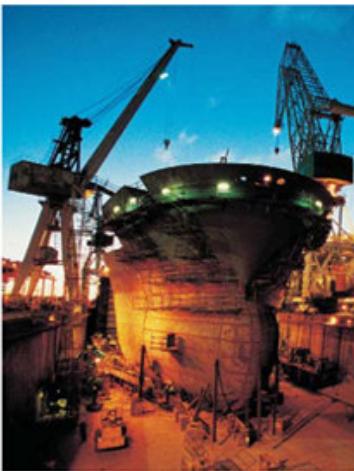
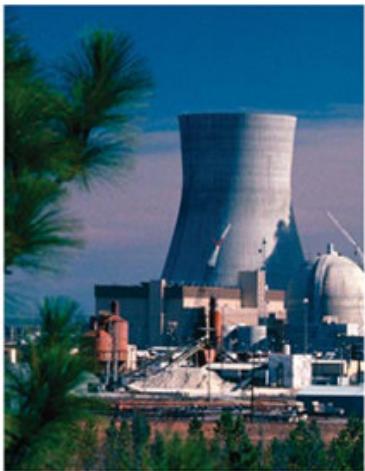


SmartPlant 3D Grids / Structure Student Workbook

April 2009

Version 2009

Process, Power & Marine



INTERGRAPH

Copyright

Copyright © 2009 Intergraph Corporation. All Rights Reserved.

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.

Restricted Rights Legend

Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c) of the *Contractor Rights in Technical Data* clause at DFARS 252.227-7013, subparagraph (b) of the *Rights in Computer Software or Computer Software Documentation* clause at DFARS 252.227-7014, subparagraphs (b)(1) and (2) of the *License* clause at DFARS 252.227-7015, or subparagraphs (c) (1) and (2) of *Commercial Computer Software---Restricted Rights* at 48 CFR 52.227-19, as applicable.

Unpublished---rights reserved under the copyright laws of the United States.

Intergraph Corporation
Huntsville, Alabama 35894-0001

Warranties and Liabilities

All warranties given by Intergraph Corporation about equipment or software are set forth in your purchase contract, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties. 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 discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license.

No responsibility is assumed by Intergraph for the use or reliability of software on equipment that is not supplied by Intergraph or its affiliated companies. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Trademarks

Intergraph, the Intergraph logo, SmartSketch, FrameWorks, SmartPlant, INtools, MARIAN, and PDS are registered trademarks of Intergraph Corporation. Microsoft and Windows are registered trademarks of Microsoft Corporation. **MicroStation is a registered trademark of Bentley Systems, Inc. ISOGEN is a registered trademark of Alias Limited.** Other brands and product names are trademarks of their respective owners.

Table of Contents

GRIDS TASK	4
LAB 1A – PLACING GRIDS/COORDINATE SYSTEMS – U02	4
LAB 1B - PLACING GRIDS - U04	10
LAB 1C – PLACING GRIDS - U03	14
LAB 2 - EDITING GRID PLANES	18
LAB 3 - OFFSHORE COORDINATE SYSTEMS – U05	21
LAB 4 - ROTATED GRID COORDINATE SYSTEM – U06.....	26
STRUCTURE TASK.....	28
LAB 1A - LINEAR MEMBER SYSTEM - STRUCTURE U04.....	28
LAB 1B - COPY / PASTE MEMBERS – STRUCTURE U02.....	35
LAB 1C - PRODUCTIVITY COMMANDS – STRUCTURE U02	40
LAB 2 - STRUCTURE MODELING - BUILDING U03	46
LAB 3 - CURVE MEMBER SYSTEM - BUILDING U03.....	63
LAB 4 - FRAME CONNECTIONS	71
LAB 5 - ASSEMBLY CONNECTIONS	75
LAB 6 - FIREPROOFING	80
LAB 7 - SLABS	103
LAB 8 - WALLS	115
LAB 9 - OPENINGS	137
LAB 10 - STAIRS / LADDERS / HAND RAILS.....	141
LAB 11 - FOOTINGS	153
LAB 12 - EQUIPMENT FOUNDATIONS.....	154
LAB 13 - DESIGNED SOLIDS.....	157
LAB 14- REPORTS	180
LAB 15 - STRUCTURAL MODELING (OFFSHORE JACKET) OPTIONAL.....	182
LAB 16 - USING BUILDING WIZARD (OPTIONAL).....	188
LAB 17 - DATA EXCHANGE FOR SOLIDS (OPTIONAL).....	194
STRUCTURAL ANALYSIS TASK	200
LAB 1 - LOADS, RELEASES, BOUNDARY CONDITIONS AND CREATING A CIS FILE	200

GRIDS TASK

Lab 1A – Placing Grids/Coordinate Systems – U02

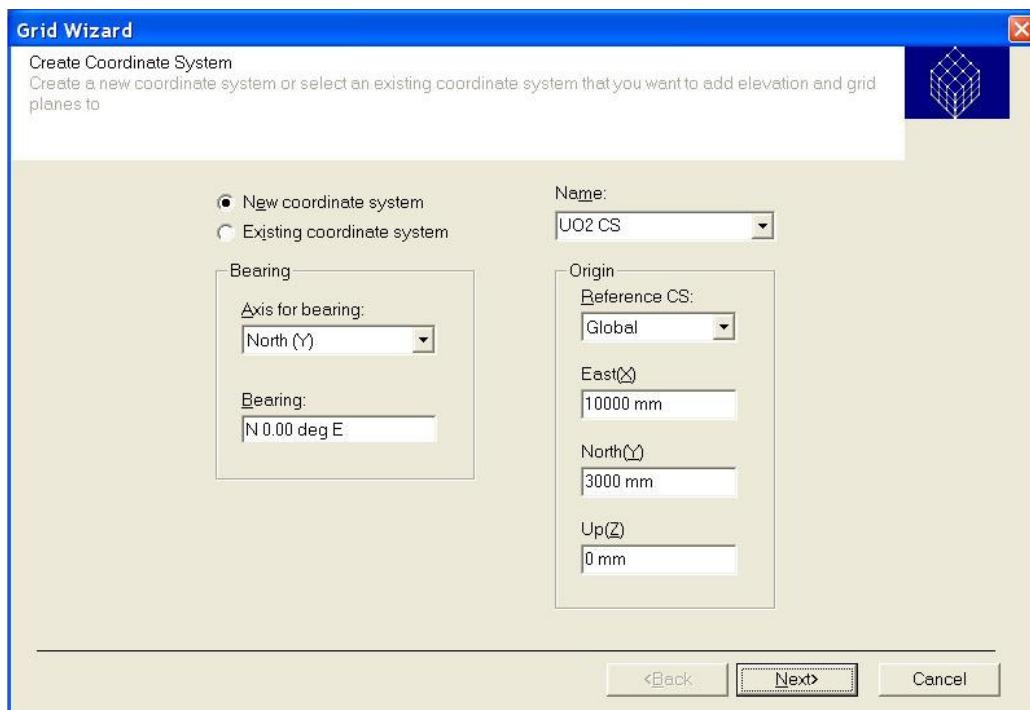
Objective

After completing this lab, you will be able to:

- Use the Grid Wizard Command to place grids and co-ordinate systems
1. Create a new session file with Metric Units and define an appropriate filter for your workspace using the SP3DtrainMet and Hide all existing graphics.
 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.

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

Name: Name: U02 CS
Bearing: Axis for Bearing: North (Y)
 Bearing: N 0.00 deg E
Origin: Reference CS: Global
 East (X): 10m
 North (Y): 3m
 Up (Z): 0m



Click the 'Next' button

Elevation Planes:

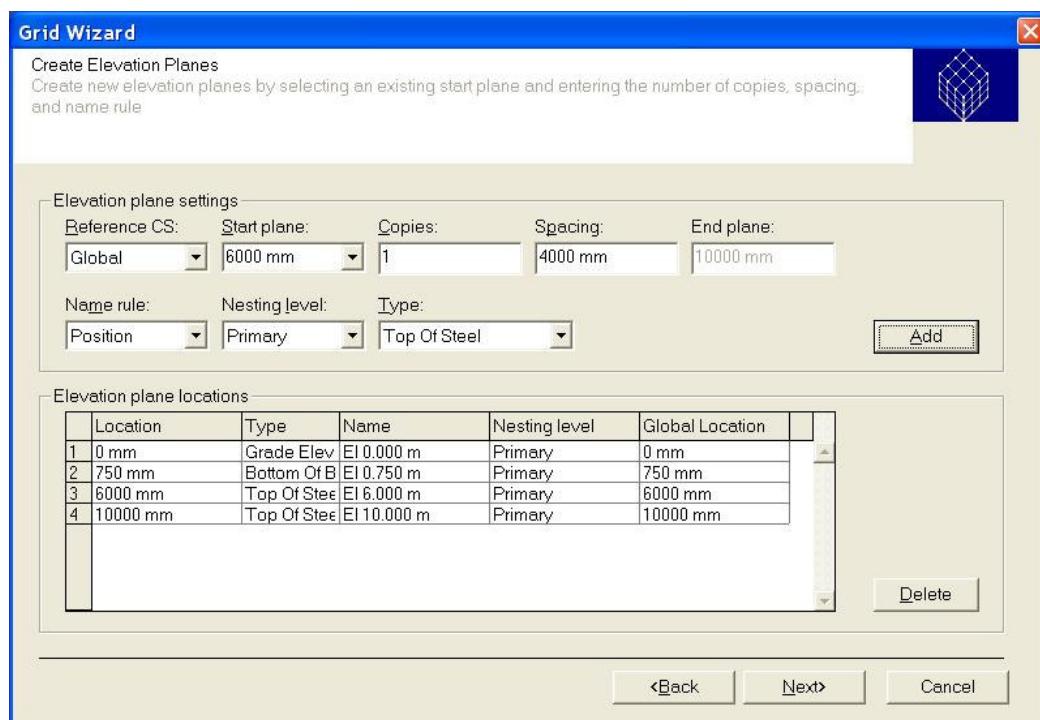
Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

Set the Name Rule as Position and Nesting level to Primary

Step	Start Plane	Copies	Spacing	End Plane	Type
1	0m	0	Ignore	0m	Grade Elevation
2	0.75m	0	Ignore	0.75m	Bottom of Baseplate
3	6m	0	Ignore	6m	Top of Steel
4	10m	0	Ignore	10m	Top of Steel

Select the ‘Add’ button between each data set keyin

The final Elevation plane locations setting should be as follows:



Select the ‘Next’ button

Grid X-Planes:

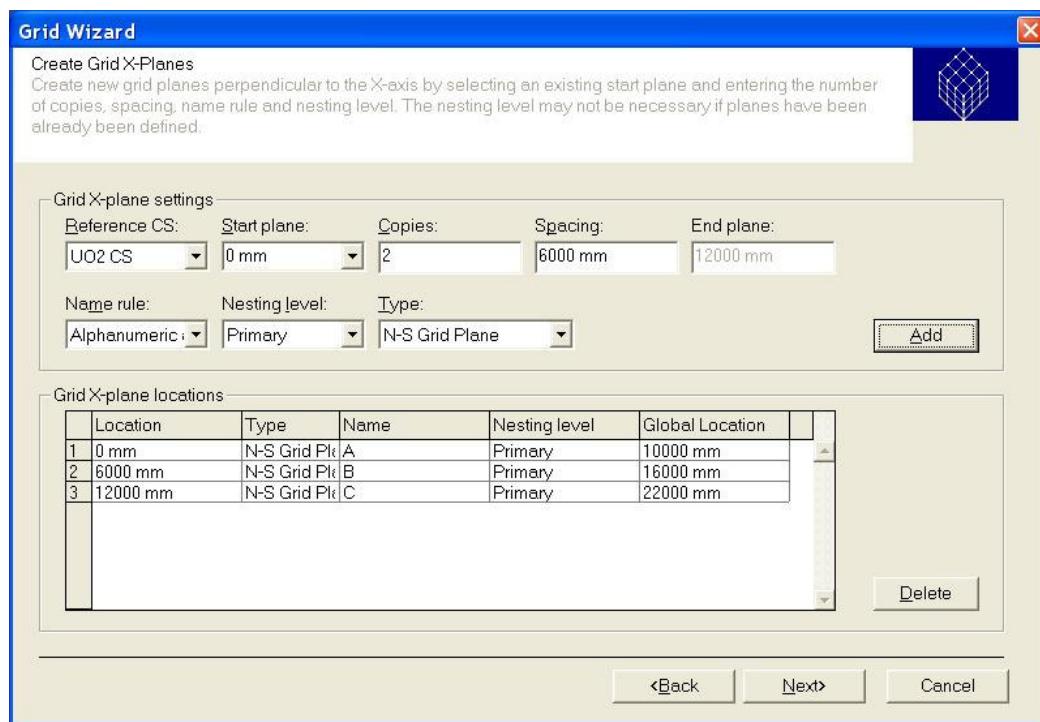
Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information:

Set the Name Rule as Alphanumeric and Percent and Reference CS to U02 CS

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	2	6m	12m	Primary	N-S Grid Plane

Set the values and select the ‘Add’ button

The final Grid-X plane locations setting should be as follows:



Click ‘Next’ button

Grid-Y Planes:

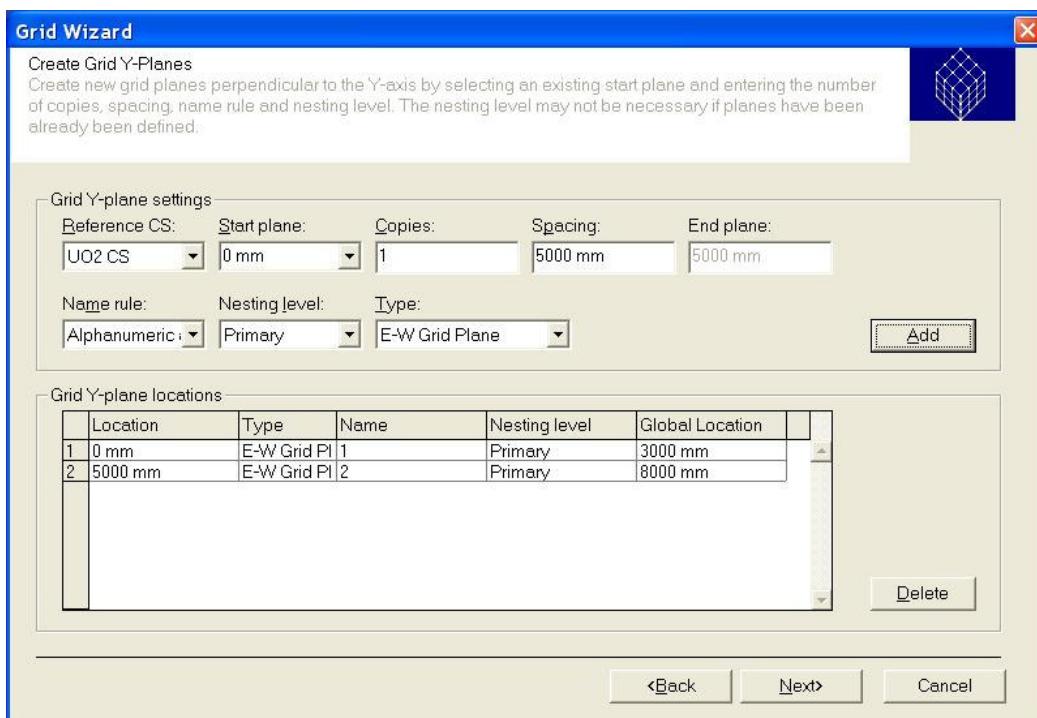
Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

Set the Name Rule as Alphanumeric and Percent, nesting level to Primary and Reference S to U02 CS.

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	1	5m	5m	Primary	E-W Grid Plane

Set the values and select the ‘Add’ button

The final Grid-Y plane locations setting should be as follows:

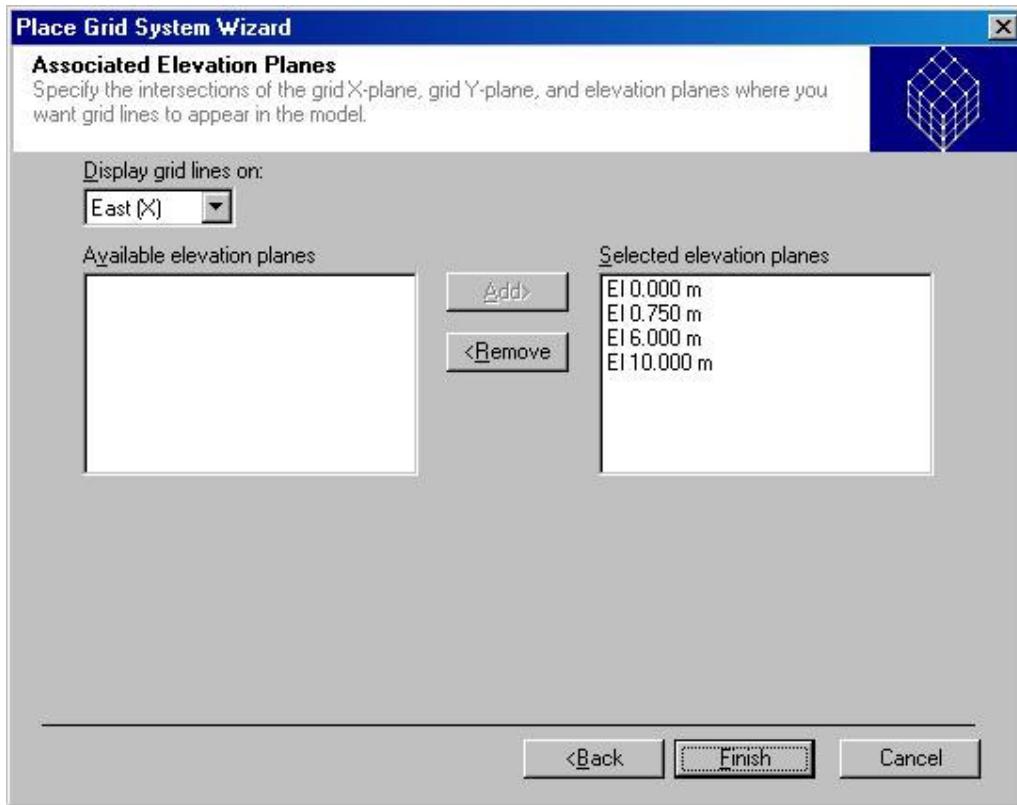


Hit Next button on step 5 Create Radial Cylinder form.

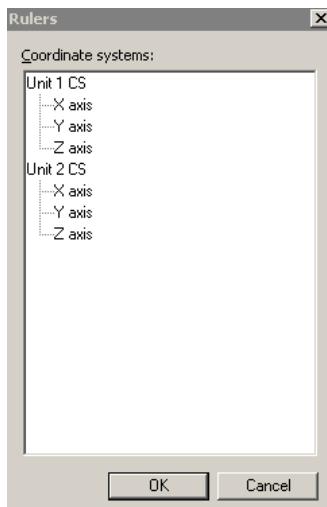
Hit Next button on step 6 Create Radial Plane form.

These two stages are skipped as no radial grid is being placed

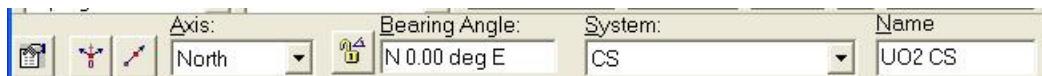
In Step 7 of the Grid Wizard command, Keep the default selection and Select the Finish Button.



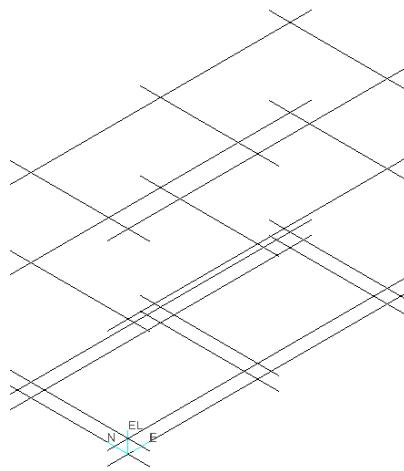
4. Go to the main menu and select View -> Rulers.. to open the ruler dialog box.
Press the <Control> key and unselect all to turn off the rulers.



5. Select the U02 CS in the Workspace Explorer to open the edit coordinate system ribbon bar.
Change the Parent System to CS



Use the Fit command from the main toolbar and the view should now resemble the following graphic in isometric.



Lab 1B - Placing Grids - U04

1. Within the Grid Task environment make sure the Active Permission Group is set to *Misc.* Select the Grid Wizard command.
2. Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:

Name: UO4 CS

Bearing:

Axis for Bearing: North (Y)

Bearing: N 0.00 deg E

Origin:

Reference CS: Global

East (X): 40m

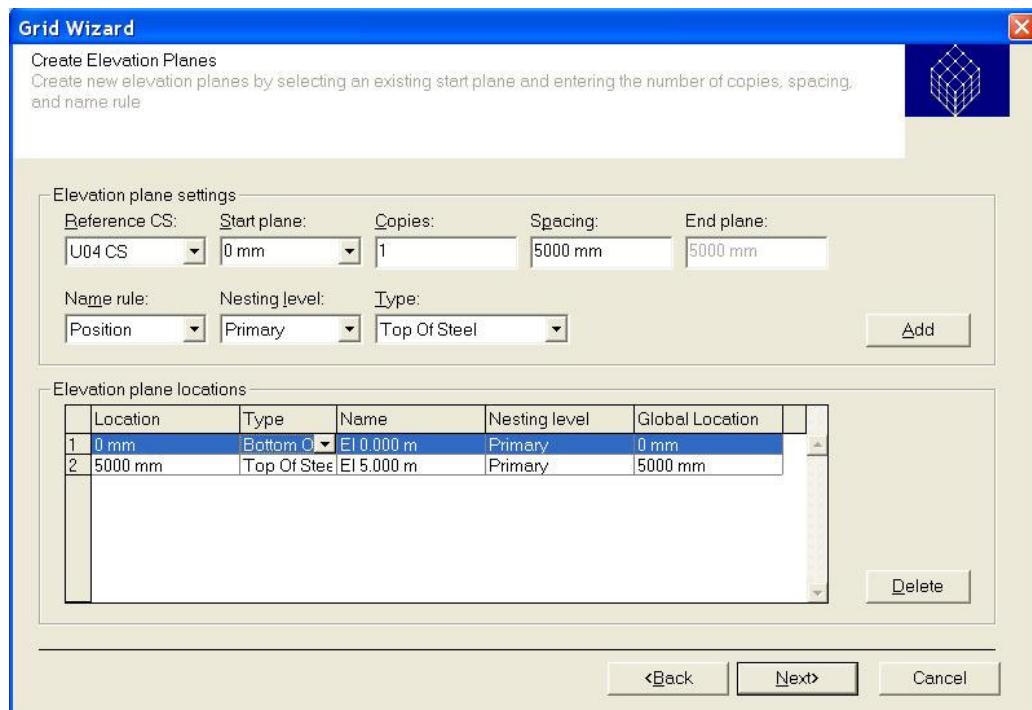
North (Y): 30m

Up (Z): 0m

3. Create the Elevation Planes based upon the following information with the name rule set to Position:

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	0	Ignore	0	Primary	Bottom of Baseplate
2	5m	0	Ignore	5m	Primary	Top of Steel

Select the “Add” button

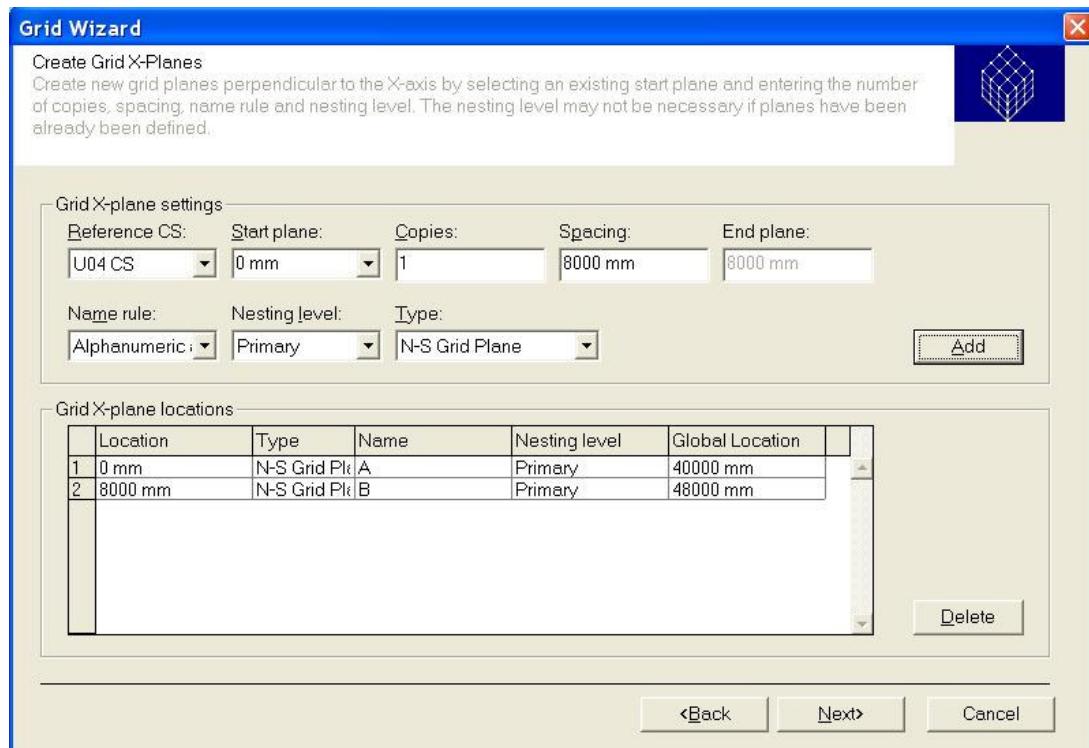


Select the “Next” button.

4. Create X-Planes based upon the following with name rule set to Alphanumeric and Percent and Reference CS to U04 CS:

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	1	8m	8m	Primary	N-S Grid Plane

Select the “Add” button

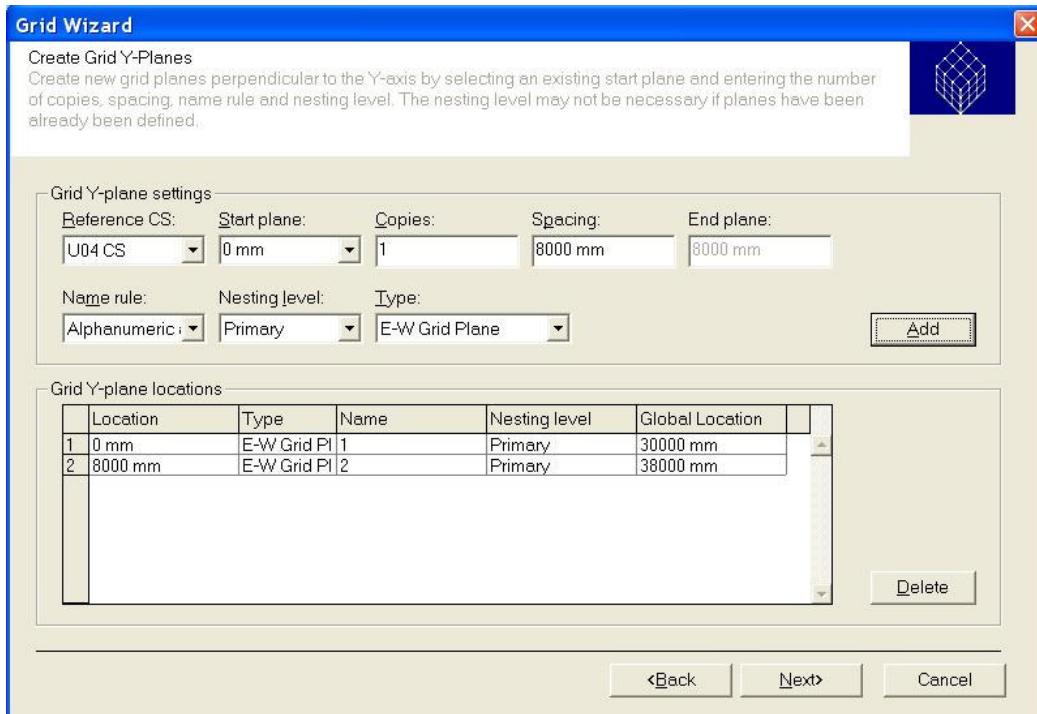


Select the “Next” button

5. Create Y-Planes based upon the following with name rule set to Alphanumeric and Percent and Reference CS to U04 CS:

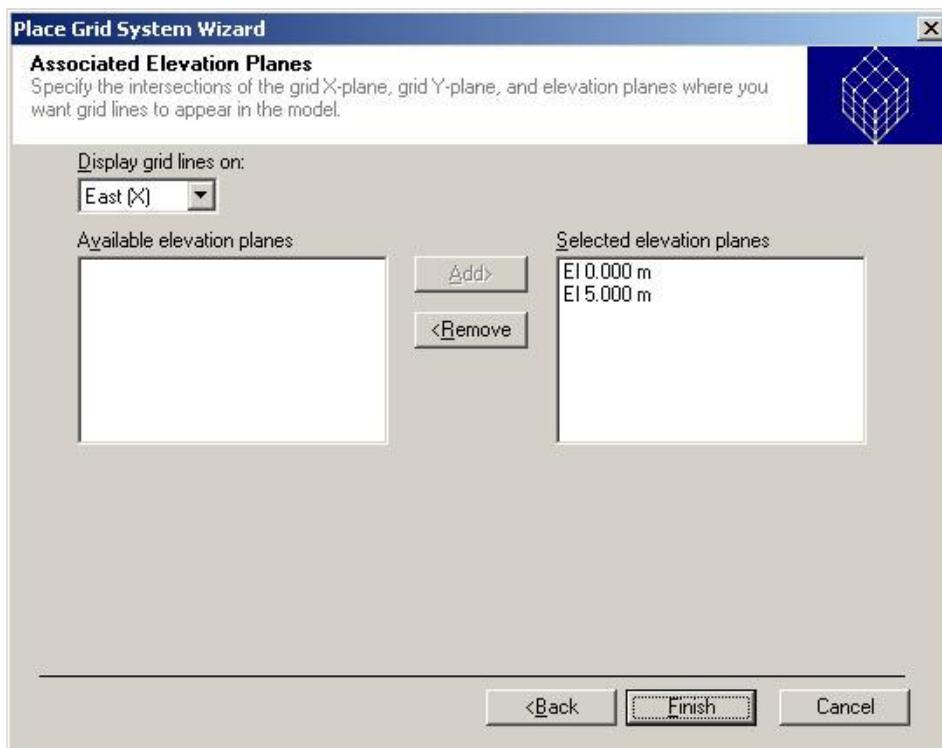
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	1	8m	8m	Primary	E-W Grid Plane

Select the “Add” button



Select the “Next” button

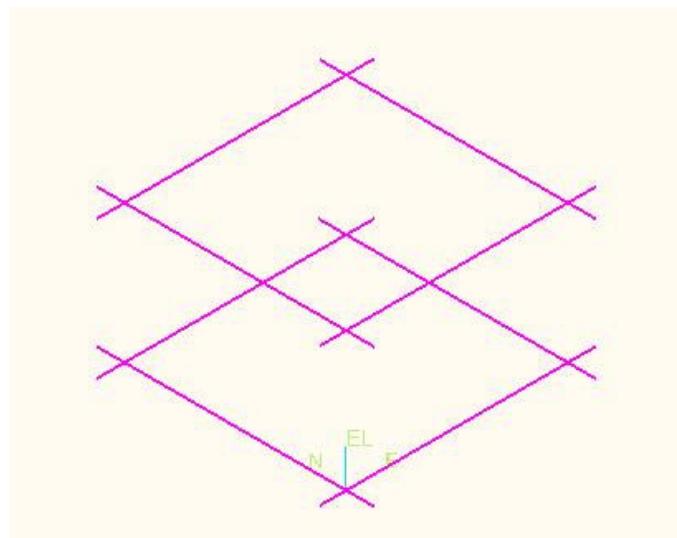
6. Select Next on the Radial Cylinder and Radial Plane forms.
7. Keep the default selection and select Finish on the final step.



8. Turn Off the rulers via the View -> Rulers option
9. Select the U04 CS in the Workspace Explorer and from the ribbon bar change the Parent System – ‘CS’



10. An Isometric View should now resemble the graphics below:



Lab 1C – Placing Grids - U03

- 1 Within the Grid Task environment make sure the Active Permission Group is set to *Misc*.
- 2 Select the Grid Wizard command
- 3 Using Step 1 in the Grid Wizard command, create a new coordinate system based on the following information:

Name:

U03 CS

Bearing:

Axis for Bearing: North (Y)

Bearing: N 0.00 deg E

Origin

Reference CS: Global

East (X): -12m

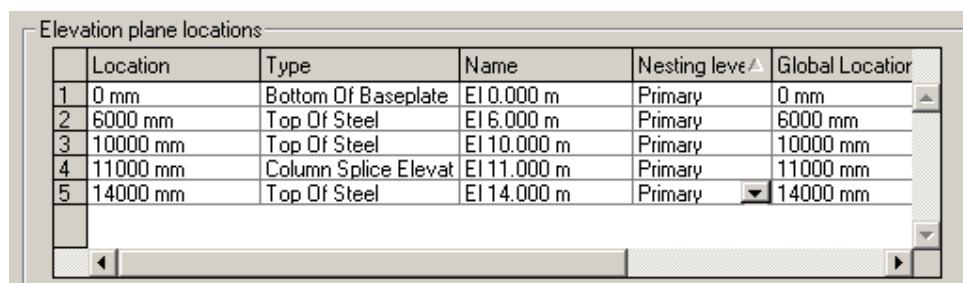
North (Y): 3m

Up (Z): 0 m

- 4 **Elevation Planes:** Using Step 2 in the Grid Wizard create the Elevation Planes based on the following information:
Set the Name Rule to Position for All Grid Planes and Reference CS as U03 CS

Step	Start Plane	Copies	Spacing	End Plane	Type
1	0m	1	6m	6m	Top of Steel
2	10m	1	1m	11m	Top of Steel
3	14m	0	1m	14m	Top of Steel

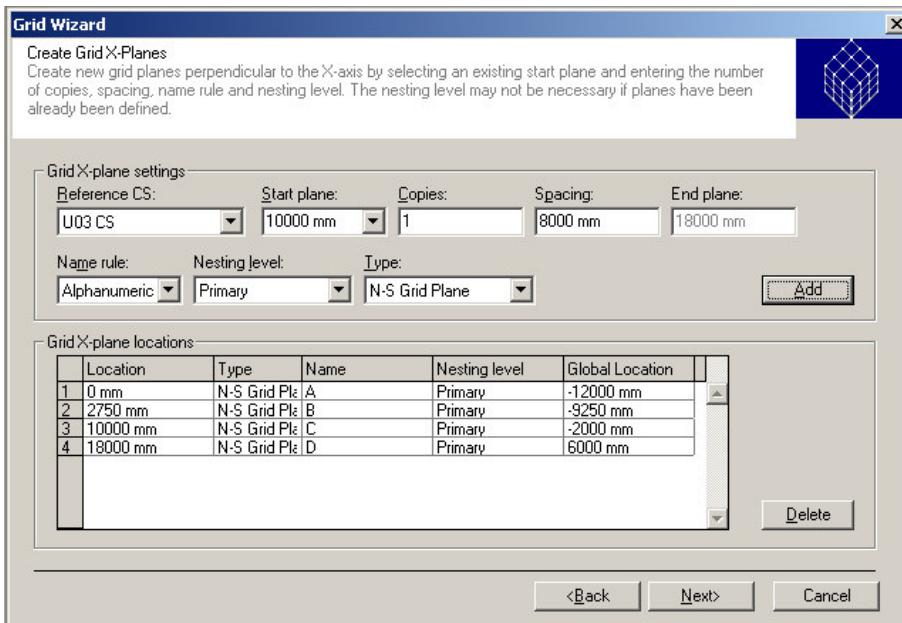
In the Elevation Plane Locations set Type to Bottom of Baseplate for the first location and Column Splice Elevation for the fourth location (11.000m) as below.



- 5 **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 and Reference CS as U03 CS

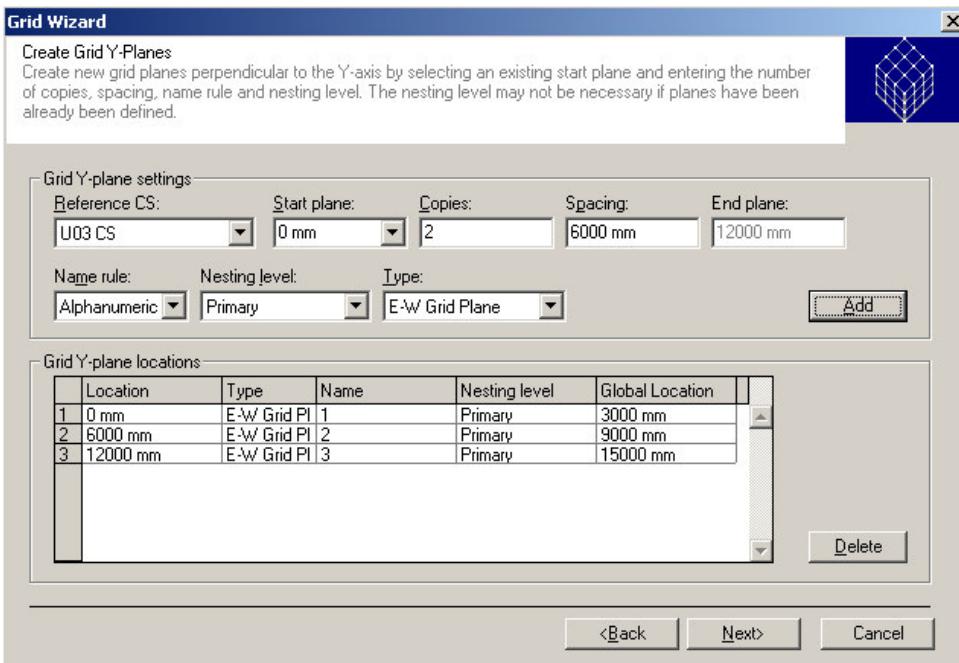
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	1	2.75m	2.75m	Primary	N-S Grid Plane
2	10m	1	8m	18m	Primary	N-S Grid Plane



- 6 **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 and Reference CS as U03 CS

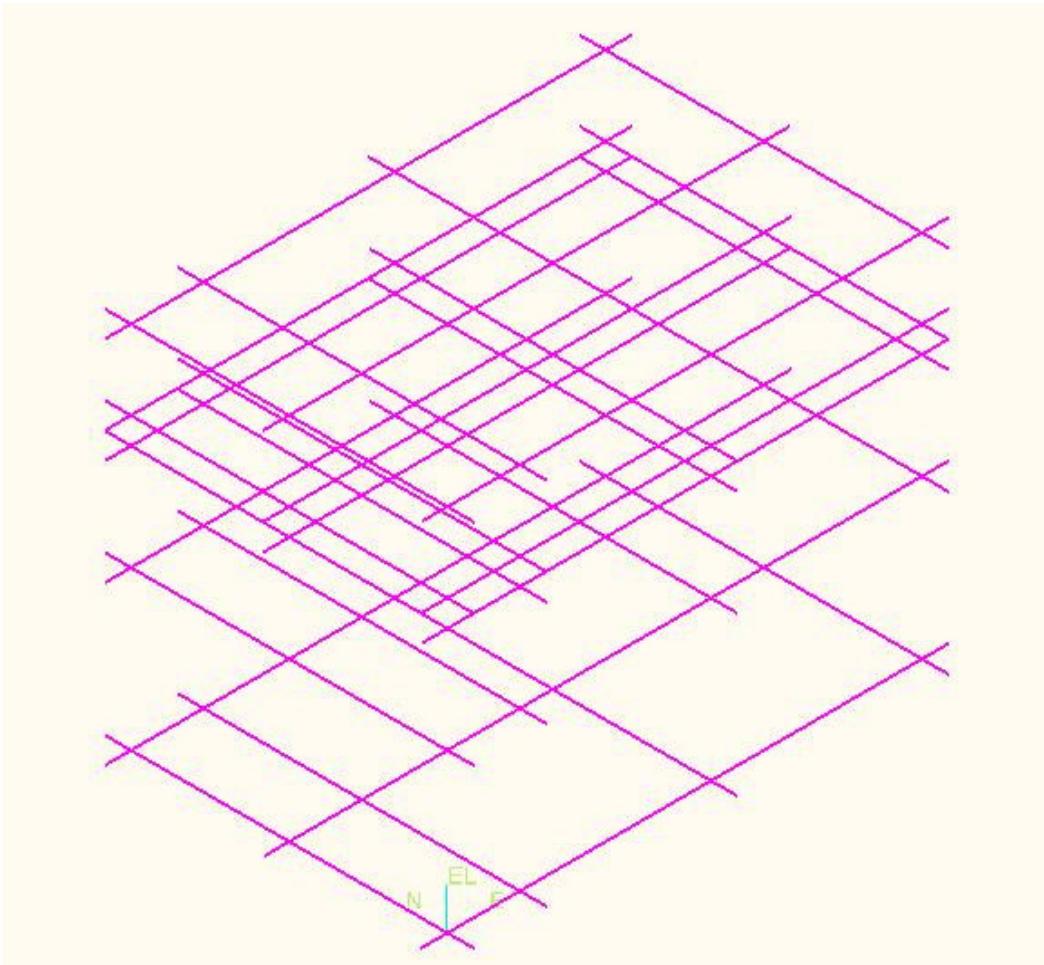
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	2	6m	12m	Primary	E-W Grid Plane



- 7 Select Next for steps 5 and 6 and in Step 7 of the Grid Wizard command, Keep the default selection and Select the Finish Button.
- 8 Go to the main menu and select View -> Ruler option to open the ruler dialog box.
9. Press the <Control> key and unselect all to turn off the rulers.
10. Select the U03 CS and move it to the System – ‘CS’



11. Your View should now resemble the following graphic.

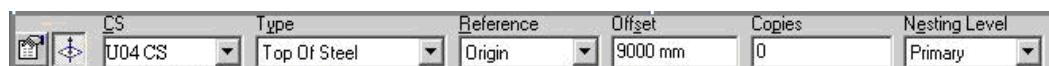


Lab 2 - Editing Grid Planes

Objective

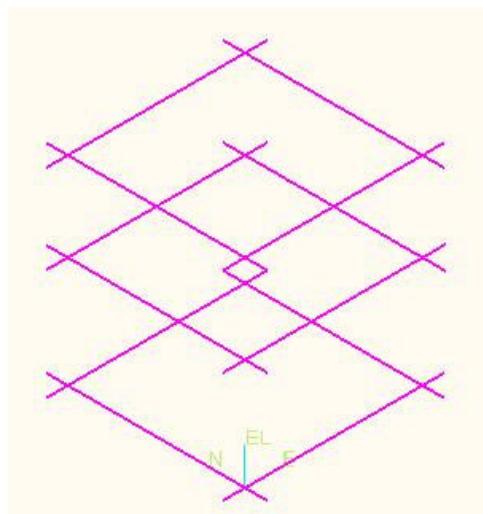
After this Lab Students will be able to edit grid planes, add planes to existing grids / coordinate systems.

- 1 Open or create a session file and define an appropriate filter for your workspace
- 2 Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc.*
- 3 Go to the Grid Task environment.
- 4 **Placing Elevation Plane:** Select the Place Elevation Plane command  button from the command toolbar.
- 5 In the Elevation Plane horizontal ribbon bar enter the following parameters:



Coordinate System: U04 CS
 Type: Top of Steel
 Name: Open the properties page and set Naming Rule to Position
 Reference: Origin
 Copies: 0
 Nesting Level: primary

- 6 Key-in the offset value and hit <Enter> to commit the transaction
 Offset value: 9000mm
- 7 The system places the elevation plane marker in the Ruler after entering the above parameters. Right mouse click to exit the command to give the following grid graphics.



8 **Placing Grid Planes X-Axis:** Select the Place Grid Plane command button  from the command toolbar

9 In the Grid Plane horizontal ribbon bar enter the following parameters:



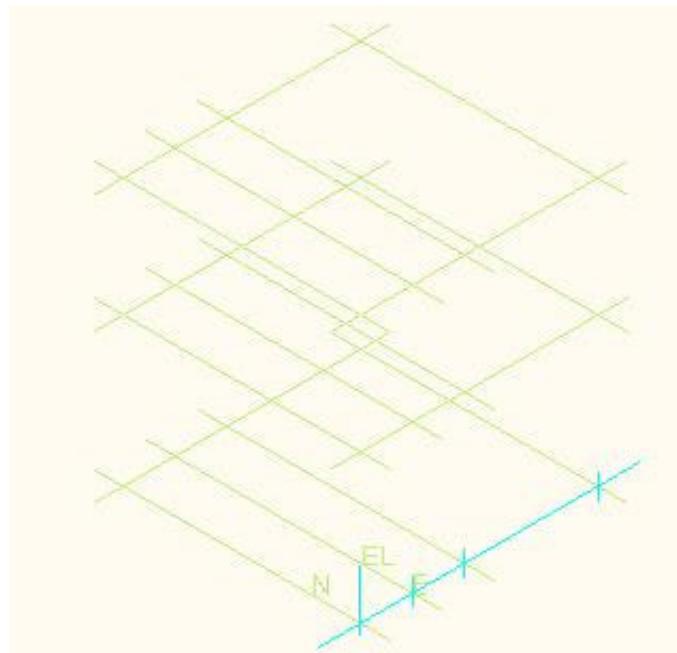
Coordinate System: U04 CS
 Axis: X
 Type: N-S Grid Plane
 Reference: Origin
 Copies: 1
 Nesting Level: Secondary

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

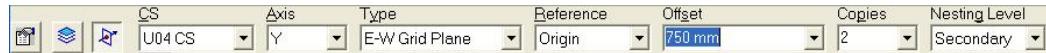
11 Offset value: 1750mm

12 Change the default name for these planes to Alphanumeric and Percent via the properties page.

13 The system places the X-Planes marker in the Ruler and the gridlines. Right mouse click to exit the command. The graphic view should appear thus:

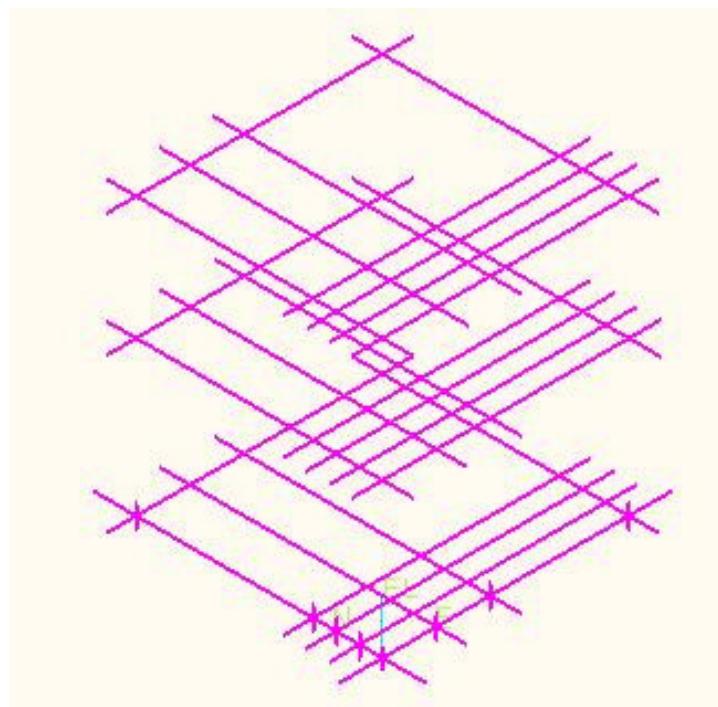


- 14 **Placing Grid Plane Y-Axis:** Select the Place Grid Plane command button  from the command toolbar.
- 15 In the Grid Plane horizontal ribbon bar enter the following parameters:



Coordinate System: U04 CS
 Axis: Y
 Type: E-W Grid Plane
 Reference: Origin
 Copies: 2
 Nesting Level: Secondary

- 16 Key-in the offset value and hit <Enter> to commit the transaction
- 17 Offset value: 750mm
- 18 Change the default name for these planes to Alphanumeric and Percent via the properties page.
- 19 The system places the Y-Planes marker in the Ruler and the gridlines. Right mouse click to exit the command. He graphic view should appear thus:



Lab 3 - Offshore Coordinate Systems – U05

Objective

To create a grid system and skew it by rotating the grid planes to create a grid suitable to define an offshore structure.

1. Open or create a session file with an appropriate filter for your workspace.

Go to the Grid Task environment. Make sure the Active Permission Group is set to *Misc* and Select the Grid Wizard command.

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

Name:

Name: U05 CS

Bearing:

Reference CS: Global

Axis for Bearing: North (Y)

Bearing: N 0.00 deg E

Origin:

East (X): -10m

North (Y): -10m

Up (Z): 0m

Elevation Planes:

3. Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information, using Position as the Name rule and U05 CS as the Reference CS:

Step	Start Plane	Copies	Spacing	End Plane	Type
1	0m	3	3m	9m	Top of Concrete

Grid X-Planes:

4. Using Step 3 in the Grid Wizard command create the Grid-X Planes based on the following information with Name rule Alphanumeric percent and Reference CS U05 CS:

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0	1	3m	3m	Primary	N-S Grid Plane

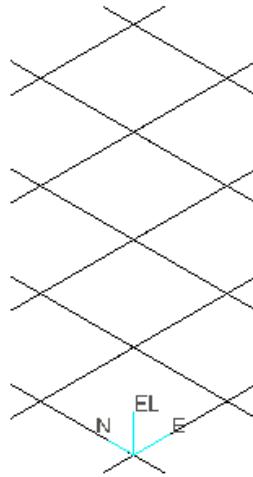
Grid-Y Planes:

6. Using Step 4 in the Grid Wizard command create the Grid-Y Planes based on the following information with Name rule Alphanumeric percent and Reference CS U05 CS:

Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	1	3m	3m	Primary	E-W Grid Plane

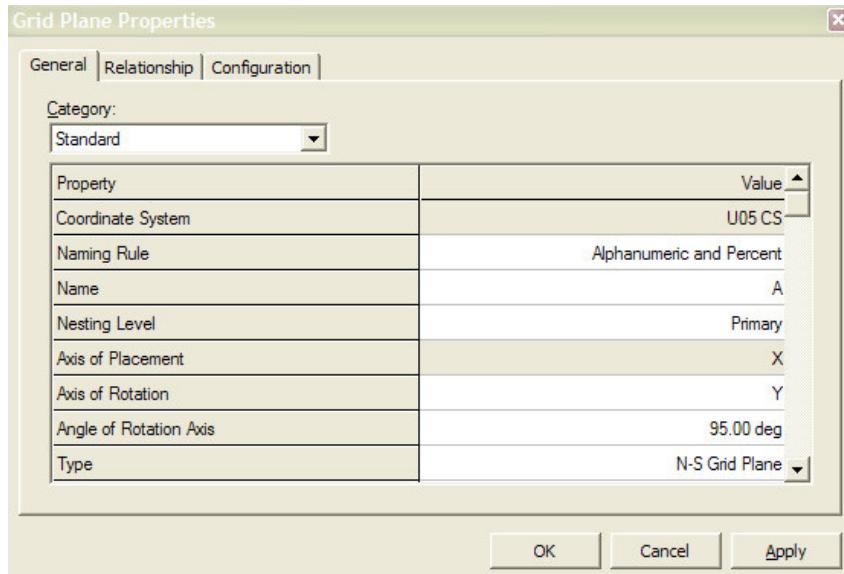
7. Skip Step 5 and 6 and in Step 7 of the Grid Wizard command, Keep the default selection and Select the Finish Button.
8. Go to the main menu and select View -> Ruler option to open the ruler dialog box. Press the <Control> key and unselect all to turn off the rulers.
9. Select the U05 CS to open the edit coordinate system ribbon bar and change the Parent System to CS

The Isometric View should now resemble the following graphic.



10. Select the Grid Plane A 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:

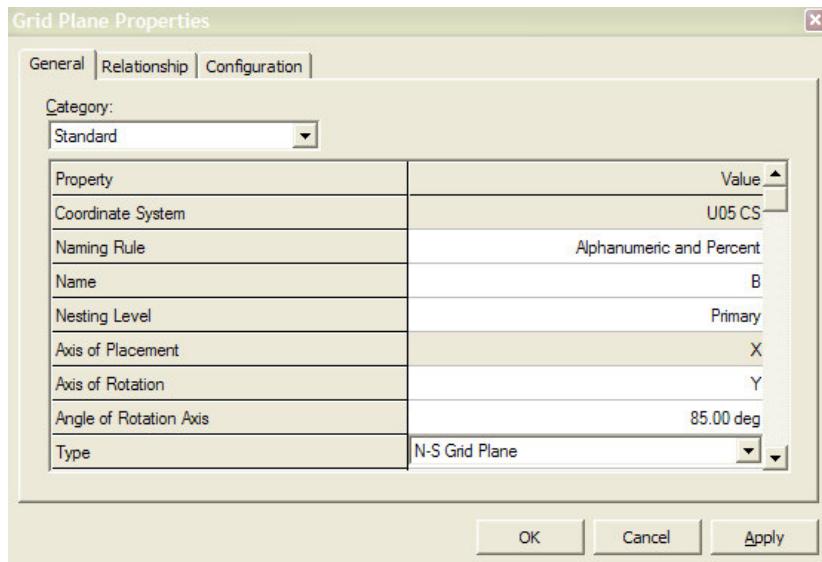
Axis of Rotation: Y
 Angle of Rotation: 95.0 deg



Select Apply button to commit the transaction.

13. Select the Grid Plane B in the Workspace Explorer
14. In the Grid Plane Properties page enter the following parameters:

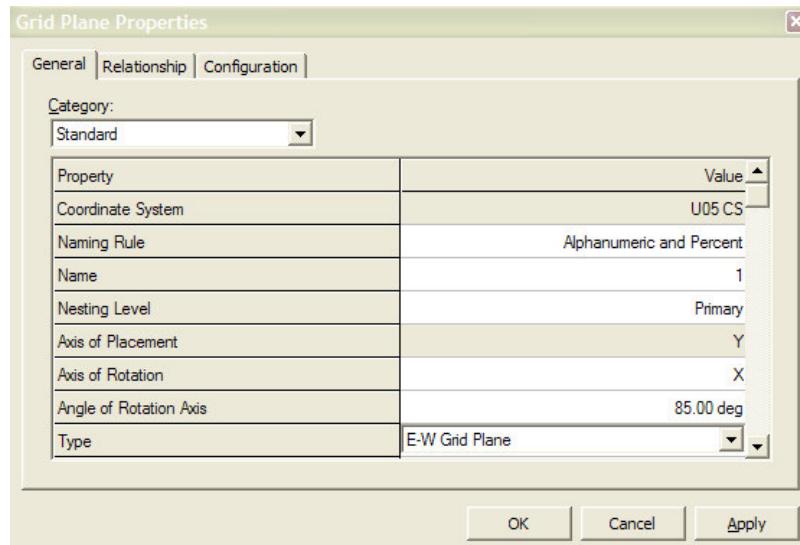
Axis of Rotation: Y
Angle of Rotation: 85.0 deg



Select Apply button to commit the transaction.

15. Select the Grid Plane 1 in the Workspace Explorer
16. In the Grid Plane Properties page enter the following parameters:

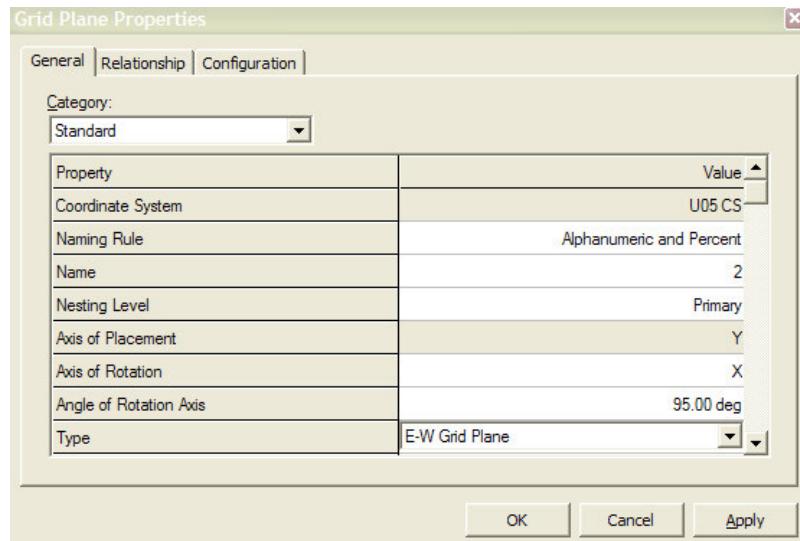
Axis of Rotation: X
 Angle of Rotation: 85.0 deg



Select Apply button to commit the transaction.

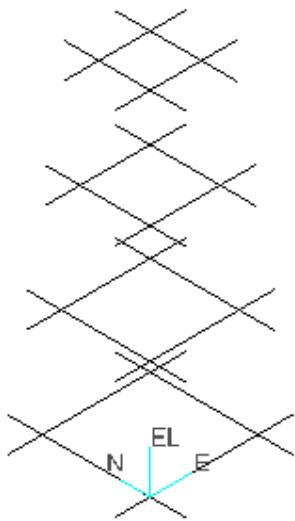
17. Select the Grid Plane 2 in the Workspace Explorer
18. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation: X
 Angle of Rotation: 95.0 deg



Select OK button to commit the transaction.

The Isometric View should now resemble the following graphic.



Lab 4 - Rotated Grid Coordinate System – U06

Objective

To create a coordinate system and grid which is rotated from the global coordinate system.

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

Name:

Name: U06 CS

Bearing:

Axis for Bearing: North (Y)
Bearing: N 20.00 deg W

Origin:

Reference CS: Global
East (X): 8 m
North (Y): -10 m
Up (Z): 0 m

4. Elevation Planes: Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information with Name rule set to Position and Reference CS to U06 CS:

Step	Start Plane	Copies	Spacing	End Plane	Type
1	0m	1	6000mm	6000mm	Top of Steel

5. Grid-X Planes: Using Step 3 in the Grid Wizard command, create the Grid-X Planes based on the following information with Name rule Alphanumeric and Percent and Reference CS as U06 CS:

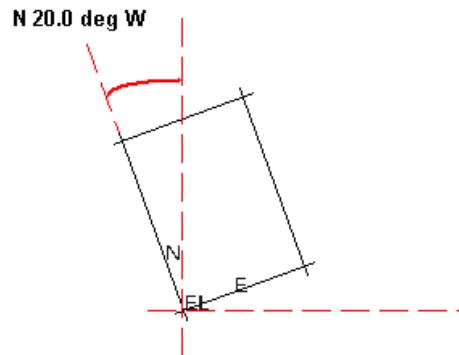
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0	1	3000mm	3000mm	Primary	N-S Grid Plane

7. Grid-Y Planes: Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information with name rule Alphanumeric and Percent and Reference CS as U06 CS:

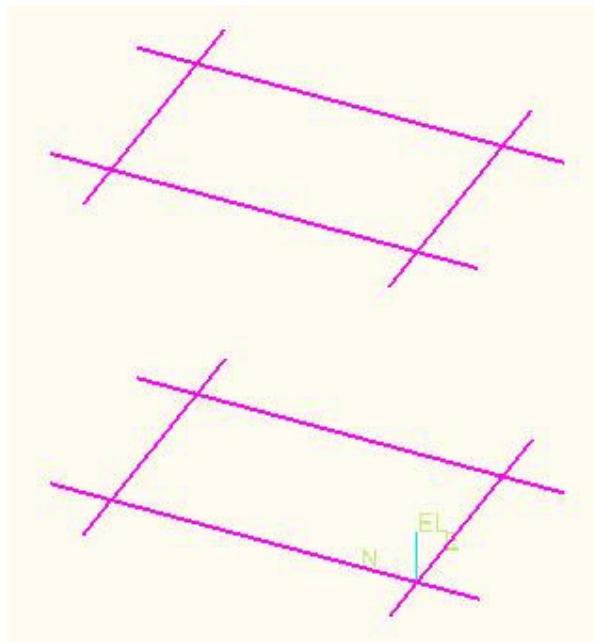
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0m	1	5000mm	5000mm	Primary	E-W Grid Plane

8. Skip steps 5 and 6 and in Step 7 of the Grid Wizard command, keep the default selection and Select the Finish Button.
9. Finally set the Parent System to CS

The Views should now resemble the following graphics.



PLAN VIEW



ISOMETRIC VIEW

STRUCTURE TASK

Lab 1A - Linear Member System - Structure U04

Objectives

After completing this lab, you will be able to:

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

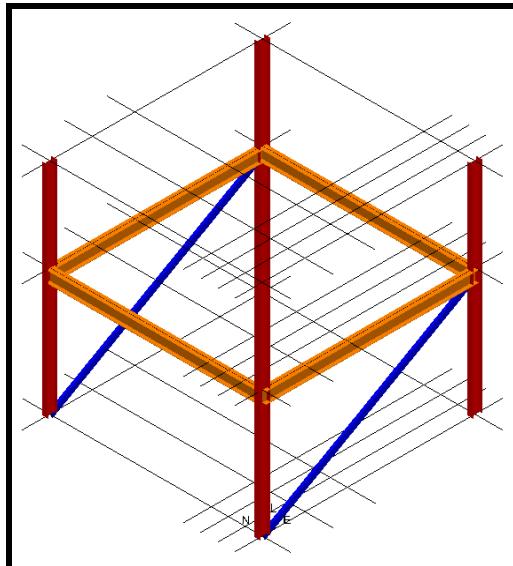
Note: Use the grid system to help you place the structural members. Though not absolutely necessary, grids help in the modeling process by providing SmartSketch intersection points at which structural members can be placed. You may also want to make a relationship with gridlines so that if a gridline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

Placing Columns / Beams / Vertical Braces

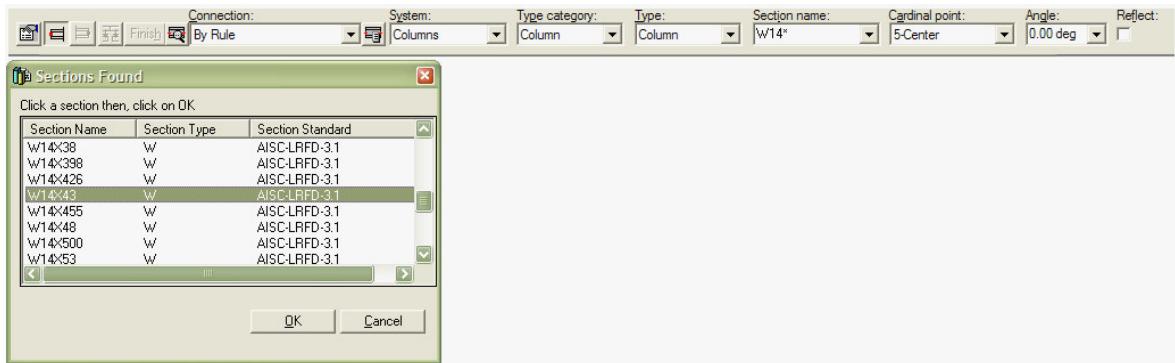
1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include A2 -> U04 and CS -> U04 CS systems.

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. Select Place Linear Member System command. Use the view shown below to place the appropriate support columns, beams and braces.



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

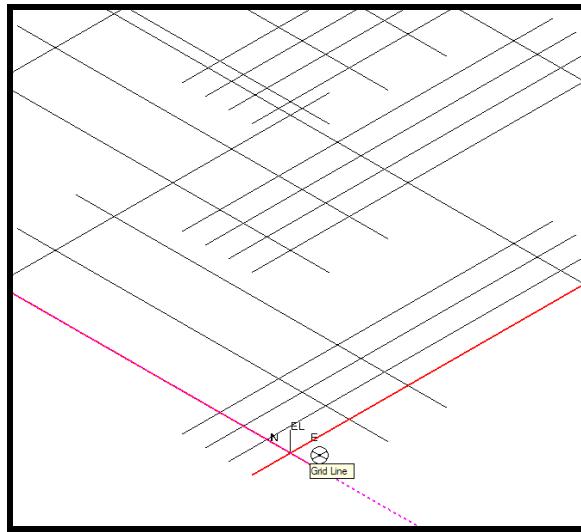
Note: Use the cross section lookup service to find the appropriate section. Key in W14 to see another dialog where you can select W14x43.*

- f. Cardinal Pt. = 5
- g. Angle = 0
- h. Reflect = Off

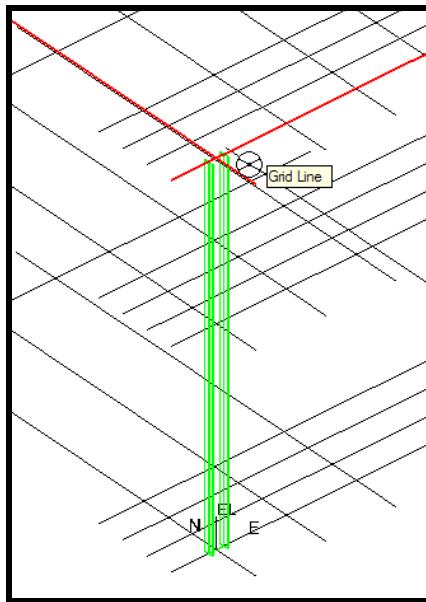
Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.

- i. Material: Steel- Carbon
- j. Grade A36

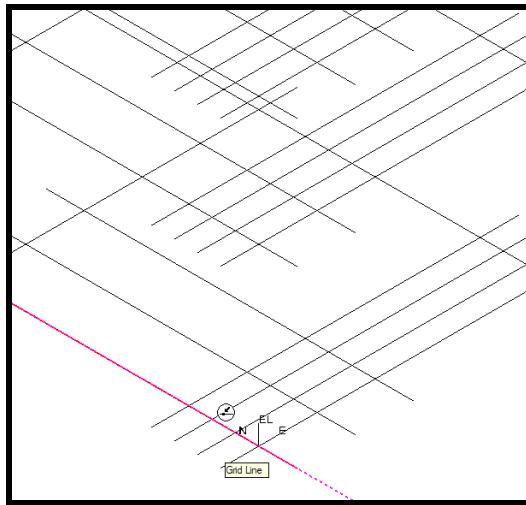
7. For “End 1” of the column, move your cursor over the intersection of the two grid lines. A glyph will appear that indicates the intersection of two lines. See figure below. Then give a Left Mouse click.



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



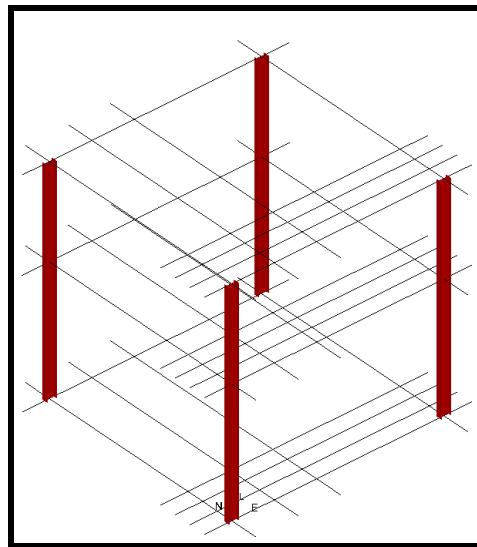
Note: An easier method of selecting an intersection is to use your middle mouse button to select one of the grids. When you do this a Purple infinite line appears telling you that you are locked onto the grid line. You can then use the perpendicular grid to indicate the intersection. See figure below.



Note: On the Place Member ribbon bar the two smartsteps 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).



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



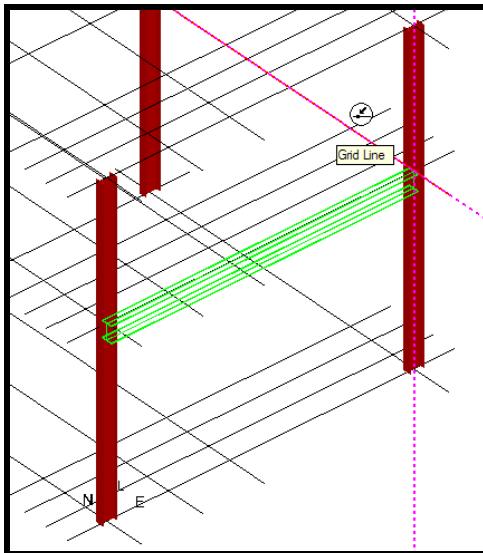
10. While still in the Place Linear Member System Command change the ribbon bar controls to:

- System = A2 -> U04 -> Structural -> Beams
- Type Category = Beam
- Type = Beam
- Section Name = W16x67
- Cardinal Point = 8

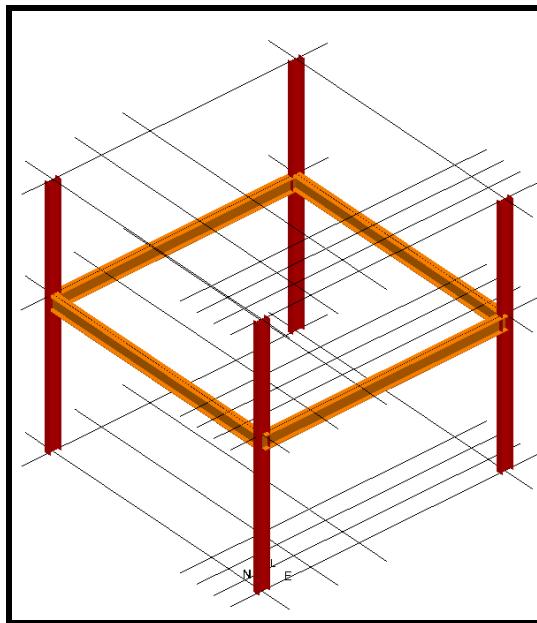
*Note: Use the properties icon in the ribbon bar to open the properties page.
Select the Cross section Tab.*

- 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.



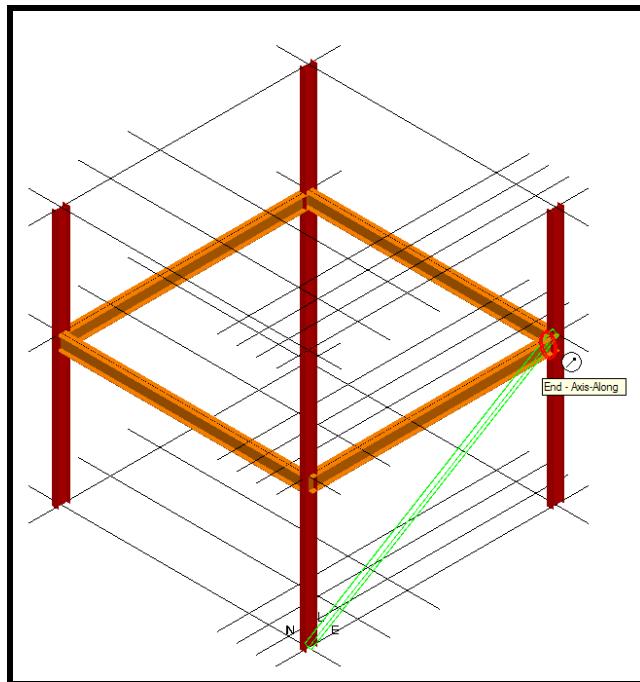
13. While still in the Place Linear Member System command change the ribbon bar controls to:

- h. System = A2 -> U04 -> Structural -> Vertical Braces
- i. Type Category = Brace
- j. Type = Vertical Brace
- k. Section Name = L8x4x1/2
- l. Cardinal Pt. = 1
- m. Angle = 180 degrees

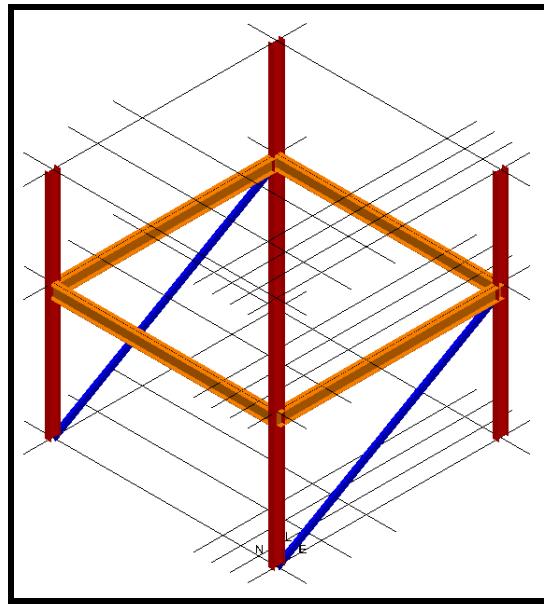
*Note: Use the properties icon in the ribbon bar to open the properties page.
Select the Cross section Tab.*

- 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 Supported and Supporting member), then a Left Mouse click.



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



Lab 1B - Copy / Paste Members – Structure U02

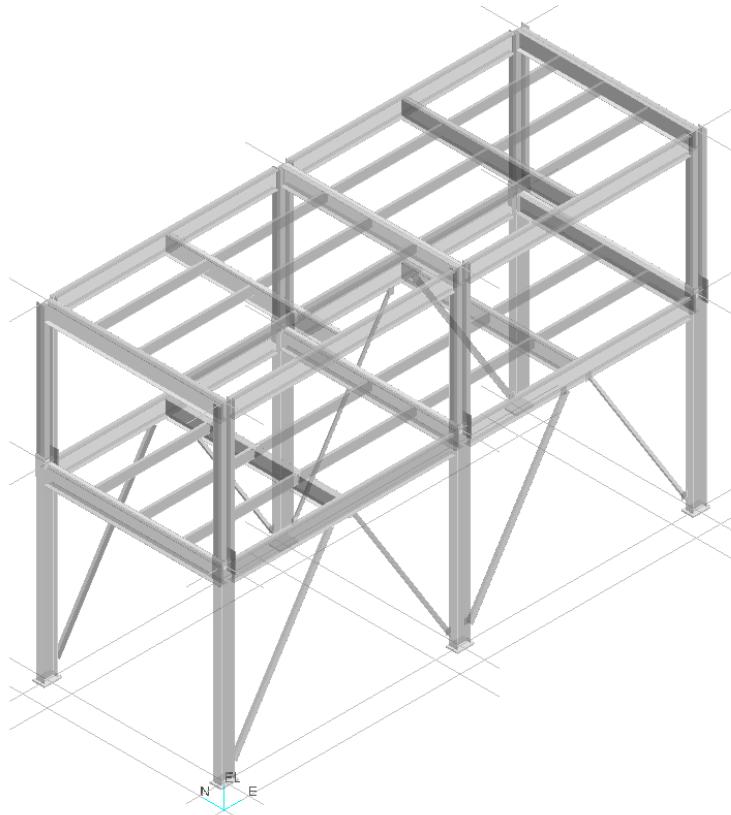
Objectives

After completing this lab, you will be able to:

1. Understand the structure entities and relationships
2. Use the place linear member system command
3. Learn to use SmartSketch service to find placement points for members
4. Use Copy/Paste functionality

In this lab, you will build a simple steel-frame two-story building.

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.



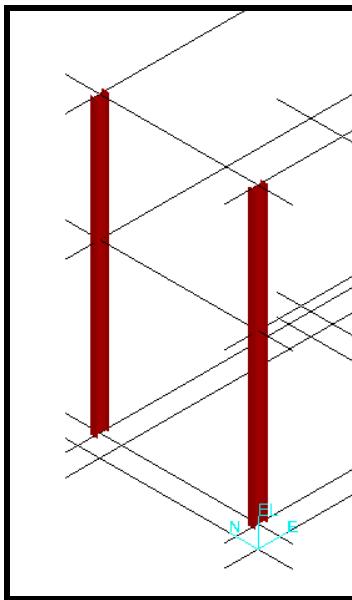
Placing Columns and Beams

1. Open or create a session file and define an appropriate filter for your workspace.
Your workspace should include A2->U02 and CS->U02 CS
2. Go to the Structure Task environment.
3. Make sure the Active Permission Group is set to **Structural**.
4. Select the place linear member system command. System displays the place linear member system smart step ribbon bar.

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

Connection:	By Rule
System:	A2 -> U02 -> Structure -> Columns.
Type Category:	Column
Type:	Column
Section Name:	W14x53
Cardinal point:	5 - Center
Angle:	0 deg
Material:	Carbon Steel
Grade:	A36

6. Place the first column end at the grid intersections **Elevation 0.75m**.
7. Place the second column end at the grid intersections **Elevation 10m**.
8. Repeat the above steps to place the other support columns. Toggle the start/end icons to set the discrete/contiguous placement method if required



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

Connection:	By Rule
System:	A2 -> U02 -> Structure -> Beams
Type Category:	Beam
Type:	Beam
Section Name:	W18x40
Cardinal point:	8 – Top Center
Angle:	0 deg
Material:	Carbon Steel
Grade:	A36

*Note: The next step is to place the first two beams at **Elevation 6m and Elevation 10m**. 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 connection of supporting members.

10. Place the beams at 6m and 10m. Toggle the start/end icons to set the discrete/contiguous placement method.

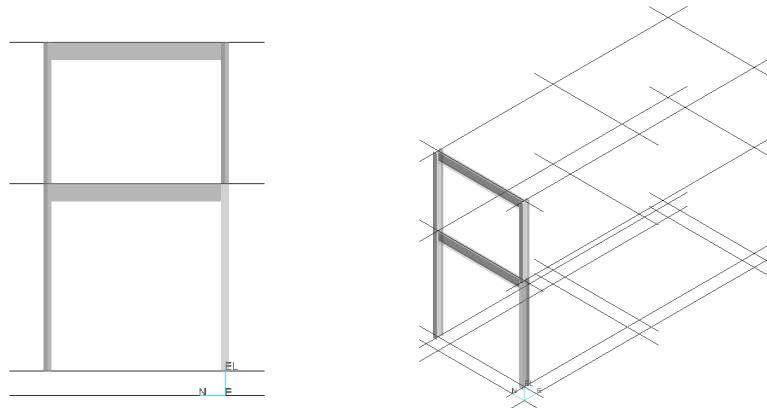


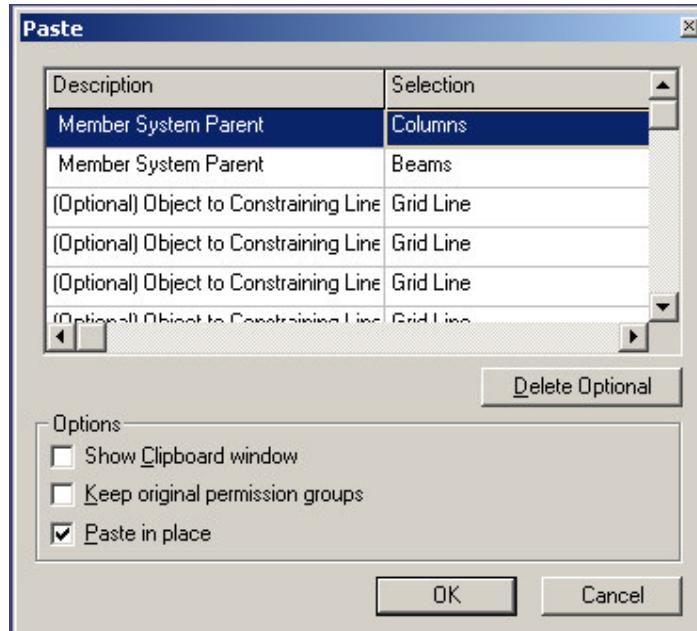
Figure 1 – Elevation and Isometric View of the Structure U02

11. Use the Copy/Paste commands to place the four members at East Plane locations 6m and 12m. See figure 2. Make sure to select the appropriate objects to re-establish the connection in the Paste Special Dialog box.

Hint: Set the select filter to Member System to select all four members

Only two grid lines related objects need to re-establish the connections

Use the Paste command twice



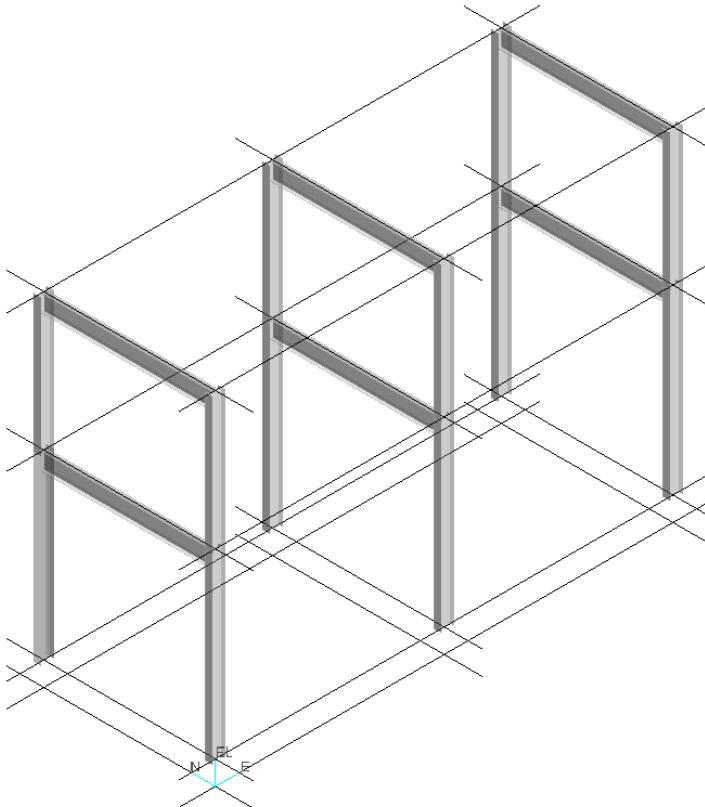


Figure 2 –Isometric View of the Structure U02

Placing Perimeter Beams

The next step is to place the perimeter beams on both first floor frame and second floor frame at **Elevation 6m** and **Elevation 10m** respectively. Beams are placed by selecting and lock constraint on to the column and then locate point-on any grid line passing through that column.

1. Select the place linear member command or toggle the start/end icons to set the discrete/contiguous placement method.
2. Use the view shown in figure 3 to place the perimeter beams. Use the place linear member system ribbon bar to set the active member parameters as follows:

Connection:	By Rule
System:	A2 -> U02 -> Structure -> Beams.
Type Category:	Beam
Type:	Beam
Section Name:	W18x40
Cardinal point:	8 – Top Center
Angle:	0 deg
Material:	Carbon Steel
Grade:	A36

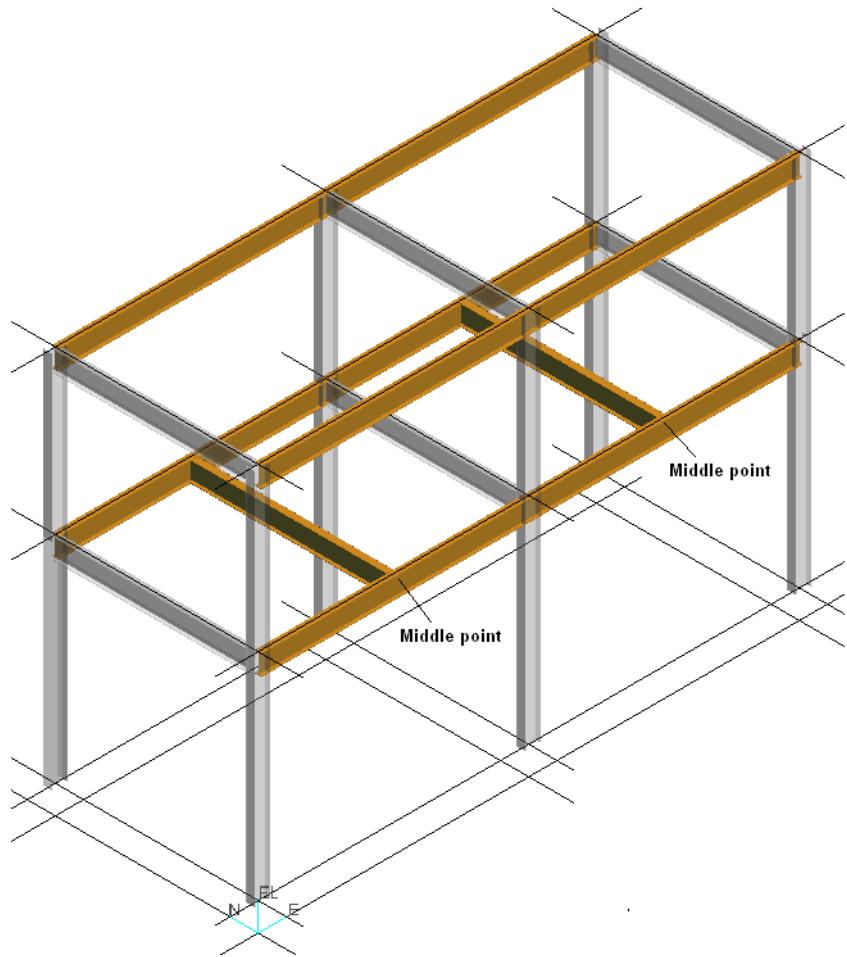


Figure 3 –Isometric View of the Structure U02

Lab 1C - Productivity Commands – Structure U02

Objectives

After completing this lab, you will be able to:

- Use Place Framing Members and Bracing Commands
- Use the Copy / Paste functionality

Framing members and vertical braces will be added to the steel frame built in lab 1B

Placing Intermediate Beams for the first floor frame



1. Select the Place Framing Members command. Use the view shown in Figure 4 to place the structural members
2. Use the Place Framing Member ribbon bar to set the active member properties as follows:

Fit Mode:	By Count
Count:	3
Connection:	By Rule
System:	A2 -> U02 -> Structure -> Beams.
Type Category:	Beam
Type:	Beam
Section Name:	C10x15.3
Cardinal point:	8
Angle:	0 deg
Reflect:	off
Offset:	0
Position:	Perpendicular
Material:	Carbon Steel
Grade:	A36

Then select the first support member, the second supporting member and Finish from the ribbon bar to commit to database.

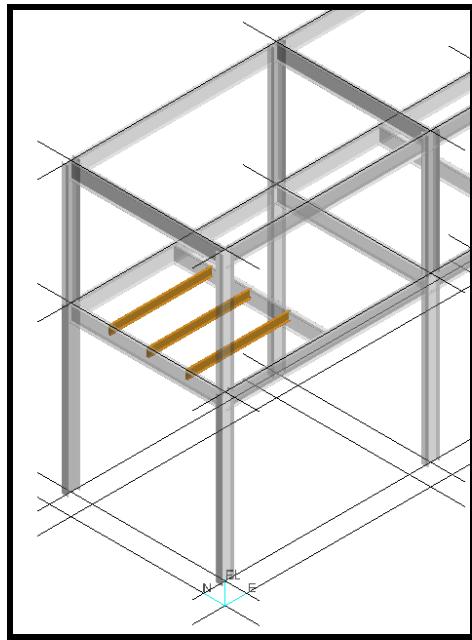


Figure 4 –Isometric View of the Structure U02

3. Repeat across elevation plane at 6m selecting next beam in turn as second supporting member as previous support member 2 automatically upgraded to support member 1.

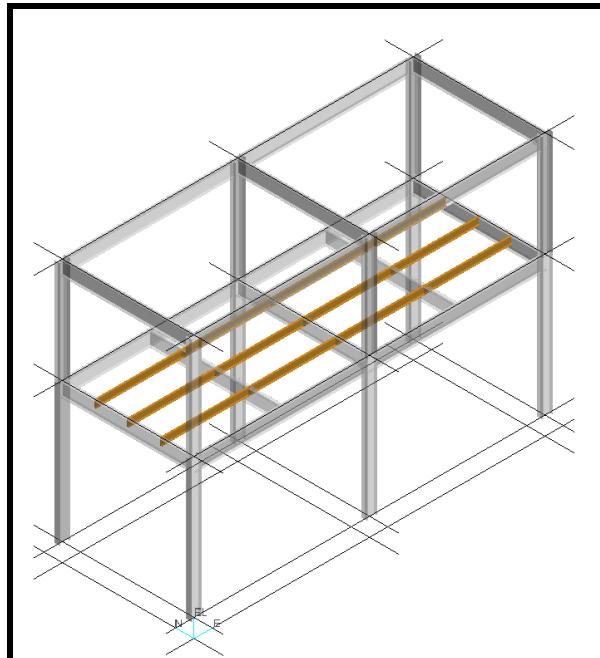


Figure 5 –Isometric View of the Structure U02

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.

4. Set the locate filter to Member Systems.
5. Select 14 members using the Fence method or Multi-select method. See figure 6 for details:

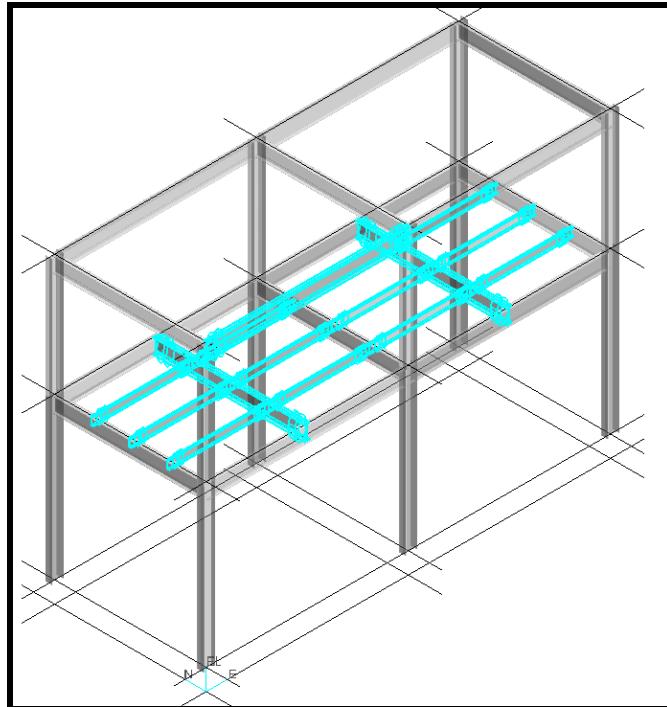
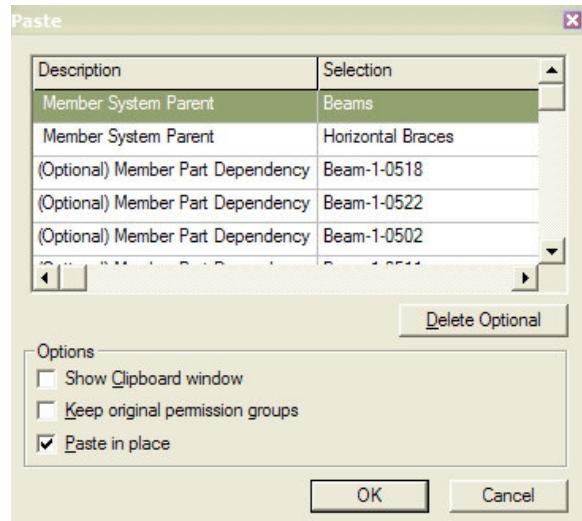


Figure 6 –Isometric View of the Structure U02 System

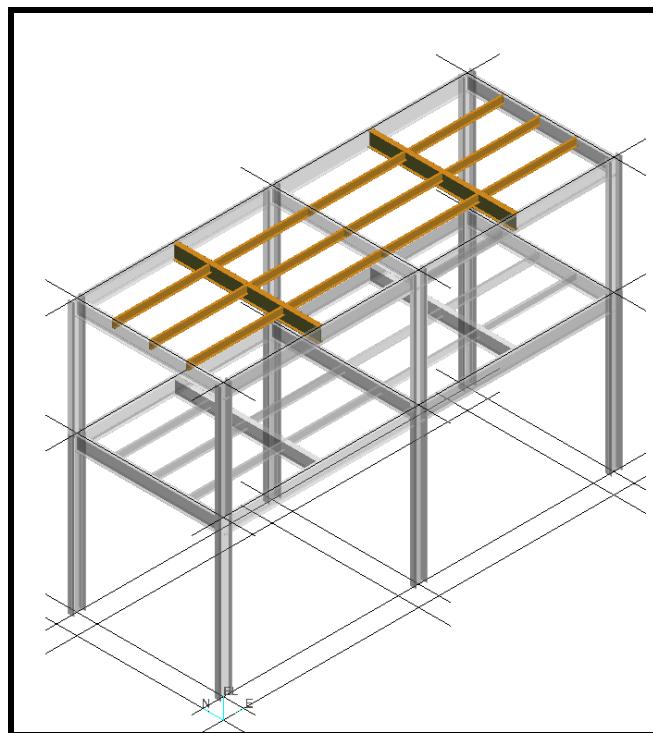
6. Go to the Main Menu and select Edit -> Copy.
7. Select the end of a beam (we just need a point on a object within the select set) as the reference point.
8. Go to the Main Menu and Select Edit -> Paste.
9. Make sure to select the appropriate objects to re-establish the connection in the Paste Dialog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

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



10. Hit “OK” button to commit the transaction

Your View should now resemble the following graphic:



Place Vertical Braces

11. Select the Place Bracing command 
12. 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
Material:	Steel- Carbon
Grade:	A36

13. Select the first column for the first SmartStep (see figure 7).

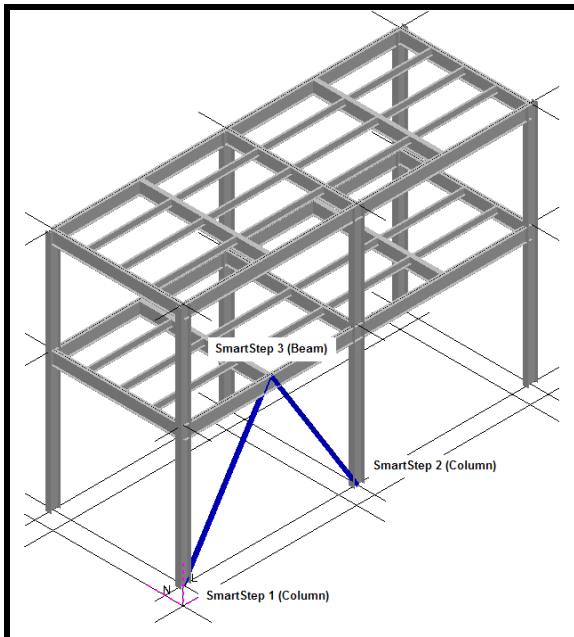


Figure 7 –Isometric View of the Structure U02

14. Select the second column for the second SmartStep

15. Select the first beam for the third SmartStep

16. Select finish  to commit to database

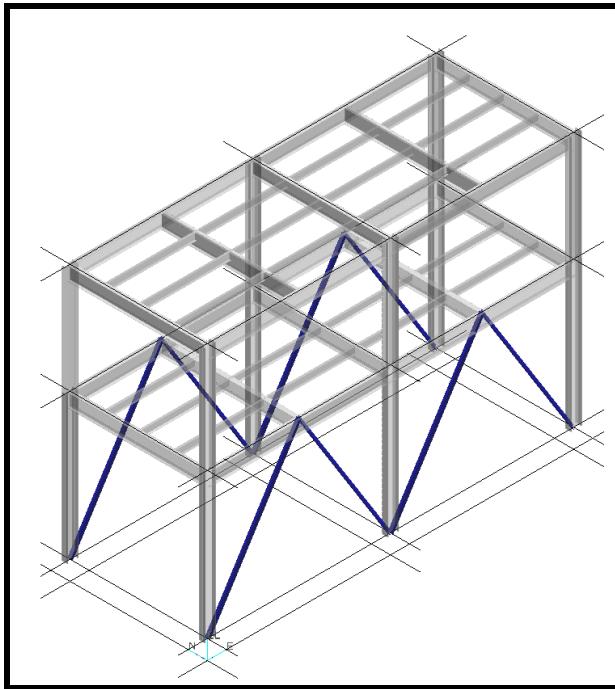


Figure 8 –Isometric View of the Structure U02

17. Repeat for other three bays

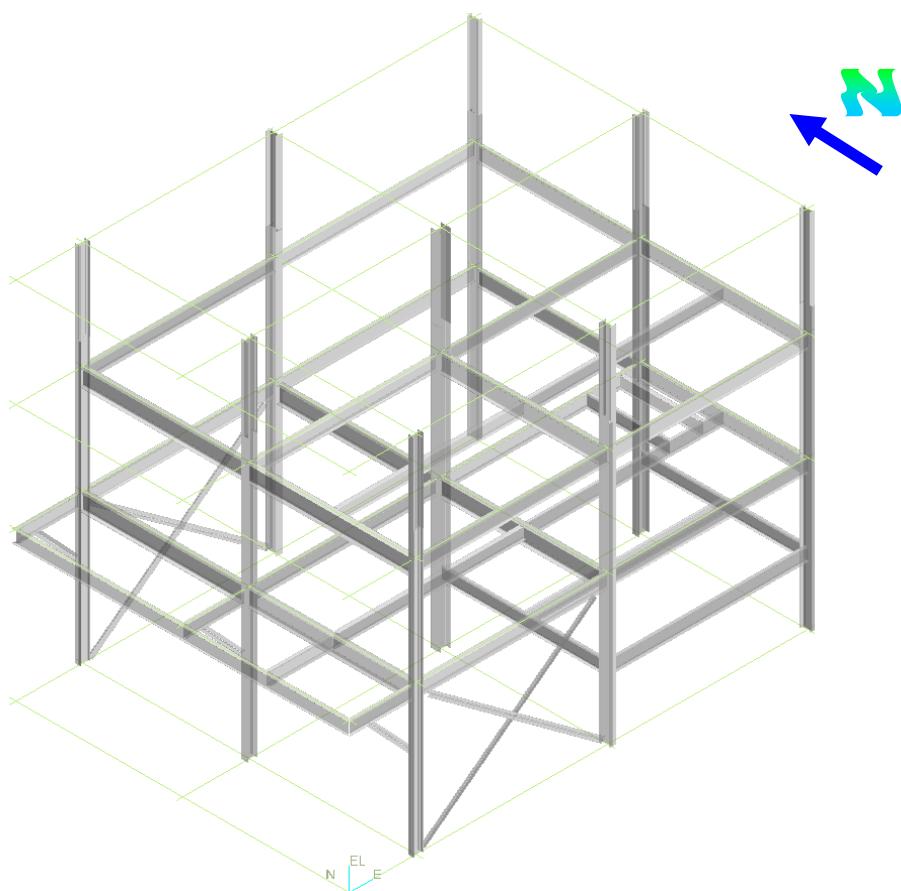
Lab 2 - Structure Modeling - Building U03

Objective

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

To build the structure below using the U03 grid previously created.

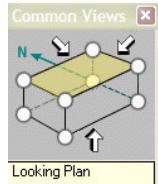


Part I: Columns

- 1 Open or create a session file and define an appropriate filter for your workspace which should include the A2 -> U03 and CS -> U03 CS systems.
- 2 Go to the Structure Task environment

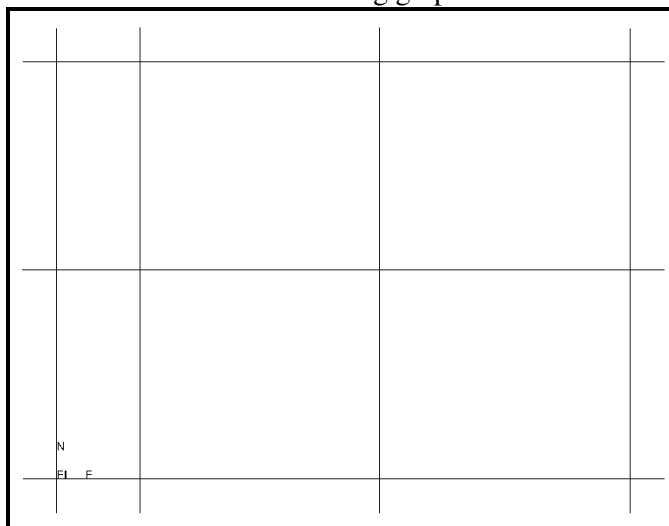
Note: Notice that the grids you created in lab 1C are in this model. Though not absolutely necessary, grids are extremely useful as references for drawing dimensions. They also help in the modeling process by providing reference points at which you can place structural members.

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 the Place Columns at Grid Intersections command. The system displays the SmartStep ribbon bar.

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

System:	A2 -> U03 -> Structural -> Columns
Type Category:	Column
Type:	Column
Section Name:	W14x53
Cardinal Point:	5
Angle:	0 deg
Reflect:	off
Material:	Steel – Carbon
Grade:	A36

7. Use Workspace Explorer and select Elevation 0m. object for the first SmartStep.
8. Use Workspace Explorer and select Elevation 14m. object for the second SmartStep.
9. Place a fence around the floor grid intersections as shown in figure 1

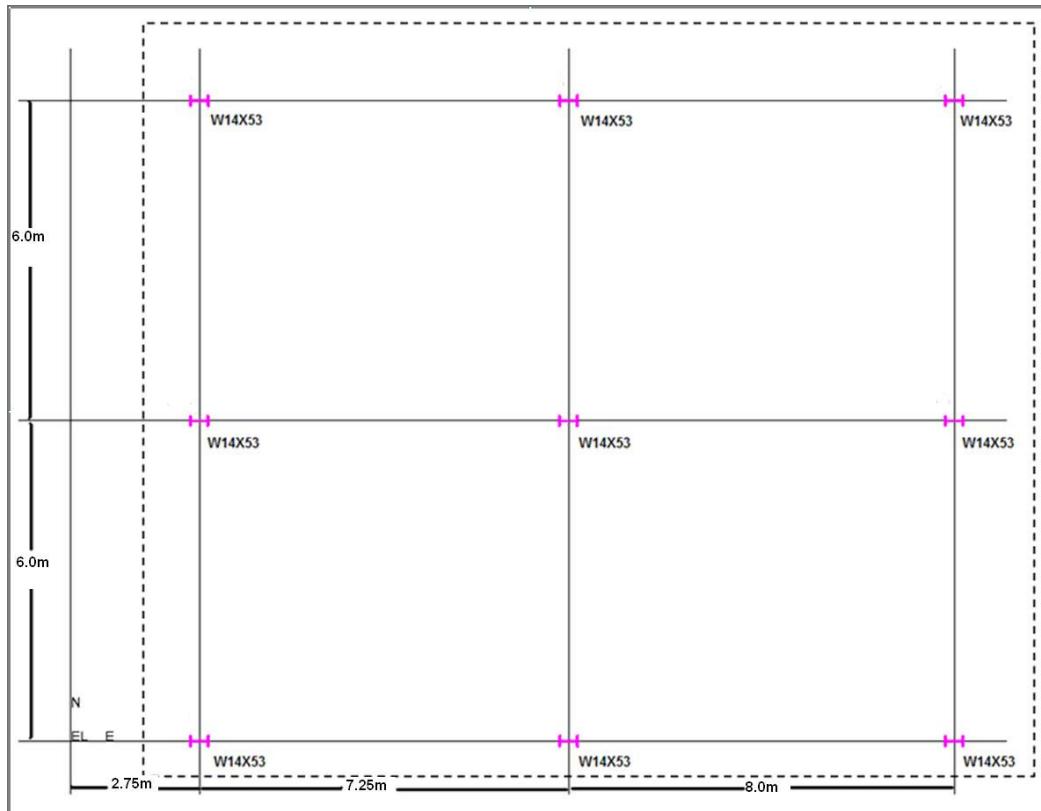


Figure 1 – Floor Plan U03

10. Click “Finish” button to commit columns to database.

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.

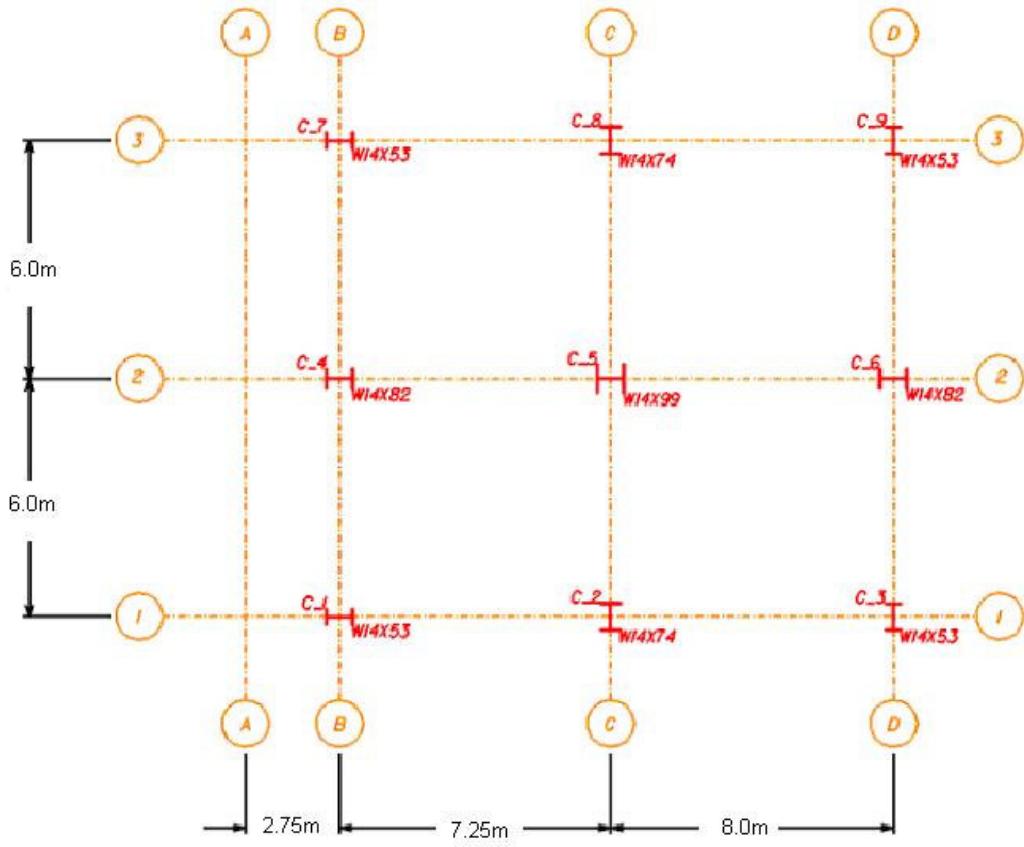


Figure 2 – Floor Plan U03

Place split connections at elevation 11m using the Place Split command

13. Select the Split command  from the task bar.
14. Make sure the Split Status is set By Rule
15. Select the Plane El11.000m from the Workspace explorer or the Ruler.
16. Select all the columns to split and click “Finish” to create the Split Connections as highlighted in Figure 3

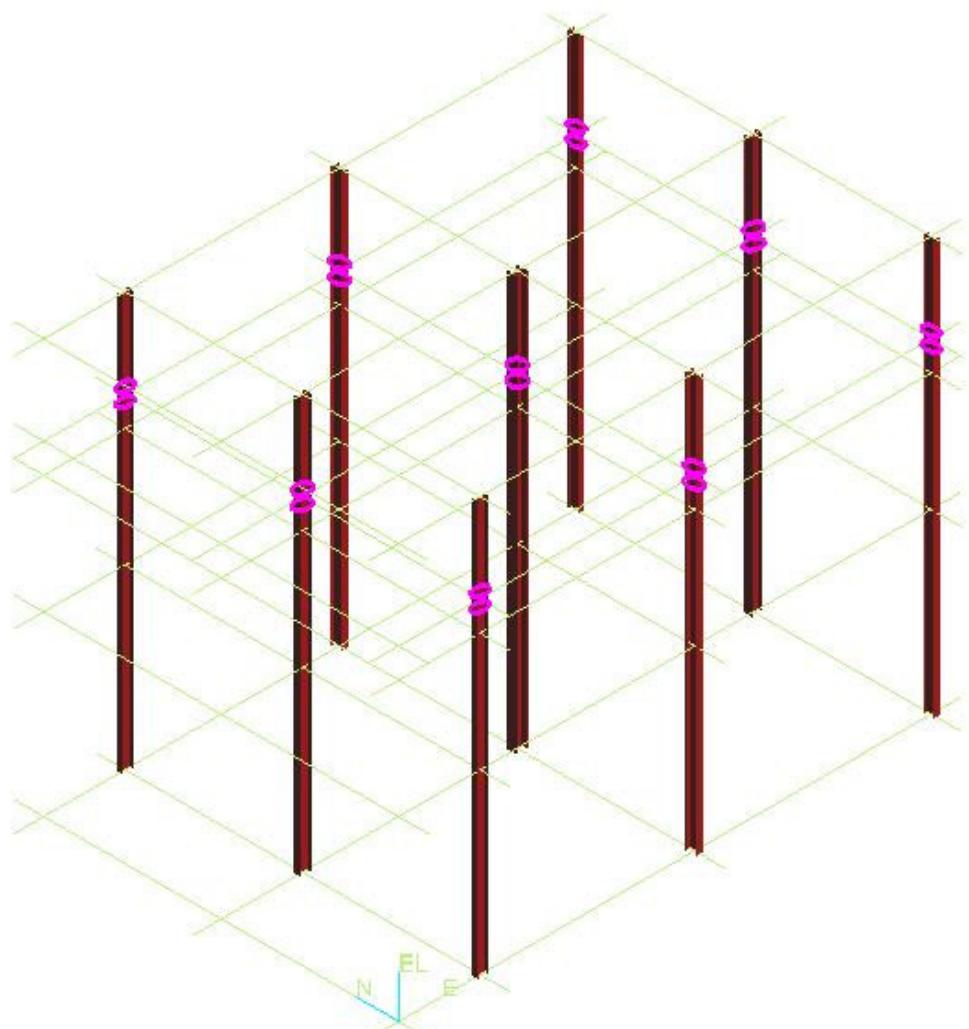
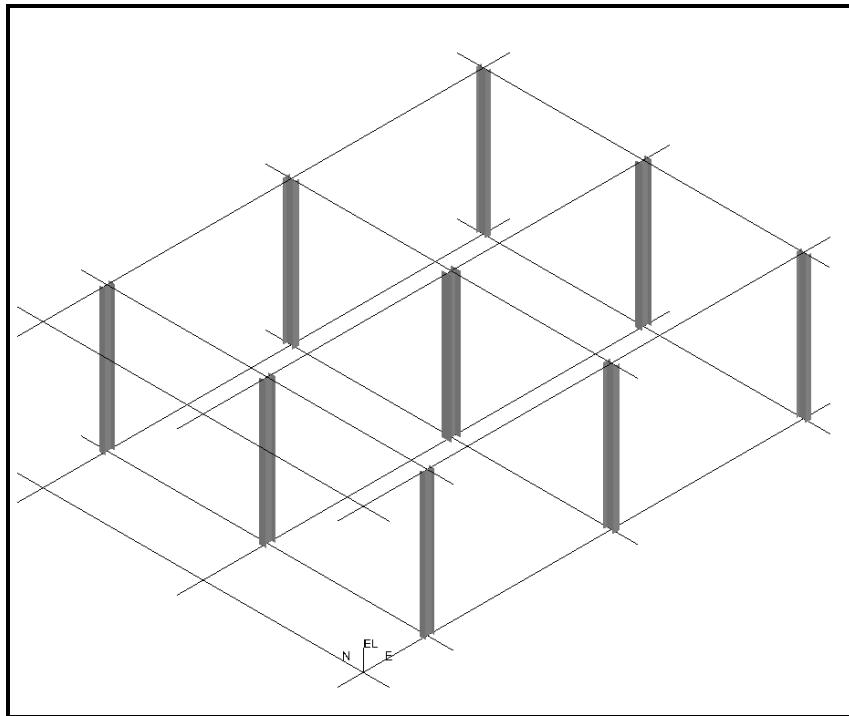


Figure 3 – Isometric View of Structure U03 showing Split Connections

Part II: First Floor Frame

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

Hint: Use the appropriate grids lines at elevations 0m and 6m to define volume



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

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

Place all perimeter members

System: A2 -> U03 -> Structural -> Beams.

Place all intermediate members in

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

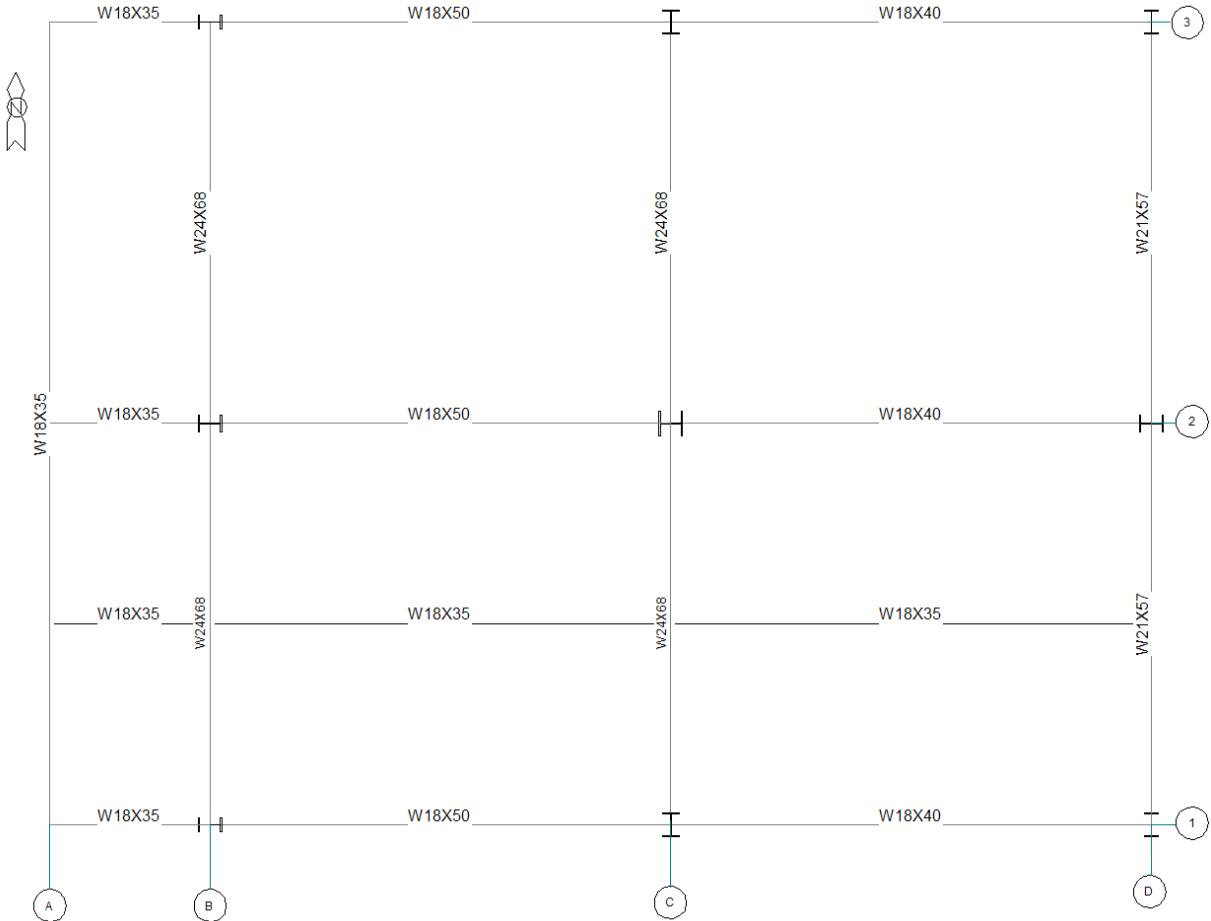


Figure 4 – U03 First Floor Plan (6m elevation)

Part III: Second Floor Frame

19. Select the Clear Clipping command to remove the clipping volume
20. Use the Clip by Volume to isolate the Second Floor as shown in Figure 5 to assist the structural member placement at elevation 10m
21. Select the place linear member command.
22. Set the active member parameters as follows

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

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

Place all intermediate members in
System: A2 -> U03 -> Structural -> Horizontal Braces.

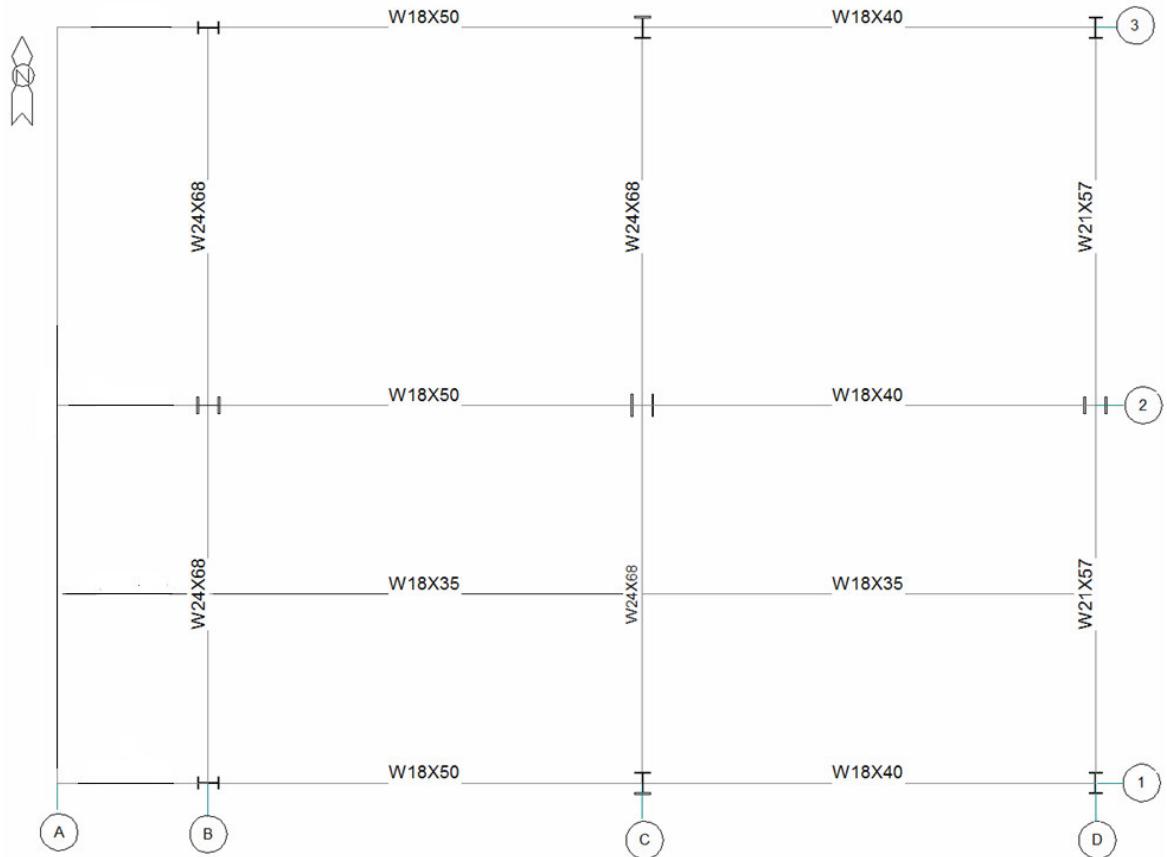


Figure 5 – U03 Second Floor Plan (10m elevation)

23. Select Clear Clipping to remove clipped volume'

Your view should resemble the following graphic:

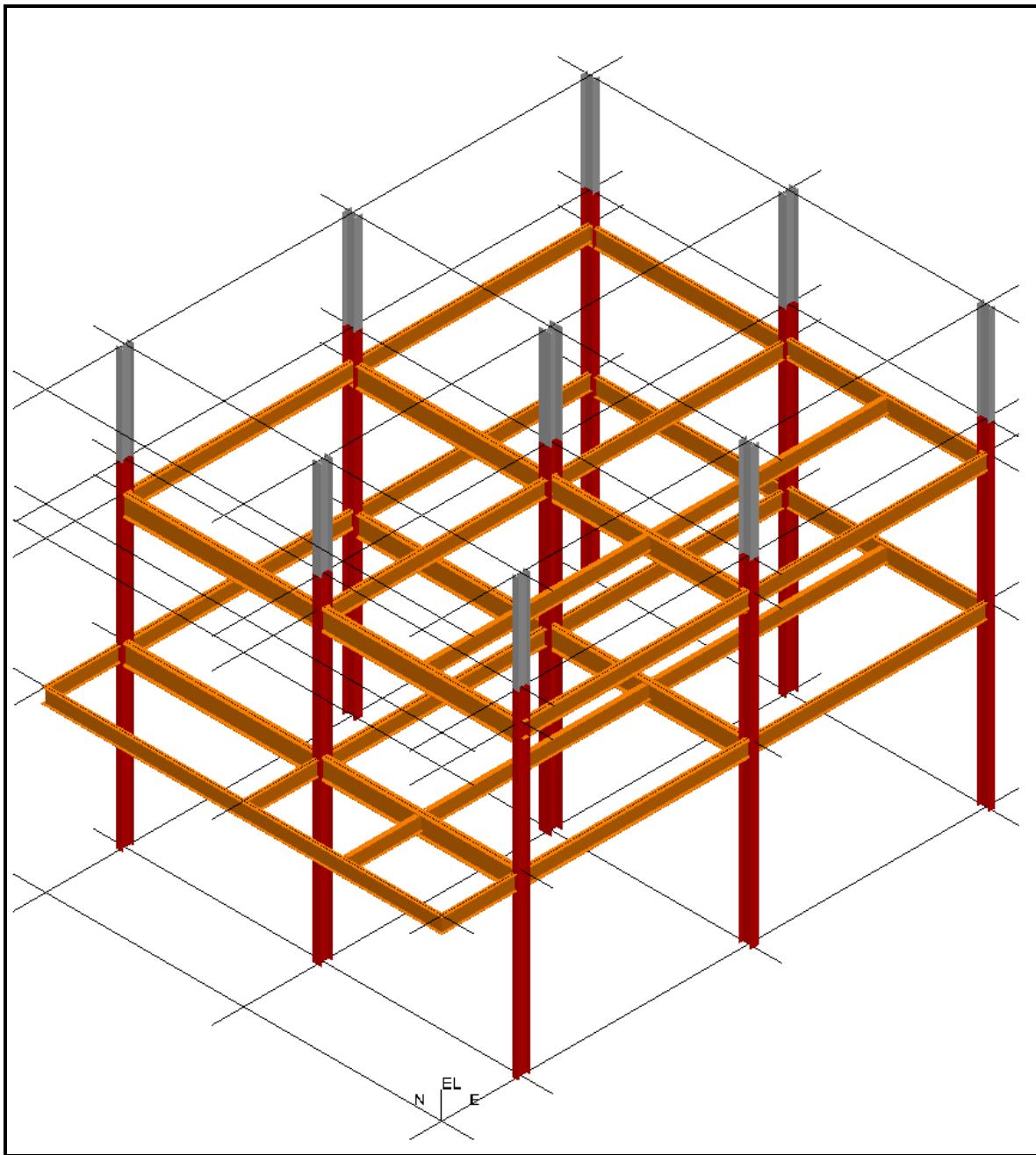


Figure 6 – ISO View of the Structure U03

Part IV: Vertical Braces

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



25. Select the place linear member command to place the support brace

26. Use Pinpoint to place Vertical bracing as per Figure 7 Bracing Detail

27. Set the parameters as

Connection:	By Rule
Type Category:	Brace
Type:	Vertical Brace
Cardinal point	5
System:	A2 -> U03 -> Structural -> Vertical Braces.
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36

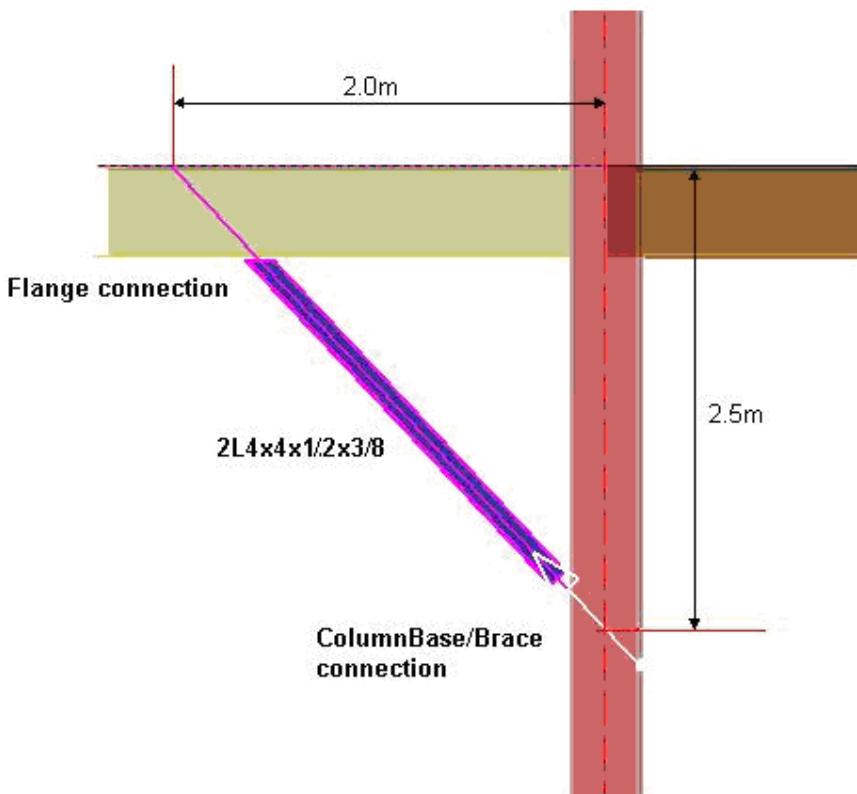
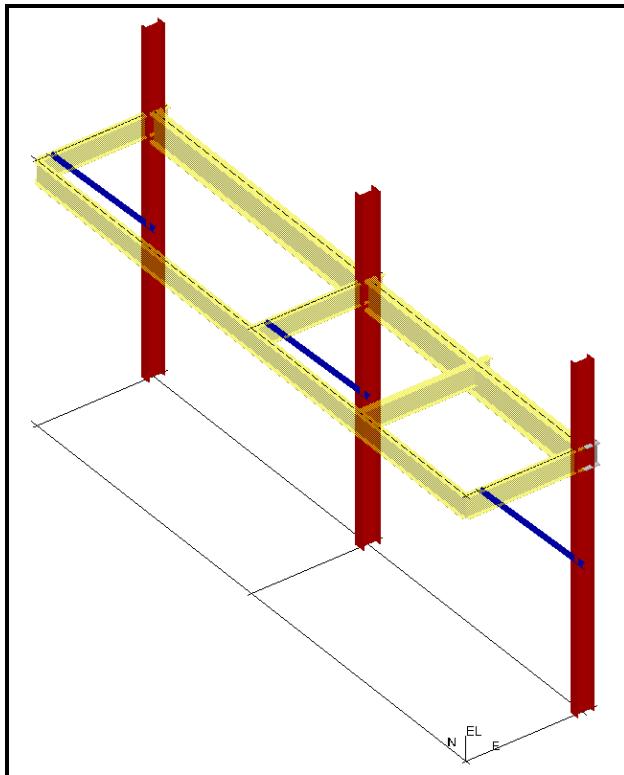


Figure 7 – Bracing Detail

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



29. Use Place Bracing Command to place cross bracings on the Structure U03 as in figure 8.

30. 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:	2L4x4x1/2x3/8
Cardinal point	5
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36

31. Select first column for the first SmartStep (See figure 8).
32. Select second column for the second SmartStep.
33. Select Beam for the third SmartStep.
34. Select “Finish” button to commit the transaction.

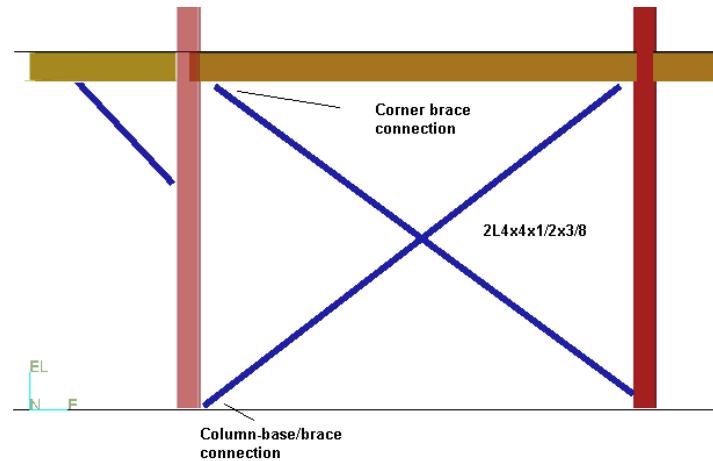


Figure 8 – North View of Building 1

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

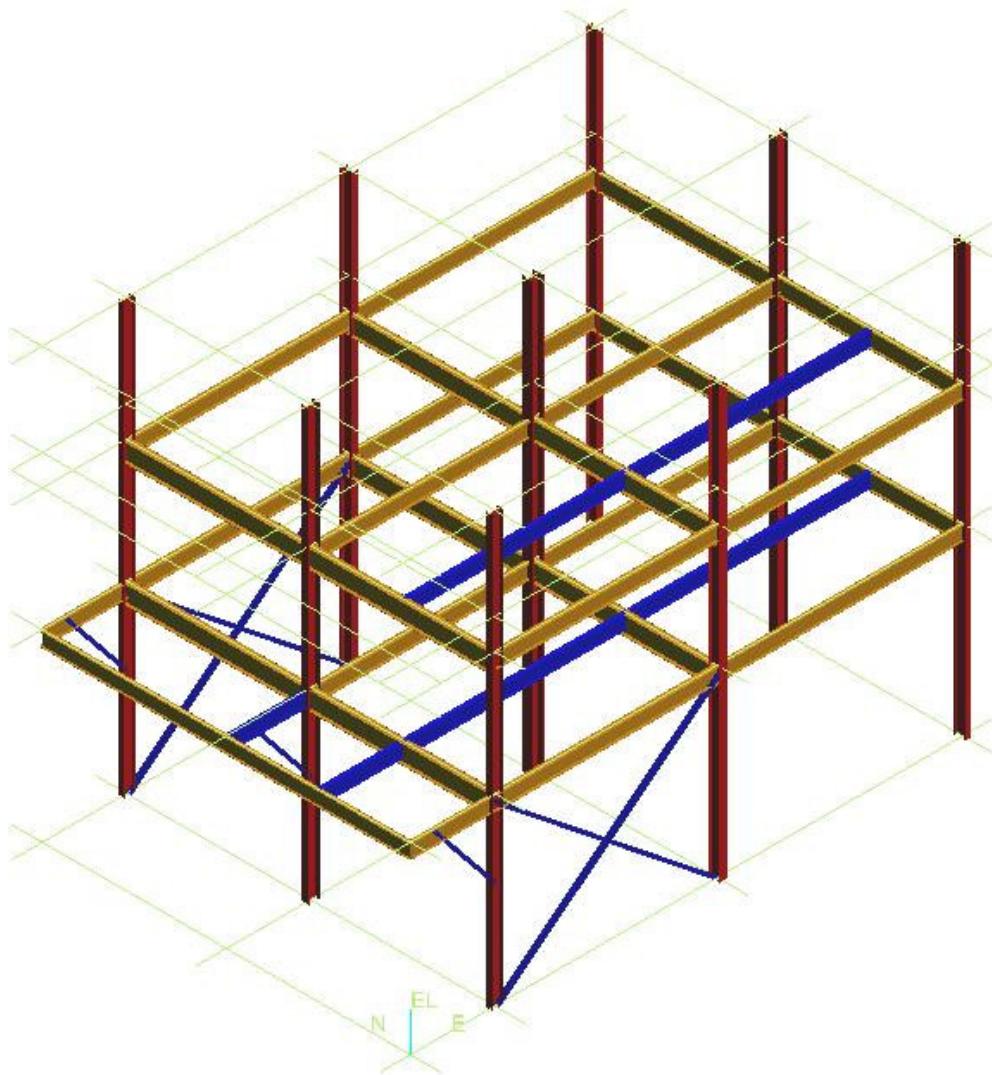


Figure 9 – ISO View of U03 after Vertical Bracing

Part V: Horizontal Braces

36. Select the place linear member command
37. Place two horizontal beams on the first floor frame as shown in Figure 10

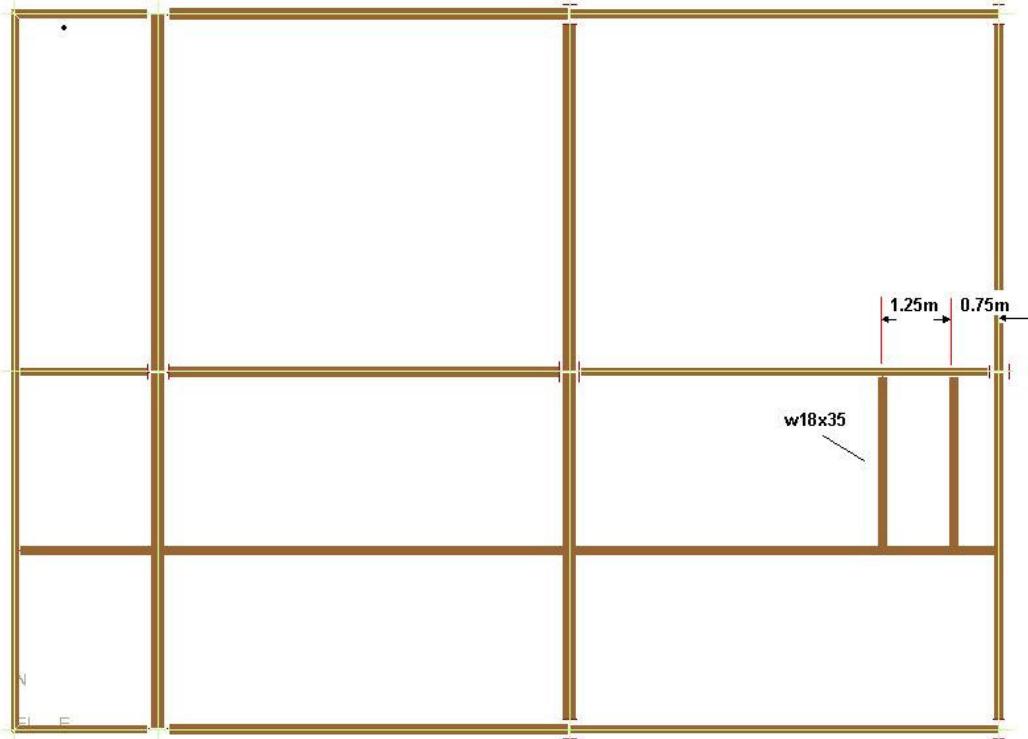
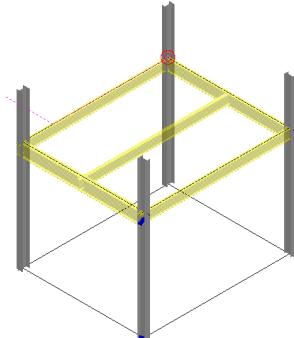


Figure 10 – U03 First Floor Frame El – 6m

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

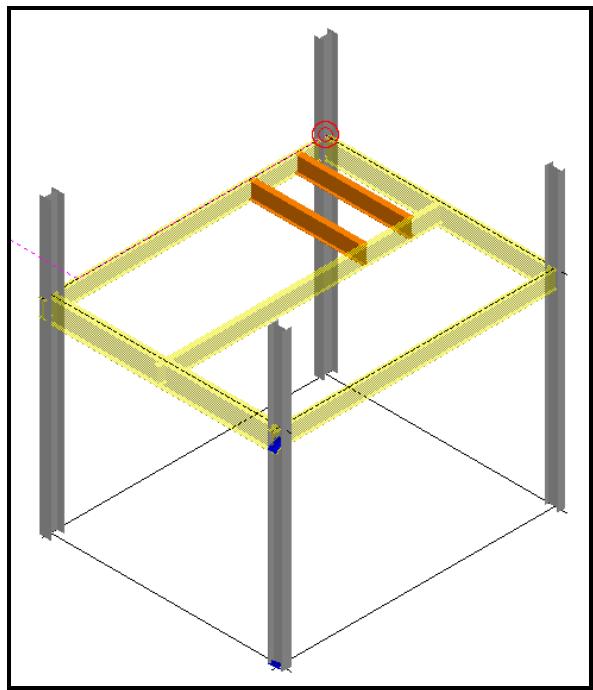


41. Use the place linear member ribbon bar to set the active member parameters as follows:

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

42. 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

43. Place four horizontal beams on the far-east bay as shown in Figure 11

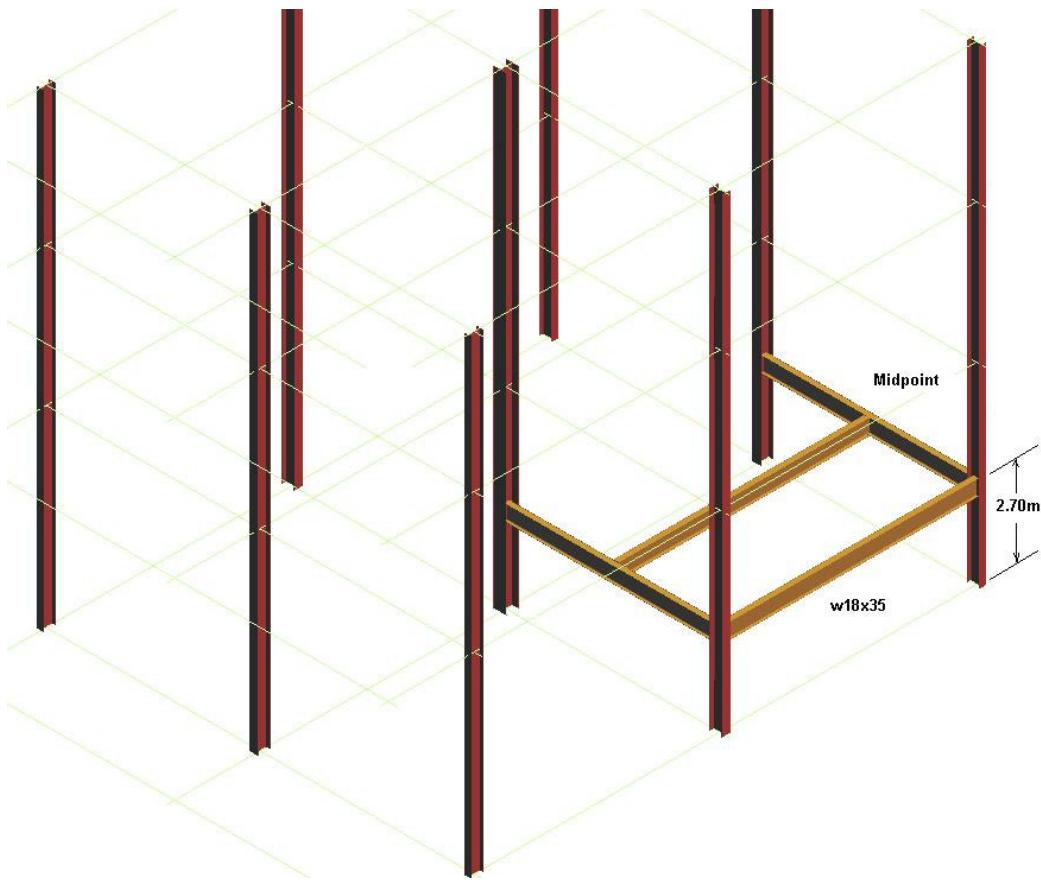


Figure 11 – ISO View of U03

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

45. Select Place Linear Member Command.

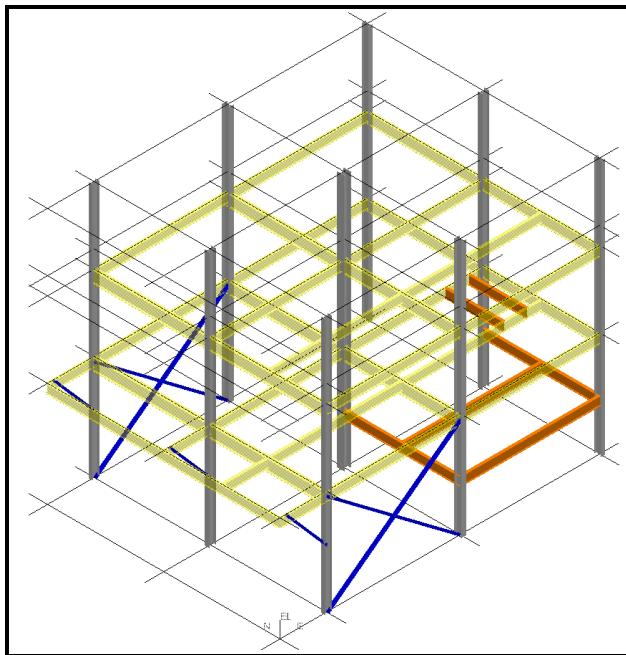
46. Set the parameters as

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

47. 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.

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

Your View should now resemble the following graphic:



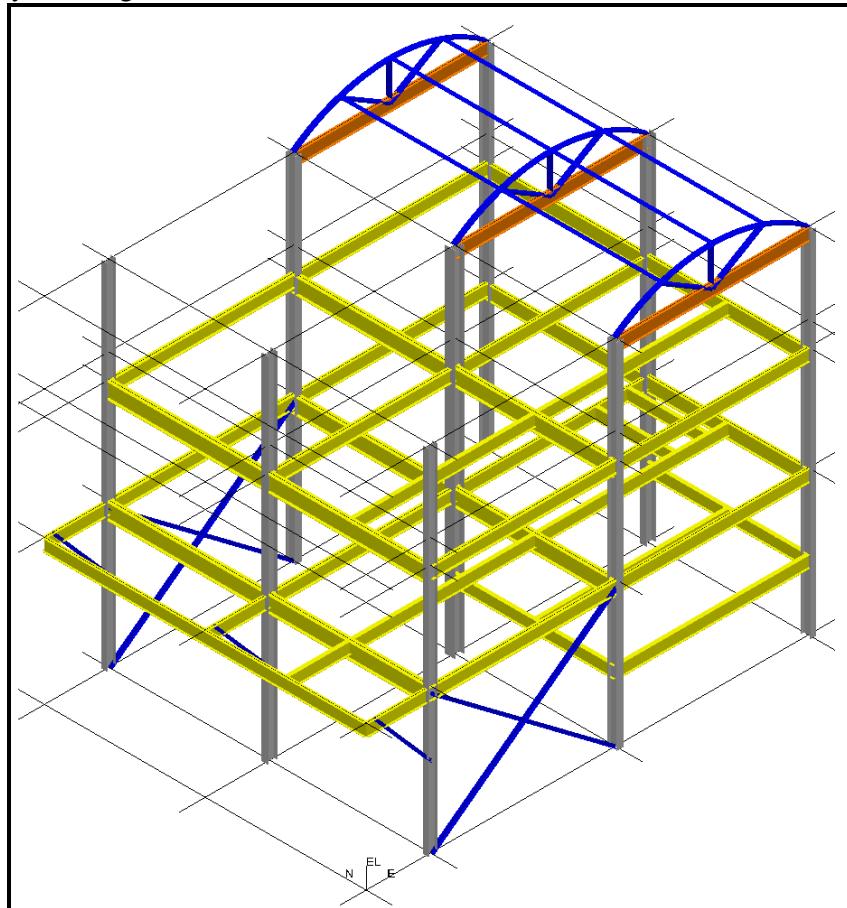
Lab 3 - Curve Member System - Building U03

Objectives

After completing this lab, you will be able to:

4. Understand the structure entities and relationships
5. Use Place Curve Member System Command
6. 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.



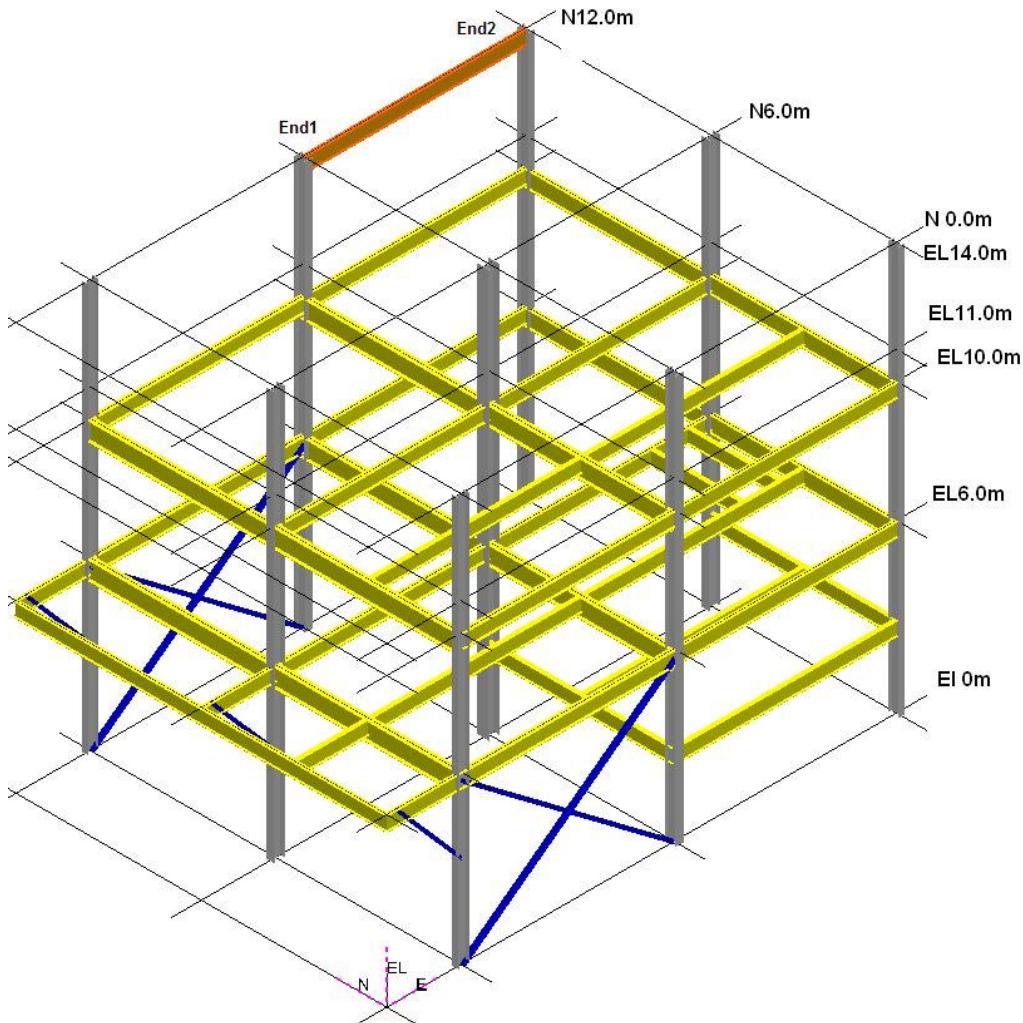
Placing Beams to support the trusses

1. Select the Place Linear Member System Command.
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 12.0m and Elevation 14.0m intersection.
4. Place Member End2 at the end of the column located at the other North Plane 12.0m and Elevation 14.0m intersection.



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

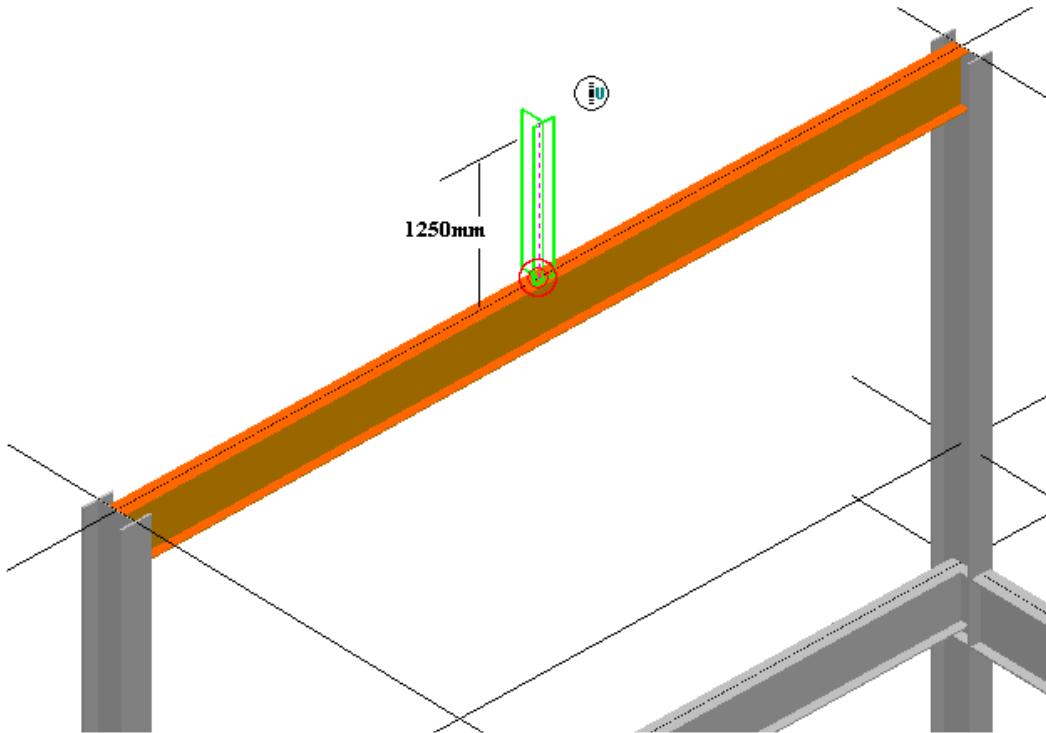
5. Select Place Linear Member System Command.
6. Set the parameters as

System:	A2->U03 -> Structural ->Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	WT8X22.5
Cardinal point	10 - Centroid
Angle:	-90 deg

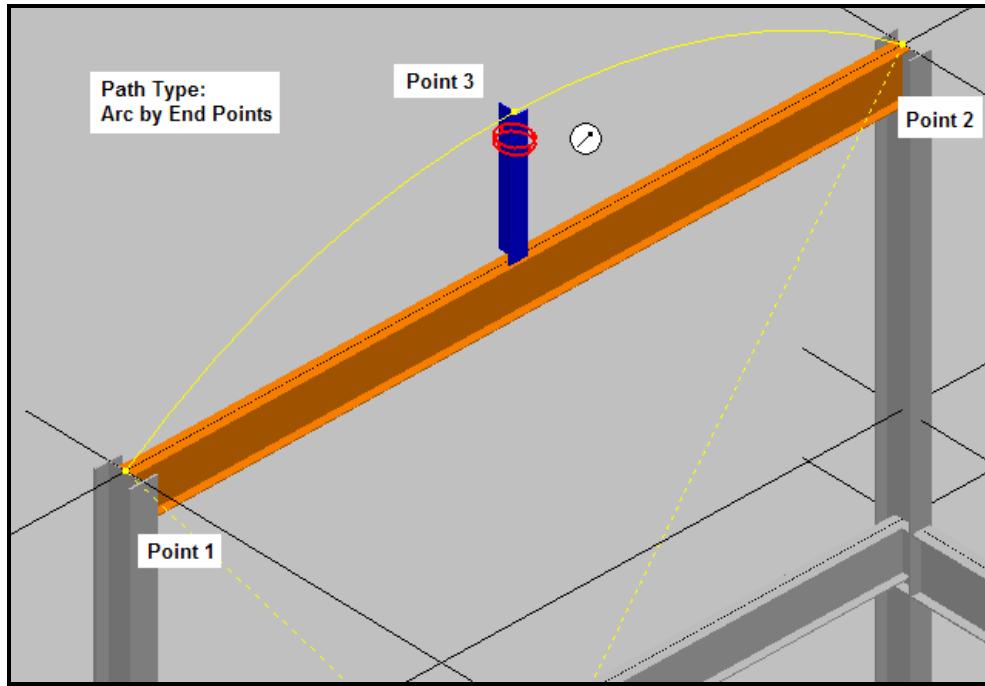
Reflect:	off
Priority:	Secondary
Material:	Steel-Carbon
Material:	A36

6. Use SmartSketch service to locate the mid-point geometric of the beam to place End1.
7. Use PinPoint key-ins to place End 2 at 1250mm 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 12.0m and Elevation 14.0m 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 12.0m and Elevation 14.0m intersection.
13. Place Point 3 at the end of the vertical brace that you have just placed.



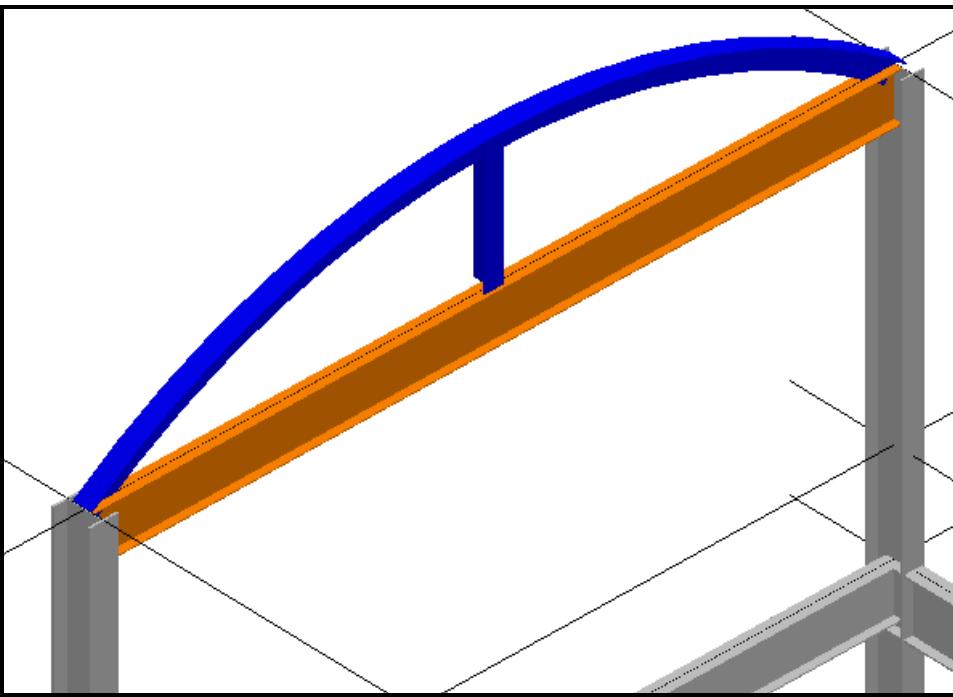
14. Hit the “Finish” Path button. System returns to the SmartStep ribbon bar.

15. Set the parameters as

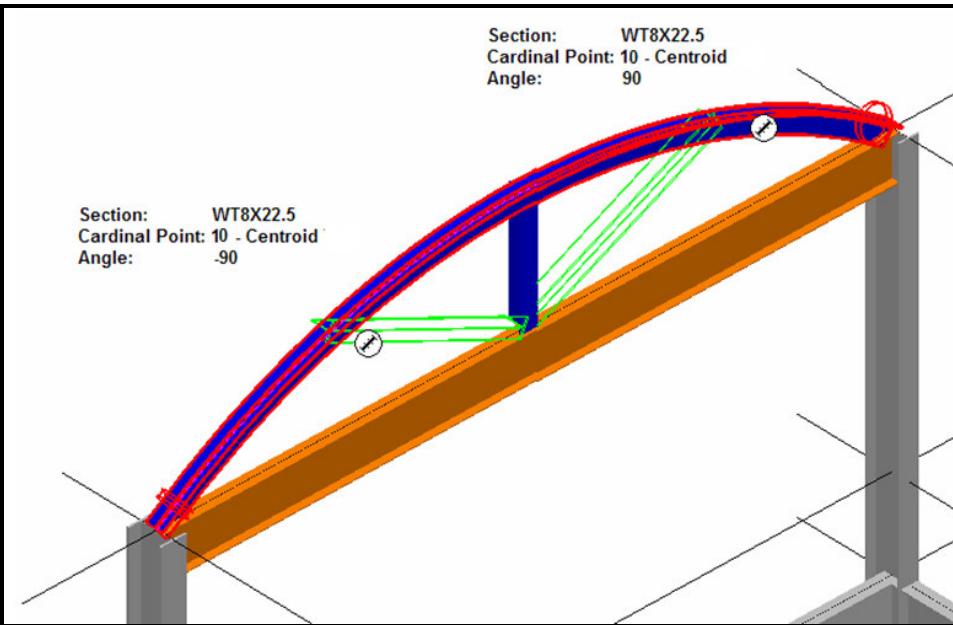
System:	A2->U03 -> Structural ->Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	WT8X22.5
Cardinal point	10 - Centroid
Angle:	0 deg
Reflect:	off
Priority:	Secondary
Material:	Steel- Carbon
Grade:	A36

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

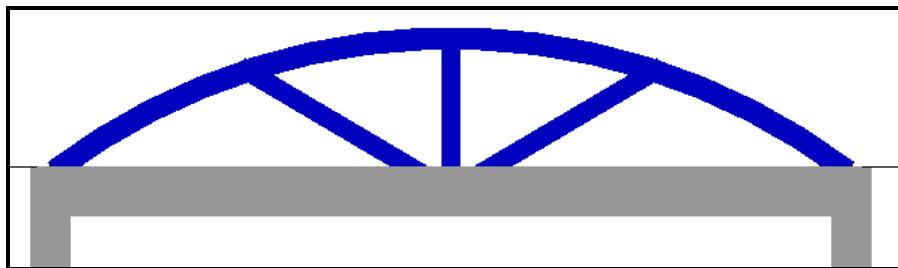
Your View should now resemble the following graphic:



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:

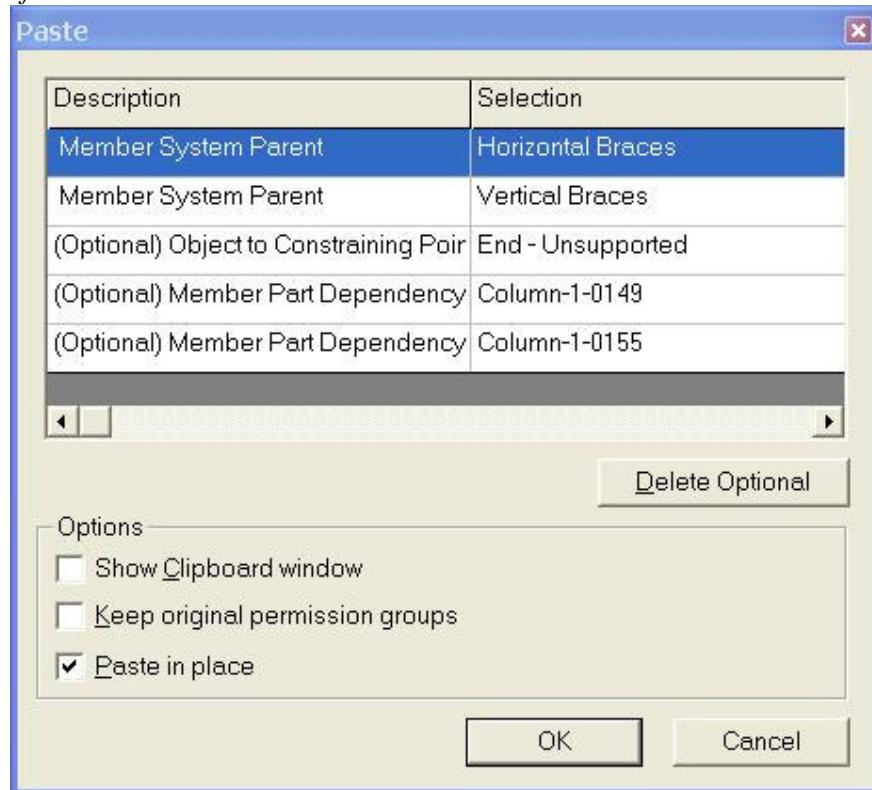


Your View should now resemble the following graphic:

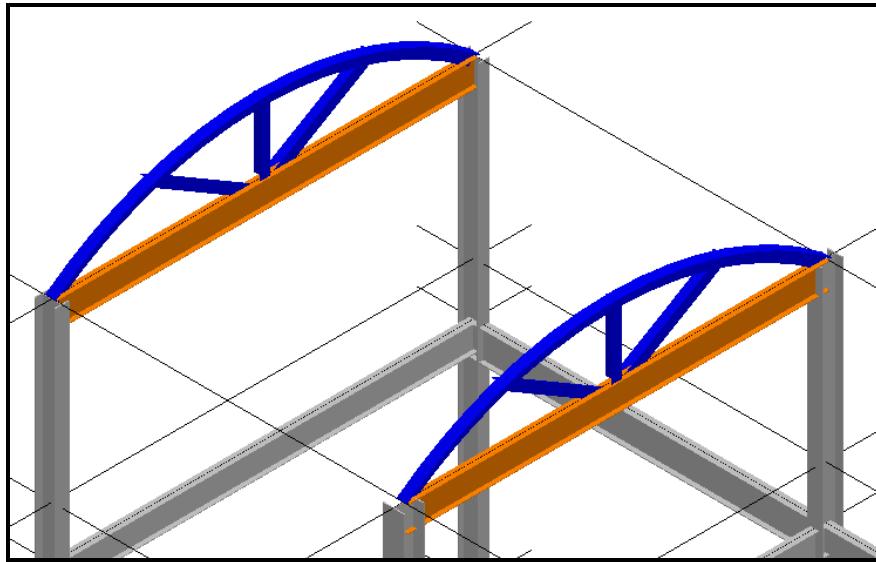


18. Set the locate filter to Member Systems.
19. Select the curve member, the support beam and the three vertical braces using the fence method or the Multi-Select method.
20. Go to the Main Menu and select Edit -> Copy.
21. Select the end of the column as the reference point.
22. Go to the Main Menu and Select Edit -> Paste.
23. Re-establish all the relationships with the columns located in North Plane 6.0m 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.

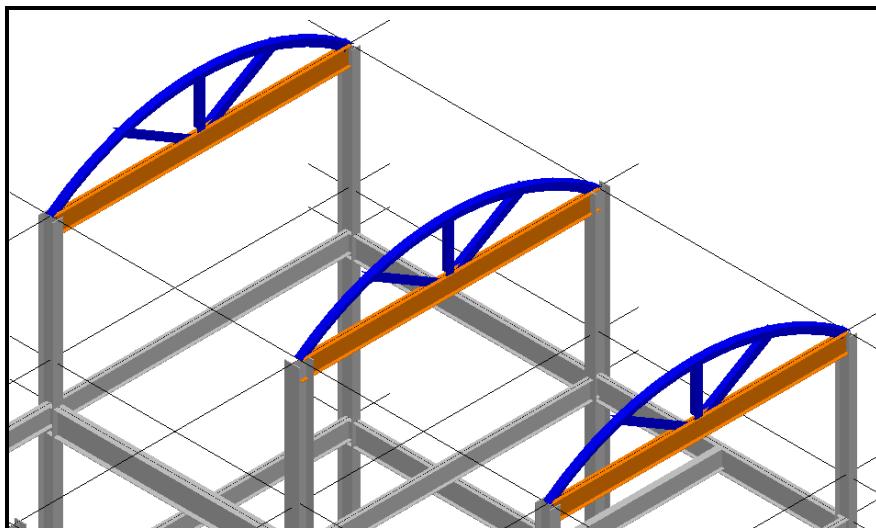


24. Hit “OK” button to commit the transaction.



25. Repeat the Paste step to place the next truss at North Plane 0.0m.

Your View should now resemble the following graphic:



26. Select Place Framing Members command. Use the view shown in Figure 12 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

Position: Toggle to Skewed

Angle: 0 deg
Priority: Secondary
Material: Steel- Carbon
Grade: A36

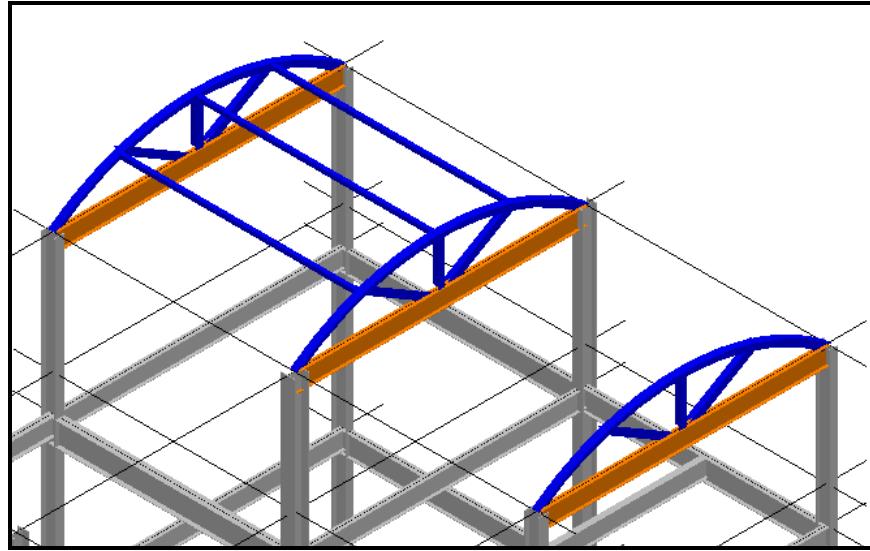


Figure 12 – ISO View of the Structure U03

28. Continue using Place Framing Members command by selecting Supporting Member2 to place framing members as shown in Figure 13.

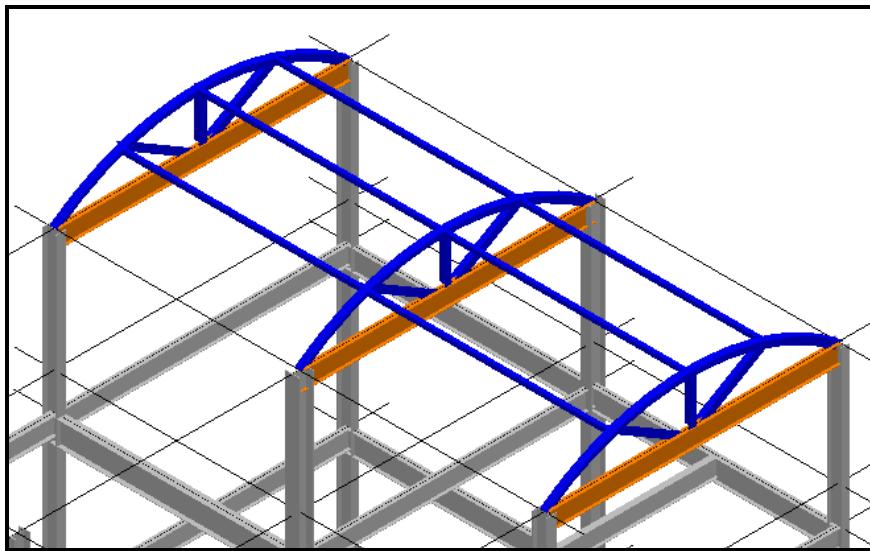


Figure 13 – ISO View of the Structure U03

Lab 4 - Frame Connections

Objectives

After completing this lab, you will be able to:

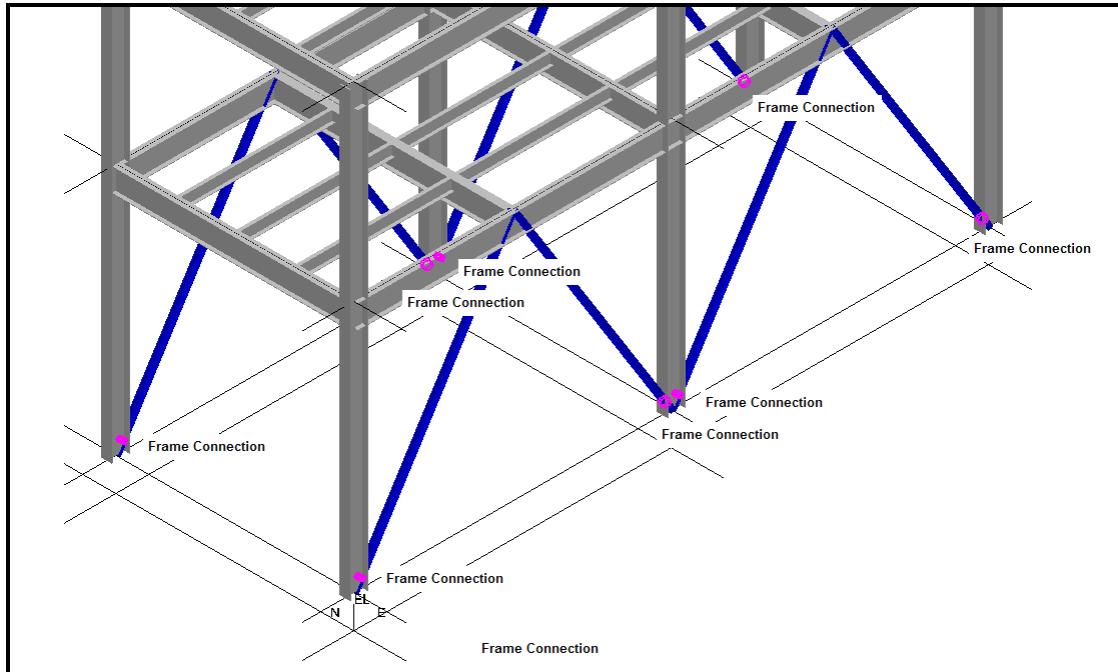
7. Understand the frame connection entities and relationships
8. Edit the Member Part and Frame Connection properties

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

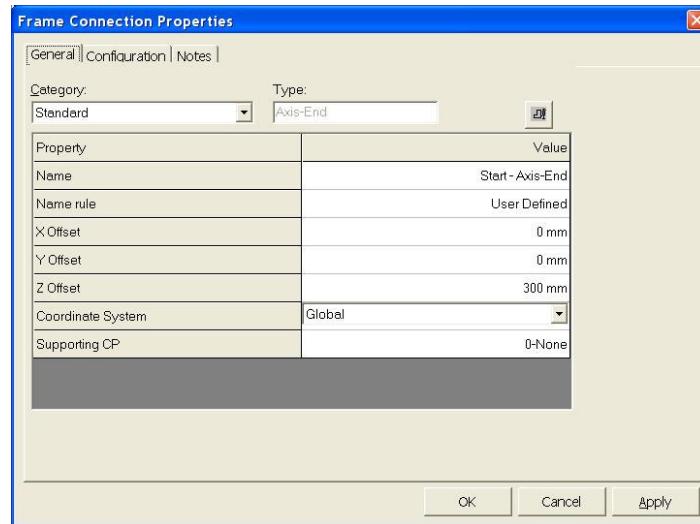
1. Open or create a session file and define an appropriate filter for your workspace.
Your workspace should include the A2 -> U02 and CS -> U02 CS systems.

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 the end of the vertical braces. See figure below:



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

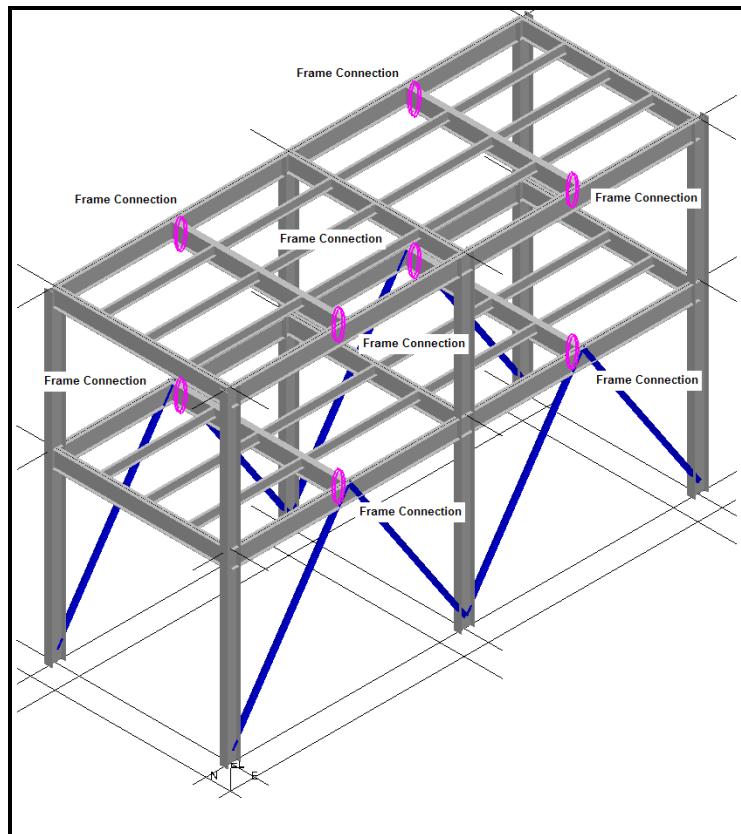


7. Keyin 300mm for Z Offset. Hit “OK” button to commit the transaction.

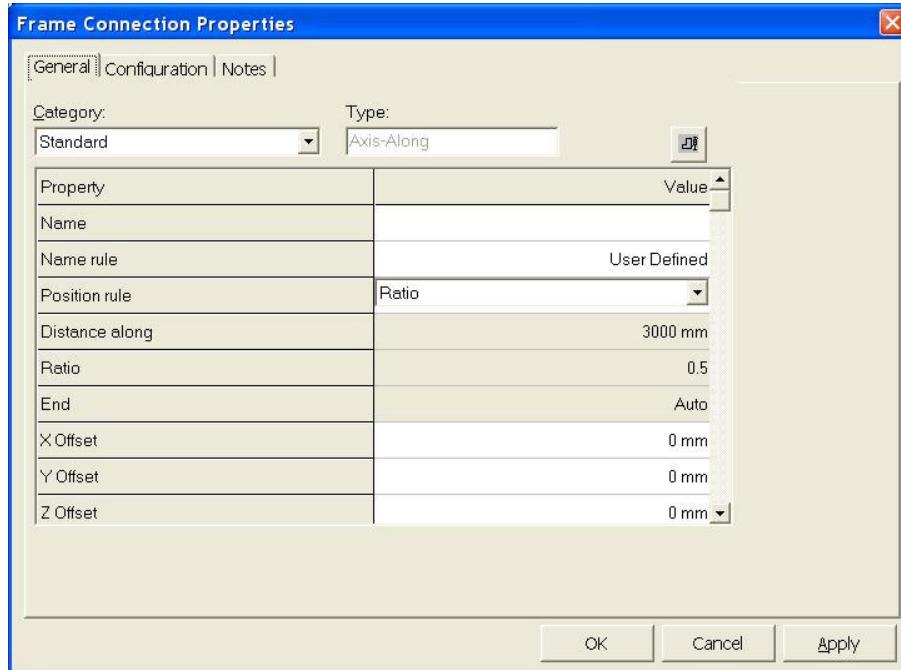
Next, you will set the Position Rule from Intersection to Ratio for the intermediate beams in the Structure U02.

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.

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



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



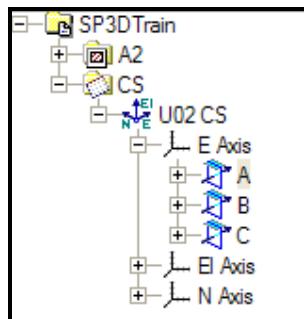
10. Change Position Rule from Intersection to Ratio. Hit “OK” button to commit the transaction.

Next, you will move the East Grid Plane 0.0m in the Structure U02.

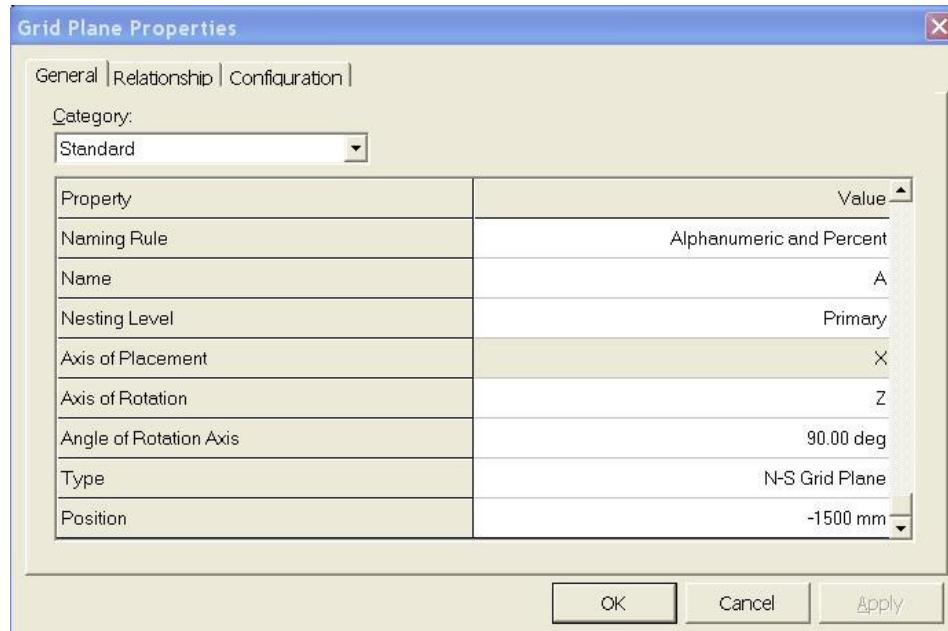
11. Set the locate filter to All.



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

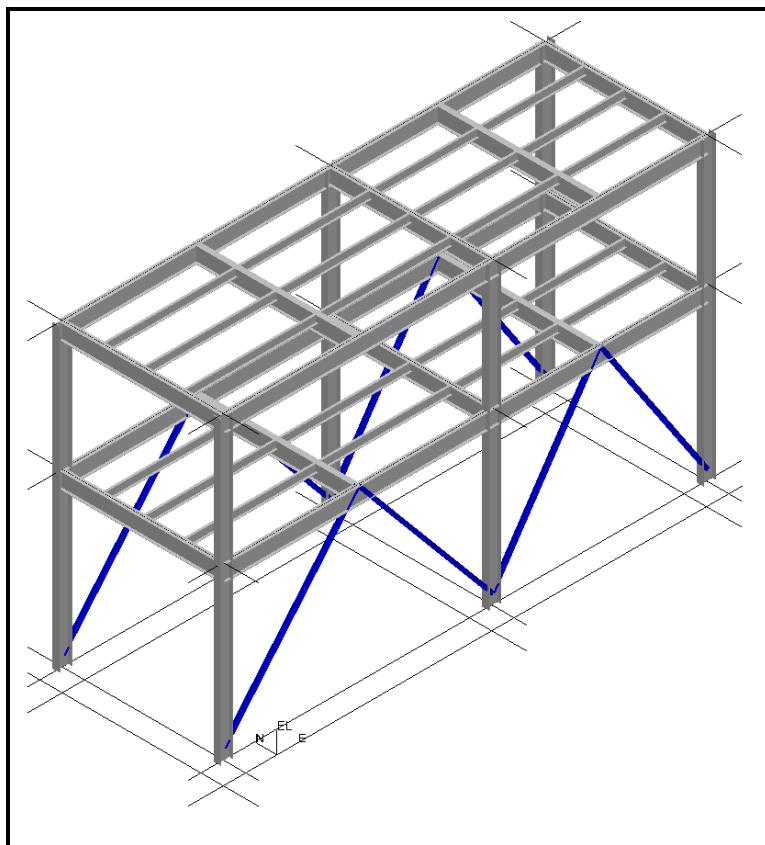


13. Go to the Main Menu and select Edit -> Properties



14. Keyin -1500mm for Position. Hit “OK” button to commit the transaction.

15. Verify that the Chevron vertical braces and all intermediate beams moved appropriately.



Lab 5 - Assembly Connections

Objectives

After completing this lab, you will be able to:

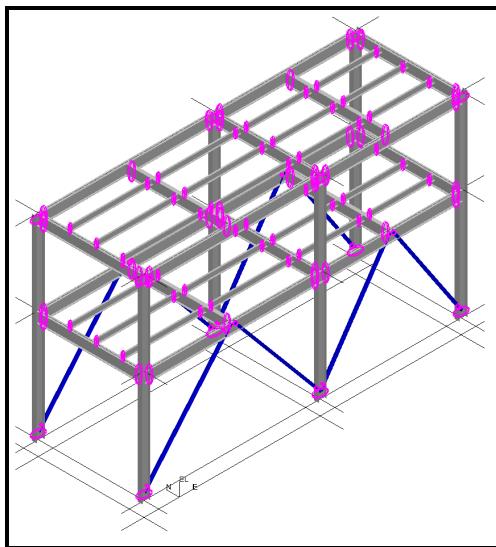
9. Understand the assembly connection entities and relationships
10. Use Place Assembly Connection command
11. Use Trim Member Command
12. Edit the assembly connection properties

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

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. Select all the frame connections in Structure U02 using the fence method. See figure below:



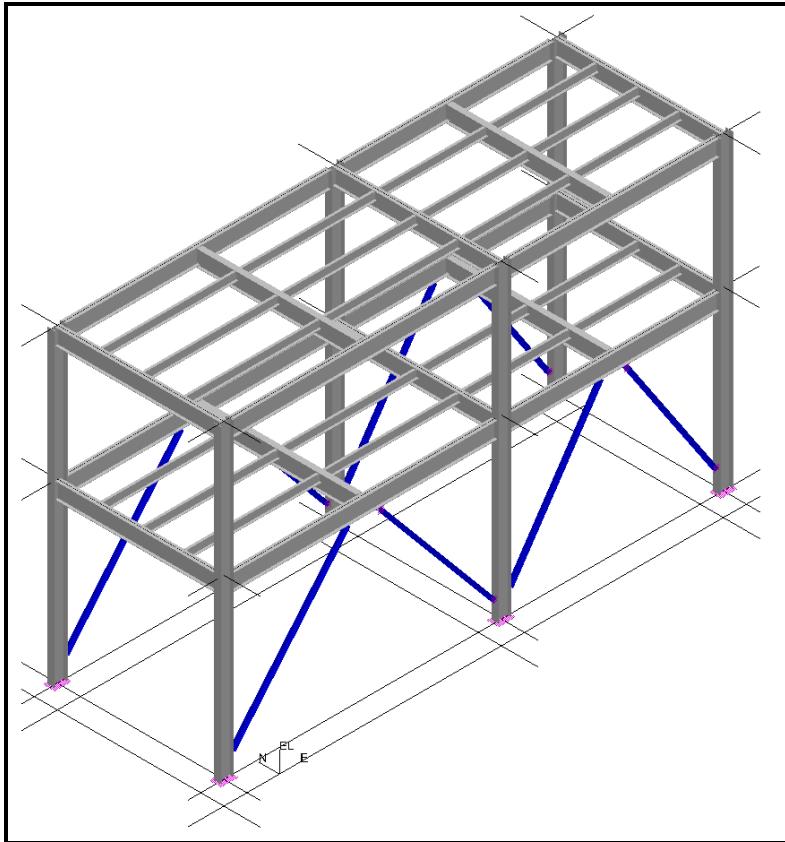
6. Select Place Assembly Connection Command.
7. Set the parameters as

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



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

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



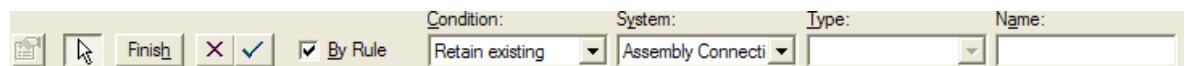
- Re-define your workspace to include the A2 -> U03 and CS -> U03 CS systems.

Repeat the above steps to create the assembly connections for structure U03.

- Select Place Assembly Connection Command.

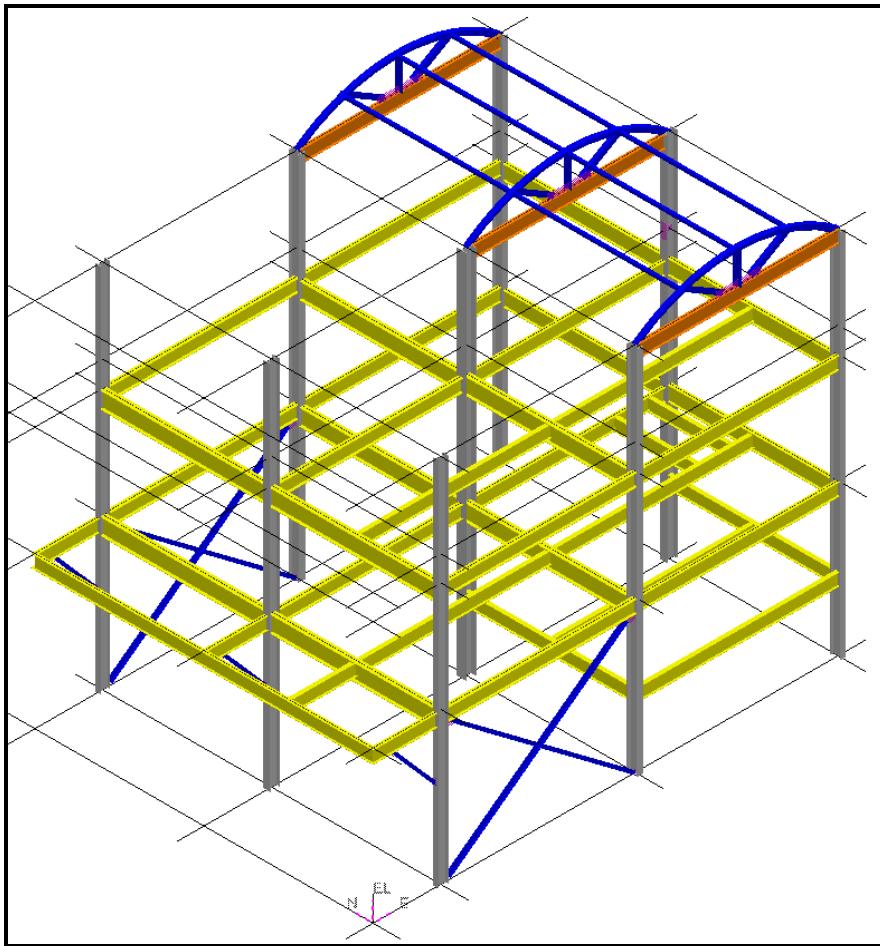
- Set the parameters as

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



- Hit "Finish" button to commit the transaction.

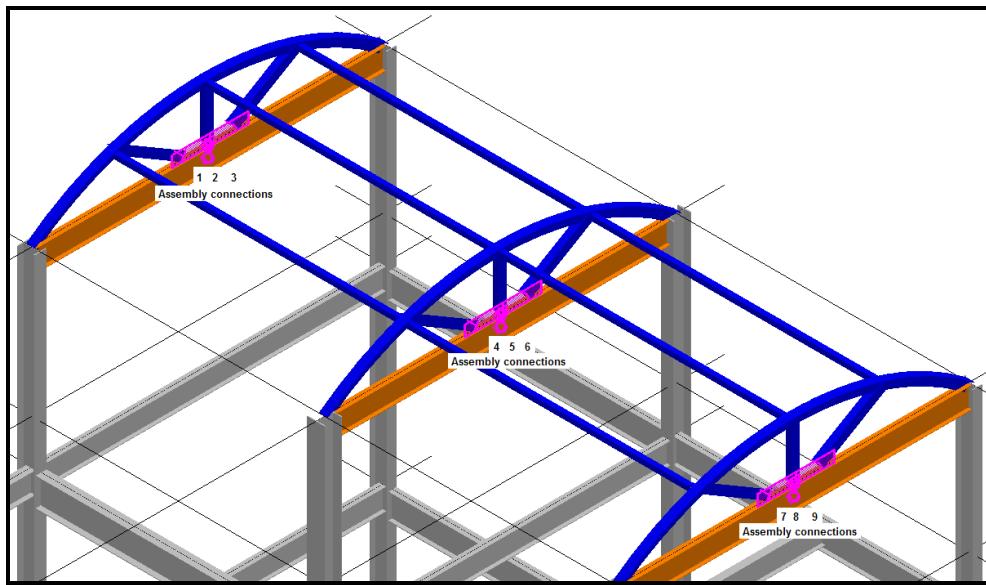
- Examine the model and verify that the planar cutbacks, base plates and gusset plates have been created.



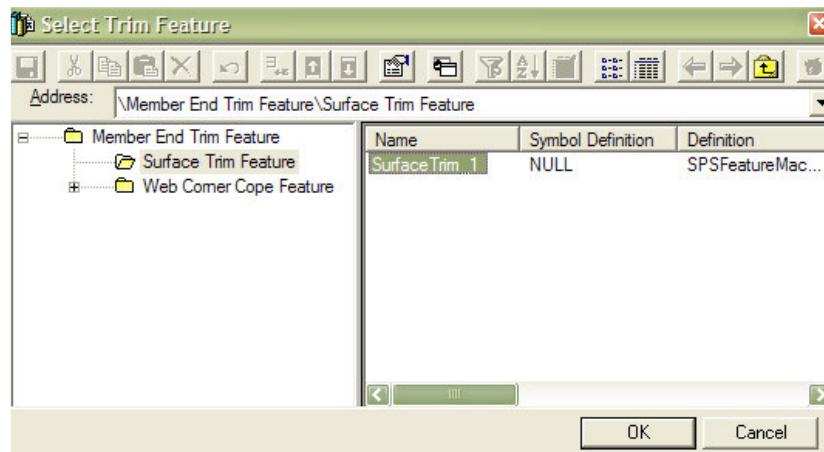
16. Use Zoom tool to zoom in around the rounded roof for the two-story building. 
17. Set the locate filter to Member Assembly Connections.

Member Assembly Connections   

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



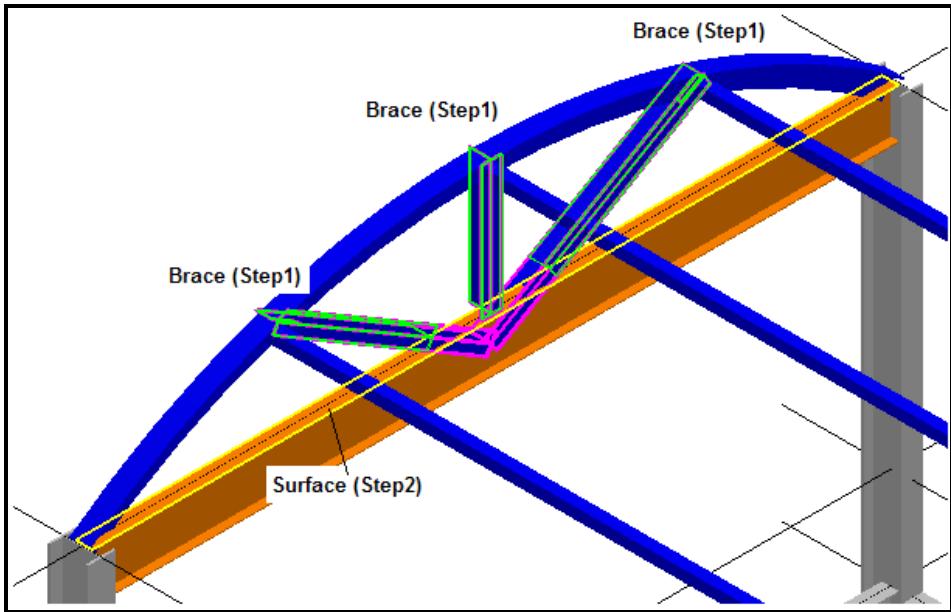
19. It should be nine objects in the select set. Click the Delete command to remove these objects.
20. Select Trim Member Command.
21. Expand the hierarchy and select the Surface Trim Feature folder.
22. Select Surface Trim 1.



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



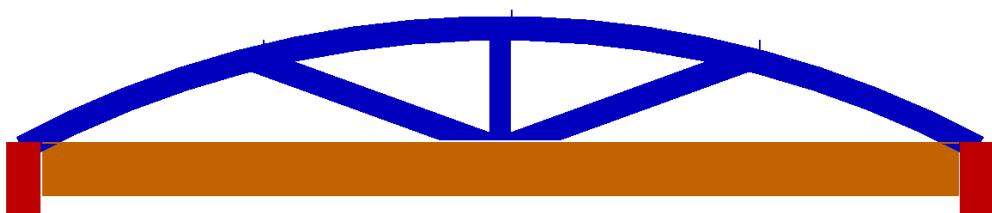
24. Select three vertical braces for the first SmartStep as shown below:



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 trims have been created.



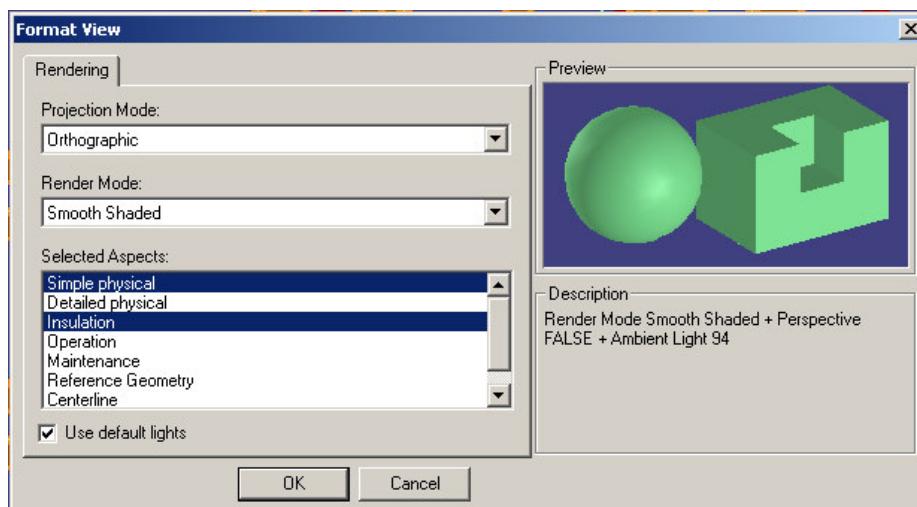
Lab 6 - Fireproofing

Objective

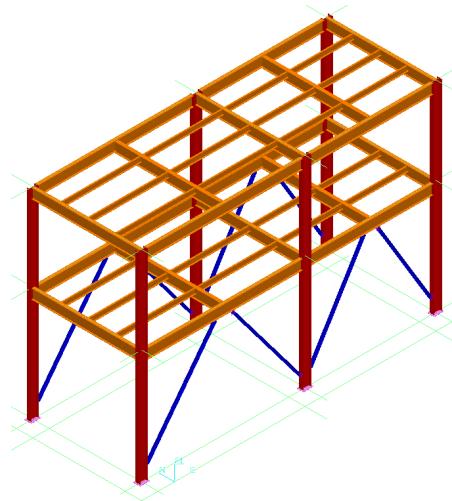
After completing this lab, you will be able to:

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

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. Go to *Format* → *View*. Select Insulation in the rendering tab.



5. Your view should resemble the following graphic. Now, hit OK.



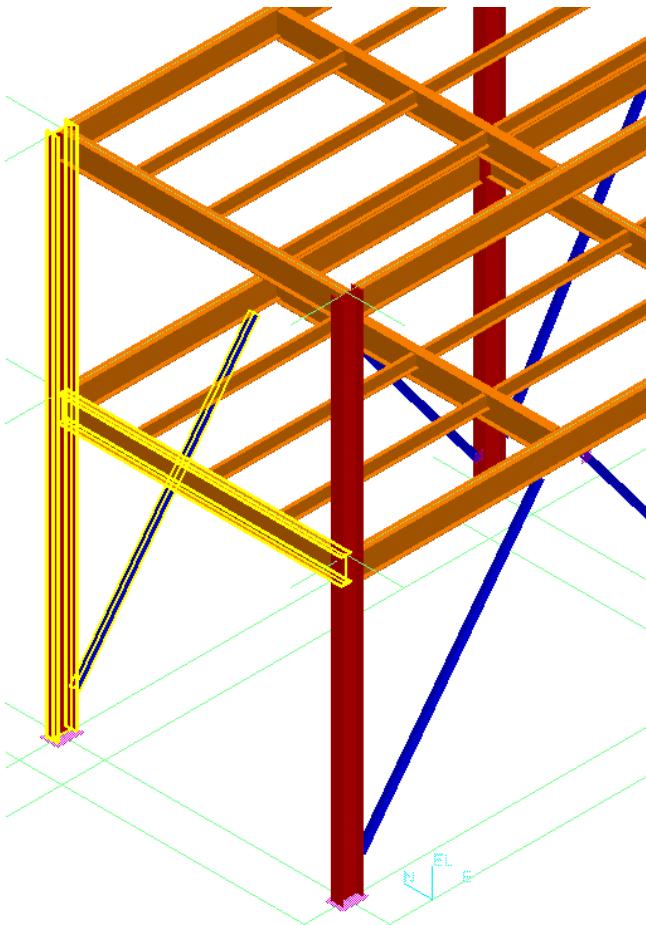
6. Go to File → Save.

Part A: Fireproofing on a pre-existing structure

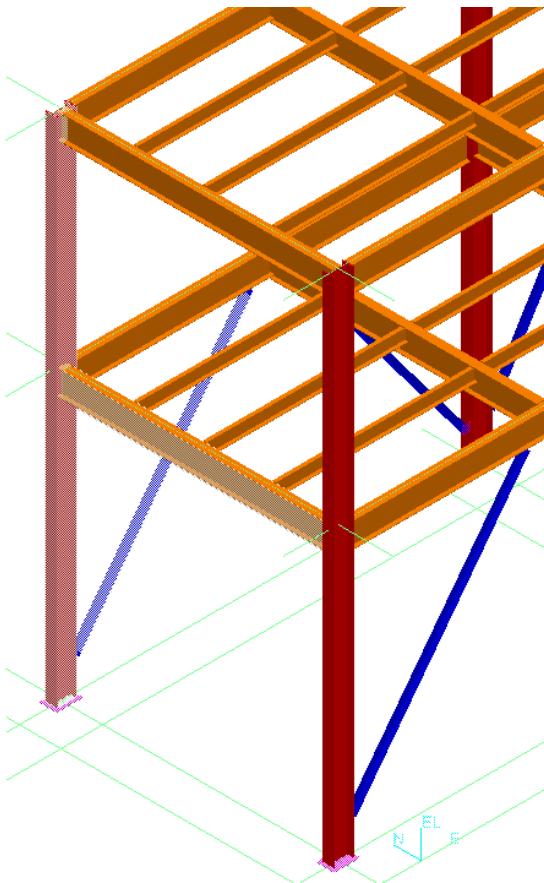
1. Select *Place Fireproofing* command. Insulation will be applied Using Place by rule. Make sure that the “*By Rule*” option is check marked.
2. Select “*Conc_1hr*” from the drop down list in the “*Insulation Spec*” menu.
3. Select “*Add*” from the drop down list in the “*condition*” menu.



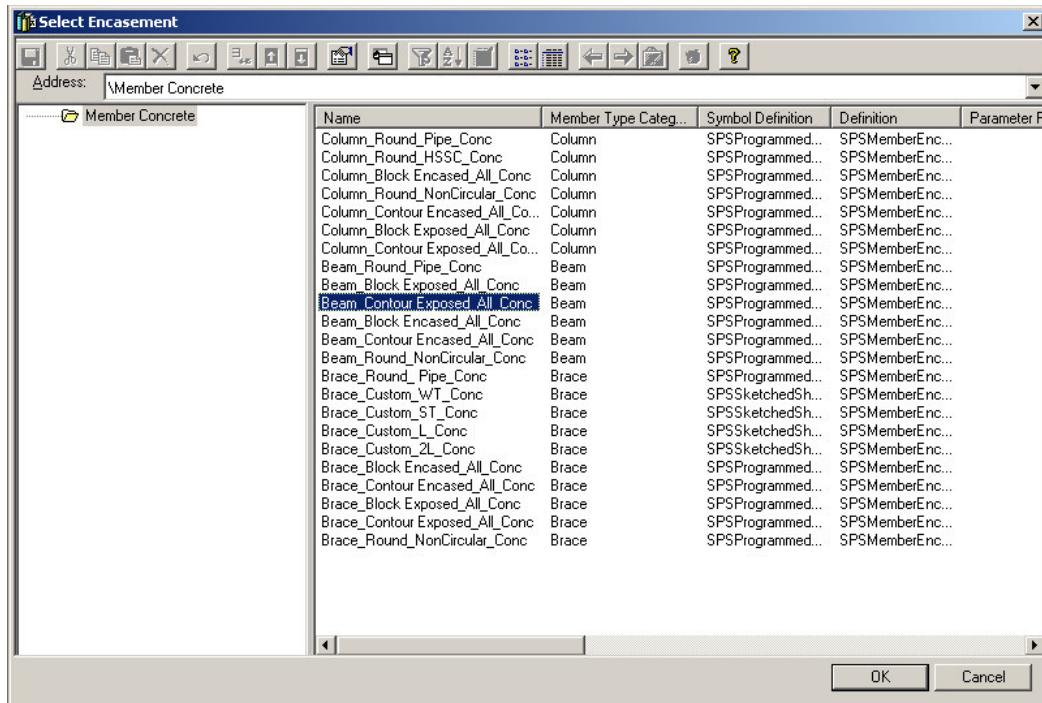
4. Select Column, Beam and Braze as shown in figure below,



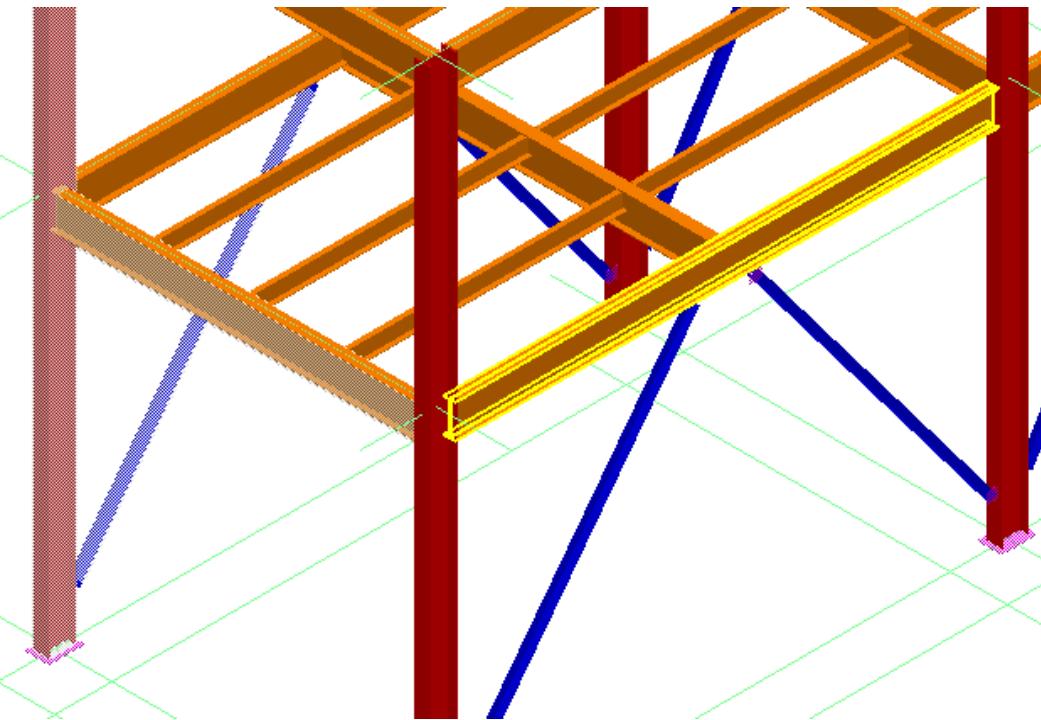
5. Hit the Accept button and then select *Finish*. Your view should resemble the following graphic.



6. Select Place Fireproofing command. Insulation will be applied with the “User Defined” option. Make sure that the “By Rule” option is unchecked.
7. Click on the drop down list in the “*Encasement*” menu Got to More... Select Encasement window will appear. Your view should resemble the following graphic.

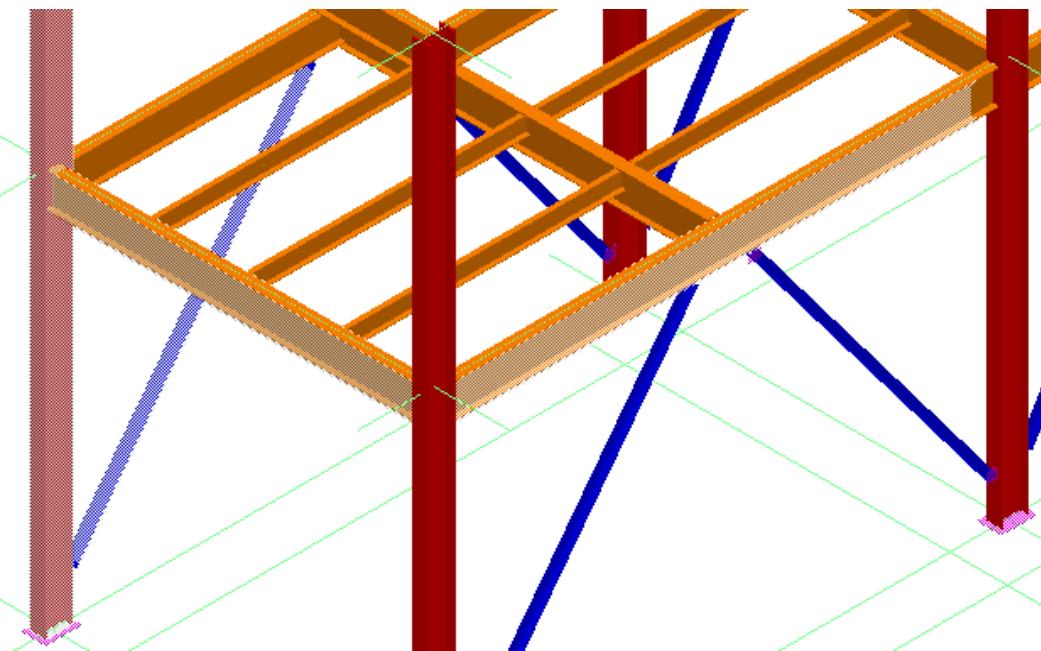


8. Select the “*Beam_Contour_Exposed_All_Conc*” Hit OK
9. Select “*Part Start*” in Reference 1.
10. Enter “0” as Distance 1.
11. Select “*Part End*” in Reference 2.
12. Enter “300mm” as Distance 2. This is the setback distance. This will not allow fireproofing to be applied on the beam at a distance of 300mm from the Part End.
13. Select the Beam as shown in the figure below,

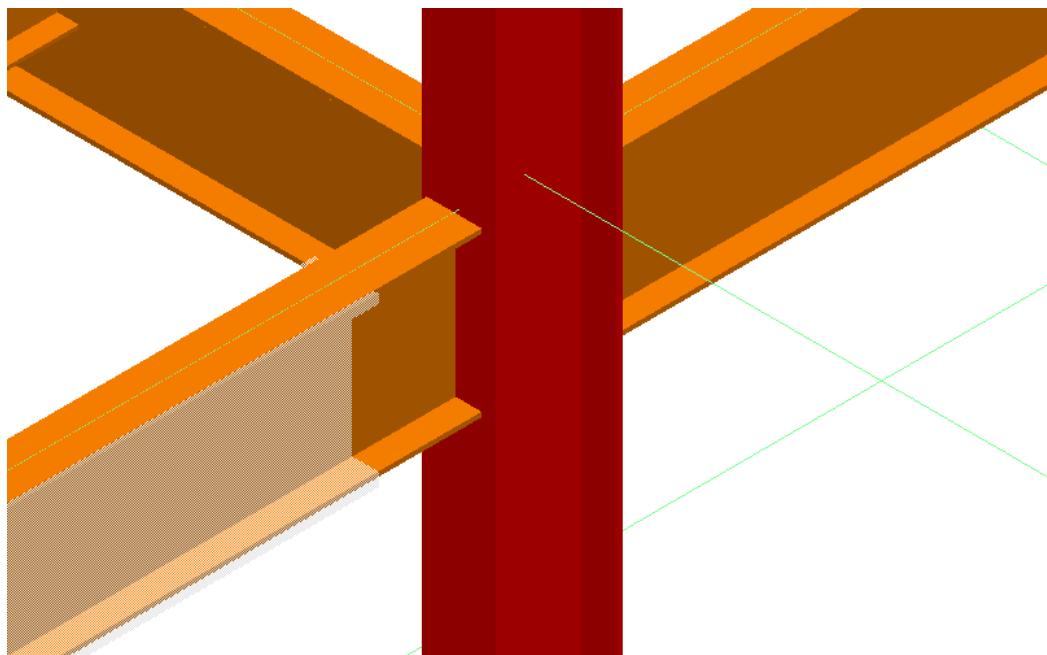


14. Hit the Accept button and then hit Finish

15. Hit the Cancel button. Your view should resemble the following graphic.

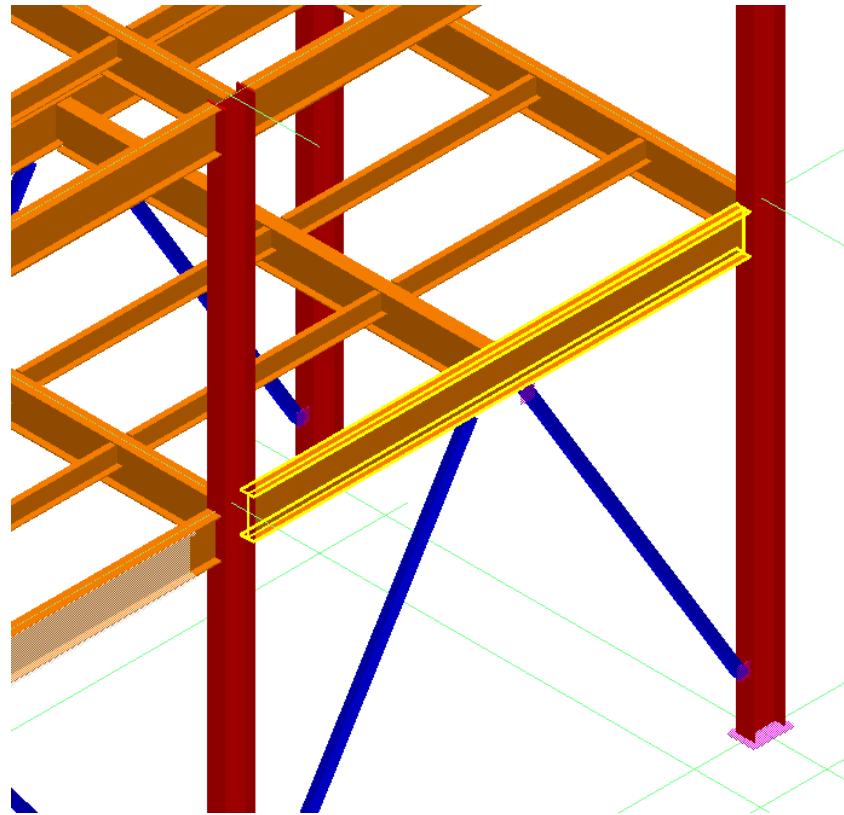


See image below for close up image. This image shows set back distance.

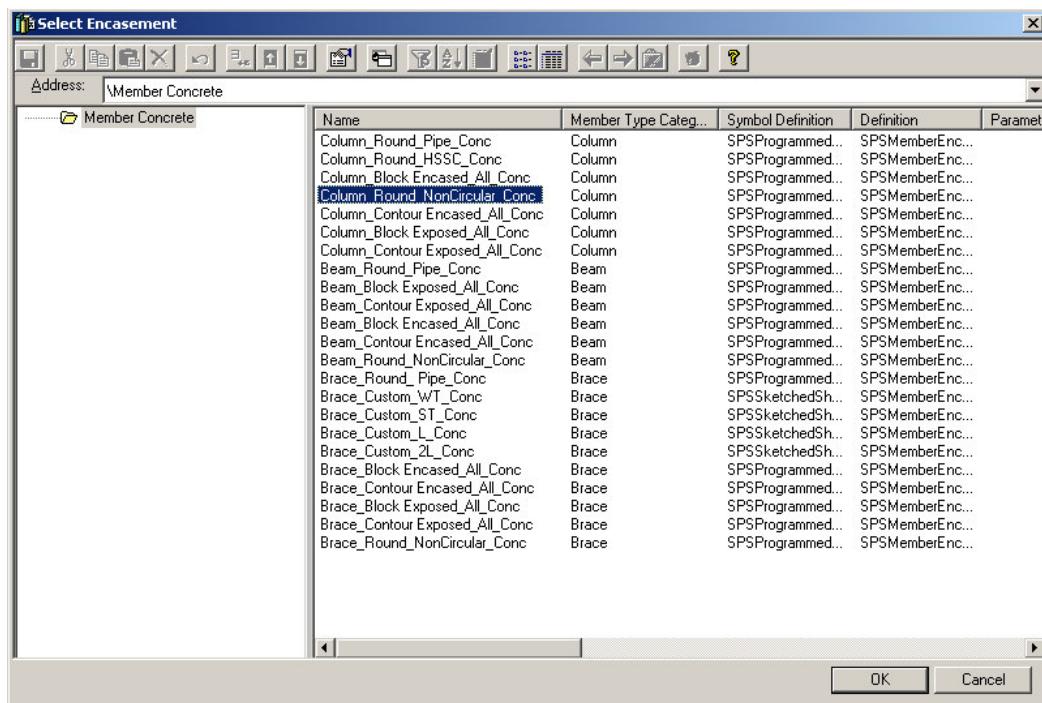


Part B: Correct encasement on a pre-existing beam using “By Rule”

1. “*Place Fireproofing*” command will always apply column encasement 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 unchecked.
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.



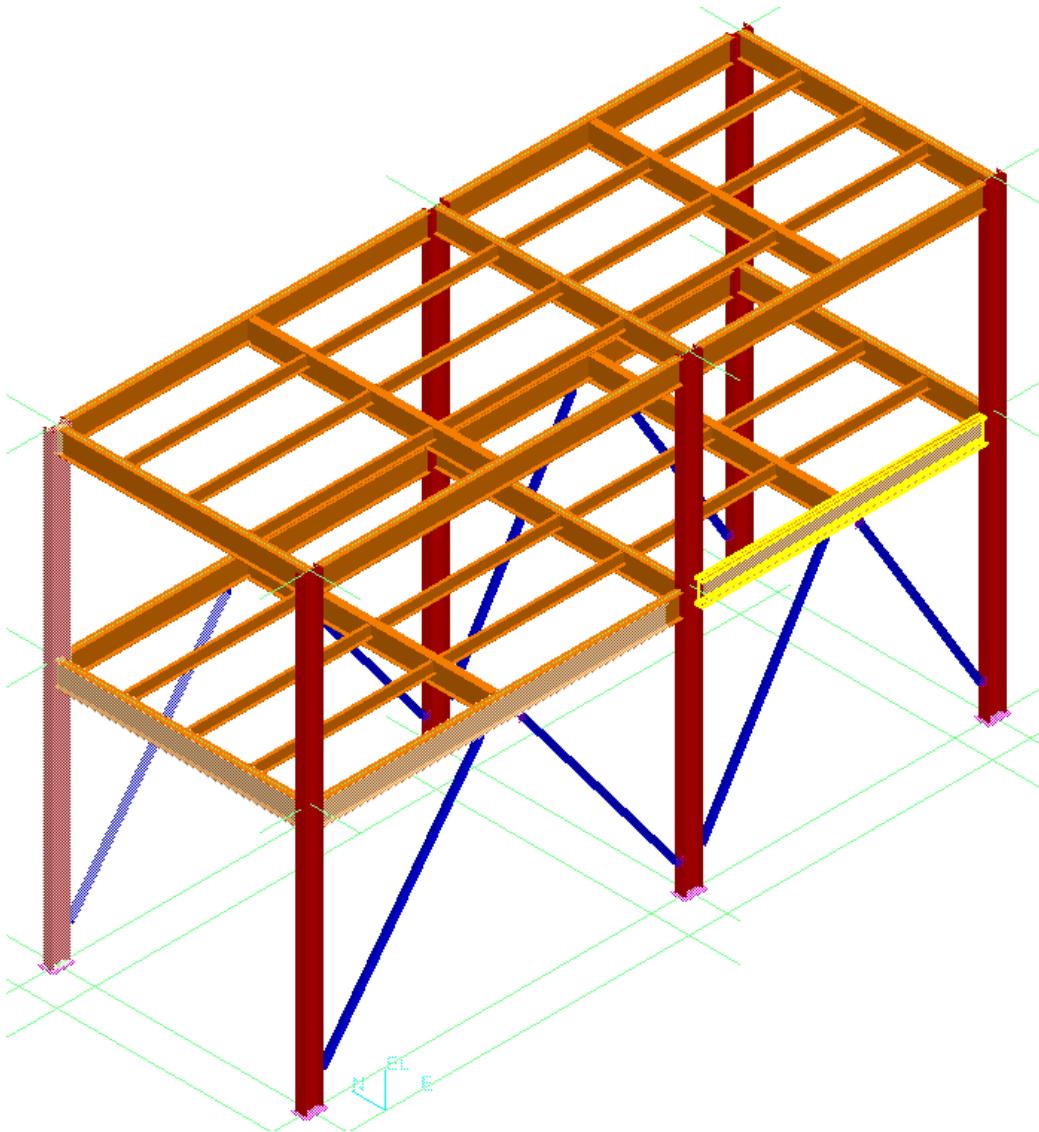
5. Select the “*Column_Round_NonCircular_Conc*” Hit OK.
6. Select “*Part Start*” in Reference 1.
7. Enter “0” as Distance 1.
8. Select “*Part End*” in Reference 2.
9. Enter “0” as Distance 2.
10. Hit Accept. Warning message will be displayed. Your view should resemble the following graphic.



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

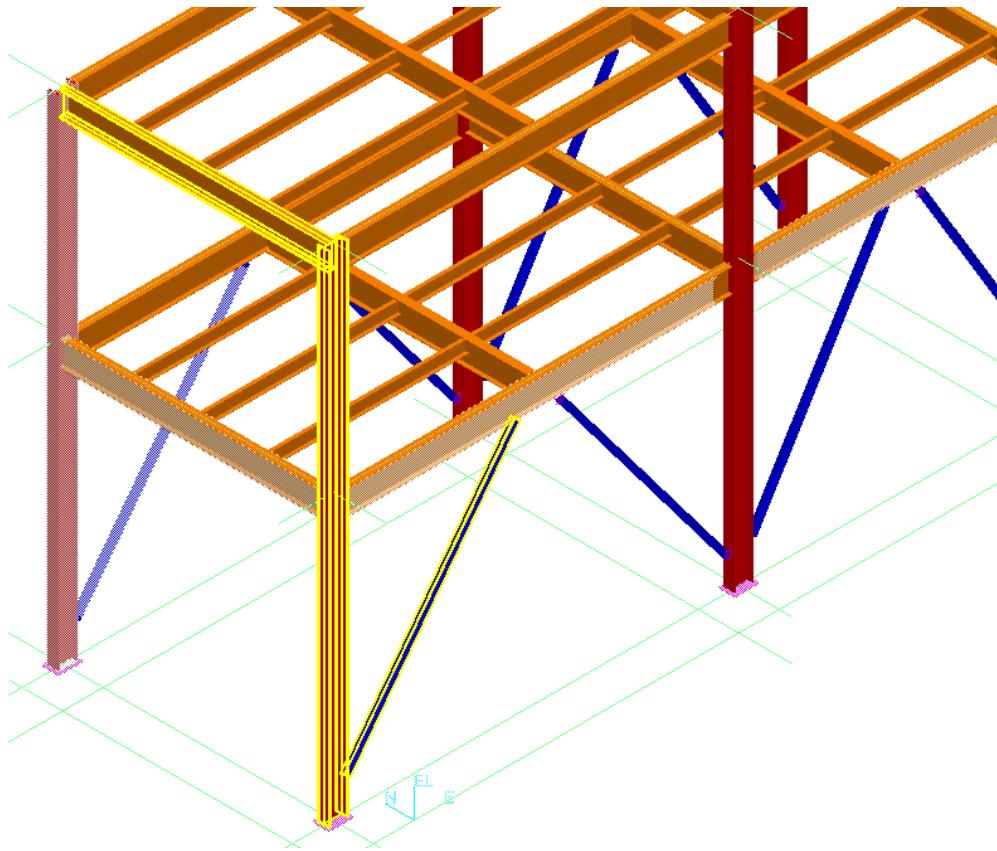


14. Hit Finish.
15. Hit Cancel. Your view should resemble the following graphic.

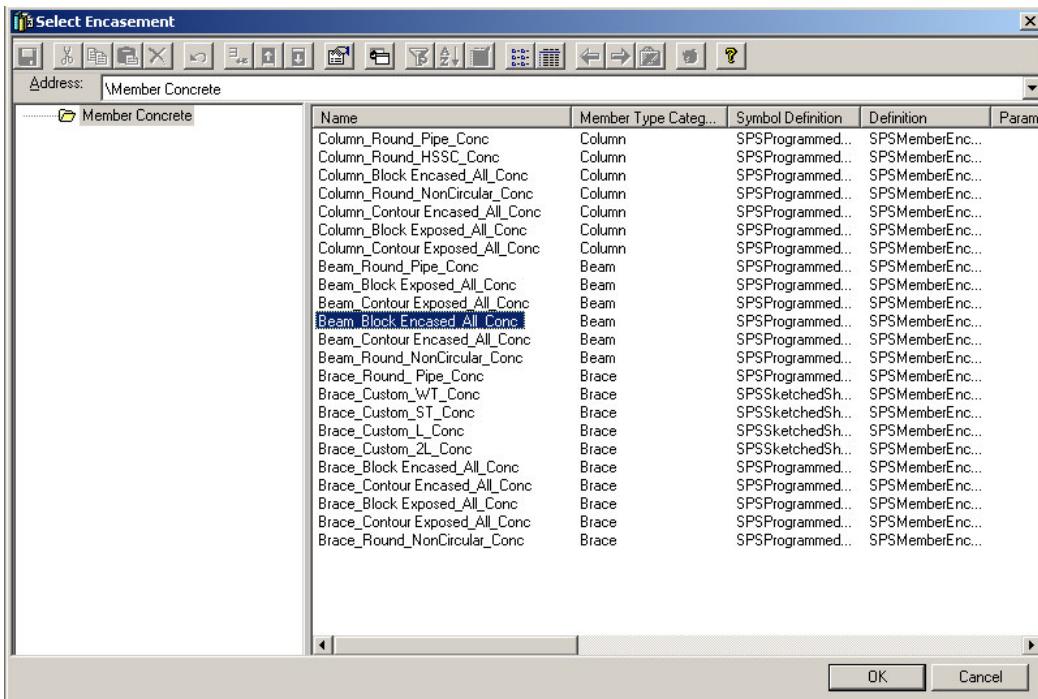


Part C: user defined correct encasement on a pre-existing member

1. “Place Fireproofing” command will always apply column 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.

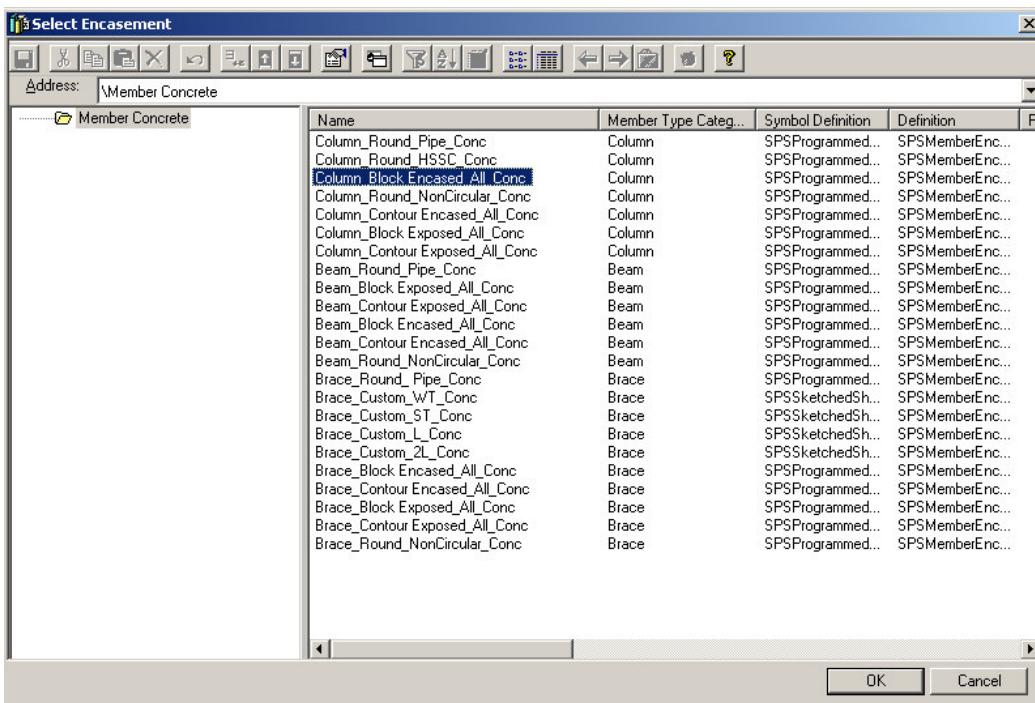
6. Hit Finish. Warning is shown.



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

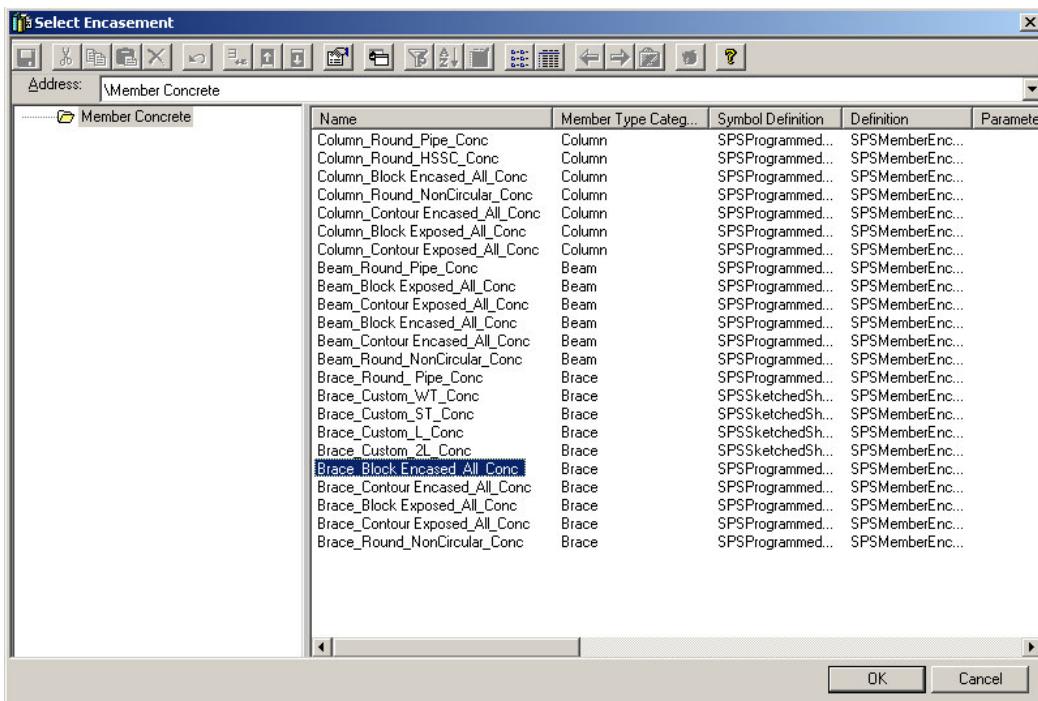
10. Hit Finish. Warning is shown.



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.

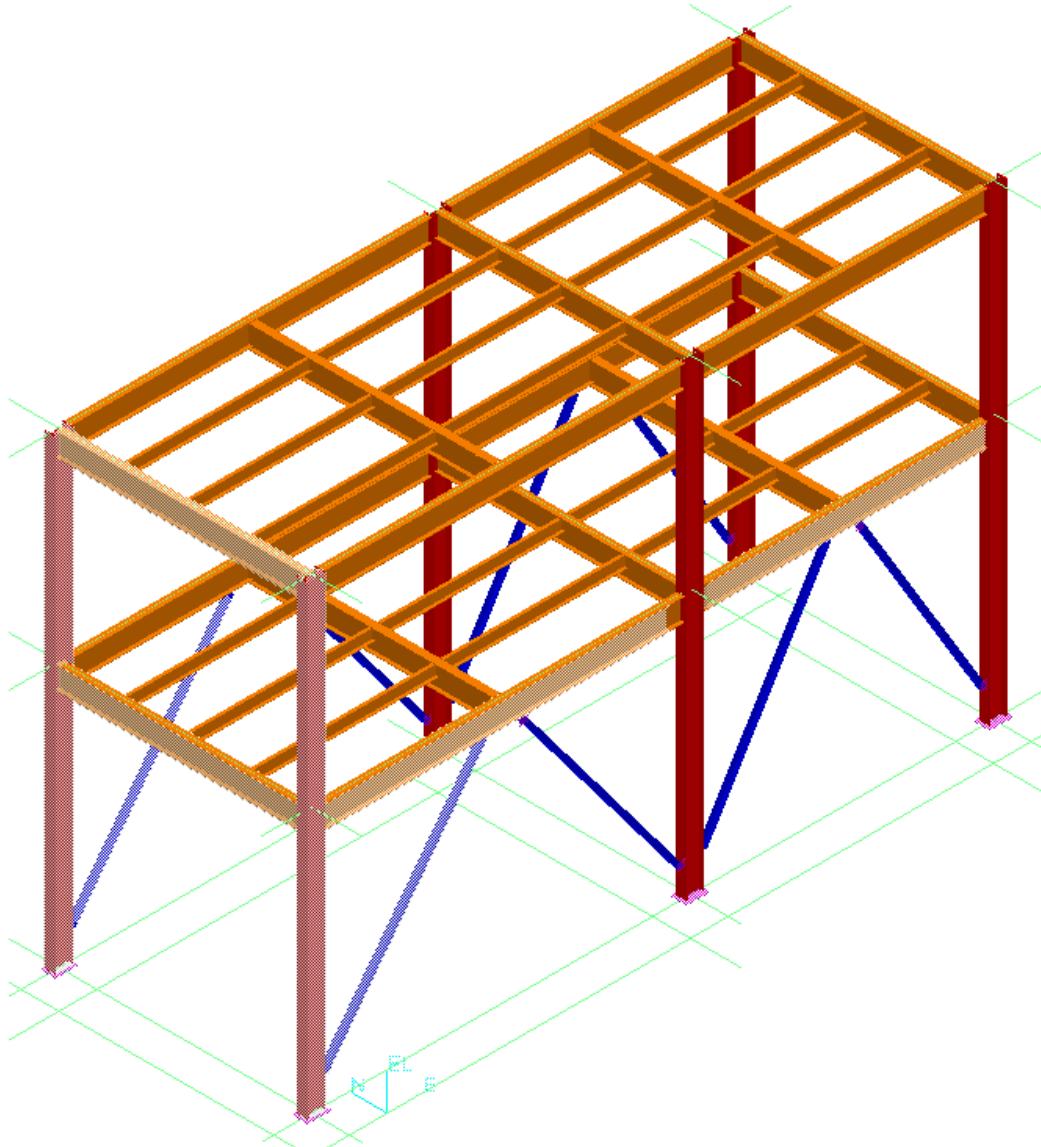
14. Hit Finish. Warning is shown.



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.

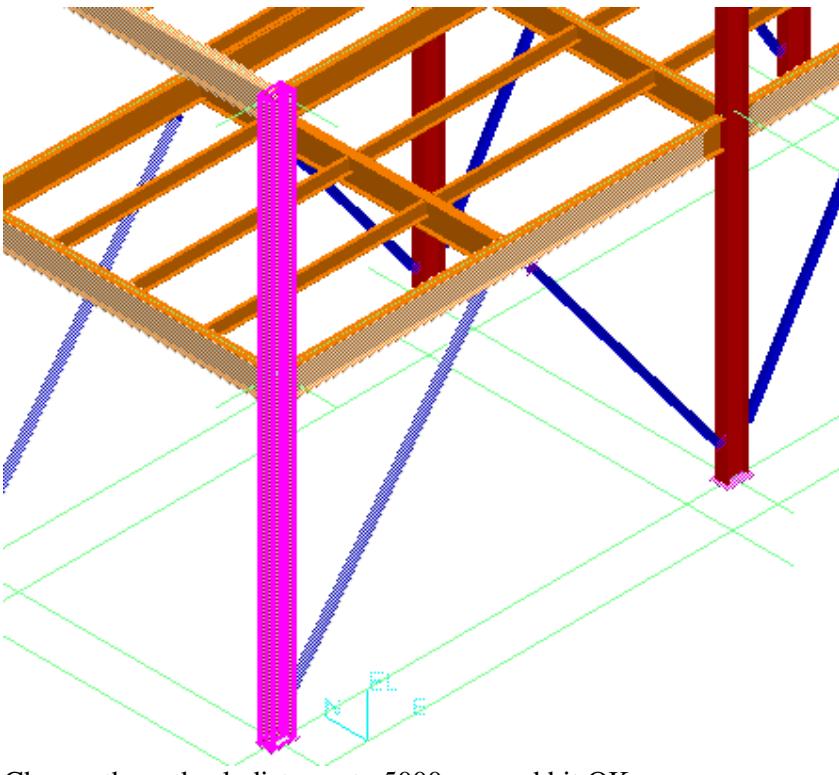


Part D: Add Fireproofing to a column that already has fireproofing.

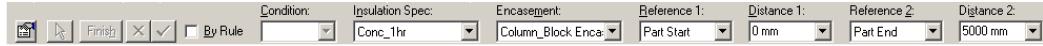
1. Set the Locate Filter to Insulation



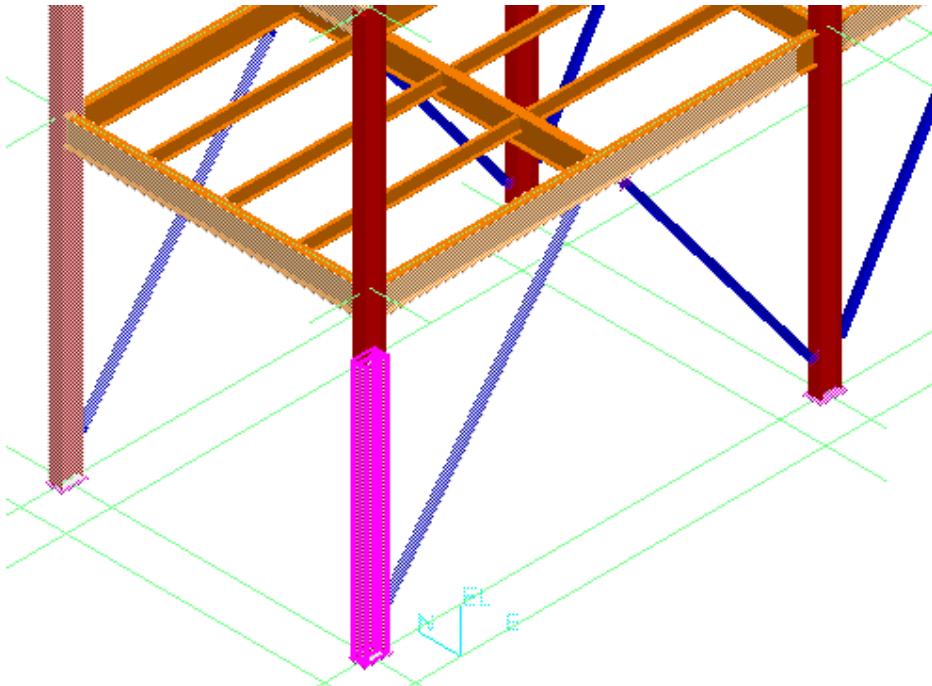
2. Select the insulation as shown in the image.



3. Change the setback distance to 5000mm and hit OK.

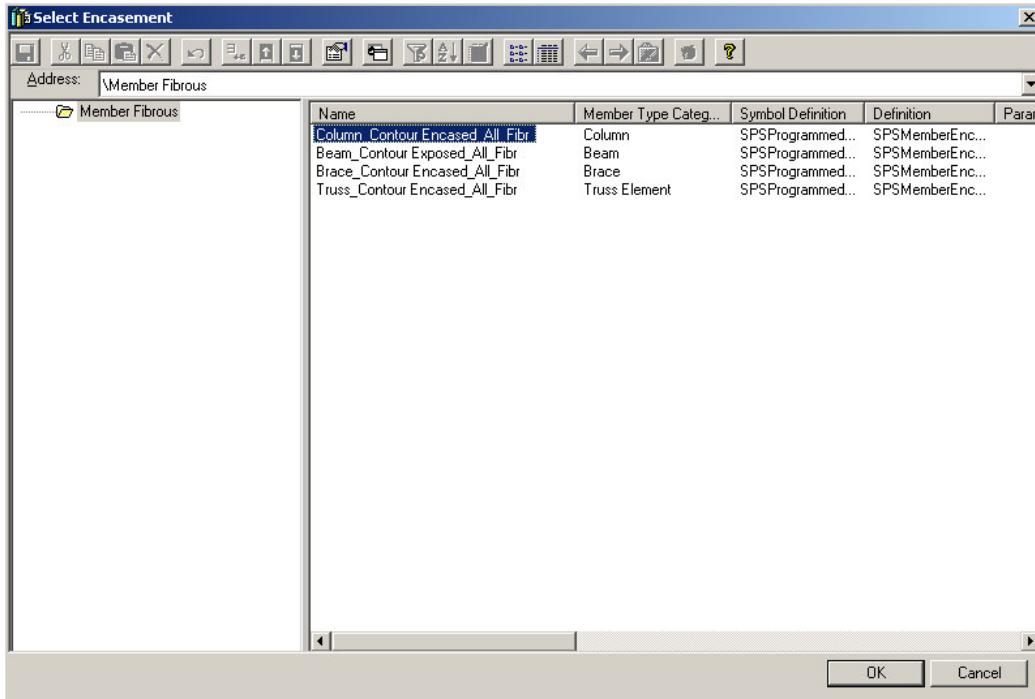


4. Your view should now resemble the following graphic.

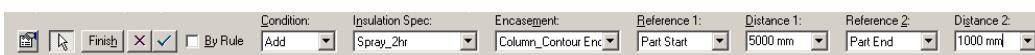


5. Select the *Place Fireproofing* command.

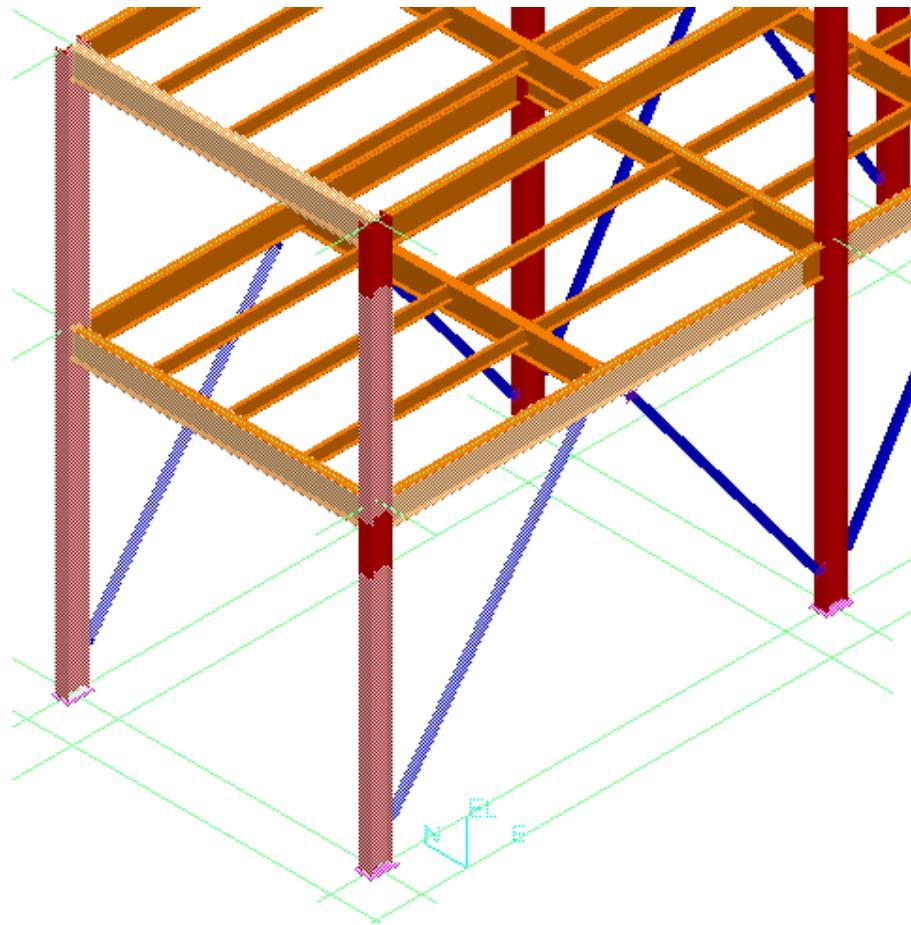
6. Change the Insulation Spec to *Spray_2hr*.
7. Go to More in the Encasement menu. Select *Column_Contour_Encasement_All_Fibr*. Your view should resemble the following graphic.



8. Hit OK.
9. Select “*Part Start*” in Reference 1.
10. Enter 5000mm as Distance 1.
11. Select “*Part End*” in Reference 2.
12. Enter 1000mm as Distance 2.

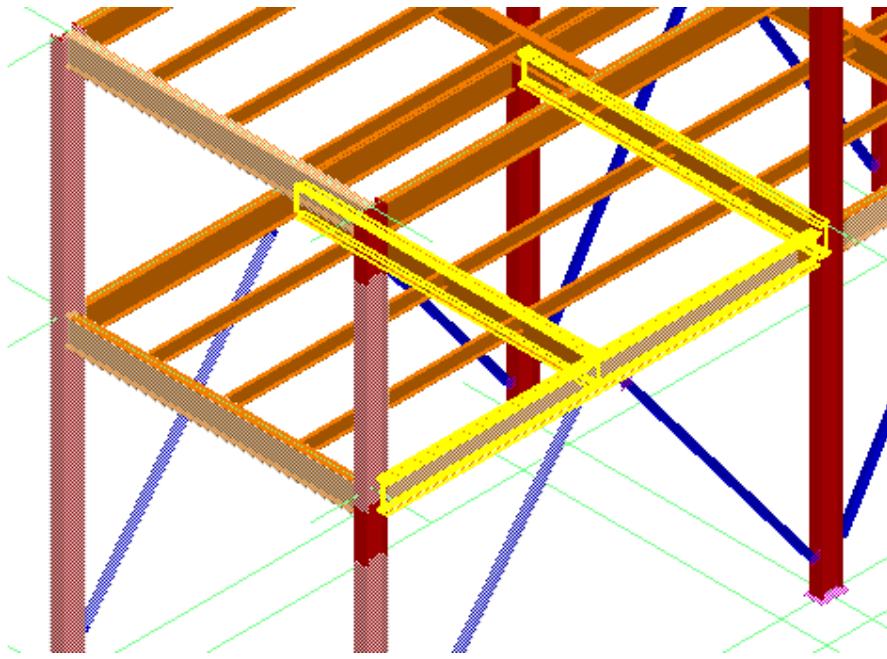


13. Hit Accept button and hit Finish.
14. Hit cancel and your view should resemble the following graphic.

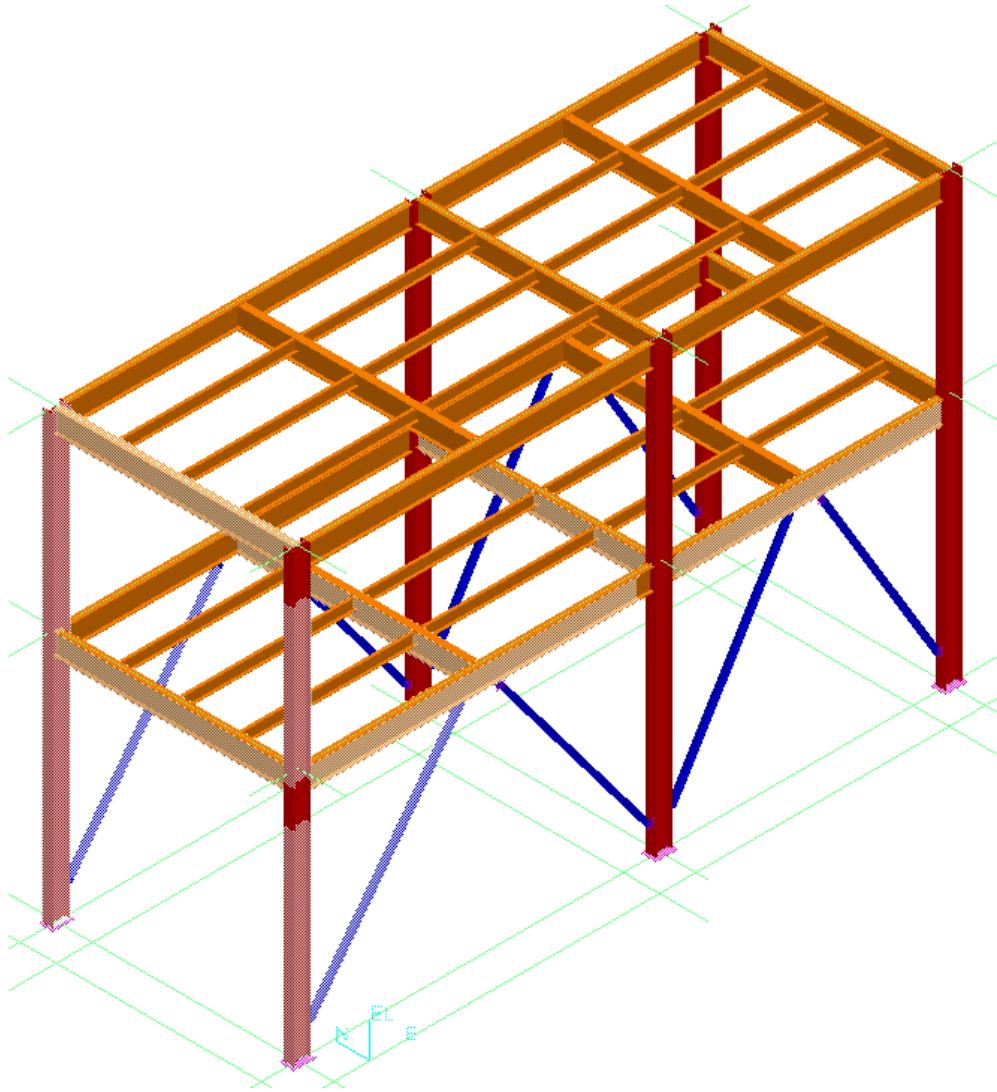


Part E: Retain fireproofing.

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

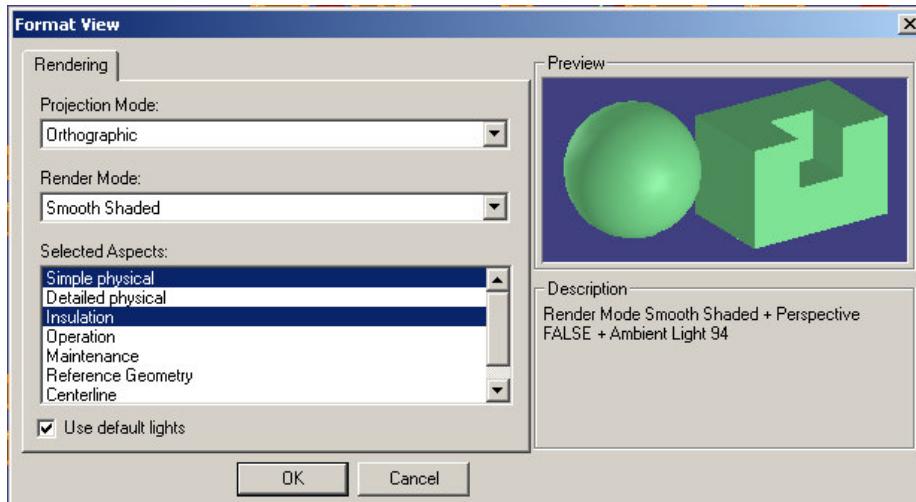


3. Change the *Condition* to “Retain” in the toolbar.
4. Select the *Insulation Spec* as Conc_1hr.
5. Select “By Rule”
6. Hit Finish. Warning will be shown. Hit OK. Hit Cancel. Fireproofing is retained on the existing member and fireproofing is applied on the other two beams. Your view should resemble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

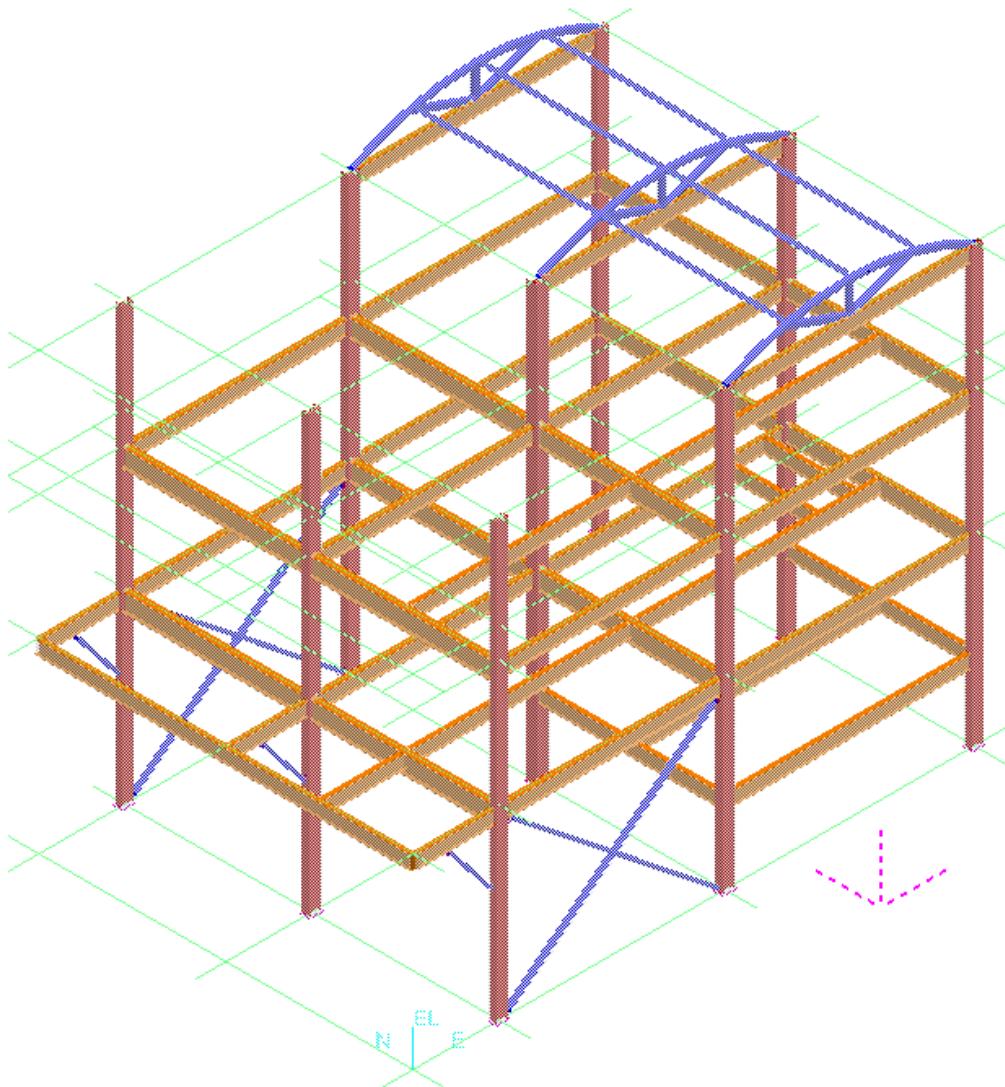


Part F: Place Fireproofing for the entire structure.

1. 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. Go to *Format ->View*. Select Insulation in the rendering tab.



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

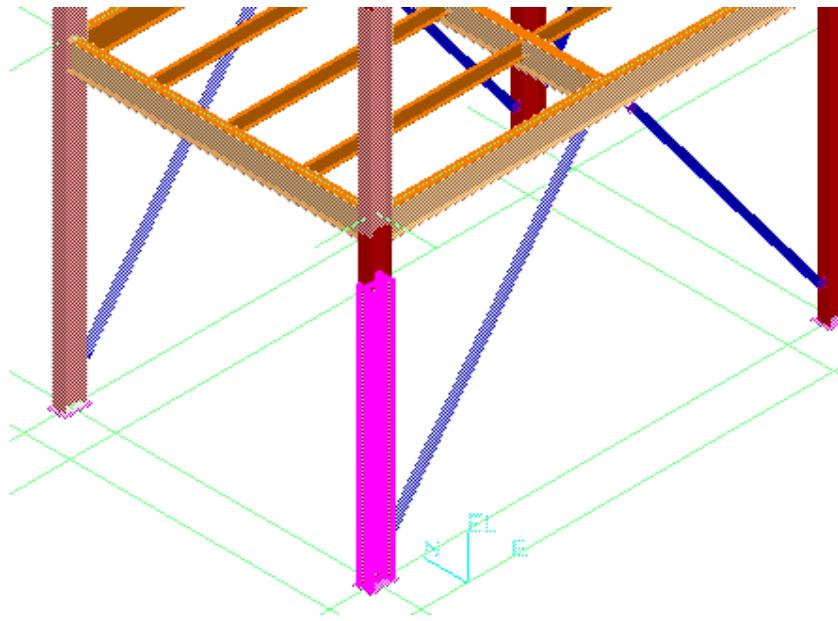


Part G: Modification

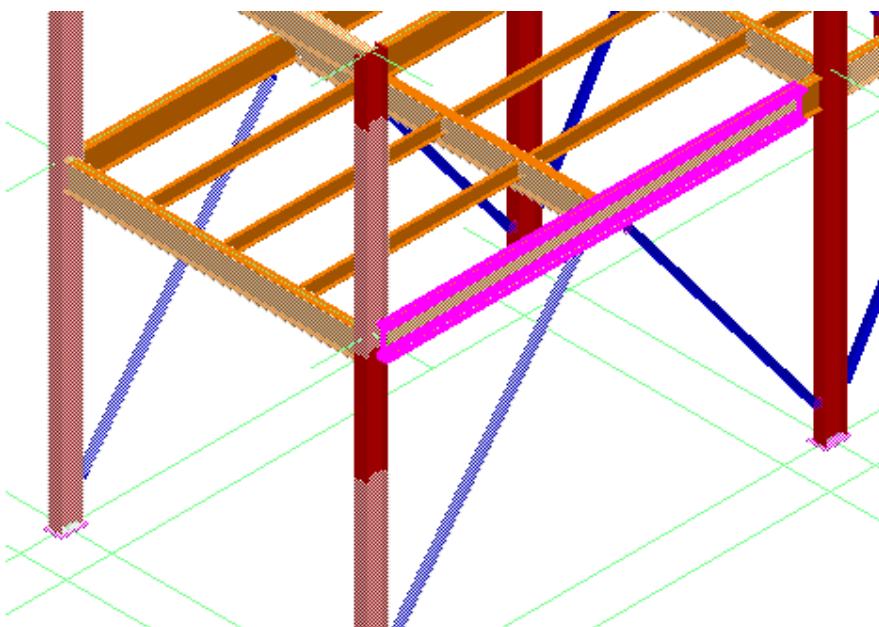
1. Define your workspace to include the A2 -> U02 and CS -> U02 CS systems.



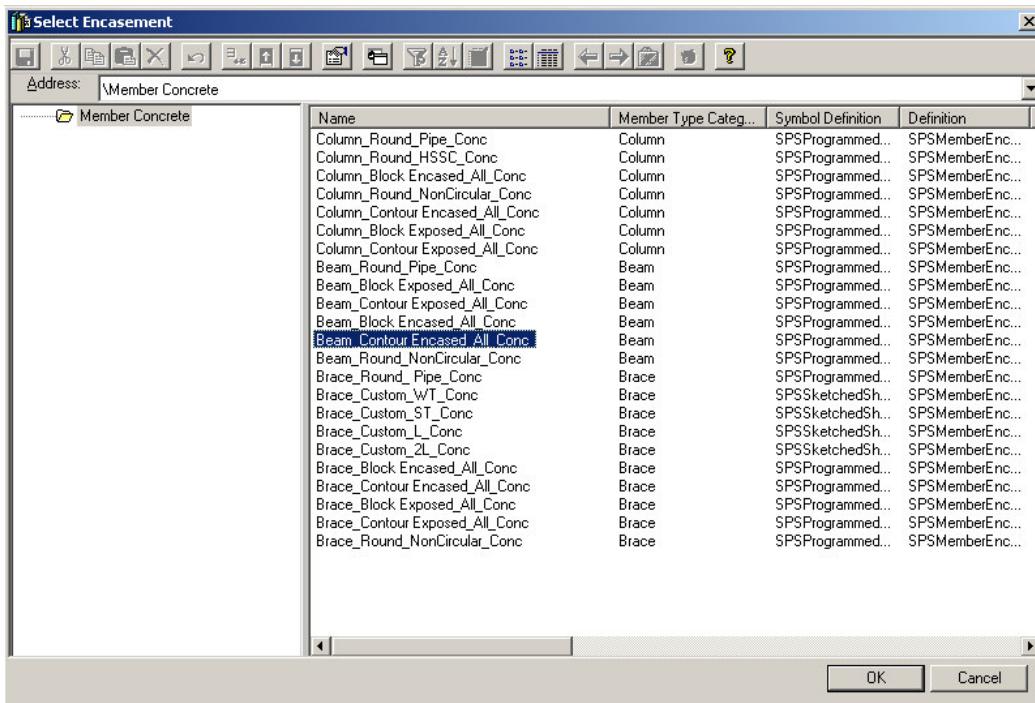
2. Set the Locate Filter to *Insulation*
3. Select the insulation as shown in the image



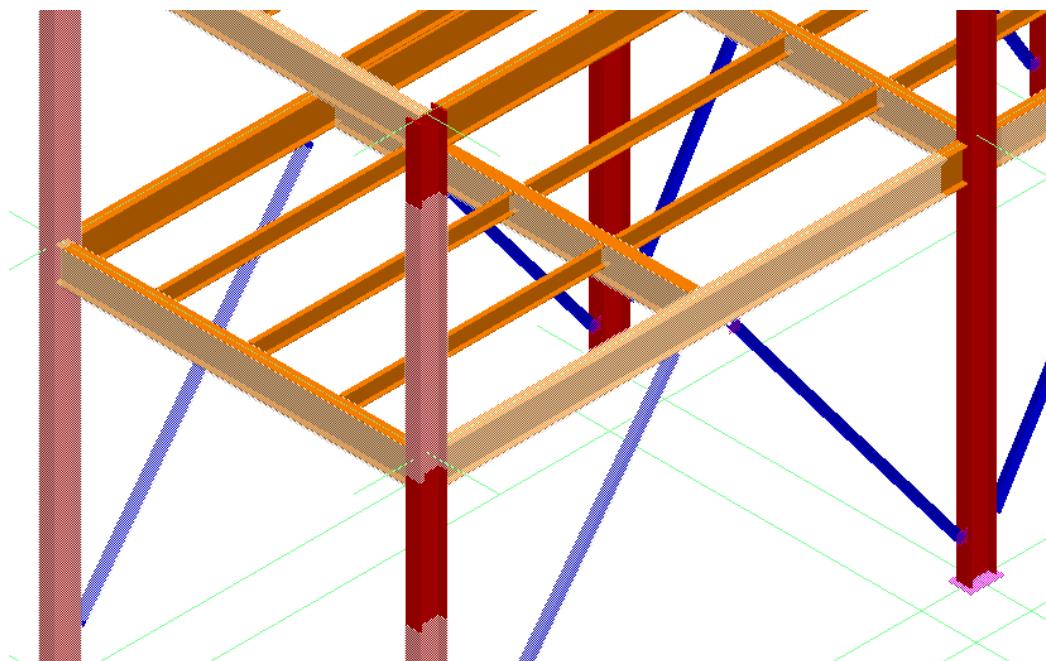
4. Change the *Part Start* distance from 0mm to 600mm.
5. Change the *Part End* distance from 5000mm to 6000mm.
6. Select the insulation for the beam as shown in the image below.



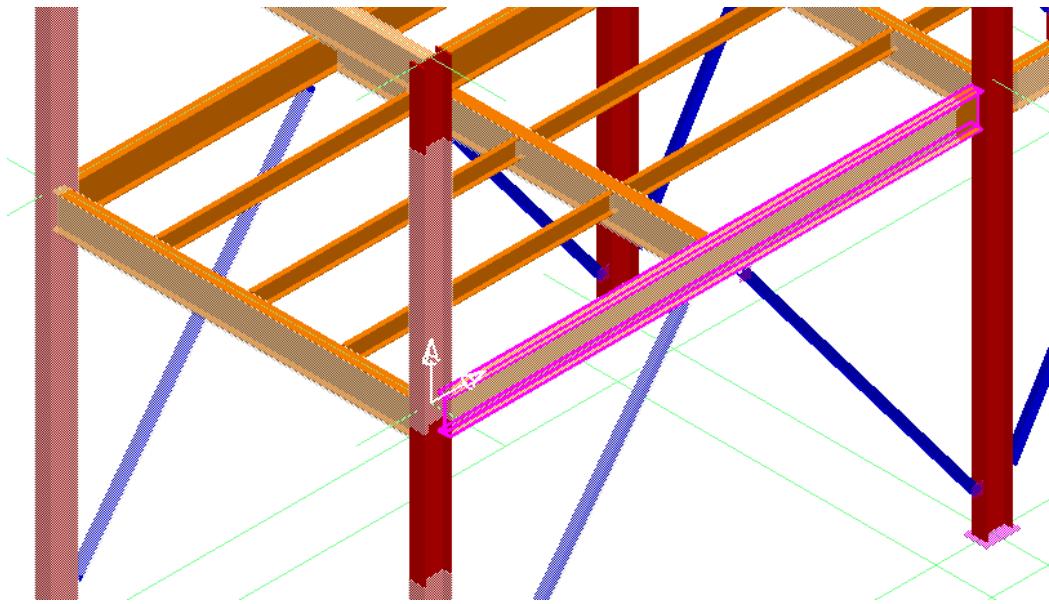
7. Go to More from the Encasement pull down list. Your view should resemble the following graphic.



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.

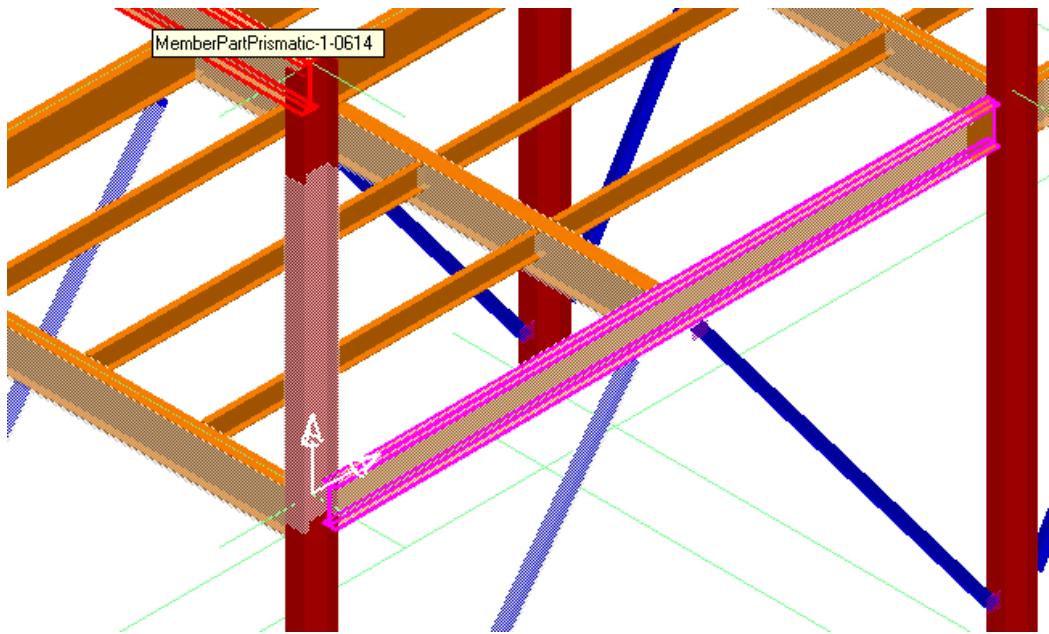


10. Set the Select Filter to *Member Parts*.
11. Select the beam as shown in the image below,



12. Change the section size to W16X26.

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 related to fireproofing.

15. Go to Edit -> Undo Modify Member or hit Ctrl + Z.

Lab 7 - Slabs

Objective

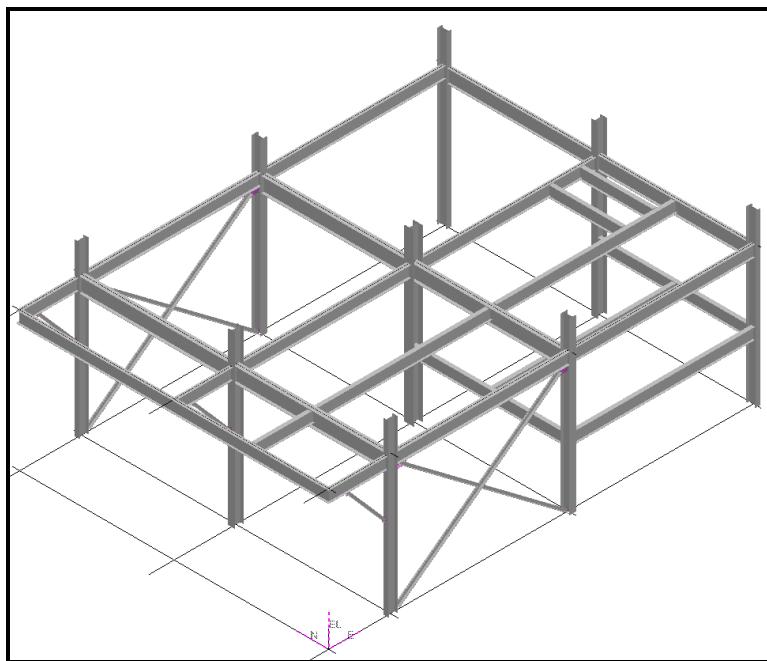
After completing this lab, you will be able to:

- Understand the slab entities and relationships
- Place Slabs using different options

You will add slab floors and a roof deck to the steel-frame done in lab 2.

Part I: Place Slab on U03 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 6.0m 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.

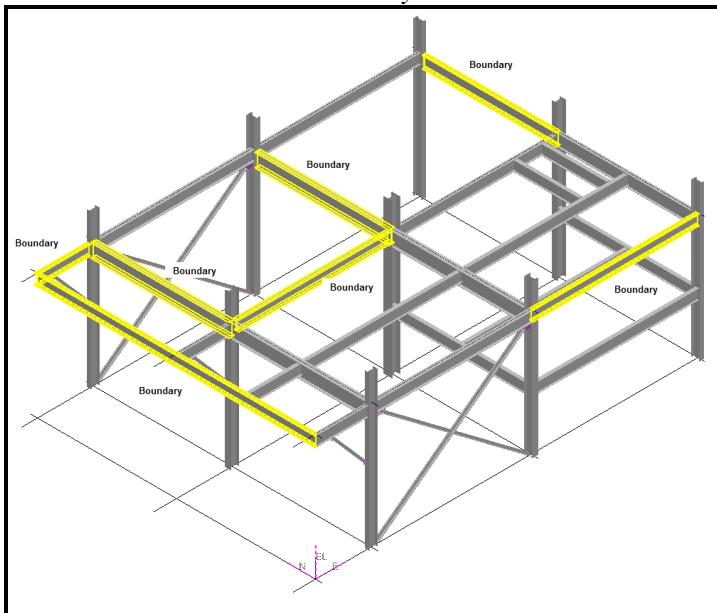
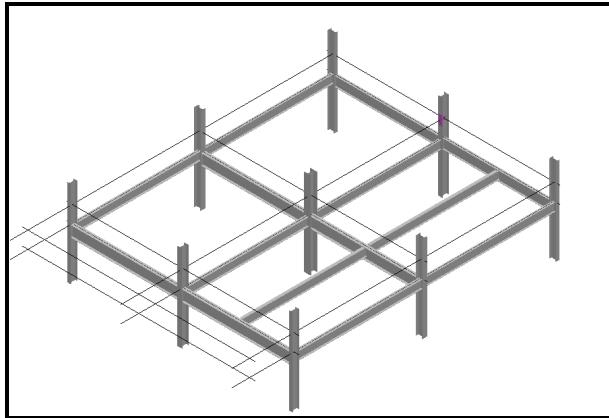


Figure 1 – ISO View of the First Floor Frame

Part II: Place Slab on U03 Second Floor:

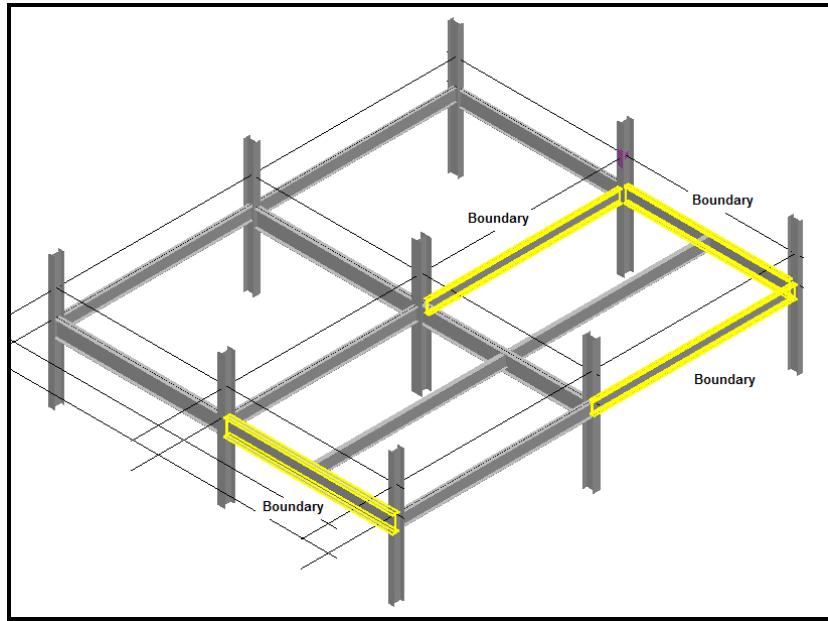
1. Select View-> Clear Clipping to remove the clipping volume.
2. Use Clip by Object command to isolate the beams and the columns in the second floor as shown below:



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

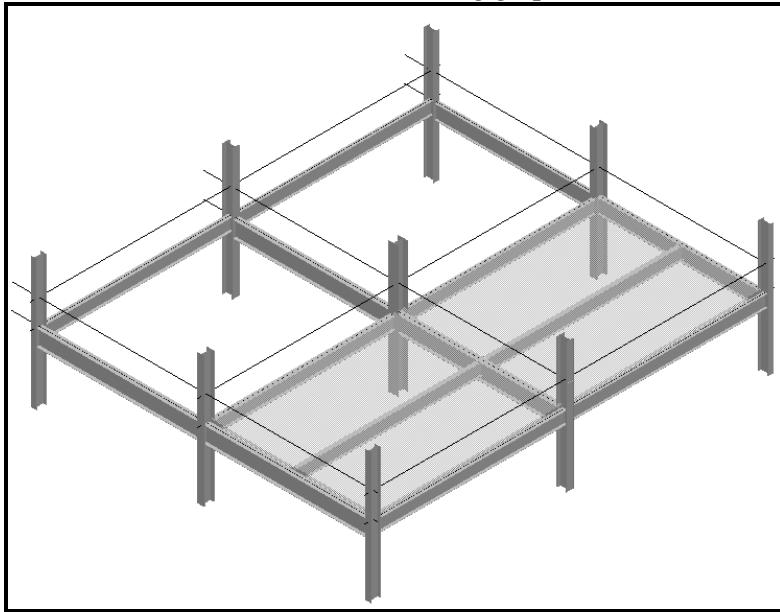
Plane Method:	Coincident
System:	A2-> U03 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4" _Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Boundary Offset Reference:	Outer port-face of a bounding member

5. Select Elevation Plane at 10.0m for the support plane. Click “Accept” button.
6. Select the boundaries as shown in Figure below.



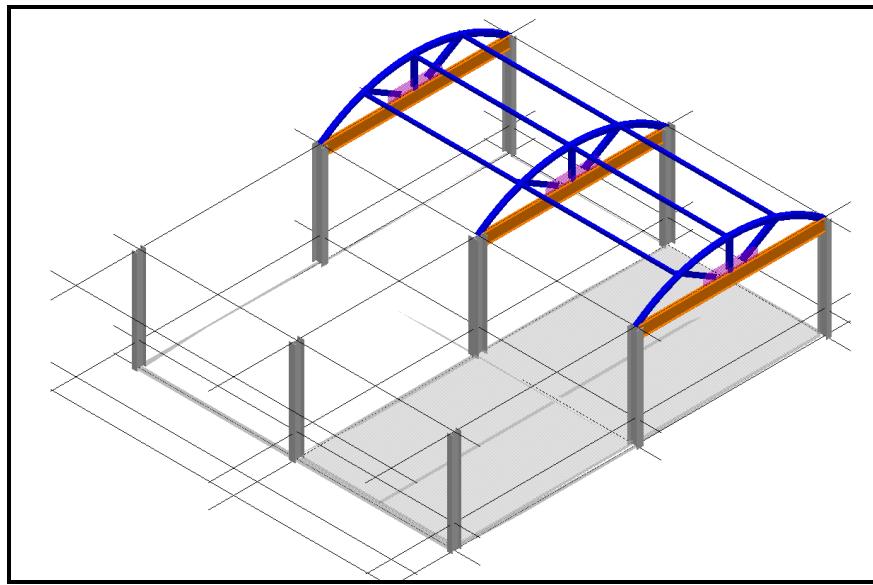
7. Skip Define the Boundaries offsets step.
8. Hit “Finish” button to commit the transaction.

Your View should now resemble the following graphic:



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 the place slab command.
4. Select Plane Method: Offset from a Plane.
5. Select the Elevation Plane at 14m for the support plane.
6. Key in -300mm 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

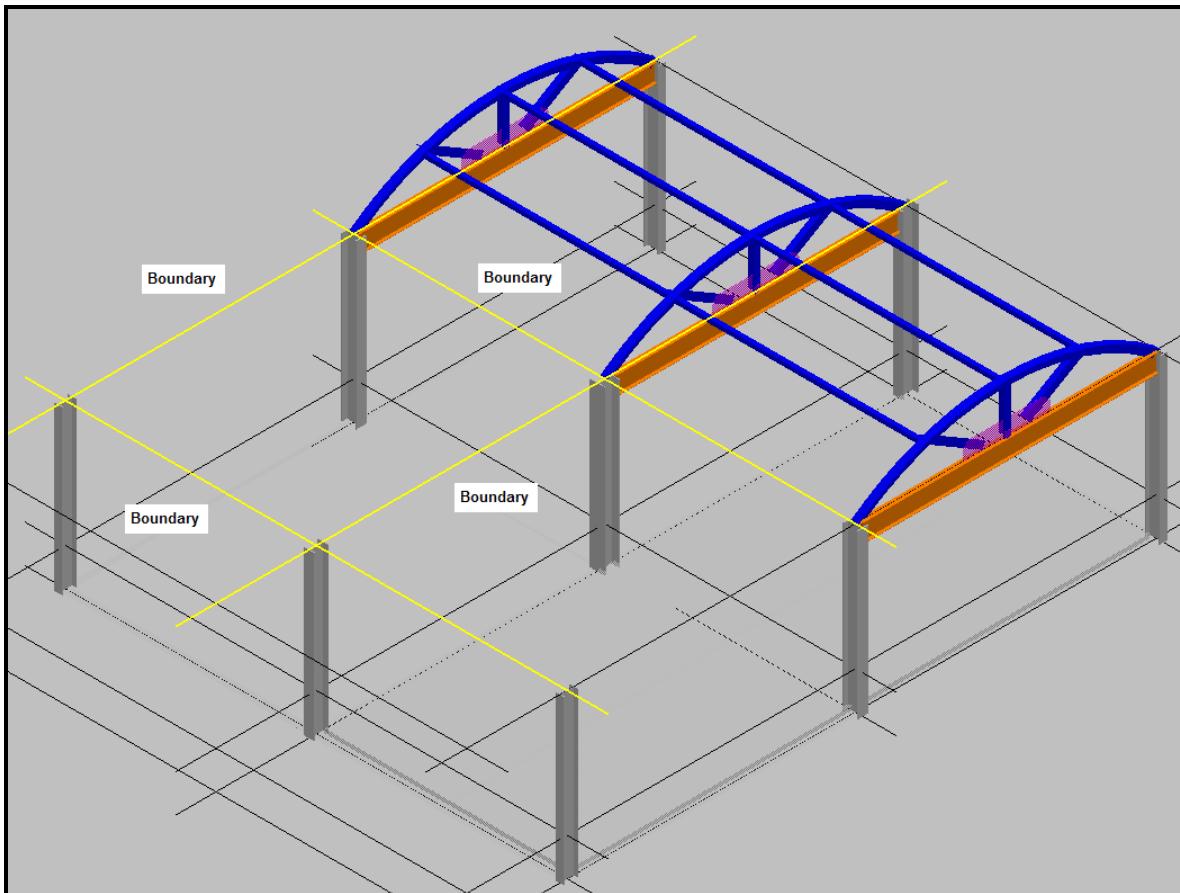
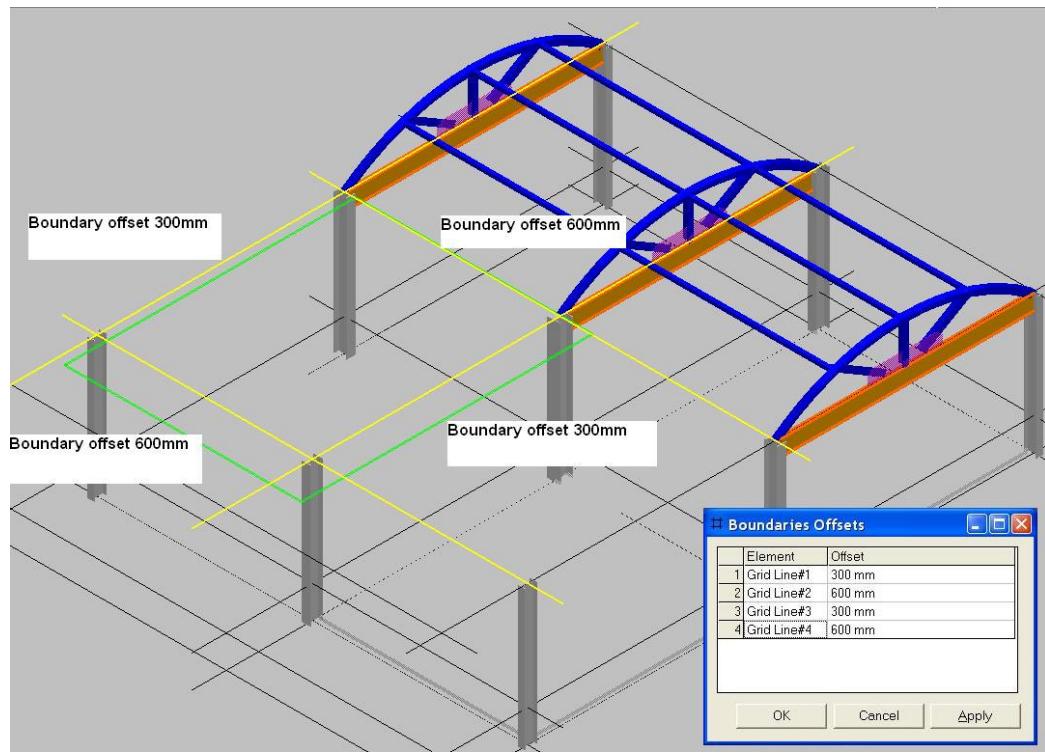


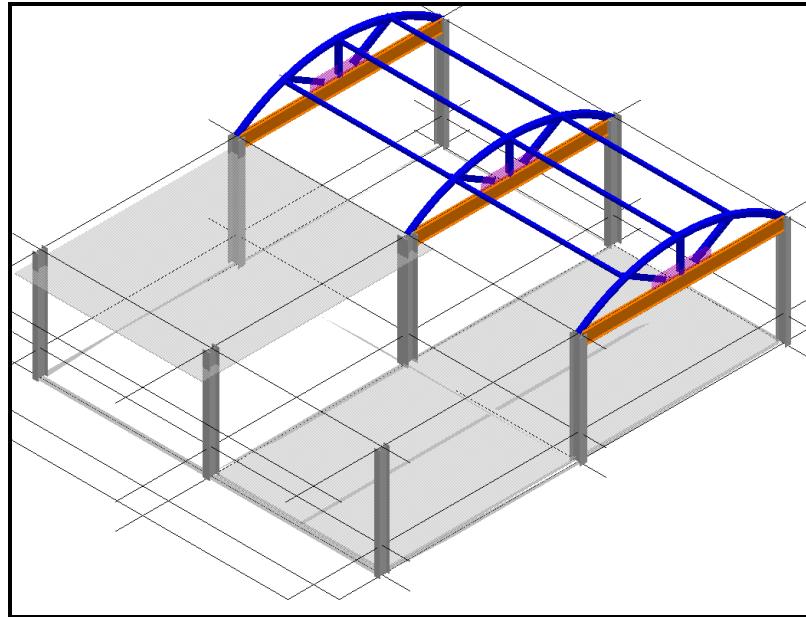
Figure 2 – ISO View of the Roof Deck

10. Select “Accept” button.
11. Define the Boundaries offsets step as shown below:



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

Your View should now resemble the following graphic:



Part IV: Place a Slab on the Far-East Bay:

- 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 in Figure 3.
- 3 Select the place slab command
- 4 Select the Top surface of the beam located at Elevation 2.70m 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 to define the edges of the slab.

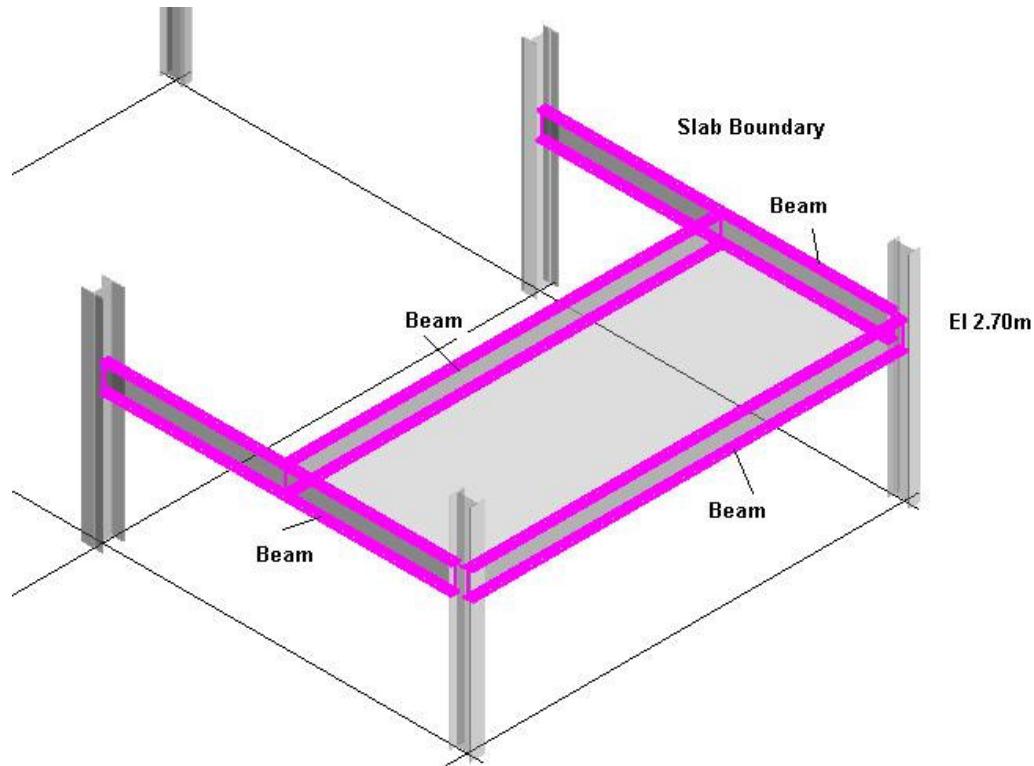
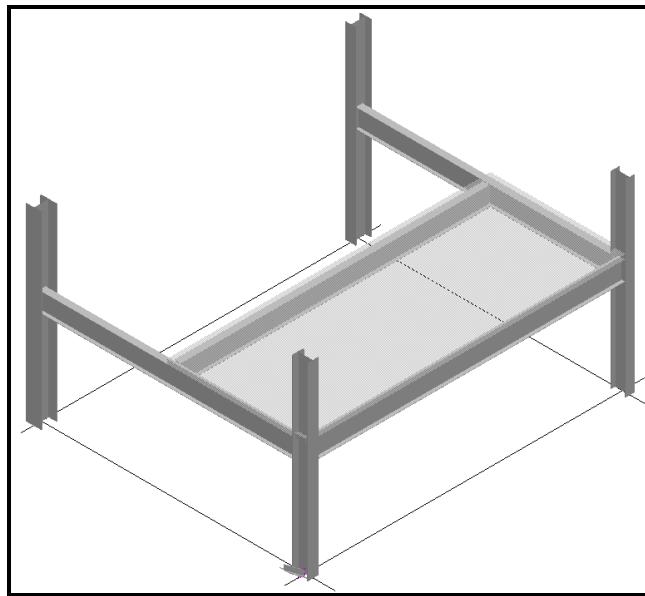


Figure 3 – ISO View of Building 1 Far-East Bay

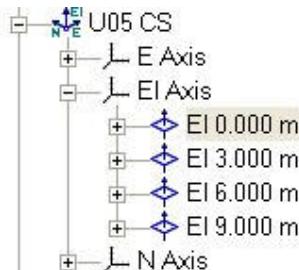
8. Select “Accept” button.
9. Hit “Finish” button to commit the transaction.

The View should now resemble the following graphic:



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

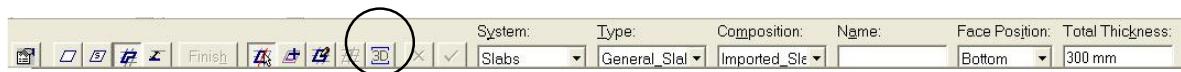
1. Select View-> Clear Clipping to remove the clipping volume.
2. Re-define your workspace to include the A2 -> U05 and CS -> U05 CS systems.
3. Select Place Slab Command.
4. Select Elevation 0.00m for the support plane using the Workspace Explorer.



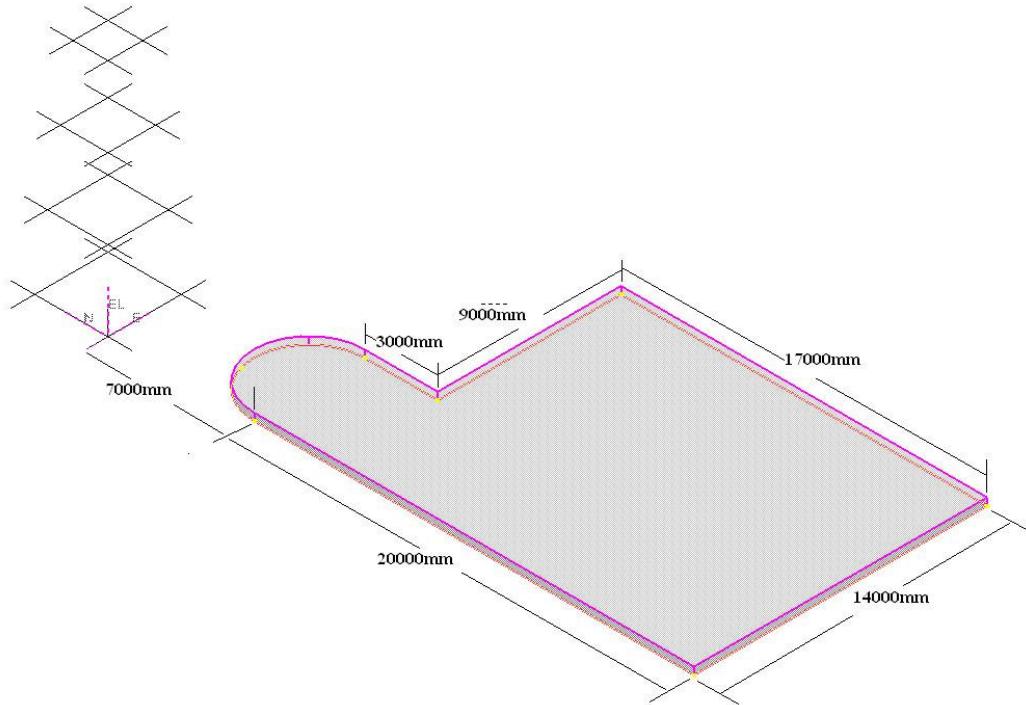
5. Select “Accept” button.
6. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U05 -> Structural -> Slabs
Slab Type:	General_Slab
Composition:	Imported_Slab
Face Position:	Bottom
Total Thickness:	300mm

7. Select the Sketch 3D step.



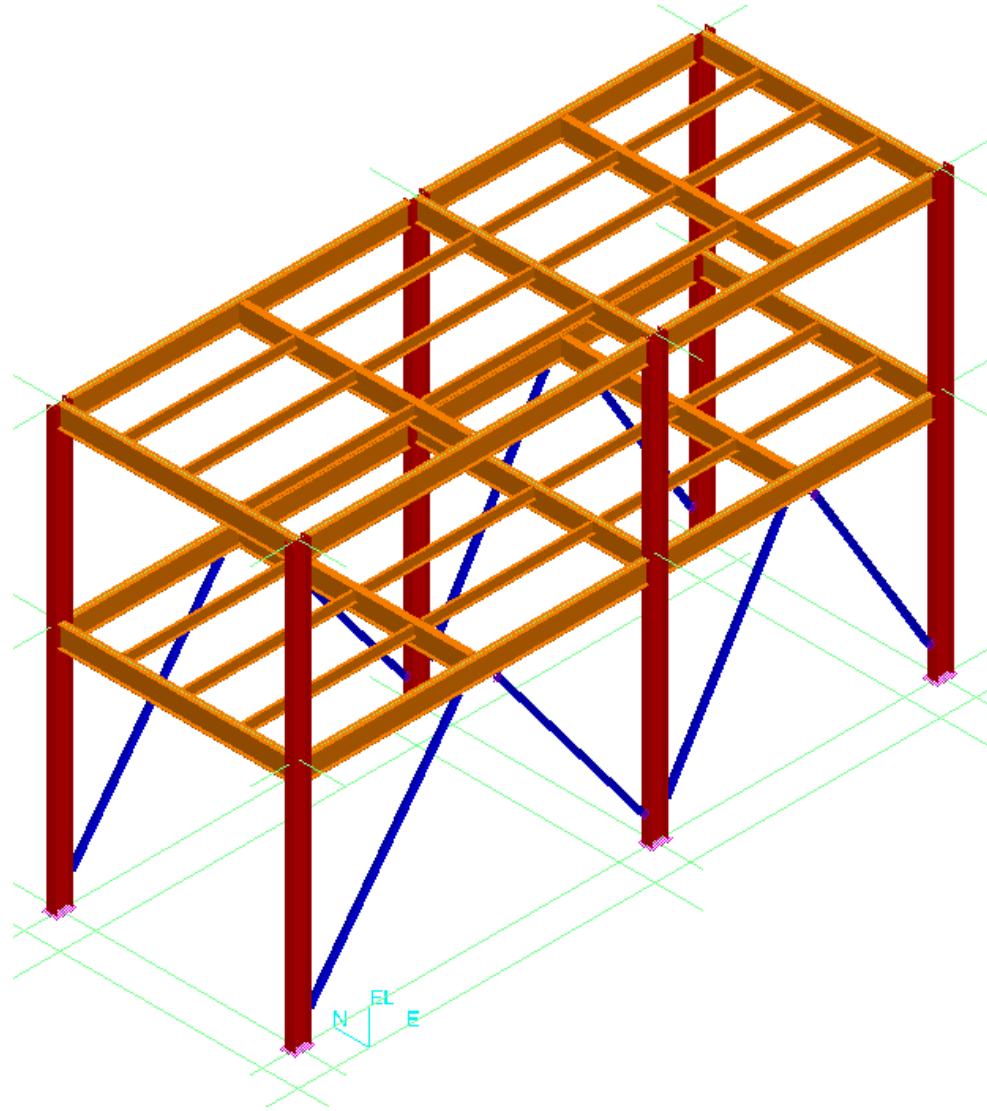
8. Sketch the boundaries of the slab as shown below:



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

Part VI – Sketch Multiple Slabs

1. Open or create a session file and define a suitable 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. Your view should resemble the following graphic.



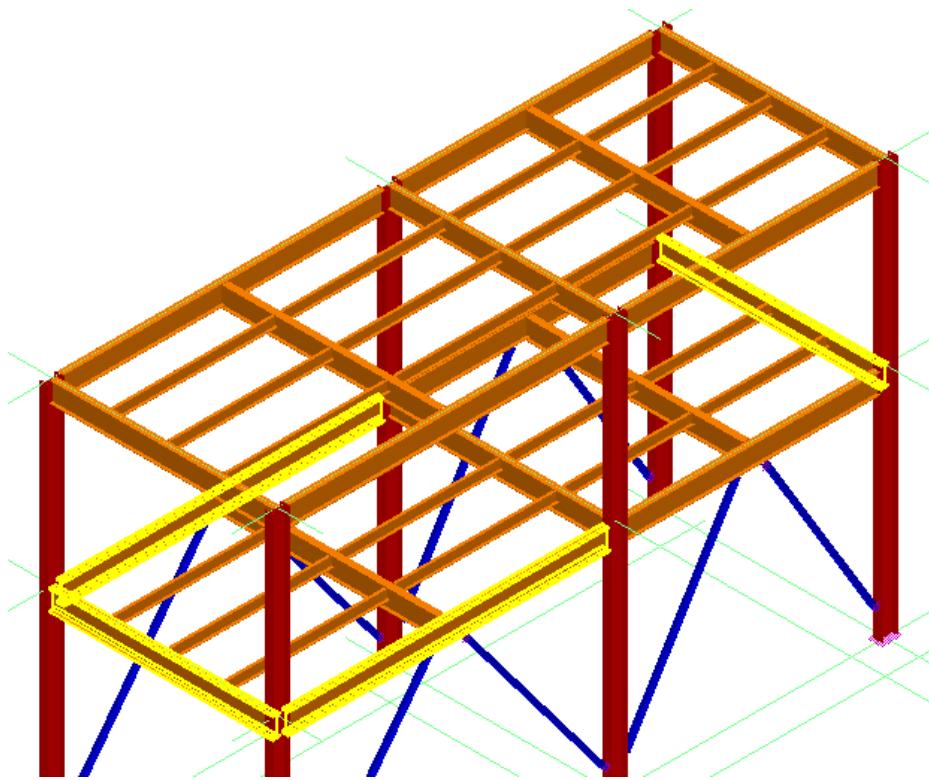
5. Select the *Slabs* command to place multiple slabs.

6. Set the slab placement parameters as follows:

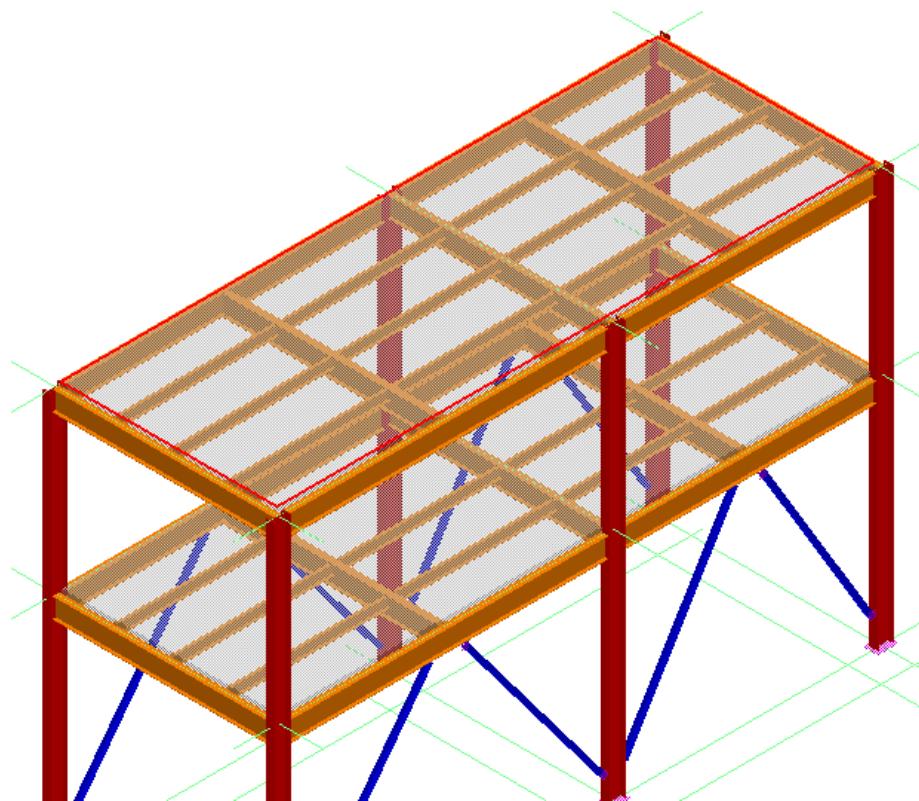
Plane Method:	Coincident
System:	A2->U02 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4" _Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Boundary Offset Reference:	Inner port-face of a bounding member

7. Select Elevation Planes at 6m and 10m for the support planes. 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.



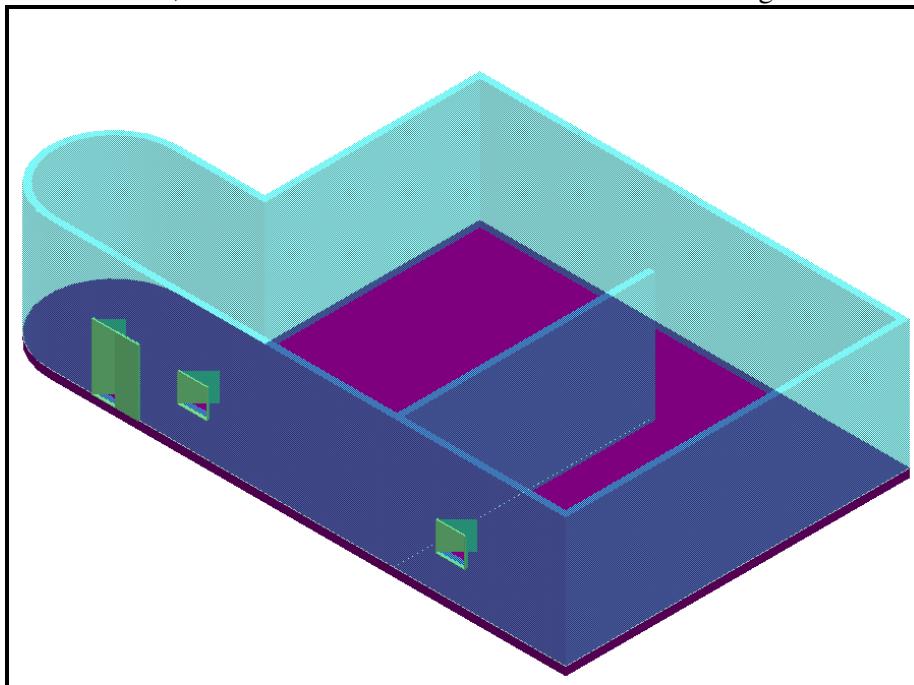
Lab 8 - Walls

Objectives

After completing this lab, you will be able to:

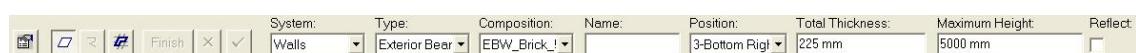
- Understand the wall entities and relationships
- Place Wall using different options
- Place Architectural objects like windows and doors in the model

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



Part I - Place wall objects to build the maintenance building:

1. Activate the PinPoint ribbon bar by Selecting Tools > PinPoint. (Make sure Active Coordinate System is set to Rectangular Coordinate mode).
2. Set the Pinpoint target to U05 CS.
3. Select Place Wall Command to open the SmartStep ribbon bar.



4. Set the wall parameters as follows:

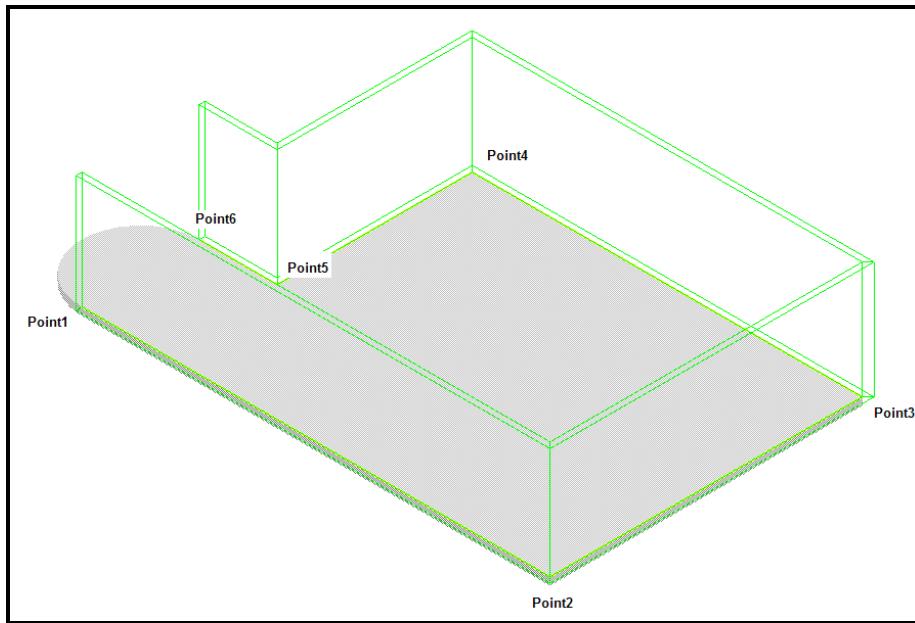
System: A2 -> U05 -> Structural -> Walls
Wall Type: Exterior Bearing Wall - Composite
Composition: EBW_Brick_5”_Air_4”_Bath&Roll_2”_Siding_Vinyl_0.0
Face Position: 3 - Bottom_Right
Total Thickness: 225mm
Maximum Height: 5000mm

5. Select the top surface of the slab for the sketching plane.

6. Select the Sketch 3D step.



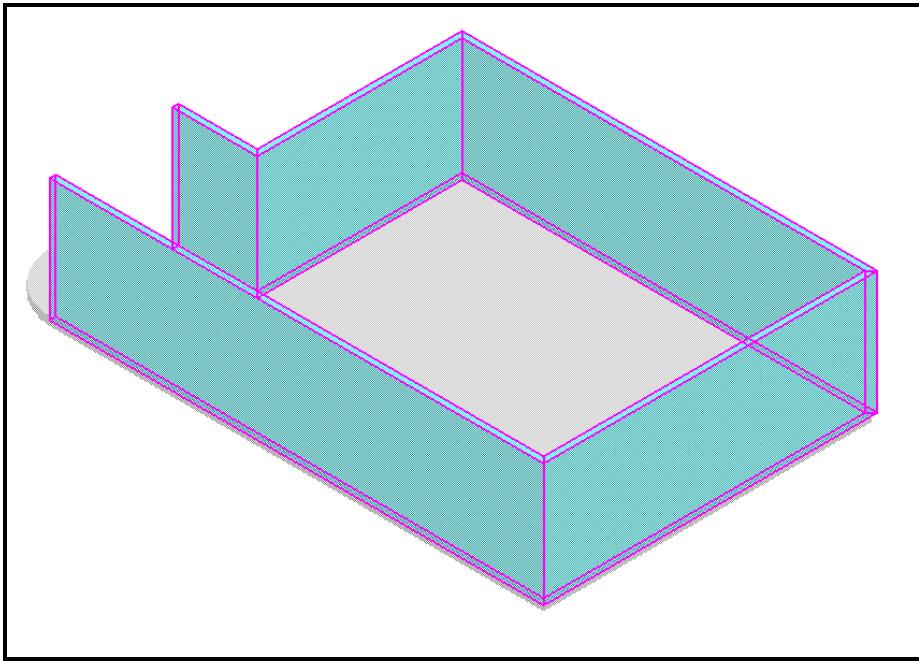
7. Sketch the boundaries of the wall as shown below:



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.

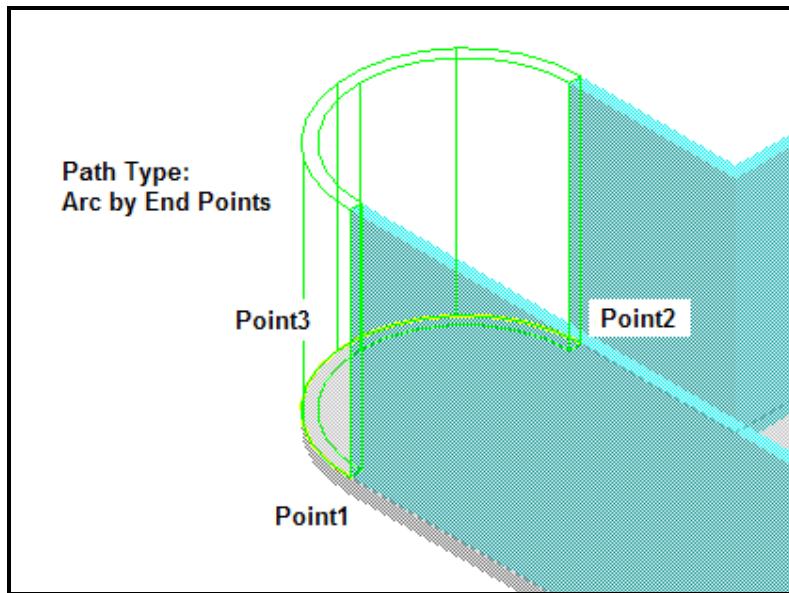
11. Set the wall parameters as follows:

System: A2 -> U05 -> Structural -> Walls
Wall Type: Exterior Bearing Wall - Composite
Composition: EBW_Brick_5"_Air_4"_Bath&Roll_2"_Siding_Vinyl_0.046
Face Position: 3 Bottom_Right
Total Thickness: 225mm
Maximum Height: 5000mm

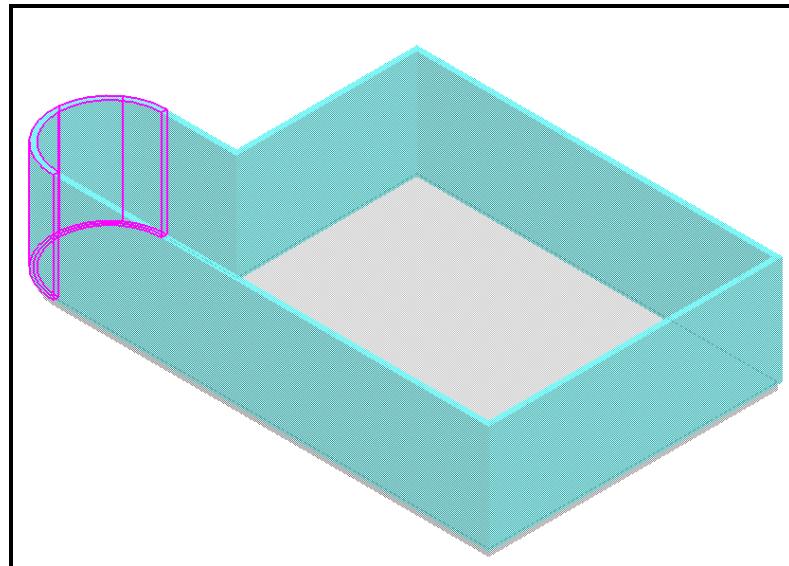
12. Select the top surface of the slab for the sketching plane.

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.

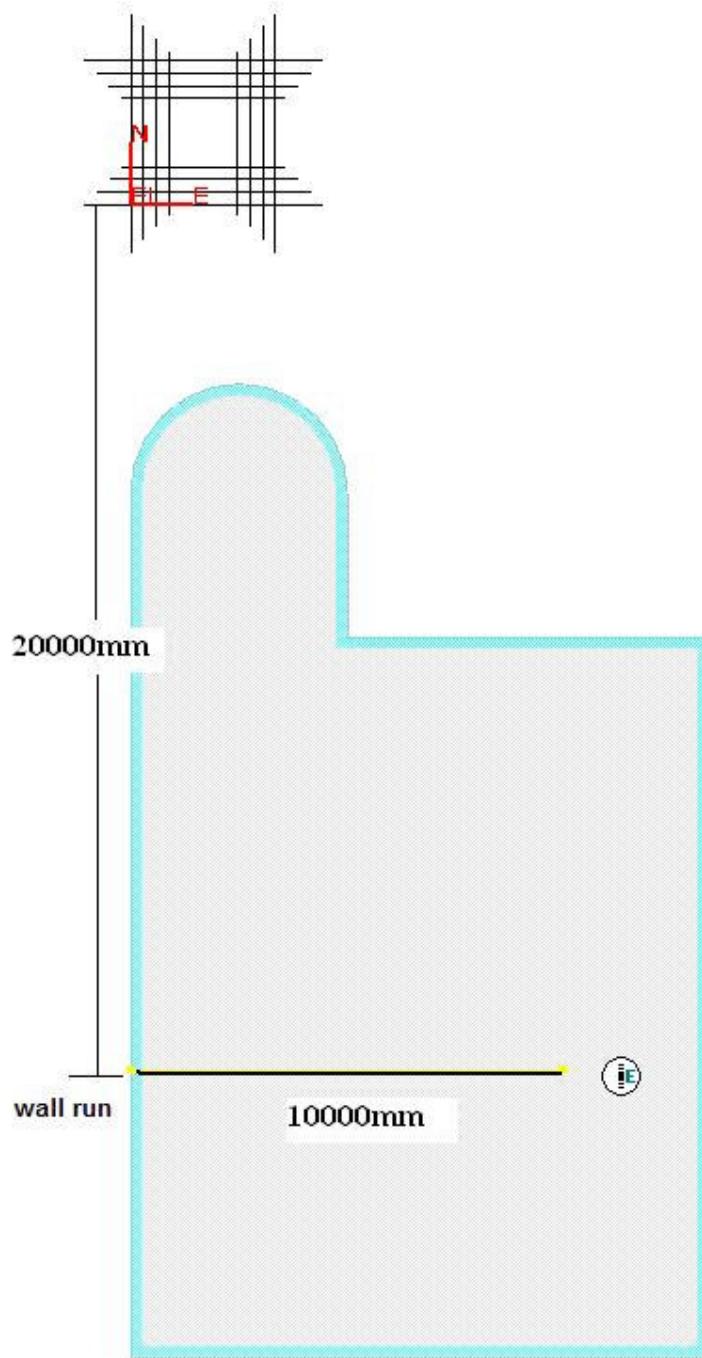


16. Repeat the above step to place a divider wall.
17. Set the wall parameters as follows:

System:	A2 -> U05 -> Structural -> Walls
Wall Type:	Interior Bearing Wall – Non-Composite
Composition:	IBW_Stone_Granite_6”
Face Position:	2 Bottom_Center
Total Thickness:	225mm
Maximum Height:	5000mm

18. Select the top surface of the slab for the sketching plane.

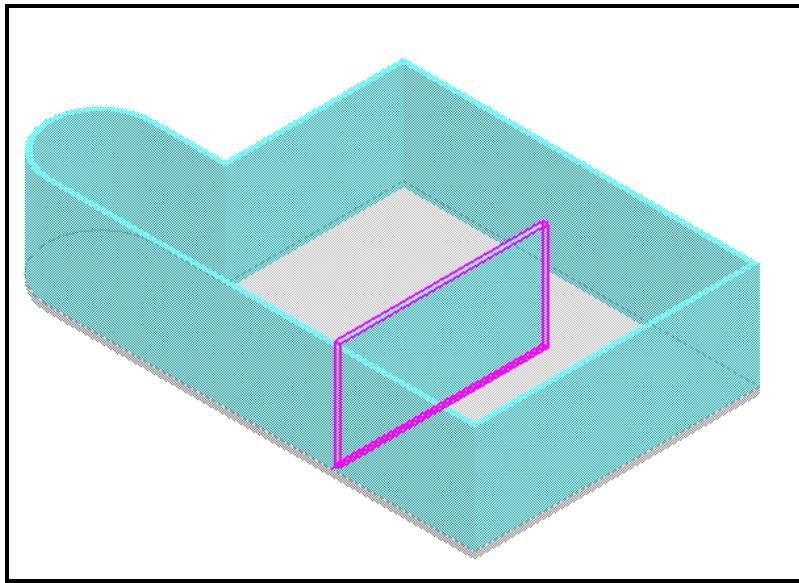
19. During the sketch step, select the path type as Line and define the two points as shown below:



Top View of the Maintenance Building

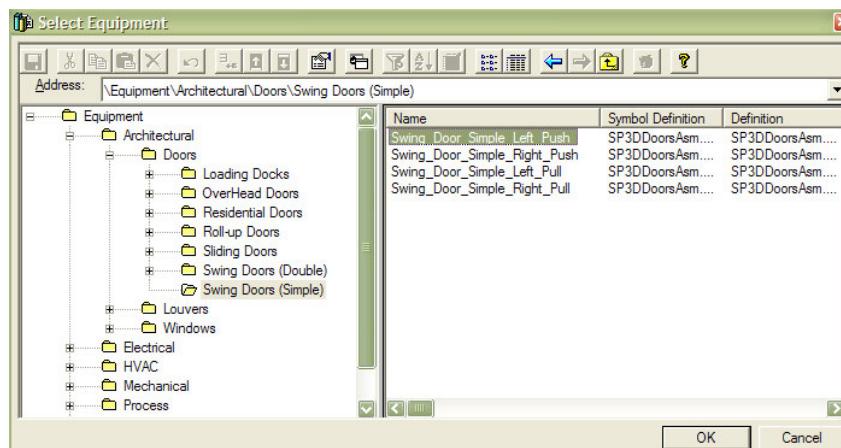
20. Select “Finish Path” button to commit the wall run (path). System returns to the SmartStep ribbon bar of the Place Wall command.
21. Select “Finish” button to persist the wall part in the model.

Your View should now resemble the following graphic:



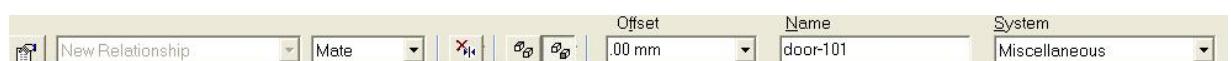
Part II - Place Door and Windows to the maintenance building:

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



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

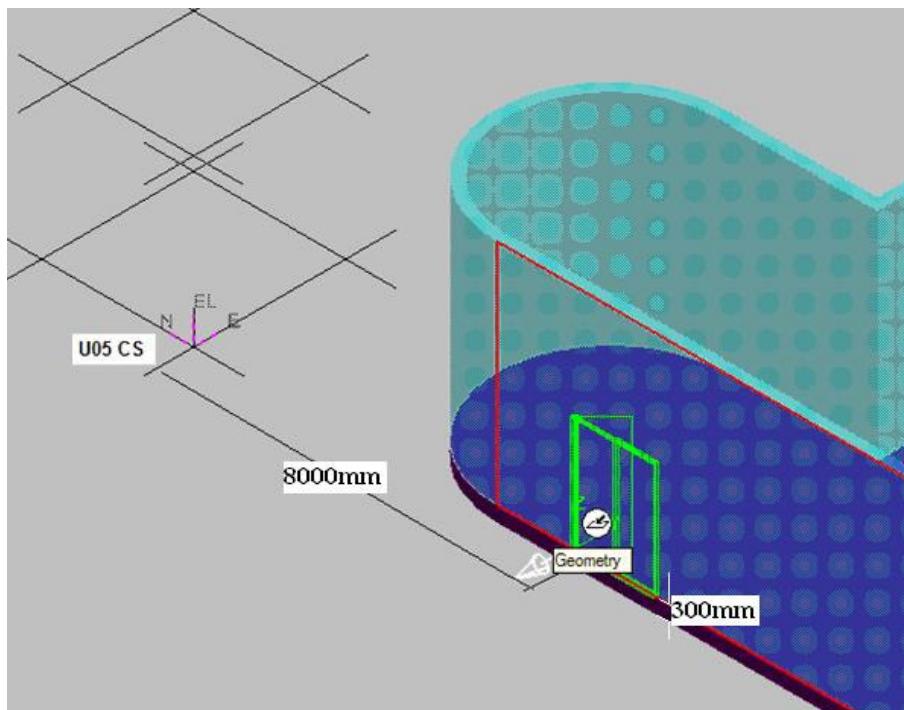


7. Go to the PinPoint ribbon bar and key in -8000mm for North and 300mm for Elevation. Don't keyin anything or lock the Easting field.

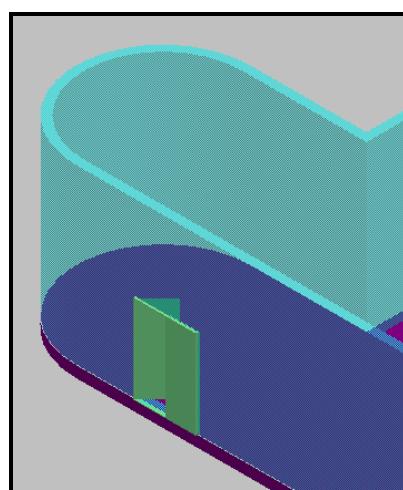


8. Use the Left Arrow key to rotate the equipment 90 deg about the z-axis.

9. Move the cursor over the wall object as shown below:



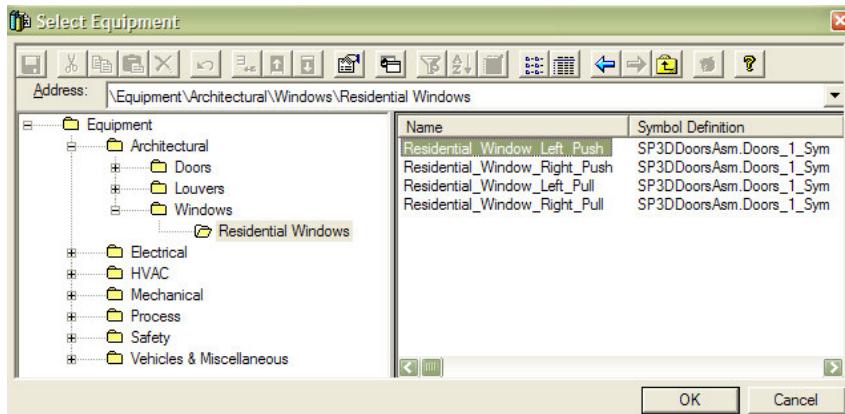
10. Left Mouse Click to place the door on the model.



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:

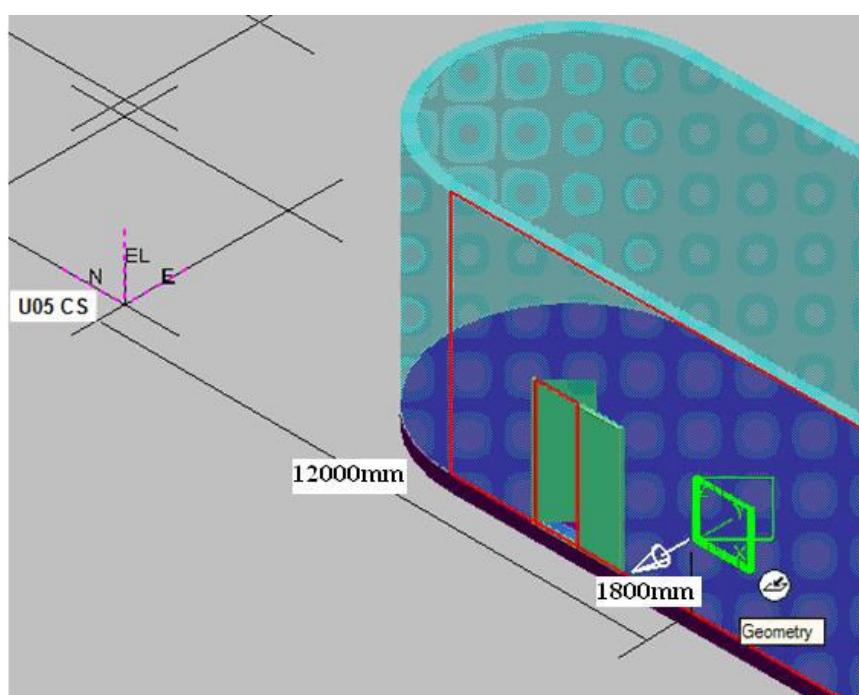
System: A2 -> U05 -> Structural -> Miscellaneous
 Name: Window-101



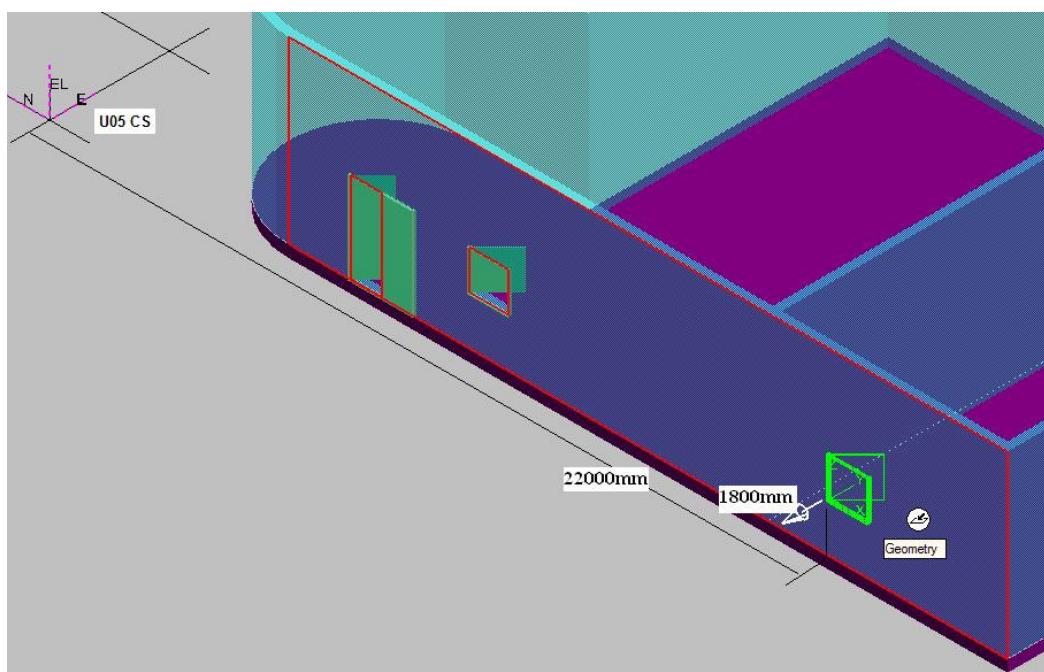
16. Go to the PinPoint ribbon bar and key in -12000mm for North and 1800mm for Elevation. Don’t keyin anything or lock the Easting field.



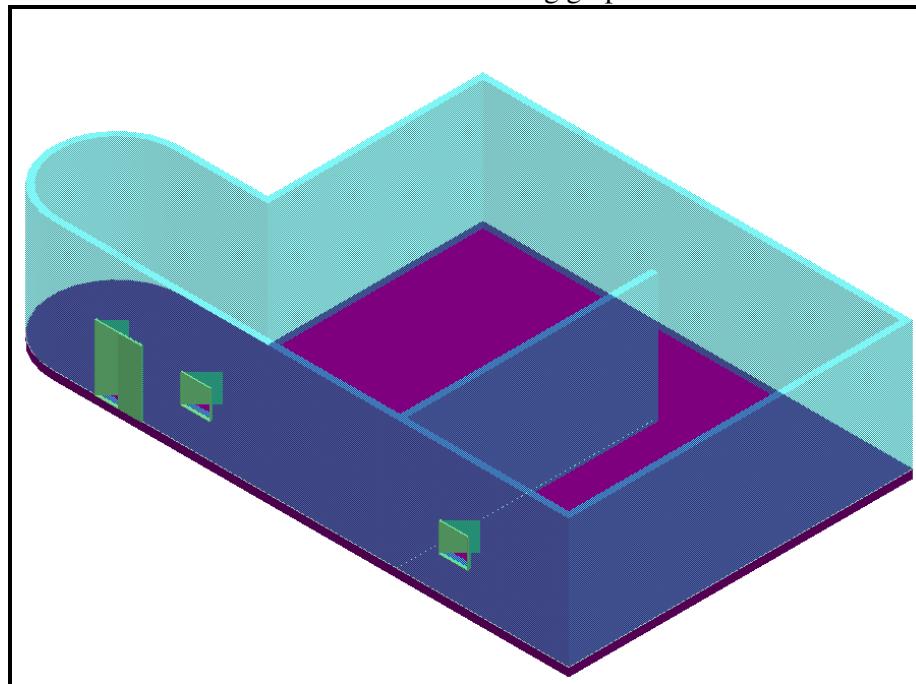
17. Use the Left Arrow key to rotate the equipment 90 deg about the z-axis.
18. Move the cursor over the wall object as shown below:



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.

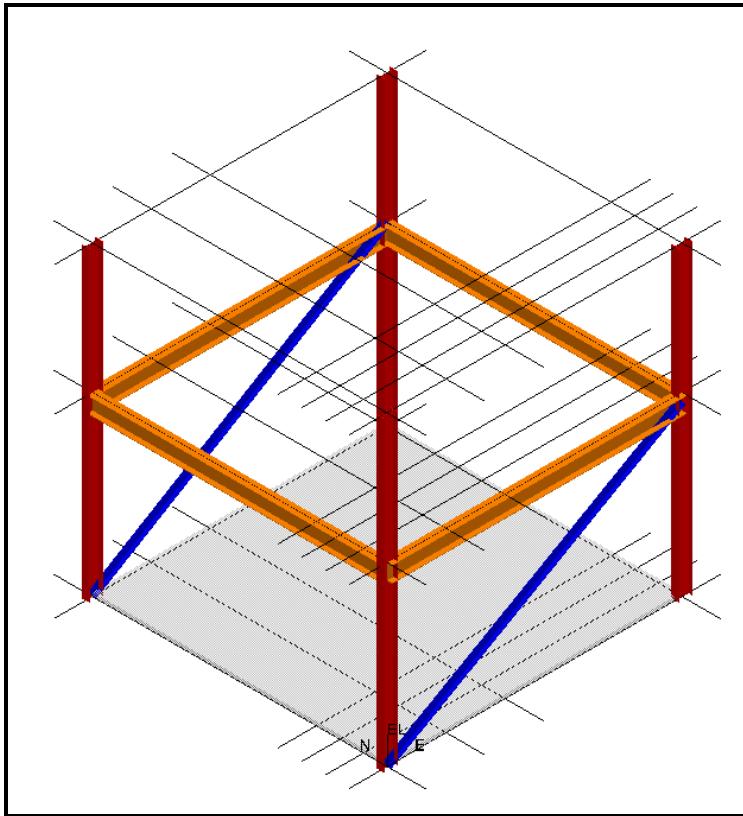


Your View should now resemble the following graphic:



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:

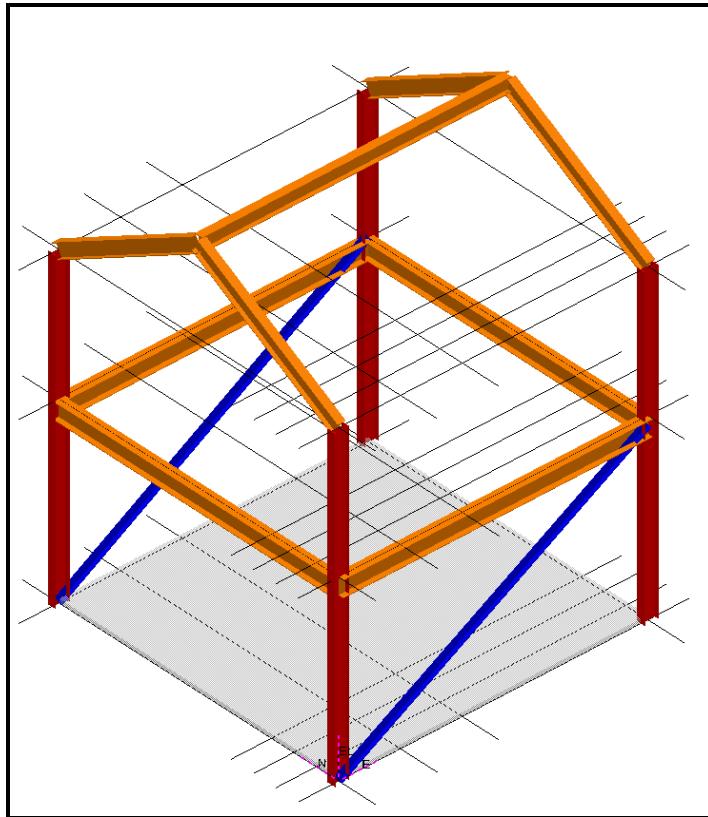


2. Select Place Linear Member System command. System displays the smart step ribbon bar.
3. Use the ribbon bar to set the active member parameters as follows:

Connection:	By Rule
System:	A2 -> 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 8m and Elevation 9m intersection.
5. Place End 2 at East 0mm North 4000mm and Elevation 11000mm
6. Continue to place the second beam (End 3) at the end of the column located at North Plane 0m and Elevation 9m

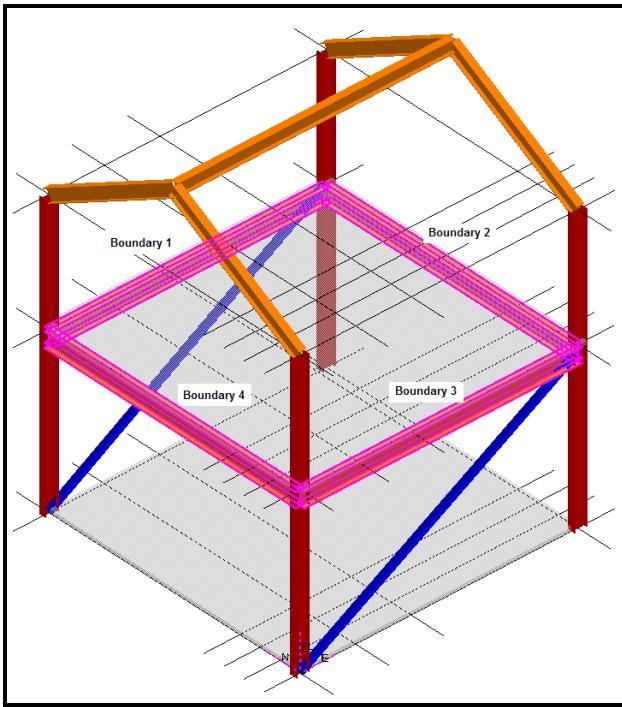
7. Repeat the above workflow to finish building the roof as shown below:



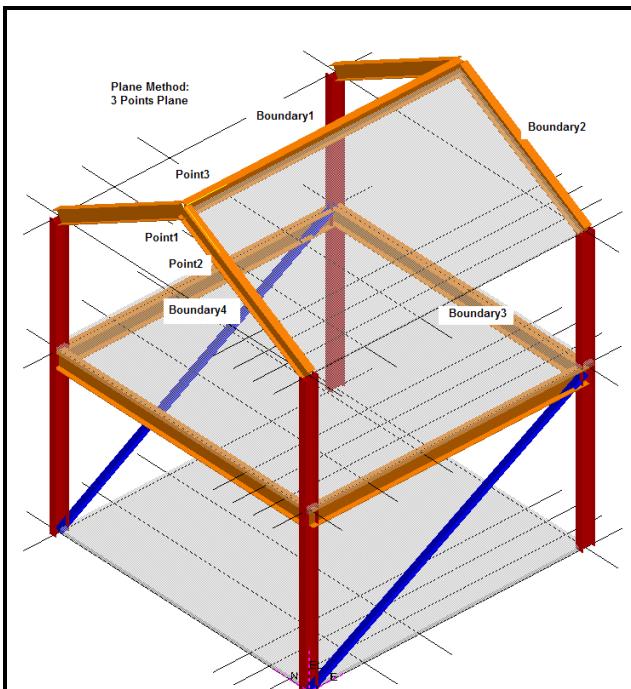
8. Select Place Slab command.
9. Select the Top surface of the beam located at Elevation 5m for the support plane.
10. Select “Accept” button.
11. Set the slab placement parameters as follows:

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:	102mm
Boundary Offset Reference:	Outer port-face of a bounding member

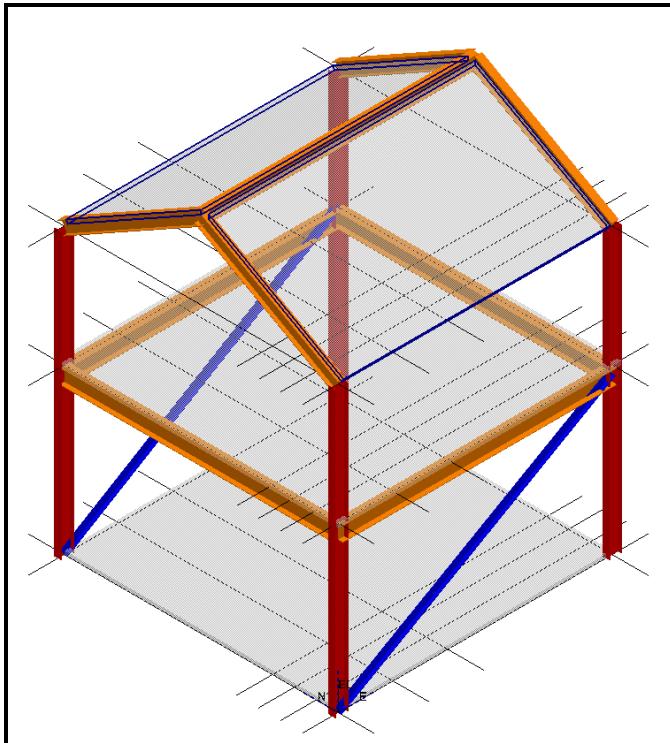
12. Select the boundaries as shown below:



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:



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



18. Set the wall parameters as follows:

System: A2 -> U04 -> Structural -> Walls
 Wall Type: Exterior Bearing Wall - Composite
 Composition: EBW_Brick_5”_Air_4”_Bath&Roll_2”_Siding_Vinyl_0.046”
 Face Position: 2 - Bottom_Center
 Total Thickness: 200mm
 Maximum Height: 6000mm

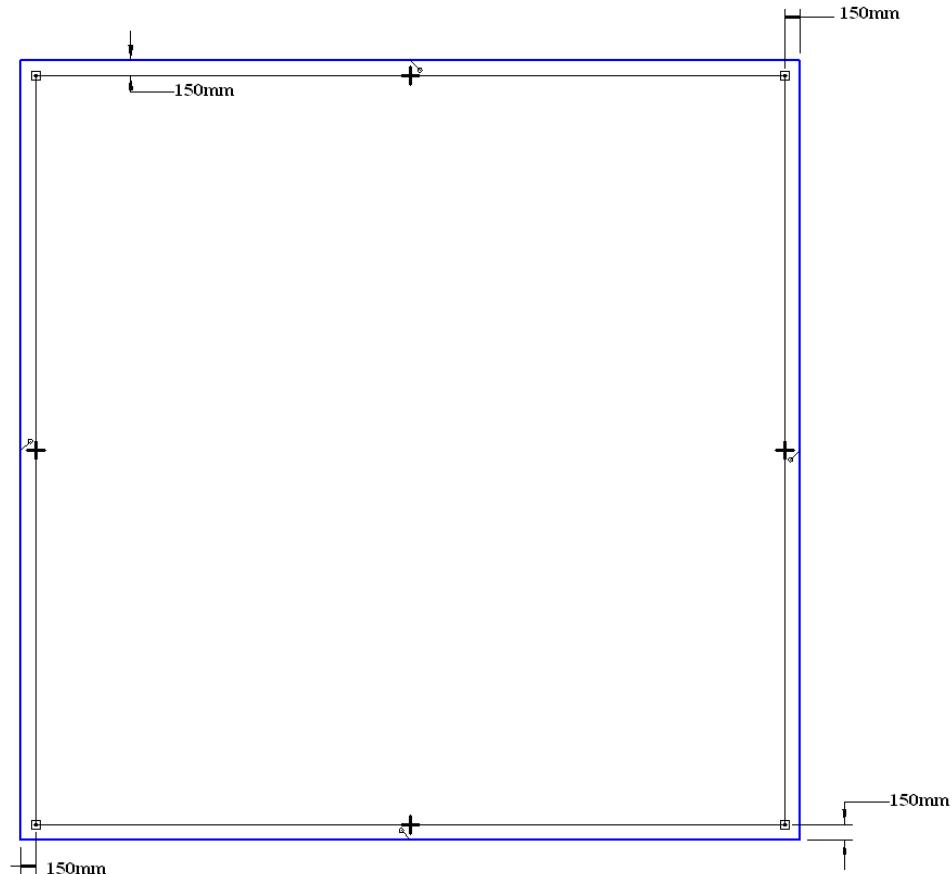
19. Select the top surface of the slab (EL 5.0m) for the sketching plane.

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:



22. Hit “Close” button to return to the SmartStep ribbon bar.

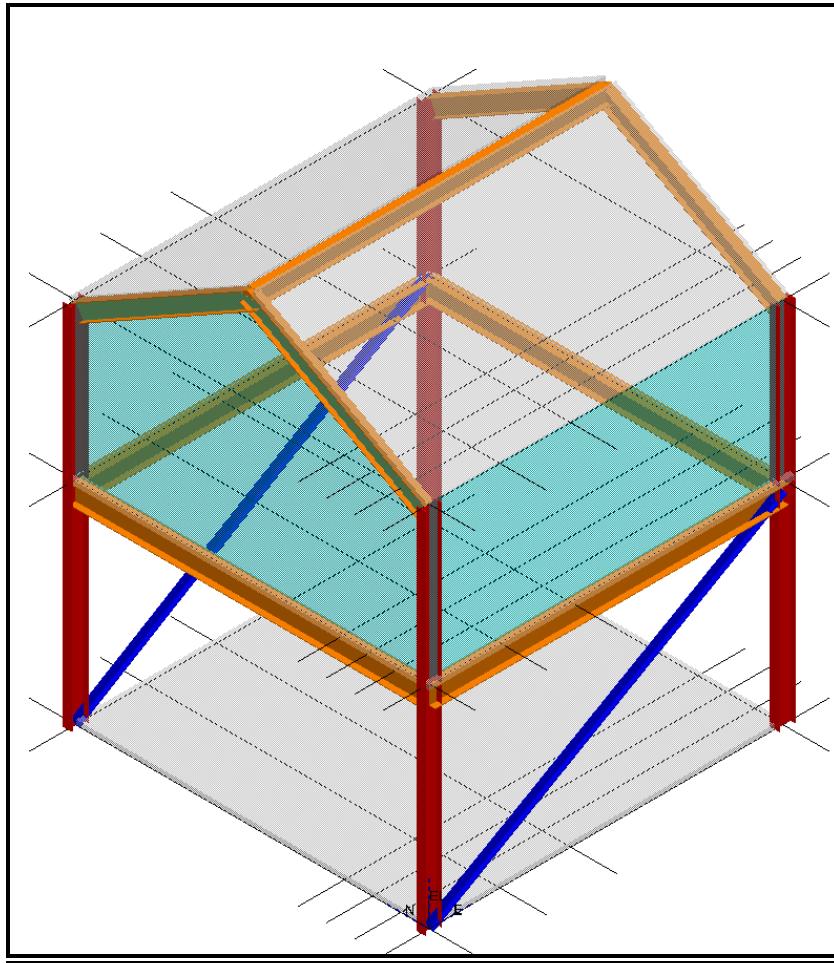


23. Select Finish button.

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:



Part IV: Split Walls 3-D sketch

- 1 Re-define your workspace to include the A2 -> U06 and CS -> U06 CS systems.
- 2 Select the Place Wall command to open the SmartStep ribbon bar

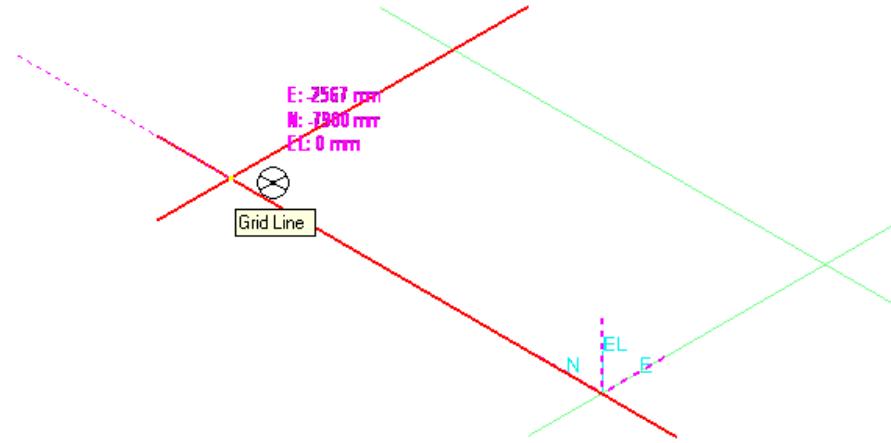


3. Set the wall parameters as follows:

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: 250mm
 Maximum Height: 6000mm

4. Select the elevation plane E10.000 m as the sketching plane.
5. Select the Sketch 3D step.

6. Select the intersection of the grids as shown below.

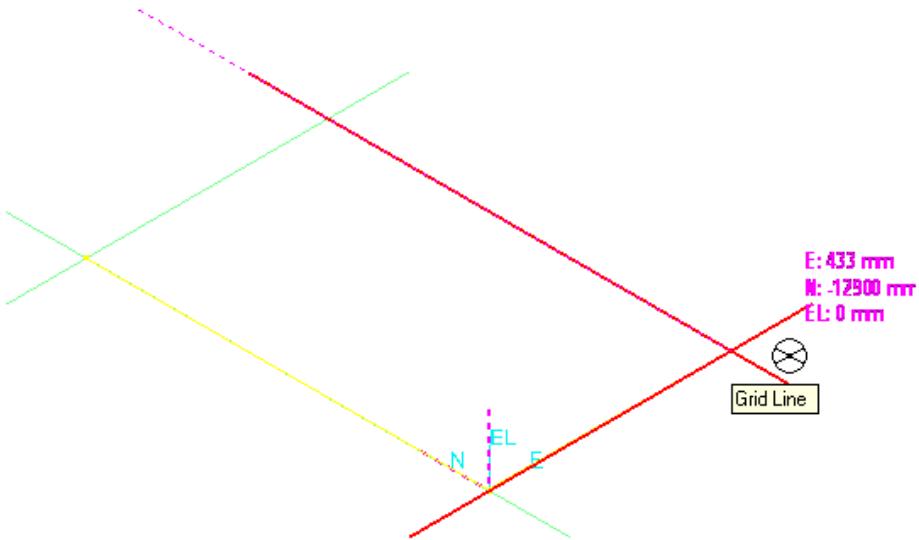


7. Select the next intersection along the North grid line and perform a left mouse click.

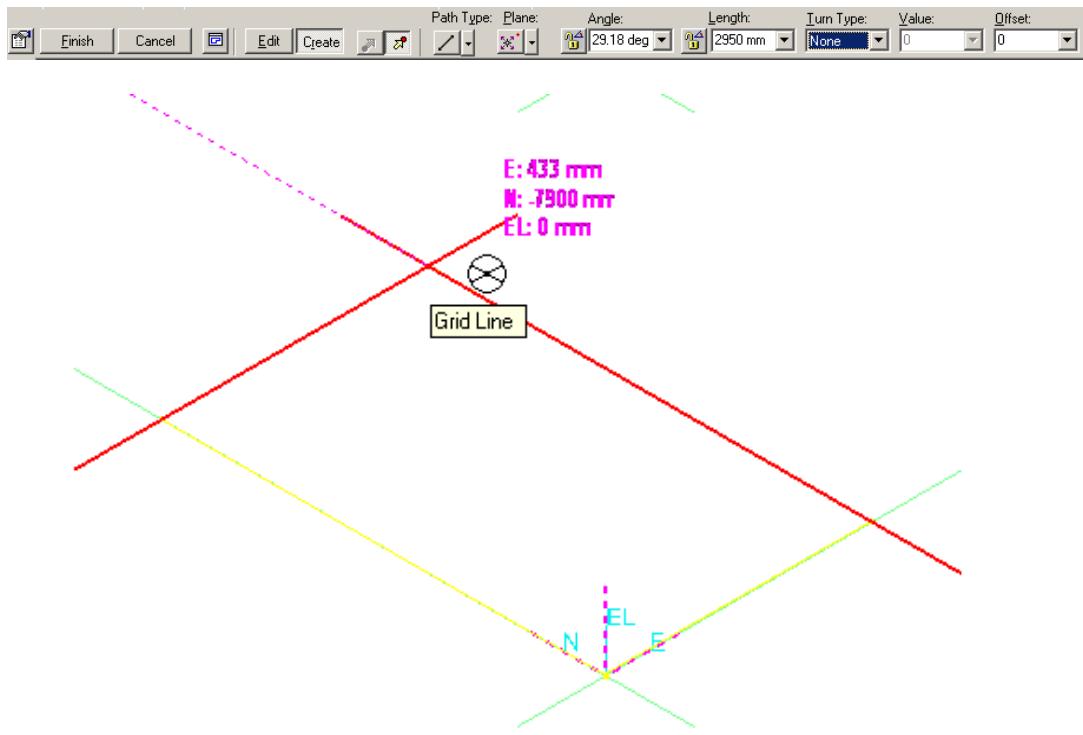
8. Go to the ribbon bar and change the Turn Type to “Split”.



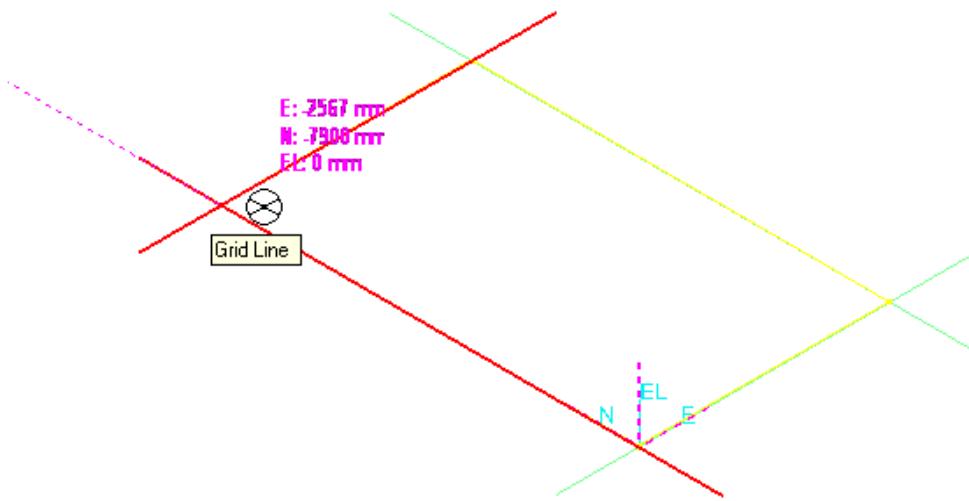
9. Now select the intersection as show below.



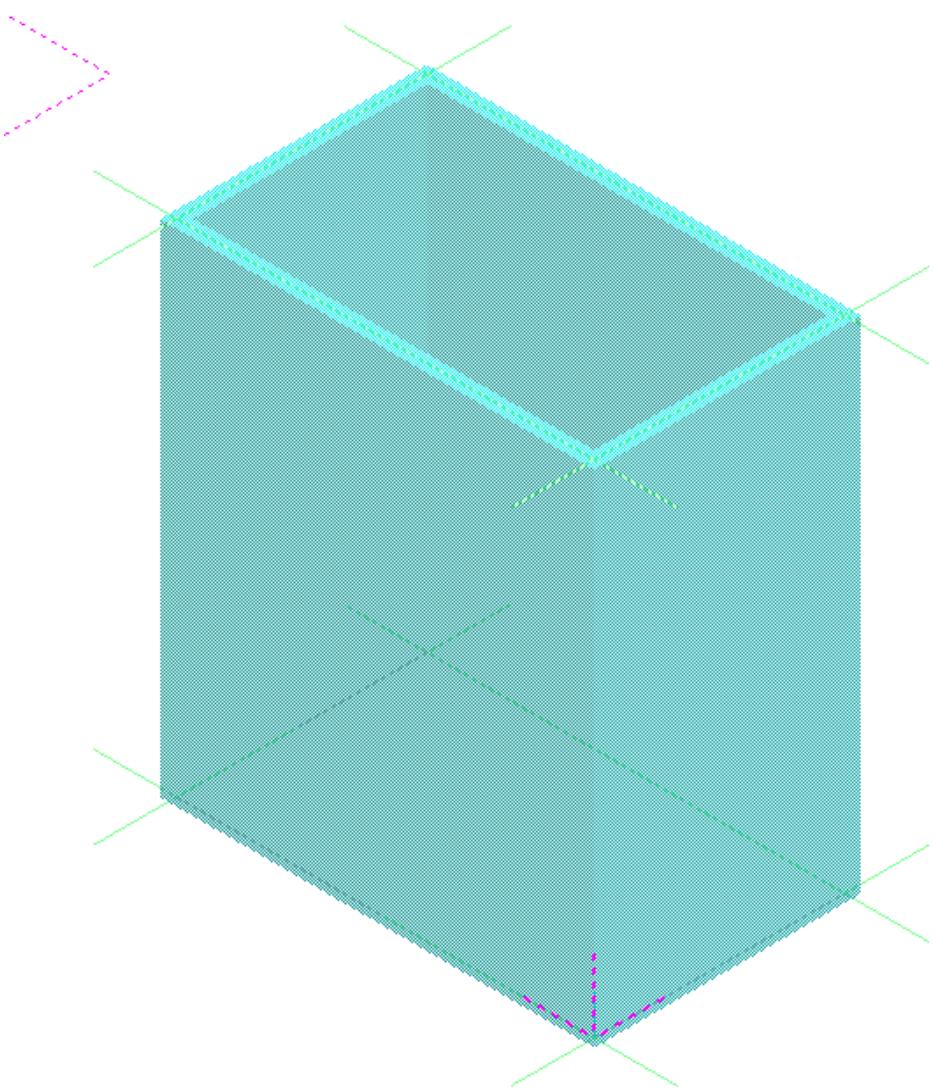
10. Change the Turn Type to “None” and then select the next intersection along the North as shown below.



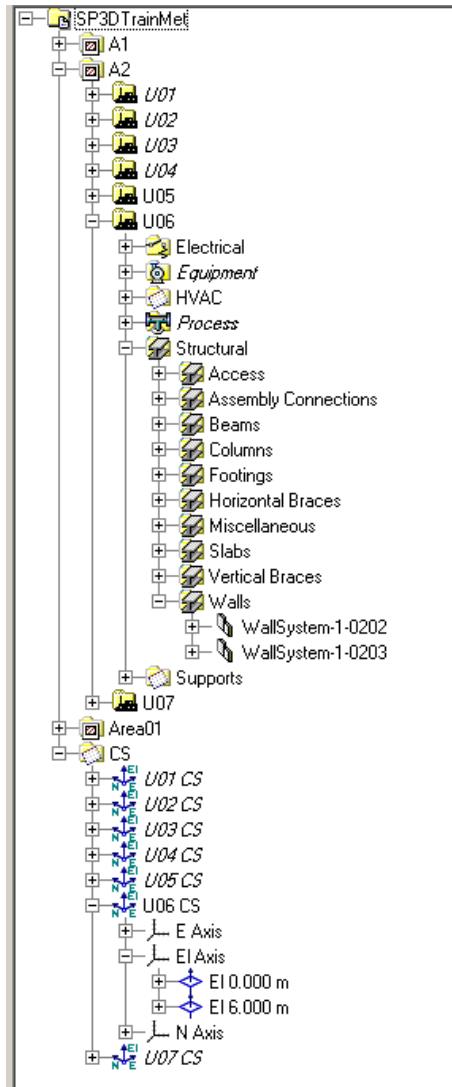
11. Change the Turn Type to Split again and select the intersection as shown below.



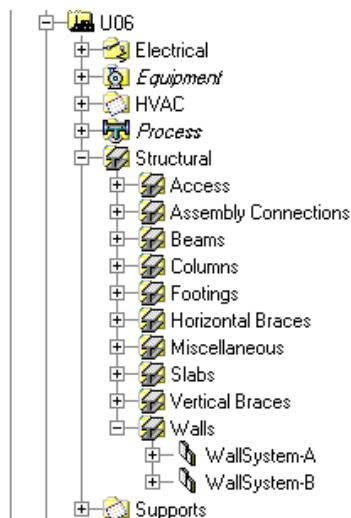
12. Hit Finish twice. Your view should resemble the following graphic.



13. The two wall systems are created as shown in the Workspace explorer window



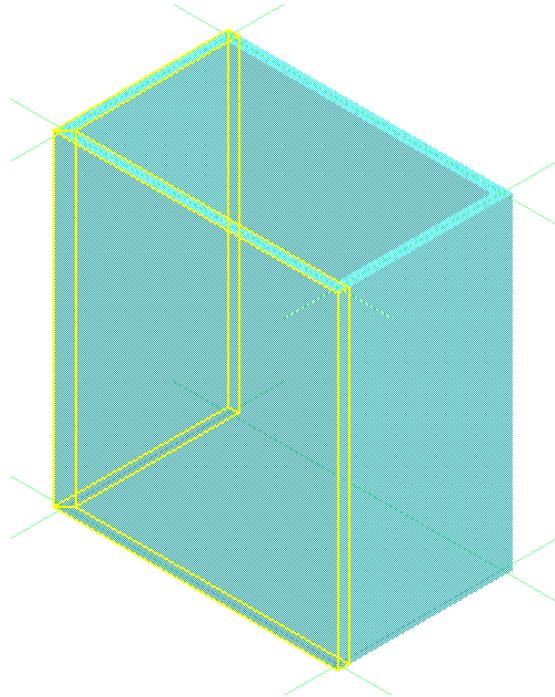
14. Rename the first as WallSystem-A and rename the second as WallSystem-B. Your workspace explorer should view as shown below.



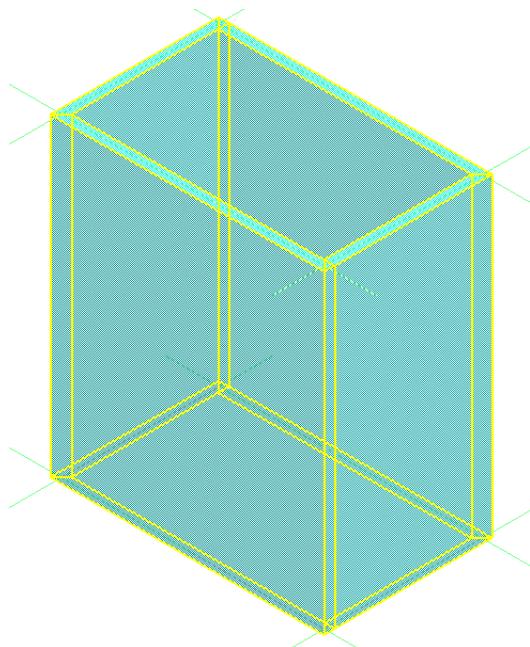
Part V: Toggle Wall Corner

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.

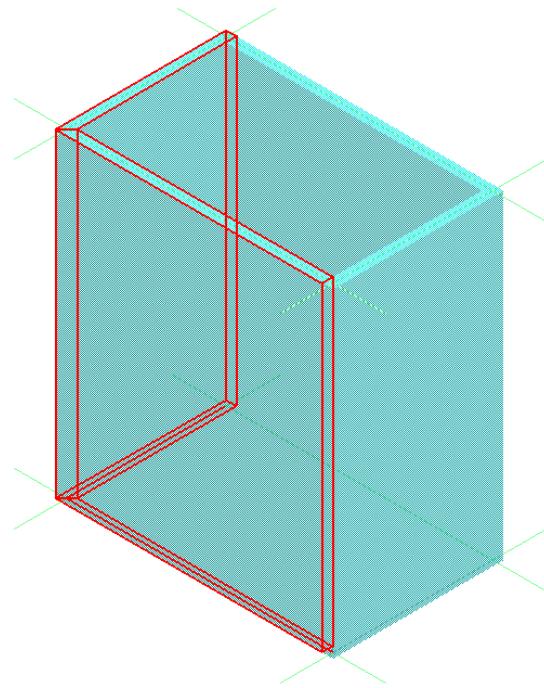


17. Select the WallSystem-B as shown below.



18. Hit Finish.

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

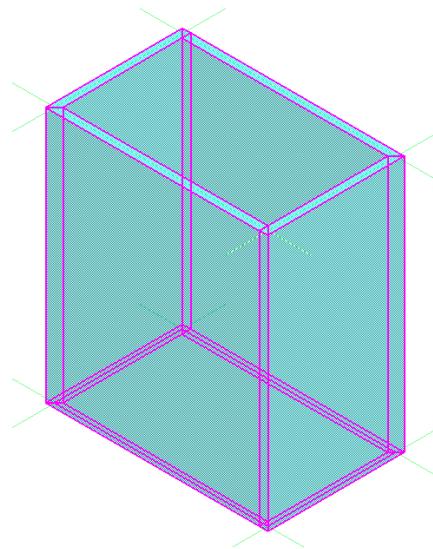


20. When two separate walls join at a corner, one wall extends past the other wall to form the corner. This command is used to toggle which wall extends past the other.

Part VI: Multiple Wall modifications

21. Multiple walls can be selected for modification. Select WallSystem-A and WallSystem-B

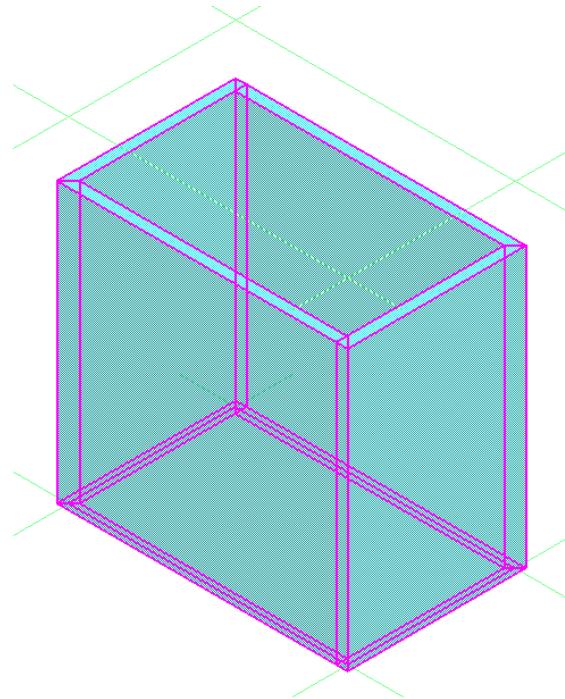
22. Your view should resemble the following graphic.



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



24. Change the Total Thickness to 200mm and Maximum Height to 5000mm. Hit Enter. Your view should resemble the following graphic.



Lab 9 - Openings

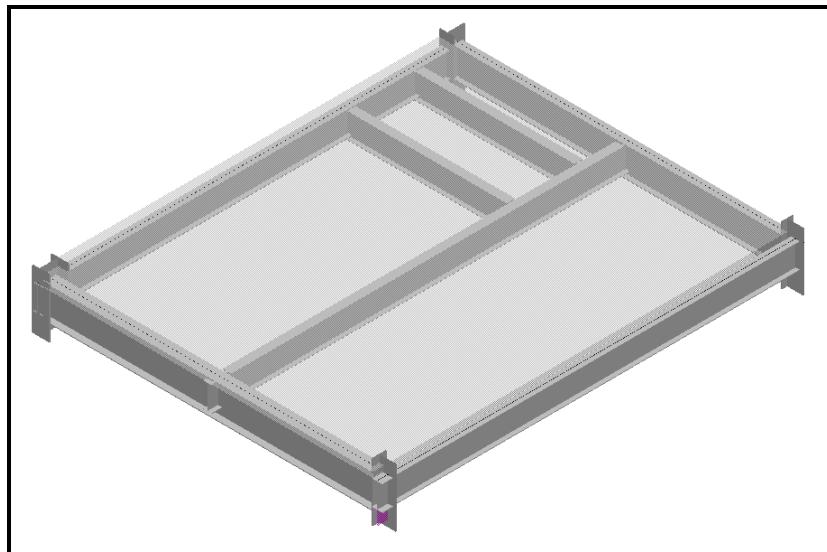
Objective

After completing this lab, you will be able to:

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

Part I: Place a Stairway Opening

- 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:



- 3 Select the place opening command
- 4 Select the slab located in U03 on the first floor frame as shown in Figure 1
- 5 Keep the default parameters in the smartstep ribbon bar.



- 6 Select the members as shown in Figure 1 to define the boundaries of the opening. Use the quick pick service to select the members.
- 7 After selecting the members, select the Finish button.

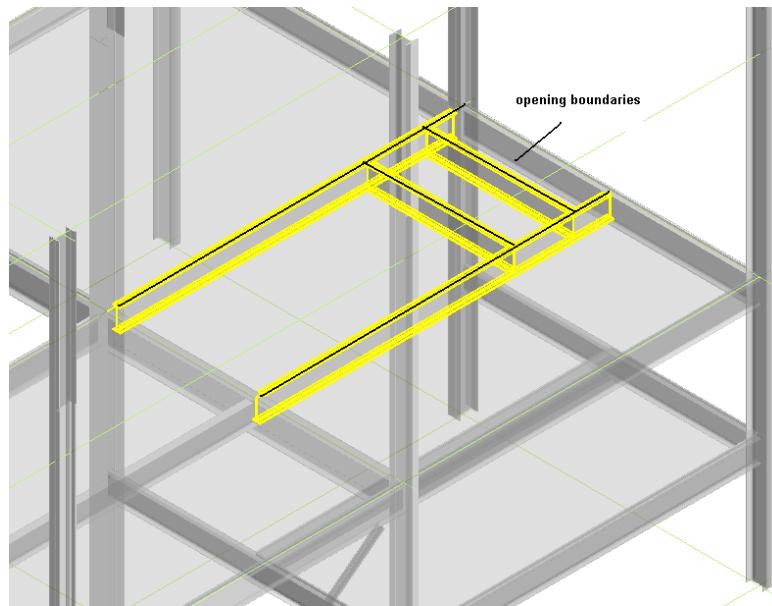


Figure 1 – ISO View of the Stairway Opening

A clipped ISO View should now resemble the following graphic:

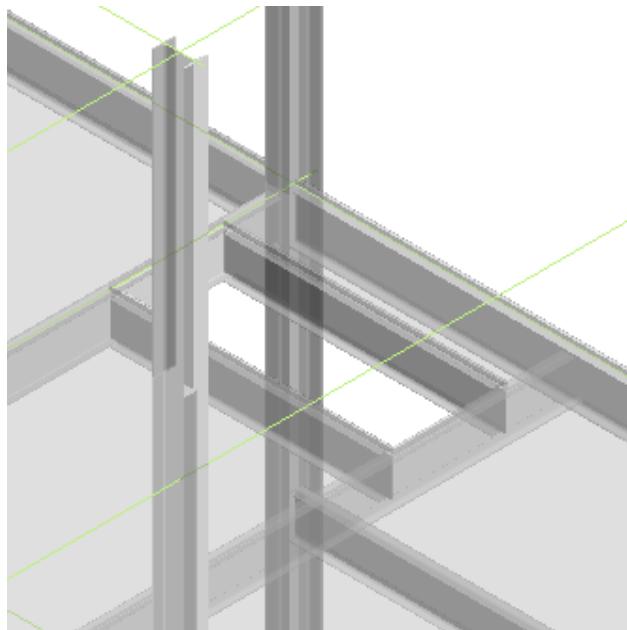
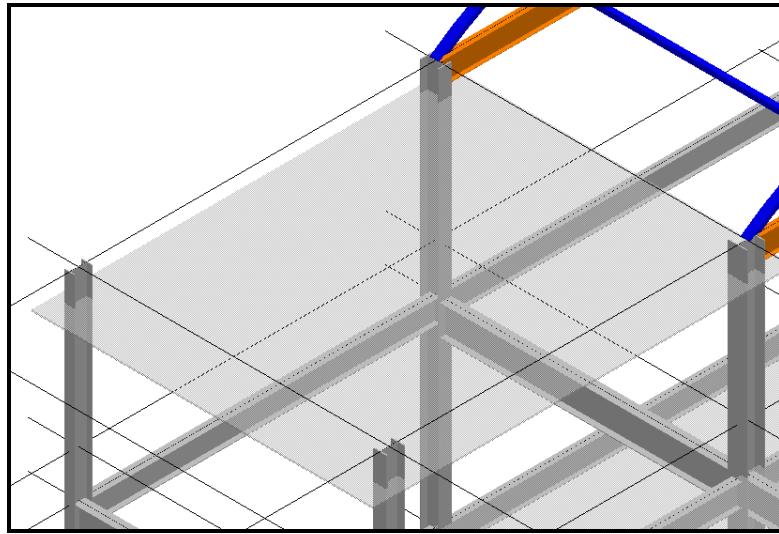


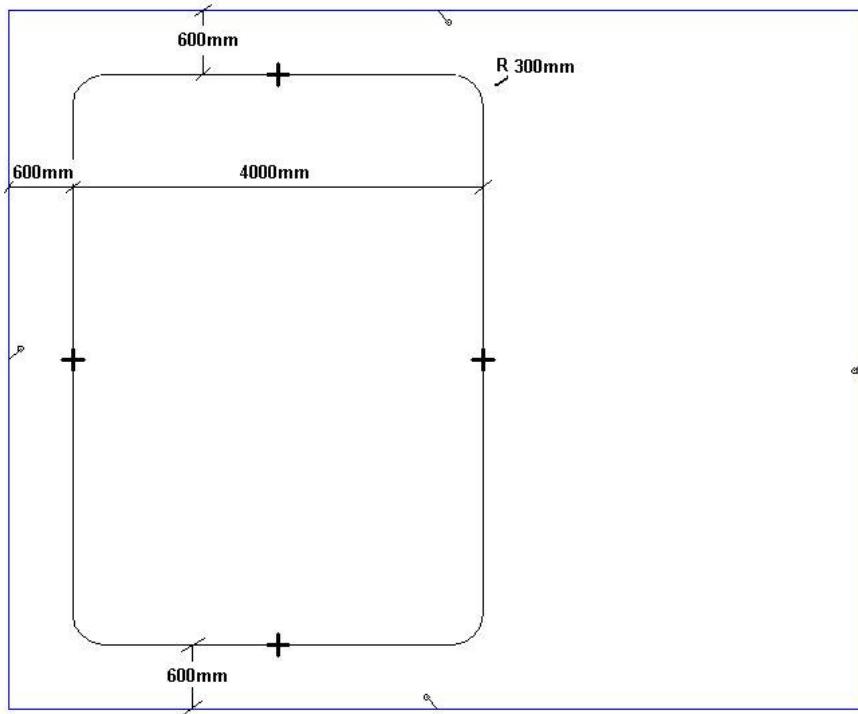
Figure 2 – ISO View of the Stairway Opening in Building 1

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 the place opening command
- 4 Select the roof deck located at Elevation 14m in Building 1 as shown in Figure 4.
- 5 Select the draw method button to define the boundaries of the opening. System automatically opens the Draft 2D Task.
4. Create the hole profile as per the sketch below.



Hint: From File -> Properties set the units to mm with no decimal places.

Use pin point, place line and fillet commands

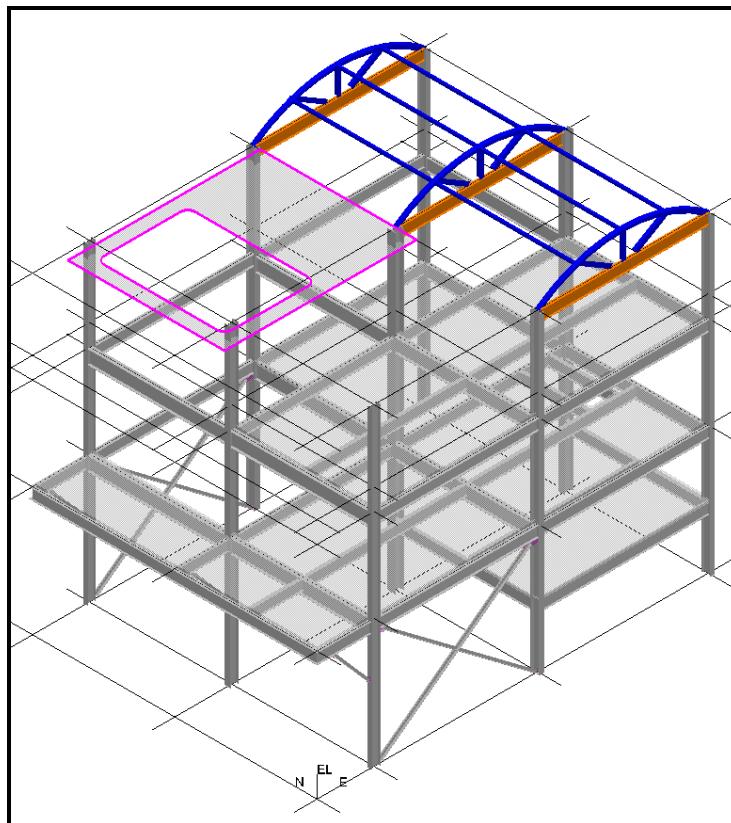
6 Select the Close button from within Draft2D

7 Keep the default parameters



8 Select the Finish button from the Slab ribbon bar.

Your View should now resemble the following graphic:



Lab 10 - Stairs / Ladders / Hand Rails

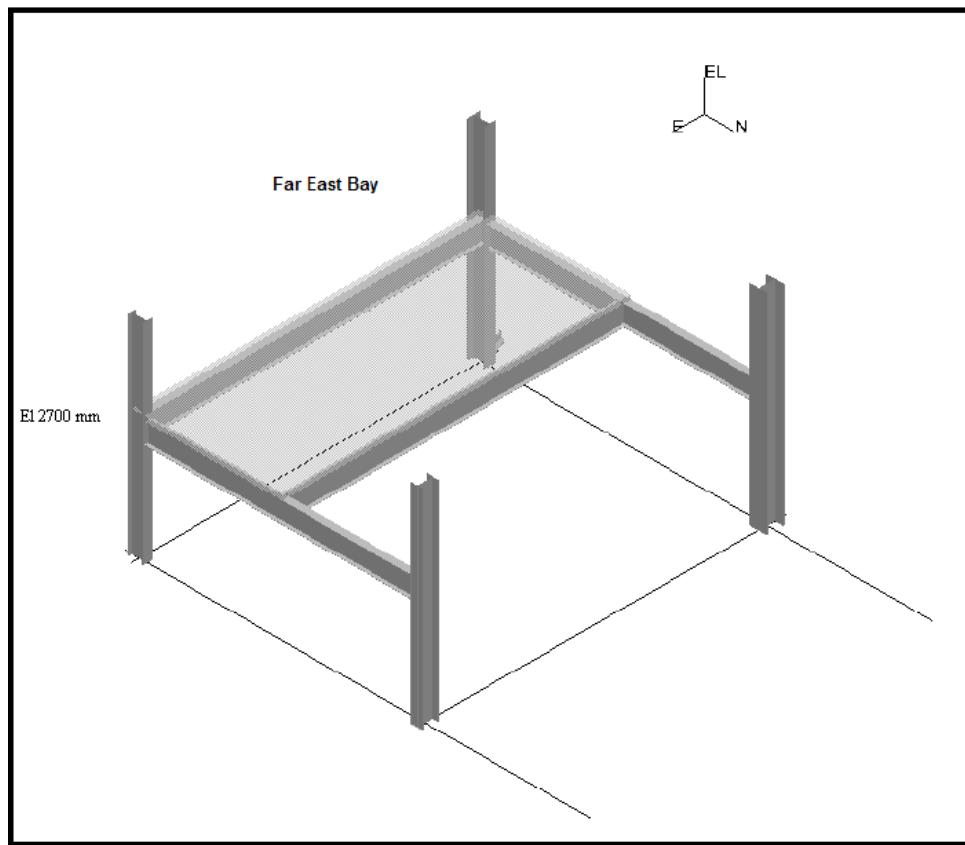
Objective

After completing this lab, you will be able to place stairs, ladders and hand rails using different methods.

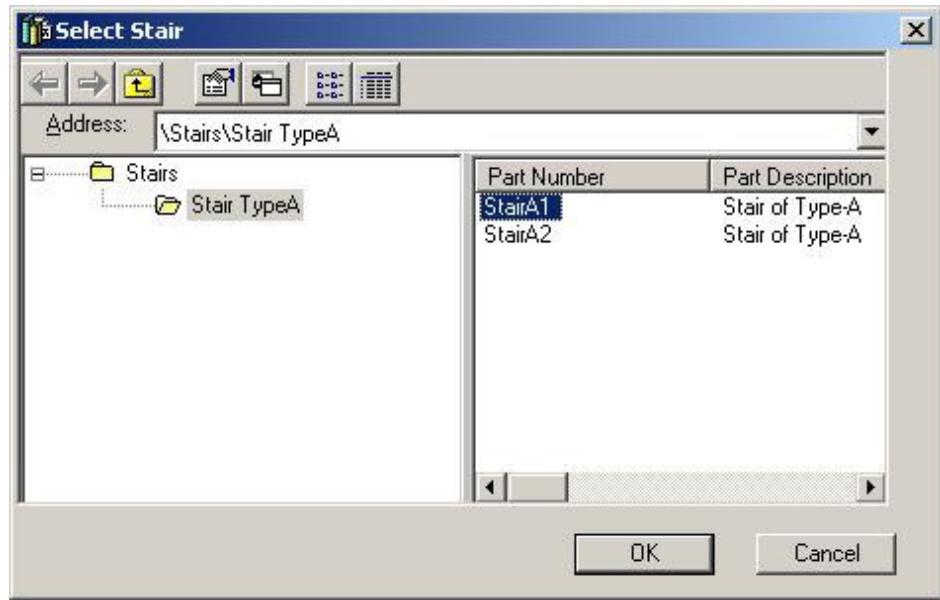
The labs use the U03 structure to place each element type and then the predefined Unit 1 structure to place more hand railing. In both cases set the Permission Group to Structural.

Part I: Place First Stair

- 1 Use Clip by Object to isolate the beams and columns for the FAR-East bay as shown below.



- 2 Select Place Stair Command on the vertical toolbar
- 3 Select stair A1 from the catalog browser dialog box



- 4 Select the top edge of the beam. This defines the top elevation of the stair
- 5 Select the elevation plane at 0m for the stair. This plane defines the bottom elevation of the stair
- 6 Select the reference edge for the stair. See Figure 1
- 7 Using the following parameters define the location of the stair along the top edge.

Width:	450mm
Angle:	32 deg
Pitch:	250mm
Horizontal Offset:	3000mm
Vertical Offset:	0mm
System	A2 -> U03 -> Structural -> Miscellaneous
- 8 Toggle Side button if necessary
- 9 Click Finish Button.

An ISO View of the area should now resemble the following graphic:

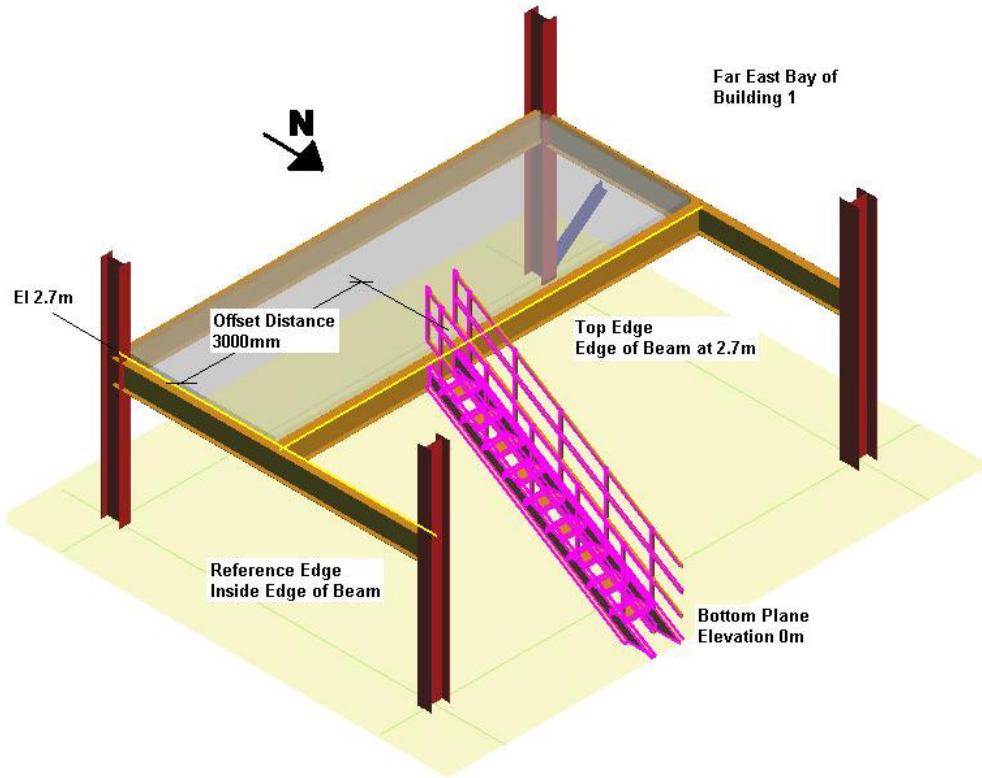
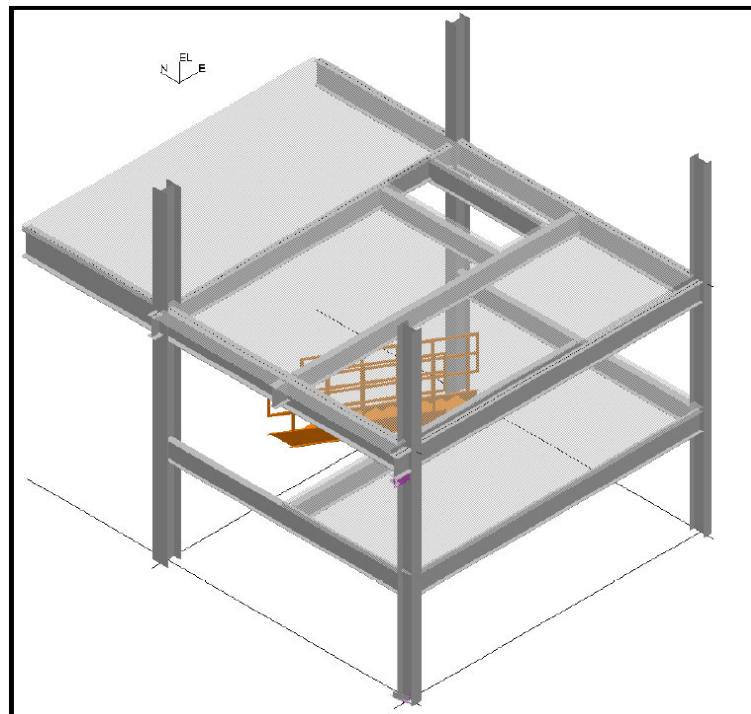


Figure 1 – ISO View of First Stair in Building 1

Part II: Place the Second Stair

- 1 Use the Clip by Volume command to adjust the clipping volume as shown below.



- 2 Select Place Stair Command on the vertical toolbar
- 3 Select stair A1 from the catalog browser dialog box
- 4 Select the top edge of the beam. This defines the top elevation of the stair
- 5 Select the top surface of the slab for the stair. This plane defines the bottom elevation of the stair
- 6 Select the reference edge for the stair. See Figure 2.
- 7 Using the following parameters to define the location of the stair along the top edge.

Width:	450mm
Angle:	40 deg.
Pitch:	250mm
Horizontal Offset:	1400mm
Vertical Offset:	0m
System	A02 -> U03 -> Structural -> Miscellaneous.
- 8 Click Finish button. An ISO View should resemble the following graphic:

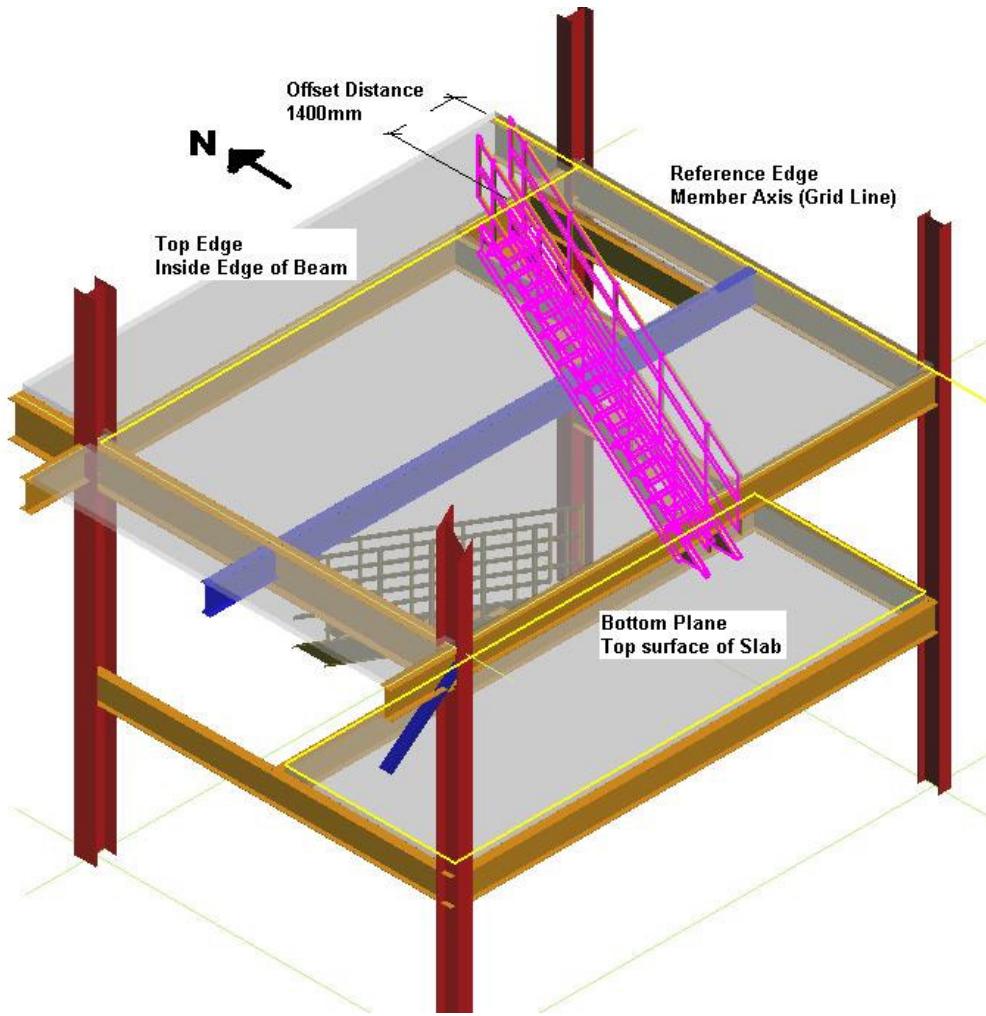
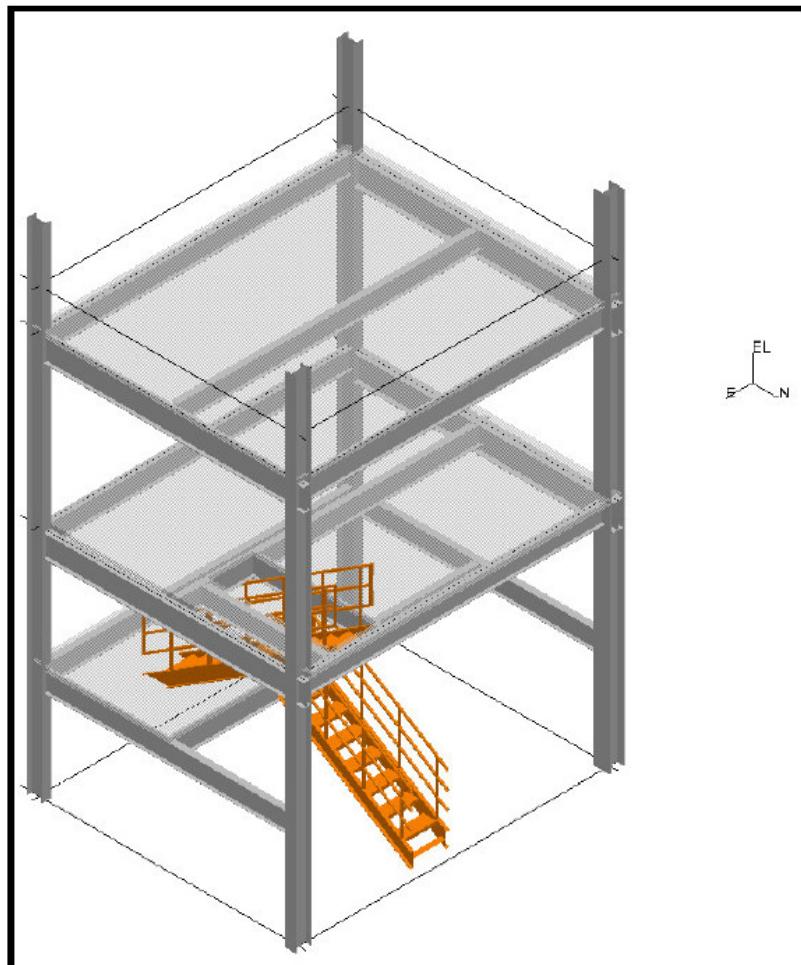


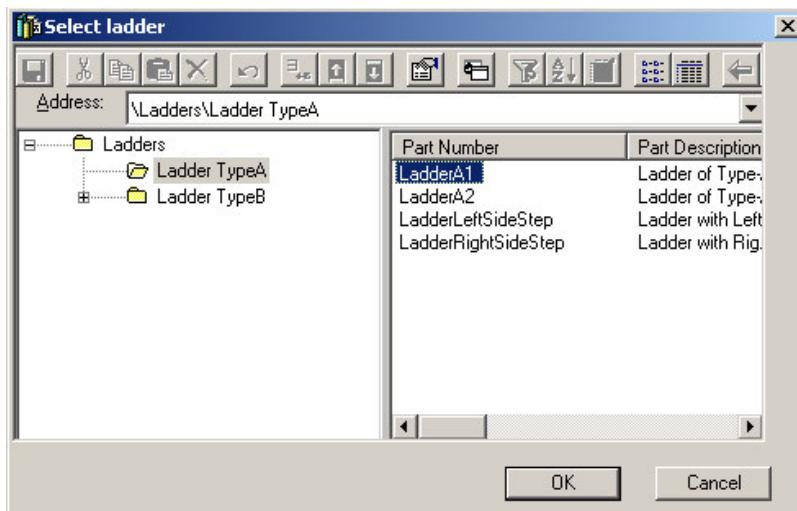
Figure 2 – ISO View of Second Stair in Building 1

Part III: Place a Ladder

- 1 Adjust the clipped volume and orientation 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
- 5 Select the top surface of the slab located at Elevation 6m. This plane defines the bottom elevation of the ladder.
- 6 Select the reference edge for the ladder. See Figures 3 and 4.
- 7 Using the following parameters to define the location of the ladder along the top edge.

Width:	600mm
Angle:	90.0 deg.
Pitch:	300mm
Horizontal Offset:	1500mm
Vertical Offset:	0m
System	A2 -> U03 -> Structural -> Miscellaneous
- 8 Click Finish button.

North and ISO Views should resemble the following graphics:

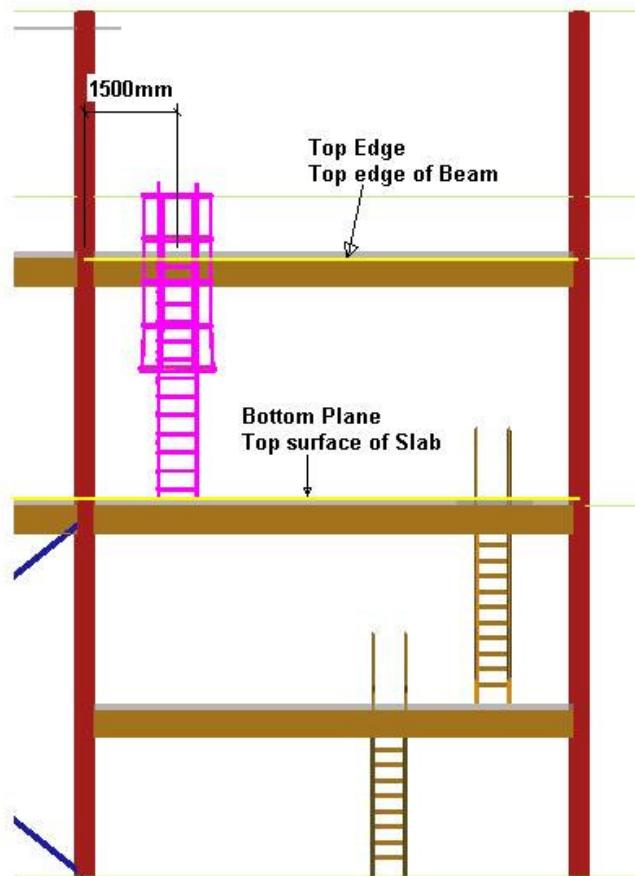


Figure 3 – North View of Building 1 East Bay

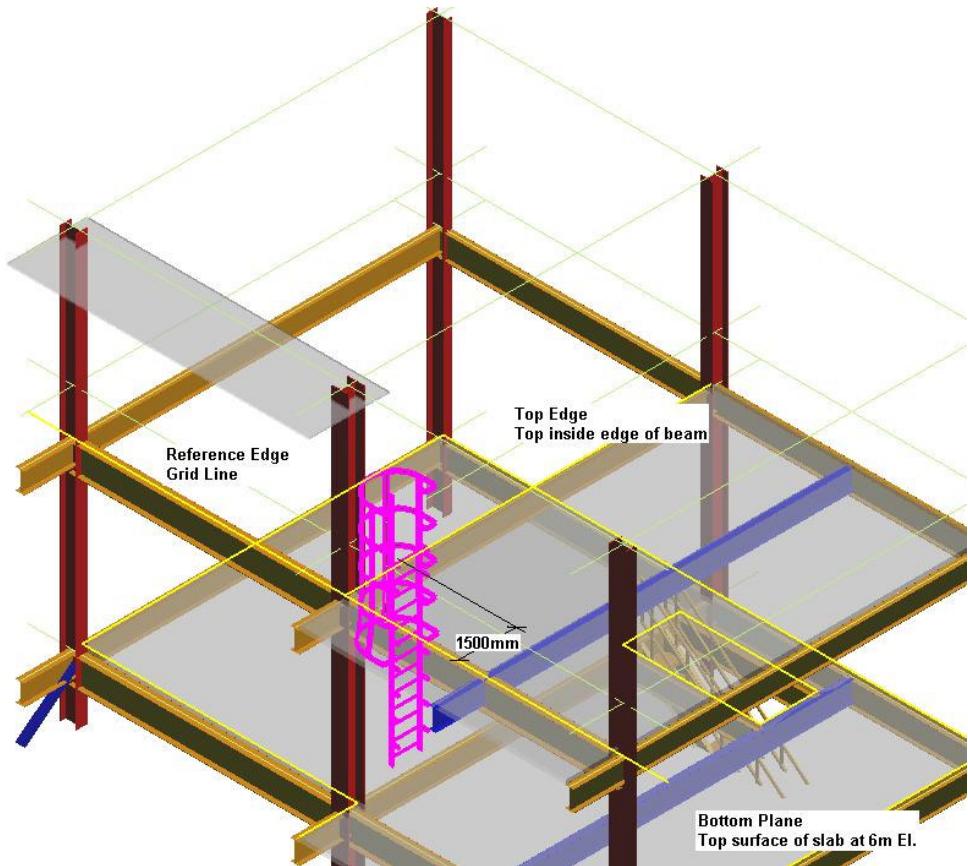
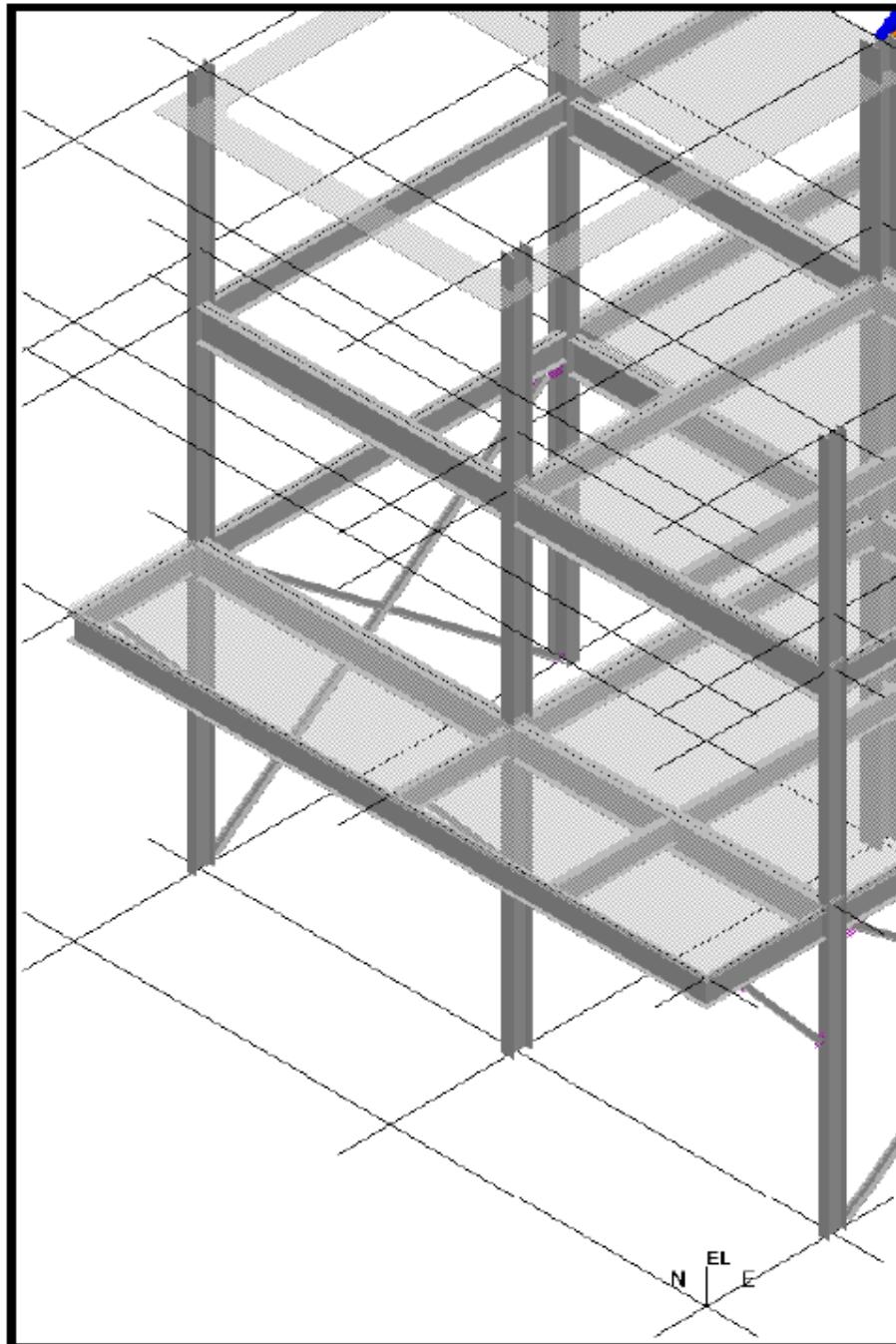


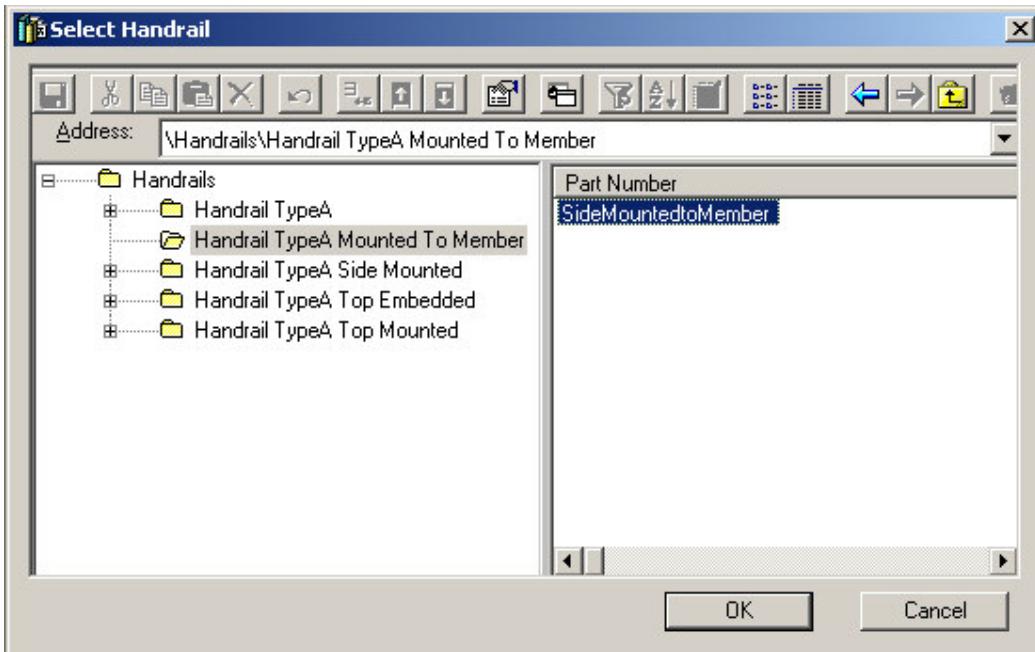
Figure 4 - ISO View of Building 1

Part IV: Placing Hand Rail

- 1 Clear the clipping and zoom to the area on the East of the structure as shown below.



- 2 Select Place Handrail by Member Command on the vertical toolbar
- 3 Select SideMountedtoMember from the catalog browser dialog box

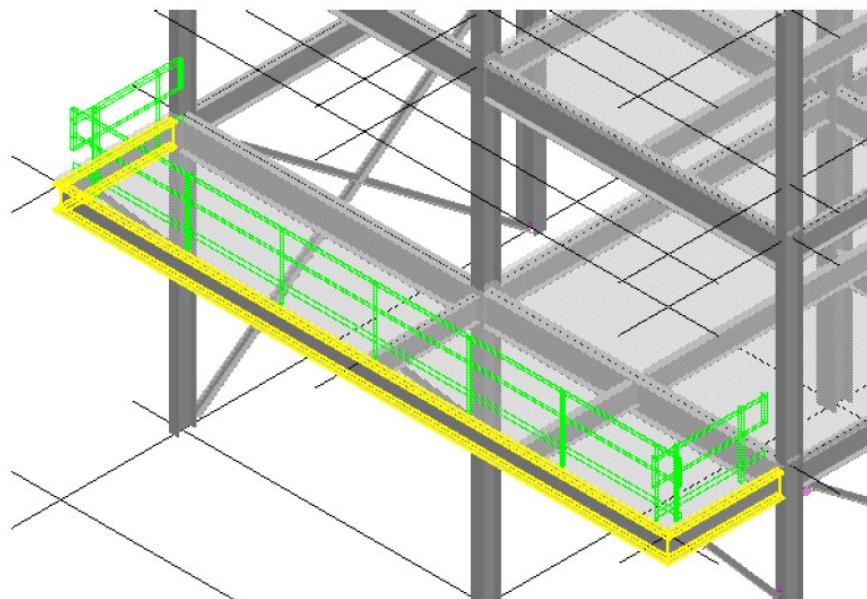


4 Set the following parameters to define the handrail representation.

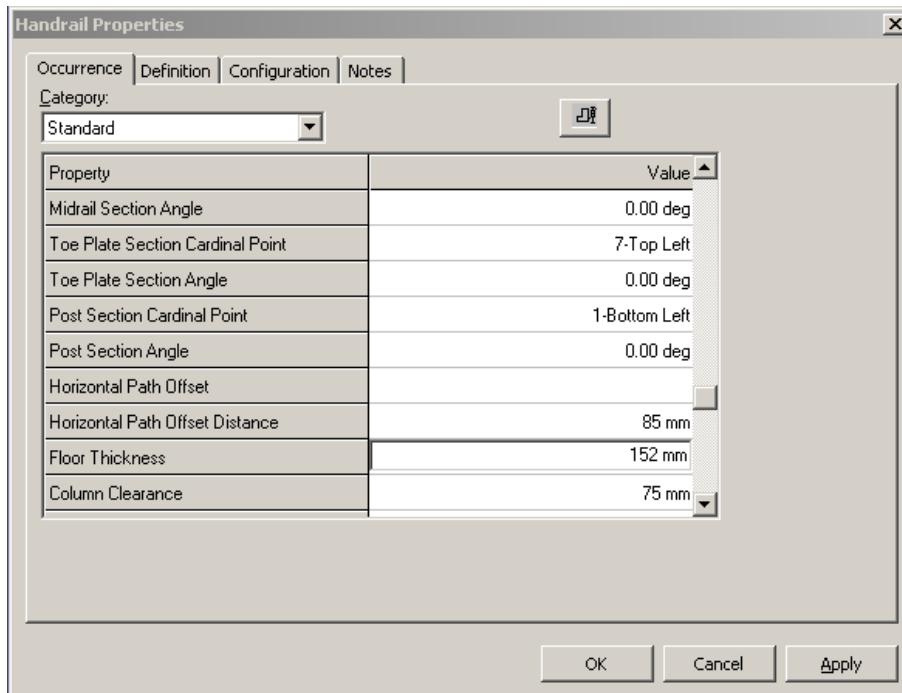
System:	A2 -> U03 -> Structural -> Miscellaneous
Begin Treatment:	Rectangular
End Treatment:	Rectangular
Offset Reference:	Centerline
Column Clearance:	75mm
End Clearance:	0mm



5 Select three beams for the SmartStep 1 and click “Accept” button.

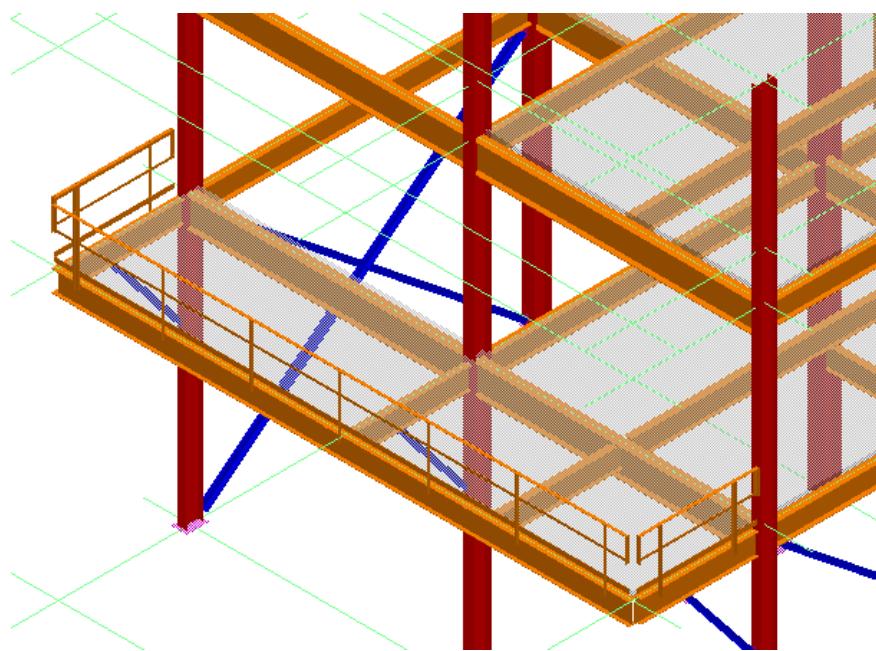


- 6 Select the slab as the walking surface location for SmartStep2.
- 7 Click “Finish”.
- 8 Click “Select Command” on the vertical toolbar and select the handrails just placed.
- 9 Open the properties page and change the Horizontal Path Offset Distance to 85mm.



- 10 Click “OK” button.

Your view should now resemble the following graphic.



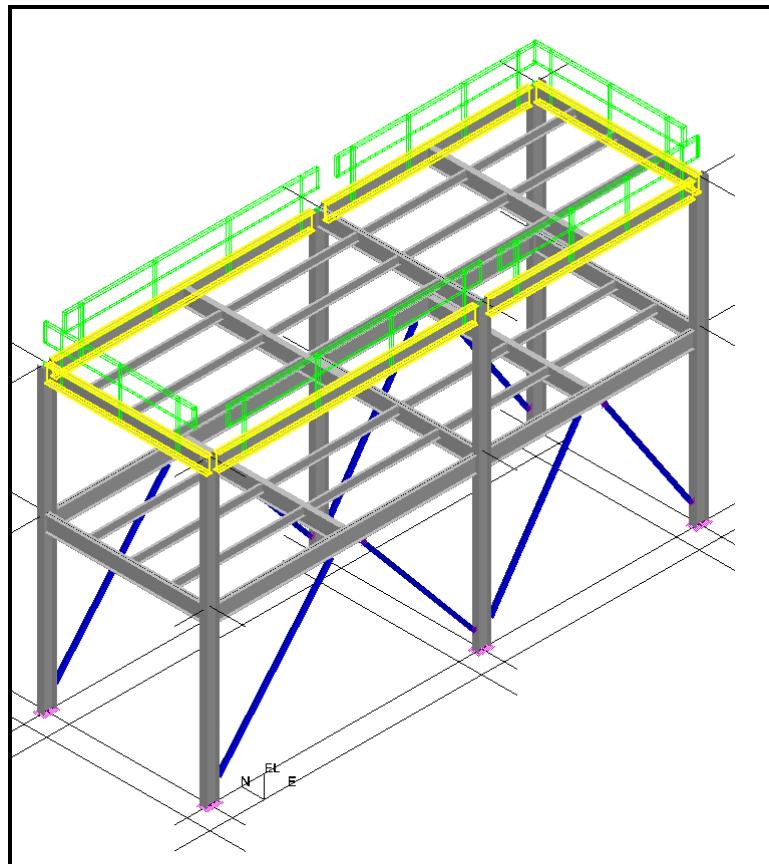
Part V: Placing Hand Rail in Structure U02

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 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:	75mm
End Clearance:	0mm

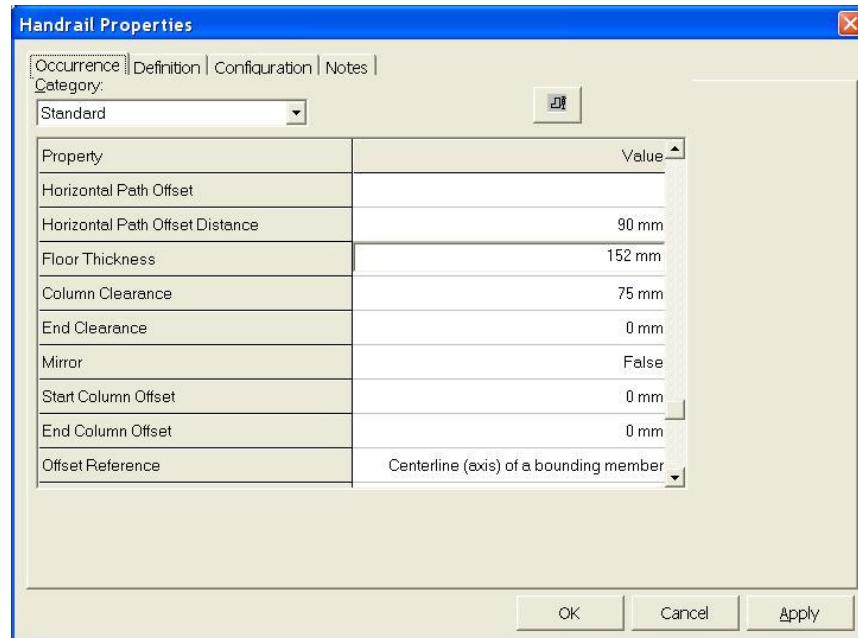


5. Select six beams for the SmartStep1 and click “Accept” button.



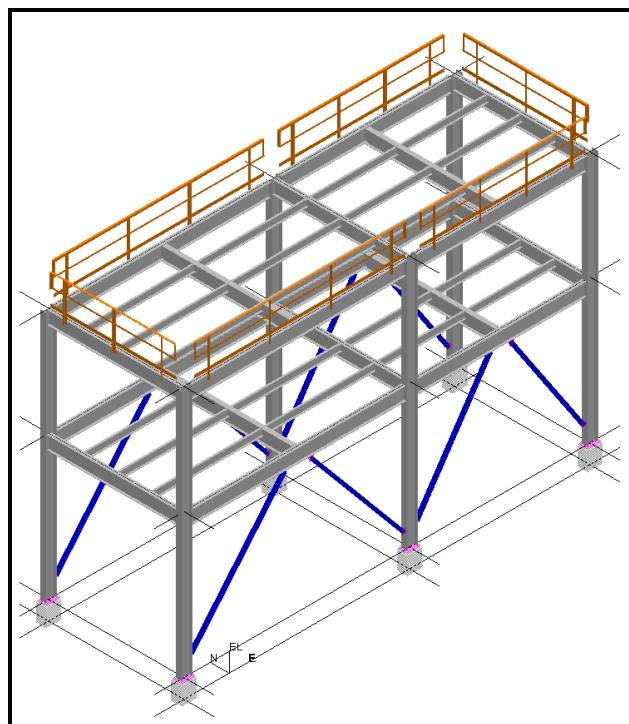
6. Select an interior beam or slab as the walking surface location for SmartStep2.
7. Click “Finish” button.
8. Click “Select command” on the vertical toolbar and select the handrails you have just placed.

9. Open the properties page and change the Horizontal Path Offset Distance to 90mm.



10. Click “OK” button.

Your View should now resemble the following graphic:



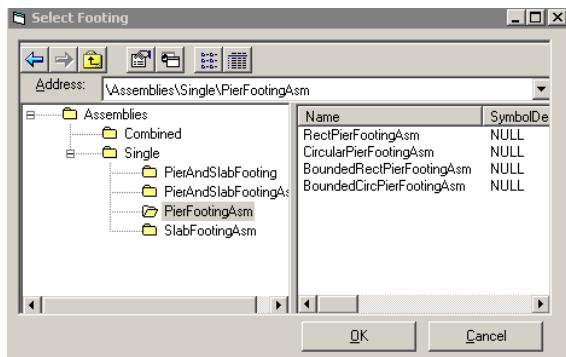
Lab 11 - Footings

Objective

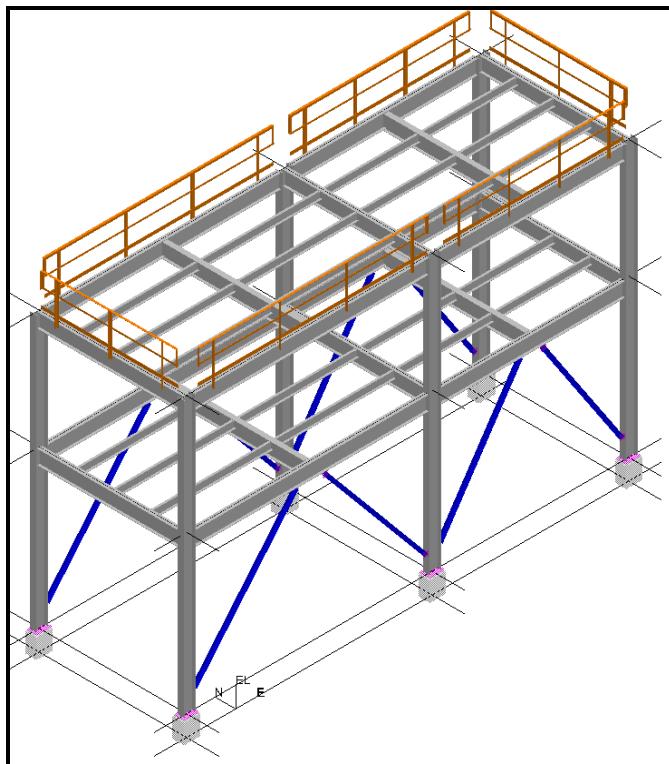
After completing this lab, you will be able to:

Use the Place Footing Command

1. Select Place Footing Command on the vertical toolbar.
2. Navigate the tree and select RectPierFootingAsm from the catalog browser dialog box.



3. Hit OK button to close the catalog browser dialog box.
4. Select all columns in U02 and then select the “Accept” button
5. Go to the System combo control and select A2 -> U02 -> Structural -> Footings.
6. Hit the Finish button to place all footings in the model



Lab 12 - Equipment Foundations

Objective

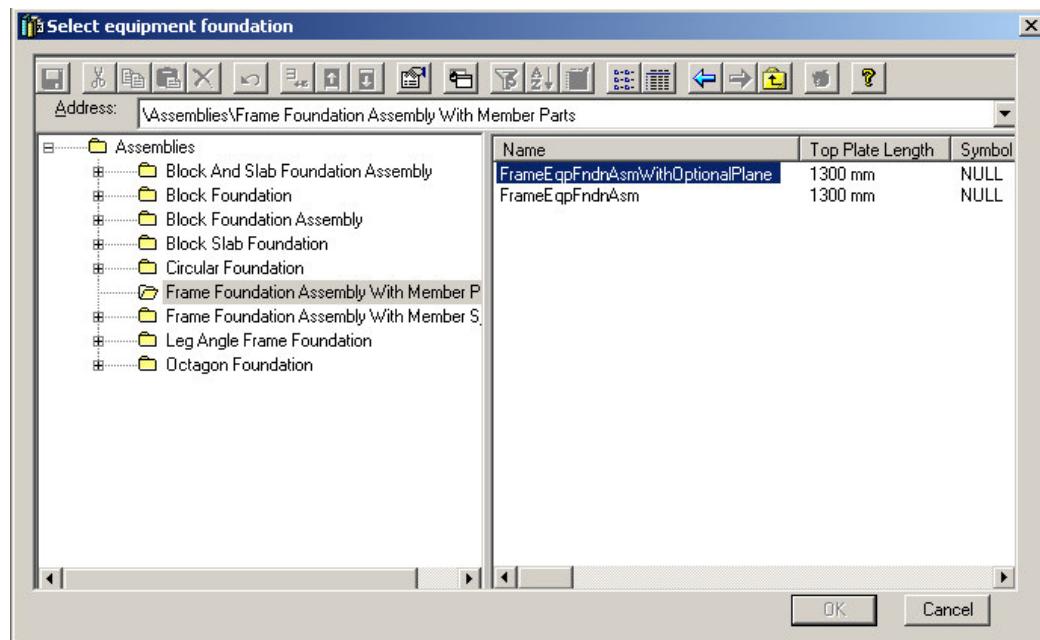
After completing this lab, you will be able to:

- Understand the footing entities and relationships
- Able to Place different type of foundations

Make sure the Active Permission Group is set to **Equipment**.

Part I: Placing Individual Foundations

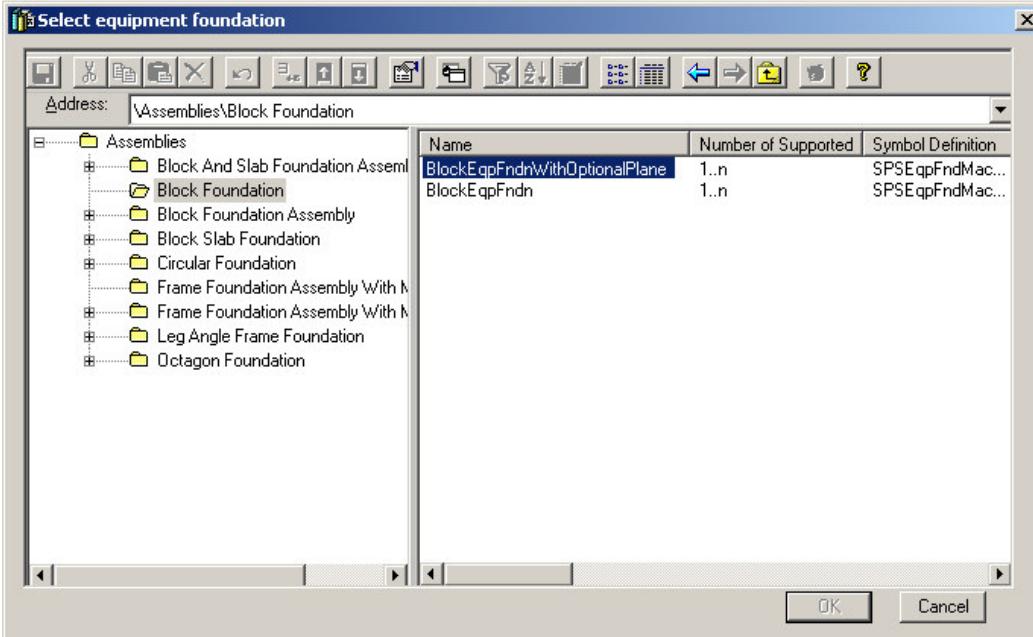
1. Set your workspace to show A -> U01 and U01 CS
2. Select Place Equipment Foundation Command on the vertical toolbar.
3. Select Pump-001
4. Ensure By Rule is unselected.
5. Select the Type combo and select More.. to open the catalog browser.
6. Navigate the tree and select FrameEqpFndnAsmWithOptionalPlane from the browser



7. Select OK to close the browser
8. Select the grade slab for the support plane
9. From the System combo on the ribbon bar select the A2 -> U01 -> Structure -> Miscellaneous system.
10. Select the Finish button to commit the assembly to the model database.

11. Select Pump-002

12. Via the Type combo select the Assembly BlockEqpFndnAsmWithOptionalPlane.

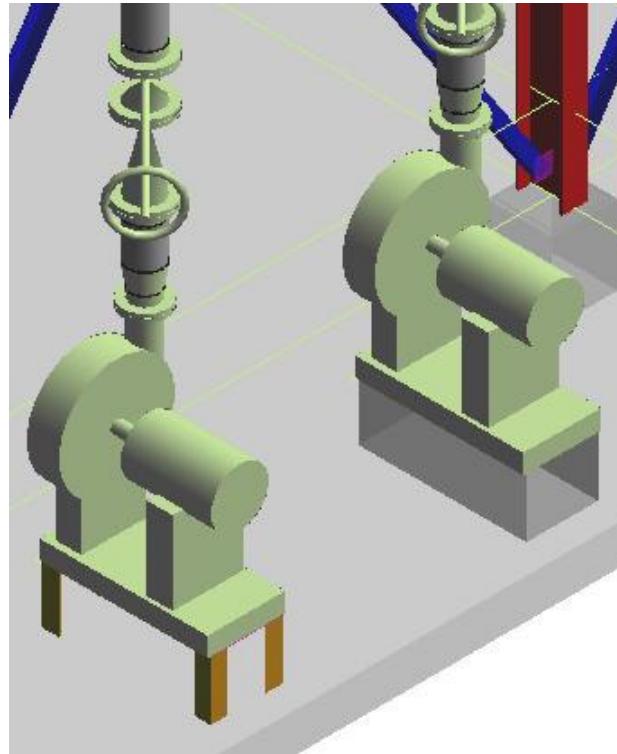


14. Green check the equipment

15. Select the slab on grade for the support plane

16. Select Finish to place the foundation

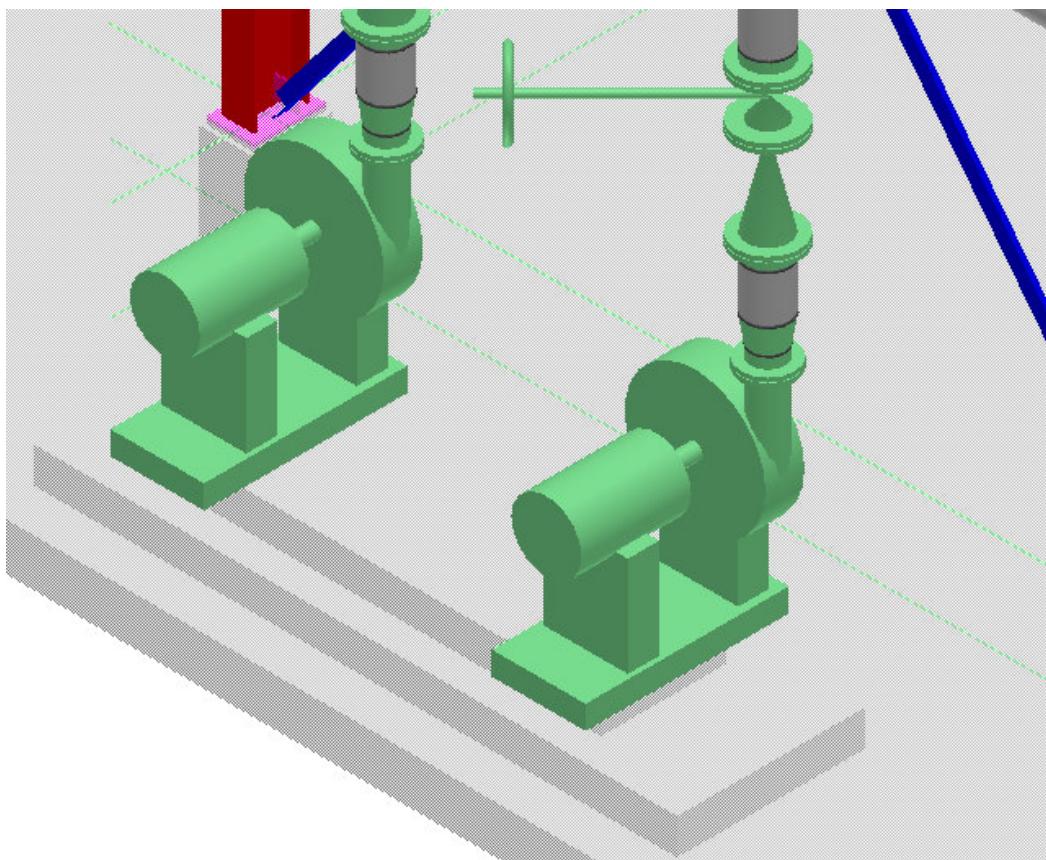
An ISO View should resemble the following graphic:



Part II: Placing Single Foundation on Multiple Pumps

- 1 Delete the foundations placed in Part I
- 2 Select Place Equipment Foundation Command on the vertical toolbar.
- 3 Ensure the By Rule Option is checked.
- 4 Select Pump-001 and Pump-002 in your workspace explorer or graphically.
- 5 Select the Green check button – this highlights the foundation ports.
- 6 Uncheck the By Rule Option and select
BlockandSlabEqpFndnAsmWithOptionalPlane
- 7 Select the Slab on Grade for the support plane
- 8 Go to the System combo control and select A2 -> U01 -> Structure ->
Miscellaneous.
- 9 Select the Finish button to place the equipment foundation in the model

An ISO View should now resemble the following graphic.



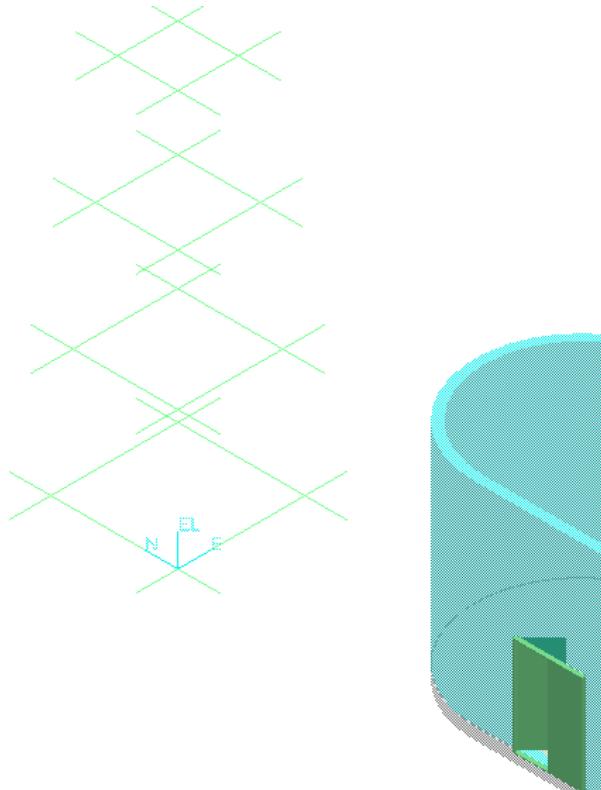
Lab 13 - Designed Solids

Objective

After completing this lab, you will be able to:

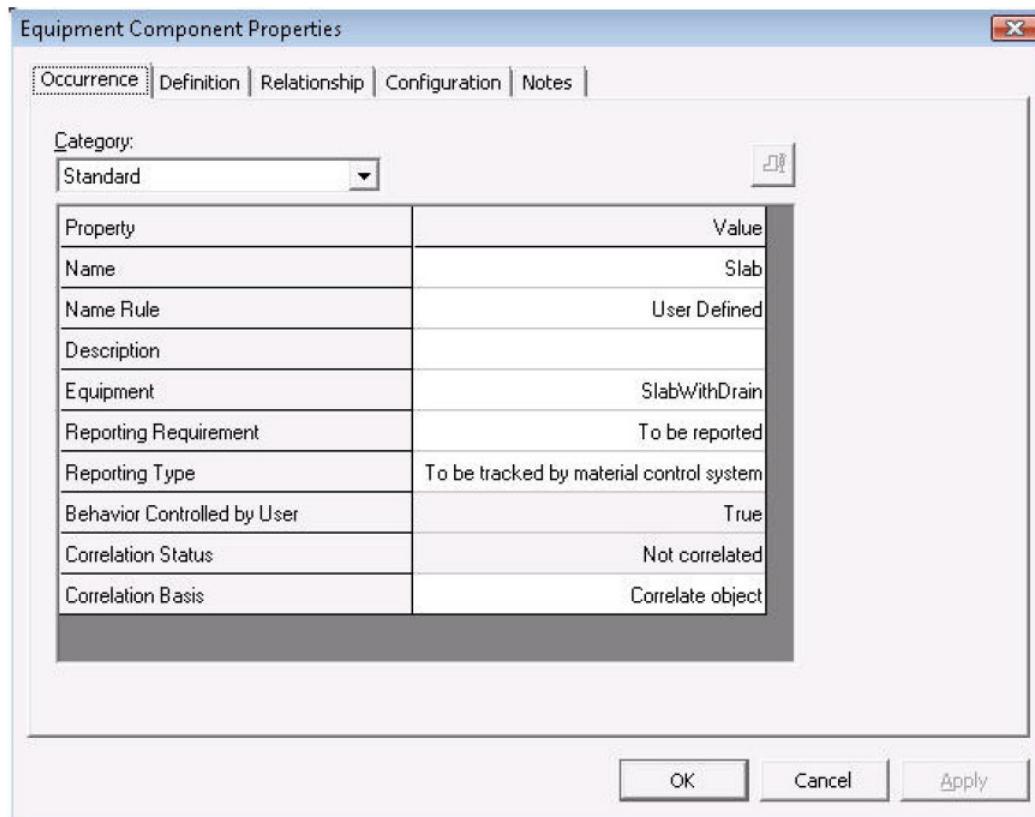
Use Connect relationships to position Shapes, rotate Shapes with the Arrow keys on the keyboard. For this lab, we will create a hierarchy using Designed Equipment Components. Understand and apply relationship. Use Boolean operations (Add, Subtract and Suppress) effectively and understand Order Shapes option.

1. Open a session file.
2. Re-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  in the ribbon bar 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 U05 CS 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*. Press OK.

5. In the Design Equipment Properties dialog, choose **Solids** as the System and give a User Defined name of *SlabWithDrain*. Press OK.
6. Place the Designed Equipment at E: -18000mm, N: --6000mm and EL: 0mm.
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 any Equipment Component. 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\ProcessComponents\Attachments\Vessel Platform\Vessel Platform and press OK.
10. In the Designed Equipment Component Properties dialog, make sure the Equipment property is set to *SlabWithDrain*. Name the Component *Slab* and press OK. See image below.

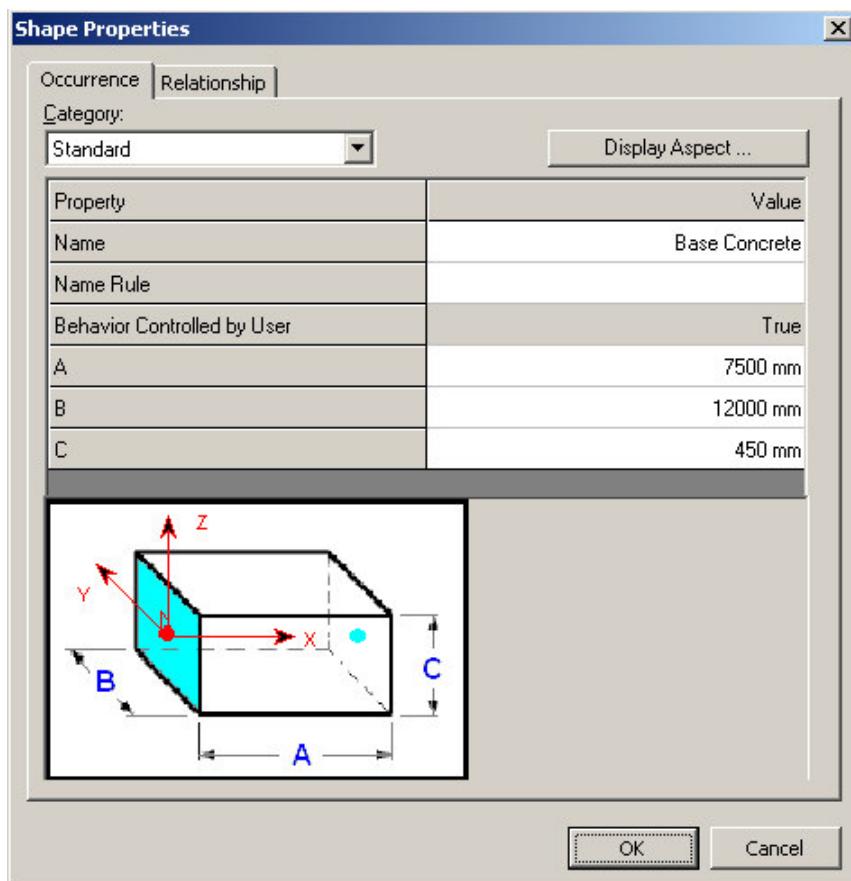


11. Place *Slab* at E: -18000mm, N: -6000mm and EL: 0mm

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 Name and Grade to Concrete and Fc 3000 for Solids, *SlopedSlab*. Your workspace explorer should resemble the following graphic.

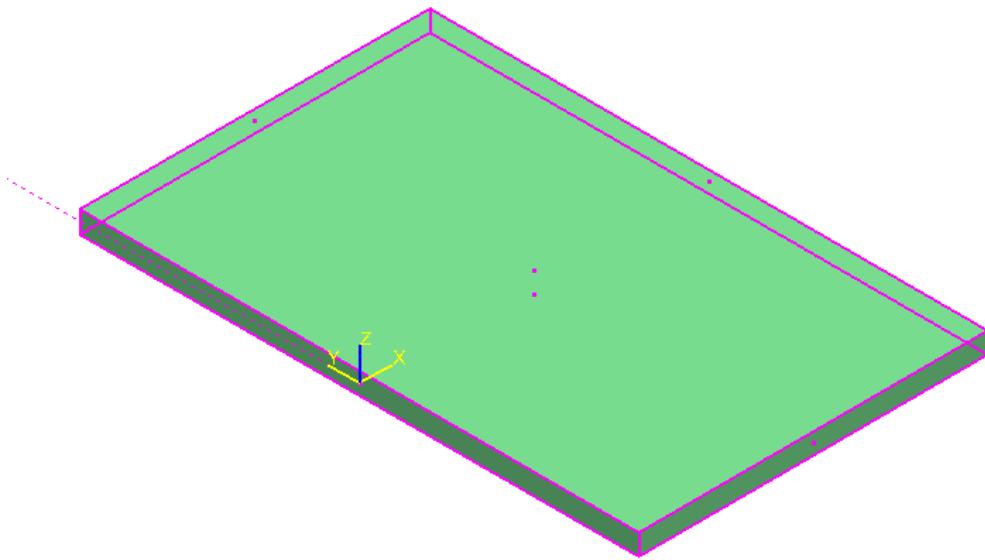


15. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid* from the Shapes dialog. Choose *SlopedSlab* as the parent of the Shape.
16. In the Shape Properties dialog, set A to 7500mm, B to 12000mm, and C to 450mm. Name the *RectangularSolid* as *Base Concrete*. Press OK.



17. Place the Shape at E: -18000mm, N: -12000mm and EL: 0mm.

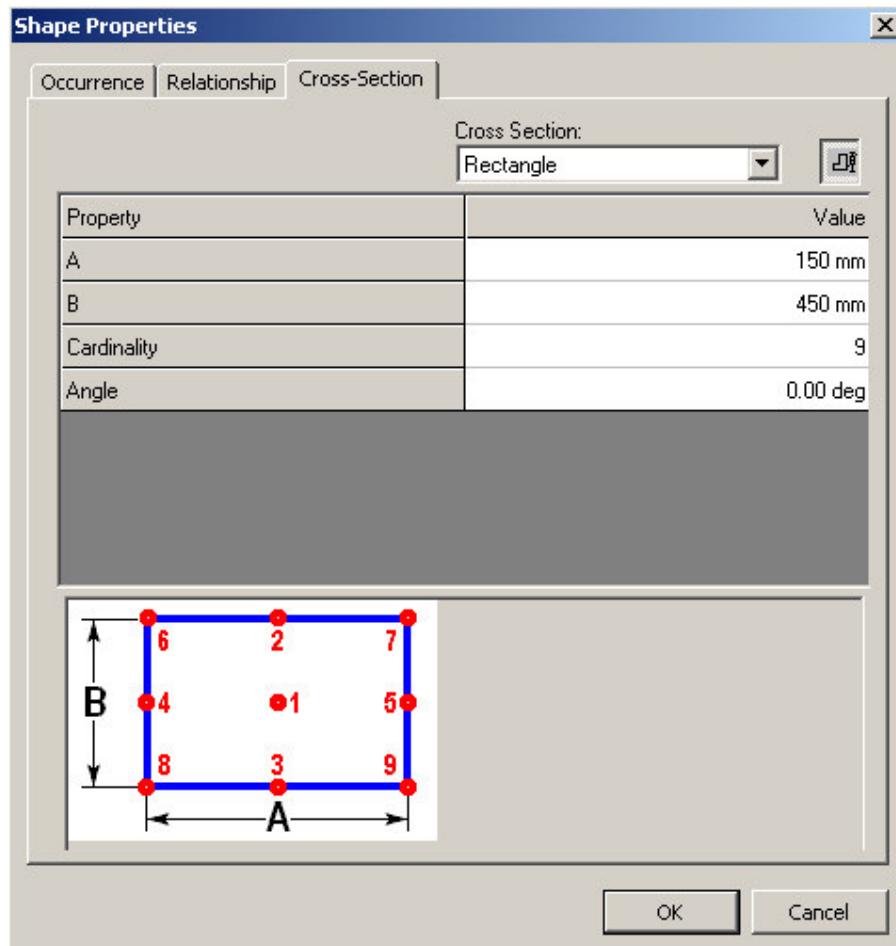
18. Your view should resemble the following graphic.



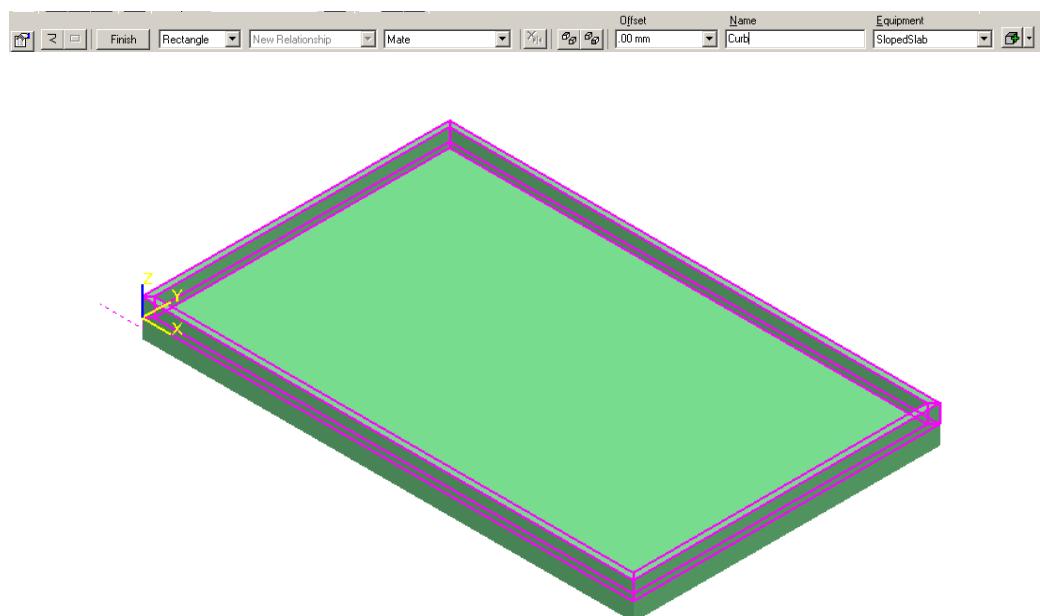
19. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *Prismatic Shape*  from the Shapes dialog. Choose *SlopedSlab* as the parent of the Shape.
20. Choose *SlopedSlab*. Shape properties window will show up. Select the Cross-Section tab. See the image below.



21. Choose Rectangle from the Cross Section pull down menu and enter the value for A = 150mm and B = 450mm. Enter cardinality = 9. Press OK. Your view should resemble the following graphic.



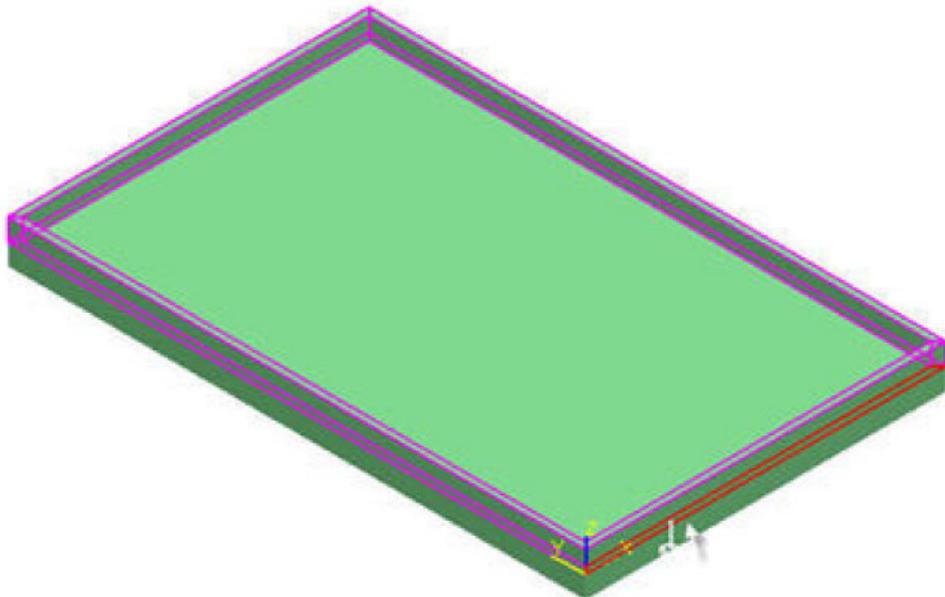
22. Select the top edges corners of the *Base Concrete* in an anticlockwise direction for the path. Name the *Prismatic Shape* as “*Curb*” and hit the Finish. Your view should now resemble the following graphic. Hit Finish.



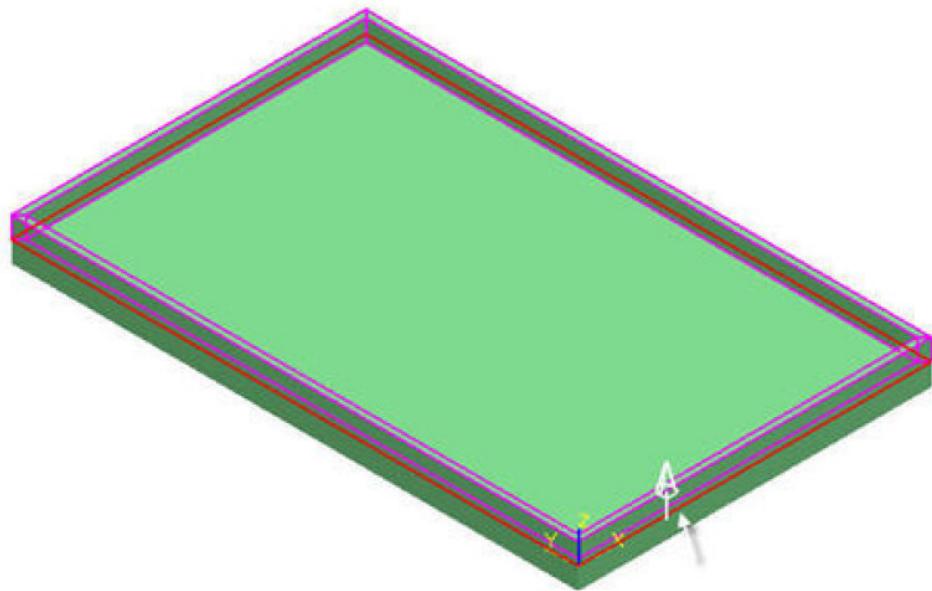
23. Select *Curb* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Mate” as the new relationship.



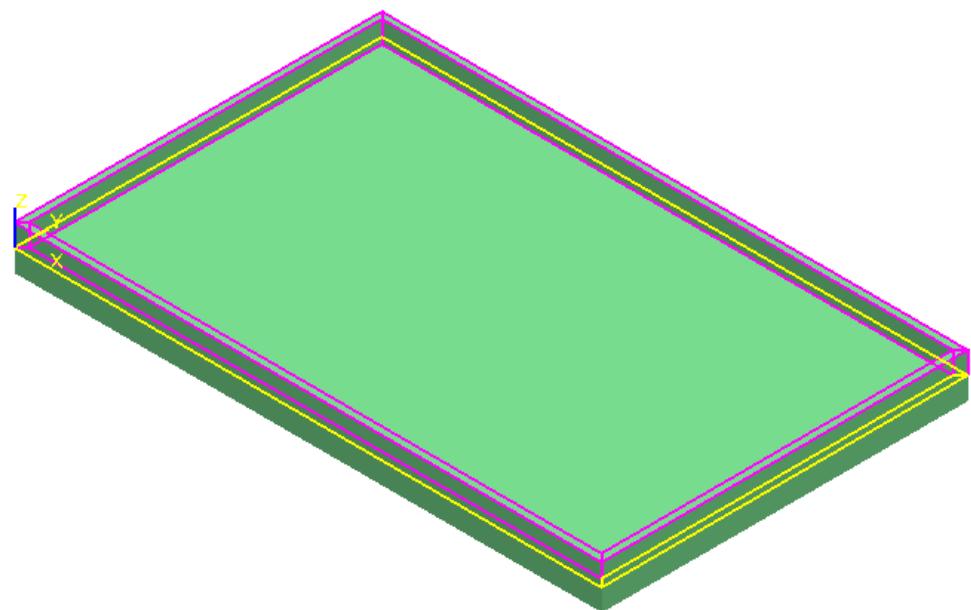
24. Select the plane as shown in the following graphic.



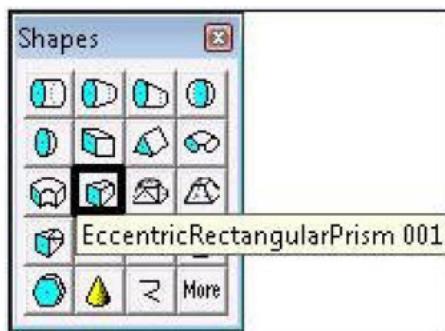
25. Select the top face of *Base Concrete* and your view should resemble as shown in the graphic below,



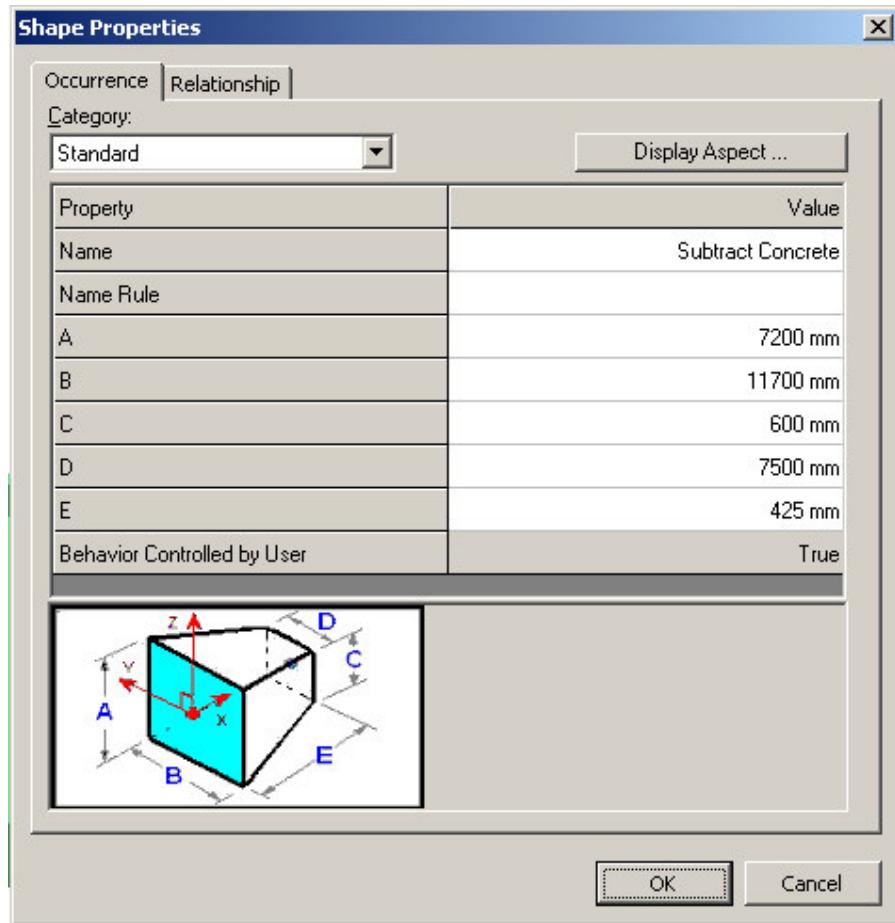
26. Your graphic should resemble as shown below.



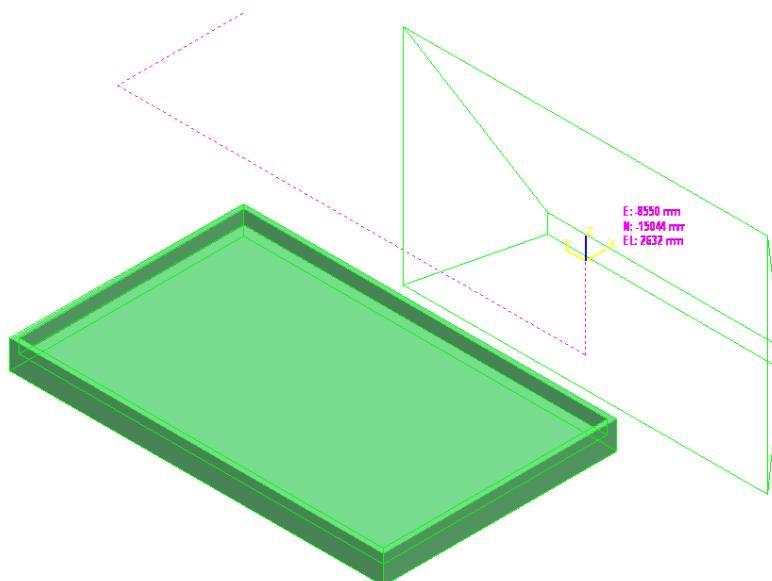
27. Select the EccentricRectangularPrism 001 from the shapes dialog box as shown below in the image.



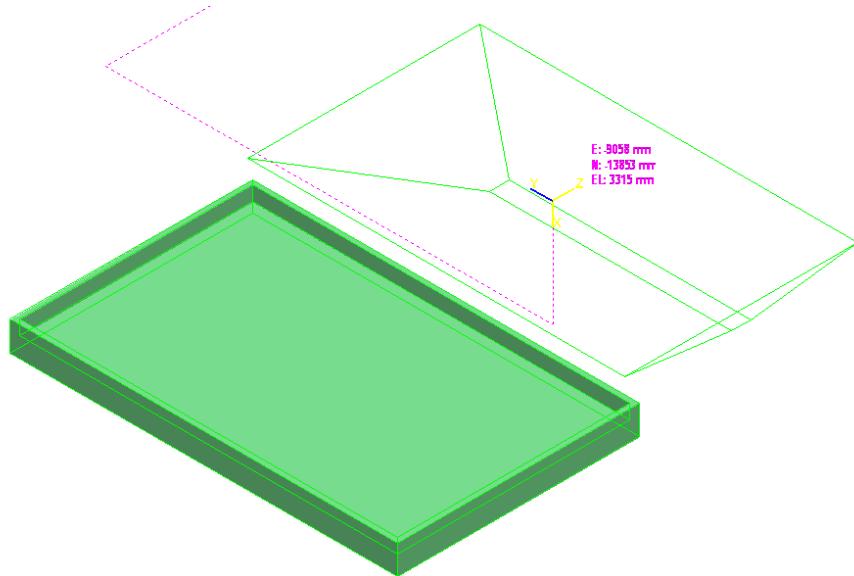
28. Choose *SlopedSlab* as the parent of the Shape.



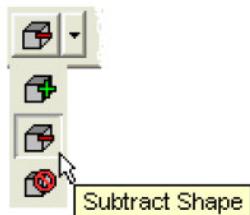
29. In the Shape Properties dialog, set A to 7200mm, B to 11700mm, C to 600mm, and D to 7500mm and E to 425mm. Name the EccentricRectangularPrism 001 as “*Subtract Concrete*”. Press OK.
30. Your view should now resemble the following graphic



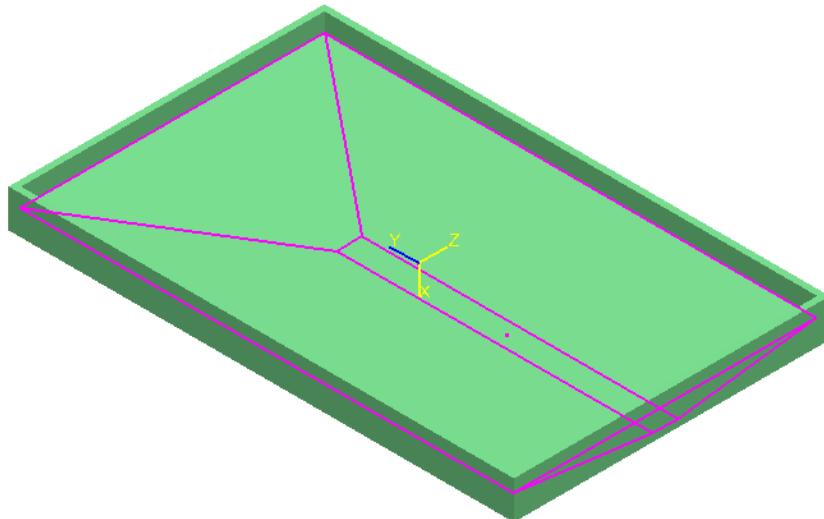
31. We need to rotate the Shape to place it correctly. This can be done before placement time using the Arrow keys on the keyboard.
32. Press Up arrow key once and then press left arrow key once. Your view should resemble the following graphic.



33. Place the Shape at E: -14250mm, N: -12000mm and EL: 225mm.
34. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.



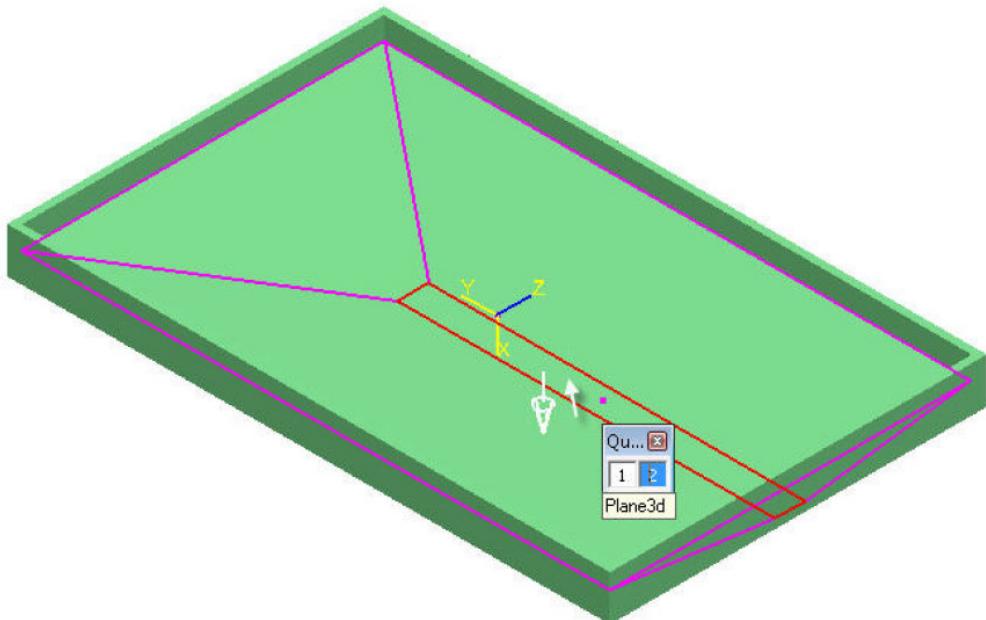
35. Your view should resemble the following graphic.



36. Select *Subtract Concrete* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Mate” as the new relationship and key in -425mm as the offset value.

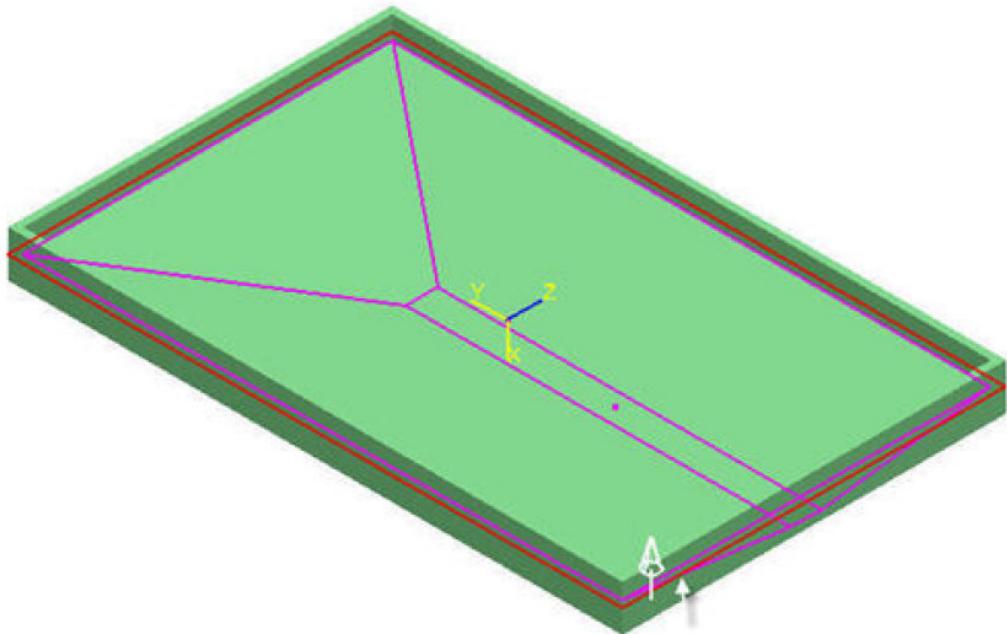


37. Move your mouse over the SlopedSlab and it should highlight *Subtract Concrete* and click the left mouse button once at the location as shown in the figure. Your view should now resemble the following graphic.

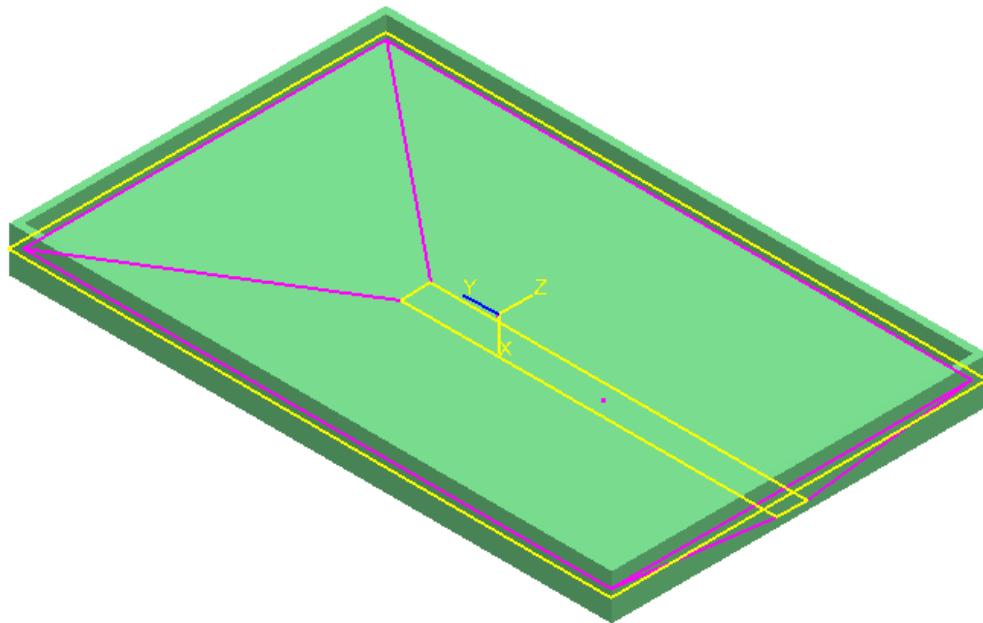


38. Select 2 from the quick pick.

39. Select the *Base Concrete*. Your view should now resemble the following graphic.



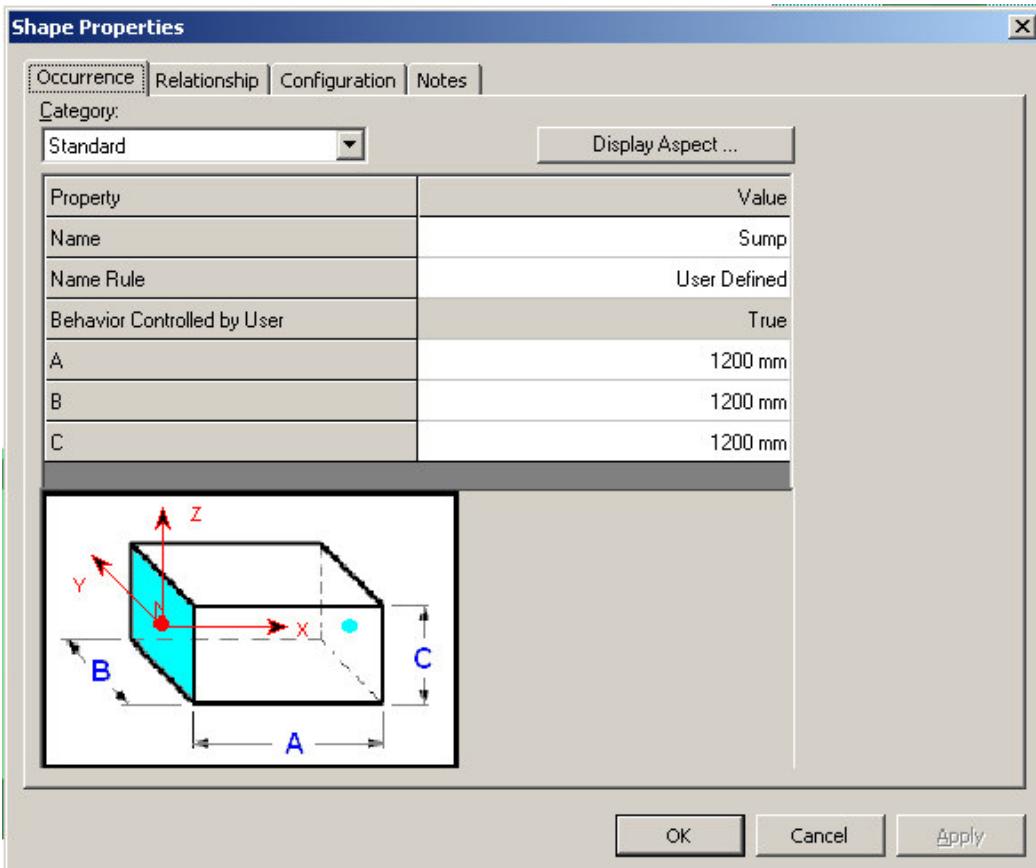
40. After selecting the highlighted plane, Mate Relationship will apply to the *Base Concrete* and *Subtract Concrete*. Your view should resemble the following graphic.



41. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid* from the Shapes dialog.

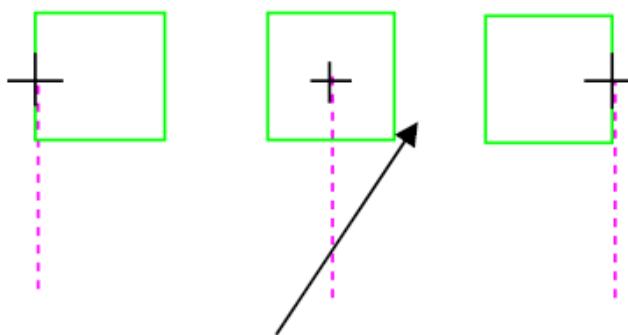
42. Choose *SlopedSlab* as the parent of the Shape.

43. In the Shape Properties dialog, set A to 1200mm, B to 1200mm, and C to 1200mm and enter the name as “*Sump*”. Press OK



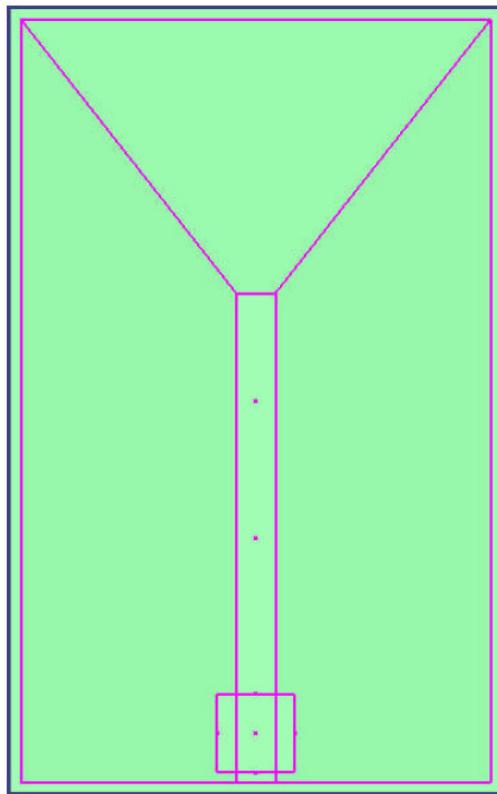
44. While looking in the Plan Plane, use the Up arrow keys (if needed) to change the axis of rotation and use the Left or Right arrow keys to rotate the shape until the point of placement is the center of the rectangle top face.

a. Looking Plan View

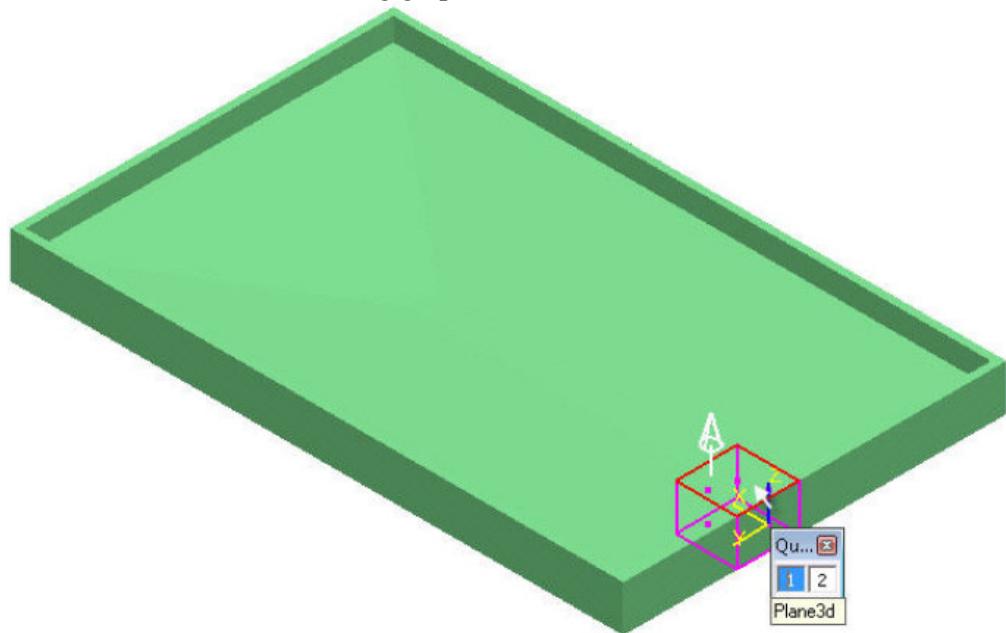


45. Place the Shape at E: -14275mm, N: -17100mm and EL: 650mm.

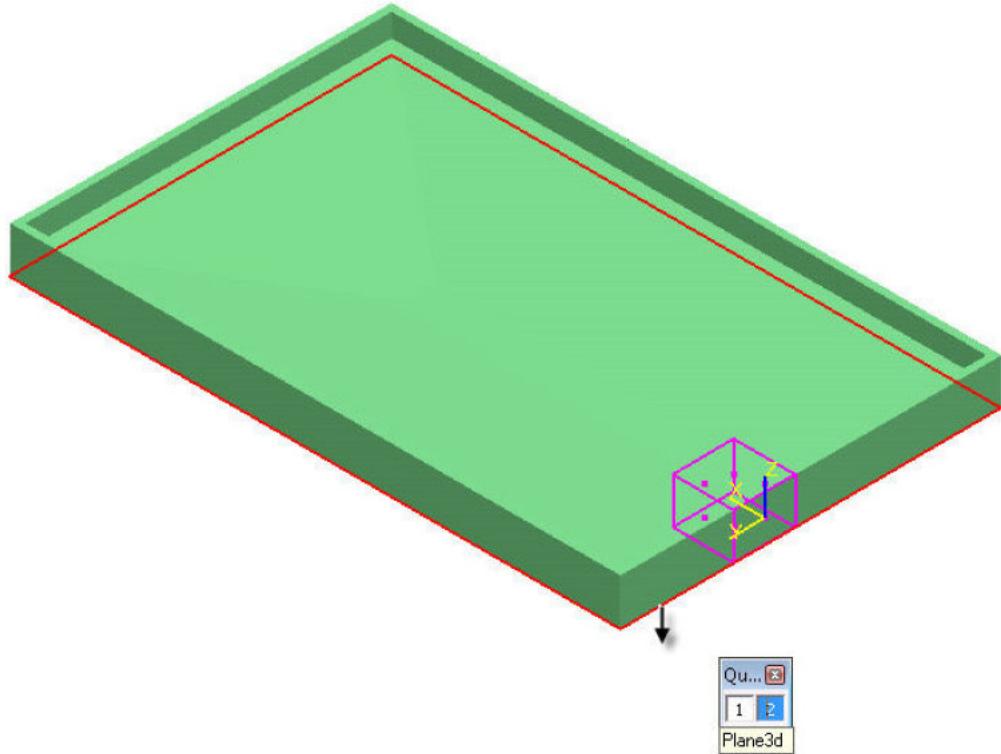
46. Change back to the Looking Plan view. Create a Connect Relationship at the point shown below and click to place the Shape. Your view should resemble the following graphic.



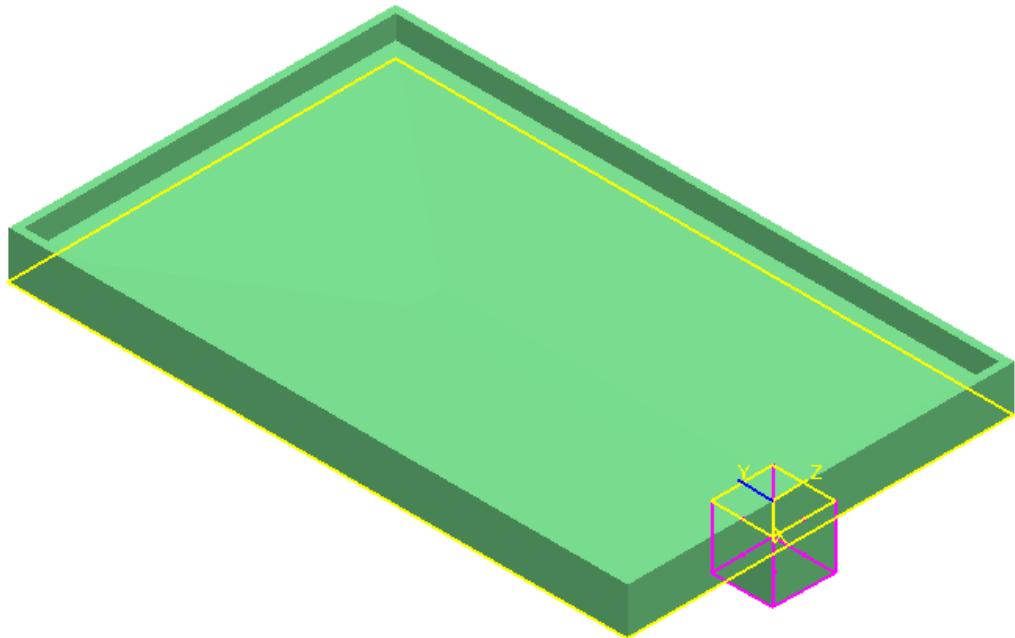
47. Change the view back to isometric. Select the *Sump* and add mate as the new relationship with 0mm offset. Select the top face of the *Sump* and your view should resemble the following graphic.



48. Select the bottom face of the *Base Concrete*. Your view should resemble the following graphic.



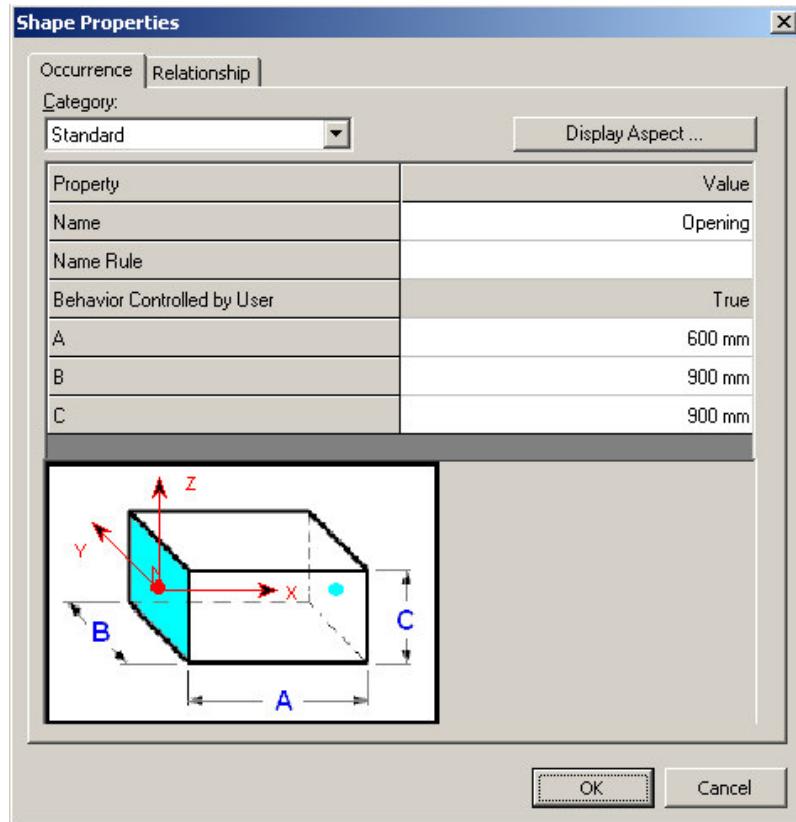
49. Once the mate relationship has been applied, your view should resemble the following graphic.



50. Change back to the Plan Plane view and start the Place Shape command again.

51. Select *RectangularSolid* from the Shapes dialog

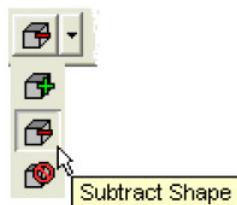
52. Choose *SlopedSlab* as the parent of the Shape and make the following additions as shown in the image below.



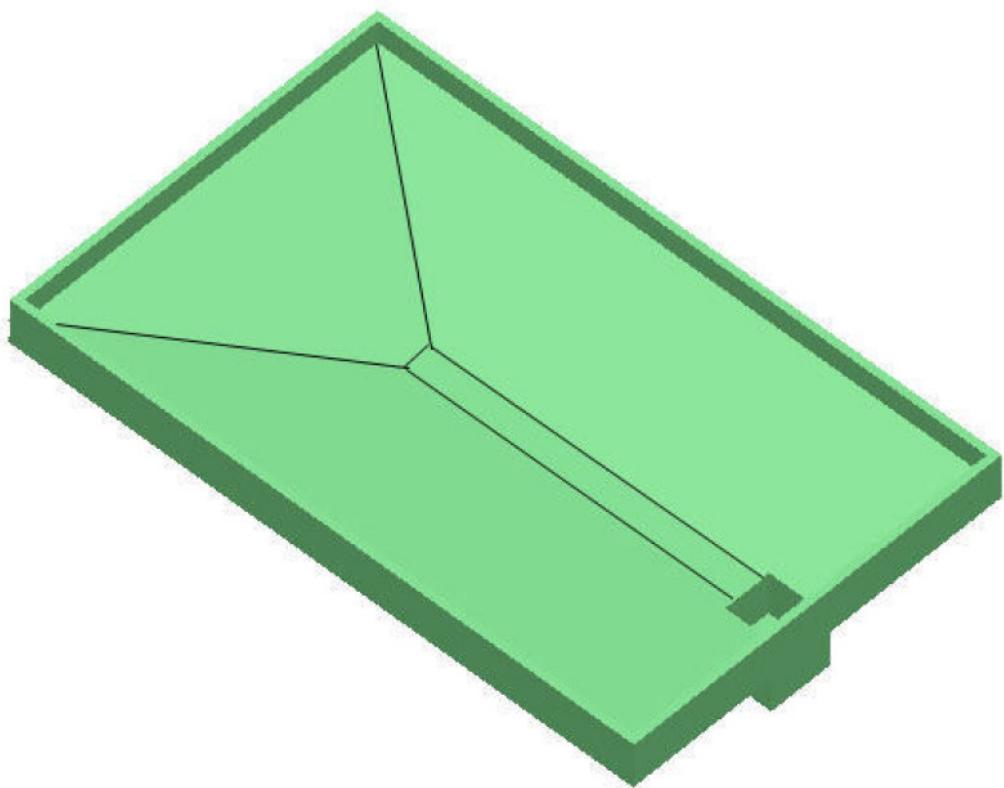
53. Press the up arrow key and then left arrow key to set the placement point in the center top face.

54. Place the Shape at E: -14275mm, N: -17100mm and EL: -800mm.

55. In the ribbon bar, change the *Add Shape* operation to a *Subtract Shape* operation.



56. 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.

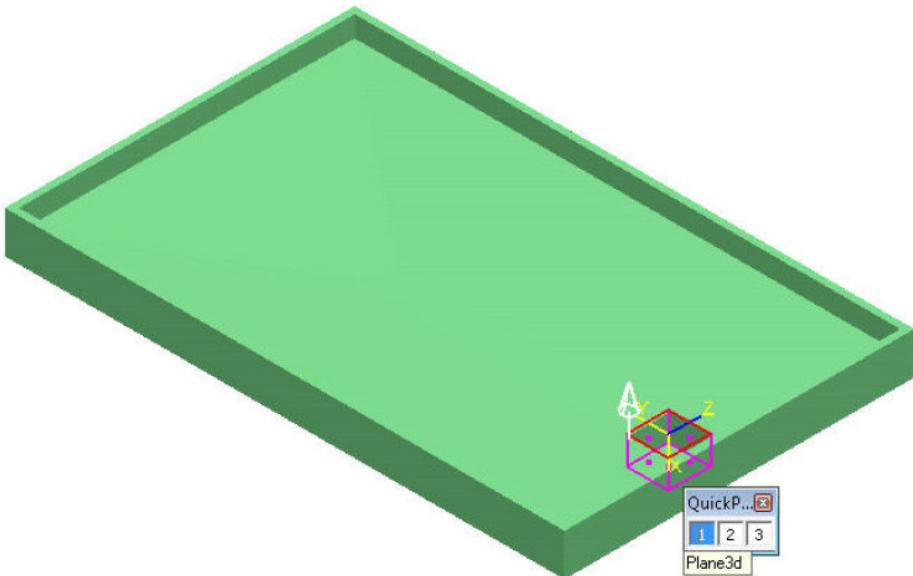


** Lines shown within this image are used only for visual enhancement of the object.

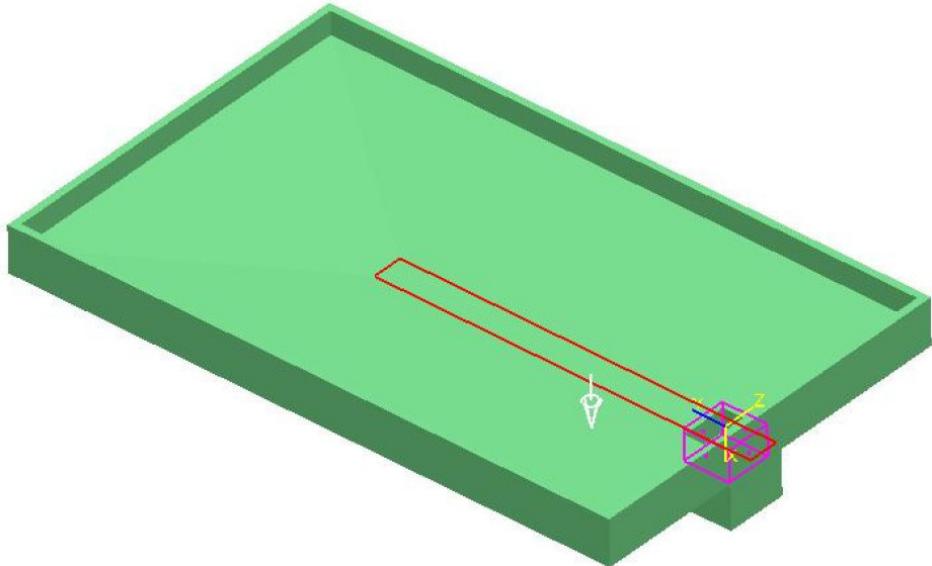
57. Select the opening and in the ribbon bar enter offset value as -125mm.



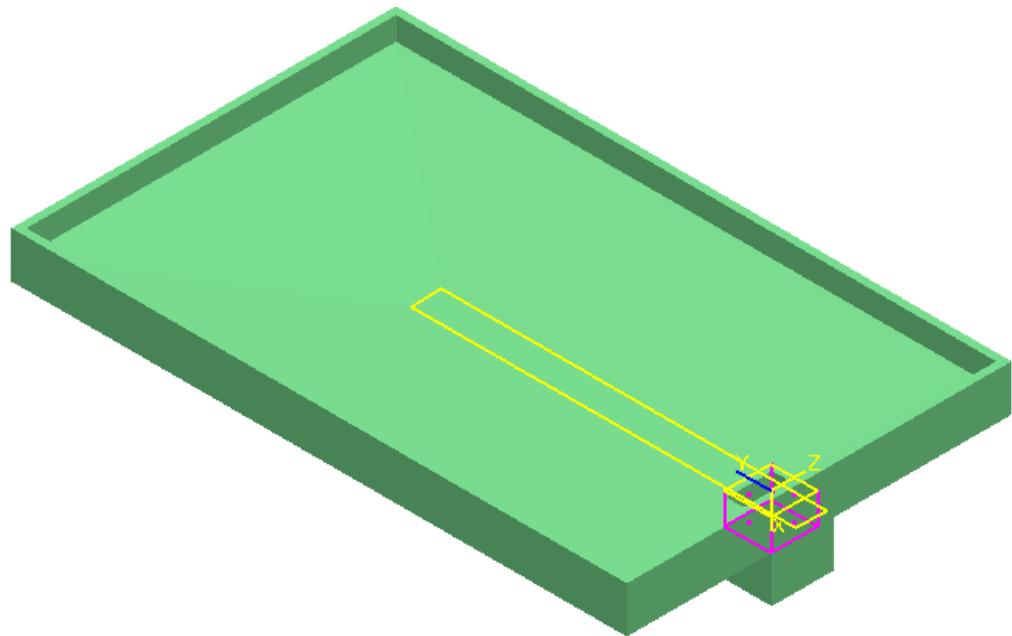
58. Select the top face of the *Opening*, using the quick pick. Your view should resemble the following graphic.



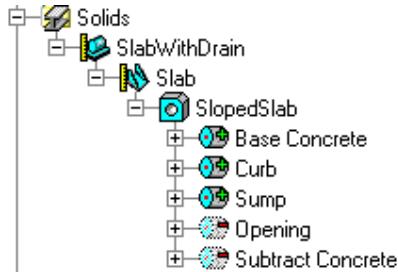
59. Select the surface of the *Subtract Concrete*. Your view should resemble the following graphic.



60. Once the mate relationship has been applied, your view should resemble the following graphic.



61. The finished hierarchy in Workspace Explorer should look like the following with the exception of the Shape Names order.



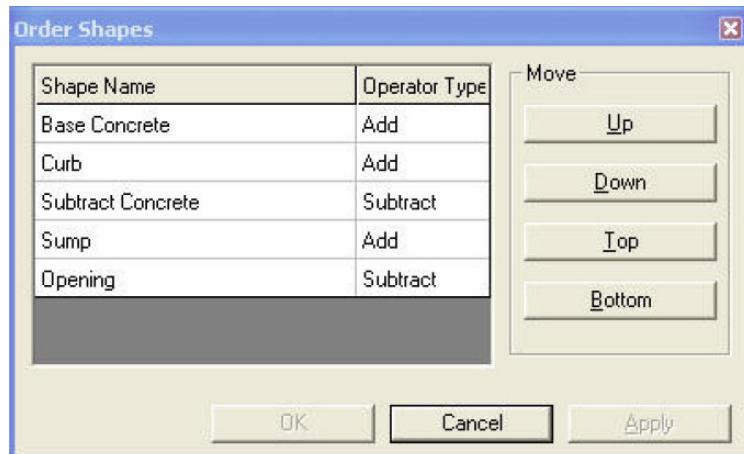
Order Shapes

62. 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

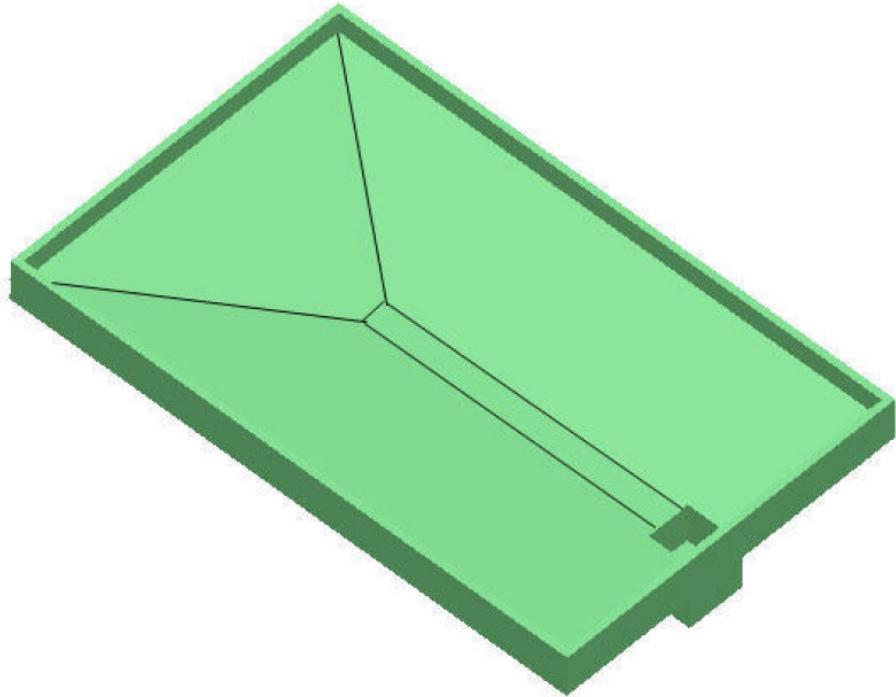


on the Modify Designed Solid ribbon.

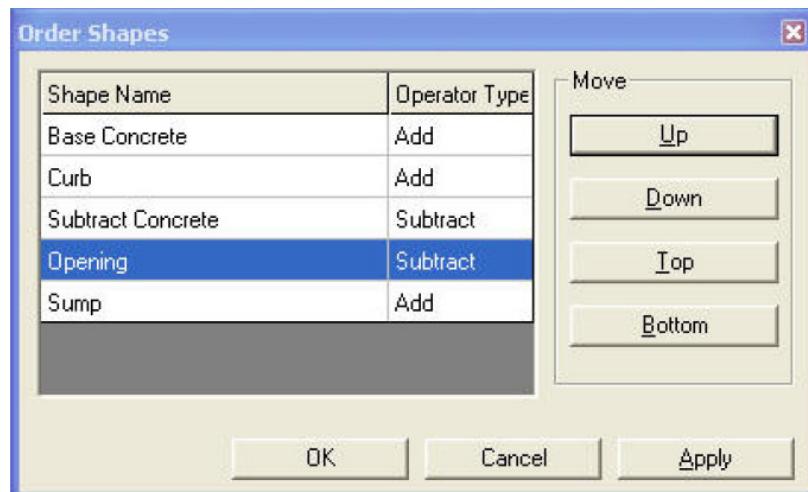
63. Select the Sloped Slab in the workspace hierarchy. Click on the Operators List and Order Shapes dialog box should appear. Your view should resemble the following graphic.



64. Your view should resemble the following graphic for the sloped slab.

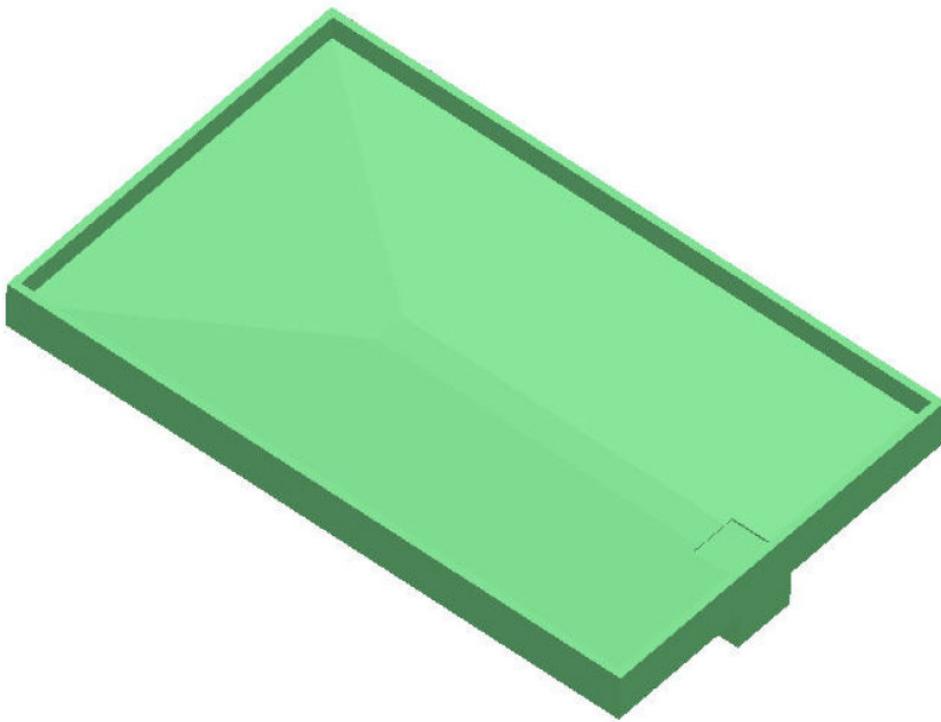


65. Select the Opening in the dialog box and hit the “Up”. Your view should resemble the following graphic.

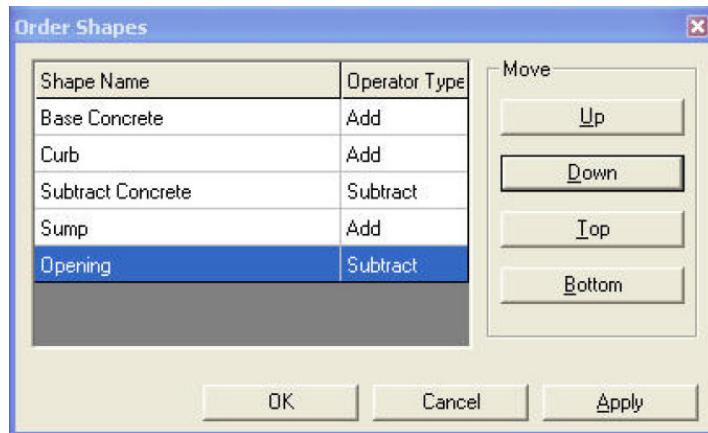


66. Hit Apply. Hit Cancel.

67. Your view should resemble the following graphic for the sloped slab.



68. Select the Opening in the dialog box and hit the “Down”. Your view should resemble the following graphic.



69. Hit Apply. Hit Cancel.

Properties

70. Now we can see the Surface Area, Volume, Weight and CG values for the *SlopedSlab*.

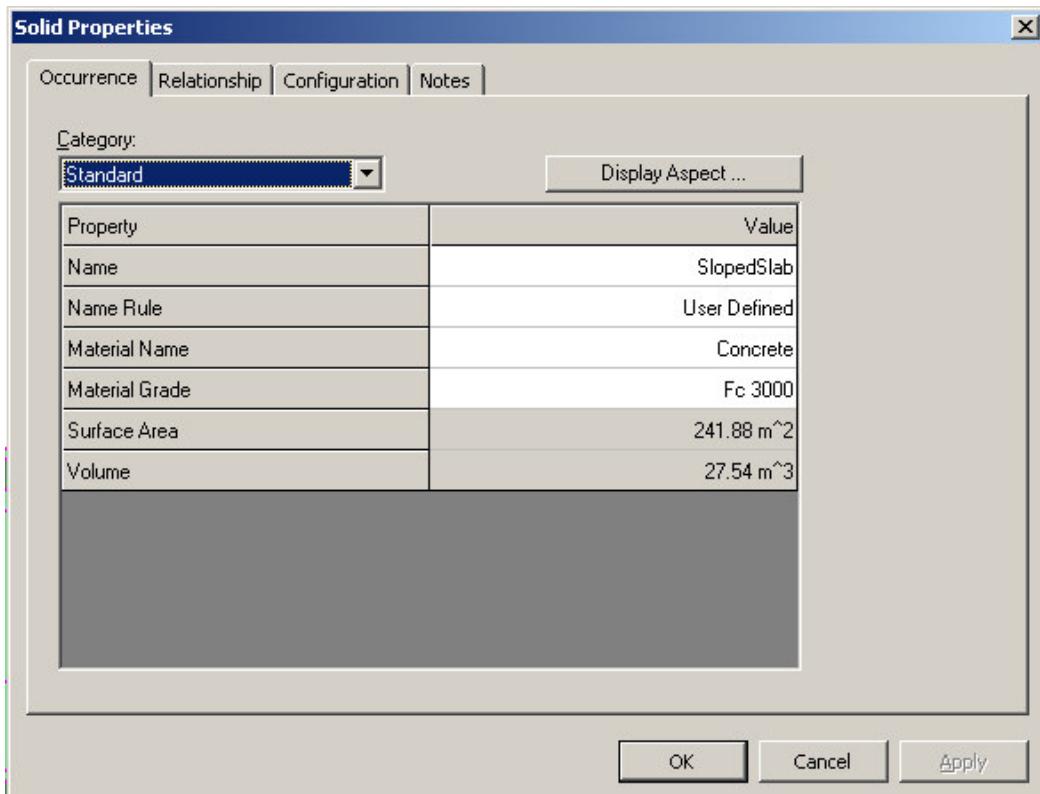
71. *SlopedSlab* with the *Sump* and without the *Sump* weight can be computed using suppress option.

72. Select *SlopedSlab* in Workspace Explorer and go to the Properties page.

73. Change the Category dropdown to Weight and CG and set Dry WCG Origin to Computed. Press Apply.

Dry Weight	63963.79 kg
DryCGX	-14251 mm
DryCGY	-11819 mm
DryCGZ	-55 mm

74. Change the Category dropdown to Standard.

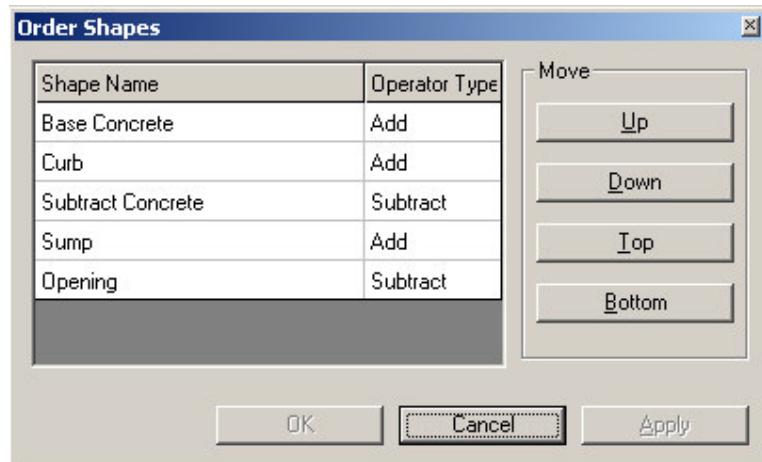


Suppress

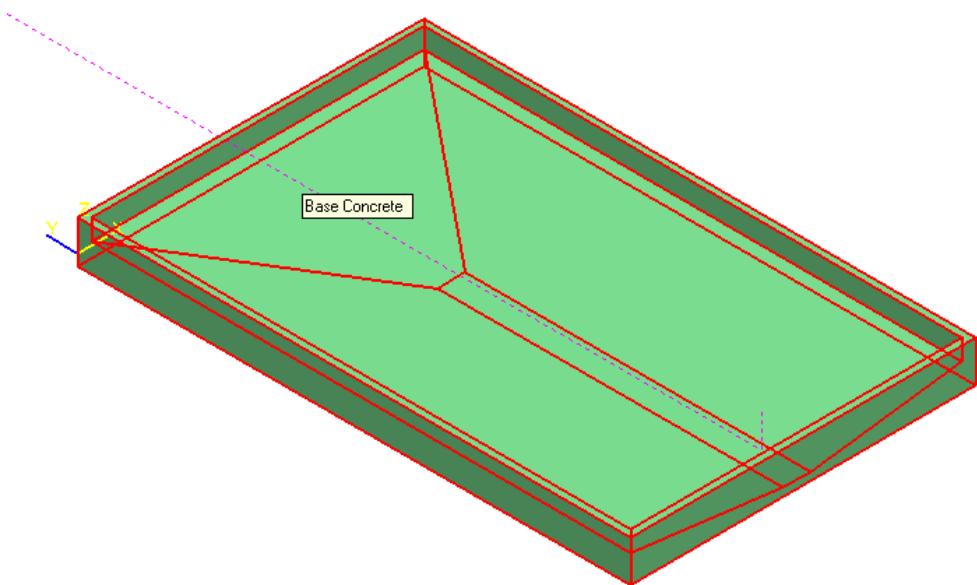
75. Select *SlopedSlab* in Workspace Explorer.

76. In the ribbon bar, select the Operators List  icon.

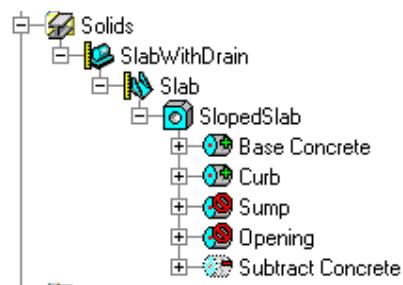
77. The Order Shapes dialog will appear. Notice that the order of the Shapes in the dialog is the same as the order in which the Shapes were placed.



78. Change the operator type to Suppress for *Sump* and *Opening*. Hit OK. Your view should resemble the following graphic.



79. Workspace hierarchy should resemble the following graphic.

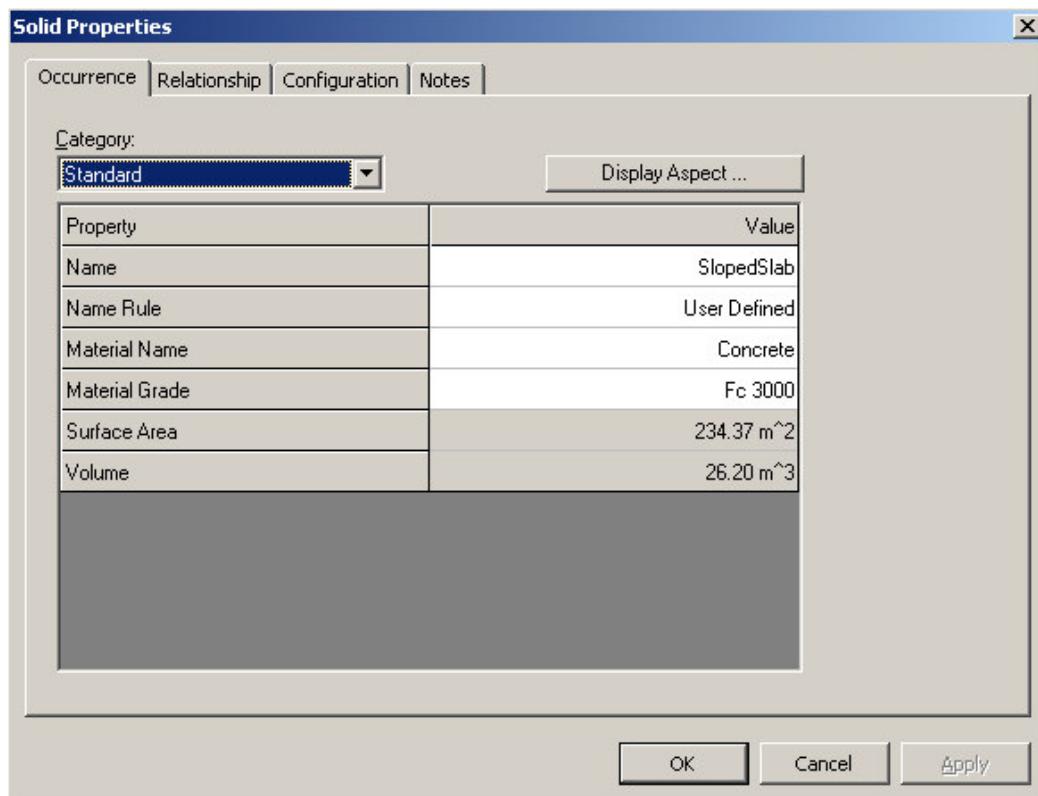


80. Select *SlopedSlab* in Workspace Explorer and go to the Properties page.

81. Change the Category dropdown to Weight and CG and set Dry WCG Origin to Computed. Press Apply.

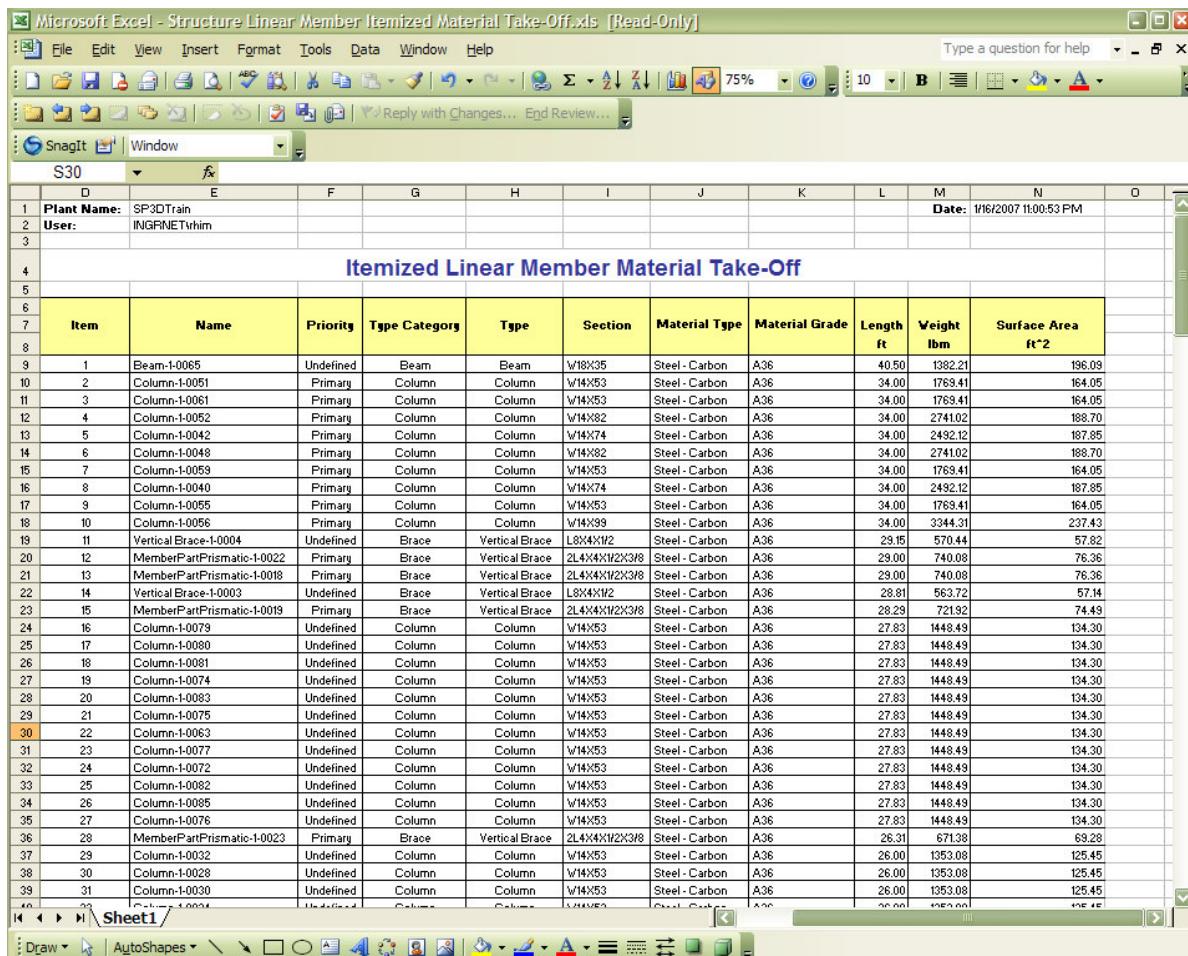
Dry Weight	60850.08 kg
DryCGX	-14250 mm
DryCGY	-11549 mm
DryCGZ	-9 mm

82. Change the Category dropdown to Standard.



Lab 14- Reports

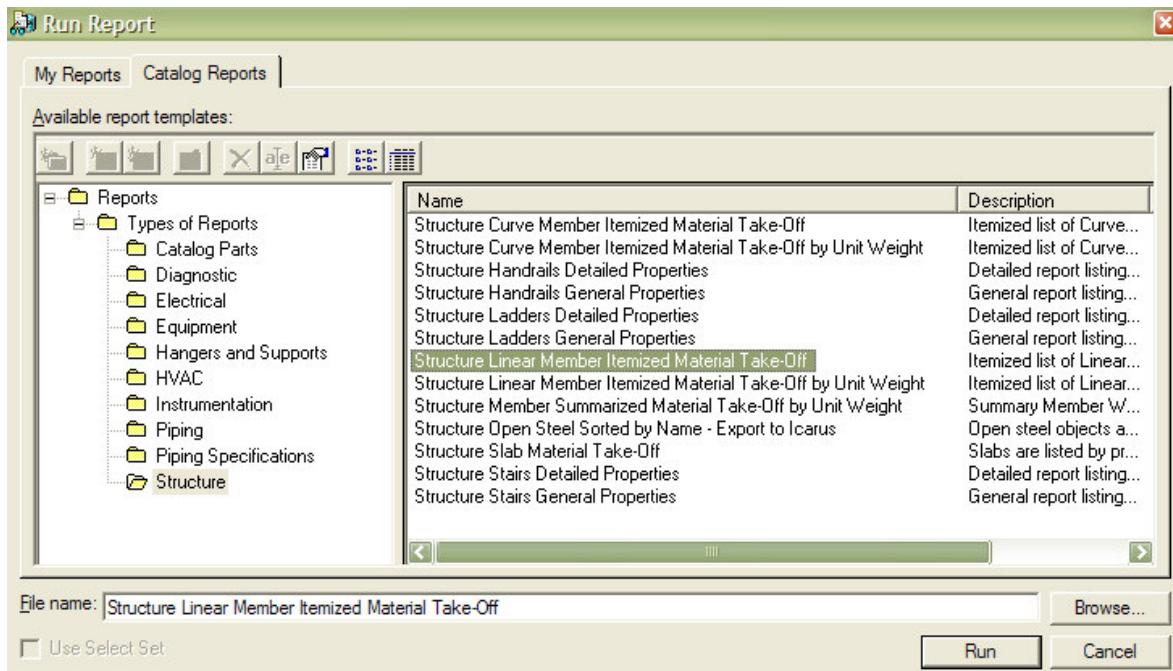
Extract a Linear Member material list from the model.



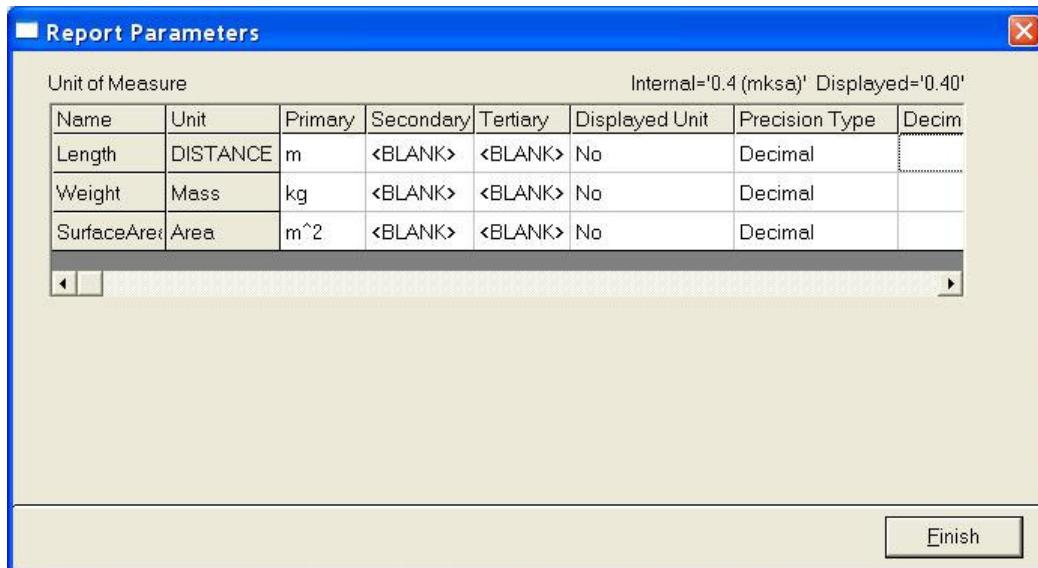
The screenshot shows a Microsoft Excel spreadsheet titled "Microsoft Excel - Structure Linear Member Itemized Material Take-Off.xls [Read-Only]". The spreadsheet contains a header section with plant and user information, followed by a table titled "Itemized Linear Member Material Take-Off". The table has columns for Item, Name, Priority, Type Category, Type, Section, Material Type, Material Grade, Length ft, Weight lbm, and Surface Area ft^2. The data includes various structural components like beams, columns, and braces, with their respective dimensions and material properties.

	D	E	F	G	H	I	J	K	L	M	N	O
1	Plant Name:	SP3DTrain								Date:	1/16/2007 11:00:53 PM	
2	User:	INGRNET\him										
Itemized Linear Member Material Take-Off												
9	Item	Name	Priority	Type Category	Type	Section	Material Type	Material Grade	Length ft	Weight lbm	Surface Area ft^2	
10	1	Beam-1-0065	Undefined	Beam	Beam	V18X35	Steel - Carbon	A36	40.50	1382.21	196.09	
11	2	Column-1-0051	Primary	Column	Column	V14X53	Steel - Carbon	A36	34.00	1763.41	164.05	
12	3	Column-1-0061	Primary	Column	Column	V14X53	Steel - Carbon	A36	34.00	1763.41	164.05	
13	4	Column-1-0052	Primary	Column	Column	V14X82	Steel - Carbon	A36	34.00	2741.02	188.70	
14	5	Column-1-0042	Primary	Column	Column	V14X74	Steel - Carbon	A36	34.00	2492.12	187.85	
15	6	Column-1-0048	Primary	Column	Column	V14X82	Steel - Carbon	A36	34.00	2741.02	188.70	
16	7	Column-1-0053	Primary	Column	Column	V14X53	Steel - Carbon	A36	34.00	1763.41	164.05	
17	8	Column-1-0040	Primary	Column	Column	V14X74	Steel - Carbon	A36	34.00	2492.12	187.85	
18	9	Column-1-0055	Primary	Column	Column	V14X53	Steel - Carbon	A36	34.00	1763.41	164.05	
19	10	Column-1-0056	Primary	Column	Column	V14X99	Steel - Carbon	A36	34.00	3344.31	237.43	
20	11	Vertical Brace-1-0004	Undefined	Brace	Vertical Brace	L8X4X12	Steel - Carbon	A36	29.15	570.44	57.82	
21	12	MemberPartPrismatic-1-0022	Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	29.00	740.08	76.36	
22	13	MemberPartPrismatic-1-0018	Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	29.00	740.08	76.36	
23	14	Vertical Brace-1-0003	Undefined	Brace	Vertical Brace	L8X4X12	Steel - Carbon	A36	28.81	563.72	57.14	
24	15	MemberPartPrismatic-1-0013	Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	28.29	721.92	74.49	
25	16	Column-1-0079	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
26	17	Column-1-0080	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
27	18	Column-1-0081	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
28	19	Column-1-0074	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
29	20	Column-1-0083	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
30	21	Column-1-0075	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
31	22	Column-1-0063	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
32	23	Column-1-0077	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
33	24	Column-1-0072	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
34	25	Column-1-0082	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
35	26	Column-1-0085	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
36	27	Column-1-0076	Undefined	Column	Column	V14X53	Steel - Carbon	A36	27.83	1448.49	134.30	
37	28	MemberPartPrismatic-1-0023	Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	26.31	671.38	69.28	
38	29	Column-1-0032	Undefined	Column	Column	V14X53	Steel - Carbon	A36	26.00	1353.08	125.45	
39	30	Column-1-0028	Undefined	Column	Column	V14X53	Steel - Carbon	A36	26.00	1353.08	125.45	
40	31	Column-1-0030	Undefined	Column	Column	V14X53	Steel - Carbon	A36	26.00	1353.08	125.45	

- 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. Select SP3DTrainMet System from the filter selection
4. Set units as below:



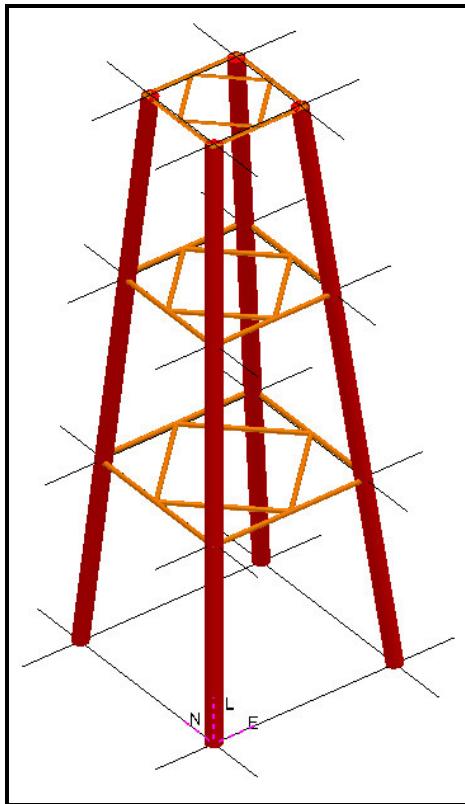
5. Select Finish
6. Review report

Lab 15 - Structural Modeling (Offshore Jacket) Optional

Part I: Offshore Jacket Pipes

In this lab, you will build a simple steel-frame offshore jacket.

Use the Offshore grid system to help 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.



1. Open or create a session file and define an appropriate filter for the workspace.
2. Go to the Structure Task environment. Make sure the Active Permission Group is set to *Structura*.
3. Select the place linear member system command. Use the views shown in figure 1 to place the appropriate support columns and beams.
4. Select the place linear member system command. System displays the place linear member system smart step ribbon bar.
5. Use the ribbon bar and set the active member parameters as follows:

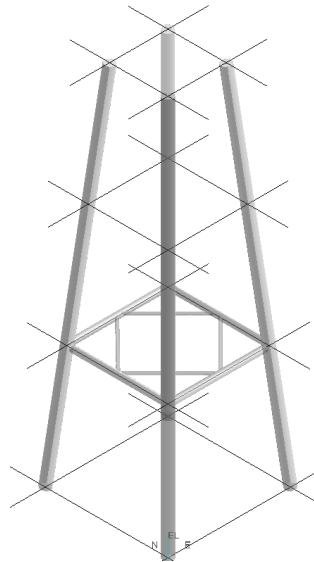
Connection:	By Rule
Type Category:	Column
Type:	Column
Section Name:	CS10
Cardinal point:	5
Angle:	0 deg
System:	A2 -> U05 -> Structural - > Column.

6. Place the first column end at the grid intersections **Elevation 0m**. Place the second column end at the grid intersections **Elevation 9m**. Repeat the above steps to place the other support columns. Toggle the start/end icons to set the discrete/contiguous placement method.
7. The next step is to place the steel frame at **Elevation 3m**. Beams are placed by selecting and lock constraint on to the column and then locate point-on any grid line passing through that column.
8. Use the ribbon bar and set the active member parameters as follows:

Type Category:	Beam
Type:	Beam
Connection:	Flush-Right for the perimeter pipe and Flush-Top for the intermediate pipe
Section Name:	CS3
Cardinal point:	5
Angle:	0 deg
System:	A2 -> U05 -> Structural -> Beams.

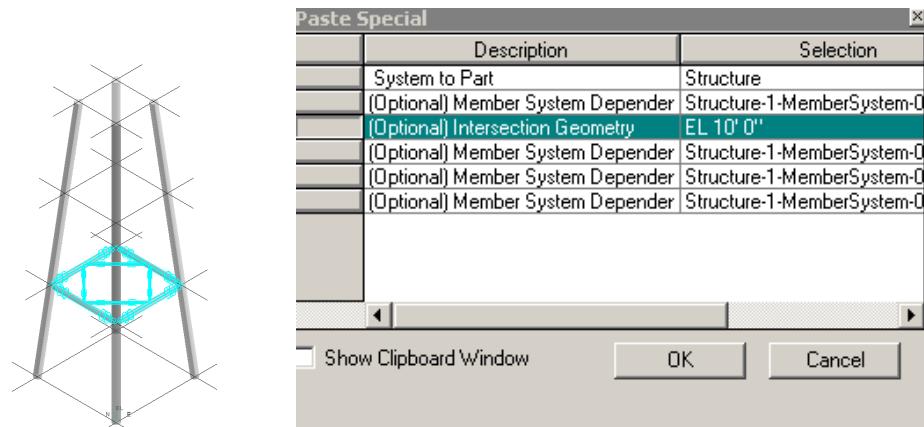
9. Toggle the start/end icons to set the discrete/contiguous placement method.

Your View should now resemble the following graphic.

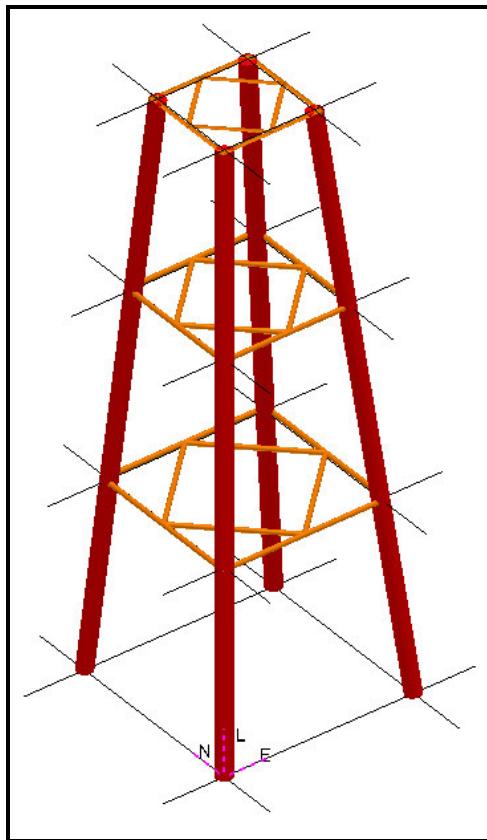


10. Select the Frame connections of the intermediate beams and set the Position Rule to Ratio
11. The next step is to place the other steel frames on the other elevations 0m, 6m and 9m. Use the Copy/Paste command to place all the intermediate members for the first frame at the other elevations. Make sure to select the appropriate objects to re-establish the connection in the Paste Special Dialog box.

Hint: Set the select filter to Member System to select 8 members
Only 1 related object is needed to re-establish the connection (El plane)
Use the Paste command two times

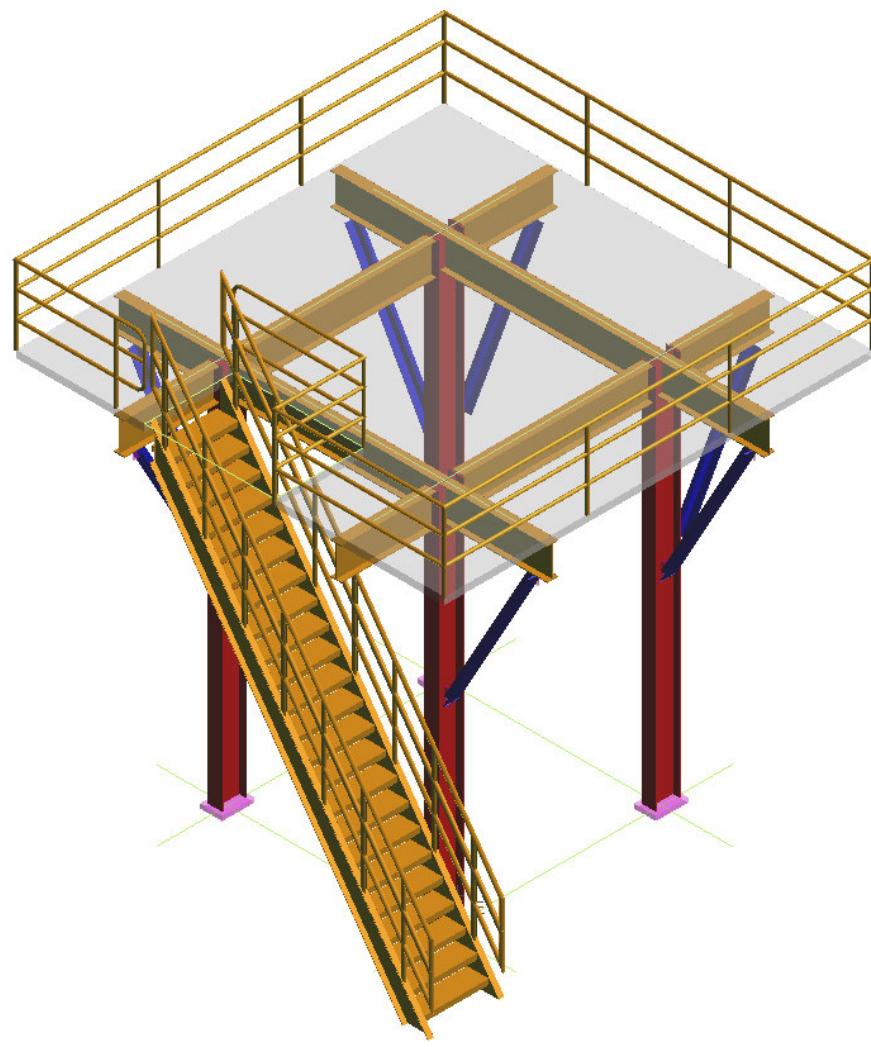


The final view should resemble the following graphic:



Part II: Rotated Coordinate System – Optional

1 Model shown Structures from Pictures and information provided:



Coordinate System: copy of U04

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 -> Structure -> Beams

Beam Extension Length: 5' (TYP)

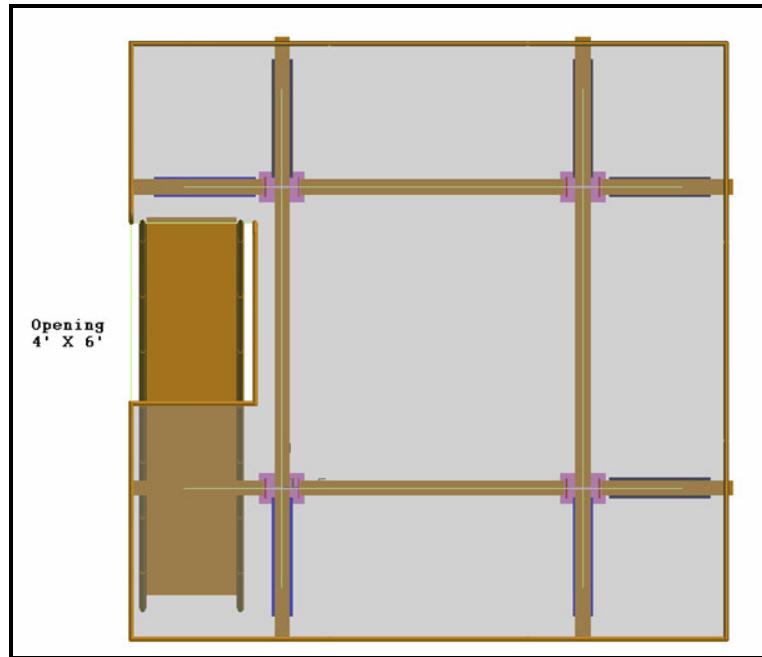
Vertical Braces:

Connection:	By Rule
Type Category:	Brace
Type:	V. Brace
Section Name:	2L4X4X1/2X3/8
Cardinal point:	5
Angle:	0
System:	A2 -> U04 -> Structural -> V. Brace

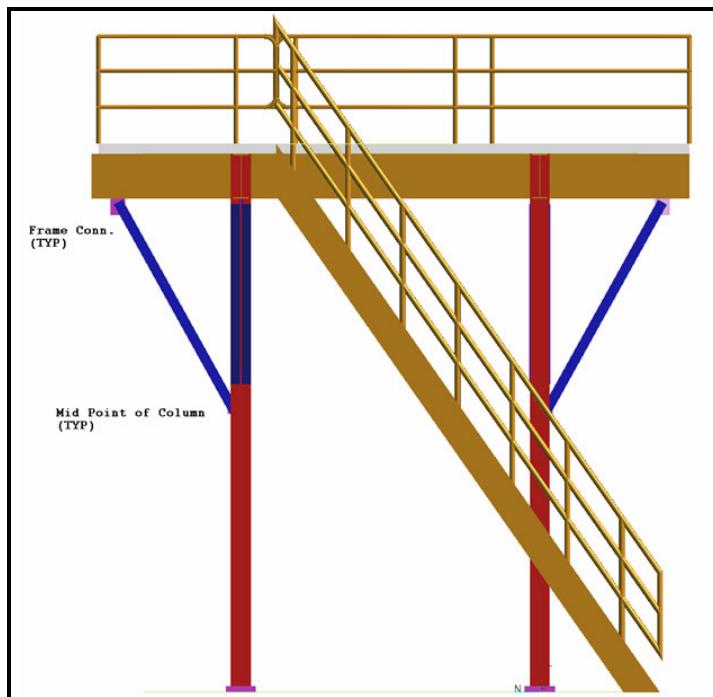
Stair:

Type:	StairA1
Width:	75mm
Angle:	55

System: A2 -> U04 -> Structural -> Miscellaneous



Top View - U04 - B Structure



West View - U04 - B Structure

Lab 16 - Using Building Wizard (optional)

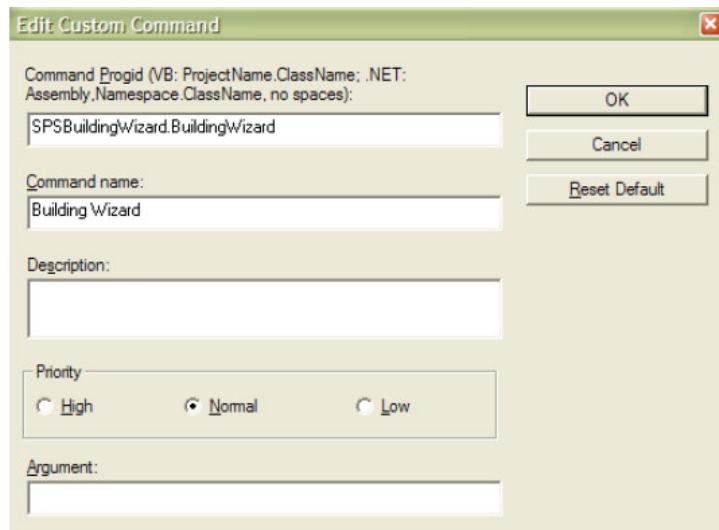
Objective

- Create Building_2 at -250m, 10m, 0m

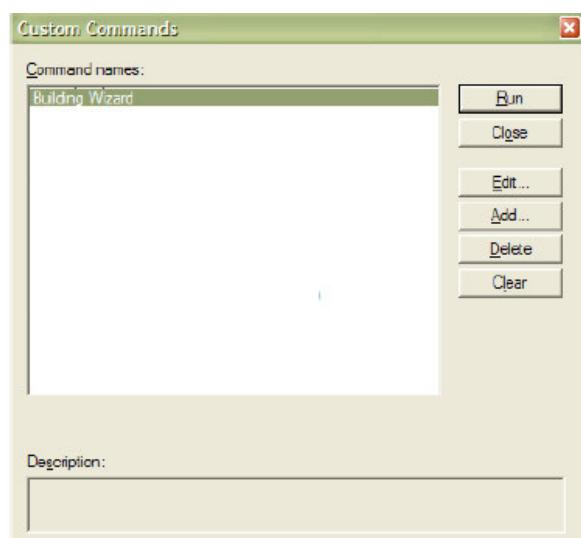
Add custom command

Add a custom command for the building wizard as follows:

1. Select Tools -> Custom Command
2. Click “Add” button
3. Enter values as shown:



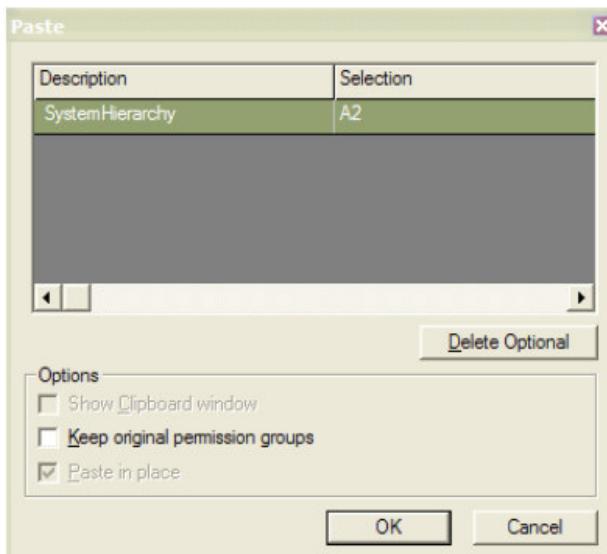
4. Click “OK” button



5. Click “Close” button

Create the systems required

1. Switch to Systems and Specification Task.
2. Expand A2 in the system hierarchy.
3. Right click U05 to open the popup menu.
4. Select the “select Nested” option
5. Select Edit -> Copy
6. Select Edit -> Paste and click “OK” button



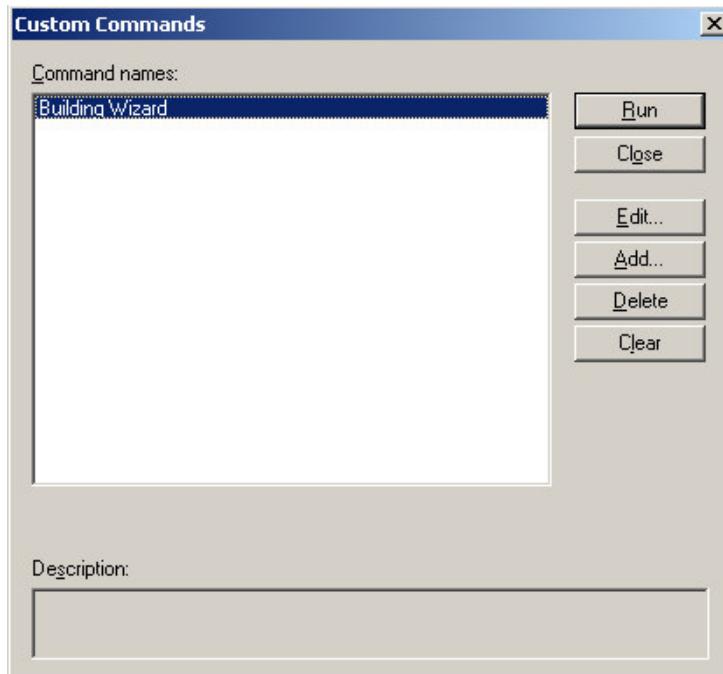
7. Select A” system in the tree view. Seelct “U05(2)” in the right window.

Name	Type
U01	Unit System
U02	Unit System
U03	Unit System
U04	Unit System
U05	Unit System
U05(2)	Unit System
U06	Unit System
U07	Unit System

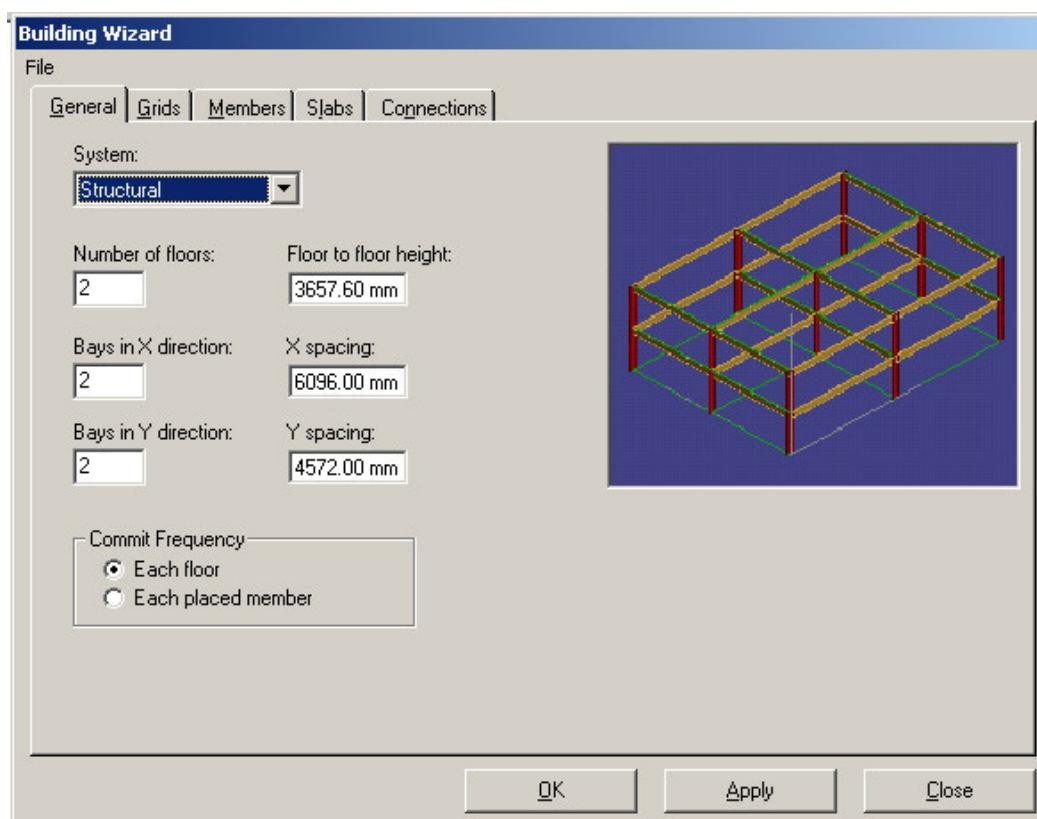
8. Rename U05(2) to U20

Run the building wizard

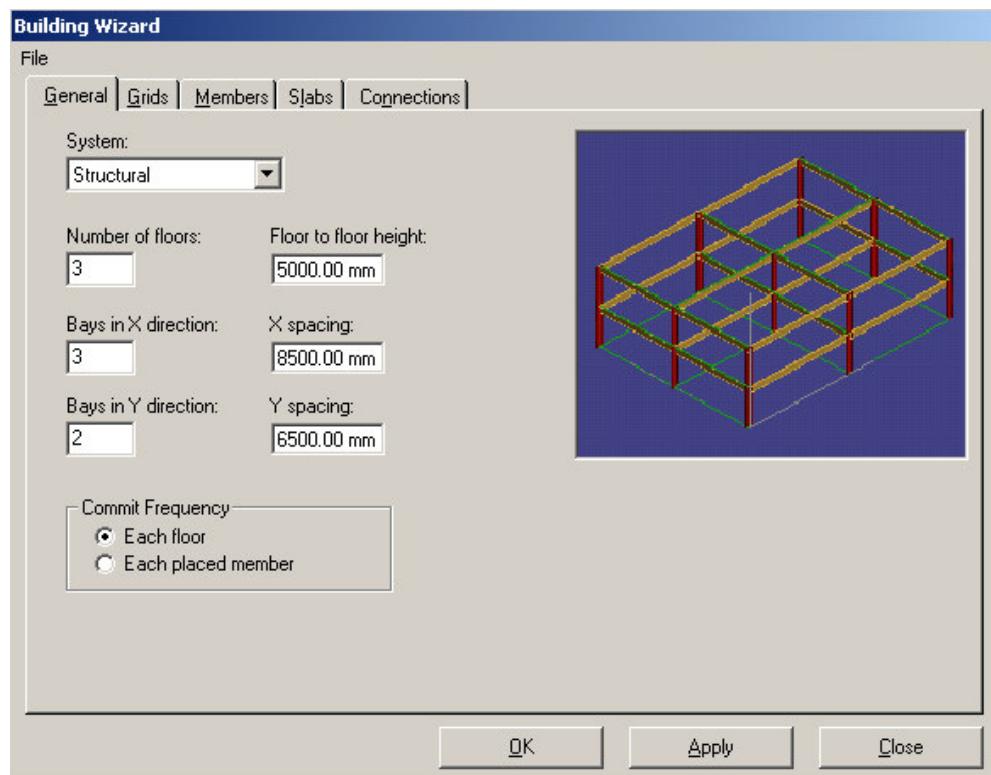
1. Switch to the Structure Task.
2. Select Tools -> Custom Commands.



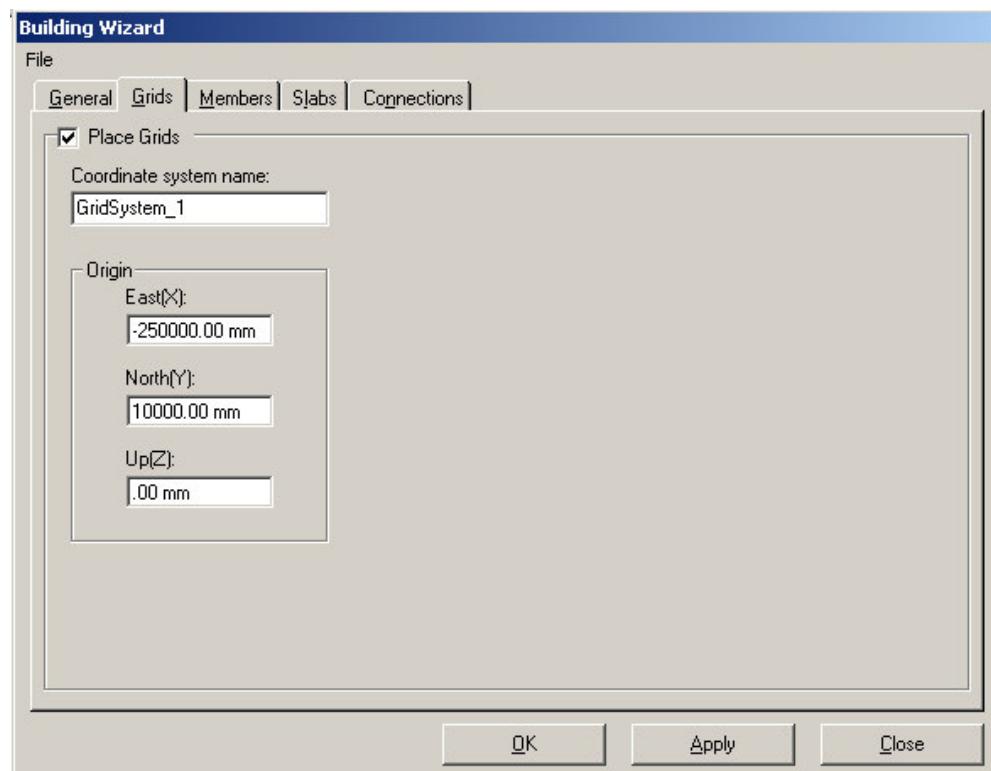
3. Select the Building Wizard.
4. Click “Run” button.
The building Wizard opens up.
5. In the System field, select More..., then pick A2 -> U20 -> Structural



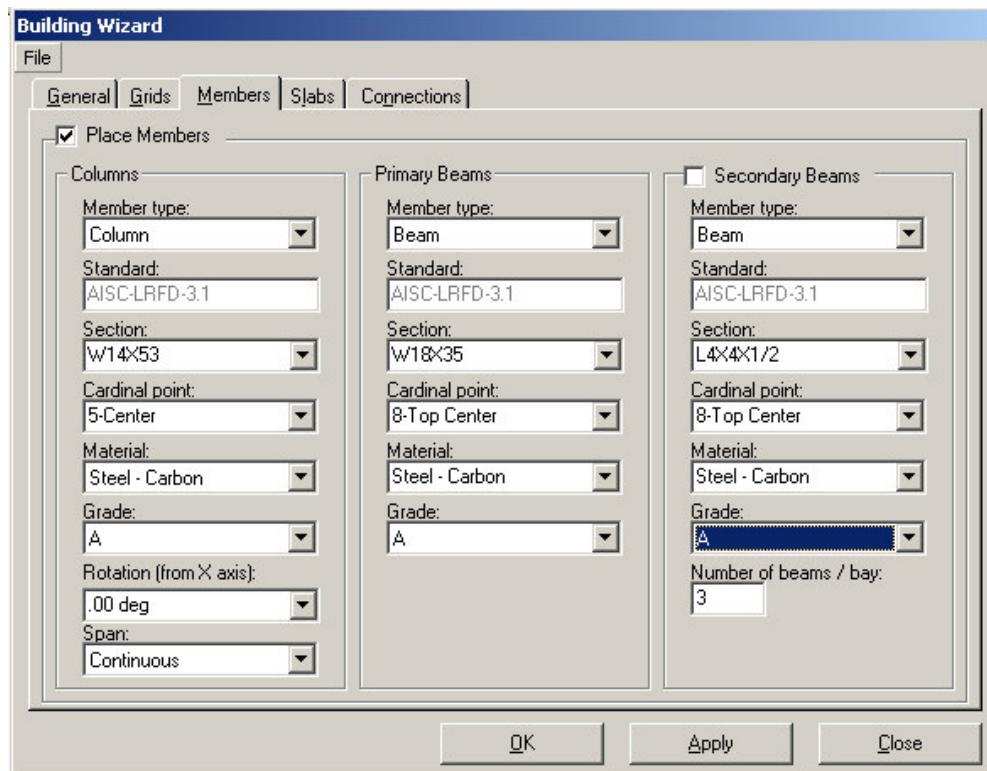
6. On the General Tab, key-in the following:



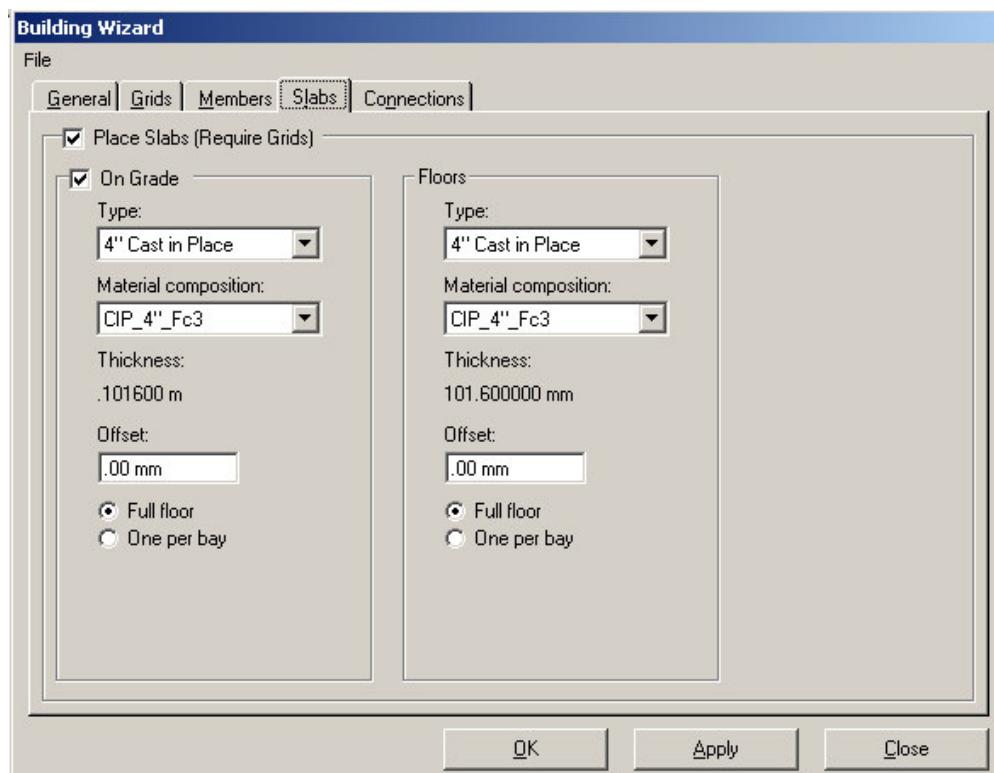
7. On the Grids tab, key-in the following:



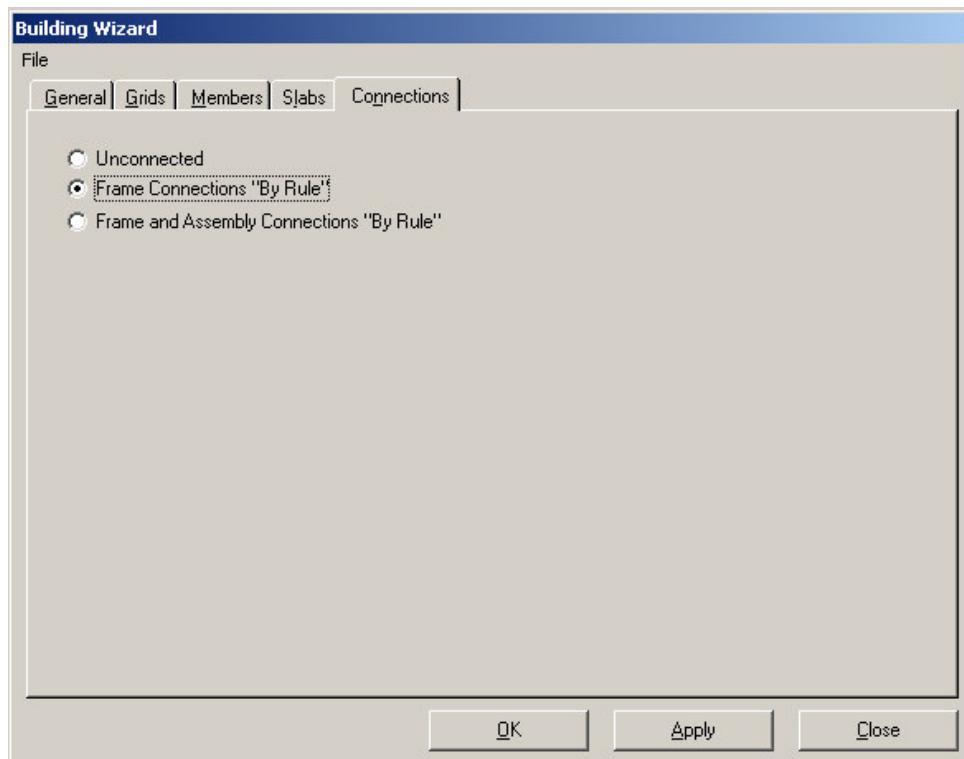
8. On the Members tab, key-in or select the following:



9. On the Slabs tab, keep the defaults.

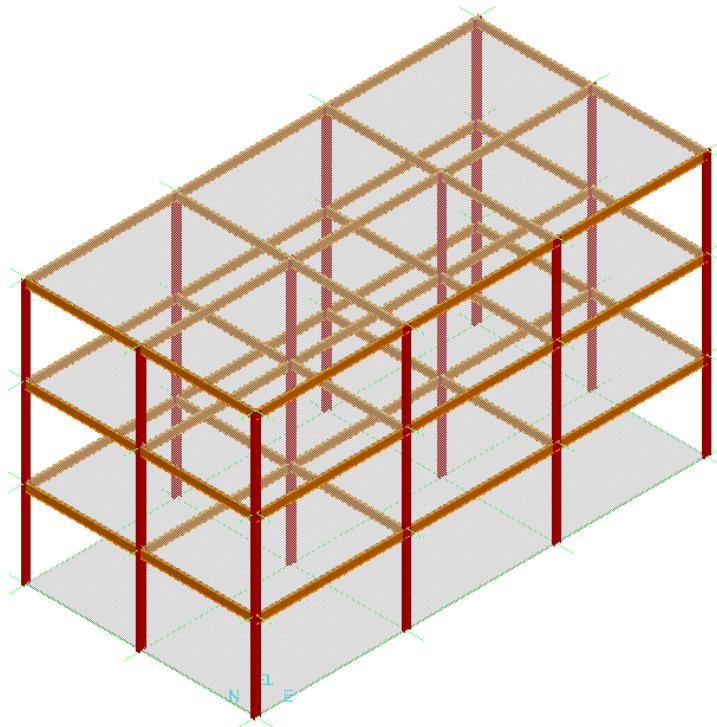


10. On the Connections tab, select “Frame Connections by Rule”.



11. Click “OK” button.

Your view of the structure should look like this:



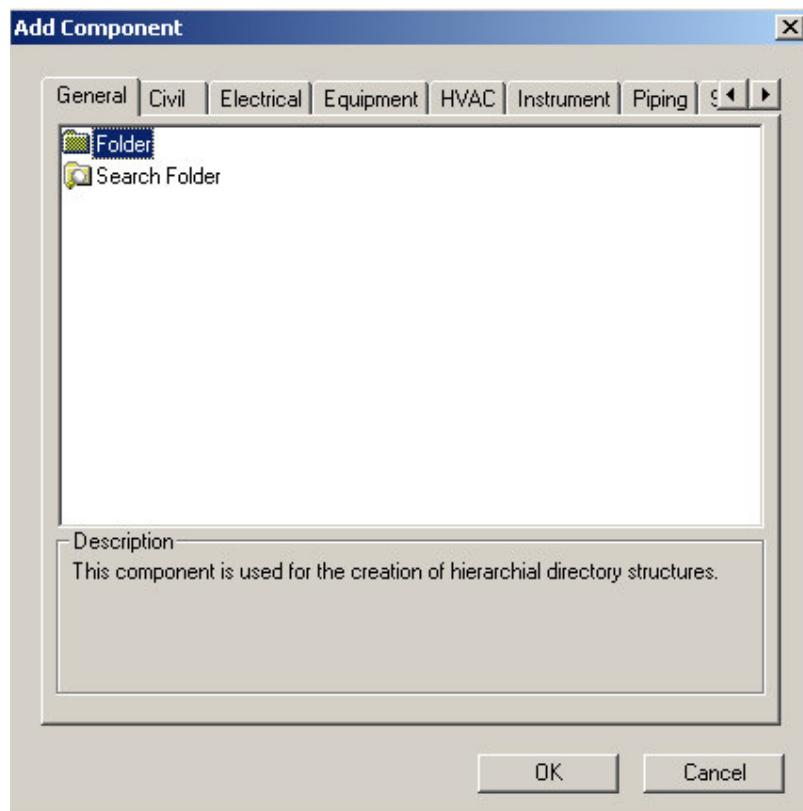
Lab 17 - Data Exchange for Solids (Optional)

Objective

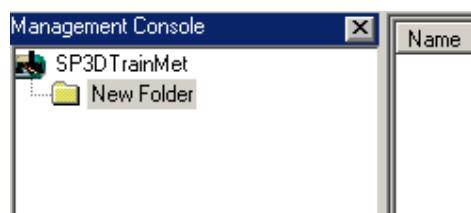
- Export solids as a .SAT file
- Import solids from a.SAT file

Part A: Export

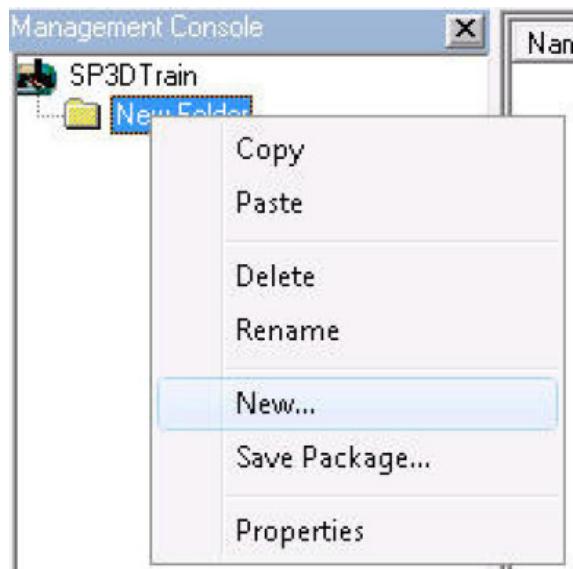
1. Open a previous session file or create a new session file containing the previously modeled Solid.
2. Go to Tasks -> Drawings and Reports
3. In the management Console window, right click on SP3DTrainmet and select New.
4. Add Component window will appear. In the General tab Select Folder. Your view should resemble the following graphic.



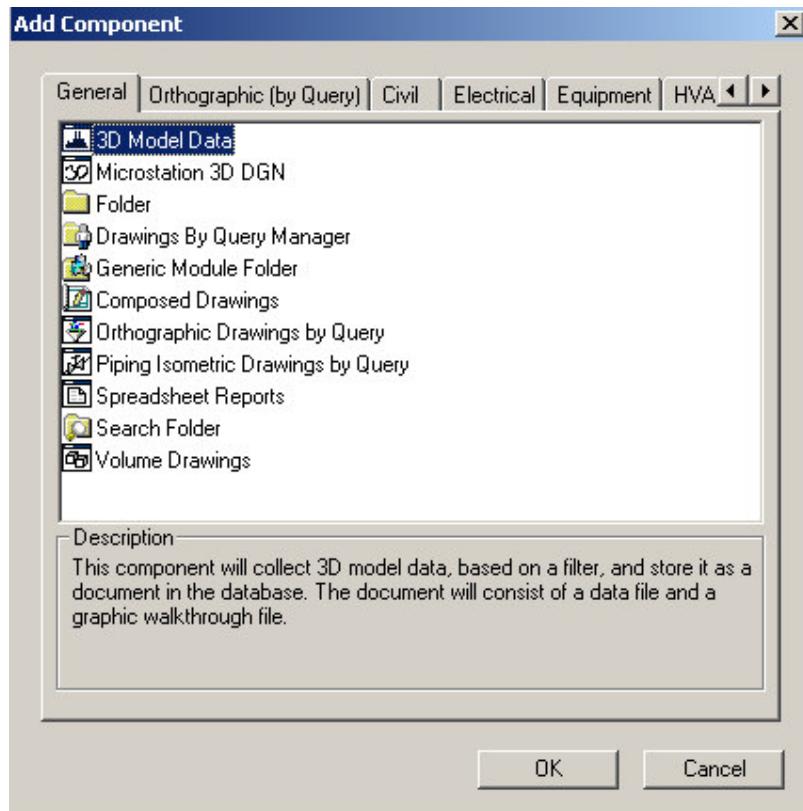
5. Hit OK. In the Management Console window New Folder is created.



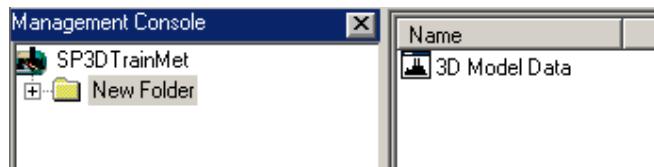
6. Right Click on the New Folder and select New as shown below.



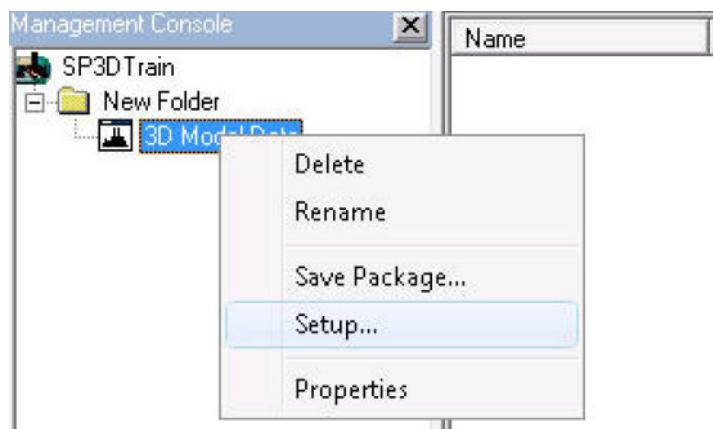
7. Choose 3D Model Data from the General Tab as shown below and hit OK.



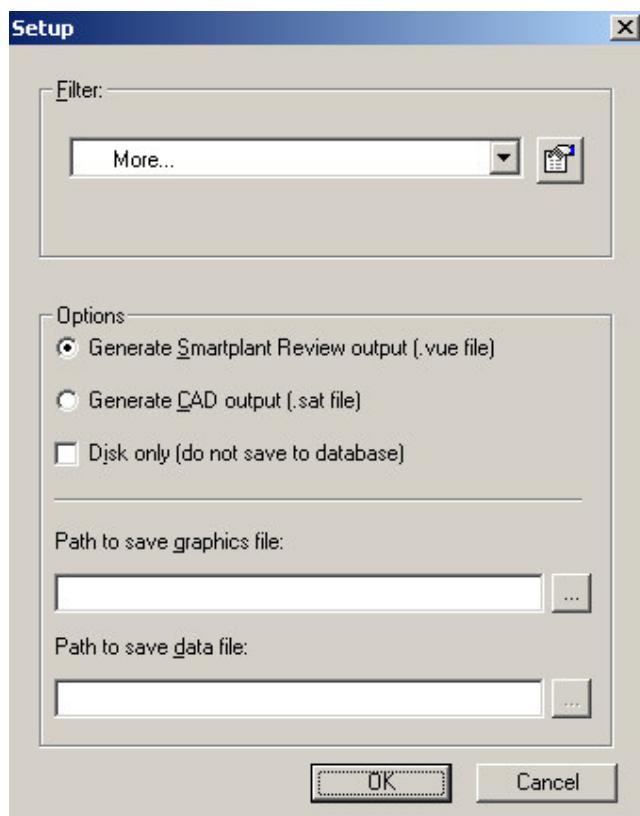
8. Your Management Console should resemble the following graphic.



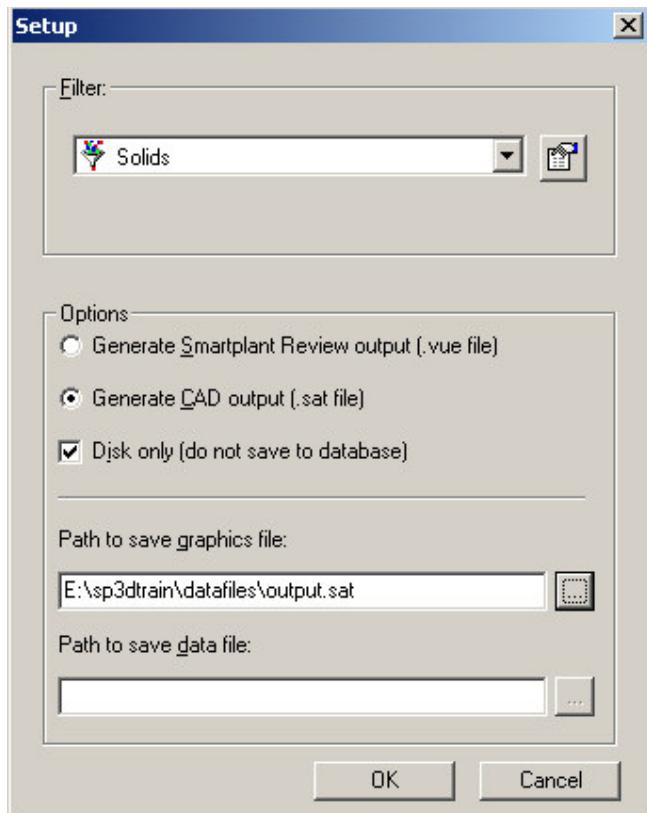
9. Right click on 3D Model Data and choose Setup...



10. Your View should resemble the following graphic.

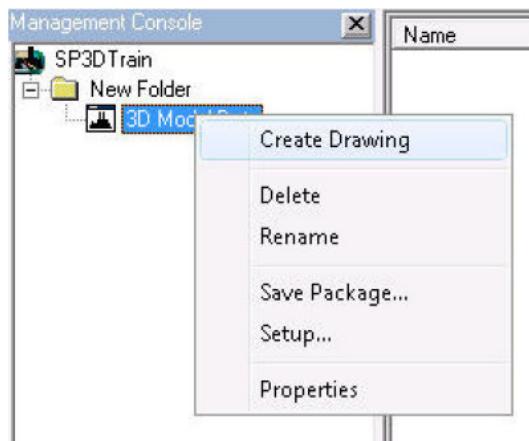


11. Go to More and select the filter that contains the *Slopedslab*. Also provide a path to save the sat file. Your view should resemble the following graphic.

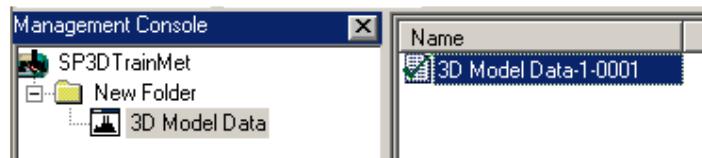


12. Hit OK.

13. Right click on the 3D Model Data and select Create Drawing.



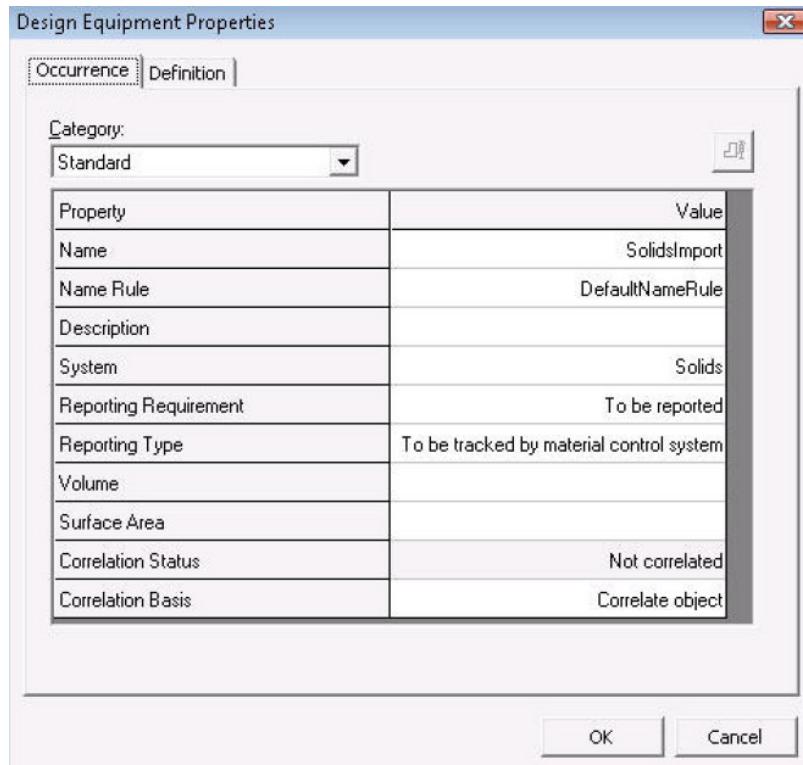
14. Right Click on 3D Model Data-1-0001 and select Update Now. When complete your view should resemble the following graphic.



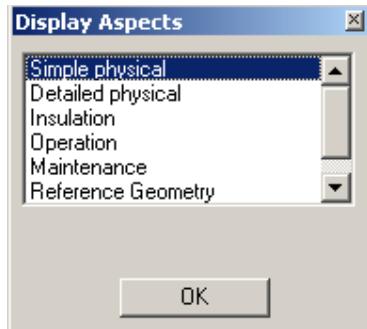
15. The file has been successfully exported.

Part B: Import

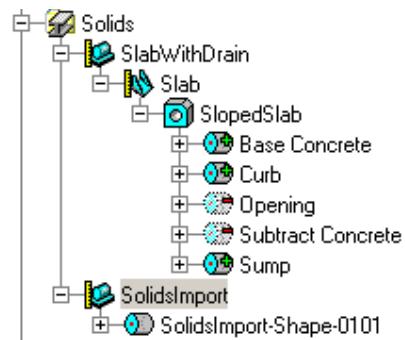
1. Create a new session file or use an existing one.
2. Create a new filter by the name Solids-import.
3. Define the workspace to include A2-> U05 -> Structural -> Solids
4. Expand the Fly-out tool bar and press the  icon on the Structure vertical toolbar to start the Place Design Equipment command. Expand *Equipment -> Civil* and select *Miscellaneous*. Press OK.
5. In the Design Equipment properties dialog, choose *Solids* as the system and give a User Defined name of *SolidsImport*. Press OK



6. Activate the PinPoint toolbar, change the coordinate system to U05 CS and set the Target to Origin.
7. Place the Design Equipment at E: 24000mm, N: 12000mm and El: 0mm
8. Select the Place Imported Shape from File command .
9. Select *SolidsImport* from the workspace explorer.
10. Select output.sat file and hit Open.
11. Select *Simple Physical* from the Display Aspects window and hit OK.



12. Key in E: 24000mm, N: 12000mm and El: 0mm using the Pin Point values and left mouse click in the workspace window.
13. The workspace explorer should resemble the following graphic.



STRUCTURAL ANALYSIS TASK

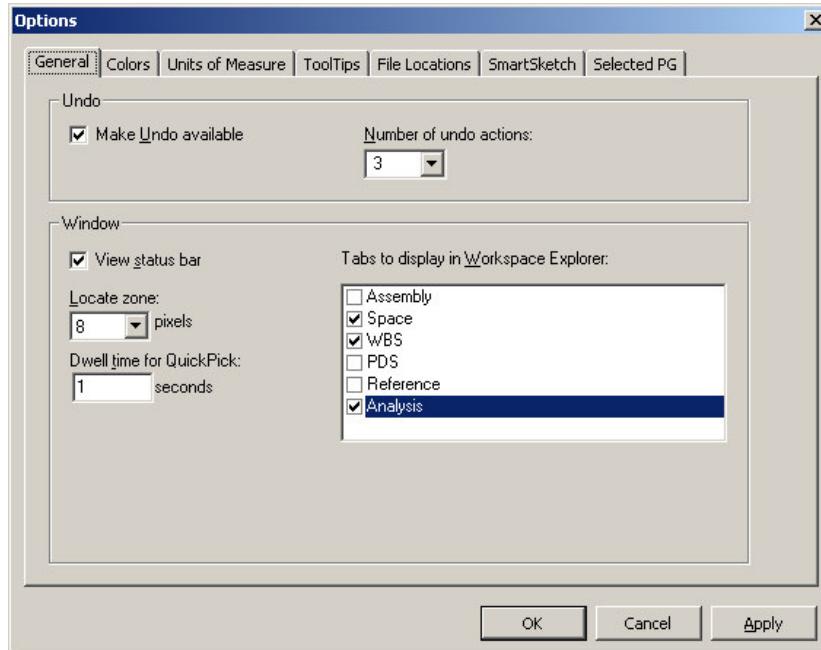
Lab 1 - Loads, Releases, Boundary Conditions and Creating a CIS file

Objective

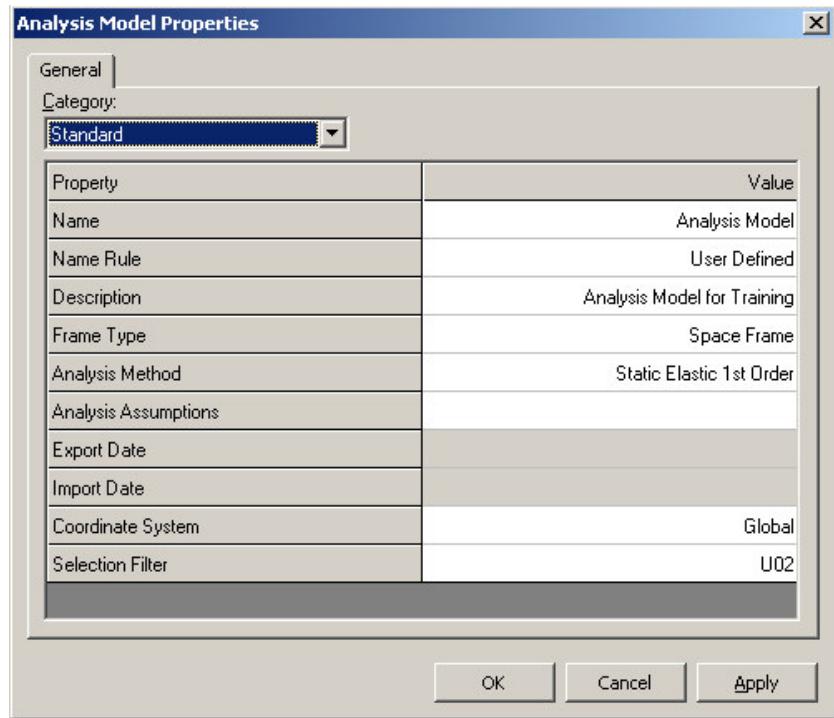
Students will be able to place dead and live distributed loads on beams. Place concentrated wind loads on one side of the structure. Define member end releases. Define member boundary conditions for the columns and create a CIS file.

Part I: Create Load Cases and Combinations

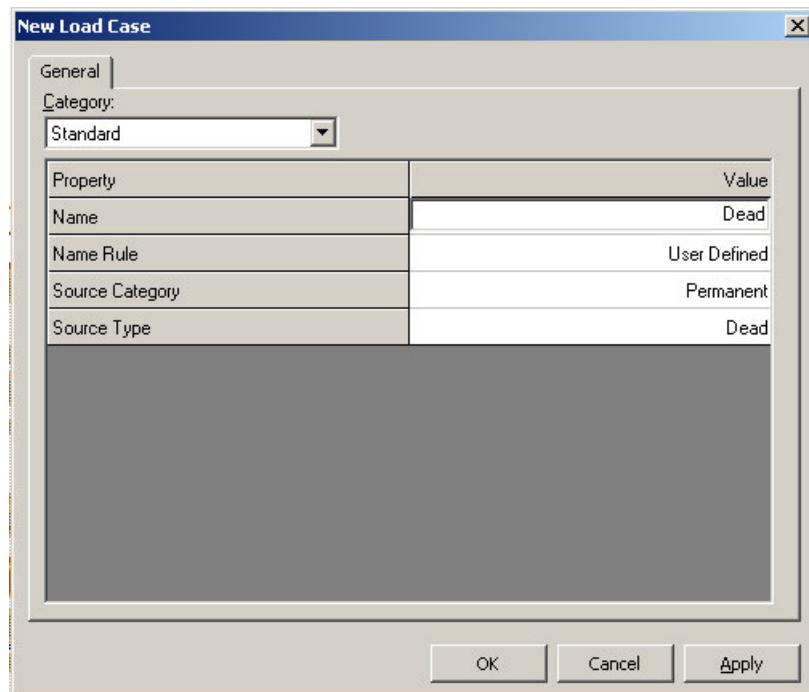
1. Open or create a session file and define an appropriate filter for your workspace that includes Grids, Structure and the Analysis model for U02 structure.
2. Go to Tools -> Option and check the Analysis.



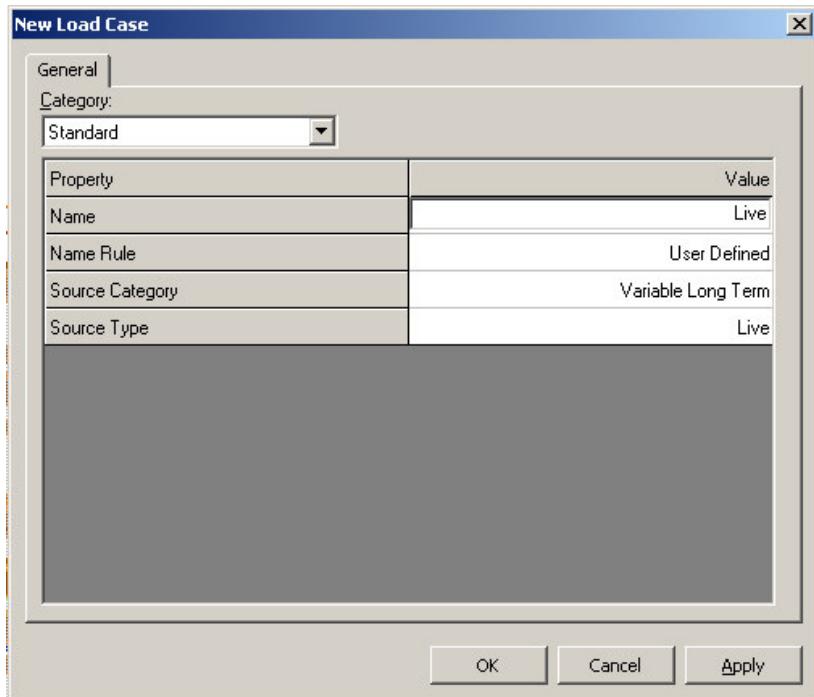
3. Hit “OK” button. Save your session.
4. Exit your session and Open it again. Notice the Analysis Tab in the Workspace Explorer.
5. Go to the Structural Analysis Task.
6. Create an Analysis Model using New Analysis Model Command.



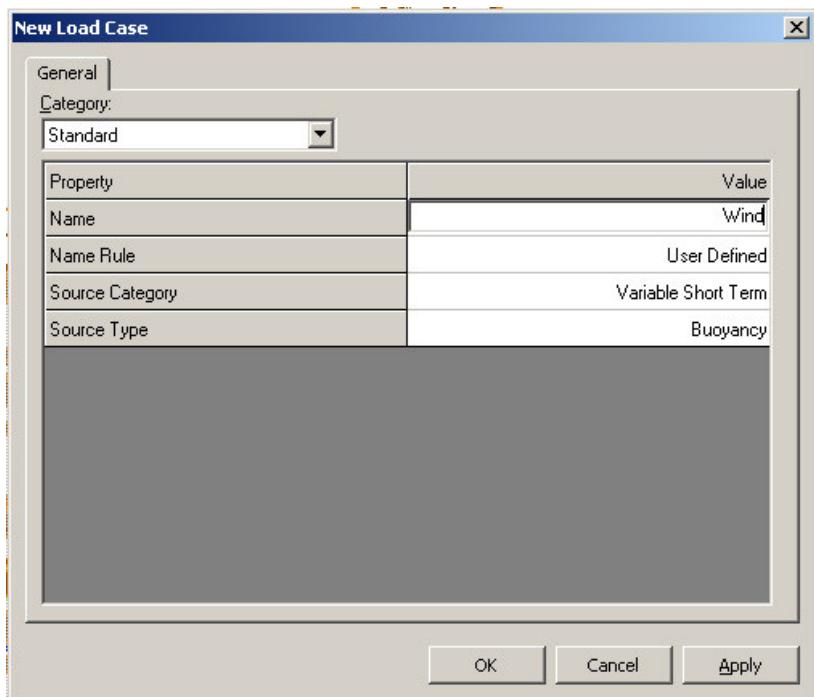
7. Hit “OK” Button.
8. Select Create New Load Case command. 
9. Create a DEAD load using New Load Case Command.



10. Hit “Apply” Button.
11. Create a LIVE load.



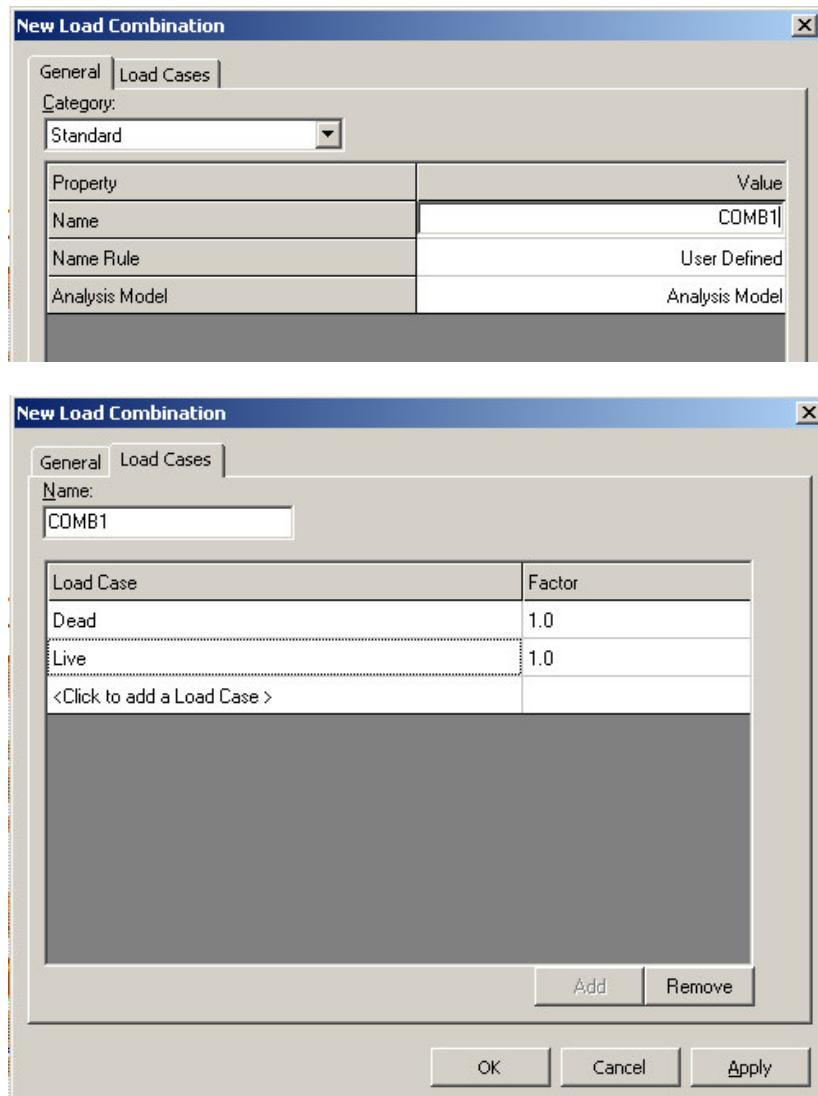
12. Hit “Apply” Button.
13. Create a WIND load.



14. Hit “OK” Button.
15. Create Load Combinations using New Load Combination Command 

COMB1: Dead + Live with the Factor as 1.0

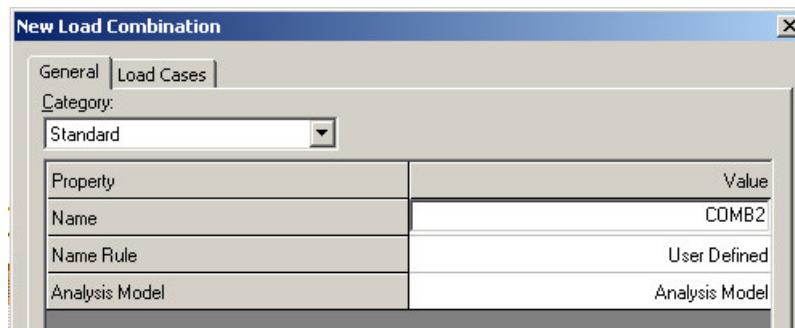
16. Set the parameter as follows:

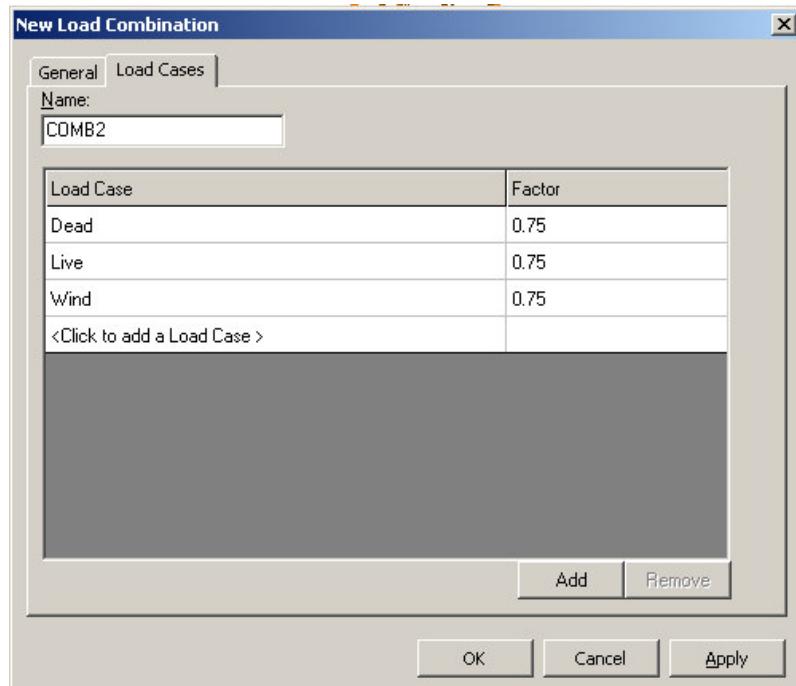


17. Hit “Apply” Button.

18. Create Load Combinations.

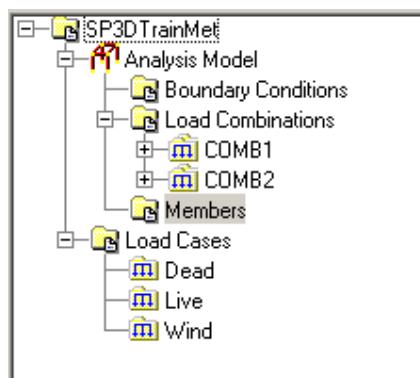
COMB2: Dead + Live + Wind with the Factor of 0.75





19. Hit “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

20. Select Place New Distributed Load Command.

21. Define the following load for the beams at Elevation 6m:

Load Case: DEAD

Load: Force Z

Reference: Global

Position as: Relative

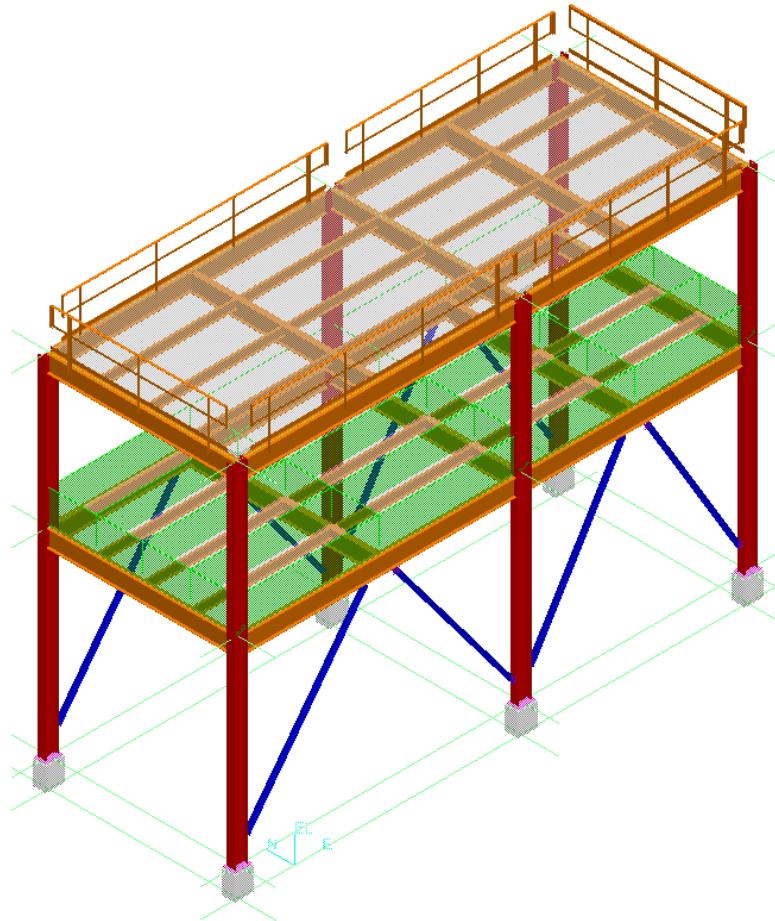
Start Position: 0

End Position: 1

Start and End Magnitude: -10000N/m

Load Case:	Load:	Reference:	Position As:	Start Position:	End Position:	Start Magnitude:	End Magnitude:
Dead	Force Z	Global	Relative	0	1	-10000.00 N/m	-10000.00 N/m

22. Select all beams at Elevation 6m and click “Finish” button. Loads will be placed as shown below:



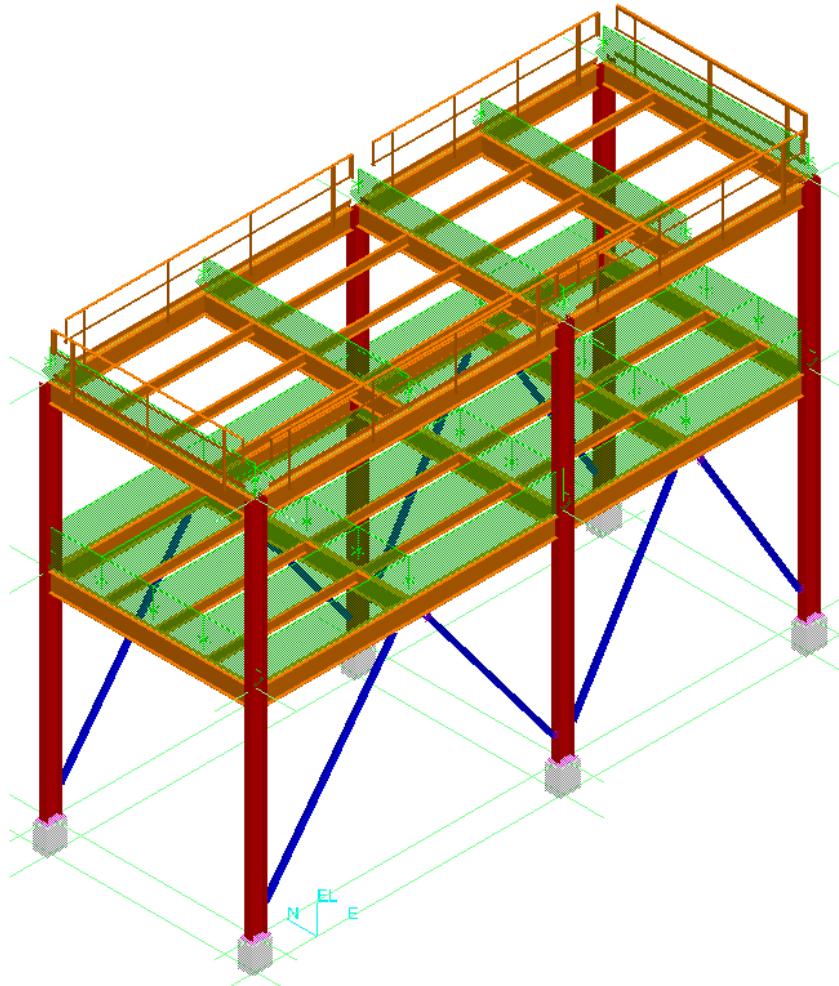
23. Define the following distributed load for the beams at Elevation 10m:

Load Case: DEAD
Load: Force Z
Reference: Global
Position as: Relative
Start Position: 0
End Position: 1
Start and End Magnitude: -1500N/m

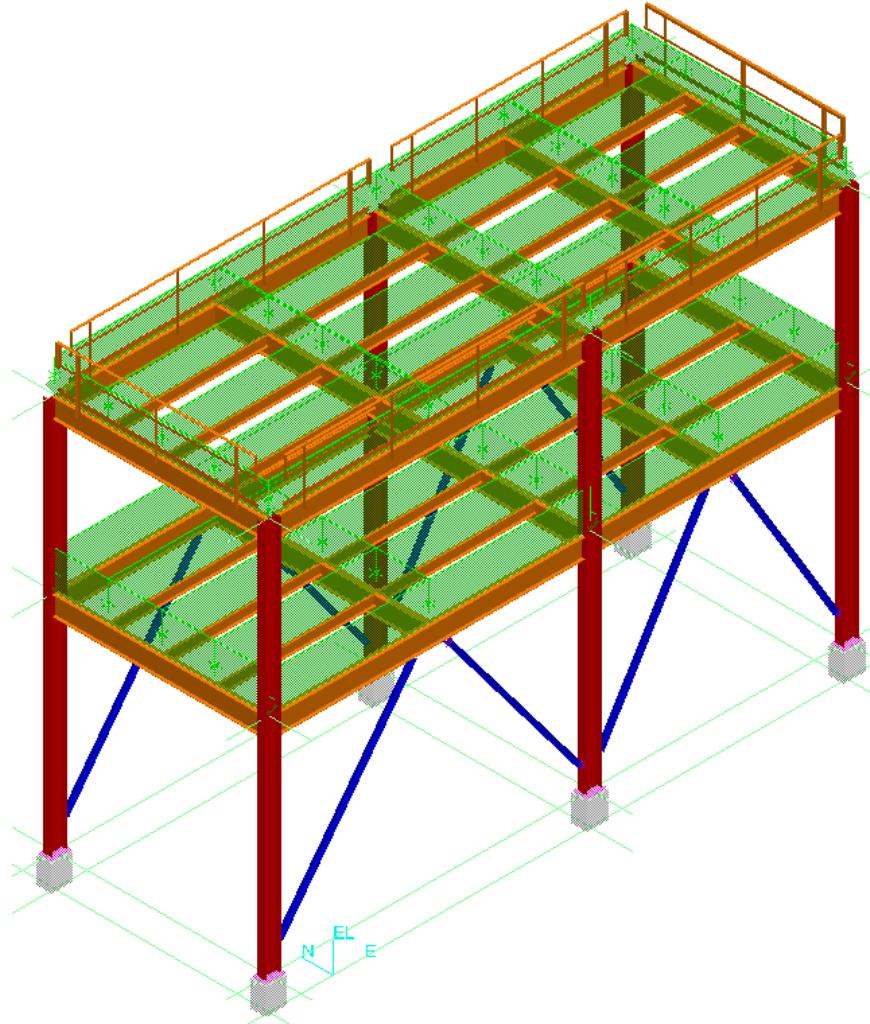
Load Case:	Load:	Reference:	Position As:	Start Position:	End Position:	Start Magnitude:	End Magnitude:
Dead	Force Z	Global	Relative	0	1	-1500.00 N/m	-1500.00 N/m

24. Select the beams running north/south at Elevation 10m and click Finish.

Loads will be placed as shown below:



25. Change the Magnitude to -3000N/m and apply this to the 4 primary beams running east/west at Elevation 10m.
26. Change the Load Case Name to Live and the Magnitude to -3600N/m and apply the load to the interior framing members and to the 4 primary beams running east/west at Elevation 10m.
27. Change the Magnitude to -1800N/m and apply the Live Load to the 3 beams running north/south at Elevation 10m.



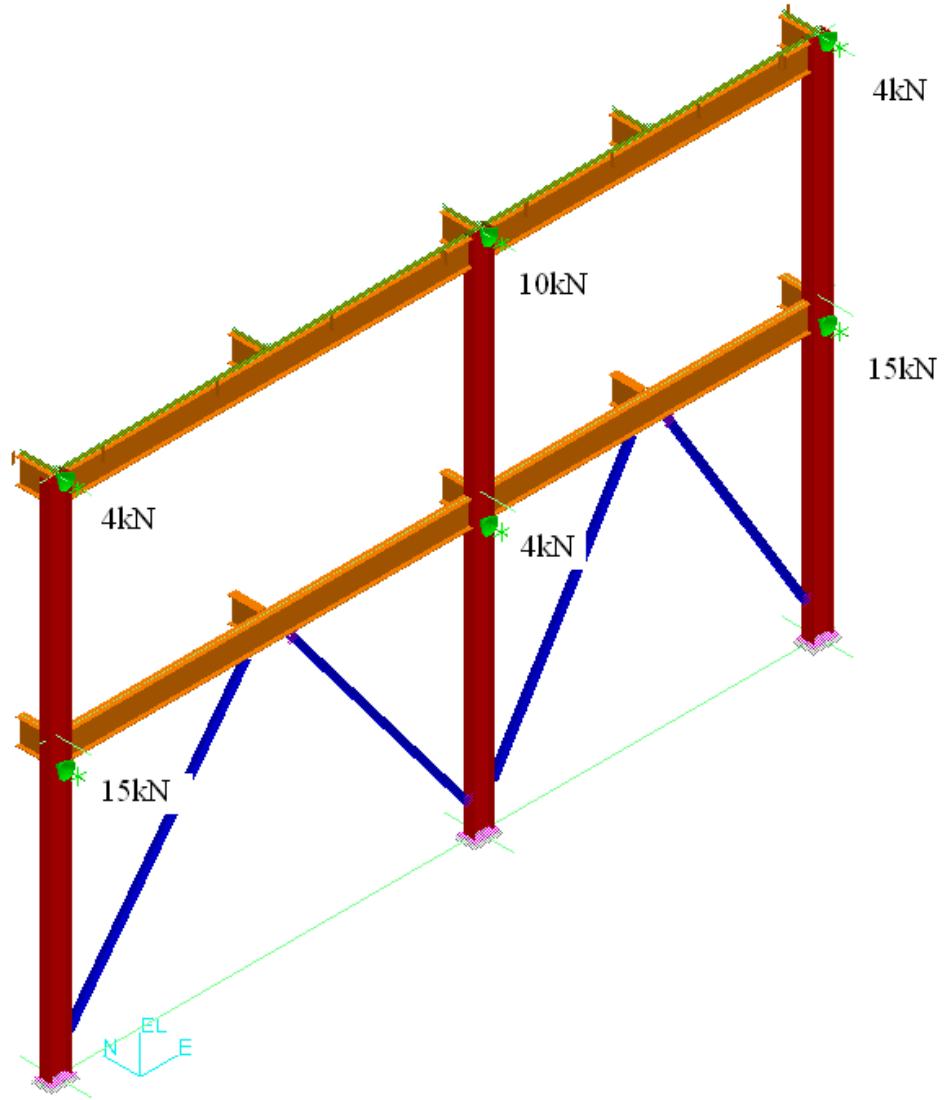
Part III: Place Wind Loads

28. Select Place New Concentrated Load Command
29. Define the load as:

Load Case: WIND
 Load: Force Y
 Reference: Global
 Position as: Relative

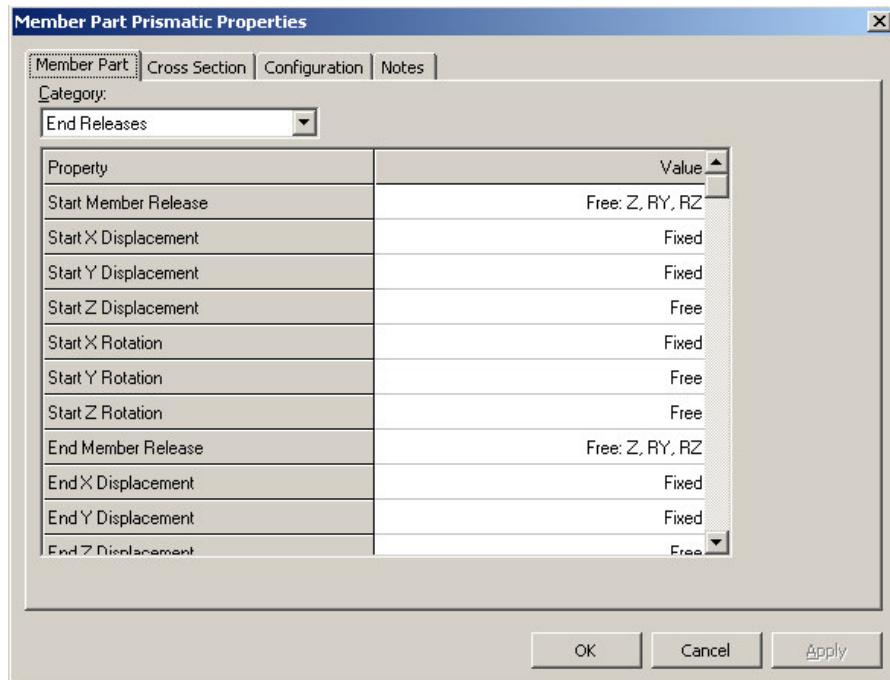
Load Case:	Load:	Reference:	Position As:	Position:	Magnitude:
Wind	Force Y	Global	Relative	1	4000.00 N

30. Place concentrated wind loads along the south side of the structure with the following magnitudes:



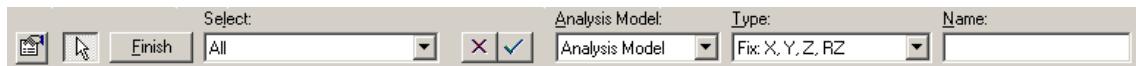
Part IV: Set Member Releases

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:Z, RY, RZ.



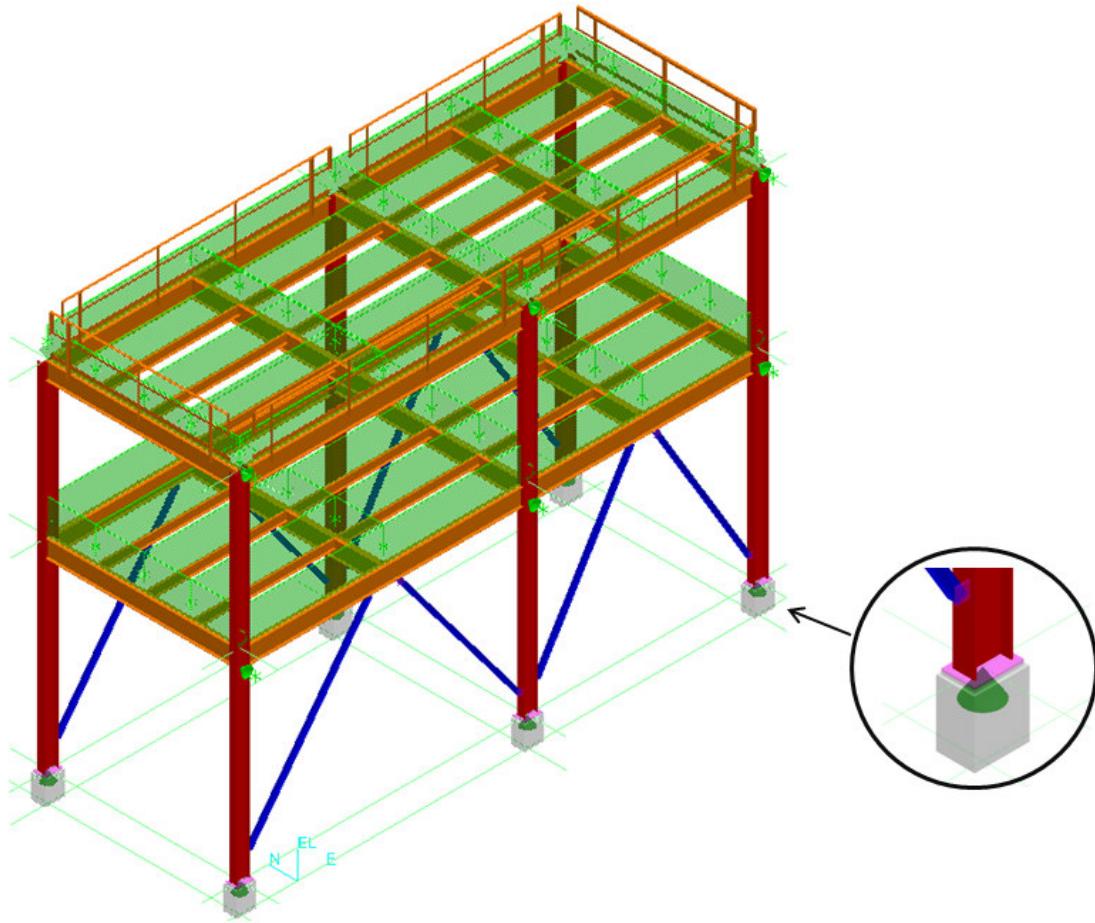
Part V: Place Boundary Conditions

35. Create a filter to include all member parts in U02. Select the Analysis Model in the Workspace Explorer. Apply the filter to the Analysis Model.
36. Select Set Boundary Condition Command .
37. Select Frame Connections at the bottom of the first floor columns
38. Set the Type to Fix: X,Y,Z,RZ



39. Select “Finish” button.

Boundary Conditions are placed as shown below:

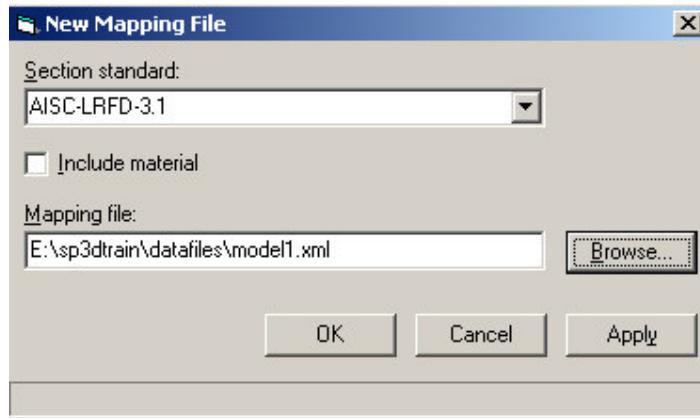


Part VI: Export Analytical Model and Update from Analysis

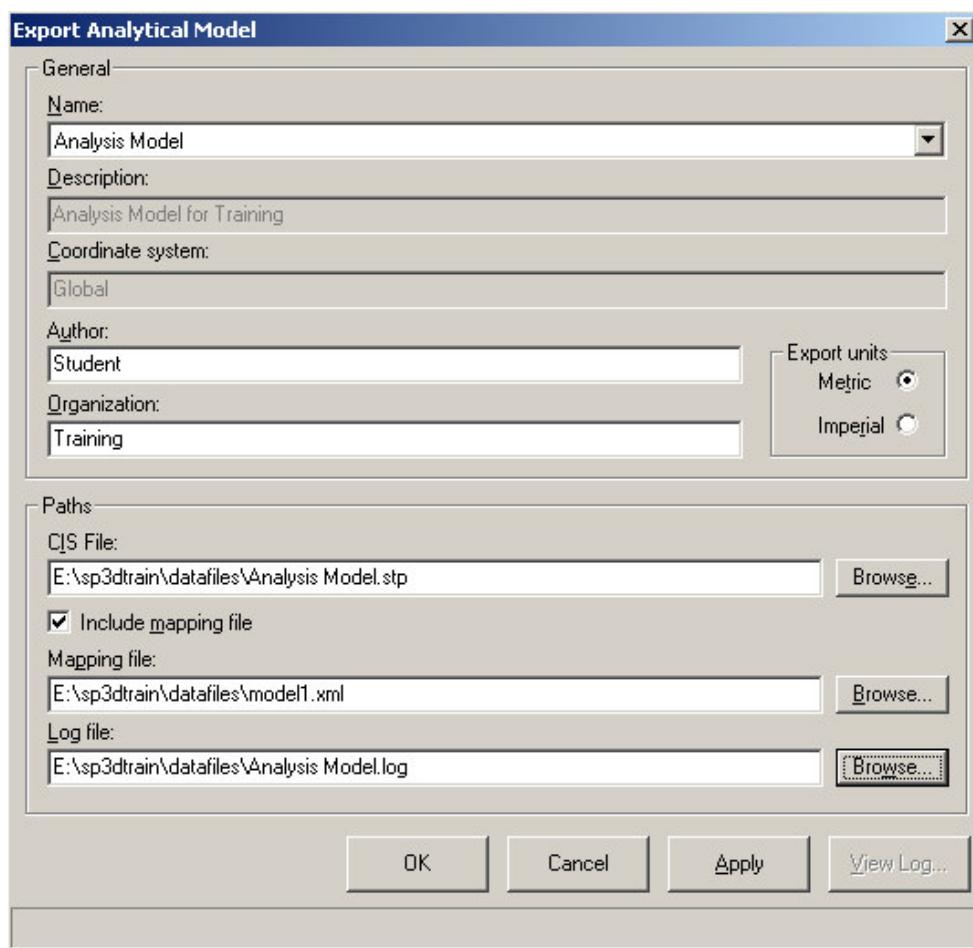
1. Select File -> New Mapping File

Note: First you need to create a mapping file. The mapping file must be used if the third party analysis program you are using calls the section standard a different name from what is delivered with SmartPlant 3D's mapping file. For example, STAAD calls its section standard 'AISC EDI Standard Nomenclature'. Therefore, the line in the mapping file that maps these two Section Standards must be changed from <Standard name="AISC-LRFD-3.0" externalname="AISC-LRFD-3.0". externalorganization="AISC" externaldate="2002" externalversion="3.0" /> to <Standard name="AISC-LRFD-3.0" externalname="AISC EDI Standard Nomenclature" externalorganization="AISC" externaldate="2002" externalversion="3.0" /> in order to get the section sizes updated.

2. Name the mapping file as follows:



3. Hit “OK” Button.
4. To create a CIS file, select File -> Export -> Analytical Model.
5. Set the followings:



6. Hit “OK” Button.
7. To update from a CIS file, select Update from Analysis/Design (Optional)