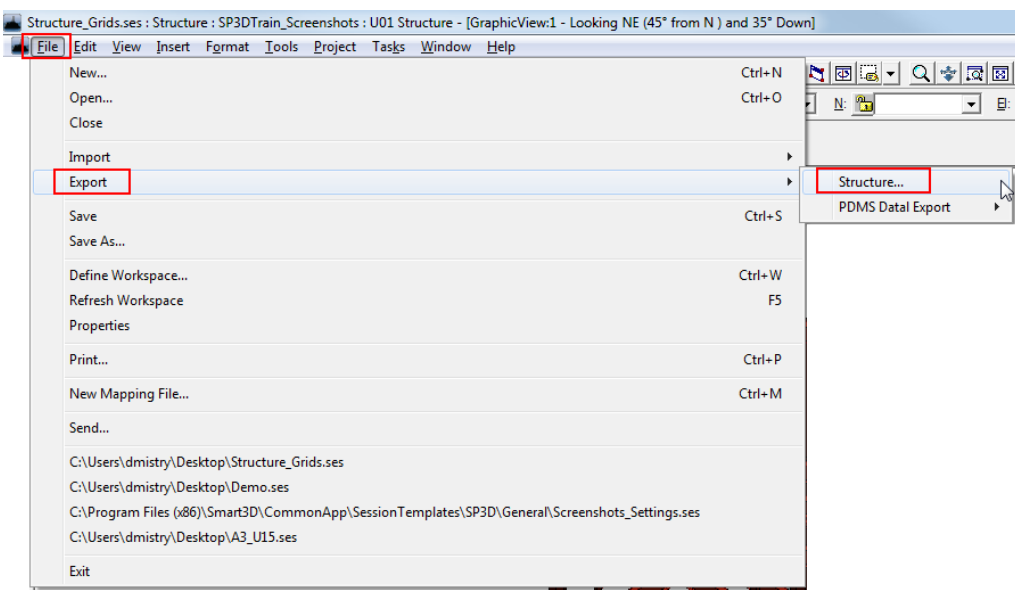
4. Go to Tools → Select By Filter… and select the “U01 Assembly Connections” filter located under the Plant Filters →Training Filters node. All the assembly connections placed in unit U01 will be selected.
5. Hit the Delete button to get rid of the assembly connections in the model.
6. Observe the structure in the workspace and note that the structural model does not have any connection parts at this point.

© Copyright 2014 Intergraph Corporation



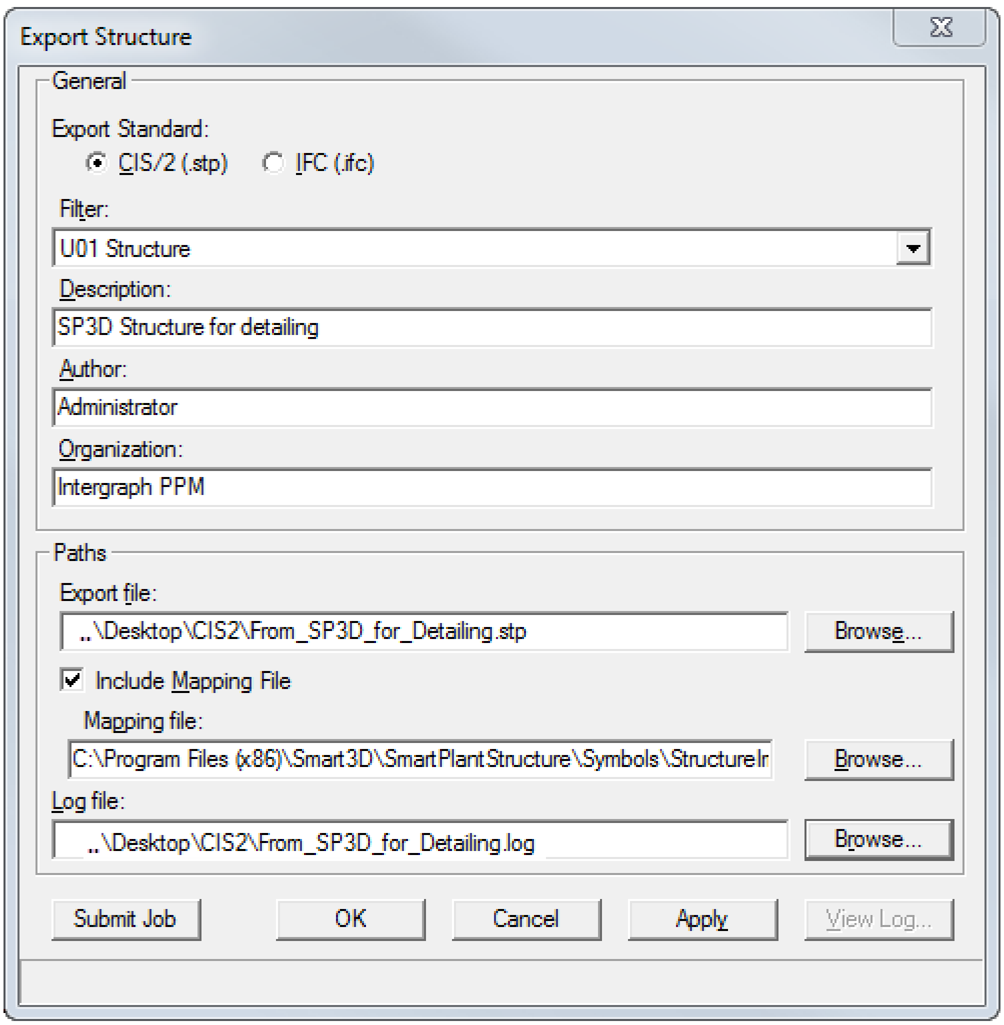
Export Structure from SP3D

7. Select File → Export → Structure…



8. On the Export Structure dialog box provide the following inputs;
   a. **Export Standard:** CIS/2 (.stp)

b. **Filter:** Select **More..** from the dropdown menu and specify the "U01_Structure" filter located under the Plant Filters →Training Filters node.
c. **Description:** SP3D Structure for detailing
d. **Author:** *Your Name*
e. **Organization:** *Your Company Name*
f. **Paths:**
   i. **Export File***: ..\Desktop\CIS2\From_SP3D_for_Detailing.stp*
   ii. **Include mapping file:** Yes. Check the box to indicate this.
   iii. **Mapping File:** *..ProductDirectory\3D\SmartPlantStructure\Symbols\StructureImportExportMaps\AISC_Master_Physical_Map.XML*
   iv. **Log File:** *..\Desktop\CIS2\From_SP3D_ for_Detailing.log*



9. After providing all the inputs on the Export Structure dialog box, click **Apply**. This will export the structure in CIS/2 format and a STP file will be created at the specified path.
10. Once the export is completed, click on **View Log** to review the log file. Close the log file after reviewing the information.
11. Click **Cancel** to close the Export Structure dialog box.

© Copyright 2014 Intergraph Corporation

LAB-25: Importing Detailed Structural Model in SP3D

Objective:

After completing this lab you will be able to

- Import CIS/2 structural model into SP3D.

Prerequisite Sessions:

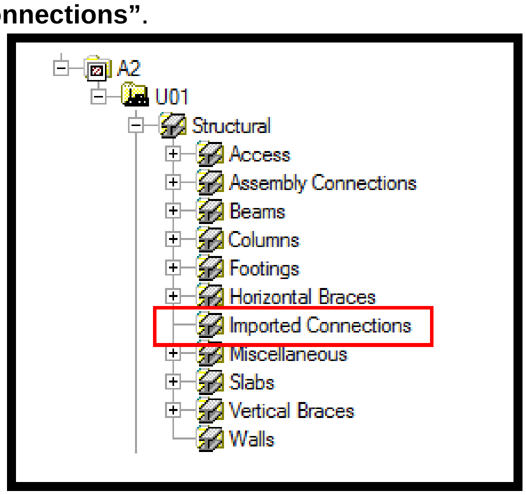
- SP3D Overview
- SP3D Common Sessions
- Structure Modeling: An Overview

Overview:

During this session we will go over the workflow to import a structure model in SP3D. The structural model was detailed in a 3rd party software and exported in CIS/2 format.

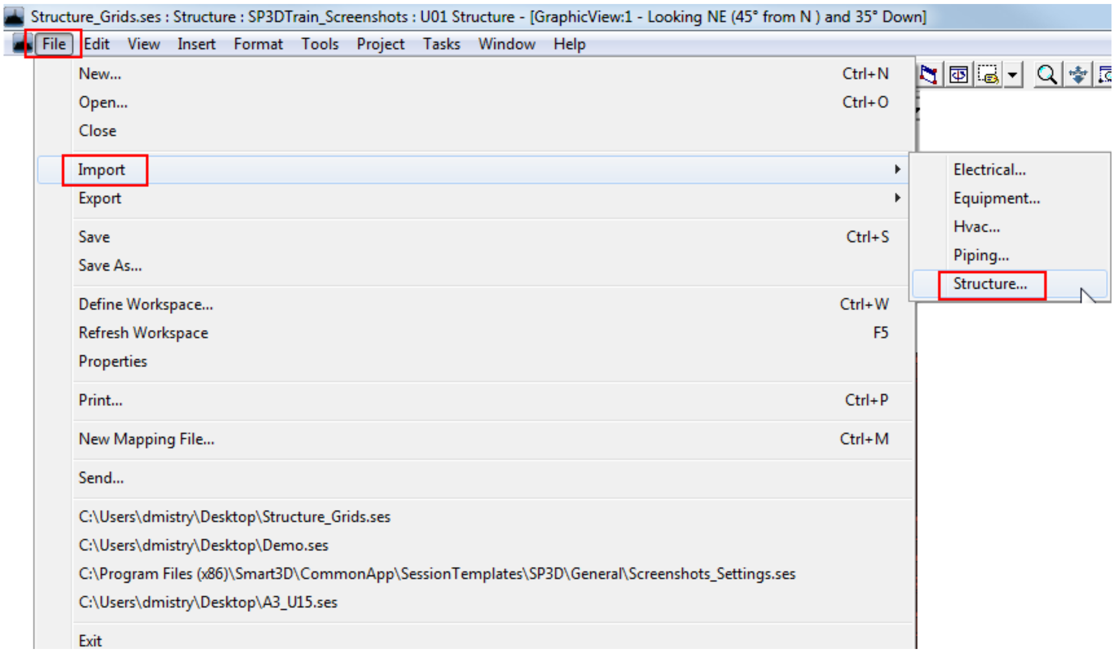
Import CIS/2 Model in SP3D:

1. Open a new session file or use an existing session and define your workspace by using the "U01_Structure" filter located under the Plant Filters →Training Filters node. Your workspace should include A2 →U01 → Structural system.
2. Go to the Structure Task environment. Click on the Fit button on the common toolbar in order to fit all the objects into the graphic view.
3. Make sure the Active Permission Group is set to *Structural.*
4. Observe the structure in the workspace and note that the structural model does not have any connection parts at this point.
5. On the workspace explorer, create a new structural system under Unit U01 → Structural and name it **"Imported Connections"**.

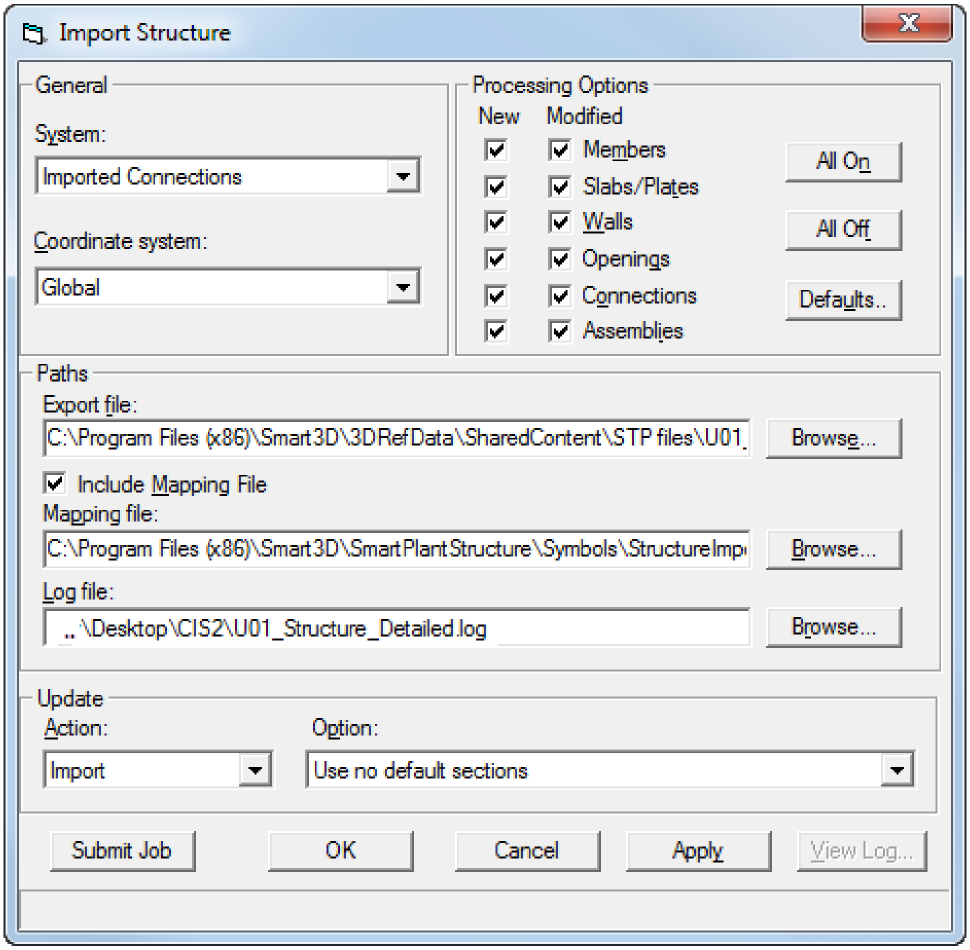


6. Select File → Import → Structure….

© Copyright 2014 Intergraph Corporation



7. On the Import Structure dialog box provide the following inputs;
    a. **System:** Imported Connections (A2→ U01 → Structural → Imported Connections)
    b. **Coordinate System:** Global.
    c. **Processing Options:** All checked.
    d. **Paths:**
        i. **Export File***: [Instructor will provide the path for the STP file] U01_Structure_Detailed.stp*
        ii. **Include mapping file:** Yes. Check the box to indicate this.
        iii. **Mapping File:** *..ProductDirectory\3D\SmartPlantStructure\Symbols\StructureImportExportMaps\AISC_Master_Physical_Map.XML*
        iv. **Log File:** *..\Desktop\CIS2\ U01_Structure_Detailed.log*
    e. **Update:**
        i. **Action:** Import
        ii. **Option:** Use no default sections



8. After providing all the inputs on the Import Structure dialog box, click **Apply**. This will start the import process.
9. Once the import is completed, click on **View Log** to review the log file. Close the log file after reviewing the information
10. Click **Cancel** to close the Import Structure dialog box.
11. You will be able to see the connections imported from the CIS/2 file in the graphics window. Zoom in on the member ends to see the imported connections as seen in the figure below;

LAB-26: New Mapping File

Objective:

After completing this lab you will be able to

- Create an XML mapping file in SP3D.
- Reference multiple mapping files in a single master XML file.

Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions

Overview:

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.

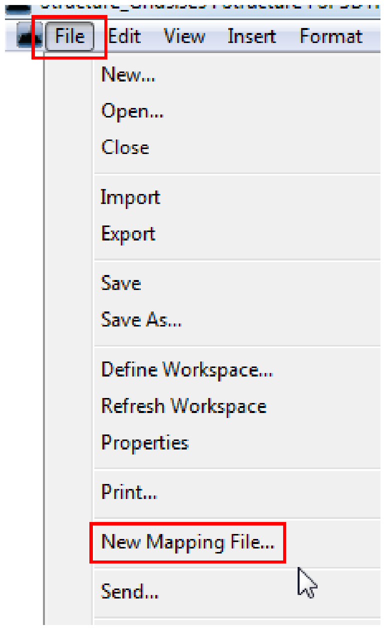
The mapping file created by this command is a template. The software does not write known-to-be-different section names to the mapping file. You are responsible for verifying, editing, updating, and maintaining the third-party application section names in the file.

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. 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.

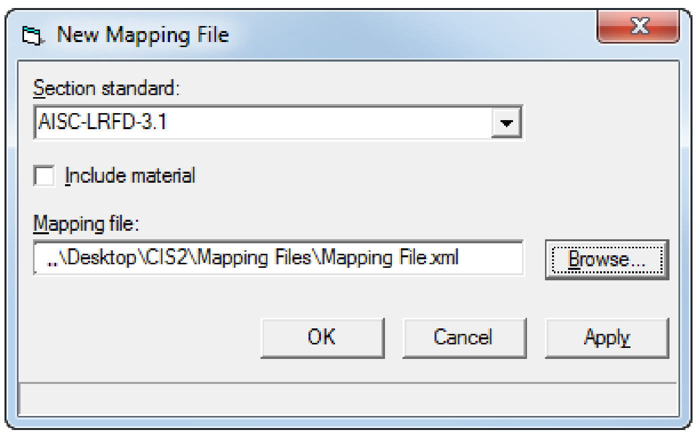
Create a Mapping File

1. Open a new SP3D session, if not already open, and define a workspace.
2. Click **File > New Mapping File**.

© Copyright 2014 Intergraph Corporation



3. In the **Section standard** box, select the section standard for the mapping file. Optionally, you can select **Include material** to write material names to the mapping file.



4. Click **Browse**, and then specify the following name and folder location for the mapping file.

   Name: Mapping_File

   Location: *..\Desktop\CIS2\Mapping Files*
5. Click **OK**.
6. Open windows explorer and browse to the location where the mapping file was saved.

   Note:You can 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.

```
<?xml version="1.0" encoding="windows-1252" ?>

<xml>

<!--

File:    C:\Users\Student\Desktop\CIS2\Mapping files\Mapping_File.xml

Description: This file provides a means to map sections from their section standard name to an alternate external section standard/name.

-->

<SectionStandard>
    <Standard name="AISC-LRFD-3.1" externalname="AISC-LRFD-3.1" externalorganization="AISC" externaldate="2003" externalversion="3.1" />
    <Sections>
        <Section name="2L5X3-1/2X1/2X3/8LLBB" externalname="2L5X3-1/2X1/2X3/8LLBB" />
        <Section name="2L4X3-1/2X3/8X3/4SLBB" externalname="2L4X3-1/2X3/8X3/4SLBB" />
        <Section name="2L5X3-1/2X1/2X3/4LLBB" externalname="2L5X3-1/2X1/2X3/4LLBB" />
        <Section name="2L4X3-1/2X5/16SLBB" externalname="2L4X3-1/2X5/16SLBB" />
        <Section name="2L5X3-1/2X3/8LLBB" externalname="2L5X3-1/2X3/8LLBB" />
        <Section name="2L4X3-1/2X5/16X3/8SLBB" externalname="2L4X3-1/2X5/16X3/8SLBB" />
        <Section name="2L5X3-1/2X3/8X3/8LLBB" externalname="2L5X3-1/2X3/8X3/8LLBB" />
        <Section name="2L4X3-1/2X5/16X3/4SLBB" externalname="2L4X3-1/2X5/16X3/4SLBB" />
```

7. Using Notepad++, open the AISC_Master_Physical_Map.XML located under the same folder.
8. Reference the Mapping_File.XML using the include statement as shown below;

AISC_Master_Physical_Map.XML

```
<?xml version="1.0" encoding="windows-1252" ?>

<xml>
<!--
Description: The master mapping file which may be used
-->
<IncludeXML href="AISC_Section_Map.XML" />
<IncludeXML href="AISC_Material_Map.XML" />
<IncludeXML href="Member_And_Slab_Type_Map.xml" />
<IncludeXML href="Mapping_File.xml" />
</xml>
```

9. Save the changes made to the AISC_Master_Physical_Map.XML.

© Copyright 2014 Intergraph Corporation

LAB-27: Exporting Concrete from SP3D using IFC

Objective:

After completing this lab you will be able to

- Export concrete objects from SP3D using Industry Foundation Class (IFC) format .

Prerequisite Sessions:

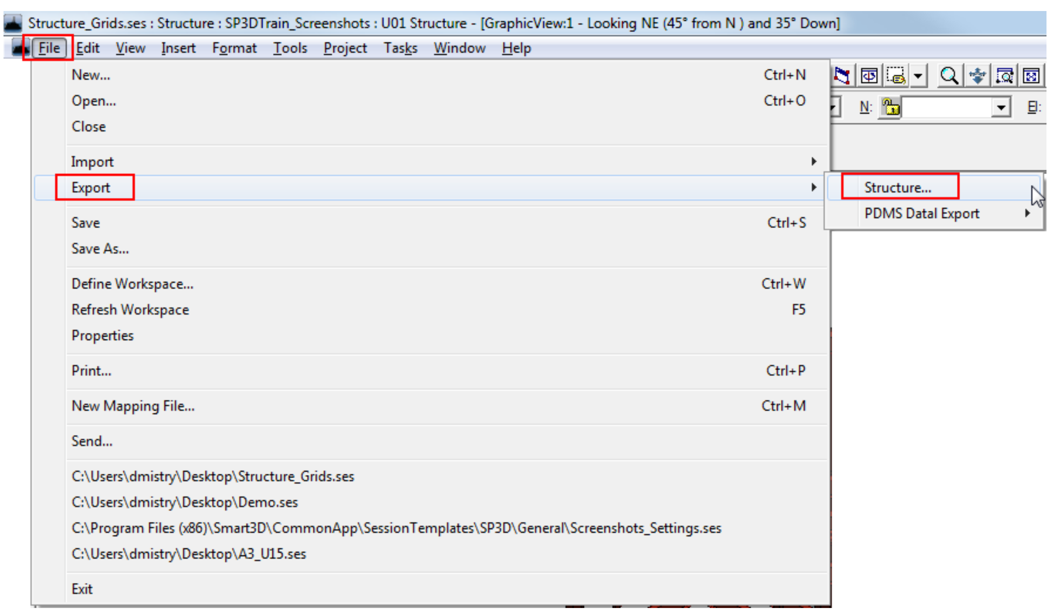
- SP3D Overview
- SP3D Common Sessions
- Structure Modeling: An Overview

Define workspace in SP3D

1. Open a new session file and define your workspace by using the “U02_Structure” filter located under the Plant Filters →Training Filters node. Your workspace should include A2 →U02 → Structural systems.
2. Go to the Structure Task environment. Click on the Fit button on the common toolbar in order to fit all the objects into the graphic view.
3. Make sure the Active Permission Group is set to *Structural.*

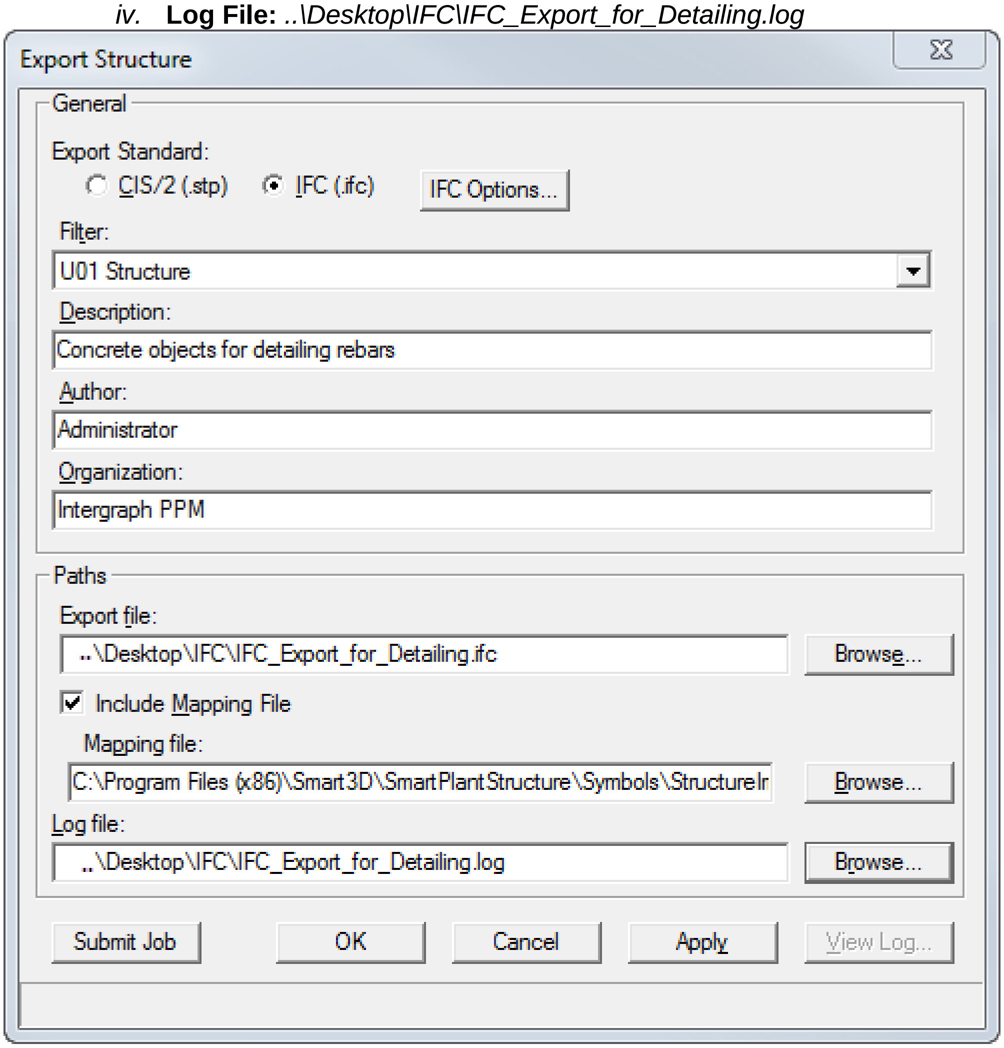
Export Concrete objects from SP3D

4. Select File → Export → Structure…



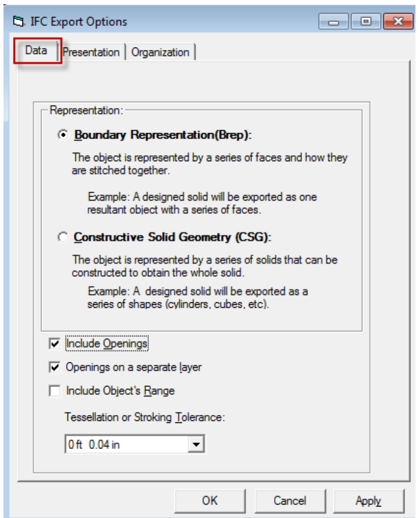
© Copyright 2014 Intergraph Corporation

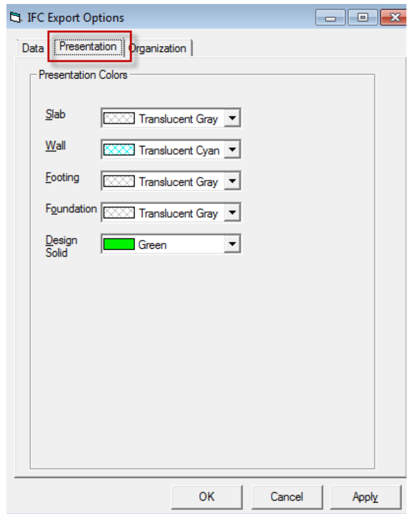
5. On the Export Structure dialog box provide the following inputs;
   a. **Export Standard:** IFC (.ifc)
   b. **Filter:** Select **More..** from the dropdown menu and specify the “U01 Structure” filter located under the Plant Filters →Training Filters node.
   c. **Description:** Concrete objects for detailing rebars
   d. **Author:** *Your Name*
   e. **Organization:** *Your Company Name*
   f. **Paths:**
      i. **Export File:** *..\Desktop\IFC\IFC_Export_for_Detailing.ifc*
      ii. **Include mapping file:** Yes. Check the box to indicate this.
      iii. **Mapping File:** *..ProductDirectory\3D\SmartPlantStructure\Symbols\StructureImportExportMaps \IFC_Master_Physical_Map.XML*
      iv. **Log File:** *..\Desktop\IFC\IFC_Export_for_Detailing.log*

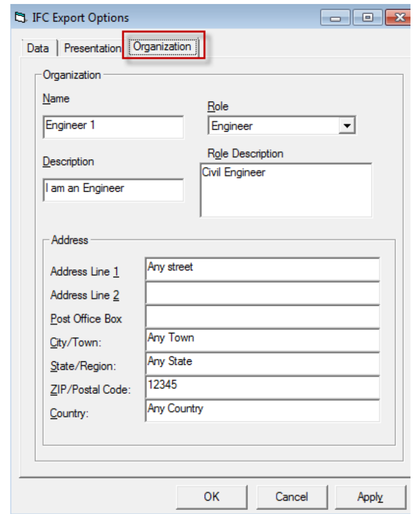


   g. **IFC Options… :**
      i. **Data Tab:** Specifies the properties for the format and accuracy of the data contained in the export file.
      ii. **Presentation Tab:** These properties specify how exported objects should render in other applications.

iii. **Organization Tab:** These properties give a brief description about the organization that owns the data and the IFC organization element**.**







6. After providing all the inputs on the Export Structure dialog box, click **Apply**. This will export the structure in IFC format and a file with an extension .ifc will be created at the specified path.
7. Once the export is completed, click on **View Log** to review the log file. Close the log file after reviewing the information
8. Click **Cancel** to close the Export Structure dialog box.

Note:

SmartPlant 3D provides limited support for IFC by exporting only concrete slabs, walls, structural systems, openings on slabs and walls, footings, equipment foundations, and design equipment (with an equipment classification of civil elements or structural elements) to the Industry Foundation Class (IFC) format. Linear and curved concrete members cannot be exported to IFC. We recommend the CIS/2 format for transferring linear and curved members.

© Copyright 2014 Intergraph Corporation

LAB-28: Loads, Releases, Boundary Conditions and Creating a CIS file

Objective

Students will be able to place dead and live distributed loads on beams. Place concentrated wind loads on one side of the structure. Define member end releases. Define member boundary conditions for the columns and create a CIS file.

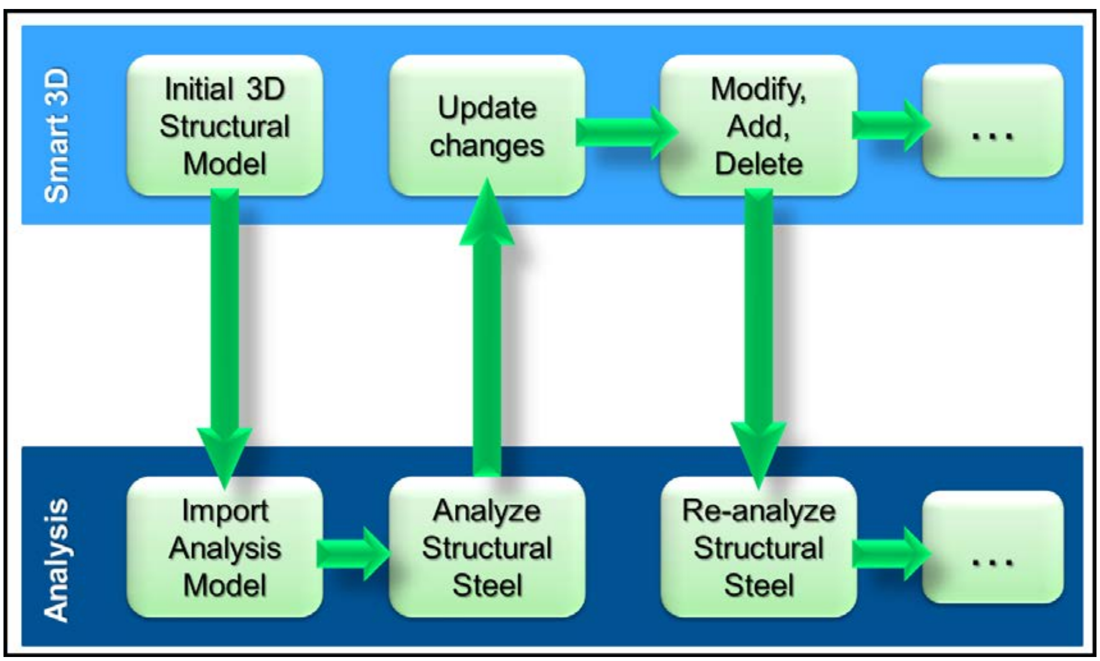
Prerequisite Sessions:

- SP3D Overview
- SP3D Common Sessions
- Structure: An Overview

Overview:

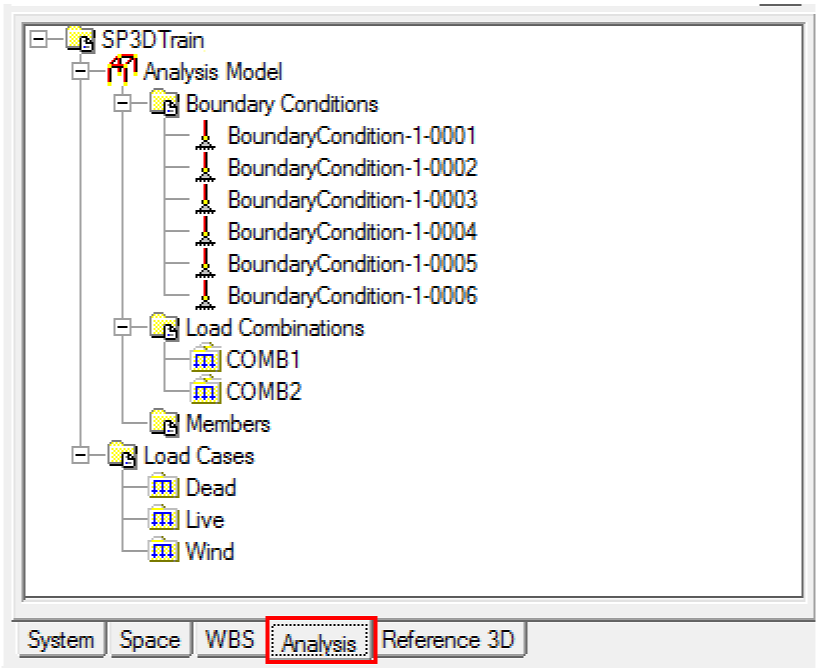
The **Structural Analysis** task allows you to create separate analysis models for all or portions of the physical model and export the model to third-party stress analysis programs. The analysis model is a non-graphical, logical grouping of boundary conditions, load combinations, and member parts. You send this information to an analysis program in a CIMsteel Integration Standard (CIS) file. A CIS file contains all the necessary data for a third-party program to create a finite element model. You can import section size changes defined in the stress analysis program, but all other editing of the structural model must be done within SP3D.

The analysis workflow is summarized in Figure below



The primary tasks involved in the structural analysis workflow are:

1. **Creating an analysis model -** All data to be analyzed in a structure is gathered and exported based on analysis models. You can define multiple analysis models. After a model is created, it is displayed on the **Analysis** tab in the Workspace Explorer. An analysis model has a fixed hierarchy of boundary conditions, load combinations, members, and load cases, as shown in Figure below.



Only member parts assigned to an analysis model are written to the CIS file. You can assign members to the analysis model by using a Standard Plant Filter. The analysis model filter is applied to the entire database, regardless of your current workspace definition.

When an analysis model is deleted, all load combinations and boundary conditions associated with the model are deleted. Member parts with loads are moved to the list of unassigned members in the Analysis hierarchy, until they are assigned to another analysis model. The member parts are not deleted from the model.

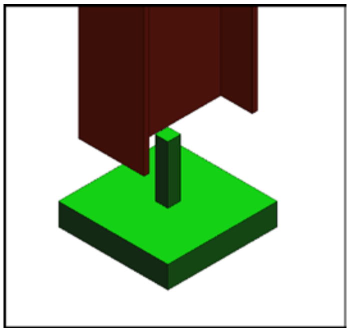
2. **Defining member releases -** Member releases are defined as properties on a member part and are relative to the local coordinate system of the member. Member releases remain the same for all analytical models that include the specific member part. You can set the release properties when you place a member by using the **Properties or Settings** command on the **Place Linear Member** ribbon. The default properties for the next member placement would be the last used properties. You can select multiple member parts and edit the releases with single edit.

Note:

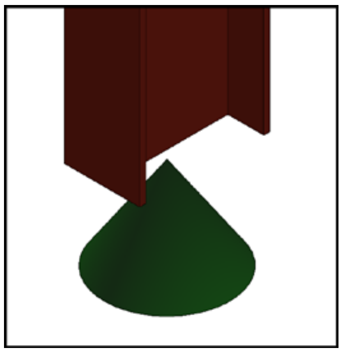
- *You can create a **Tool Tip** that displays the member releases when the cursor is positioned over the member.*

3. **Defining boundary conditions to be included in the analysis model** – Boundary conditions are defined for each analytical model. It sets six degrees of freedom—X, Y, Z,

RX, RY, and RZ—relative to the global coordinate system. A unique indicator displayed at the end of each member indicates the type of boundary condition. Indicators for fully fixed and RX, RY, and RZ fixed supports resemble Figure below.



Indicators for supports where RX, RY, RZ, or a combination of the three moment directions are released resemble Figure below.



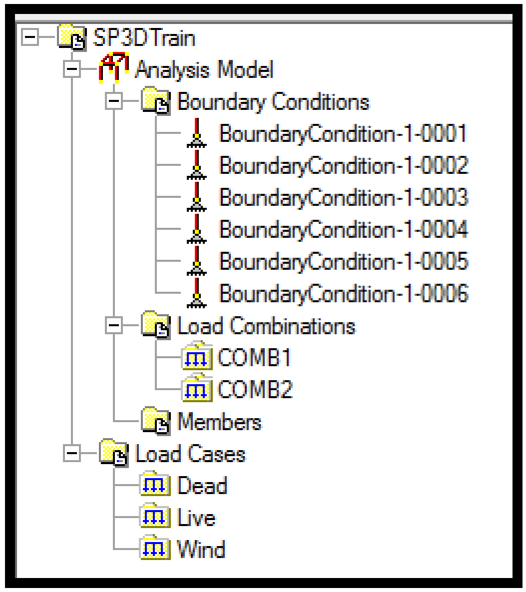
Tip:

- *You can create a **Tool Tip** to display the details of the boundary condition releases.*

4. **Defining load cases, load combinations, and loads to be included in an analysis model** – A load case is a grouping of loads that have the same source category and source type. The default Catalog provides the common source category and source type identifications. A load combination is a grouping of load cases with the assigned multiplication factor for each load case. Stress results for the structure can be calculated for each load combination. Load cases and load combinations appear in the Workspace Explorer under the analysis model, as shown in Figure below.

   Concentrated or distributed loads are placed on physical members. You can place multiple loads on a member. These loads appear below the member on which they are placed, as shown in Figure below. The **Load Placement** or **Modification** ribbons allow you to specify the load magnitudes and choose the load type, case, and direction. After loads are placed, you can edit any property, except the load type (which can be concentrated or distributed). You can also delete any individual load, if required.

© Copyright 2014 Intergraph Corporation



5. **Creating a CIS file** – You need to create a CIS file to be analyzed and designed by a third-party application by selecting the **File → Export → Analytical Model** command. The export uses a mapping file to create a CIS file with the section and material names that the third-party analysis application will understand.

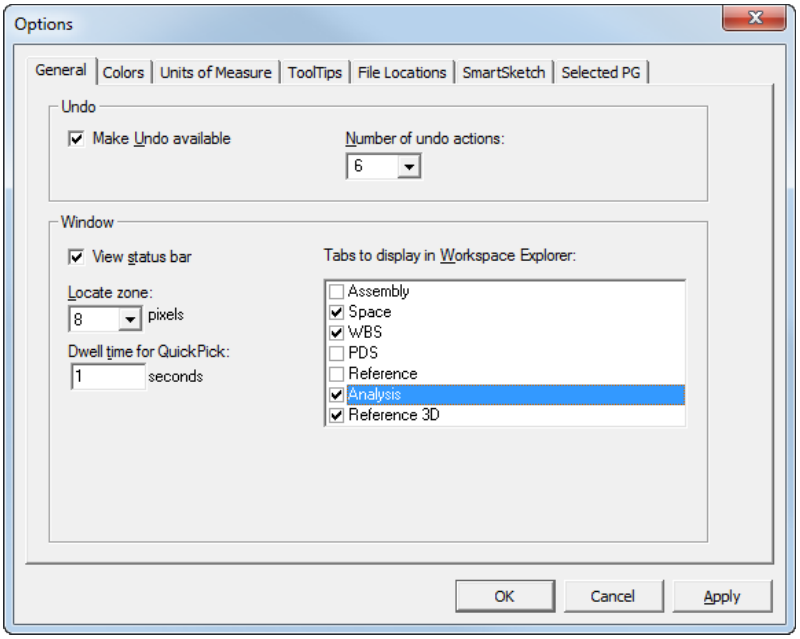
6. **Updating member section sizes:** You update member section sizes in the structure, based on the third-party application analysis by using the **Update from Analysis/Design** command on the vertical toolbar. Importing will only update the section sizes.

Part I: Create Load Cases and Combinations

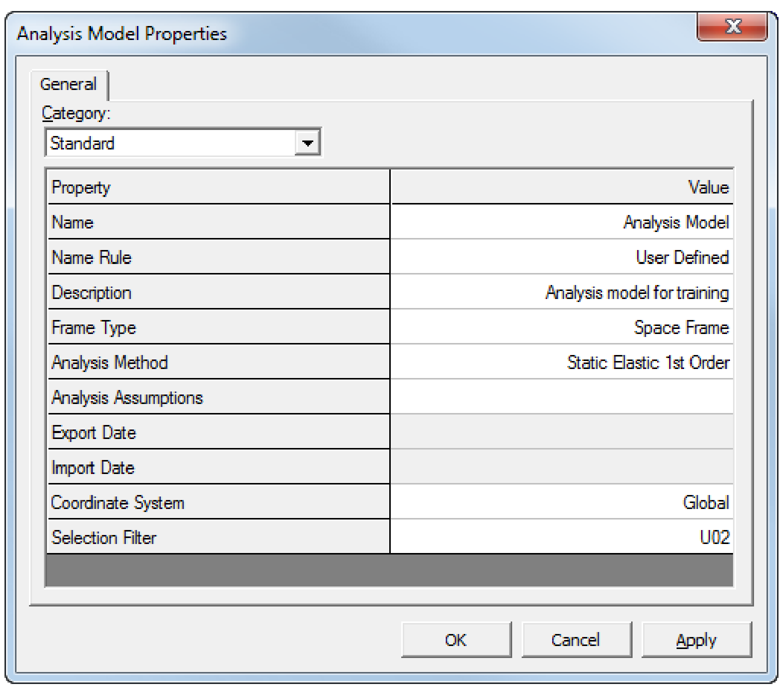
1. Open or create a session file and define an appropriate filter for your workspace that includes Grids, Coordinate System, Structural Steel, Frame Connections and the Analysis model for U02 structure.

2. Go to Tools → Option and check the Analysis.

© Copyright 2014 Intergraph Corporation



3. Click “OK” button. Save your session.
4. Exit your session and Open it again. Notice the Analysis Tab in the Workspace Explorer.
5. Go to the Structural Analysis Task.
6. Create an Analysis Model using New Analysis Model Command 



7. The **Analysis Model Properties** dialog box appears, as shown in Figure above. Specify the following values in the Analysis model properties:

© Copyright 2014 Intergraph Corporation

Name:	Analysis Model
Description:	Analysis model for training
Frame Type:	Space Frame
Analysis Method:	Static Elastic 1^{st} Order
Coordinate System:	Global
Selection Filter:	Plant Filter → Training Filters → U02

Note:

a. *In the **Name** field, specify a name for the analysis model. If you do not specify a name, SP3D generates a name based on default naming rule.*

b. *In the **Description** field, specify a description for the analysis model.*

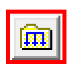
c. *In the **Frame Type** field, select a frame type. The options that appear for this property are defined in the Catalog and can be customized by the Catalog administrator.*

d. *In the **Analysis Method** field, select an analysis method. The options that appear for this property are defined in the Catalog and can be customized by the Catalog administrator.*

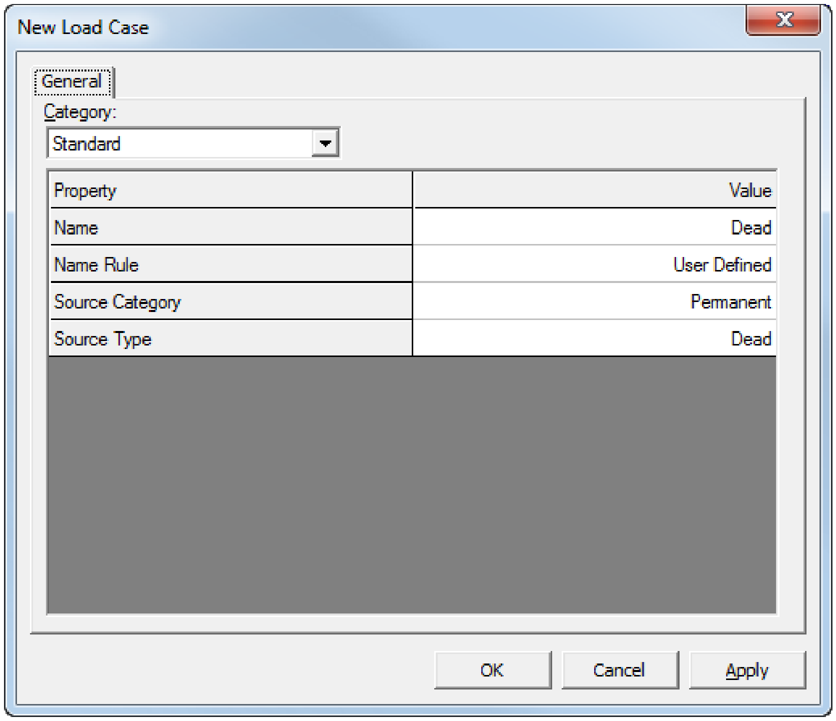
e. *In the **Coordinate System** field, select a coordinate system. Coordinates written to the CIS/2 file will be relative to this coordinate system.*

f. *In the **Selection Filter** field, select a filter that identifies the members to include in the analysis model. You can either create a new filter or select an existing one. Typically, you will select a filter that is stored under the Plant Filters rather than using a personal filter.*

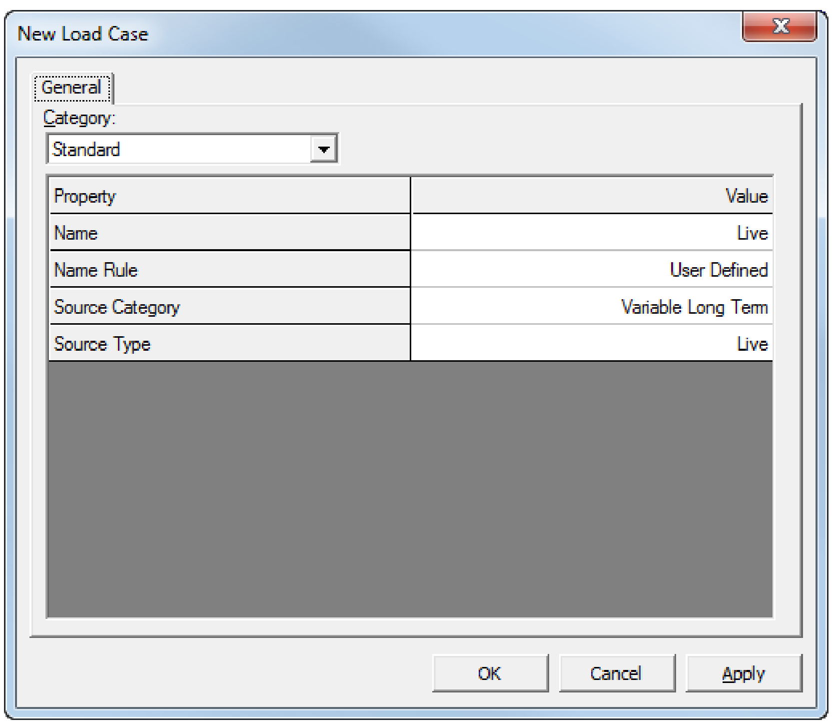
8. Click “OK” Button.

9. Select Create New Load Case command. 

10. Create a DEAD load using New Load Case Command.



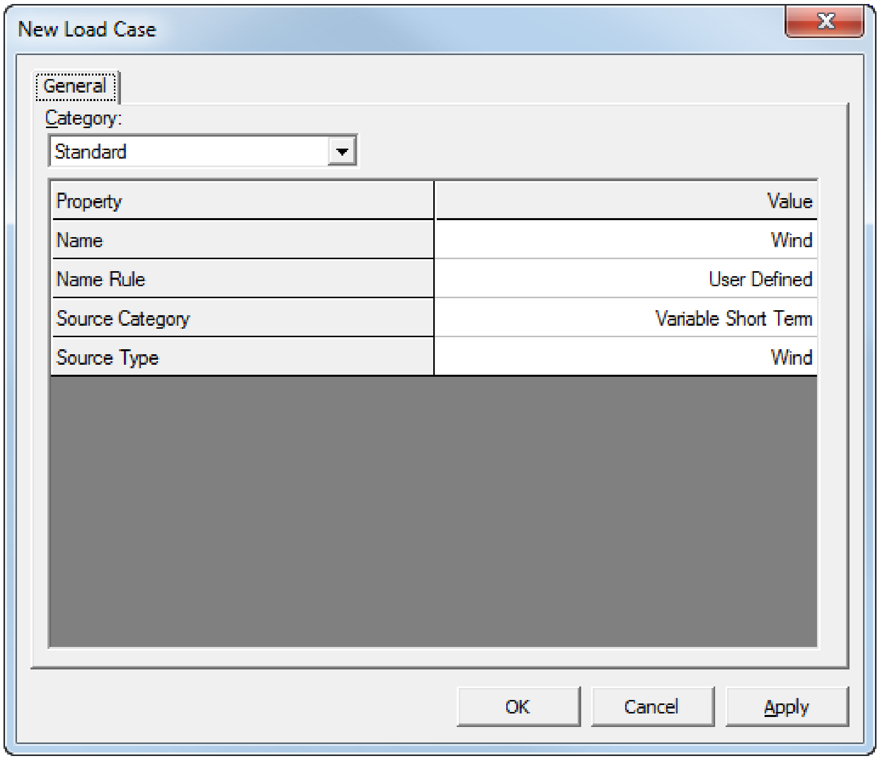
10. Click “Apply” Button.
11. Create a LIVE load.



12. Click “Apply” Button.

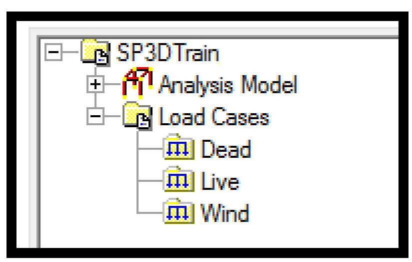
© Copyright 2014 Intergraph Corporation

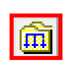
13. Create a WIND load.



14. Click “OK” Button.

15. Switch to the Analysis tab in the workspace explorer to see the Analysis model and Load cases that were created.

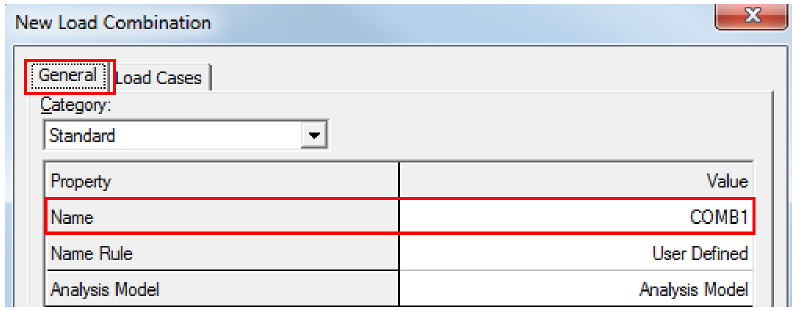


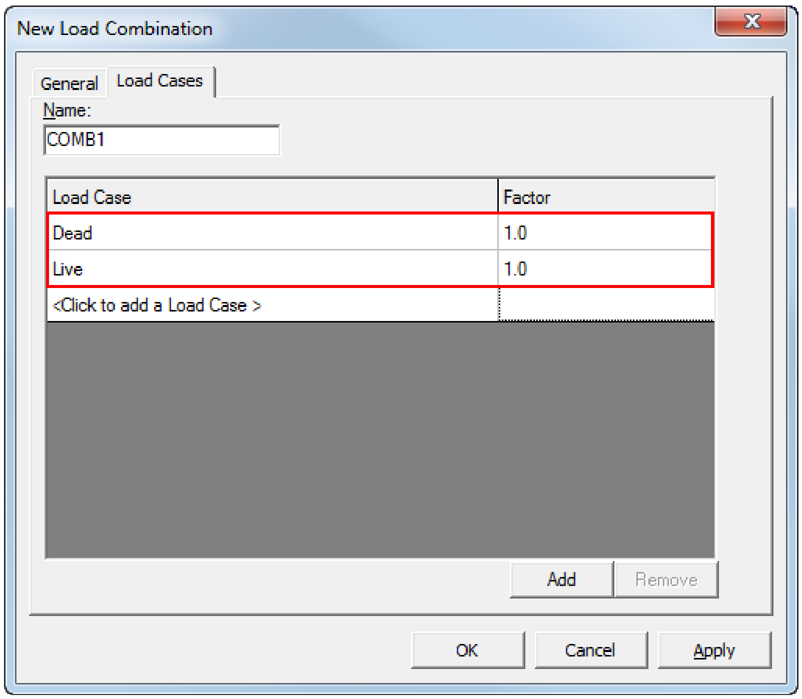
16. Create Load Combinations using New Load Combination Command 

COMB1: Dead + Live with the Factor as 1.0

17. Set the parameter as follows:

© Copyright 2014 Intergraph Corporation

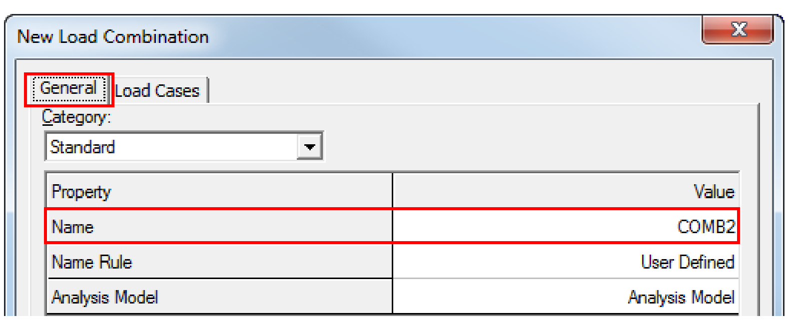


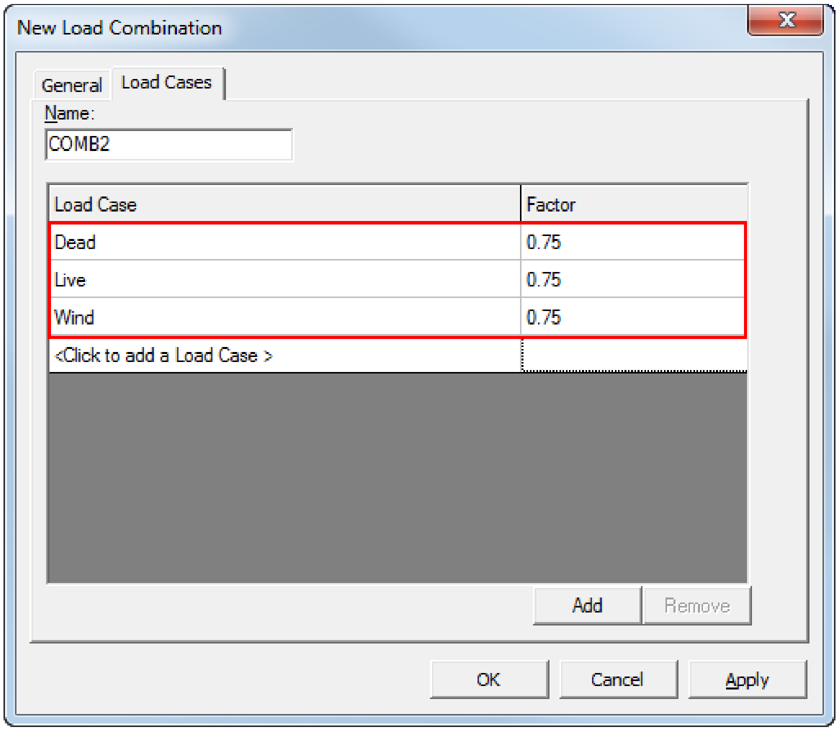


18. Click "Apply" Button.

19. Create Load Combinations. 

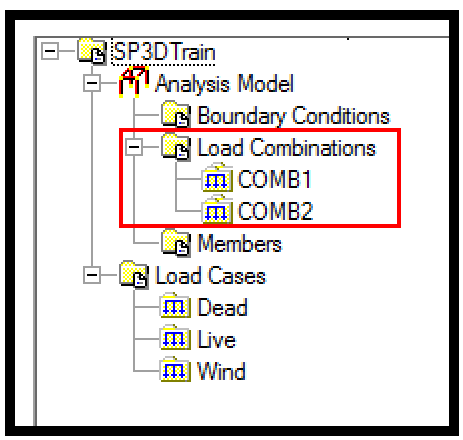
COMB2: Dead + Live + Wind with the Factor of 0.75





19. Click "OK" Button.

The Workspace Explorer now includes the Load Cases and Combinations created above on the Analysis Tab.

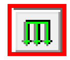


Part II: Place Live and Dead Loads

After creating the load cases and load combinations for your analysis model, you place loads on the model. You can place two types of loads:

a. Concentrated load - A concentrated load is applied at a user-defined location along the member's length.

b. Uniformly distributed load - A uniformly distributed load is applied along the entire member length or between two points on the member. The load per unit length can be constant or can vary linearly from the start to the end.

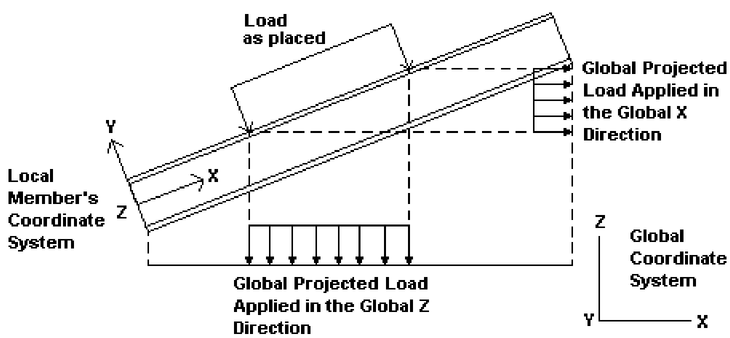
20. Select Place New Distributed Load Command. 

21. Define the following load for the beams at Elevation 18'- 0":

Load Case: DEAD
Load: Force Z
Reference: Global
Position as: Relative
Start Position: 0
End Position: 1
Start and End Magnitude: -0.65 kpf

Notes:

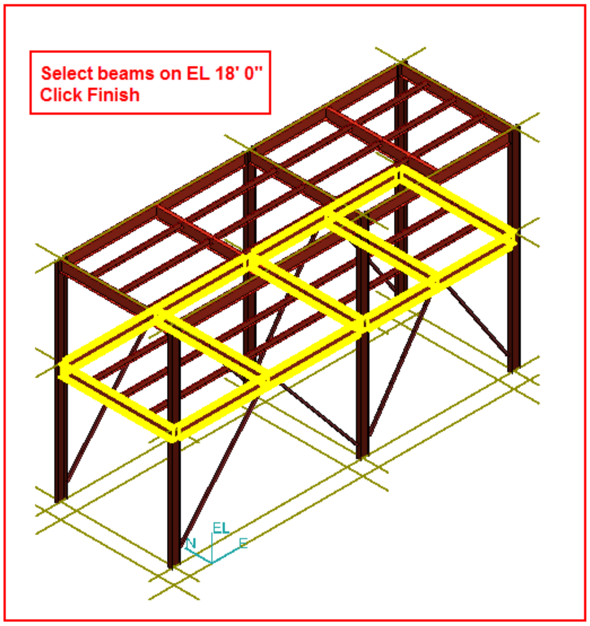
- *The **Load** field defines the direction of the load using the reference specified in the **Reference** field. The **Reference field** has three options: **Local**, **Global**, and **Projected Global**. **Local** option is relative to the member's local coordinate system. **Global** option is relative to the model's global coordinate system (not the active coordinate system). The **Projected Global** load option applies the load to the projection of the member on the plane perpendicular to the global direction specified and acting in the direction specified as shown in Figure 15.*

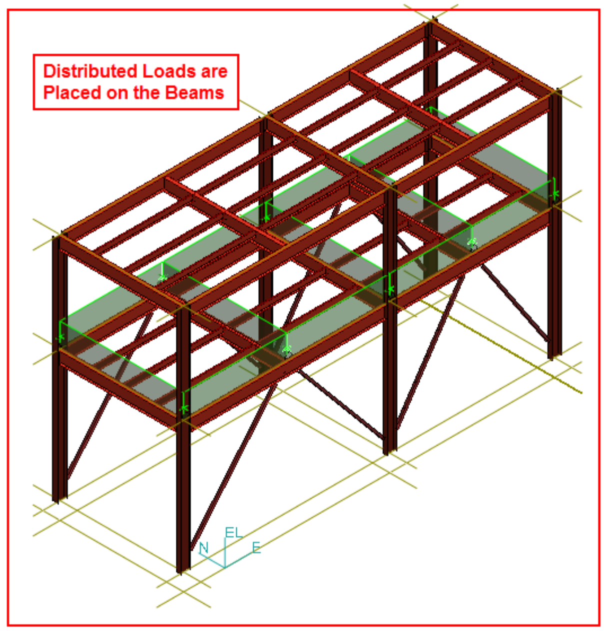


- *The **Position As** field defines how the start and stop positions are specified. If you select the **Relative** option in the **Position As** field, the start and end positions are defined as the fraction of the member length. If you select the **Absolute** option in the **Position As** field, the start and end positions are defined as actual distance from the member end.*
- *You can define the location of the start and end positions graphically on one member by clicking on the start and end point smartsteps after selecting the member. This graphic option works best if you use it when applying a load to just a member.*
- *The **Start Magnitude** and **End Magnitude** fields allow you to define a constant or linearly varying load per unit length.*

© Copyright 2014 Intergraph Corporation

22. Select all beams at Elevation 18'-0" and click "Finish" button. Loads will be placed as shown below:





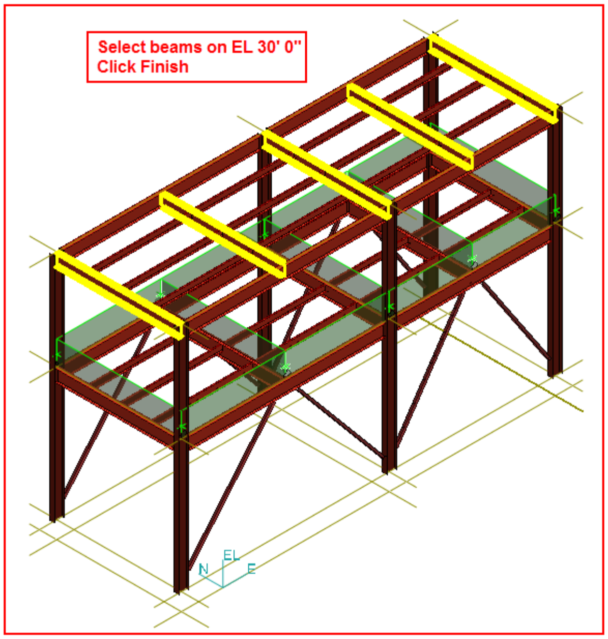
23. Define the following distributed load for the beams at Elevation 30'-0":

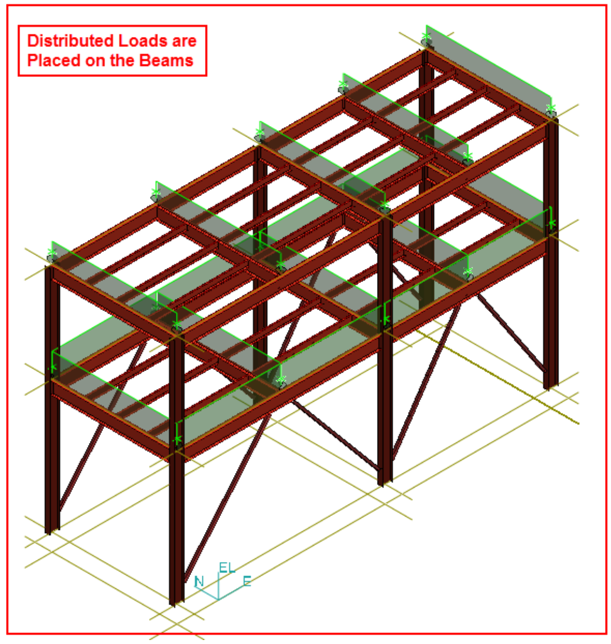
Load Case: DEAD
Load: Force Z
Reference: Global
Position as: Relative
Start Position: 0
End Position: 1
Start and End Magnitude: -0.100 kpf

24. Select the beams running north/south at Elevation 30'-0" and click Finish.

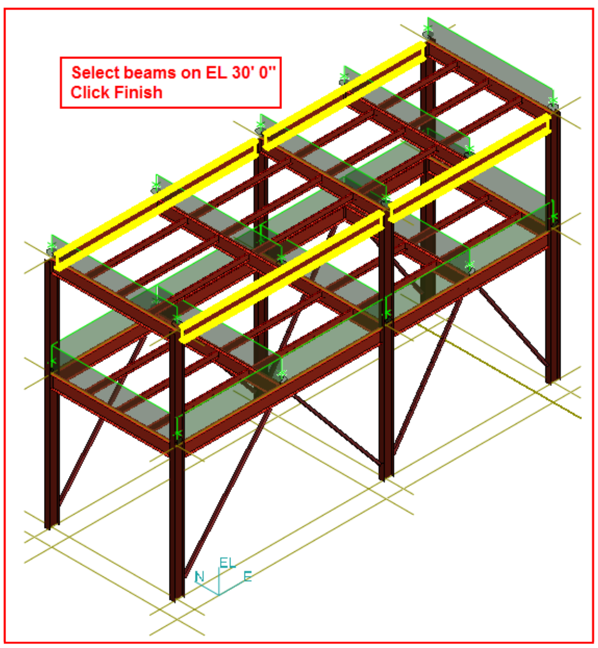
Loads will be placed as shown below:

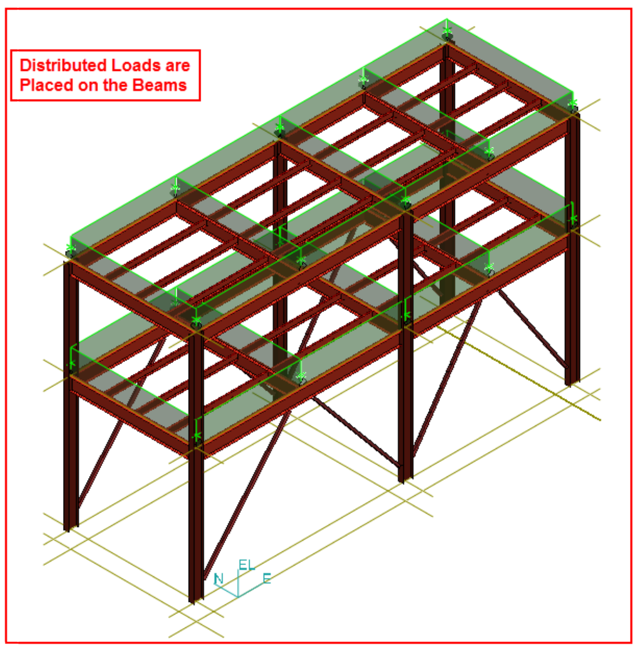
© Copyright 2014 Intergraph Corporation



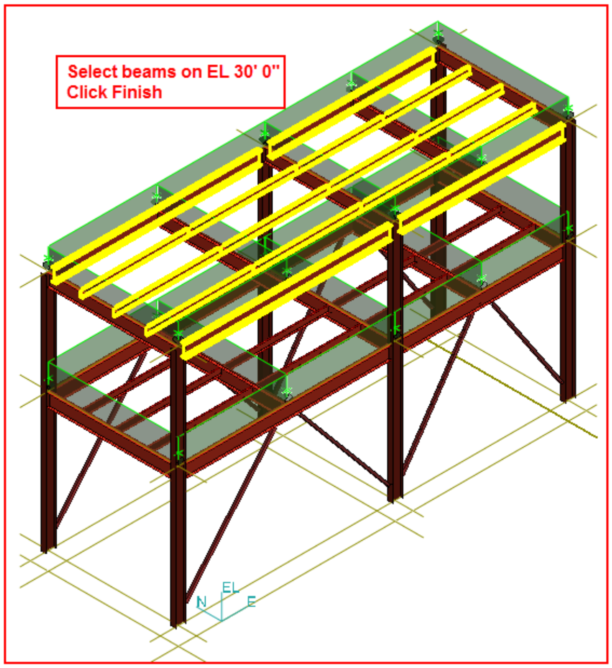


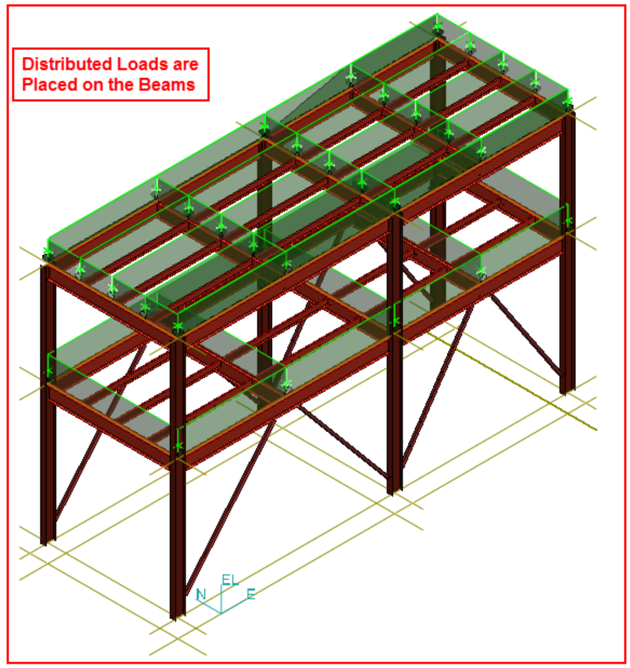
25. Change the Start and End Magnitude to –0.200 kpf and apply this to the 4 primary beams running east/west at Elevation 30'-0".



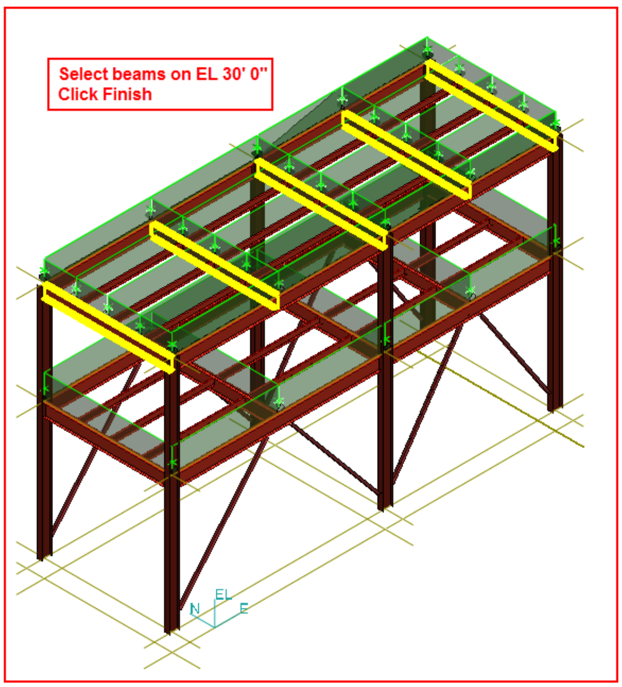


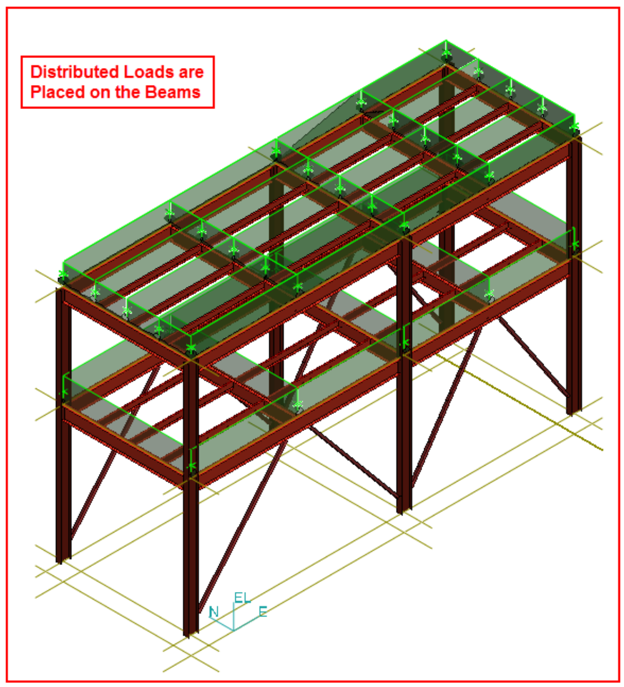
26. Change the Load Case Name to Live and the Magnitude to -.220 kpf and apply the load to the interior framing members and to the 4 primary beams running east/west at Elevation 30'-0.



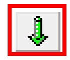


27. Change the Start and End Magnitude to –0.110 kpf and apply the Live Load to the 5 beams running north/south at Elevation 30'-0".





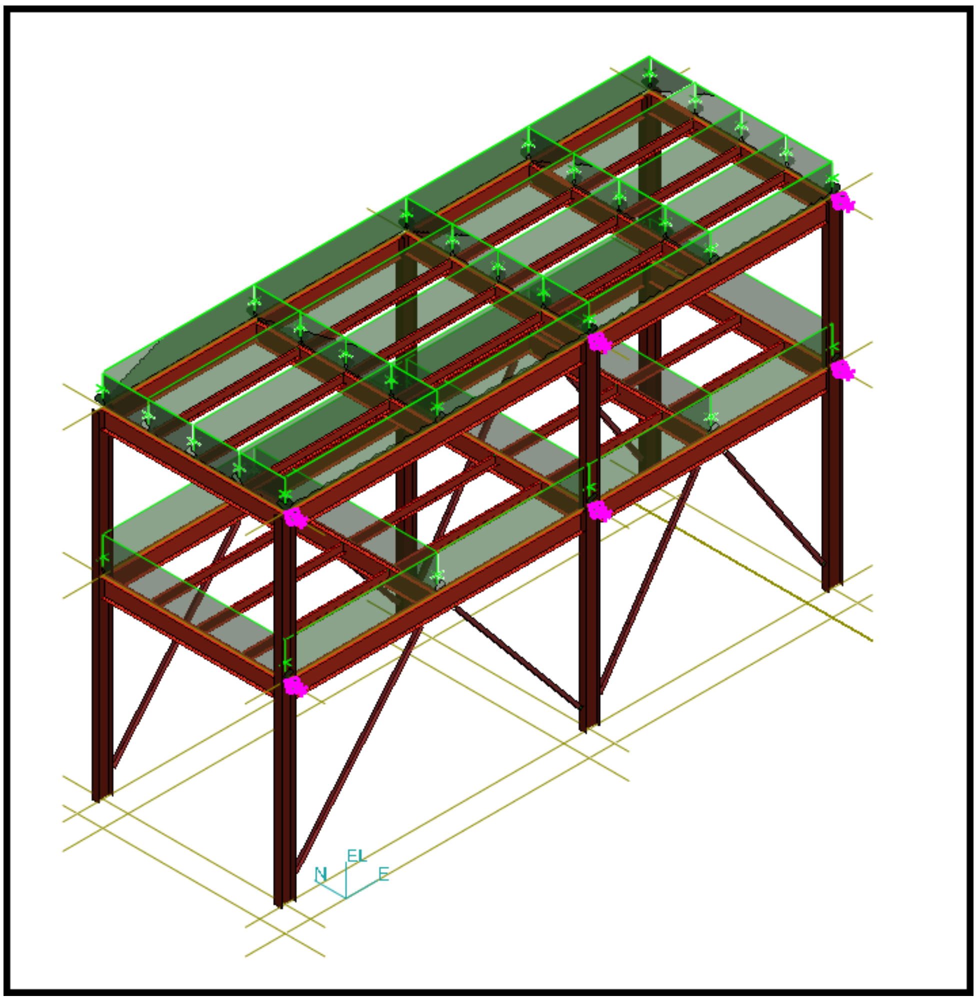
Part III: Place Wind Loads

28. Select Place New Concentrated Load Command. 

29. Define the load as:

    Load Case: WIND
    Load: Force Y
    Reference: Global
    Position as: Relative

30. Place concentrated wind loads along the south side of the structure with the following magnitudes:

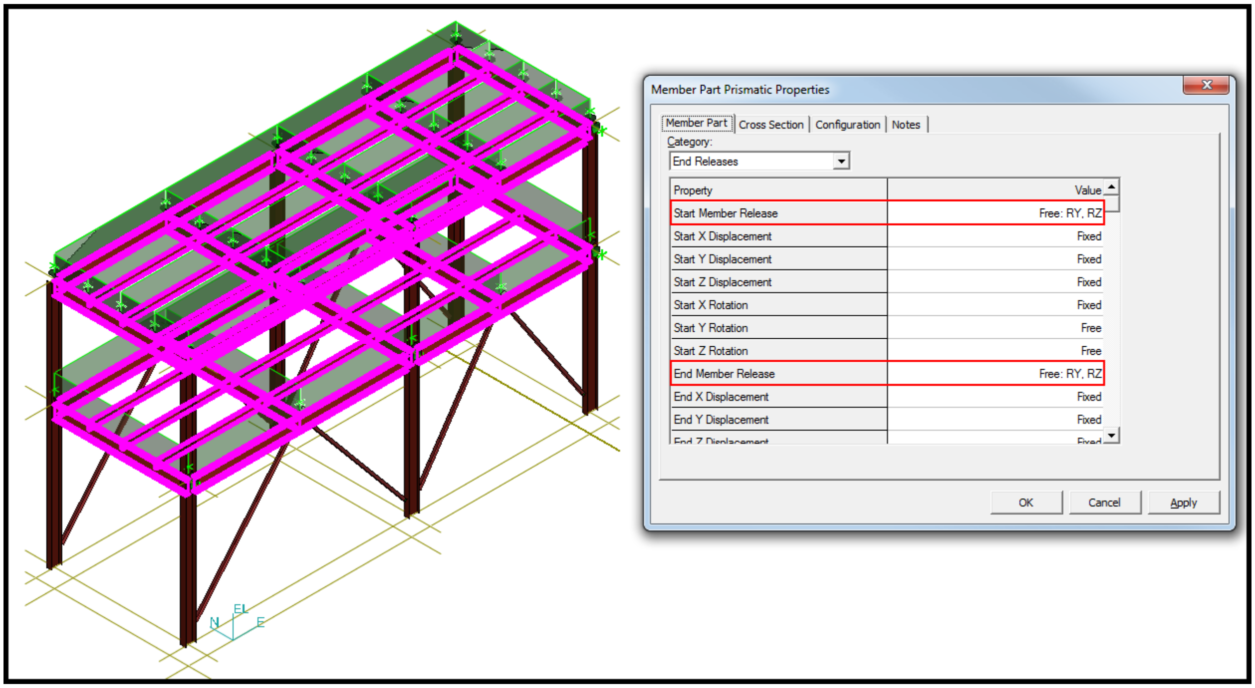


Part IV: Set Member Releases

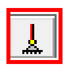
31. Select all beams in U02 with the Locate Filter set to Member Parts.

32. Select Edit → Properties in Menu to open the Properties dialog box

33. Choose the End Releases Category.
34. Set Start and End Member Release to Free: RY, RZ.



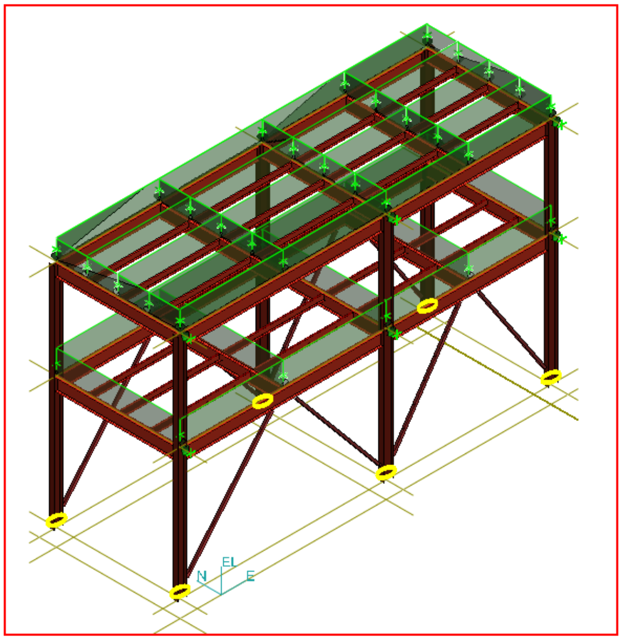
Part V: Place Boundary Conditions

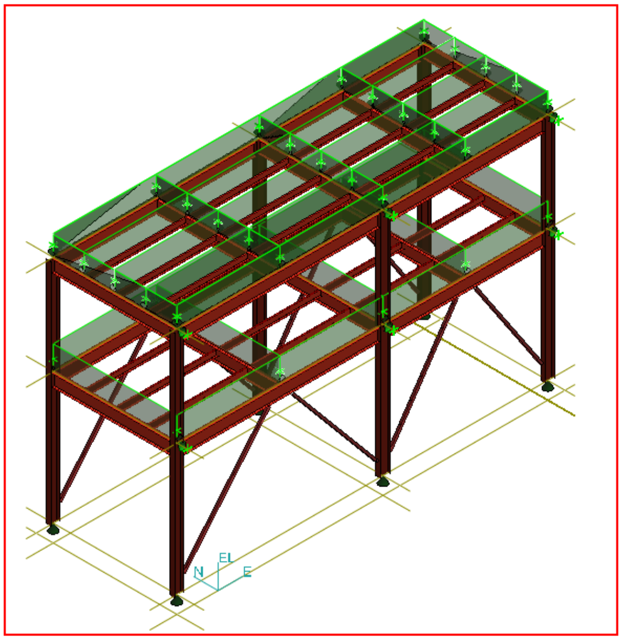
35. Select Set Boundary Condition Command. 
36. Select Frame Connections at the bottom of the first floor columns
37. Set the Type to Fix: X,Y,Z,RZ

38. Select "Finish" button.

Boundary Conditions are placed as shown below:

© Copyright 2014 Intergraph Corporation


N
EL
E


N
EL
E

Part VI: Export Analytical Model and Update from Analysis

CIMSTEEL Overview:

The STandard for the Exchange of Product model data (STEP - ISO 10303) provides a neutral computer-interpretable representation of product data throughout the life cycle of a product, independent of any particular system.

The CIMSteel Integration Standards (CIS/2) is the product model and electronic data exchange file format for structural steel project information. CIS/2 is intended to create a seamless and integrated flow of information among all parties of the steel supply chain involved in the construction of steel framed structures. It has been adopted by the American Institute of Steel Construction as their format for Electronic Data Interchange (EDI). CIMsteel stands for the Computer Integrated Manufacturing of Constructional Steelwork. The technical basis for CIS/2 is STEP (ISO 10301).

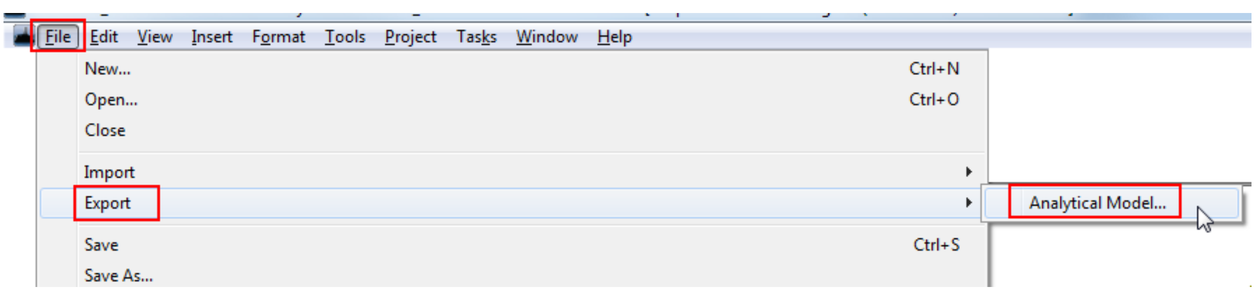
CIS/2 has been implemented as a file import or export capability by many steel design, analysis, engineering, fabrication, and construction software packages. A CIS/2 file exported by an analysis or design program could be imported into a detailing program to detail the connections. The user will see it as file format, just like DXF is a file format that is imported or exported in steel related CAD software.

The CIS/2 standard covers everything from nuts and bolts to materials and loads to frames and assemblies. Structures can be represented as analysis, design, or manufacturing (detailed) models. There is a logical relationship between the different types of models. For example, a beam in an analysis model that has to be subdivided for analysis is logically only one beam in the detailed model.

An analysis model is a logical collection of member parts, load cases, load combinations, boundary conditions, and end releases. Once you create the analysis model, you can export it to a third-party analysis program for stress analysis. You can edit section sizes in the analysis program and then import the section size changes to update your SP3D model. All other modifications to the structural model such as addition and deletion of members or modification to the member positions should be done in the SP3D model and exported again to the third-party analysis program for stress analysis.

Procedure to export Analysis model from SP3D:

39. To create a CIS2 file, select File -> Export -> Analytical Model.



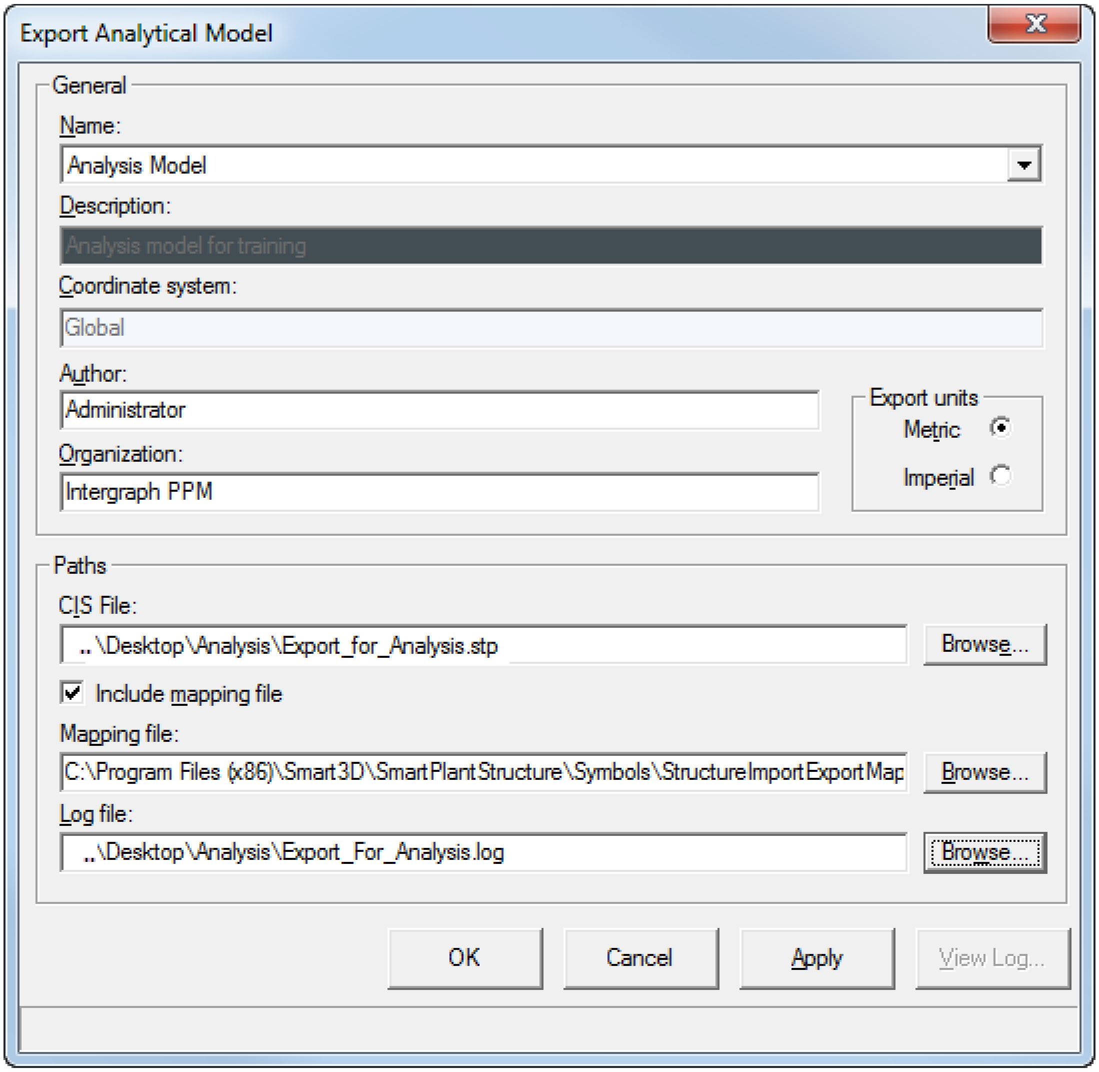
40. Specify the following on the Export Analytical Model dialog

a. **Name**: Analysis Model
b. **Author:** *Your Name*
c. **Organization:** *Your Company Name*
d. **Export Units:** Metric
e. **Paths:**
   i. **Export File:** *..\Desktop\Analysis\Export_for_Analysis.stp*

ii. **Include mapping file:** Yes. Check the box to indicate this.

iii. **Mapping File:** **..***ProductDirectory\3D\SmartPlantStructure\Symbols\StructureImportExportMaps \AISC_Master_Analysis_Map.XML*

iv. **Log File:** *..\Desktop\Analysis\Export_for_Analysis.log*



41. Click “OK” Button.

© Copyright 2014 Intergraph Corporation