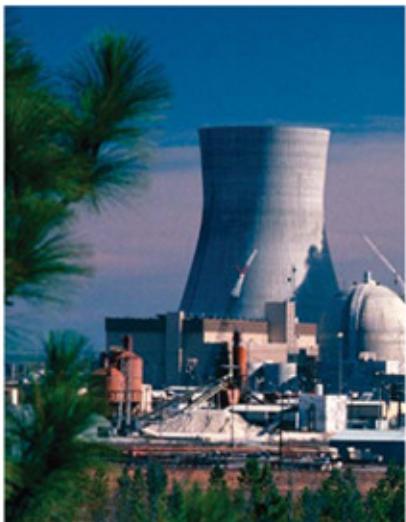


# SmartPlant 3D Grids /Structure/Solids

## *Student Workbook*

---

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Unpublished---rights reserved under the copyright laws of the United States.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

<b>GRIDS TASK .....</b>	<b>4</b>
LAB-1A: Placing Grids/Coordinate Systems - U02 .....	4
LAB-1B: Placing Grids – U04.....	10
LAB-1C: Placing Grids – U03.....	15
LAB-2: Editing Grid Planes .....	19
LAB-3: Offshore Coordinate System - U05 .....	23
LAB-4: Rotated Grid Coordinate System - U06.....	29
<b>STRUCTURE TASK.....</b>	<b>32</b>
<b>MEMBERS</b>	
LAB-1A: Linear Member System - Structure U04.....	32
LAB-1B: Copy/Paste Members - Structure U02 .....	40
LAB-1C: Productivity Commands - Structure U02.....	45
LAB-2: Structure Modeling - Building U03 .....	51
LAB-3: Curve Member System - Building U03 .....	68
LAB-4: Frame Connections .....	77
LAB-5: Assembly Connections .....	82
LAB-6: Fireproofing .....	87
<b>CONCRETE</b>	
LAB-7: Slabs .....	111
LAB-8: Walls.....	124
LAB-9: Openings.....	147
<b>MISCELLANEOUS</b>	
LAB-10: Stairs / Ladders / Handrails .....	151
LAB-11: Footings .....	166
LAB-12: Equipment Foundations.....	168
<b>SOLIDS</b>	
LAB-13: Designed Solids.....	171
<b>STRUCTURE REPORTS</b>	
LAB-14: Reports.....	196
<b>OPTIONAL (MEMBERS / CUSTOM COMMANDS / DATA EXCHANGE)</b>	
LAB-15: Structural Modeling (offshore Jacket) - Optional .....	198
LAB-16: Using Building Wizard (Optional) .....	205
LAB-17: Data Exchange for Solids (Optional) .....	211
<b>STRUCTURAL ANALYSIS TASK.....</b>	<b>218</b>
LAB-1: Loads, Releases, Boundary Conditions and Creating a CIS file .....	218

# GRIDS TASK

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

### Objectives

After completing this lab, you will be able to:

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

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

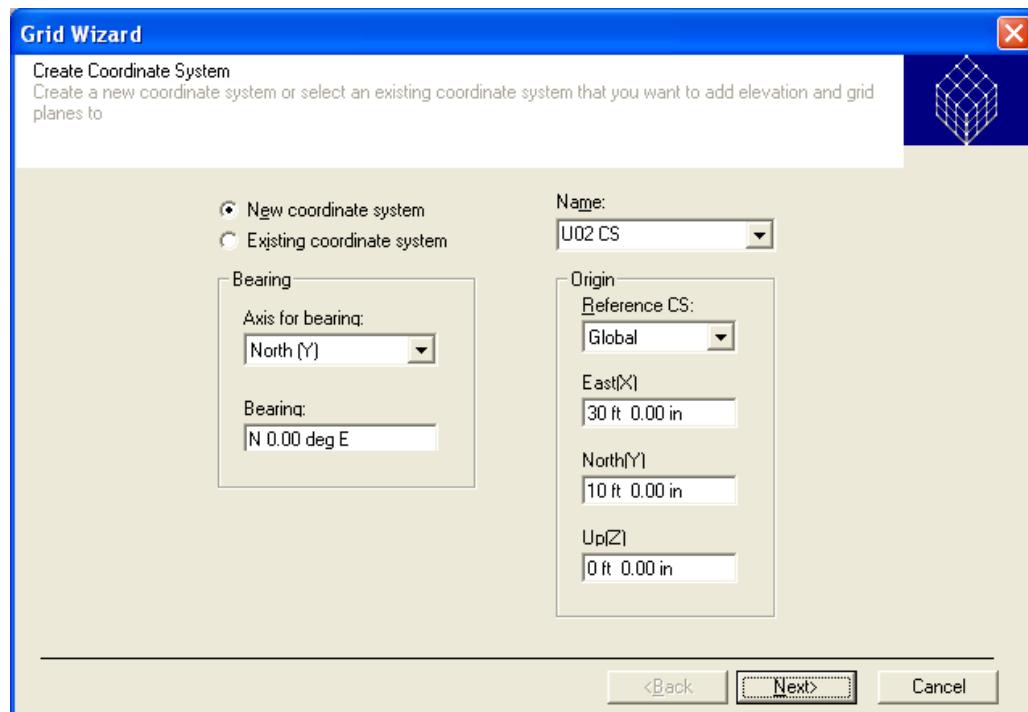
*Note: Session file stores settings from the last time you were in SmartPlant 3D. The name of the current session file appears in the title bar of the application, along with the name of the task, model and filter. One of the settings saved in the session file is the workspace. See the previous common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

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

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

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

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



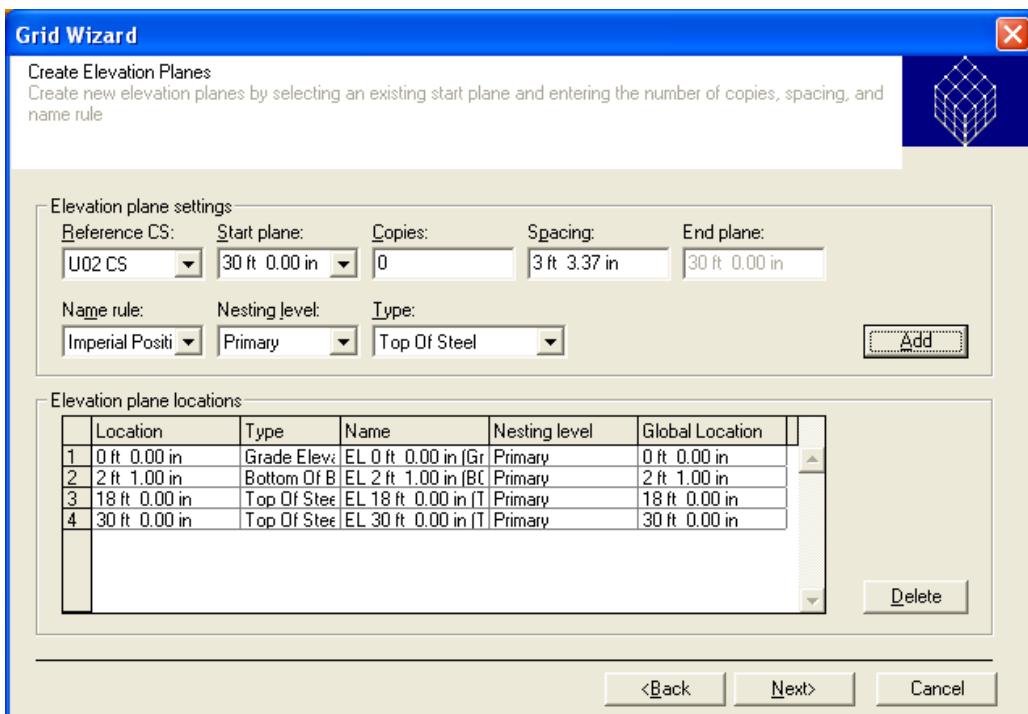
Next, click “Next” button.

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

Set the Nesting level to Primary for All Elevation Planes.

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

Select the “Add” button.

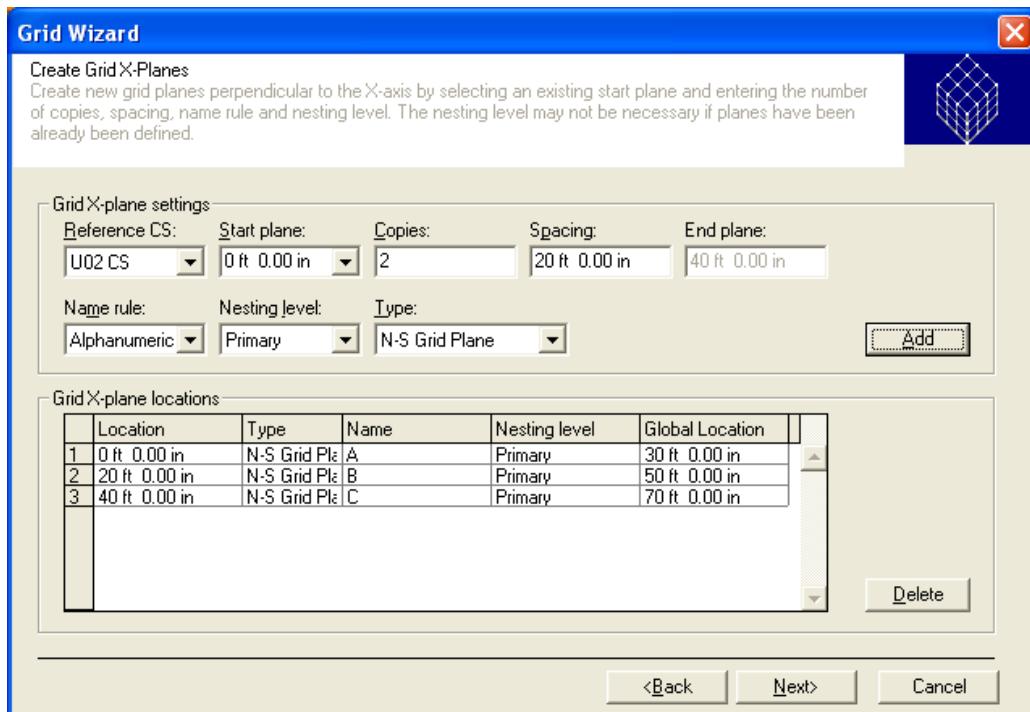


Next, click “Next” button.

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

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

Select the “Add” button.



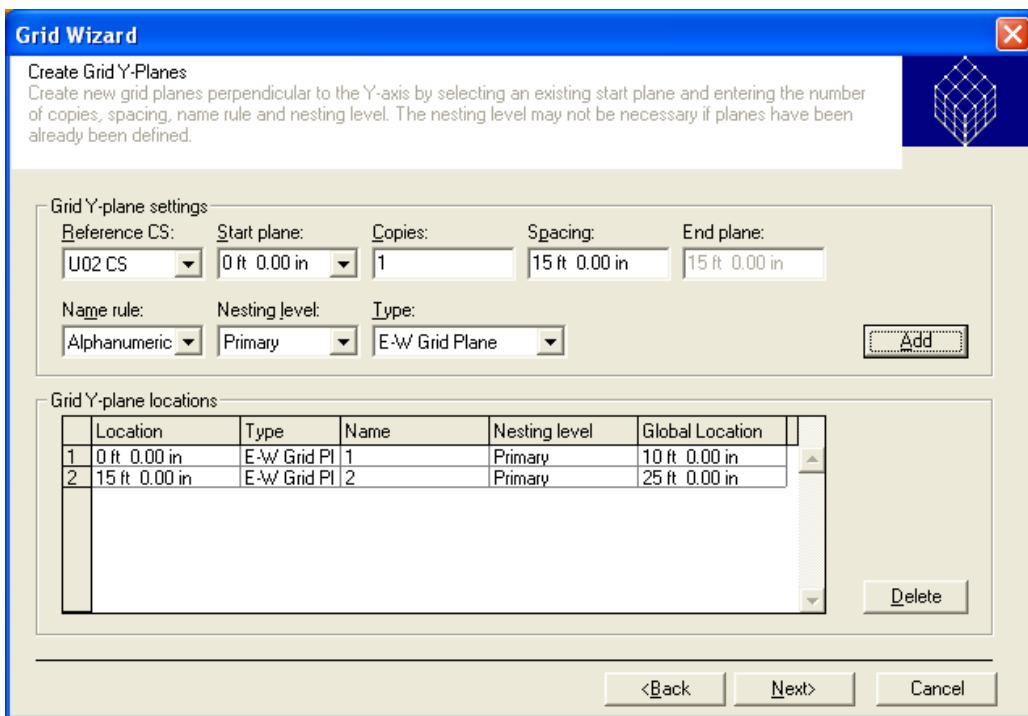
Next, click “Next” button.

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

Set the Name rule to Alphanumeric and Percent for All Grid Planes.

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

Select the “Add” button.



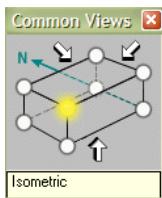
Next, click “Next” button.

8. Hit “Next” button on Create Radial Cylinder form.
9. Hit “Next” button on Create Radial Plane form.

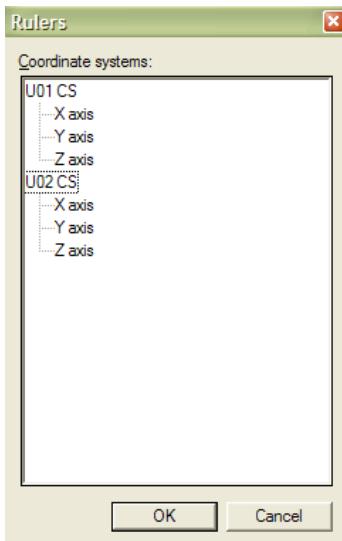
*Note: We are skipping these two forms because we are not placing a Radial Grid set.*

10. Hit “Finish” button on Associated Elevation Plane form. The wizard will create the grids lines at all elevations.

11. Use the Fit command from the main toolbar to fit all graphics into the view.
12. Use the Common View control to look at an ISO view. Select the yellow corner indicated below.

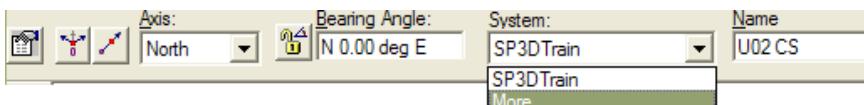


13. Go to the main menu and select View -> Ruler option to open the ruler dialog box.
14. Hold the <Control> key down and de-highlight the coordinate system rulers called U02 CS. Select the “OK” button.

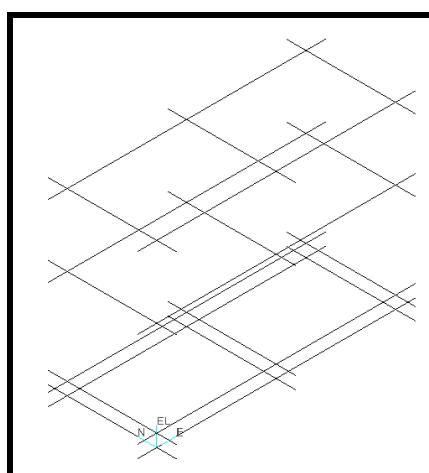


15. Select the U02 CS object in the Workspace Explorer to open the edit coordinate system ribbon bar.
16. Change the Parent System to CS. Select the “More” option to open “Select System” dialog box. Select CS in the system hierarchy tree. Hit “Ok” button.

*Note: Objects that you place directly in the model are associated to a system in the System Hierarchy. Therefore, you are responsible for making sure the object is associated to the appropriate System.*



Your View should now resemble the following graphic.



# LAB-1B: Placing Grids – U04

## Objectives

After completing this lab, you will be able to:

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

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

Name: U04 CS  
 Axis for bearing: North (Y)  
 Bearing: N 0.00 deg E  
 Origin  
 East (X): 120 ft  
 North (Y): 80 ft  
 Up (Z): 0 ft

Next, click “Next” button

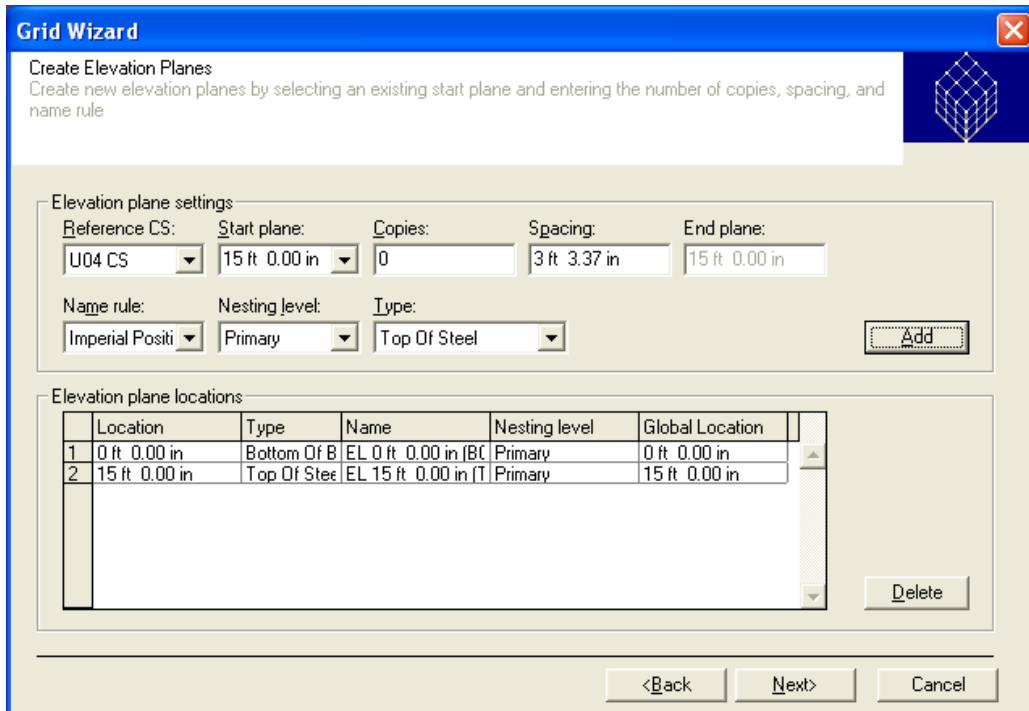
5. **Elevation Planes:** Using Step 2 in the Grid Wizard command, create the Elevation Planes based on the following information:

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

Set the Nesting level to Primary for All Elevation Planes.

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

Select the “Add” button.



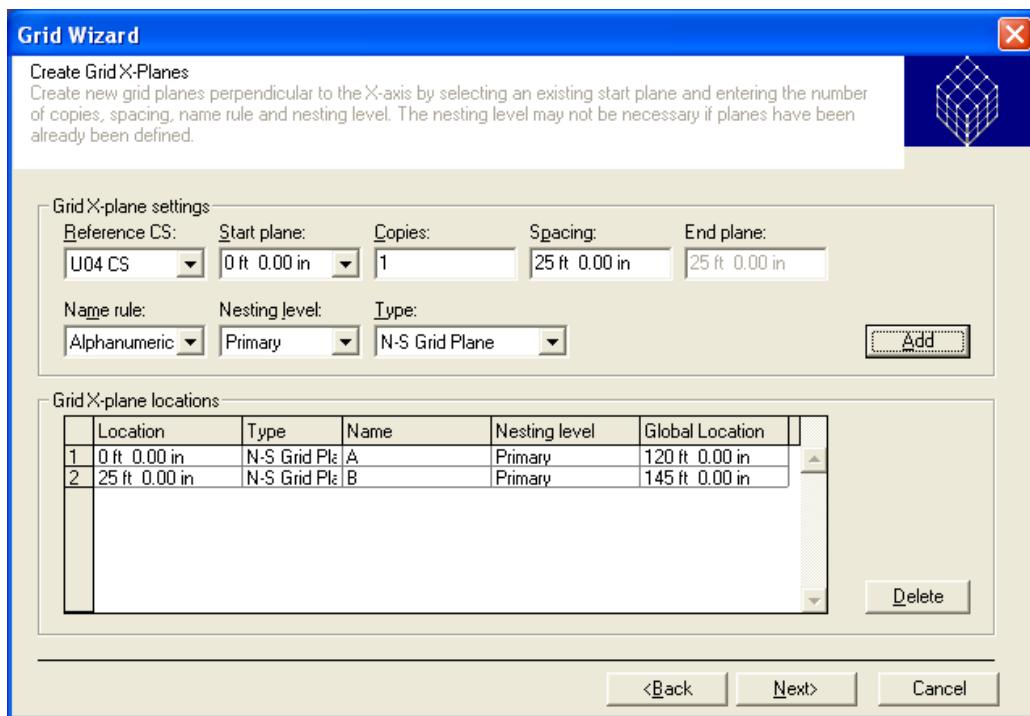
Next, click “Next” button.

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

Set the Name rule to Alphanumeric and Percent for All Grid Planes.

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

Select the “Add” button.

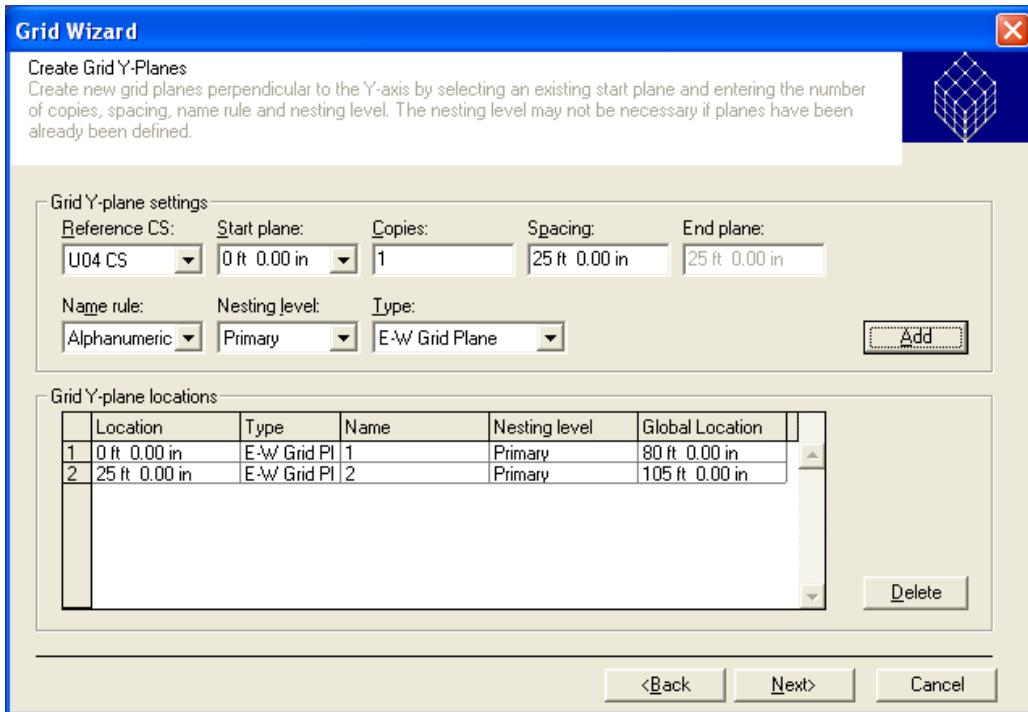


Next, click “Next” button.

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

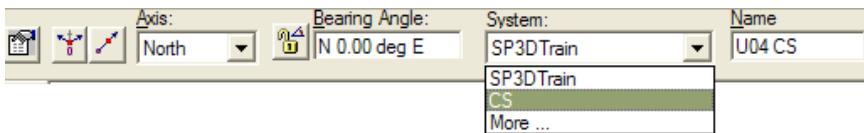
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	1	25 ft 0 in	25 ft 0 in	Primary	E-W Grid Plane

Select the “Add” button.

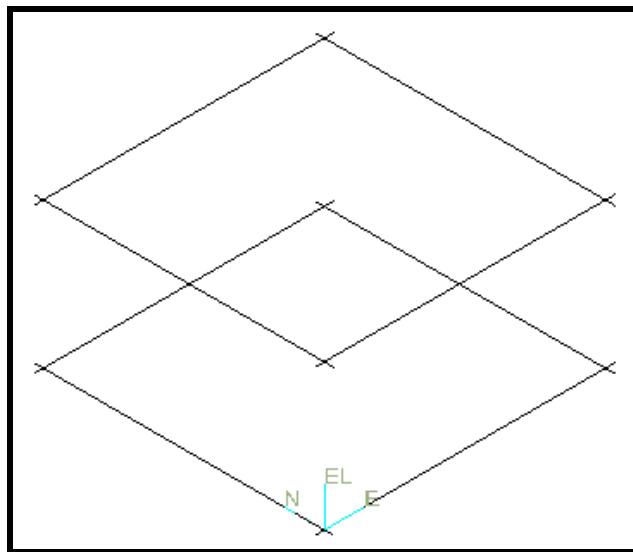


Next, click “Next” button.

8. Hit “Next” button on Create Radial Cylinder form.
9. Hit “Next” button on Create Radial Plane form.
10. Hit “Finish” button on Associated Elevation Plane form.
11. Use the Fit command from the main toolbar to fit all graphics into the view.
12. Go to the main menu and select View -> Ruler option to open the ruler dialog box.
13. Press the <Control> key and unselect all items. Select the “OK” button to turn off the rulers.
14. Select the U04 CS to open the edit coordinate system ribbon bar.
15. Change the Parent System to CS.



16. Your View should now resemble the following graphic.



# LAB-1C: Placing Grids – U03

## Objectives

After completing this lab, you will be able to:

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

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

Name: U03 CS  
 Axis for Bearing: North (Y)  
 Bearing: N 0.00 deg E

Origin

East (X): -35 ft  
 North (Y): 10 ft  
 Up (Z): 0 ft

Next, click “Next” button.

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

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

Select the “Add” button.

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

Elevation plane locations					
	Location	Type	Name	Nesting level	Global Location
1	0 ft 0.00 in	Grade Elev:	EL 0 ft 0.00 in (Gr)	Primary	0 ft 0.00 in
2	18 ft 0.00 in	Top Of Stee	EL 18 ft 0.00 in (T)	Primary	18 ft 0.00 in
3	31 ft 0.00 in	Top Of Stee	EL 31 ft 0.00 in (T)	Primary	31 ft 0.00 in
4	34 ft 0.00 in	Column Splt	EL 34 ft 0.00 in (S)	Primary	34 ft 0.00 in
5	44 ft 0.00 in	Top Of Stee	EL 44 ft 0.00 in (T)	Primary	44 ft 0.00 in

Next, click “Next” button.

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

Set the Name rule to Alphanumeric and Percent for All Grid Planes.

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

Select the “Add” button.

**Grid Wizard**

Create Grid X-Planes  
Create new grid planes perpendicular to the X-axis by selecting an existing start plane and entering the number of copies, spacing, name rule and nesting level. The nesting level may not be necessary if planes have been already been defined.

**Grid X-plane settings**

Reference CS:	Start plane:	Copies:	Spacing:	End plane:
U03 CS	31 ft 0.00 in	1	24 ft 0.00 in	55 ft 0.00 in

Name rule: Nesting level: Type:  
Alphanumeric Primary N-S Grid Plane Add

**Grid X-plane locations**

	Location	Type	Name	Nesting level	Global Location
1	0 ft 0.00 in	N-S Grid Plk	A	Primary	-35 ft 0.00 in
2	8 ft 0.00 in	N-S Grid Plk	B	Primary	-27 ft 0.00 in
3	31 ft 0.00 in	N-S Grid Plk	C	Primary	-4 ft 0.00 in
4	55 ft 0.00 in	N-S Grid Plk	D	Primary	20 ft 0.00 in

Delete

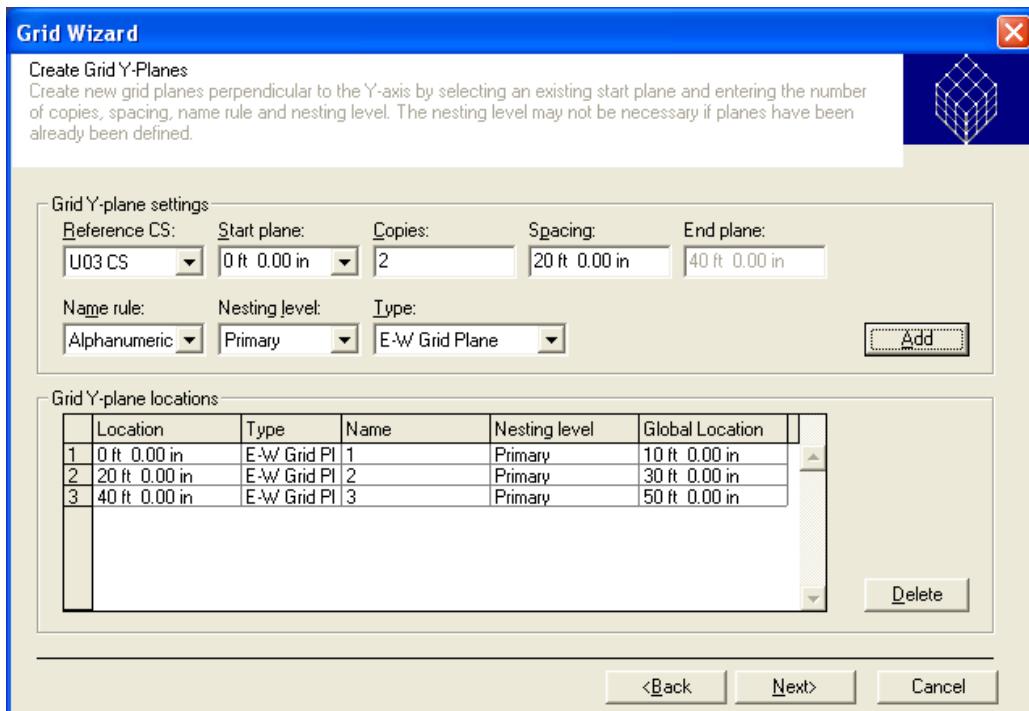
<Back Next Cancel

Next, click “Next” button.

8. **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.

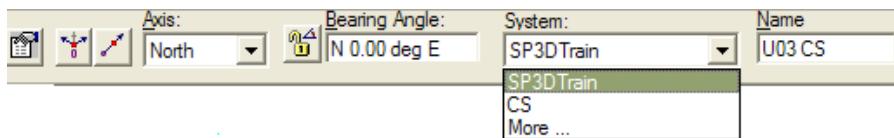
Step	Start Plane	Copies	Spacing	End Plane	Nesting Level	Type
1	0 ft 0 in	2	20 ft 0 in	40 ft 0 in	Primary	E-W Grid Plane

Select the “Add” button.

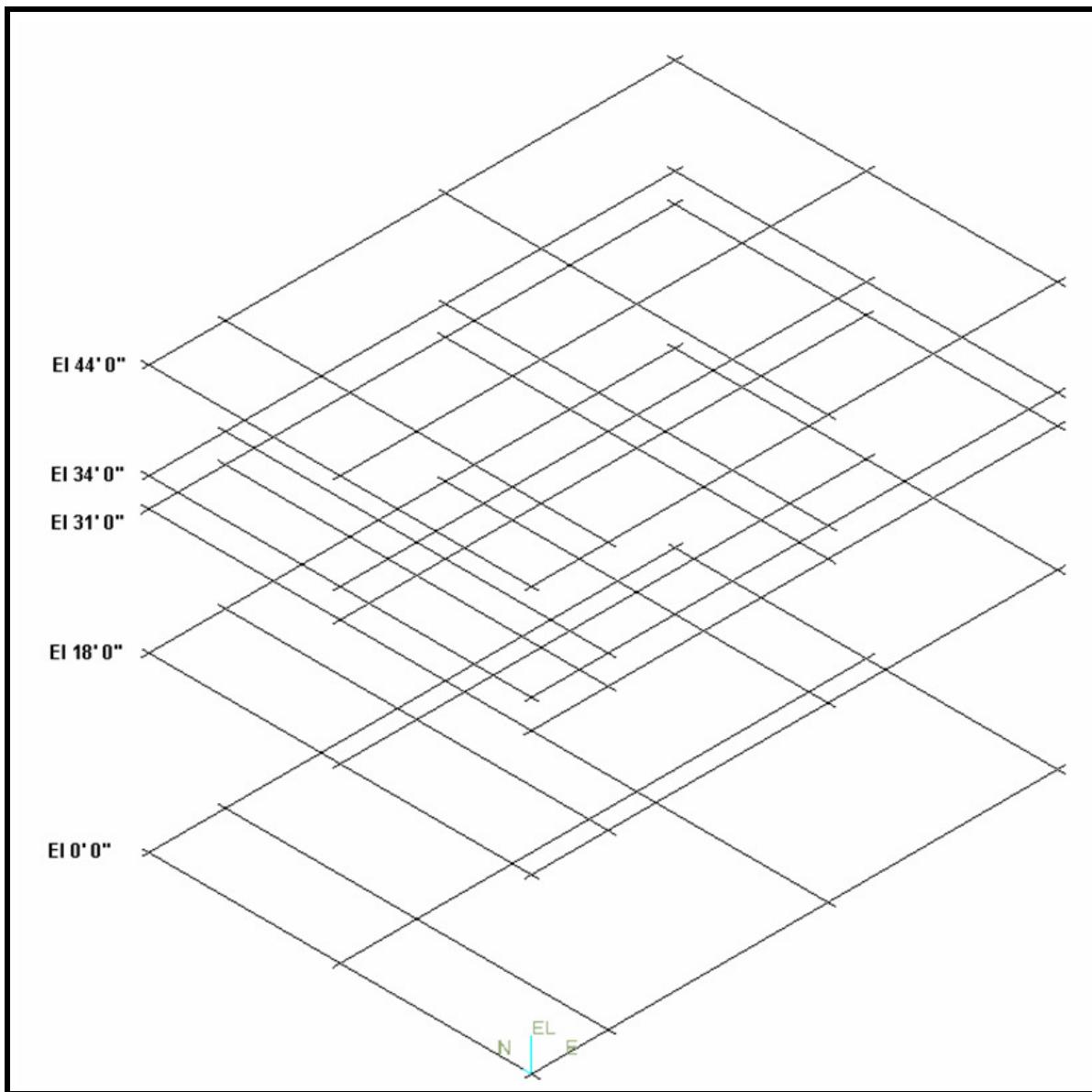


Next, click “Next” button.

9. Hit “Next” button on Create Radial Cylinder form.
10. Hit “Next” button on Create Radial Plane form.
11. Hit “Finish” button on Associated Elevation Plane form.
12. Go to the main menu and select View -> Ruler option to open the ruler dialog box.
13. Press the <Control> key and unselect all items. Select the “OK” button to turn off the rulers.
14. Select the U03 CS to open the edit coordinate system ribbon bar.
15. Change the Parent System to CS.



Your View should now resemble the following graphic.



# LAB-2: Editing Grid Planes

## Objectives

After completing this lab, you will be able to:

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

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

### Placing Elevation Plane:

4. Select Place Elevation Plane command button from the vertical toolbar.
5. In the Elevation Plane horizontal ribbon bar enter the following parameters:



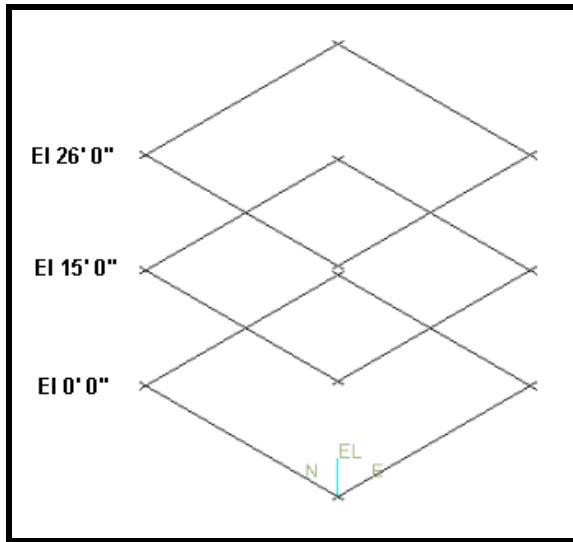
Coordinate System:	Pick U04 CS
Type:	Pick Top of Steel
Name:	Open the setting properties page and change the name rule to Imperial Position
Nesting level:	Primary
Reference:	Origin
Copies:	0

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

Offset value: 26 ft

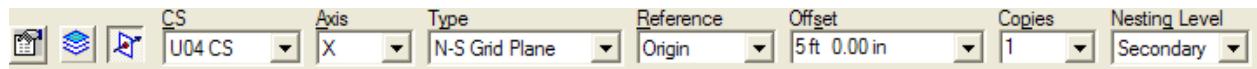
7. The system places the elevation plane marker in the Ruler after entering the above parameters. Right mouse click to exit the command.

Your View should now resemble the following graphic.



### Placing Grid Planes: X-Axis

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

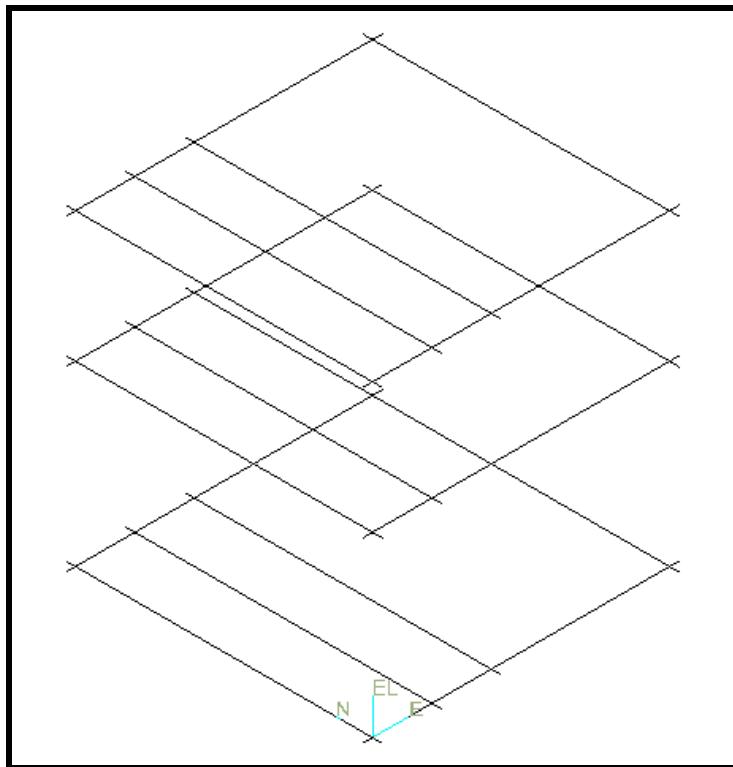


Coordinate System:	U04 CS
Axis:	X
Type:	N-S Grid Plane
Reference:	Origin
Copies:	1
Nesting Level:	Secondary
Name:	Open the properties page and change the name rule to Alphanumeric and Percent

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

Offset value: 5 ft

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



### Placing Grid Planes: Y-Axis

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



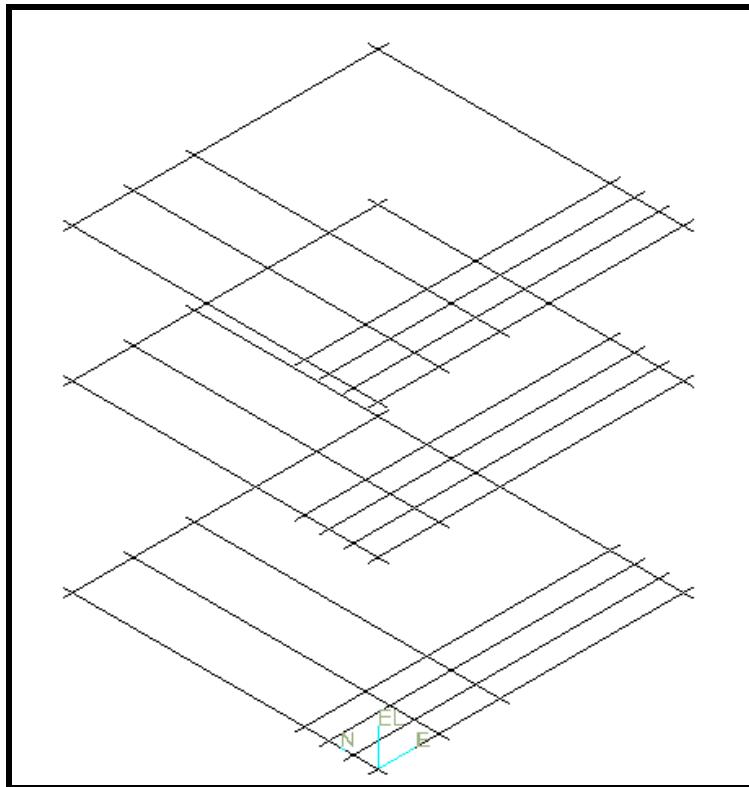
Coordinate System: U04 CS  
 Type: E-W Grid Plane  
 Reference: Origin  
 Copies: 2  
 Nesting Level: Secondary  
 Name: Open the properties page and change the name rule to Alphanumeric and Percent

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

Offset value: 2 ft

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

Your View should now resemble the following graphic.



# LAB-3: Offshore Coordinate System - U05

## Objectives

After completing this lab, you will be able to:

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

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

Name: U05 CS  
 Axis for Bearing: North (Y)  
 Bearing: N 0.00 deg E

Origin:  
 East (X): -35 ft  
 North (Y): -30 ft  
 Up (Z): 0 ft

Next, click “Next” button.

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

Use Imperial Position as Name Rule.

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

Next, click “Next” button.

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

Use Alphanumeric and Percent as Name Rule.

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

Next, click “Next” button.

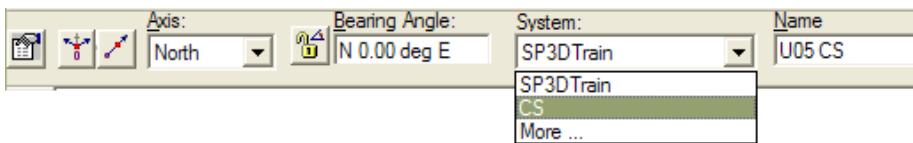
7. Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

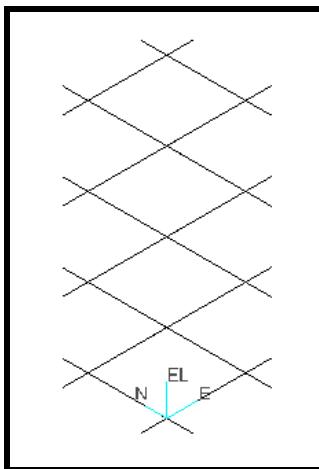
Use Alphanumeric and Percent as Name Rule.

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

Next, click “Next” button.

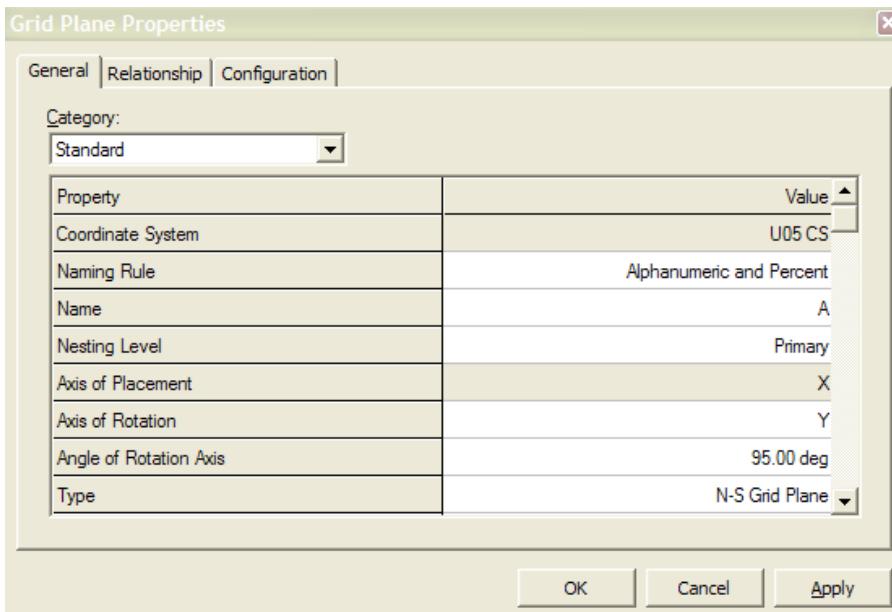
8. Hit “Next” button on Create Radial Cylinder form.
9. Hit “Next” button on Create Radial Plane form.
10. Select the “Finish” Button.
11. Turn off all Rulers.
12. Select the U05 CS to open the edit coordinate system ribbon bar.
13. Change the Parent System to CS.





14. Select Grid Plane A in the Workspace Explorer to open the Edit Grid Plane ribbon bar.
15. Select the properties icon to open the properties page.
16. In the Grid Plane Properties page enter the following parameters:

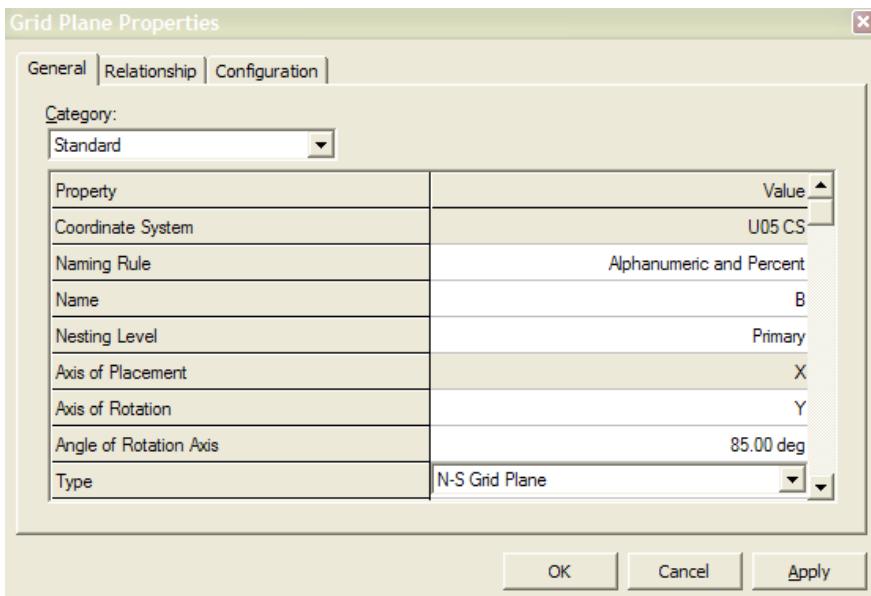
Axis of Rotation: Y  
 Angle of Rotation: 95.0 deg



17. Hit “Apply” button to commit the transaction.
18. Select Grid Plane B in the Workspace Explorer.
19. Select the properties icon to open the properties page.

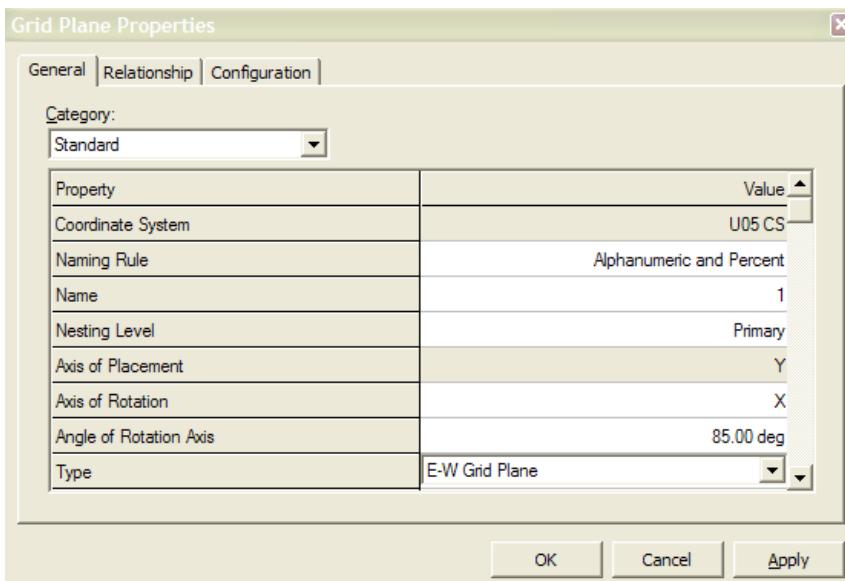
20. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation: Y  
Angle of Rotation: 85.0 deg



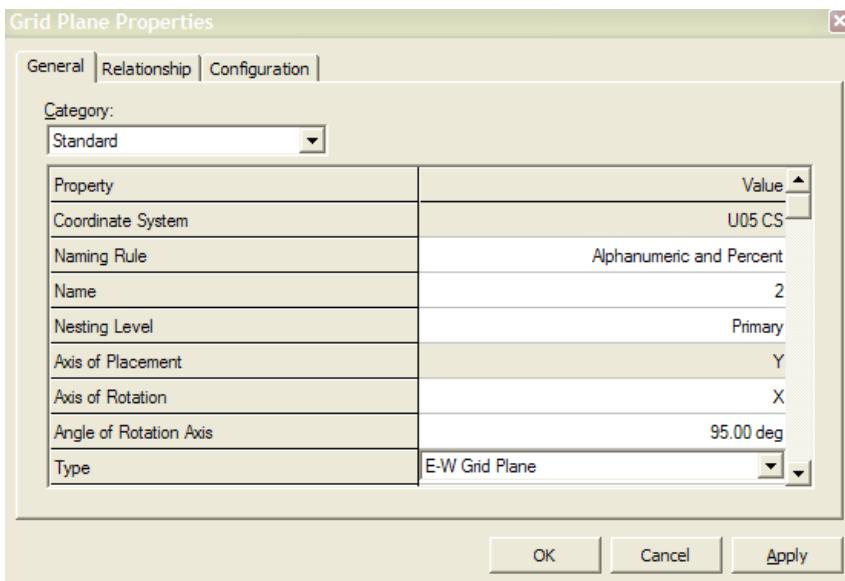
21. Hit “Apply” button to commit the transaction.  
22. Select Grid Plane 1 in the Workspace Explorer.  
23. Select the properties icon to open the properties page.  
24. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation: X  
Angle of Rotation: 85.0 deg



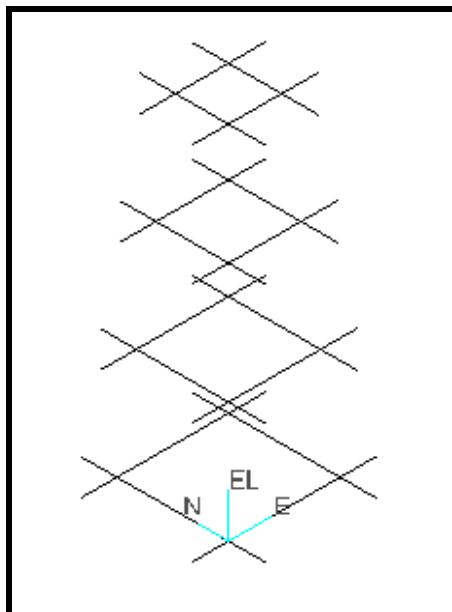
25. Hit “Apply” button to commit the transaction.
26. Select Grid Plane 2 in the Workspace Explorer.
27. Select the properties icon to open the properties page.
28. In the Grid Plane Properties page enter the following parameters:

Axis of Rotation: X  
 Angle of Rotation: 95.0 deg



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

Your View should now resemble the following graphic.



# LAB-4: Rotated Grid Coordinate System - U06

## Objective

After completing this lab, you will be able to:

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

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

Name: U06 CS

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

Origin:  
East (X): 20 ft  
North (Y): -30 ft  
Up (Z): 0 ft

Next, click “Next” button.

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

Use Imperial Position as Name Rule.

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

Next, click “Next” button.

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

Use Alphanumeric and Percent as Name Rule.

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

Next, click “Next” button.

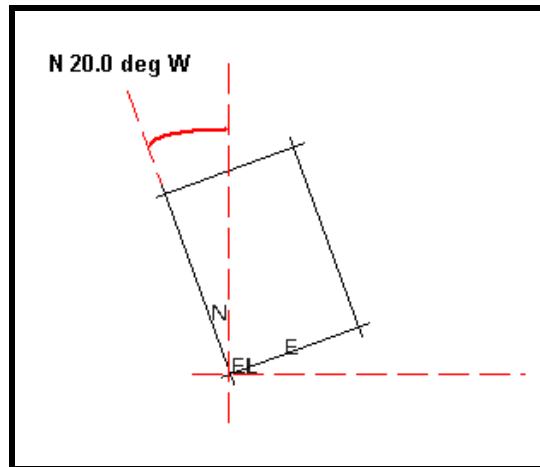
- Using Step 4 in the Grid Wizard command, create the Grid-Y Planes based on the following information:

Use Alphanumeric and Percent as Name Rule.

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

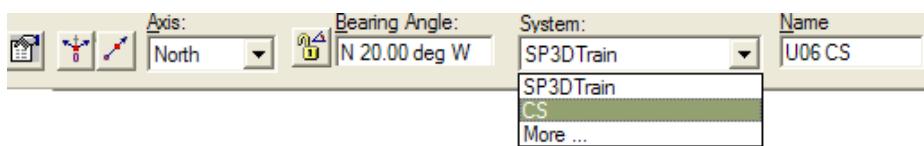
Next, click “Next” button.

- Hit “Next” button on Create Radial Cylinder form.
- Hit “Next” button on Create Radial Plane form.
- Hit “Finish” button on Associated Elevation Plane form.
- Your View should now resemble the following graphic.



**PLAN VIEW**

- Turn off all Rulers.
- Select the U06 CS to open the edit coordinate system ribbon bar.
- Change the Parent System to CS.



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

# STRUCTURE TASK

## LAB-1A: Linear Member System - Structure U04

### Objectives

After completing this lab, you will be able to:

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

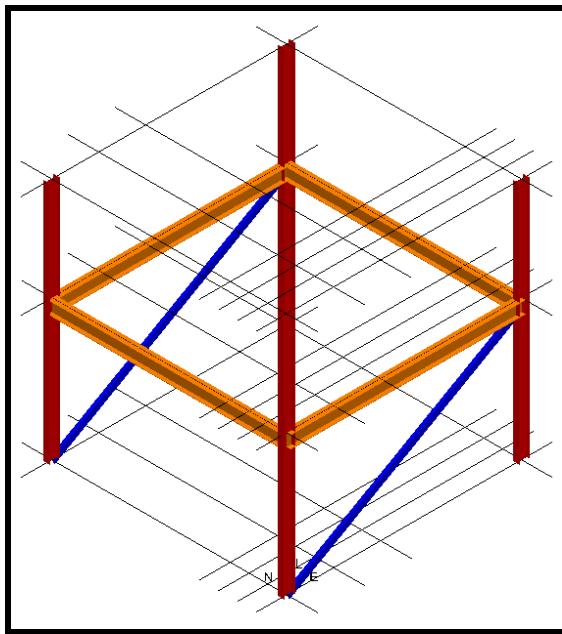
*Note: Use the grid system to help you place the structural members. Though not absolutely necessary, grids help in the modeling process by providing SmartSketch intersection points at which structural members can be placed. You may also want to make a relationship with gridlines so that if a gridline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

#### Placing Columns / Beams / Vertical Braces

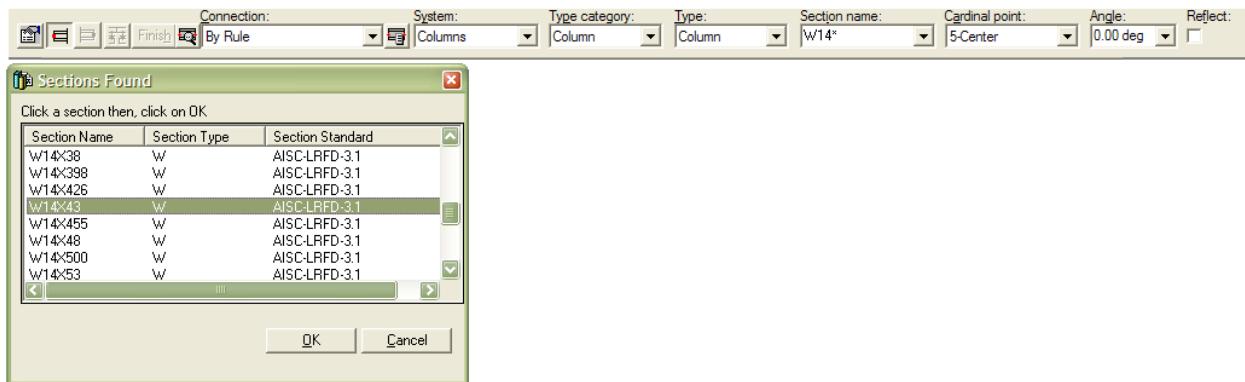
1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include A2 -> U04 and CS -> U04 CS systems.

*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:



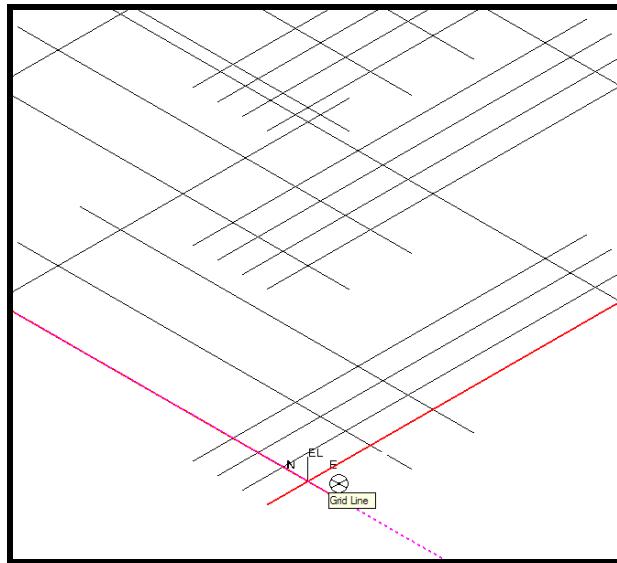
- a. Connection = “By Rule”
- b. System = A2 -> U04 -> Structural -> Columns
- c. Type Category = Column
- d. Type = Column
- e. Section Name = W14x43

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

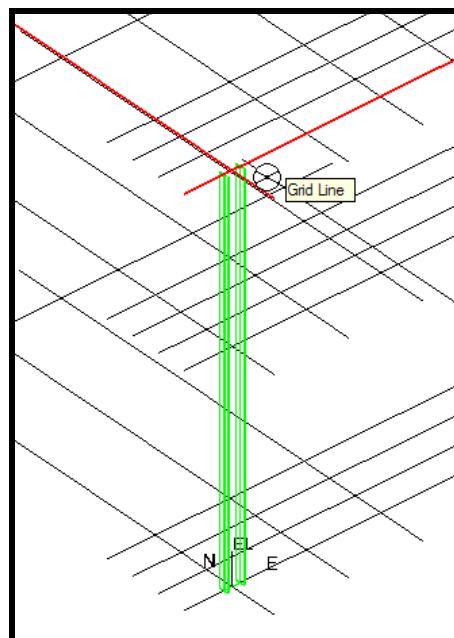
- f. Cardinal Pt. = 5
- g. Angle = 0
- h. Reflect = Off

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

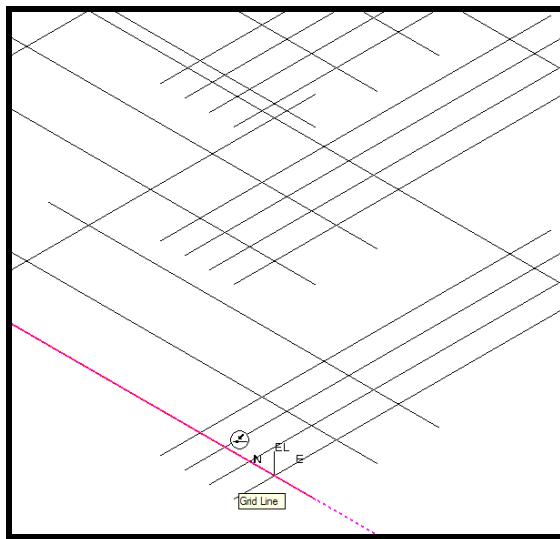
- i. Material: Steel- Carbon
  - j. Grade A36
7. For “End 1” of the column, move your cursor over the intersection of the two grid lines. A glyph will appear that indicates the intersection of two lines. See figure below. Then give a Left Mouse click.



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



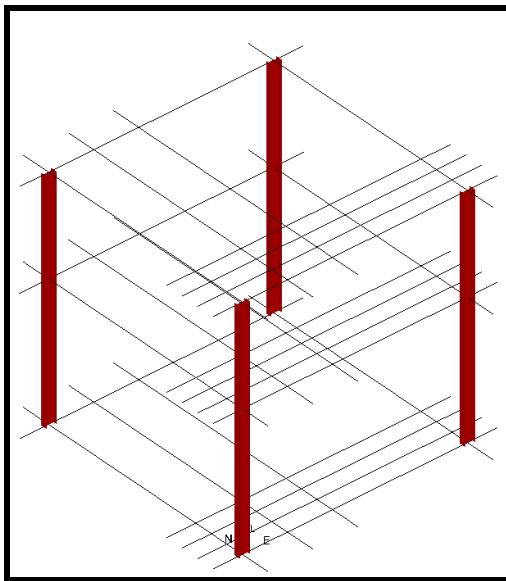
Note: An *easier* method of selecting an intersection is to use your middle mouse button to select one of the grids. When you do this a Purple infinite line appears telling you that you are locked onto the grid line. You can then use the perpendicular grid to indicate the intersection. See figure below.



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:



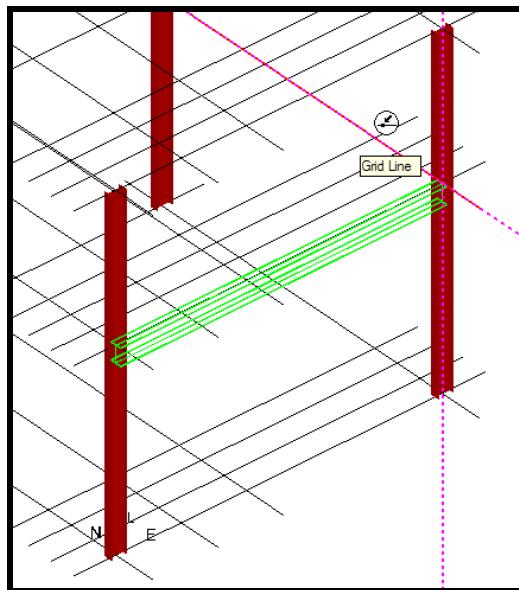
10. While still in the Place Linear Member System Command change the ribbon bar controls to:

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

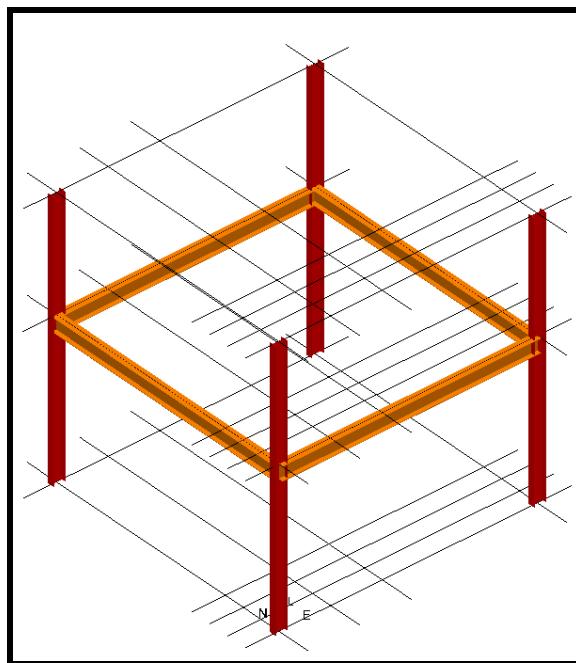
*Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.*

- f. Material: Steel- Carbon
- g. Grade A36

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



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



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

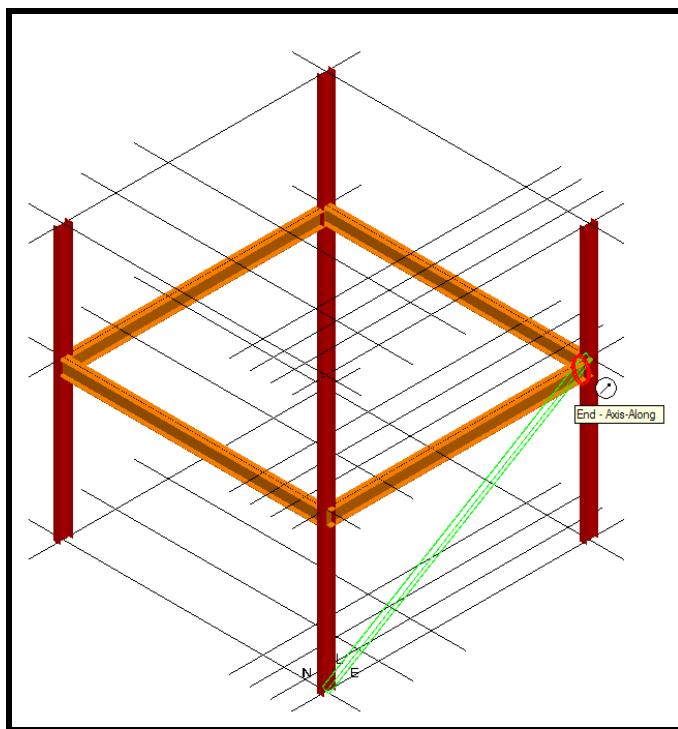
- h. System = A2 -> U04 -> Structural -> Vertical Braces
- i. Type Category = Brace
- j. Type = Vertical Brace
- k. Section Name = L8x4x1/2
- l. Cardinal Pt. = 1

m. Angle = 180 degrees

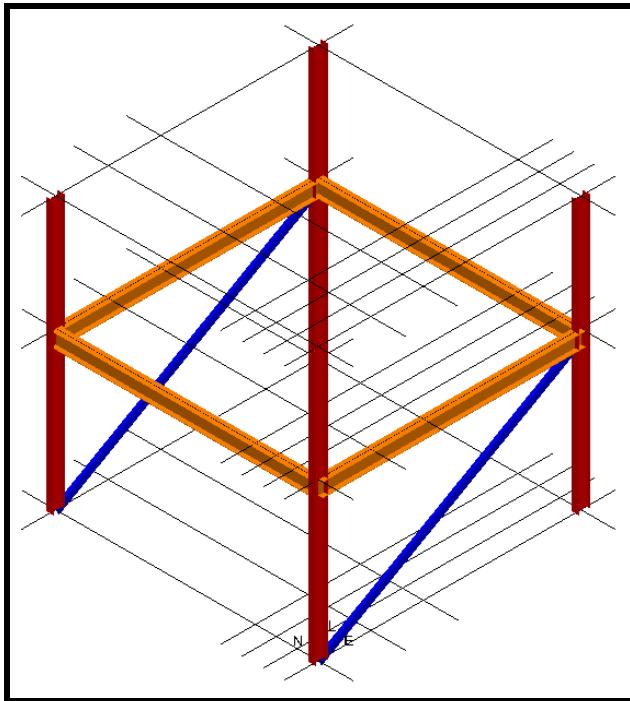
*Note: Use the properties icon in the ribbon bar to open the properties page. Select the Cross section Tab.*

- 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.



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

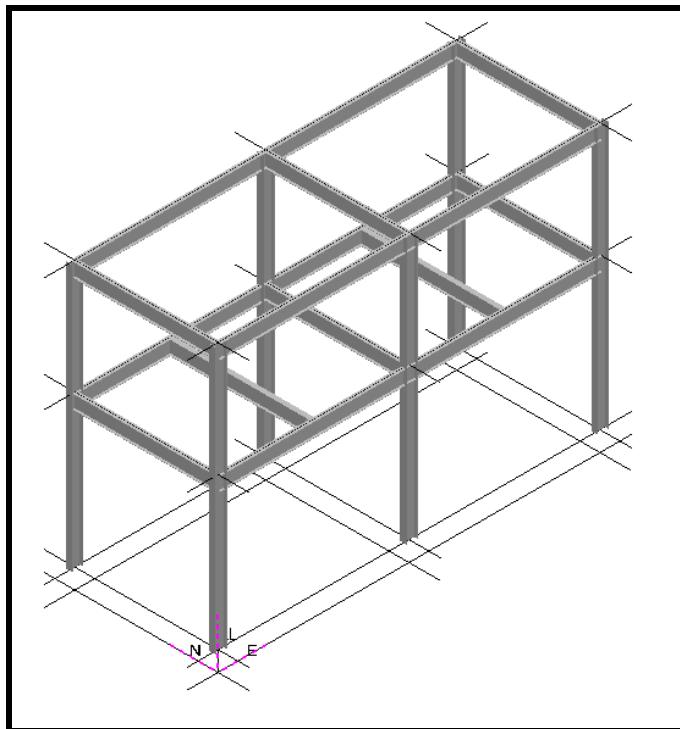
# LAB-1B: Copy/Paste Members - Structure U02

## Objectives

After completing this lab, you will be able to:

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

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



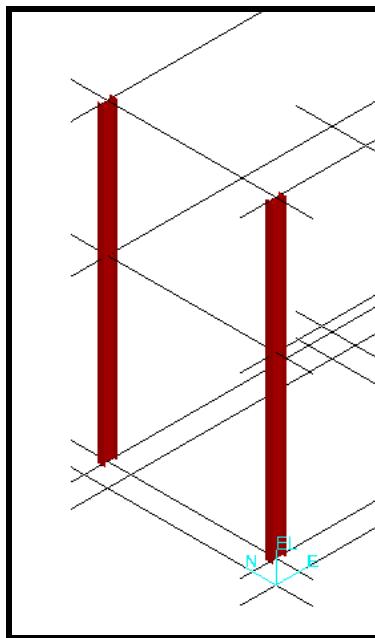
### Placing Columns and Beams

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

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

6. Place the first column end at the grid intersections Elevation 2' 1".
7. Place the second column end at the grid intersections Elevation 30' 0".
8. Repeat the above steps to place the other supporting column.



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

Connection:	By Rule
System:	A2 -> U02 -> Structural -> Beams

Type Category:	Beam
Type:	Beam
Section Name:	W18x40
Cardinal point:	8
Angle:	0 deg
Material:	Steel- Carbon
Grade:	A36

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

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

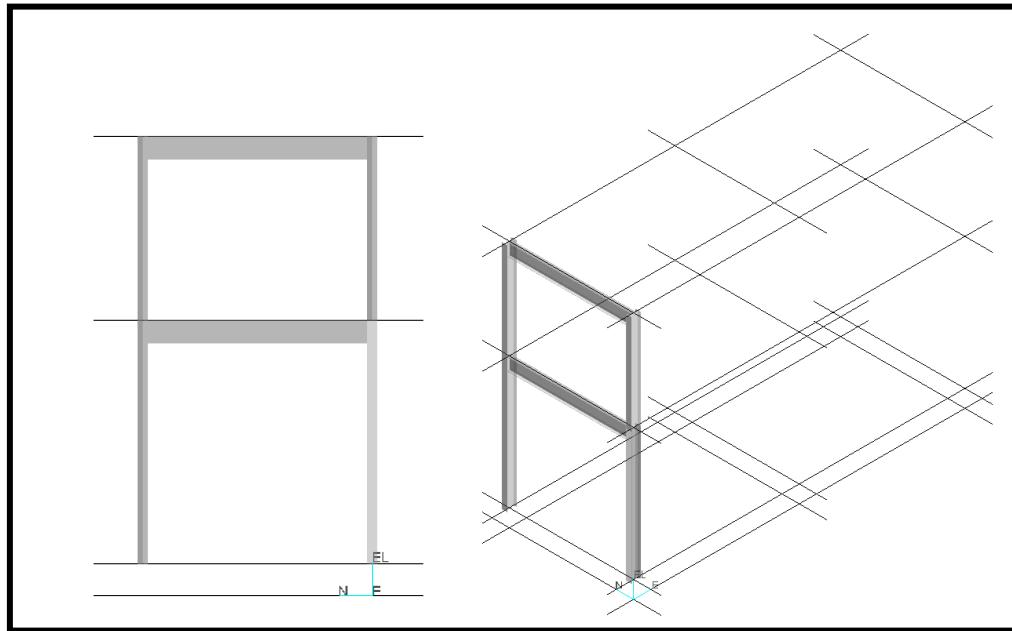
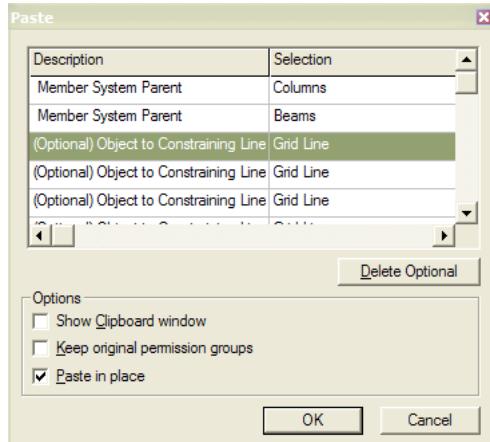


Figure 1 – Elevation and Isometric View of the Structure U02

11. Set the locate filter to Member Systems. 
12. Select all the members using the fence method.
13. Go to the Main Menu and select Edit -> Copy.
14. Select the end of the column as the reference point.
15. Go to the Main Menu and Select Edit -> Paste.

16. Re-establish all the relationship with East Plane location 20' 0" using the Paste dialog. See figure 2.

*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 only two grid lines.*



17. Hit "OK" button to commit the transaction.  
18. Repeat the Paste step to place the next frame at East Plane location 40' 0".

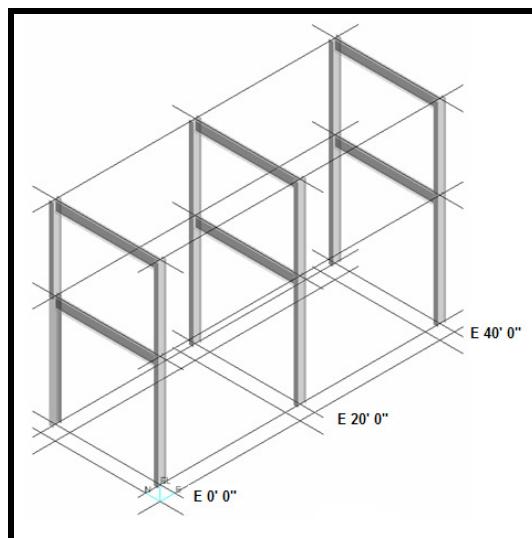


Figure 2 –Isometric View of the Structure U02

### **Placing Perimeter Beams:**

19. Select Place Linear Member System command to place beams as shown in figure 3. Use following properties for perimeter beams:

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

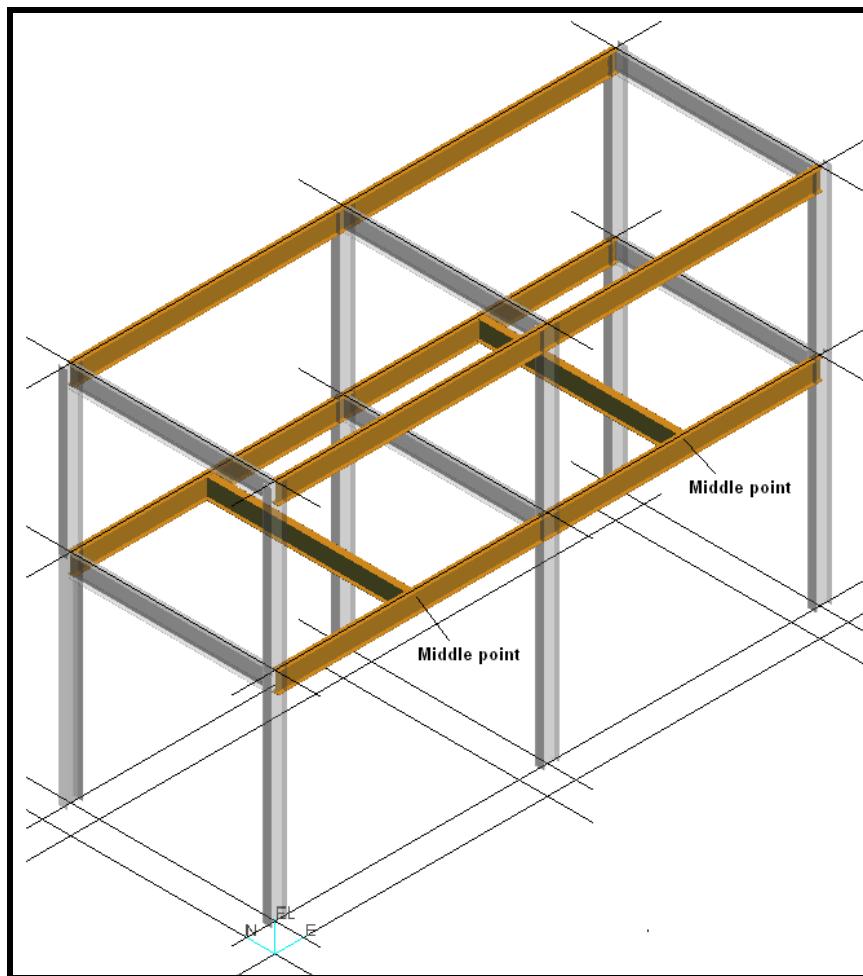


Figure 3 –Isometric View of the Structure U02

# LAB-1C: Productivity Commands - Structure U02

## Objectives

After completing this lab, you will be able to:

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

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

### **Placing Intermediate Beams for the first floor frame**

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

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

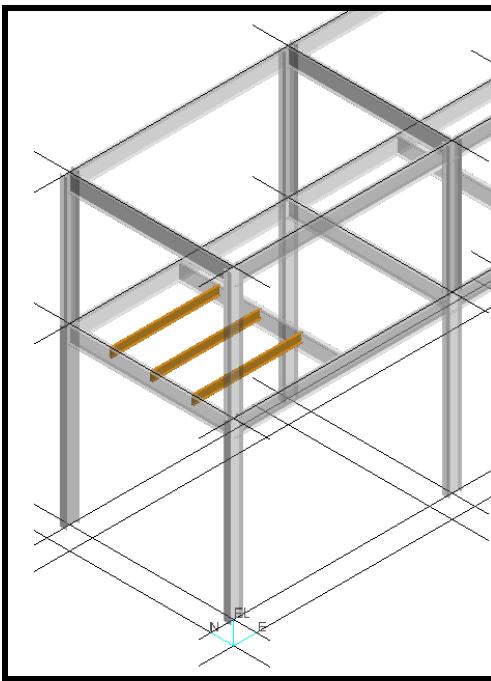


Figure 4 –Isometric View of the Structure U02

- 3 Continue using Place Framing Members command by selecting Supporting Member2 to place framing members as shown in Figure 5.

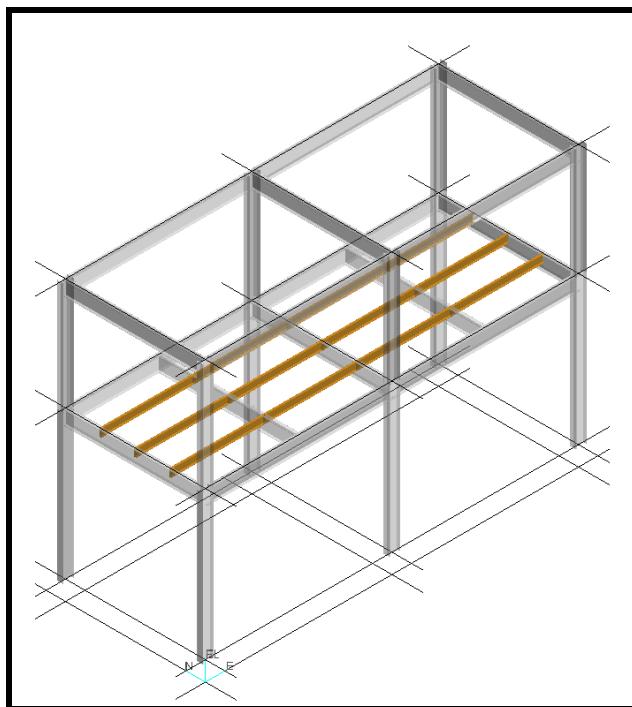


Figure 5 –Isometric View of the Structure U02

### **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:

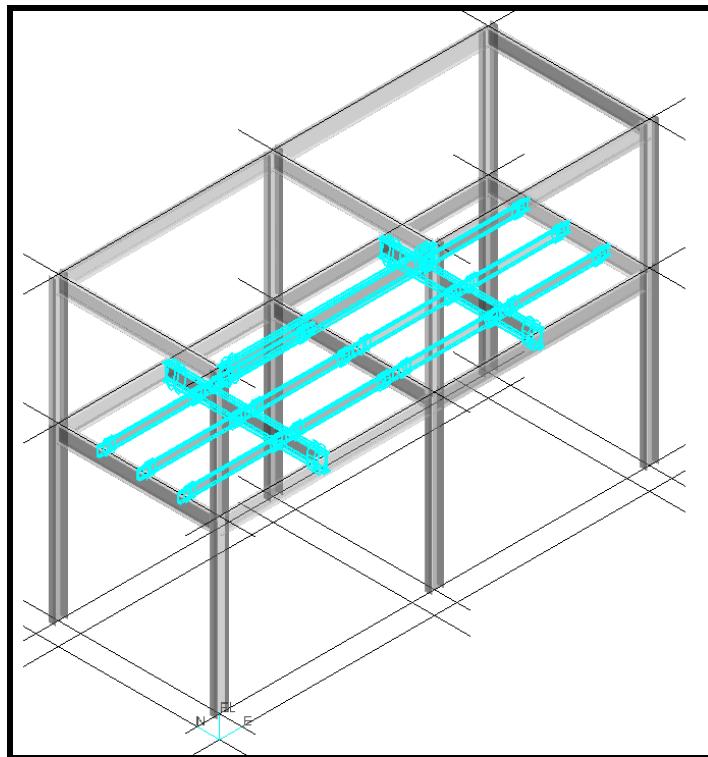
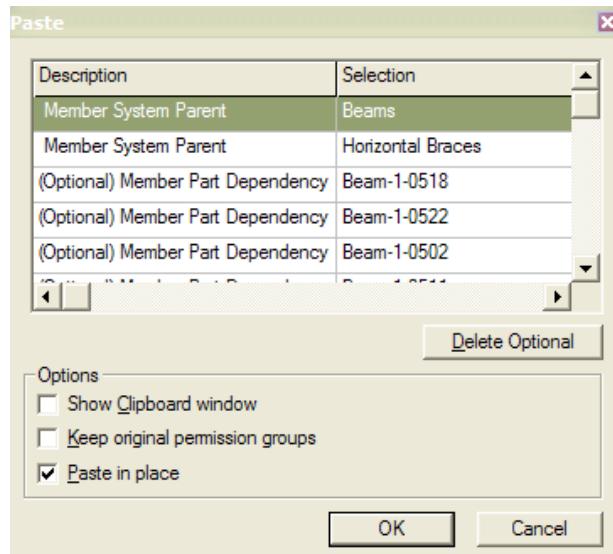


Figure 6 –Isometric View of the Structure U02 System

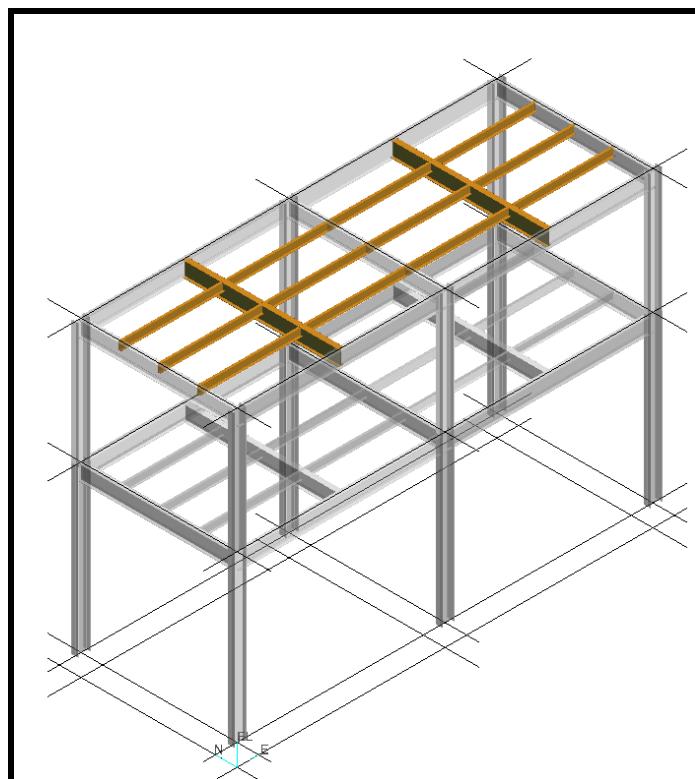
- 6 Go to the Main Menu and select Edit -> Copy.
- 7 Select the end of a beam (we just need a point on a object within the select set) as the reference point.
- 8 Go to the Main Menu and Select Edit -> Paste.
- 9 Make sure to select the appropriate objects to re-establish the connection in the Paste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



### **Placing Vertical Braces**

11. Select 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).

14. Select the second column for the second SmartStep.

15. Select the Beam for the third SmartStep.

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

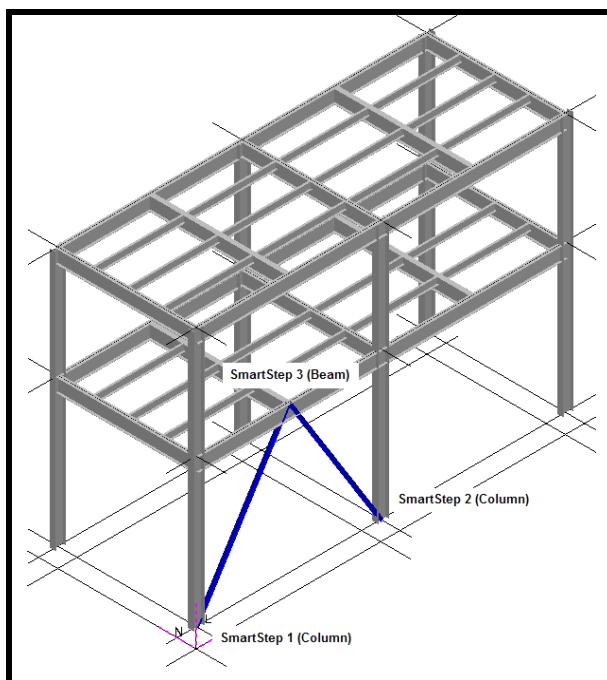


Figure 7 –Isometric View of the Structure U02

17. Repeat the above workflow to place the other vertical braces as shown in figure 8.

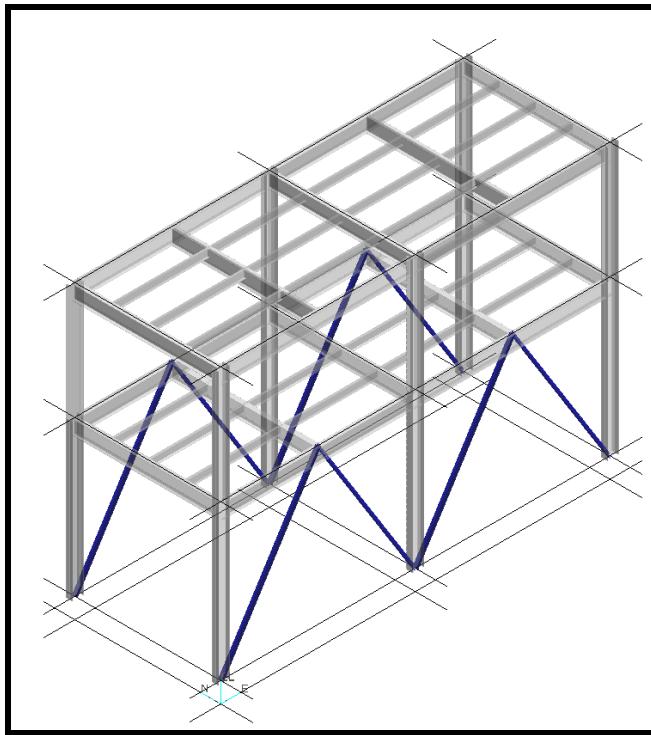


Figure 8 –Isometric View of the Structure U02

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

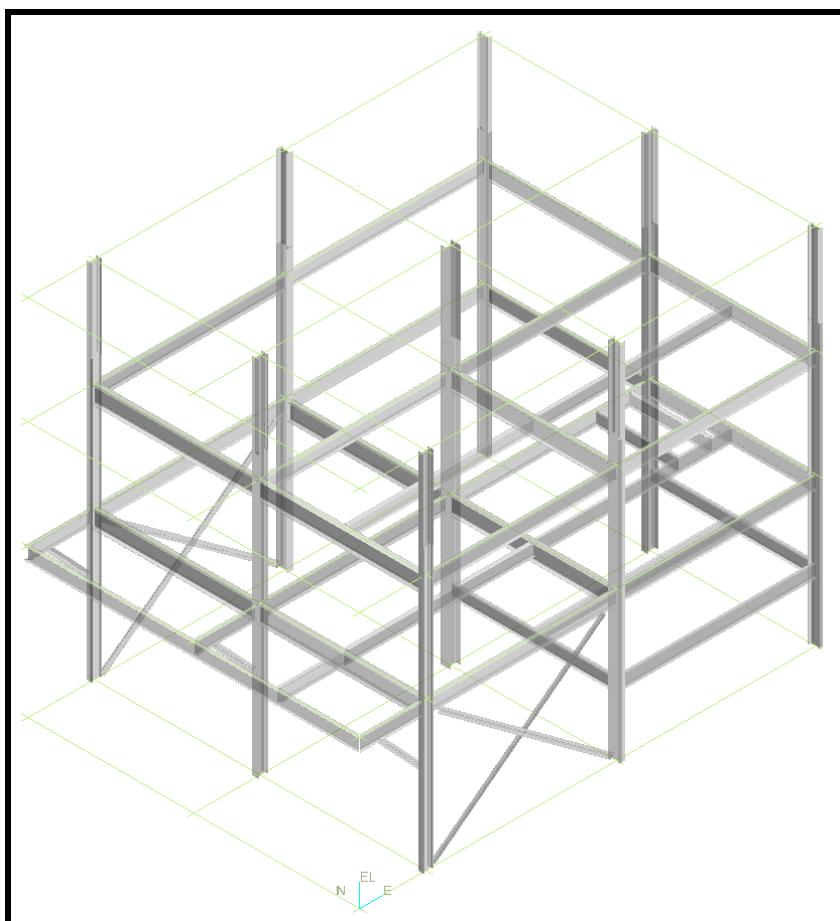
# LAB-2: Structure Modeling - Building U03

## Objectives

After completing this lab, you will be able to:

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

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

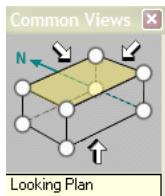


### **Part I: Columns**

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

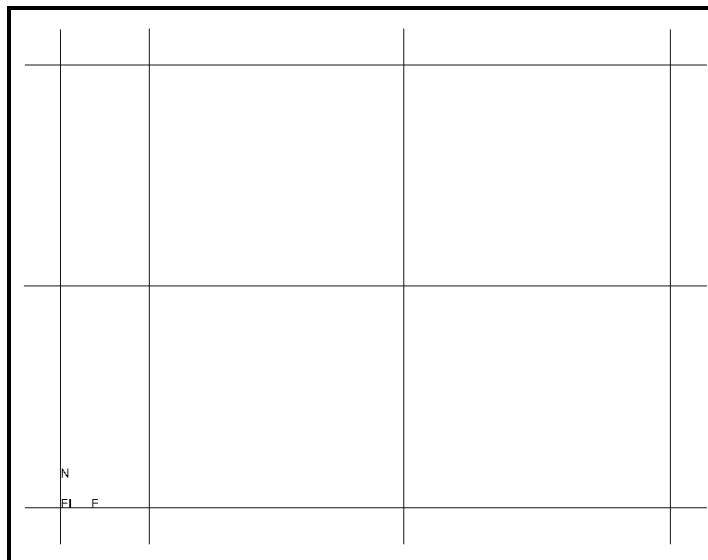
*Note: Notice that the grids you created in lab 1C are in this model. Though not absolutely necessary, grids are extremely useful as references for drawing dimensions. They also help in the modeling process by providing reference points at which you can place structural members.*

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



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

Your View should now resemble the following graphic.



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

System:	A2 -> U03 -> Structural -> Columns
Type Category:	Column
Type:	Column

Section Name: W14x53  
 Cardinal point: 5  
 Angle: 0 deg  
 Reflect: off  
 Material: Steel- Carbon  
 Grade: A36

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

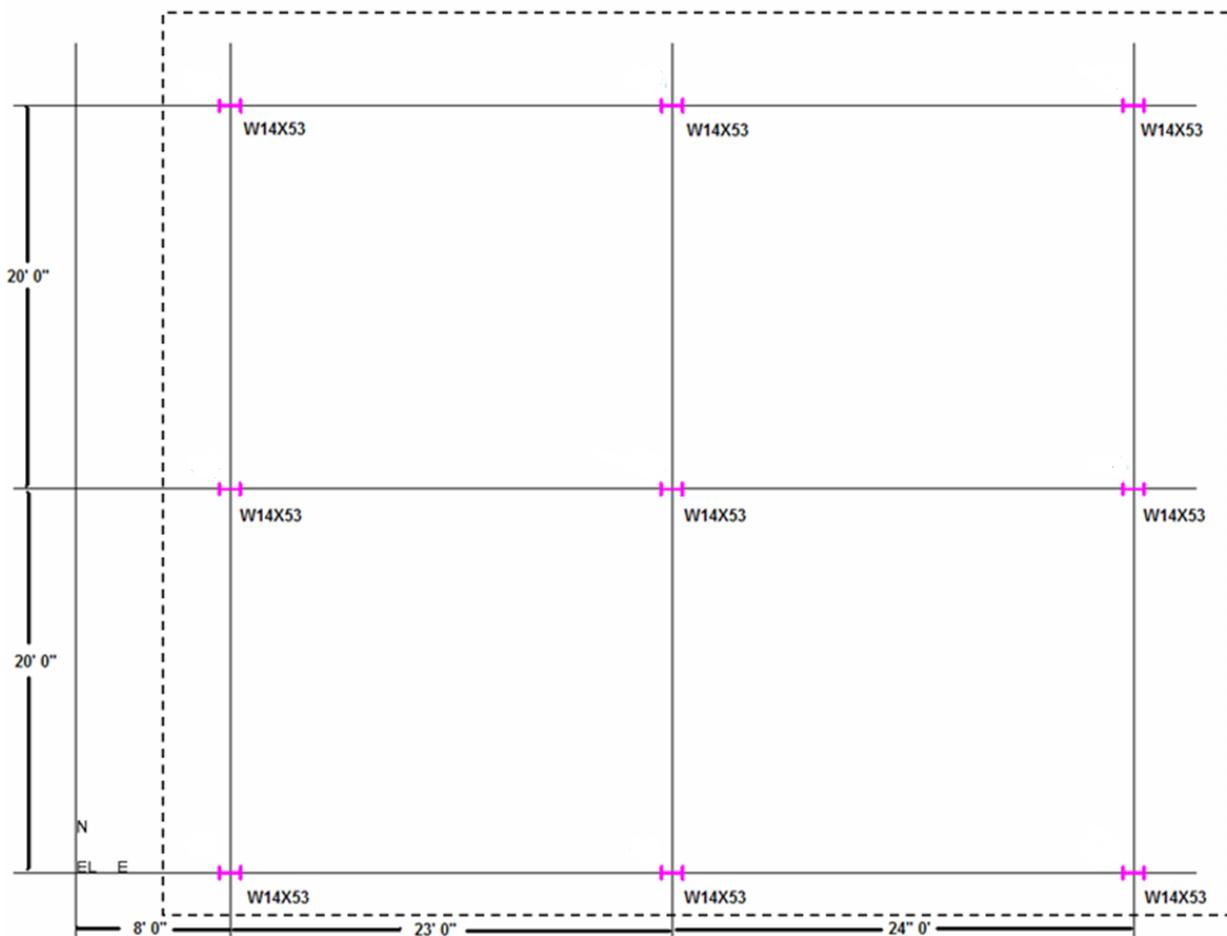


Figure 1 - Floor Plan

Change the column section size and orientation

11. Set the locate filter to Member Parts.



12. Multi-select the appropriate columns and use the SmartStep ribbon bar to change section size and the orientation as shown in figure 2.

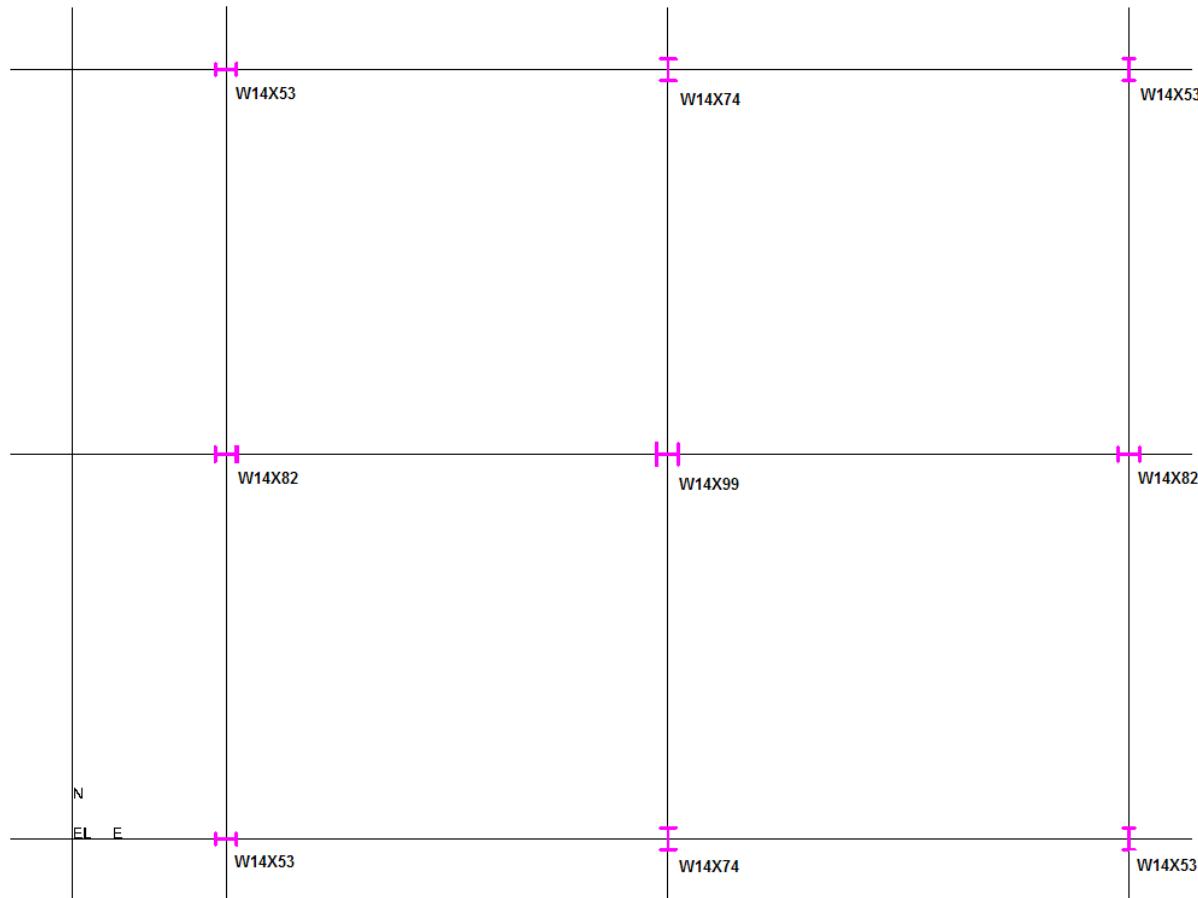


Figure 2 - Floor Plan

Place split connections at elevation 34'-0" using the Place Split command

13. Select Place Split command.

14. Make sure the Split Status is set By Rule.

15. Select 34'-0" Elevation Plane object in the Workspace Explorer or the Ruler.

16. Select all columns to split and click "Finish" button. Figure 3 shows Split connections highlighted.

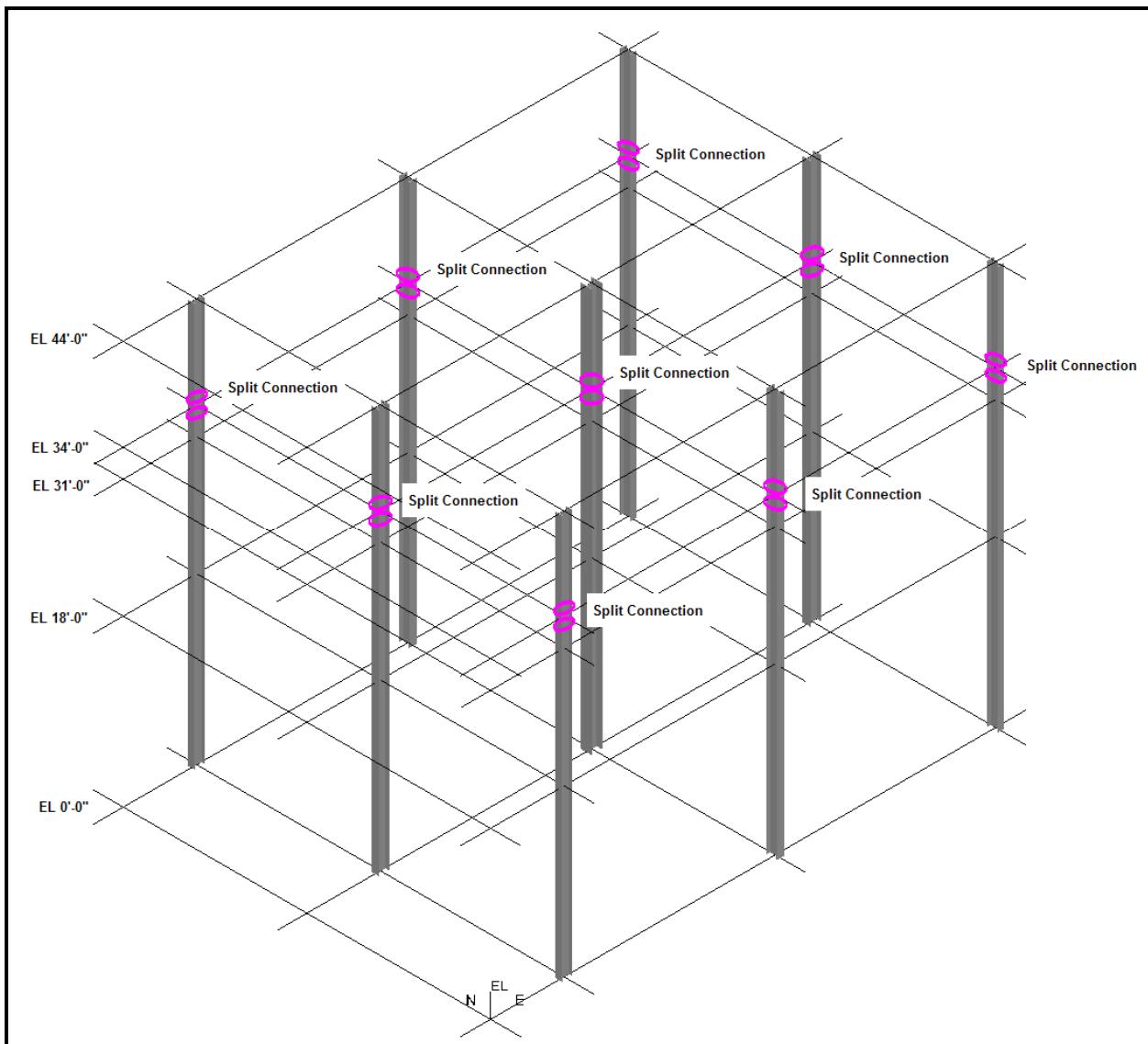


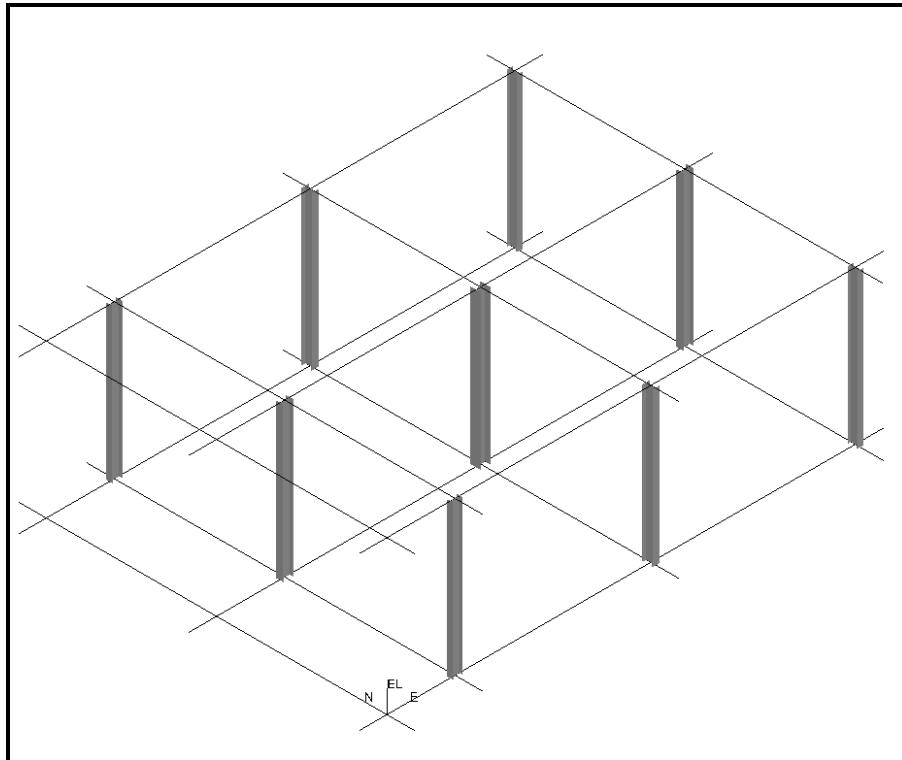
Figure 3 – Isometric View of the Structure U03

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

## **Part II: First Floor Frame**

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

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



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

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

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

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

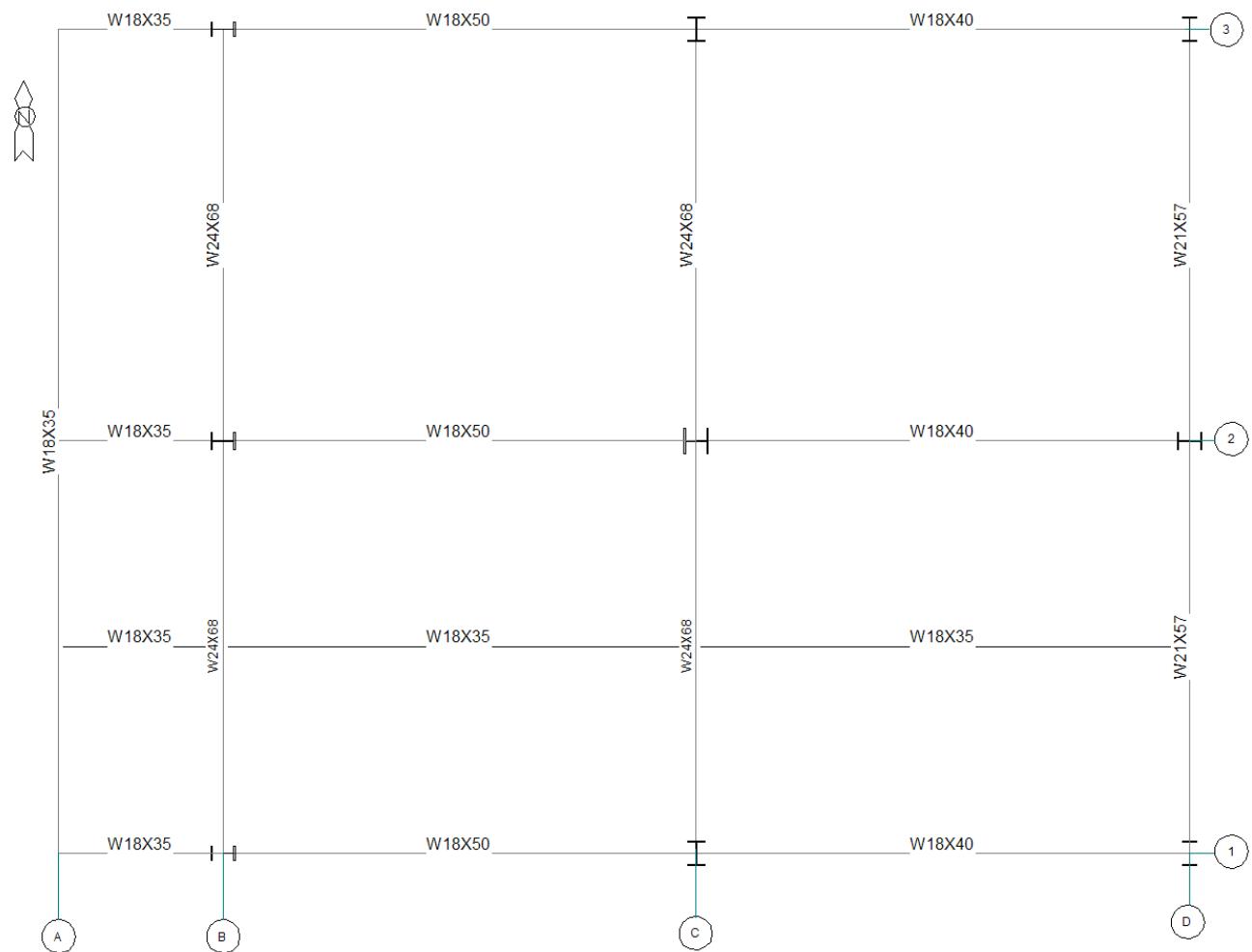
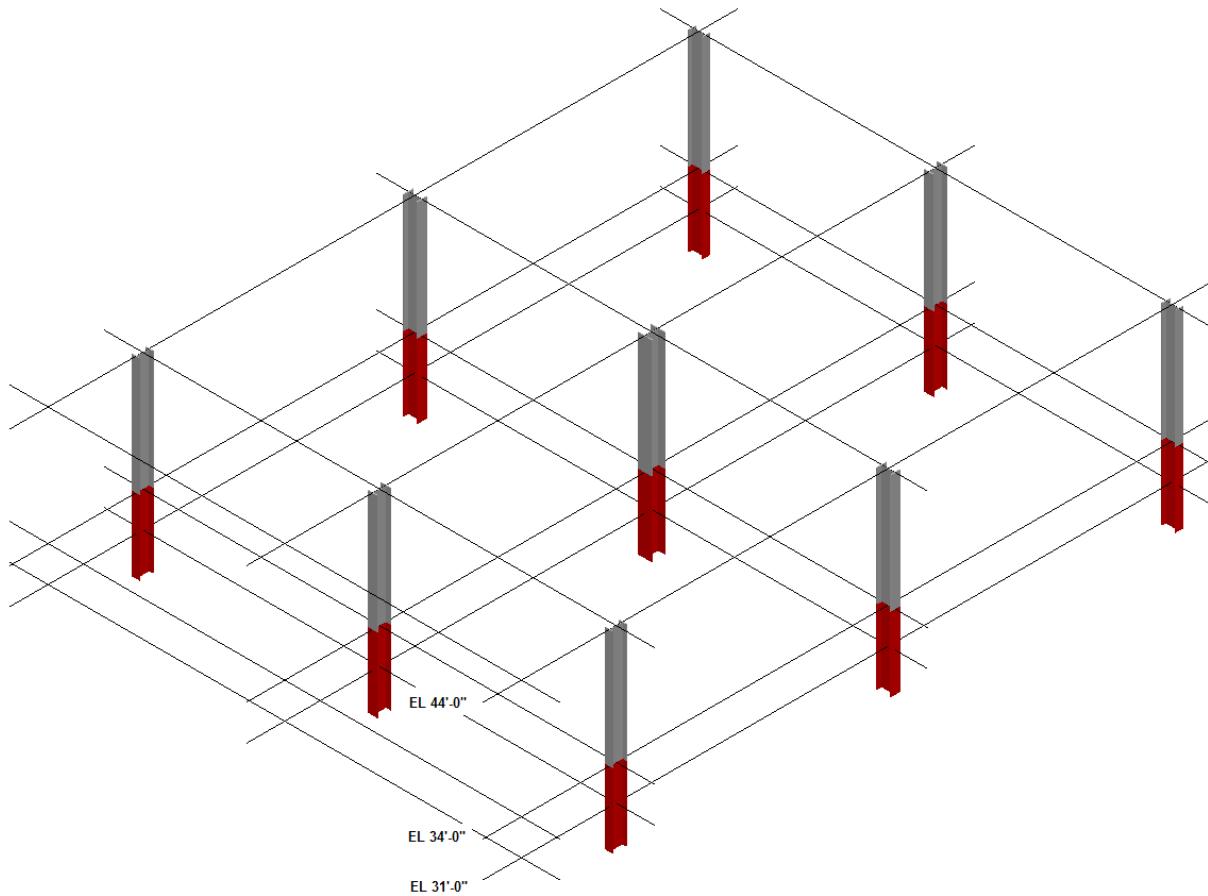


Figure 4 – First Floor Plan

### **Part III: Second Floor Frame**

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



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

Use following properties:

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

Place all perimeter beam in  
A2 -> U03 -> Structural -> Beams.

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

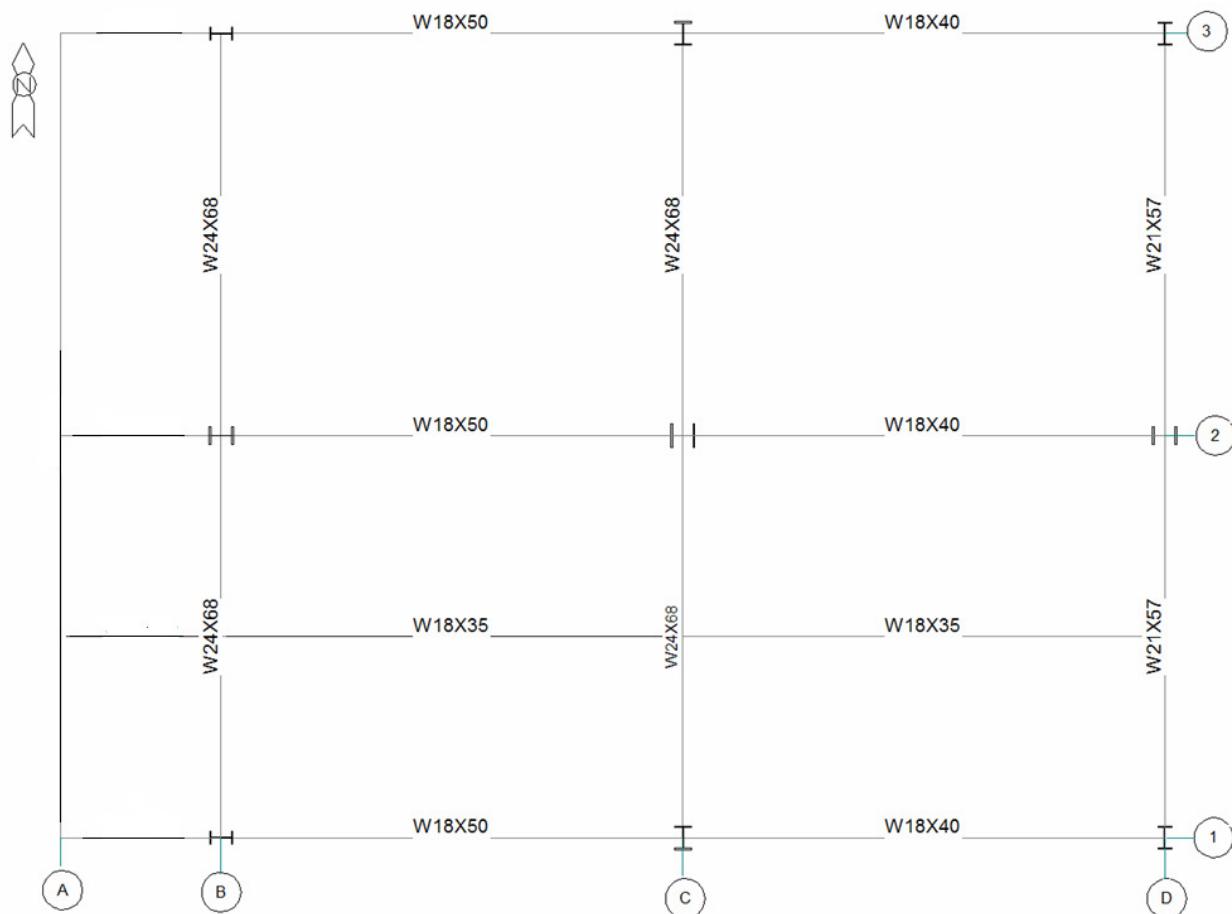


Figure 5 – Second Floor Plan

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

Your View should now resemble the following graphic:

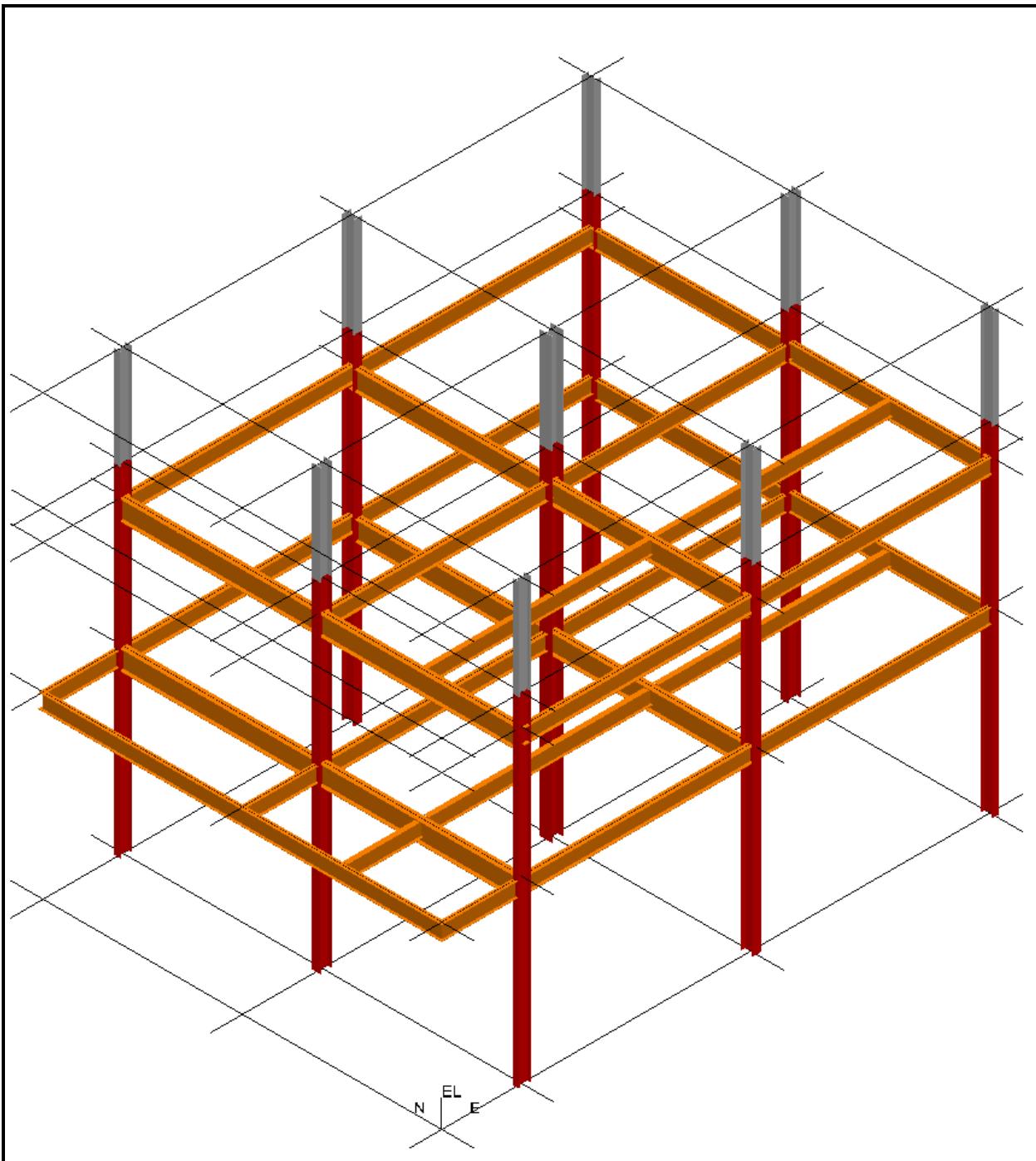


Figure 6 – Isometric View of the Structure U03

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

## Part IV: Vertical Braces

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

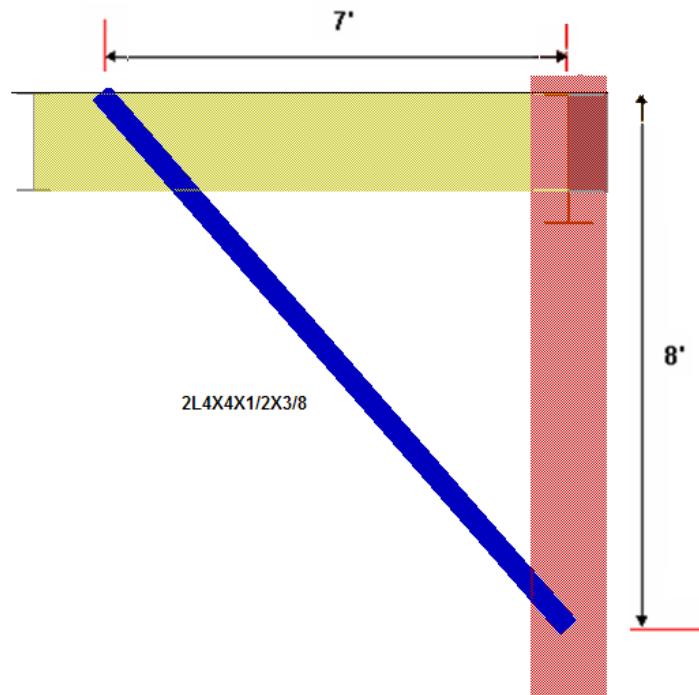


2. Select Place Linear Member System Command to place the supported brace.

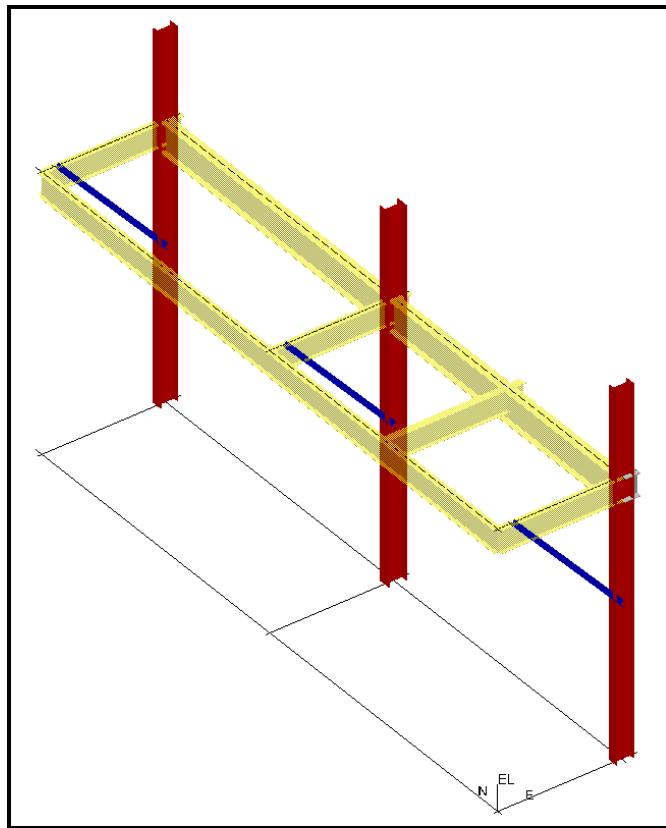
*Note: Use Pinpoint tool to place supported bracing at a given distance. Place the PinPoint target at the intersection of the column and the beam.*

3. Set the parameters as

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



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



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

Bracing Type:	Cross
Connection:	By Rule
System:	A2->U03 -> Structural ->Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	2L4x4x1/2x3/8
Cardinal point	5
Angle:	0 deg
Reflect:	off
Material:	Steel- Carbon
Grade:	A36

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

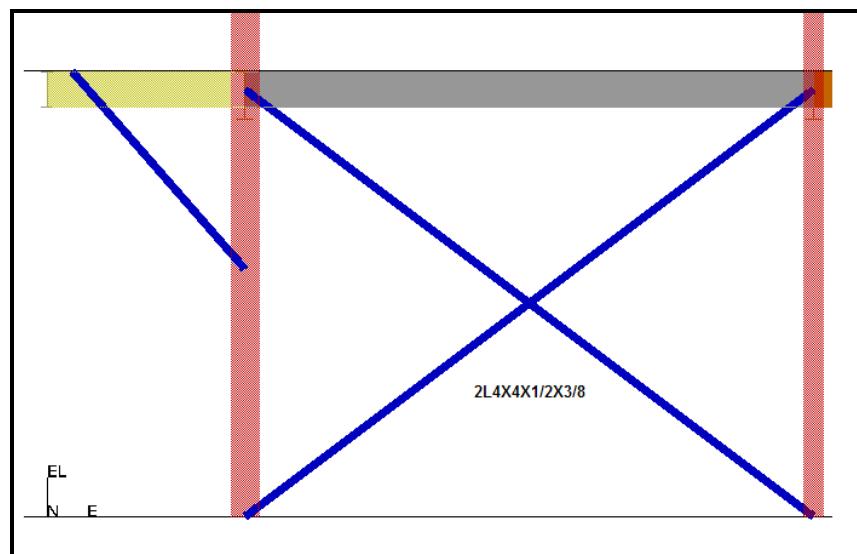


Figure 7 – North View of the Structure U03

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

Your View should now resemble the following graphic:

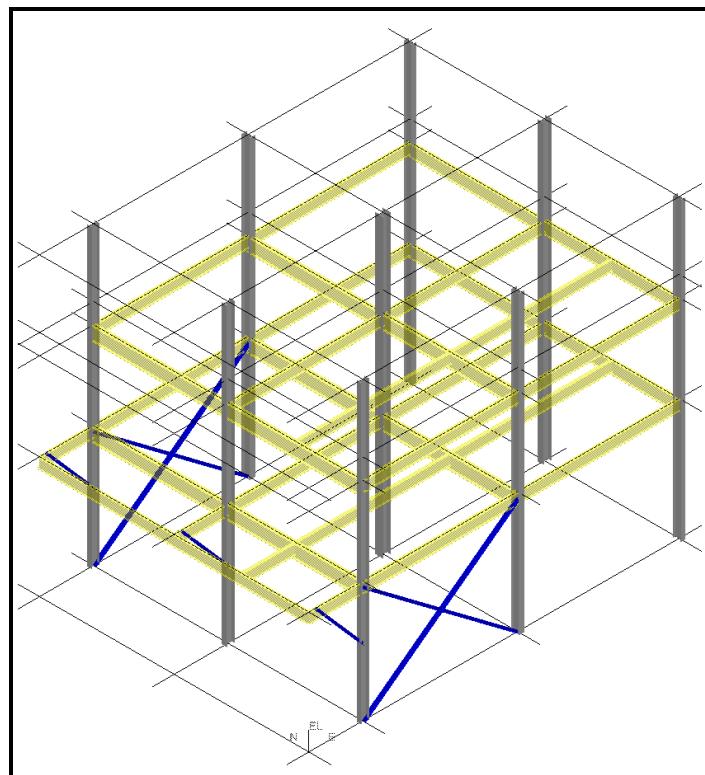


Figure 8 – ISO View of the Structure U03

## **Part V: Horizontal Braces**

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

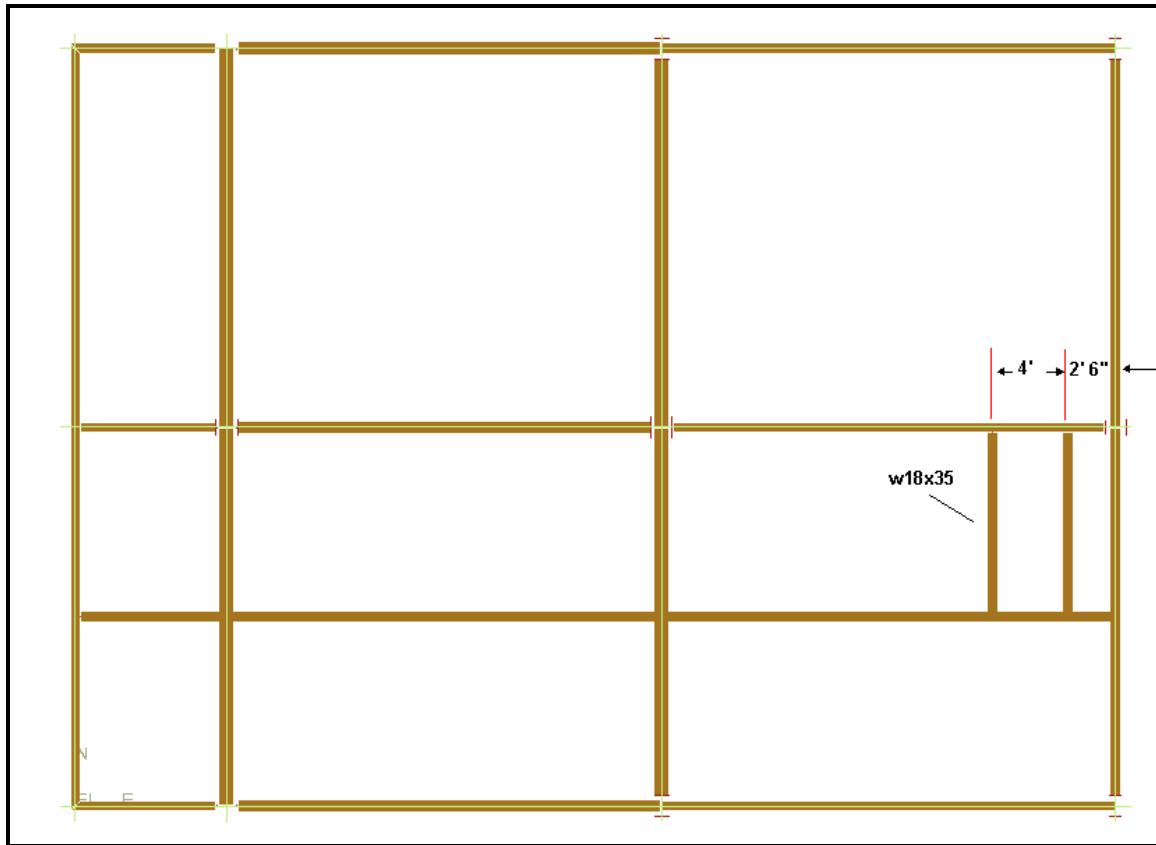
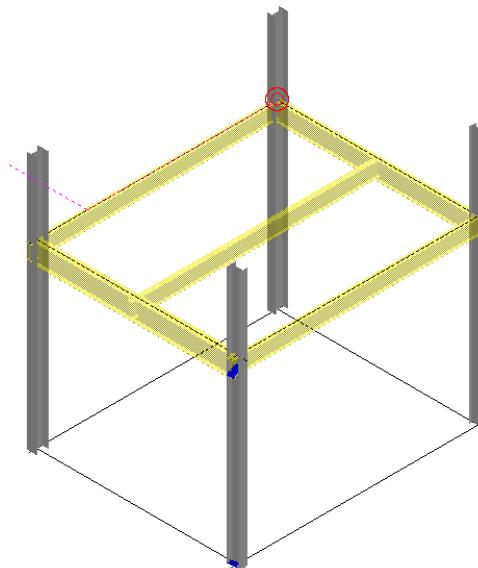


Figure 9 – First Floor Frame El -18' 0”

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

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



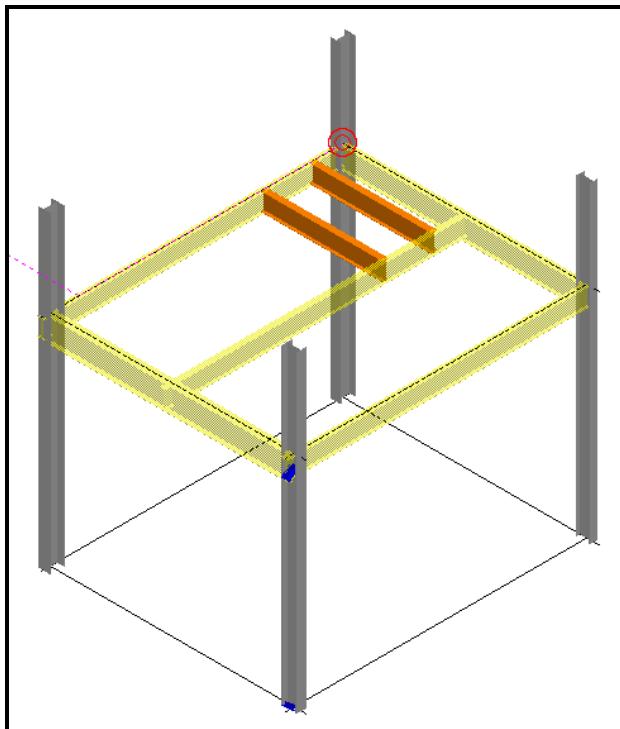
4. Select Place Linear Member System Command

5. Set the parameters as

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

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

Your View should now resemble the following graphic:



#### **Part VI: Horizontal Beams on the Far-East Bay**

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

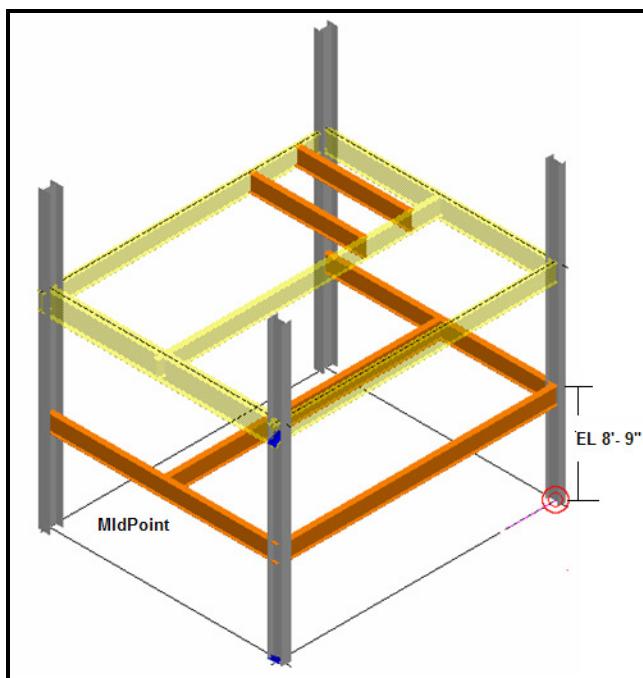


Figure 10 – ISO View of the Structure U03

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

2. Select Place Linear Member Command.

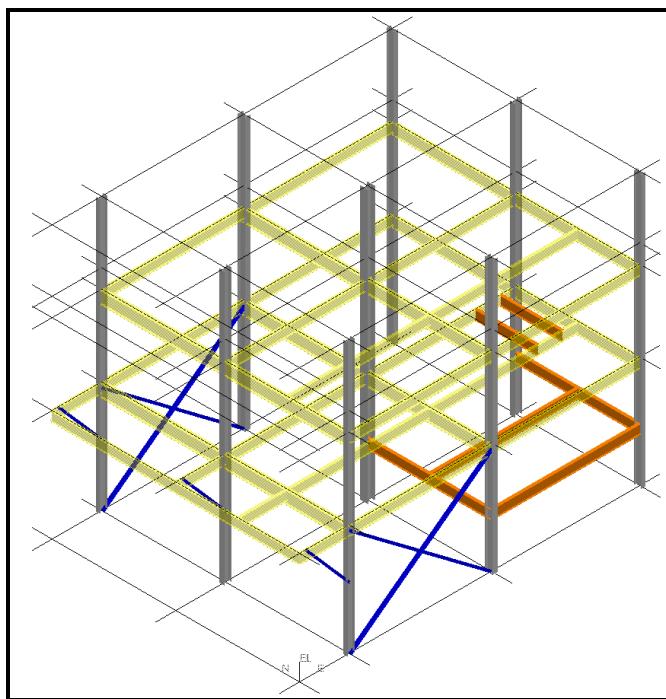
3. Set the parameters as

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

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

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

Your View should now resemble the following graphic:



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

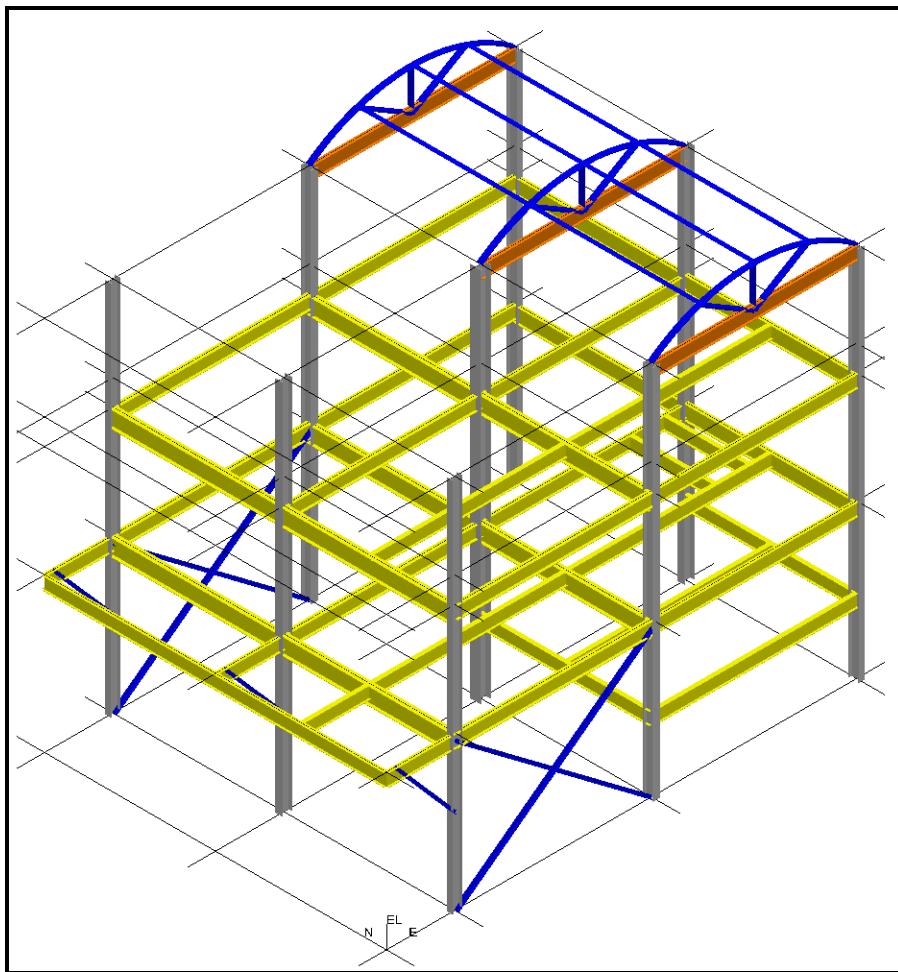
# LAB-3: Curve Member System - Building U03

## Objectives

After completing this lab, you will be able to:

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

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



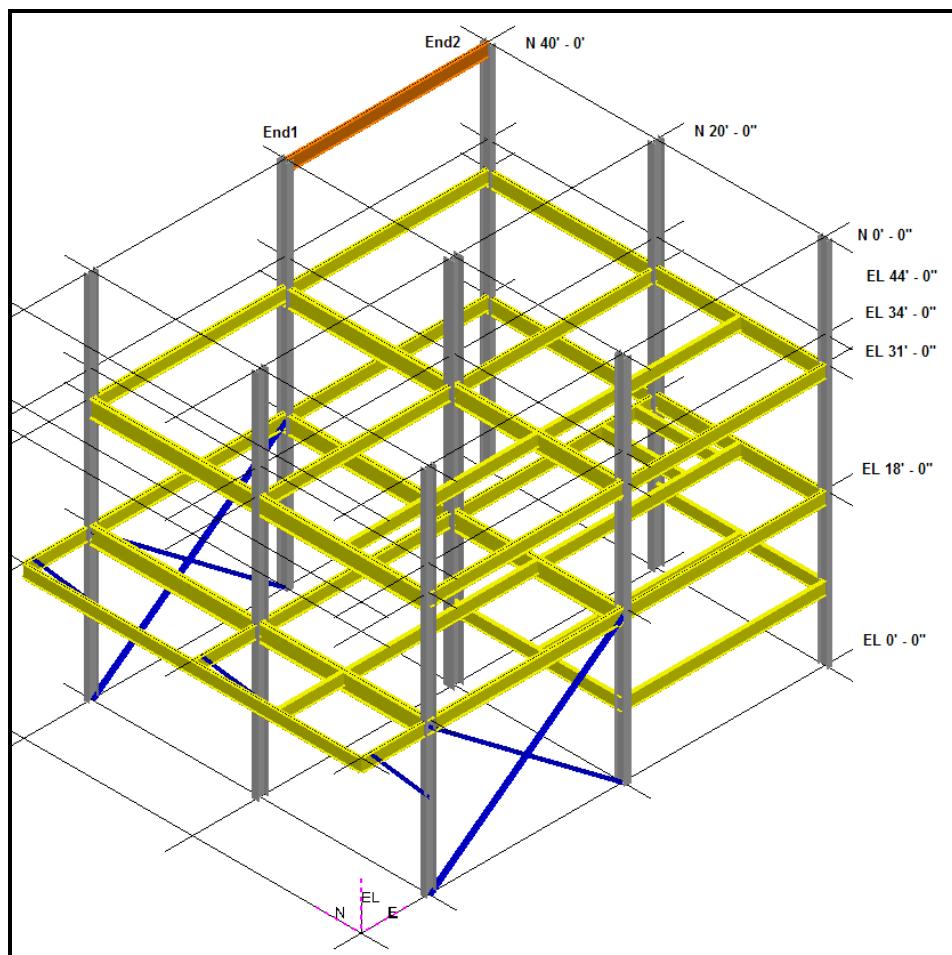
### Placing Beams to support the trusses

1. Select the Place Linear Member System Command.

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

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



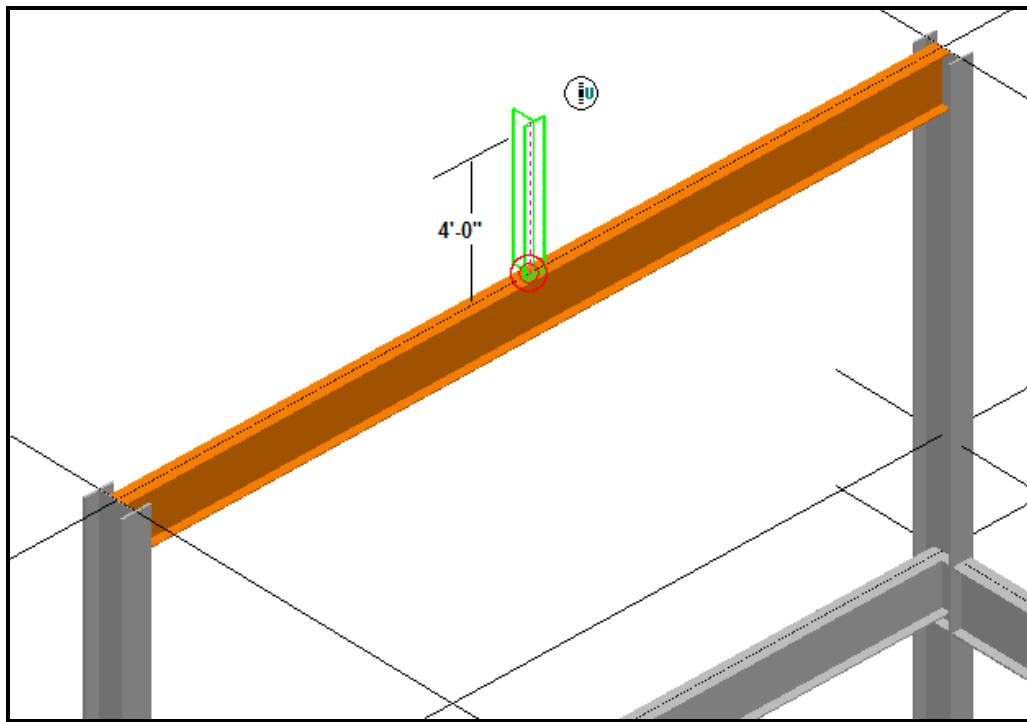
Next, you will be placing a vertical brace to support the actual curve member.

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

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

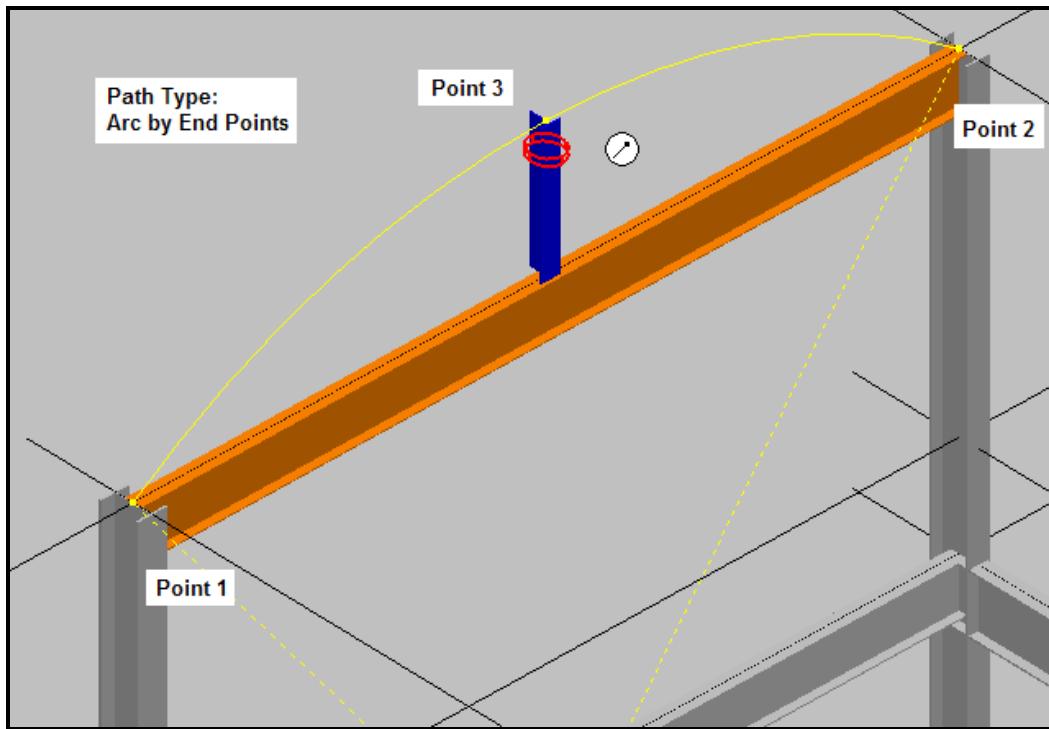
7. Use SmartSketch service to locate the mid-point geometric of the beam to place End1.
8. Use PinPoint key-ins to place End 2 at 4'- 0" up along the Z-axis.

Your View should now resemble the following graphic:



9. Select Place Curve Member System Command. System displays the 3D Sketch Path ribbon bar.
10. Place Point 1 at the end of the column located at North Plane 40'- 0" and Elevation 44'- 0" intersection.

11. Select Path Type as Arc by End Points.
12. Place Point 2 at the end of the column located at the other North Plane 40'- 0" and Elevation 44'- 0" intersection.
13. Place Point 3 at the end of the vertical brace that you have just placed.



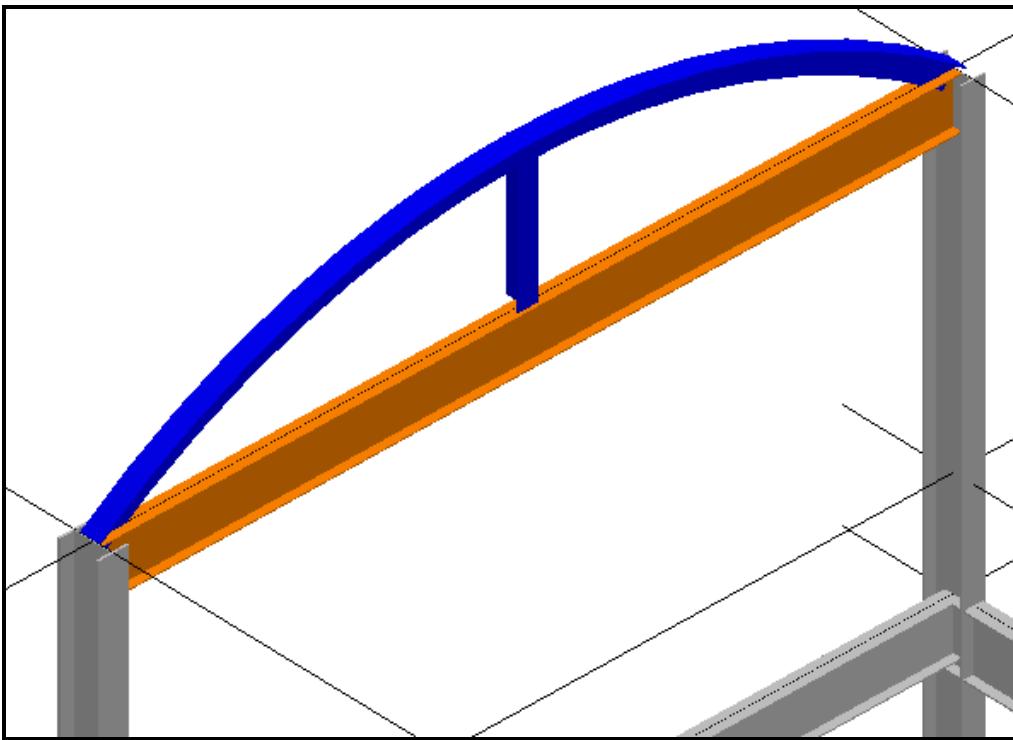
14. Hit the “Finish” Path button. System returns to the SmartStep ribbon bar.

15. Set the parameters as

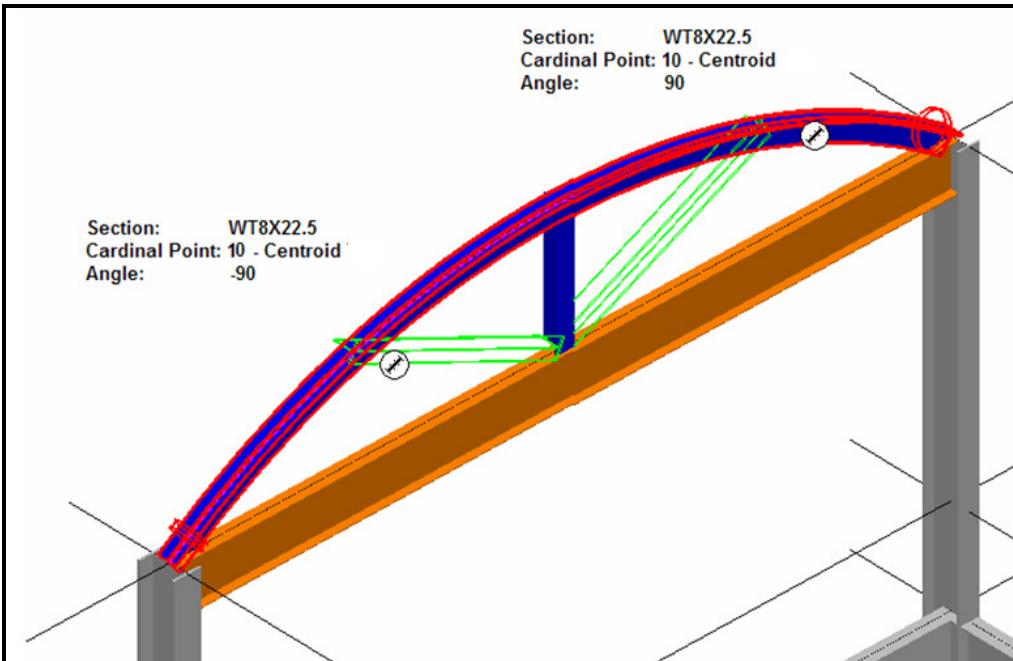
System:	A2->U03 -> Structural ->Vertical Braces
Type Category:	Brace
Type:	Vertical Braces
Section Name:	WT8X22.5
Cardinal point	10 - Centroid
Angle:	0 deg
Reflect:	off
Priority:	Secondary
Material:	Steel- Carbon
Grade:	A36

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

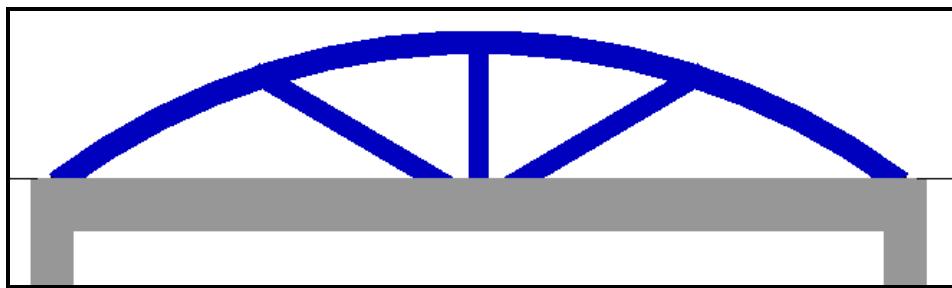
Your View should now resemble the following graphic:



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:

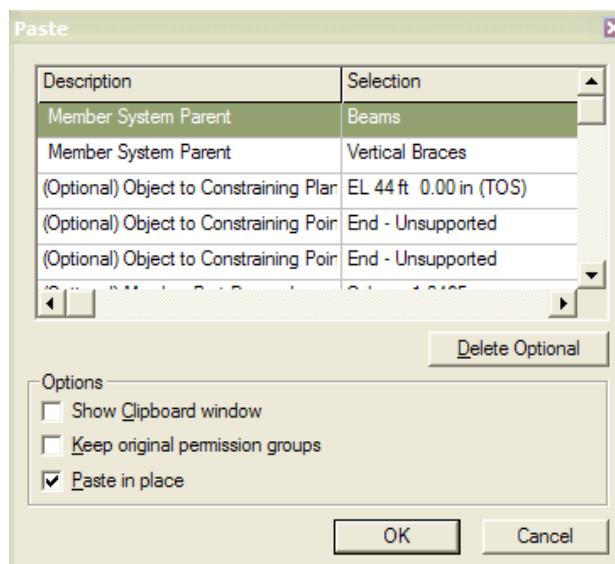


Your View should now resemble the following graphic:

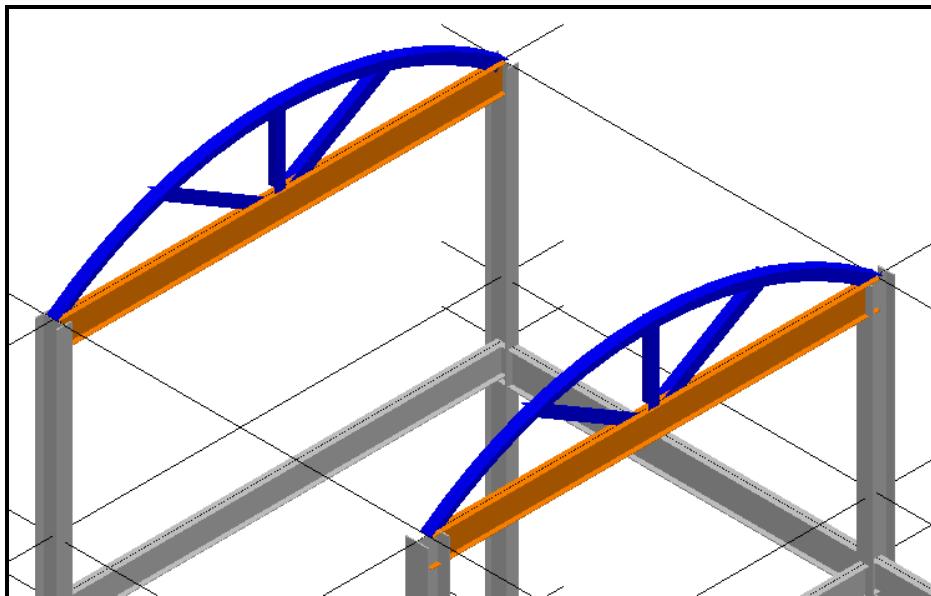


18. Set the locate filter to Member Systems. 
19. Select the curve member, the support beam and the three vertical braces using the fence method or the Multi-Select method.
20. Go to the Main Menu and select Edit -> Copy.
21. Select the end of the column as the reference point.
22. Go to the Main Menu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

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

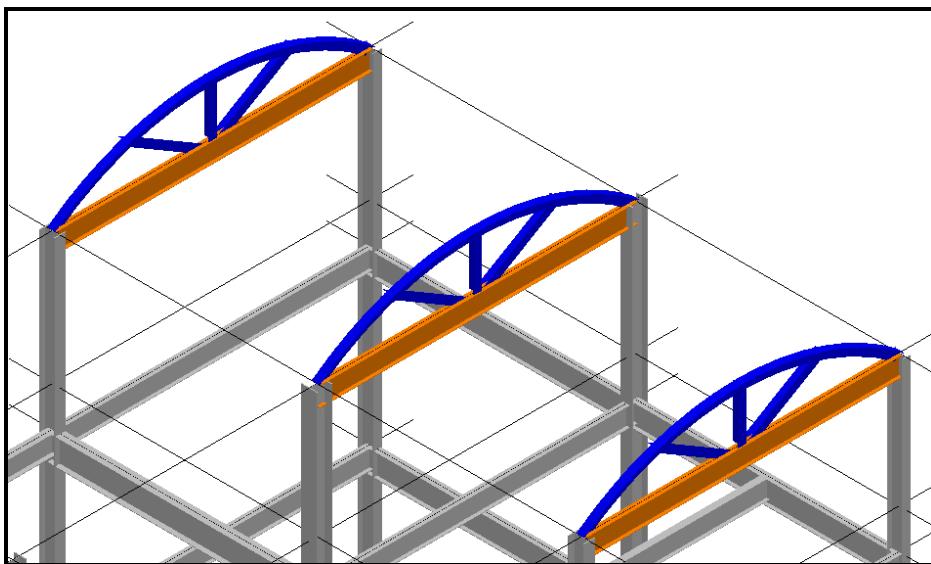


24. Hit "OK" button to commit the transaction.



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

Your View should now resemble the following graphic:



26. Select Place Framing Members command. Use the view shown in Figure 11 to place the structural members.

27. Set the active member parameters as follows:

Fit Mode:	By Count
Count:	3
Connection:	By Rule
System:	A2->U03 -> Structural -> Horizontal Braces
Type Category:	Brace

Type: Horizontal Brace  
Section Name: L4X4X1/2  
Cardinal point: 3  
Reflect: off  
Offset: 0' 0"  
  
Position: Toggle to Skewed  
  
Angle: 0 deg  
Priority: Secondary  
Material: Steel- Carbon  
Grade: A36

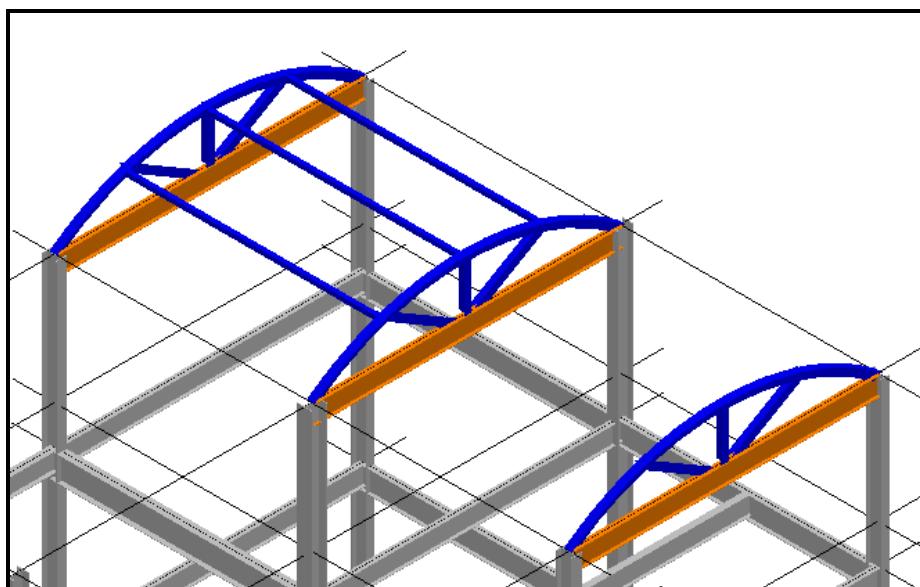


Figure 11 – ISO View of the Structure U03

28. Continue using Place Framing Members command by selecting Supporting Member2 to place framing members as shown in Figure 12.

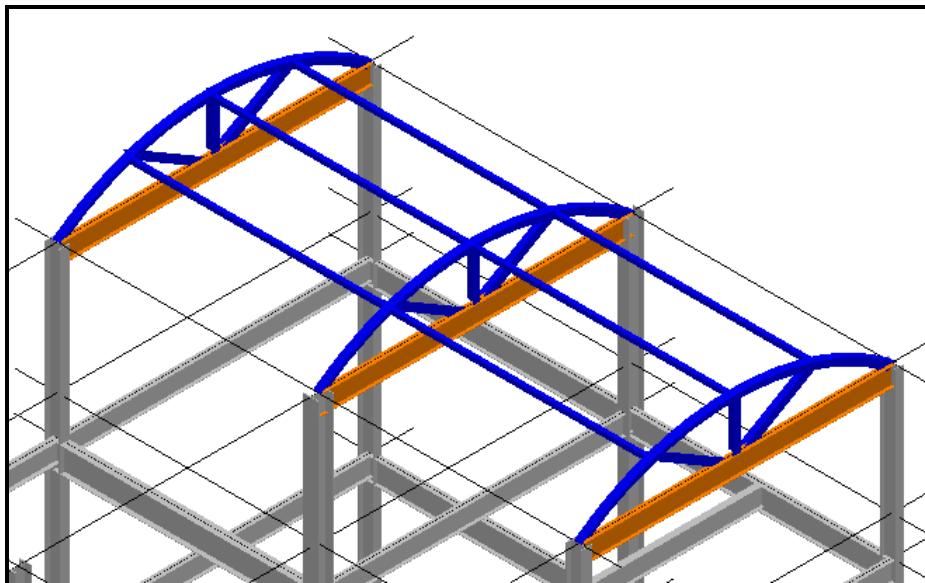


Figure 12 – ISO View of the Structure U03

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

# LAB-4: Frame Connections

## Objectives

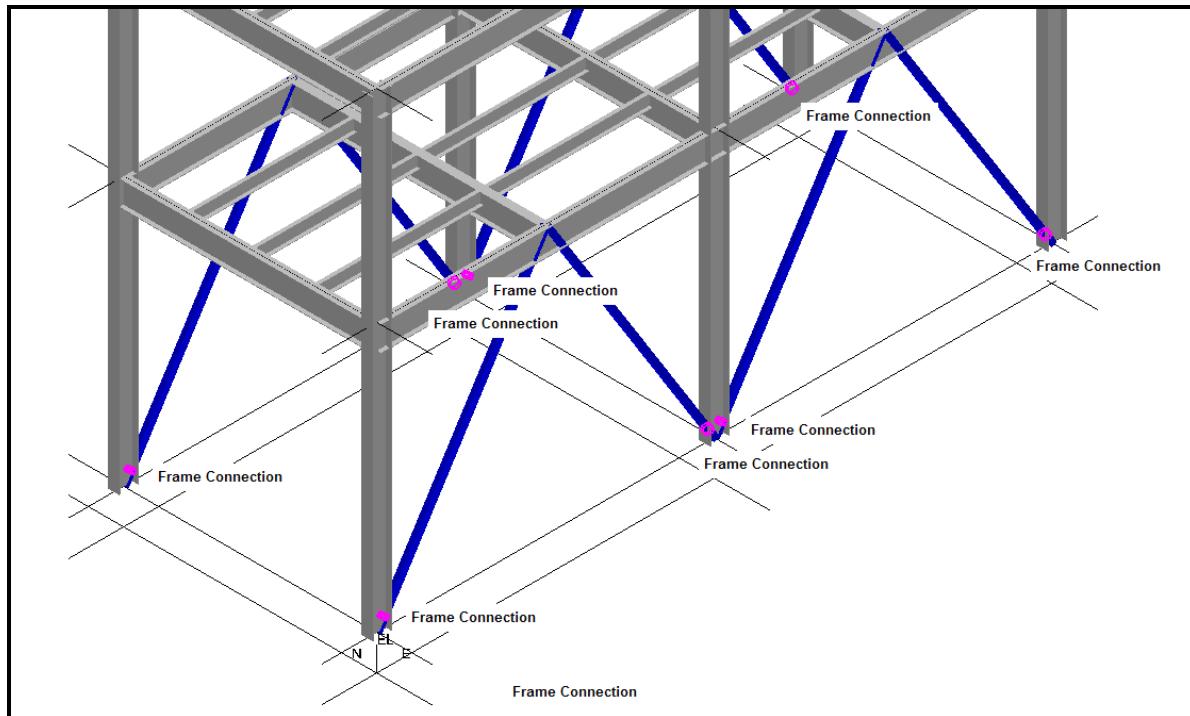
After completing this lab, you will be able to:

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

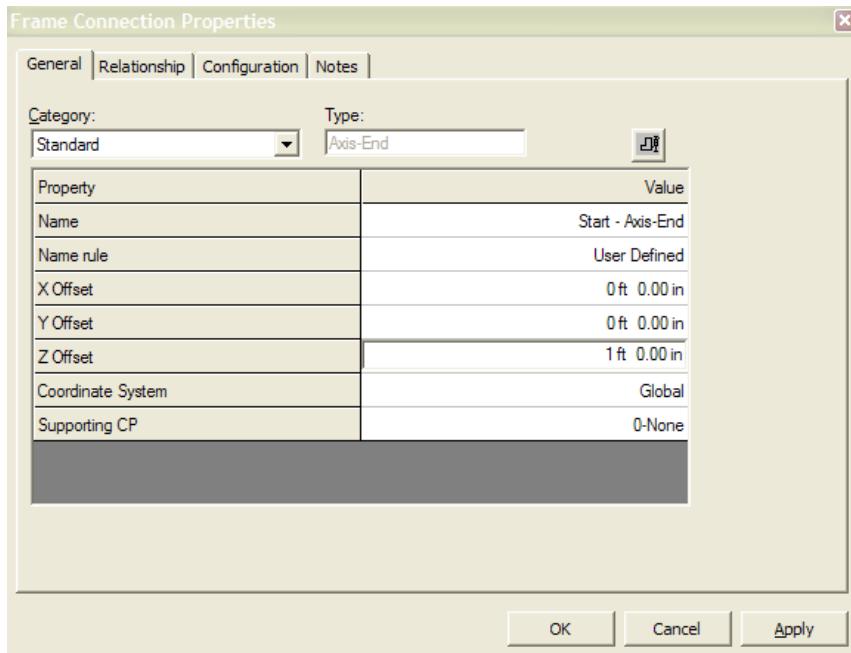
You will apply a working point offset for the vertical braces in the Structure U02.

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

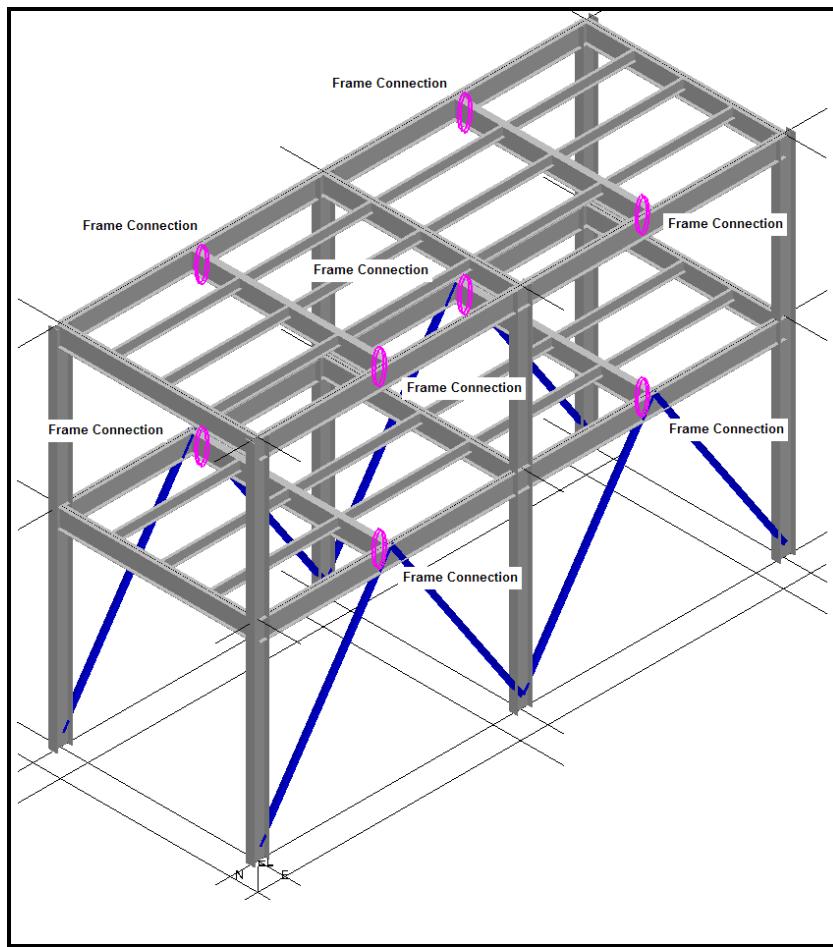


7. Keyin 1'- 0" for Z Offset. Hit "OK" button to commit the transaction.

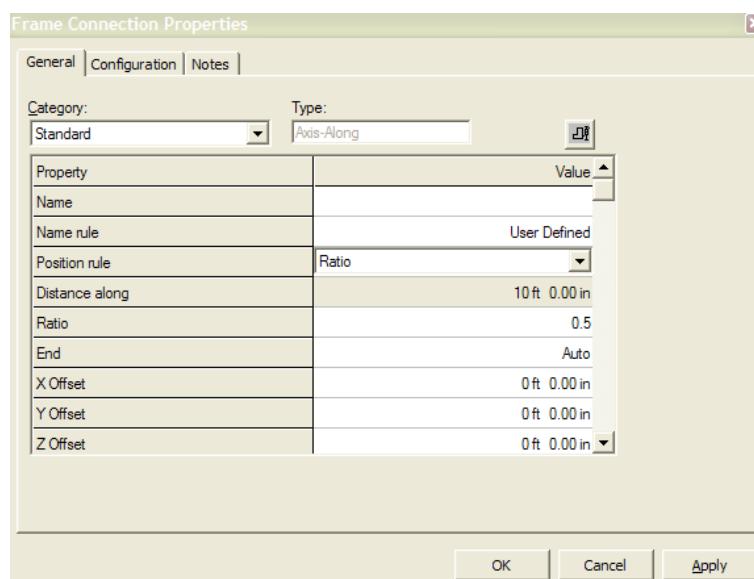
Next, you will set the Position Rule from Intersection to Ratio for the intermediate beams in the Structure U02.

*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



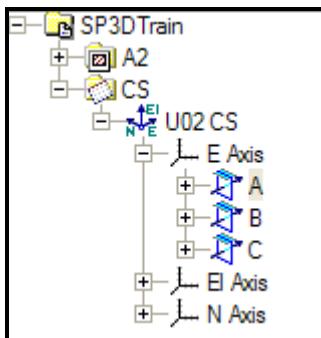
10. Change Position Rule from Intersection to Ratio. Hit "OK" button to commit the transaction.

Next, you will move the East Grid Plane 0'- 0" in the Structure U02.

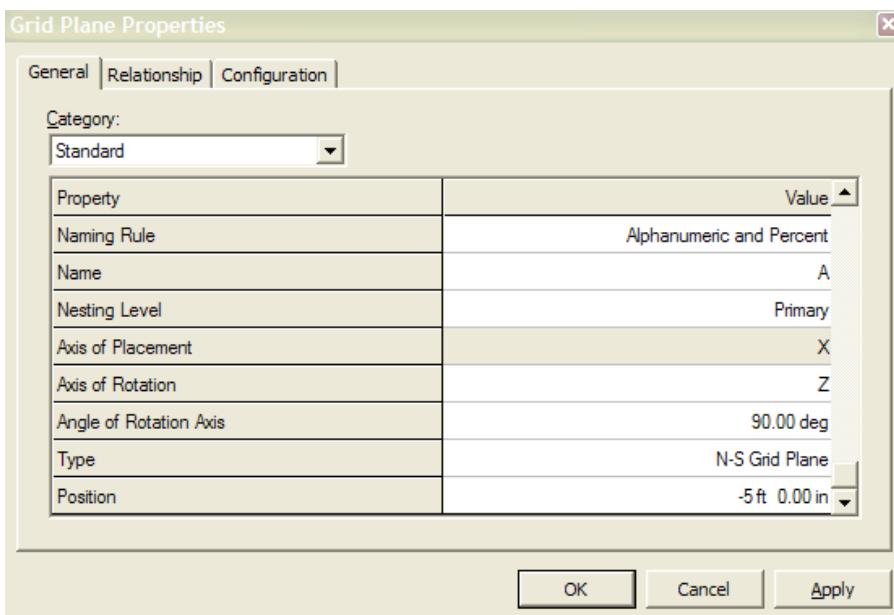
- Set the locate filter to All.



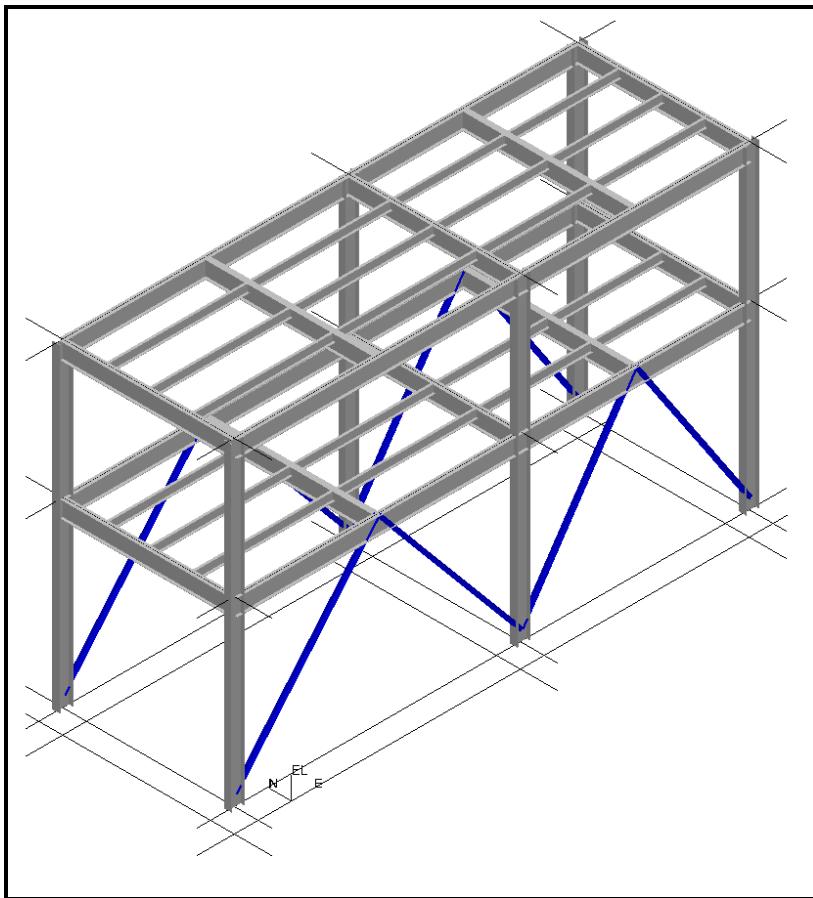
- Go to the Workspace Explorer and select Grid Plane A



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



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



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

# LAB-5: Assembly Connections

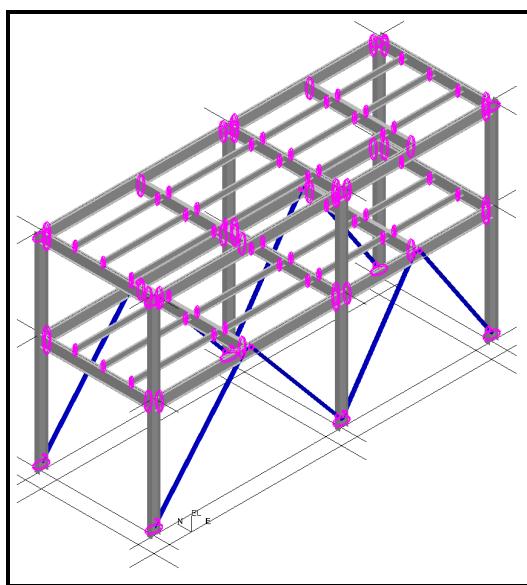
## Objectives

After completing this lab, you will be able to:

- Understand the assembly connection entities and relationships
- Use Place Assembly Connection command
- Use Trim Member Command
- Edit the assembly connection properties

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

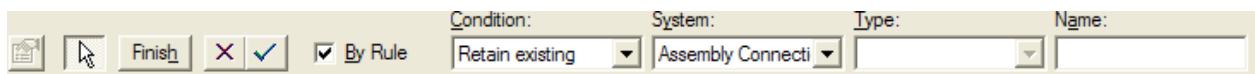
1. Open or create a session file and define an appropriate filter for your workspace. Your workspace should include the A2 -> U02 and CS -> U02 CS systems.
2. Go to the Structure Task environment.
3. Make sure the Active Permission Group is set to *Structural*.
4. Set the locate filter to Frame Connections. 
5. Select all the frame connections in Structure U02 using the fence method. See figure below:



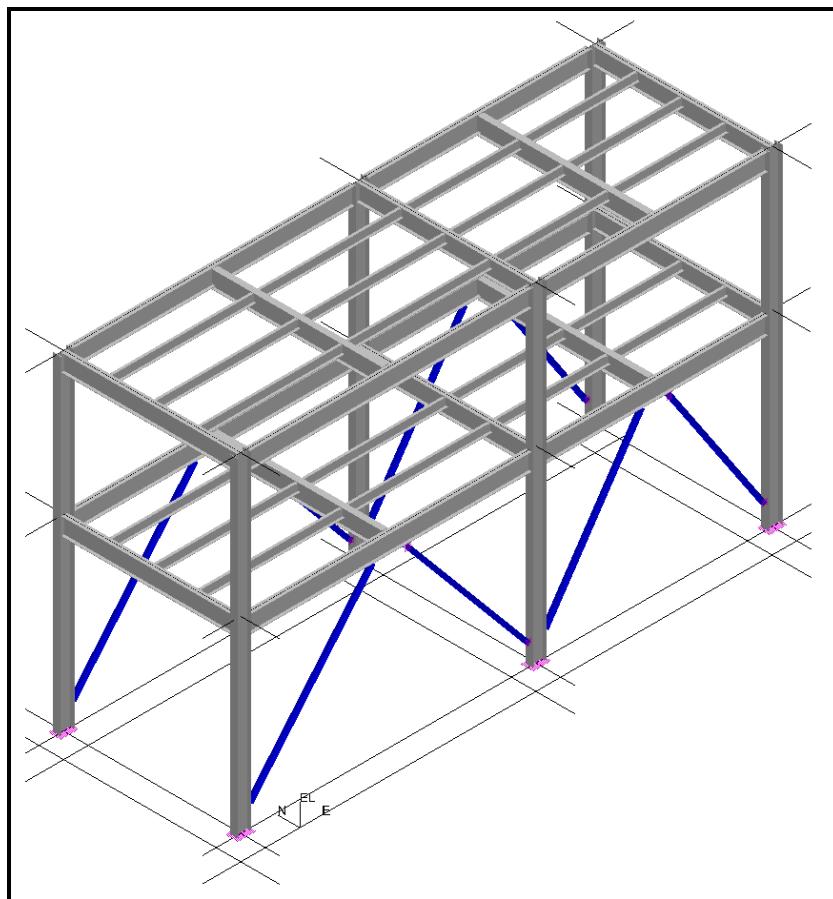
6. Select Place Assembly Connection Command.
7. Set the parameters as

By Rule:  
Condition:  
System:

Check  
Retain existing  
A2-> U02 -> Structural -> Assembly Connections.



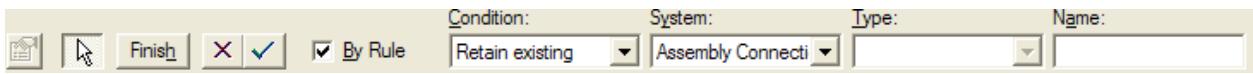
8. Hit “Finish” button to commit the transaction.
9. Examine the model and verify that the planar cutbacks, base plates and gusset plates have been created.



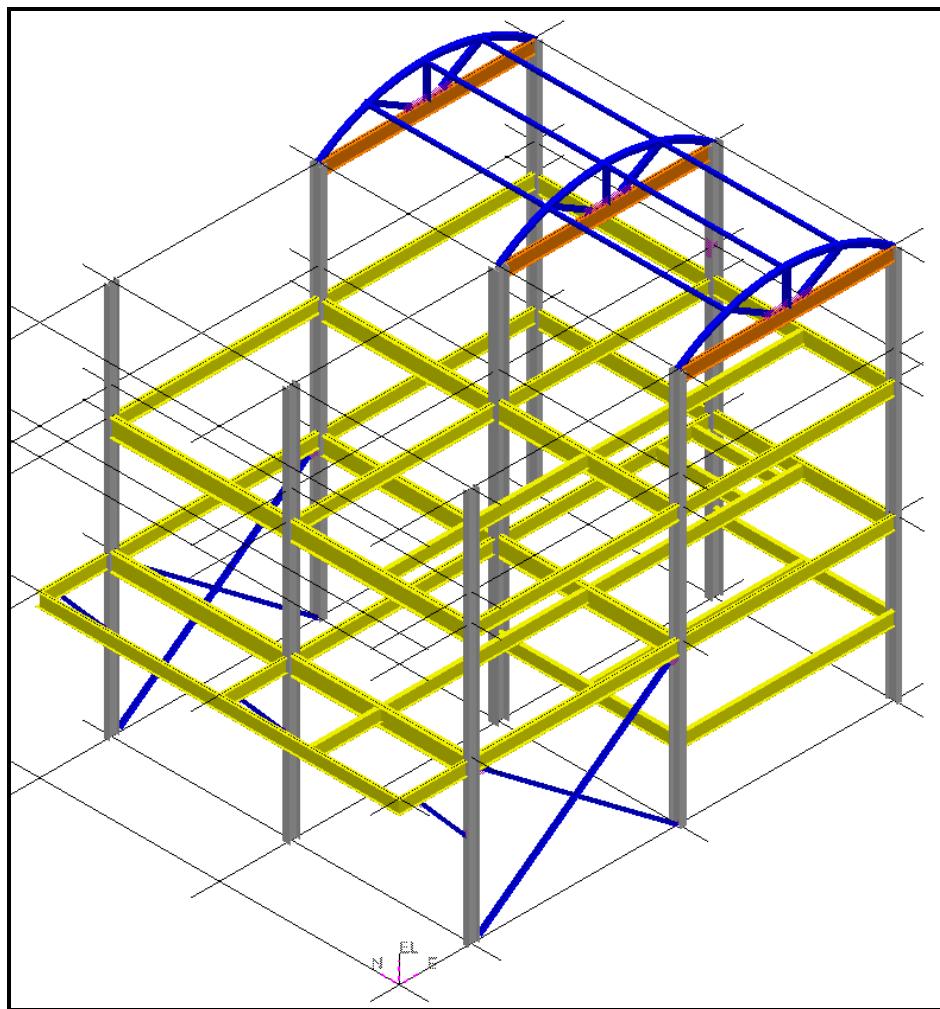
10. Save your session. Select File -> Save.
11. Re-define your workspace to include the A2 -> U03 and CS -> U03 CS systems.  
Repeat the above steps to create the assembly connections for structure U03.
12. Select Place Assembly Connection Command.
13. Set the parameters as

By Rule:  
Condition:  
System:

Check  
Retain existing  
A2-> U03 -> Structural -> Assembly Connections.



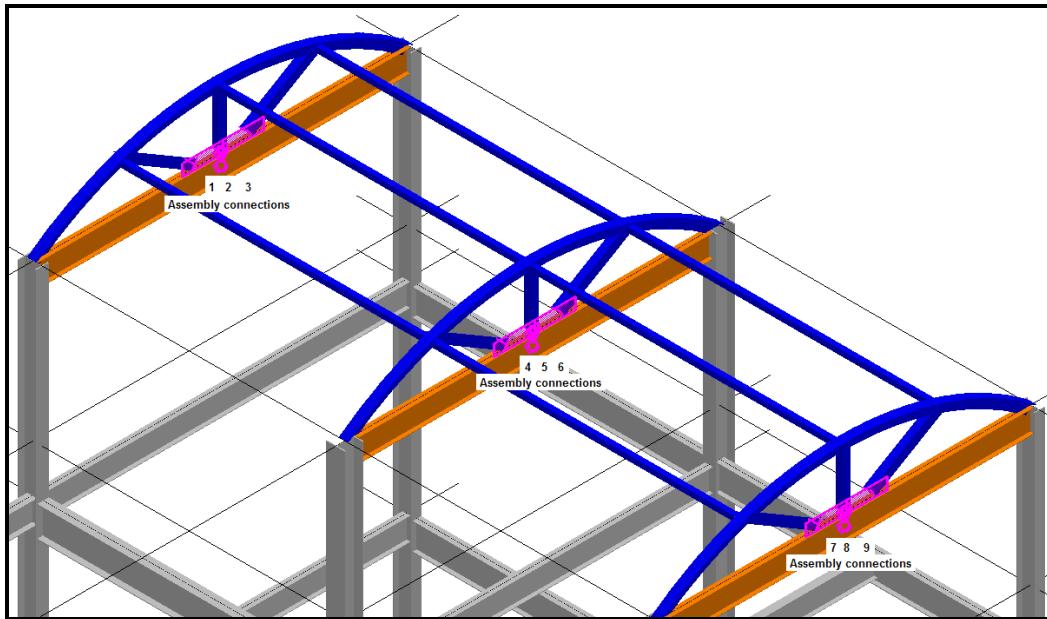
14. Hit “Finish” button to commit the transaction.
15. Examine the model and verify that the planar cutbacks, base plates and gusset plates have been created.



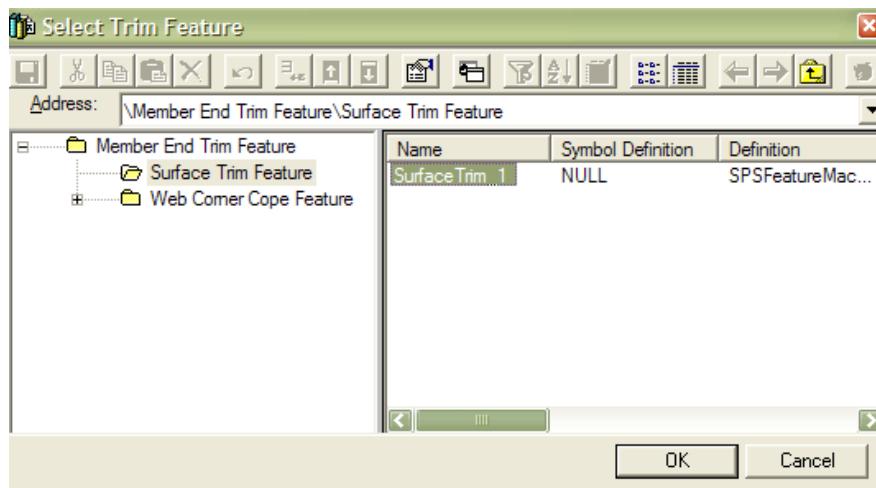
16. Use Zoom tool to zoom in around the rounded roof for the two-story building.
17. Set the locate filter to Member Assembly Connections.



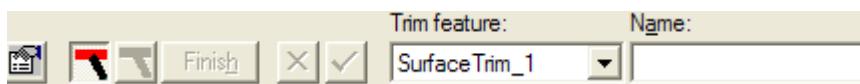
18. Multi-select the assembly connections located at the end of the braces. See figure below:



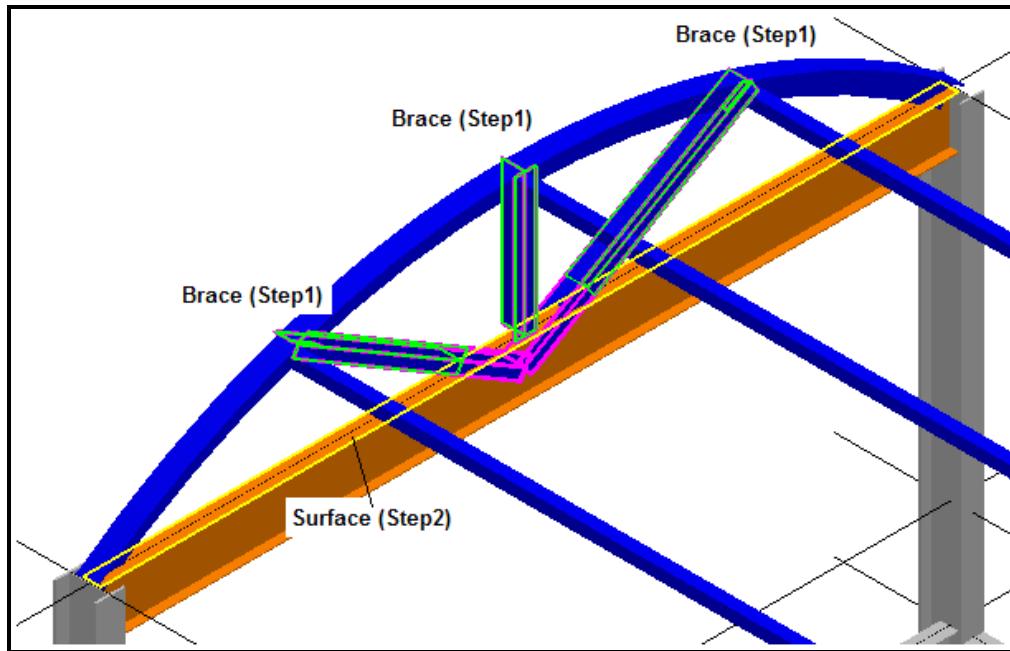
19. It should be nine objects in the select set. Click the Delete command to remove these objects.  
 20. Select Trim Member Command.  
 21. Expand the hierarchy and select the Surface Trim Feature folder.  
 22. Select Surface Trim 1.



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



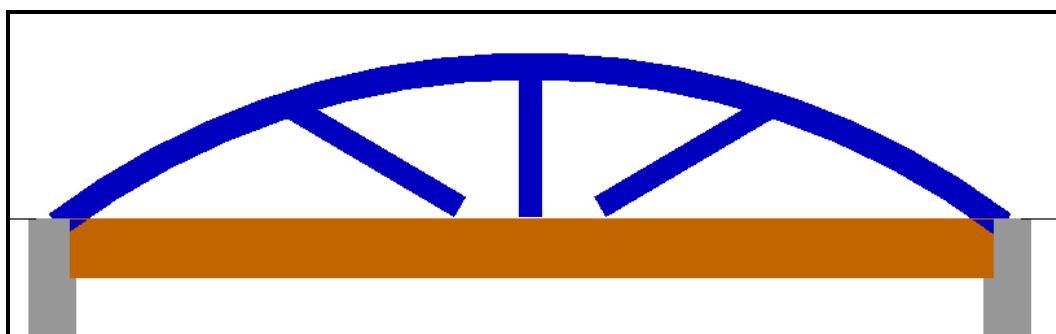
24. Select three vertical braces for the first SmartStep as shown below:



25. Select “Accept” button.
26. Select the top surface of the beam for the second SmartStep.
27. Select “Finish” button to commit the transaction.

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

28. Verify the cutbacks have been created.



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

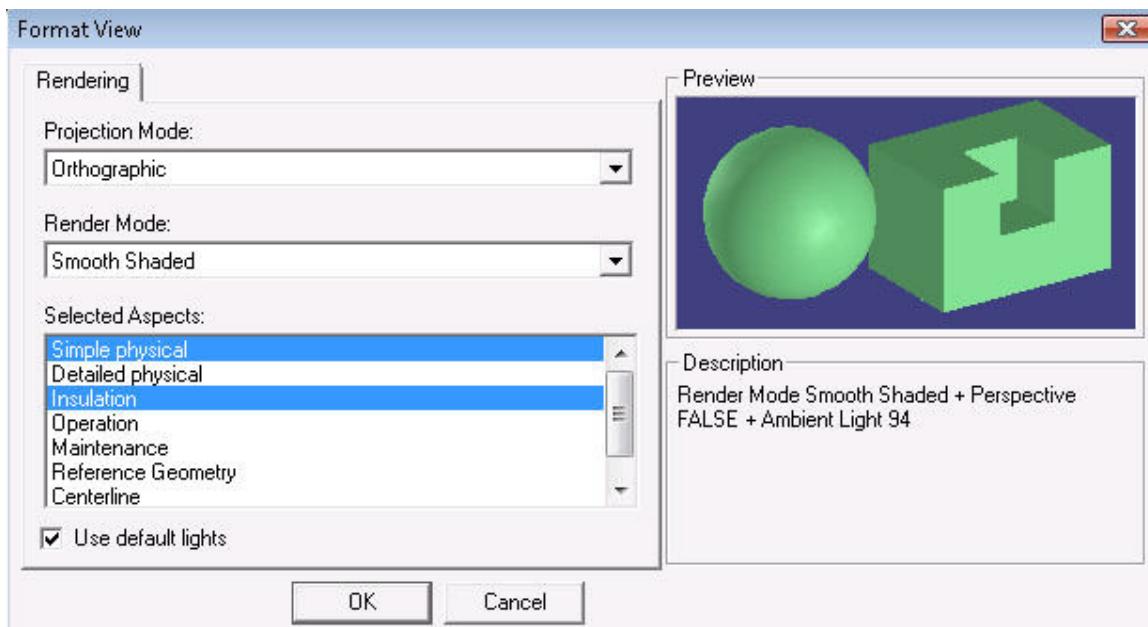
# LAB-6: Fireproofing

## Objectives

After completing this lab, you will be able to:

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

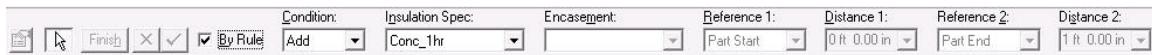
1. Open a session file.
2. Define your workspace to include the A2 → U02 and CS → U02 CS system.
3. Set your active permissions to “*Structural*”
4. Go to *Format* → *View*. Select Insulation in the rendering tab.



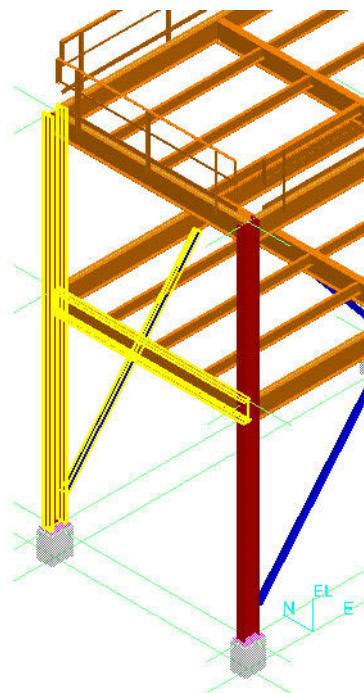
5. Your view should resemble the following graphic. Now, hit OK.
6. Go to File → Save.

### Part A: Fireproofing on a pre-existing structure

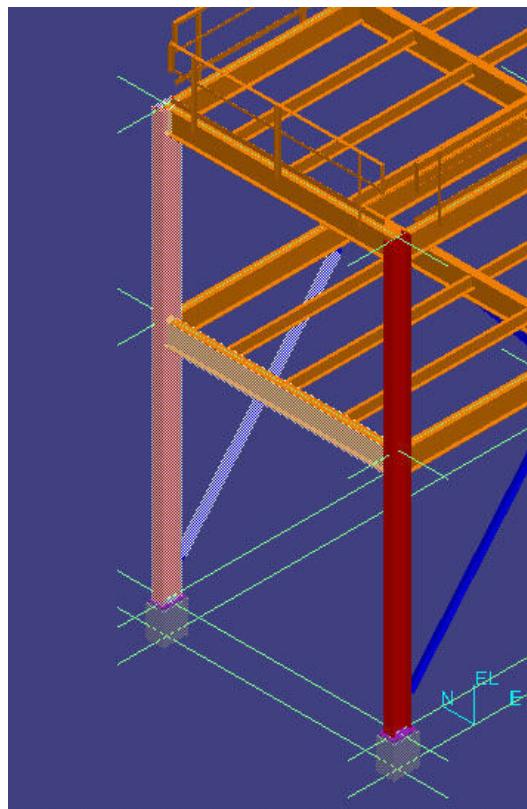
1. Select *Place Fireproofing* command. Insulation will be applied Using Place by rule. Make sure that the “*By Rule*” option is check marked.
2. Select “*Conc\_1hr*” from the drop down list in the “*Insulation Spec*” menu.
3. Select “*Add*” from the drop down list in the “*Condition*” menu.



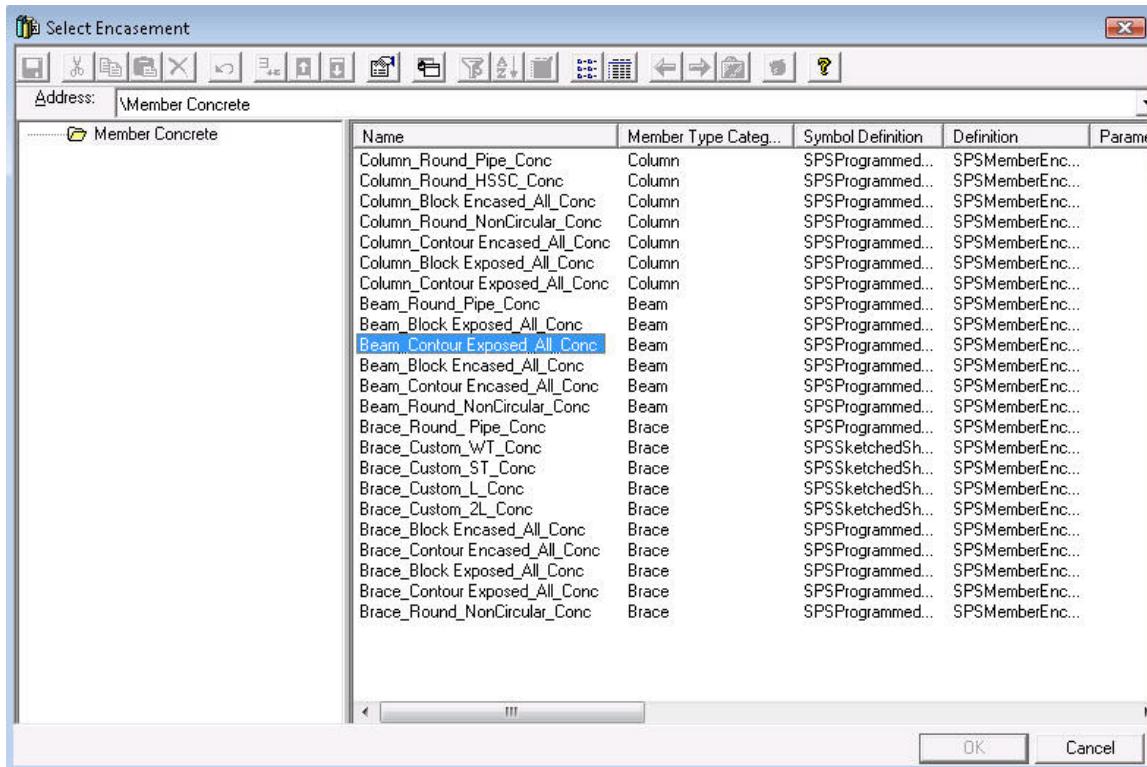
4. Select Column, Beam, Brace as shown in figure below,



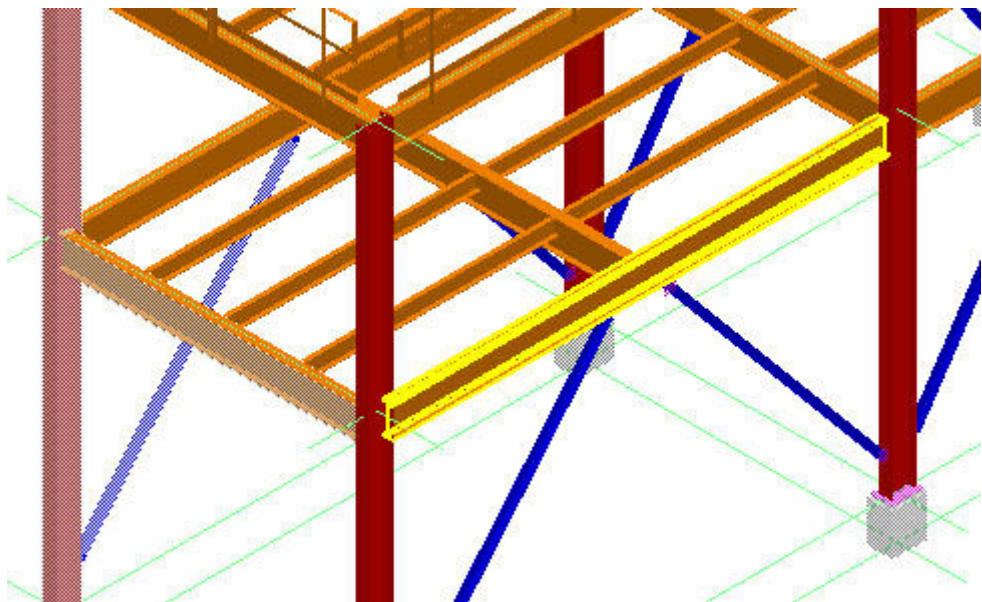
5. Hit the Accept button and then hit *Finish*. Your view should resemble the following graphic.



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

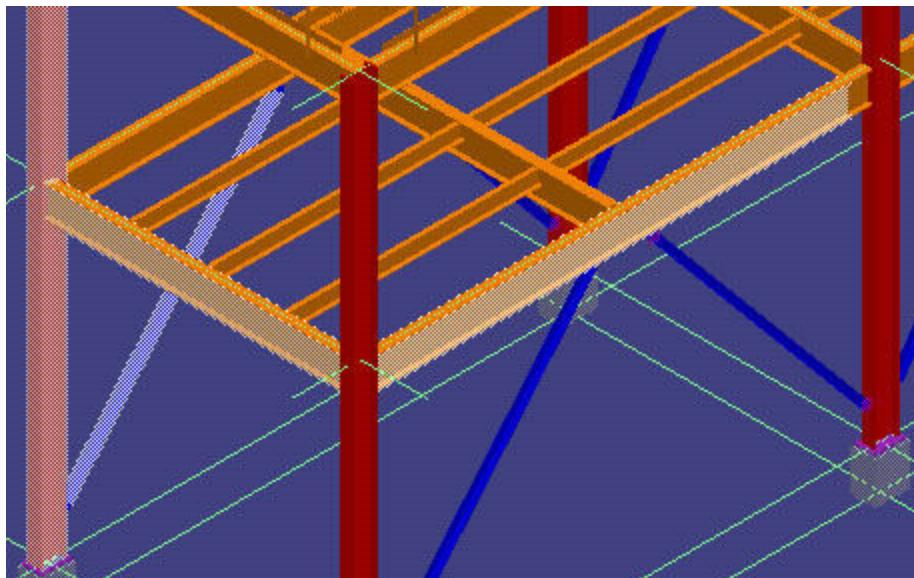


8. Select the “*Beam\_Contour\_Exposed\_All\_Conc*” Hit OK.
9. Select “*Part Start*” in Reference 1.
10. Enter “0” as Distance 1.
11. Select “*Part End*” in Reference 2.
12. Enter “1 ft” as Distance 2. This is the setback distance. This will not allow fireproofing to be applied on the beam at a distance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

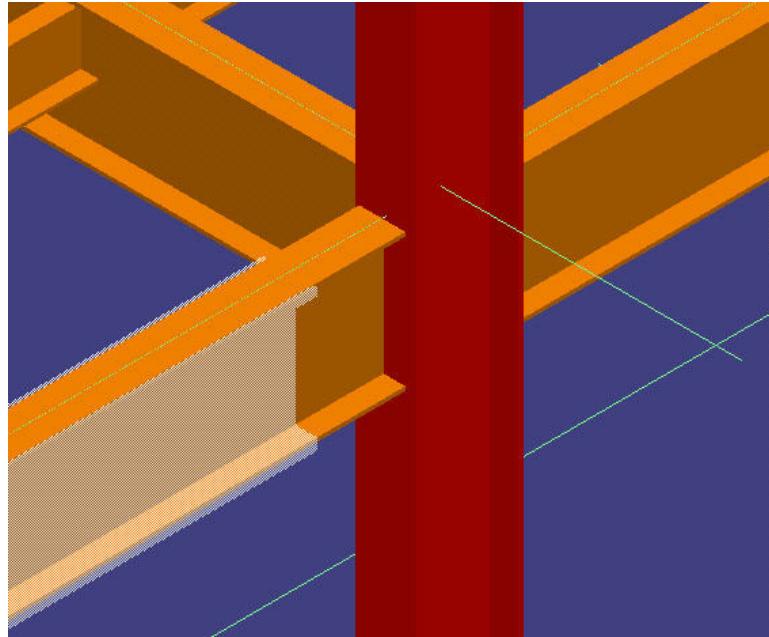


14. Hit the Accept button and then hit *Finish*.

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

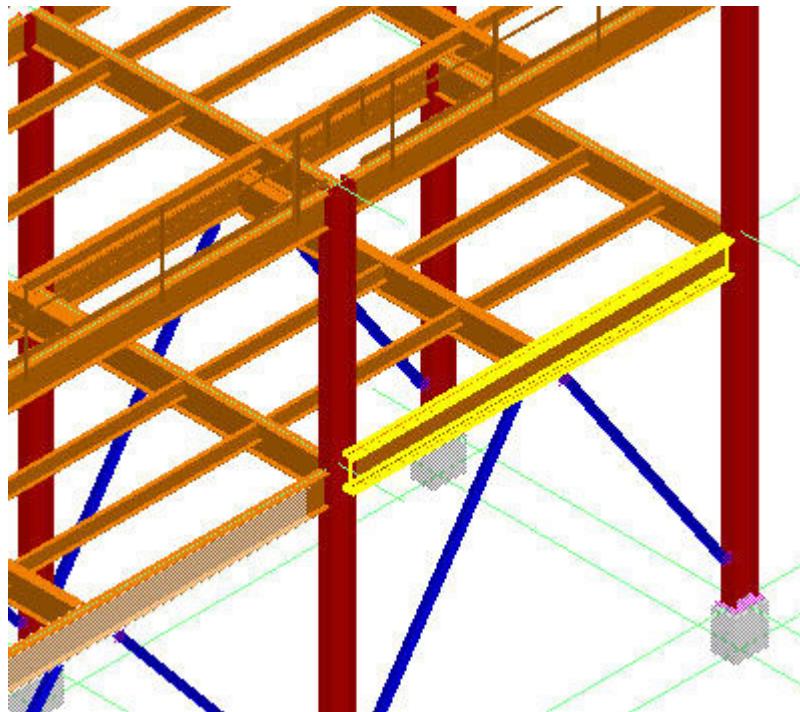


Please see below for close up image. This close up image shows the set back distance.

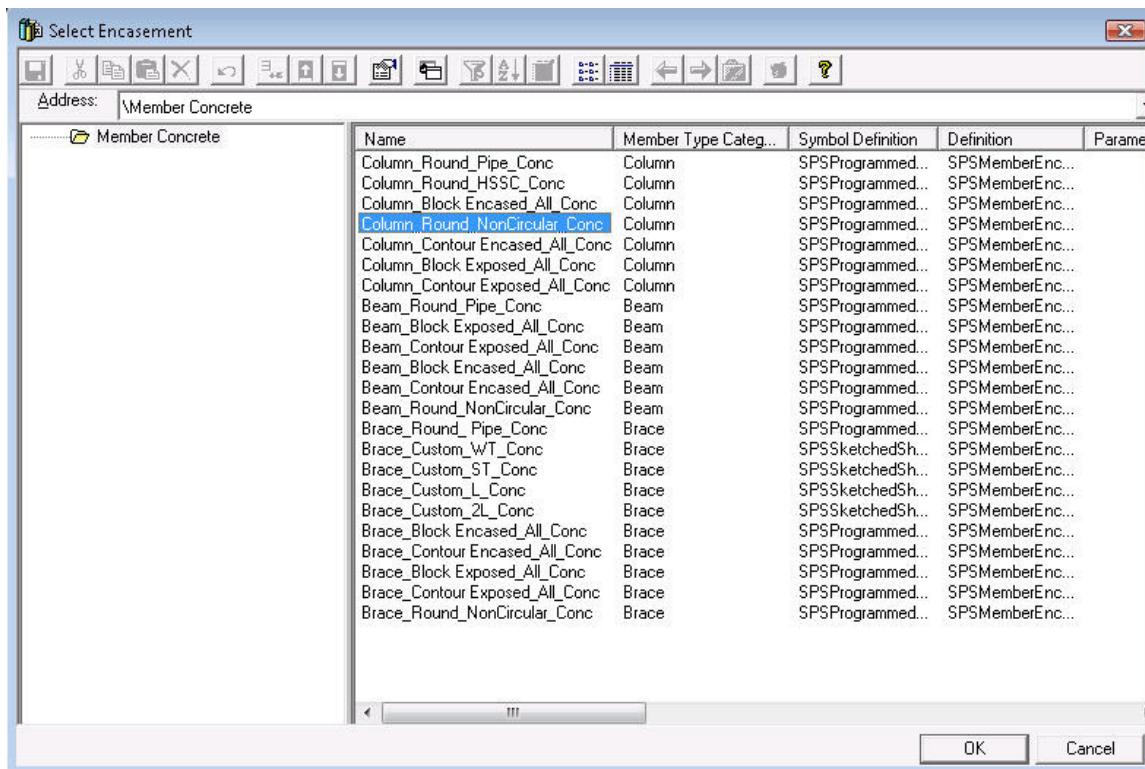


### Part B: Correct encasement on a pre-existing beam using “By Rule”

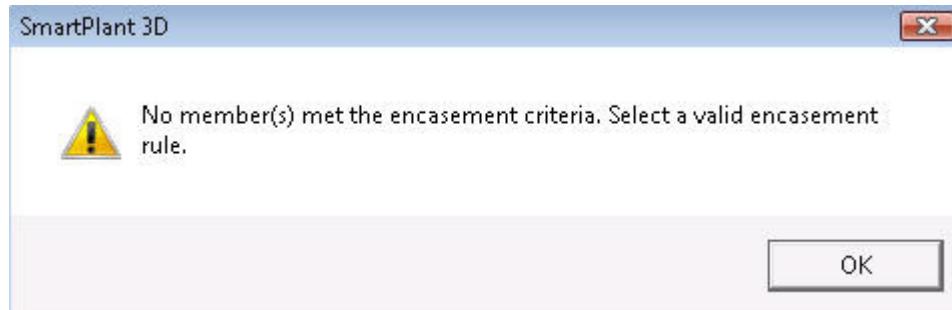
1. “*Place Fireproofing*” command will always apply column encasement to column and beam encasement to beam using “By Rule”.
2. Select the beam as shown in the figure below,



3. Insulation will be applied with the “User Defined” option. Make sure that “By Rule” option is 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.



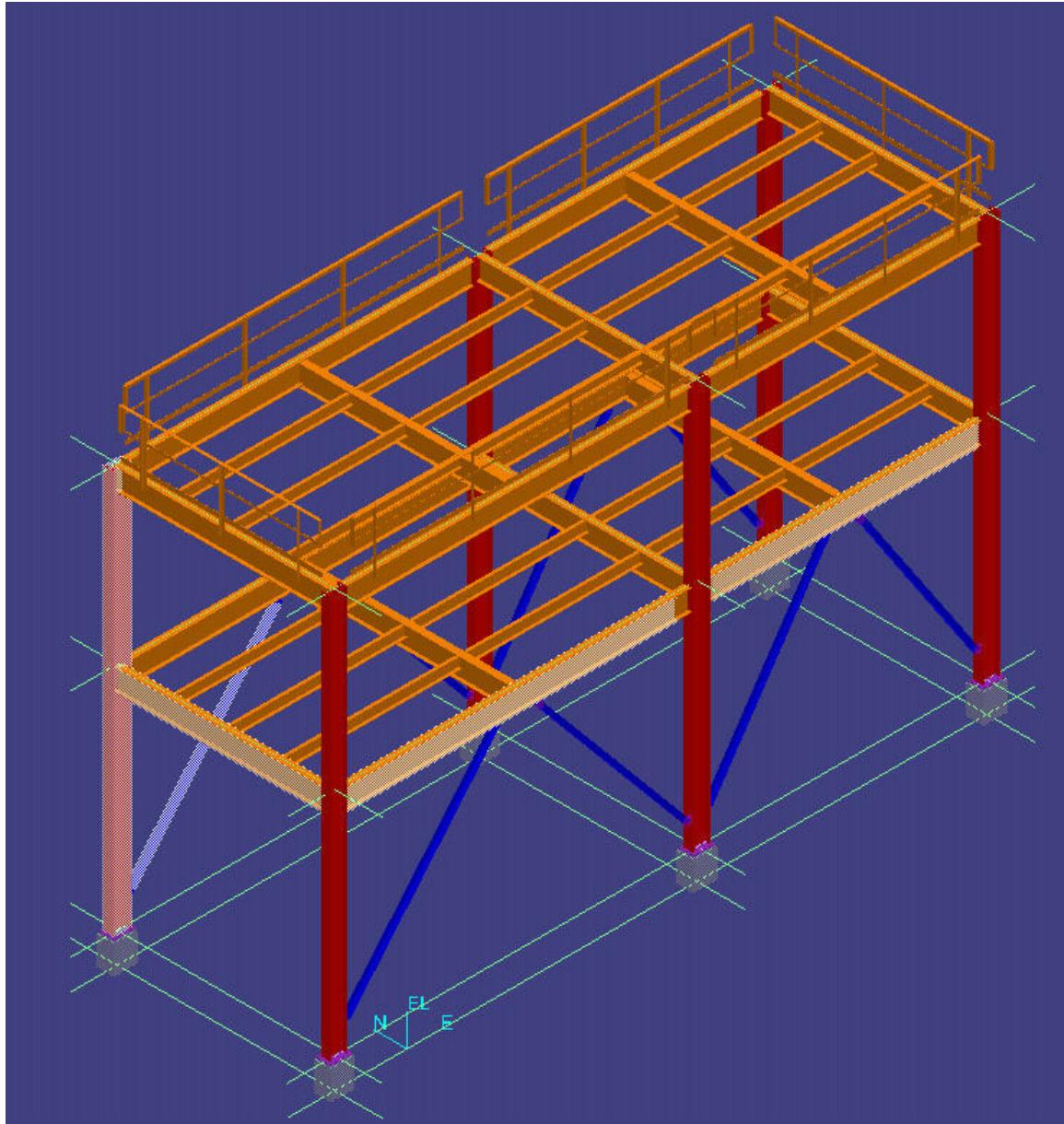
5. Select the “*Column Round NonCircular Conc*” Hit OK.
6. Select “*Part Start*” in Reference 1.
7. Enter “0” as Distance 1.
8. Select “*Part End*” in Reference 2.
9. Enter “0” as Distance 2.
10. Hit Accept. Warning message will be displayed. Your view should resemble the following graphic.



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.

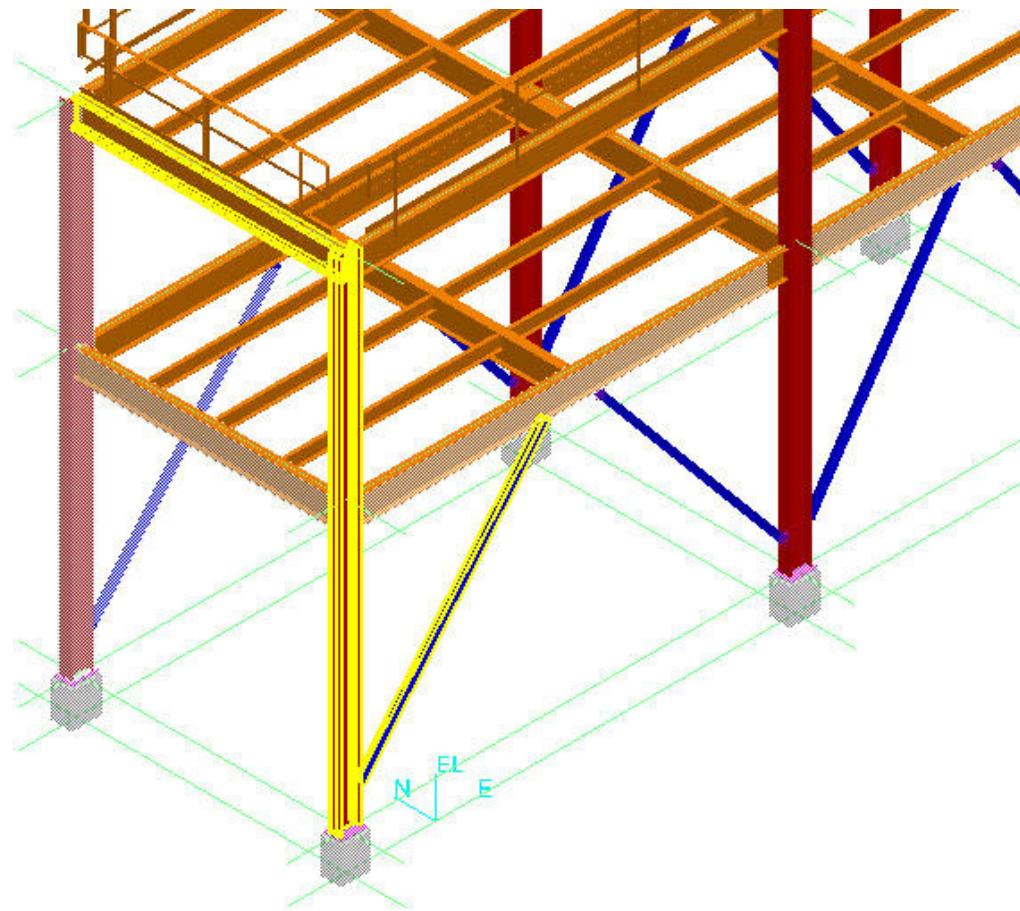


14. Hit Finish.
15. Hit Cancel. Your view should resemble the following graphic.

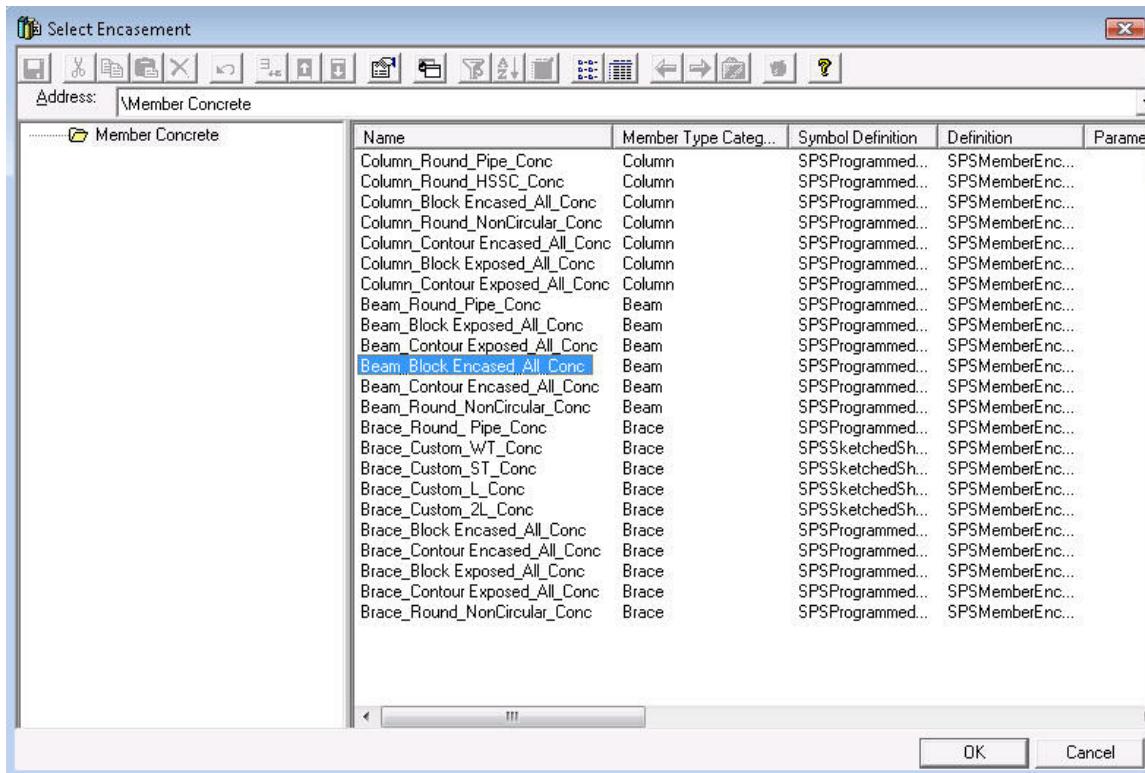


### Part C: User defined correct encasement on a pre-existing members

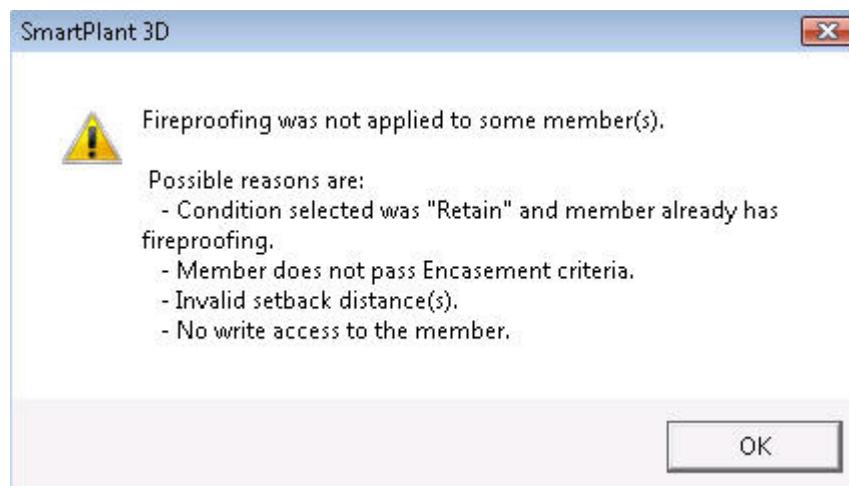
1. “*Place Fireproofing*” command will always apply column encasement to column and beam encasement to beam by selecting the correct encasement without using By Rule.
2. Select the “*Place Fireproofing*” command. Make sure that By Rule is unchecked.
3. Select the column, beam and brace as shown in the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



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



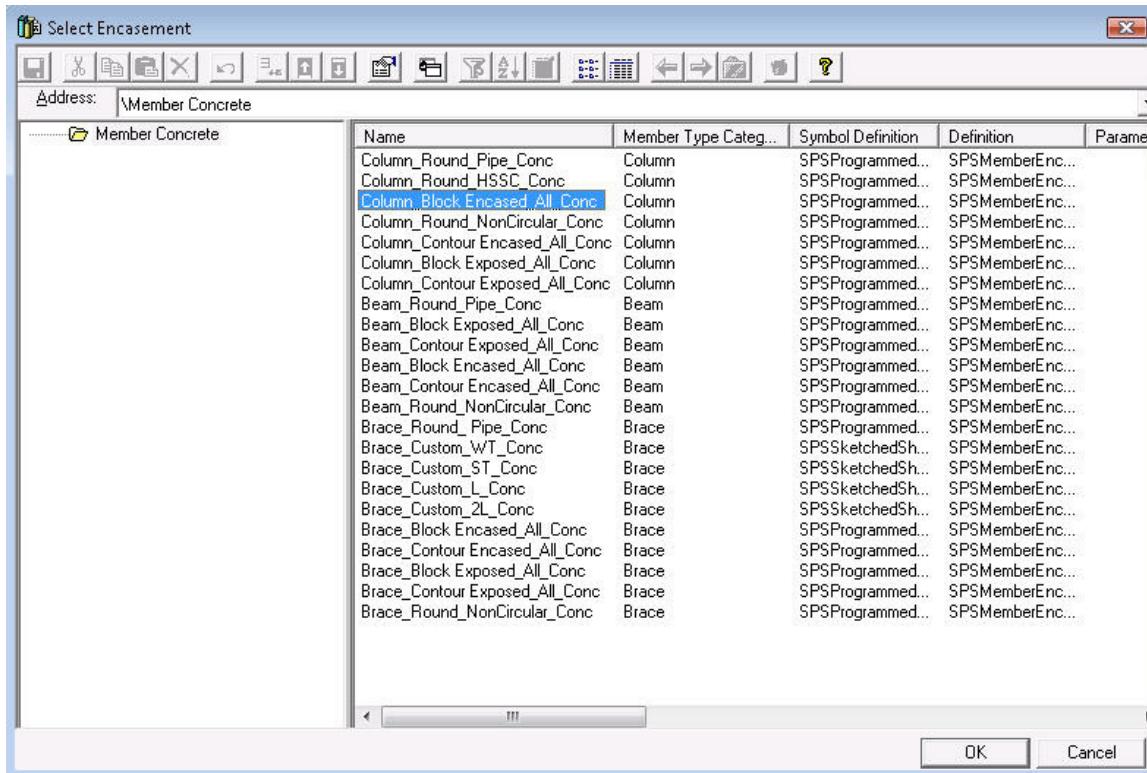
5. Select “Beam\_Block\_Encased\_All\_Conc” and Hit OK.
6. Hit Finish. Warning is shown.



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

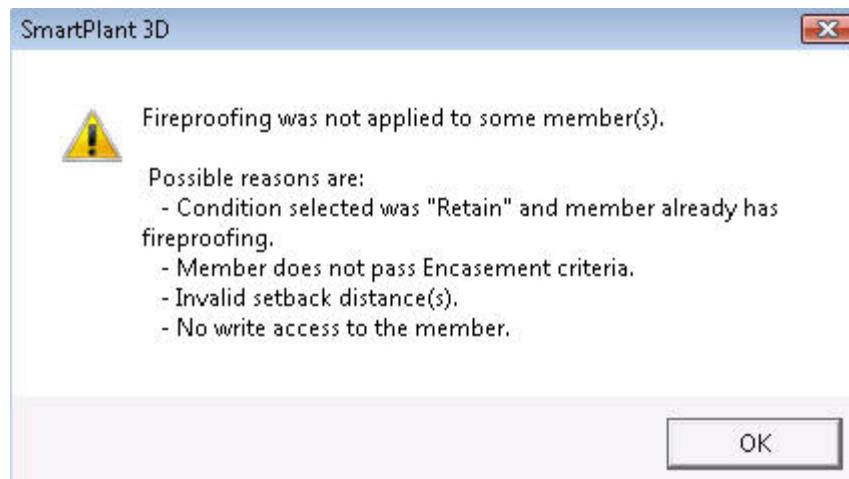
*Note: Selection is not lost.*

8. Go to more in the “Encasement” pull down list. Select Encasement window is shown.



9. Select “*Column\_Block\_Encased\_All\_Conc*” and Hit OK.

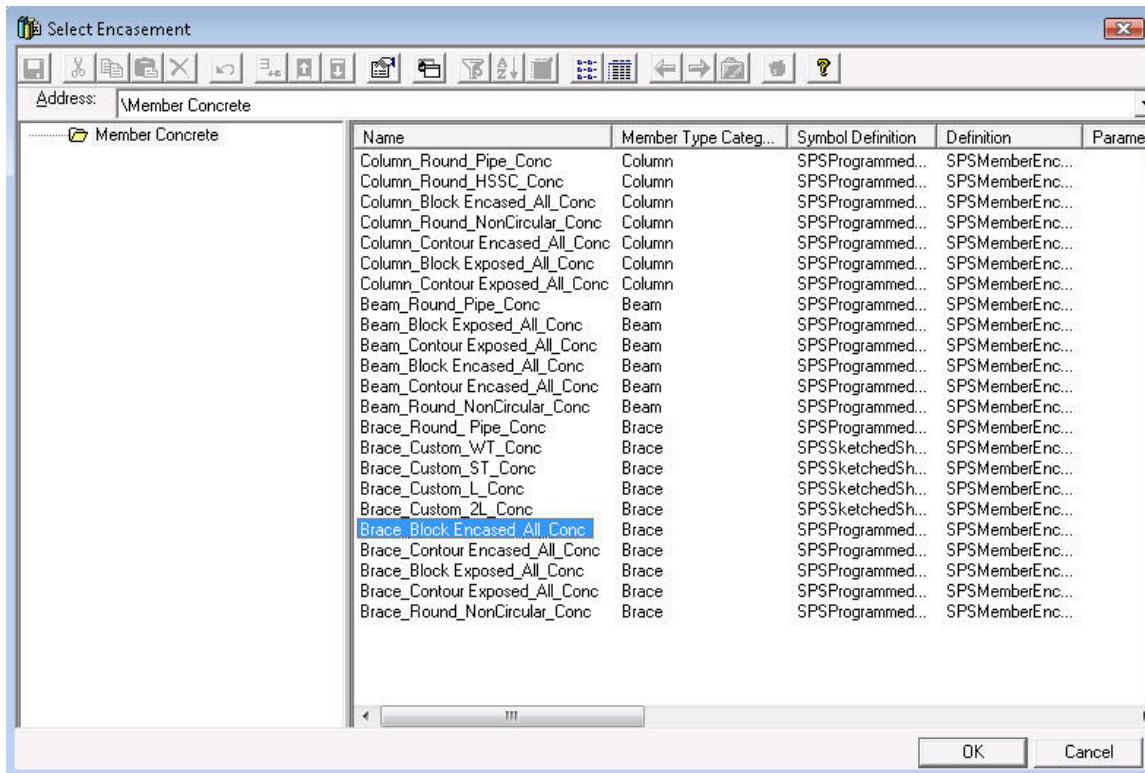
10. Hit Finish. Warning is shown.



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

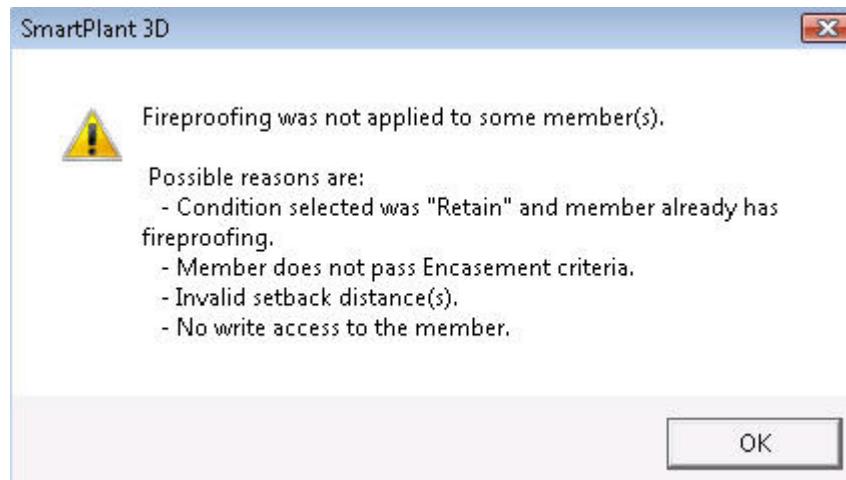
*Note: Selection is not lost.*

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.

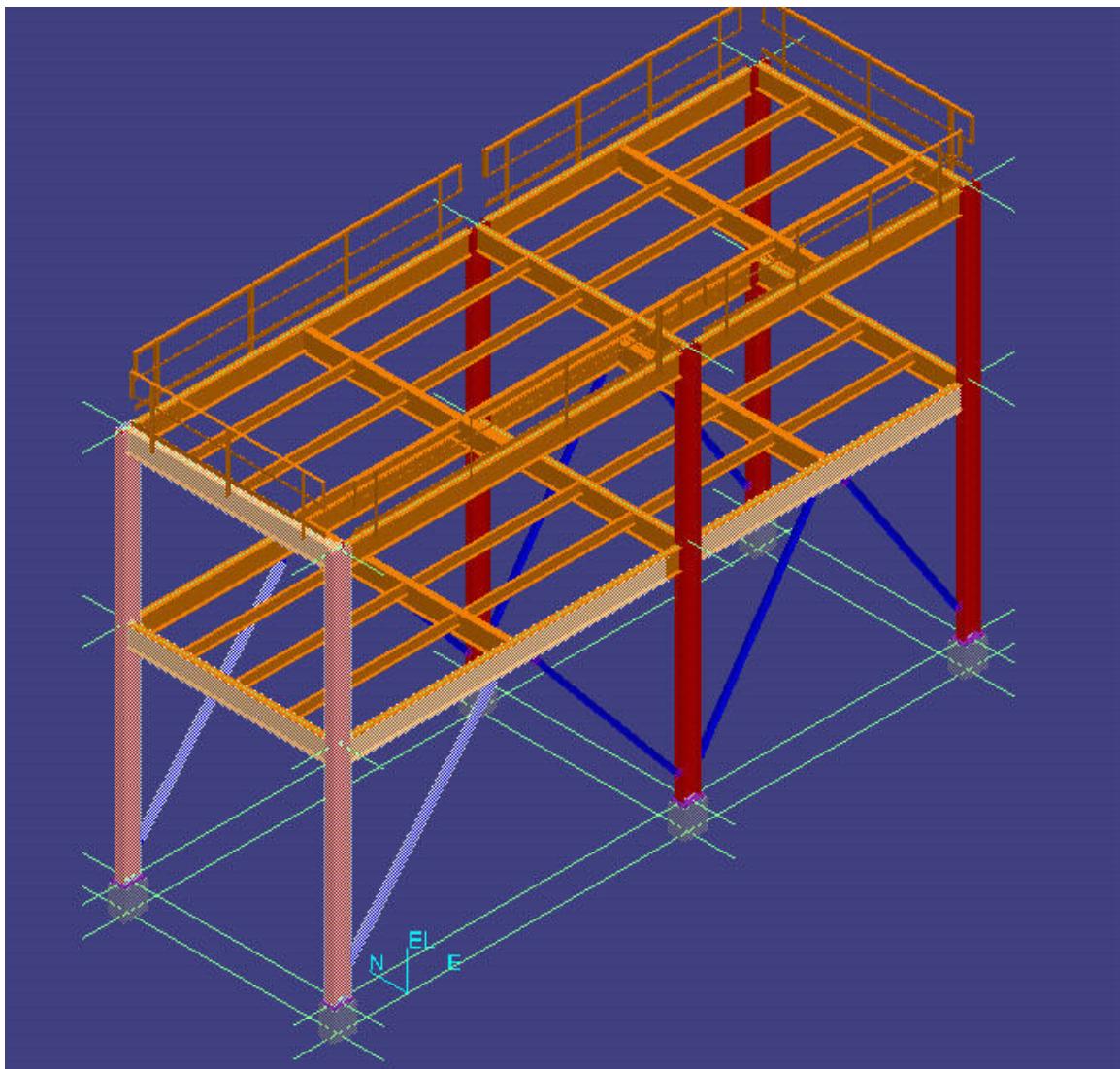
14. Hit Finish. Warning is shown.



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

*Note: Selection is not lost.*

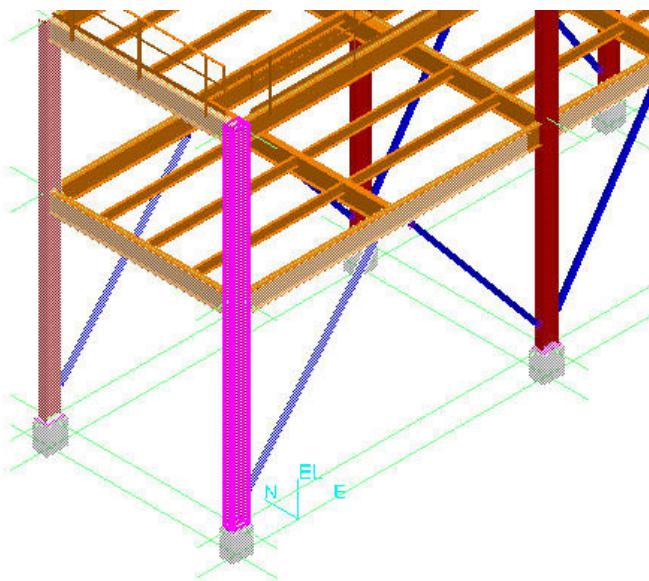
16. Hit Cancel. Your view should resemble the following graphic.



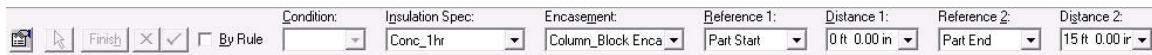
**Part D: Add Fireproofing to a column that already has fireproofing.**

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

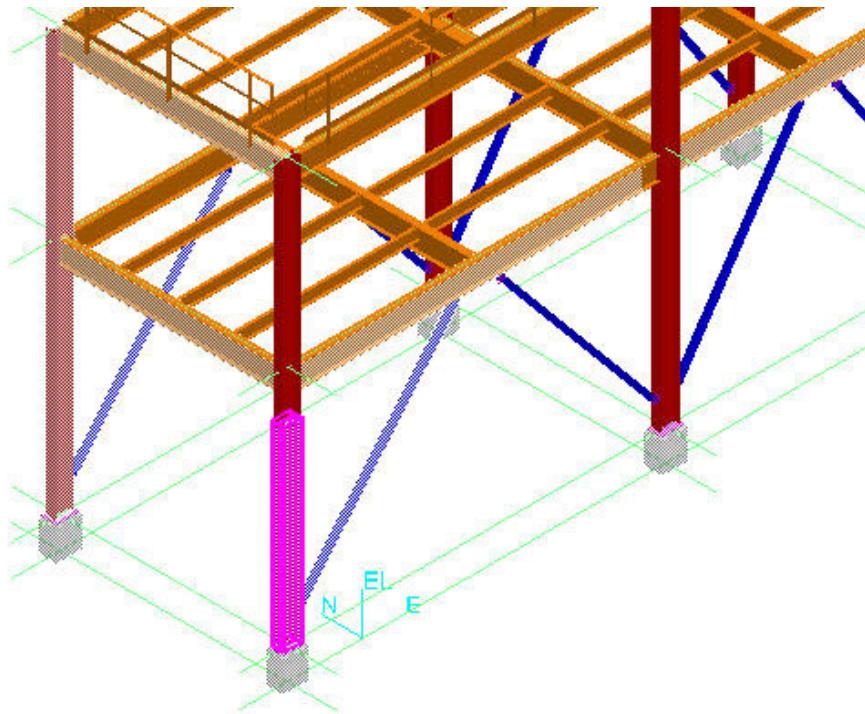




3. Change the setback distance to 15 ft and hit OK.

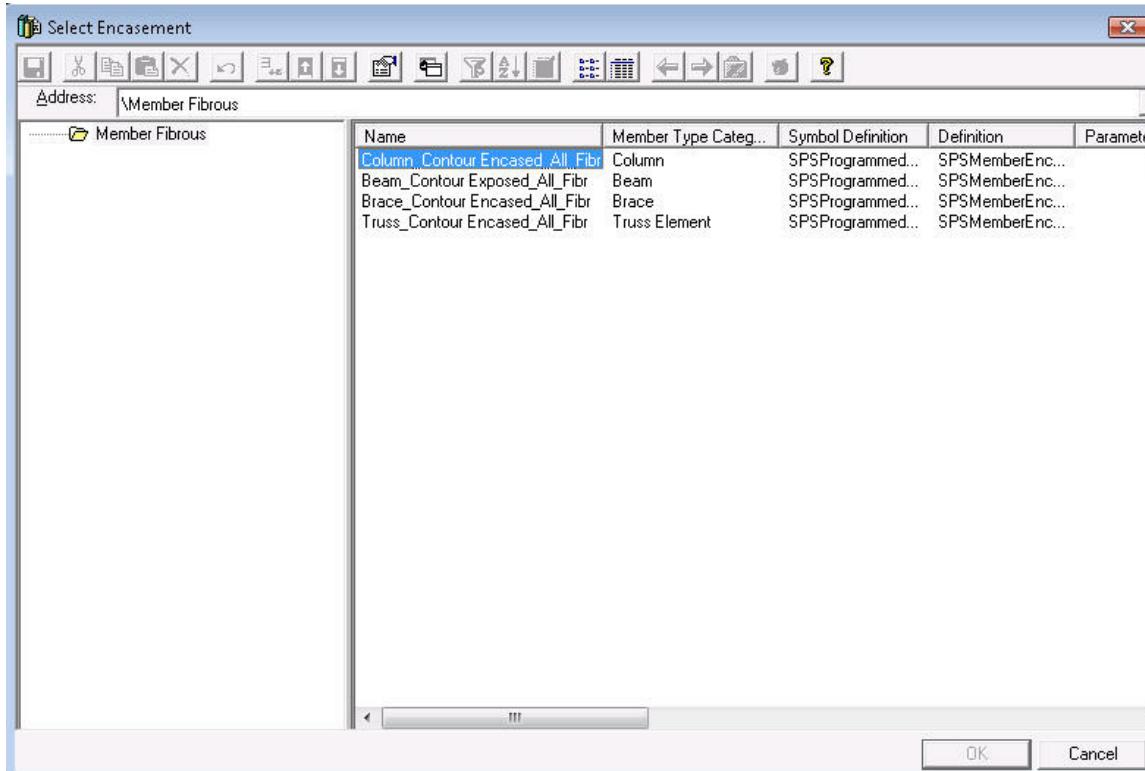


4. Your view should resemble the following graphic.



5. Place fireproofing command.
6. Change the Insulation Spec to Spray\_2hr.

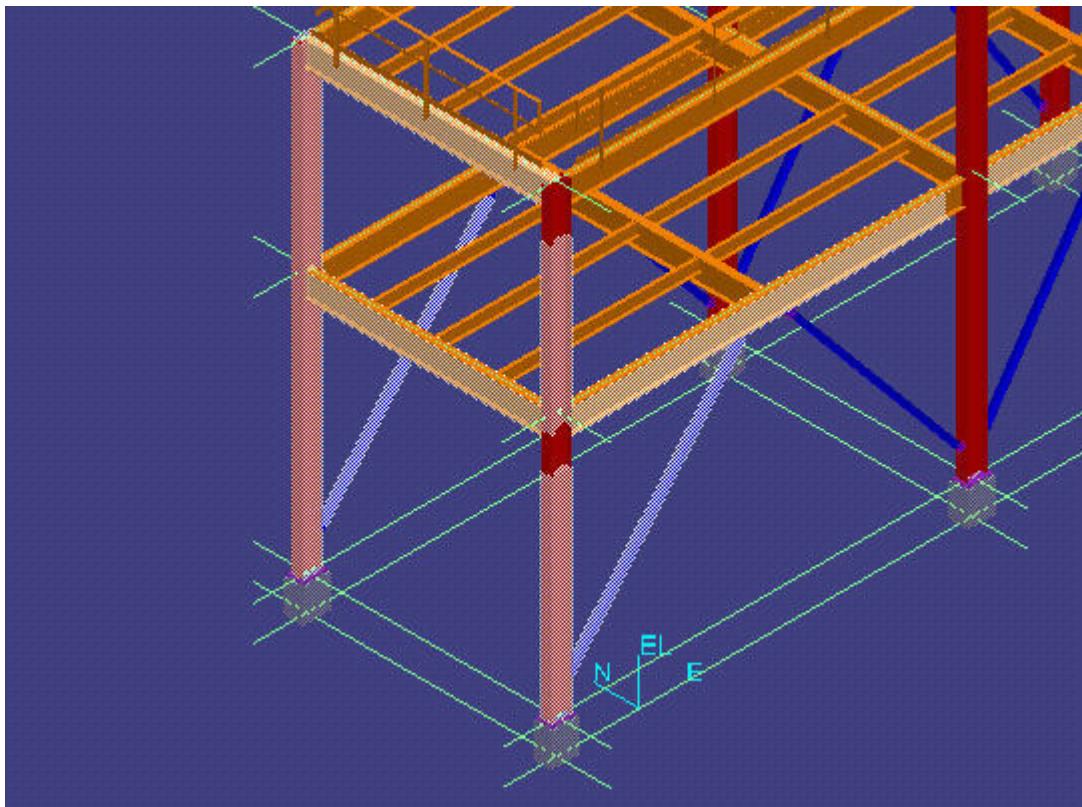
7. Go to More in the encasement menu. Select *Column\_Contour\_Encasement\_All\_Fibr*. Your view should resemble the following graphic.



8. Hit OK.  
 9. Select “*Part Start*” in Reference 1.  
 10. Enter “15 ft” as Distance 1.  
 11. Select “*Part End*” in Reference 2.  
 12. Enter “3 ft” as Distance 2.

<input type="button" value="New"/>	<input type="button" value="Edit"/>	<input type="button" value="Finish"/>	<input type="button" value="X"/>	<input type="checkbox"/> By Rule	Condition:	Insulation Spec:	Encasement:	Reference 1:	Distance 1:	Reference 2:	Distance 2:
						Spray_2hr	Column_Contour Enc...	Part Start	15 ft 0.00 in	Part End	3 ft 0.00 in

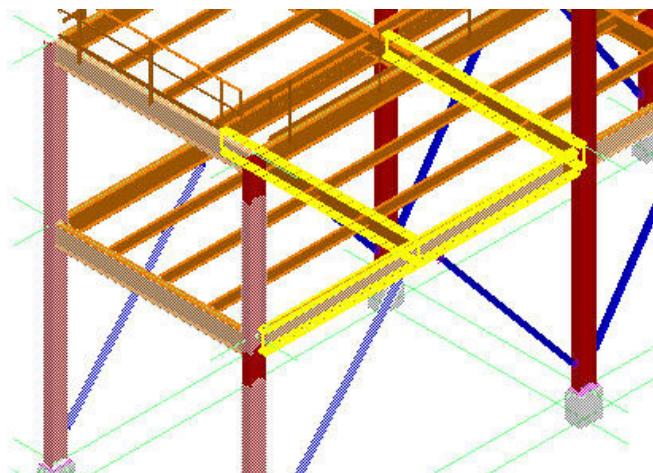
13. Hit the Accept button and hit Finish. Your view should resemble the following graphic.



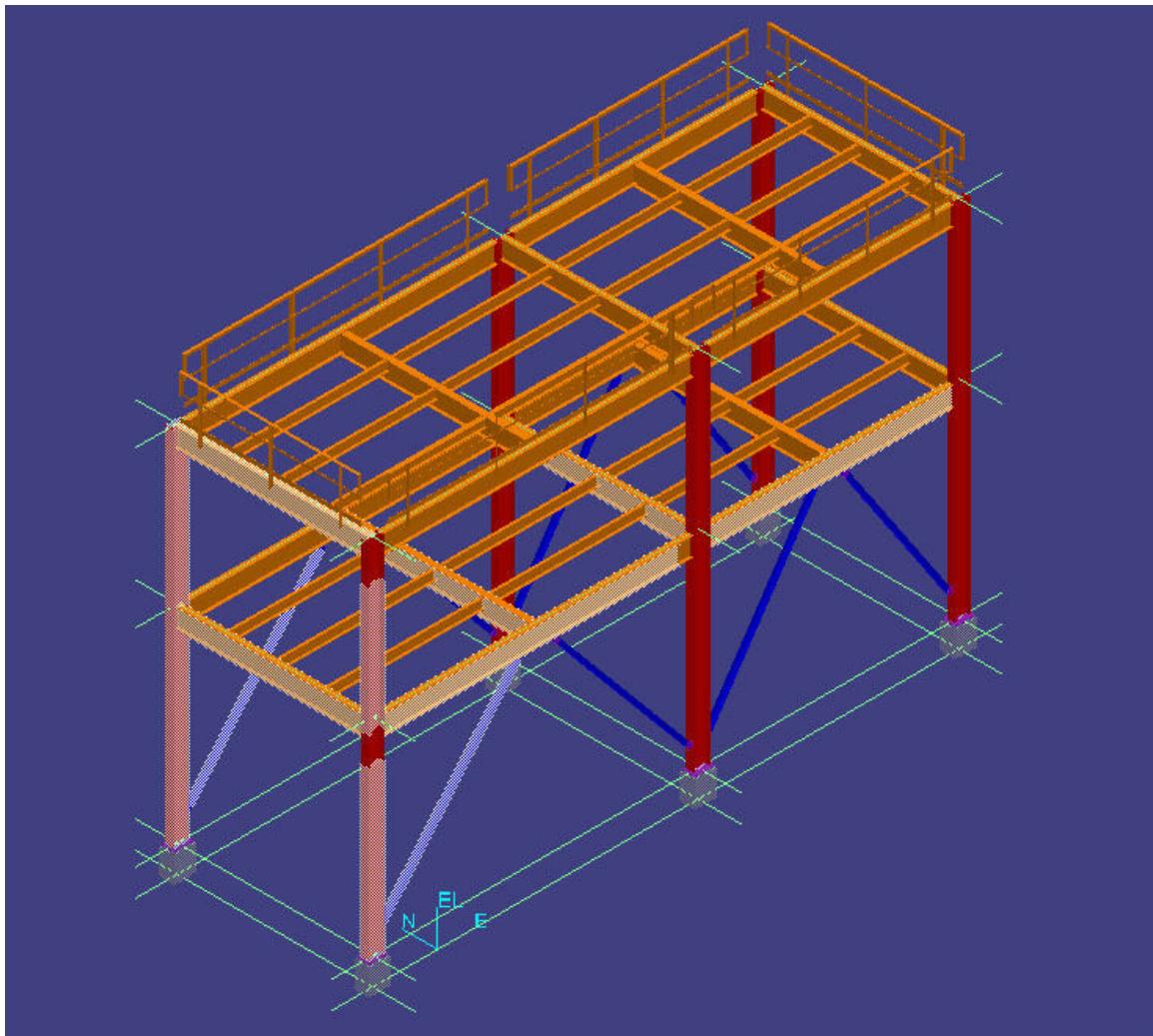
14. Hit Cancel.

### Part E: Retain fireproofing.

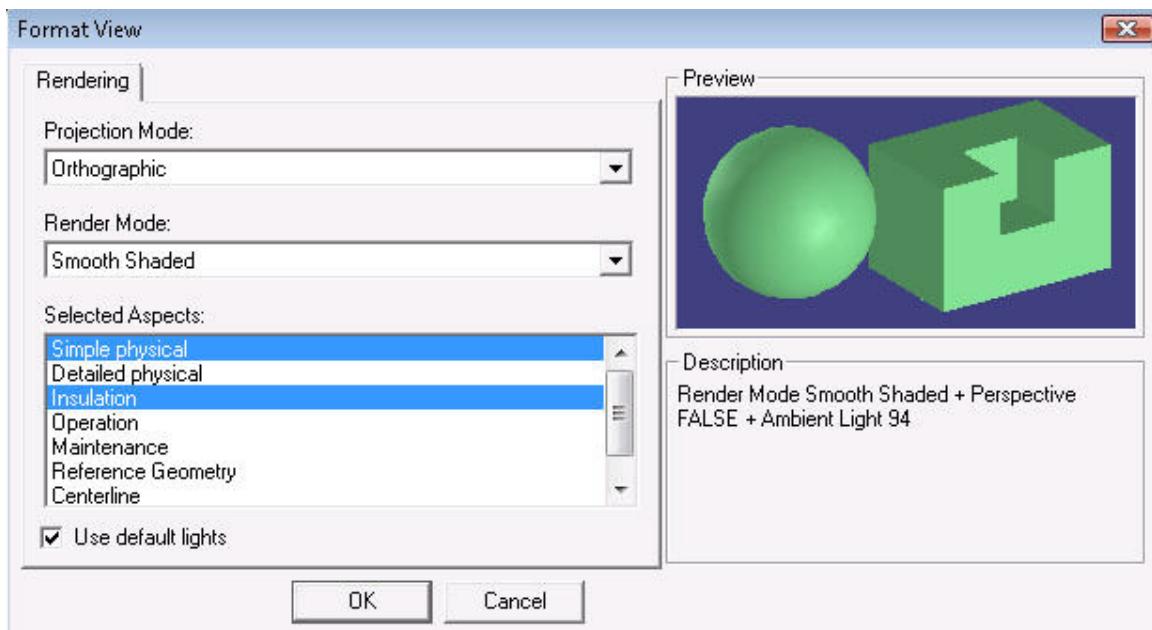
1. Select the *Place Fireproofing* command.
2. Select beams (one beam on which fireproofing has already been applied and two other beams that have no fireproofing). Your view should resemble the following graphic.



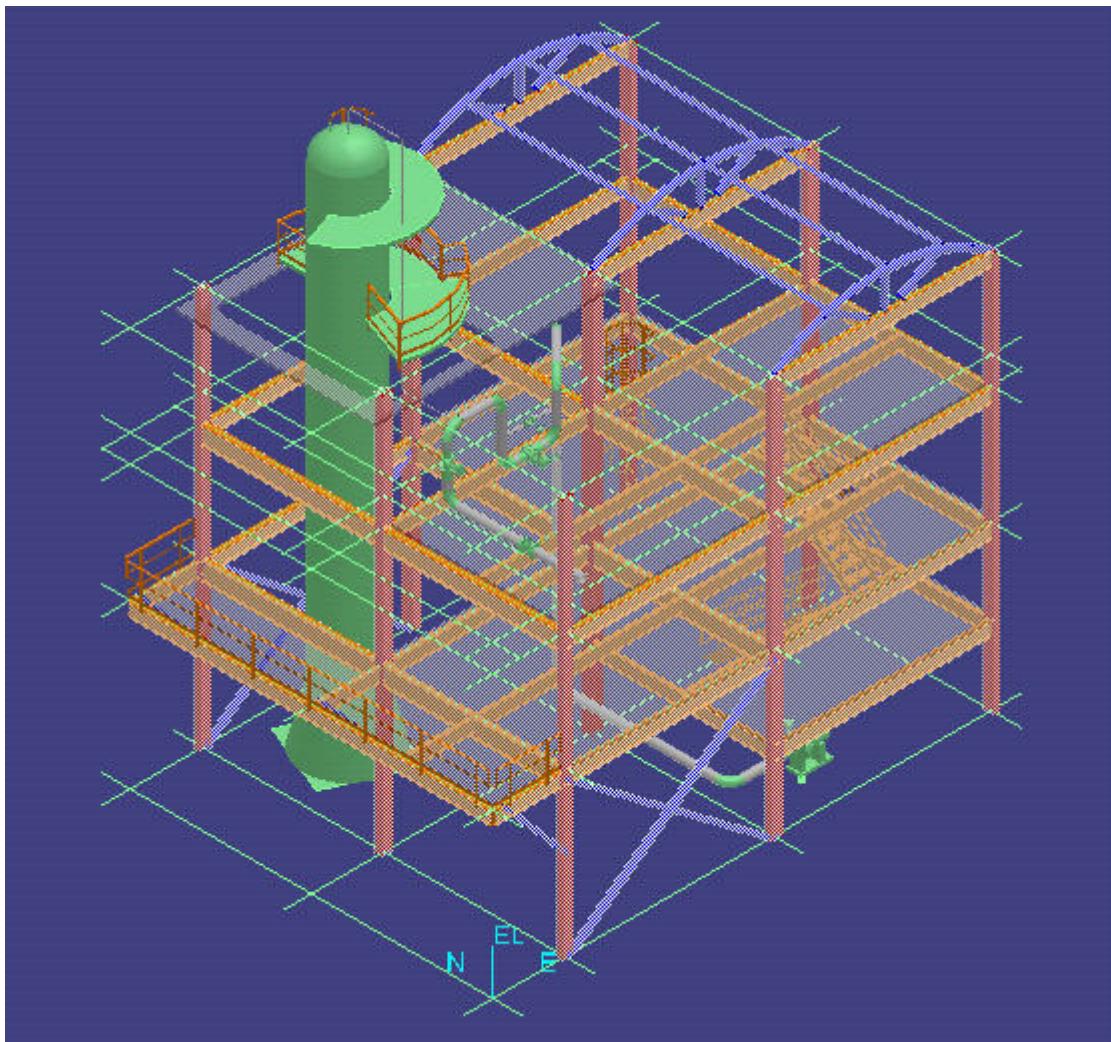
3. Change the *Condition* to “*Retain*” in the toolbar.
4. Select the *Insulation Spec* as Conc \_1hr.
5. Select “By Rule”.
6. Hit Finish. Warning will be shown. Hit OK. Hit Cancel. Fireproofing is retained on the existing beam and fireproofing is applied on the other two beams. Your view should resemble the following graphic.

**Part F: Place fireproofing for the entire structure.**

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

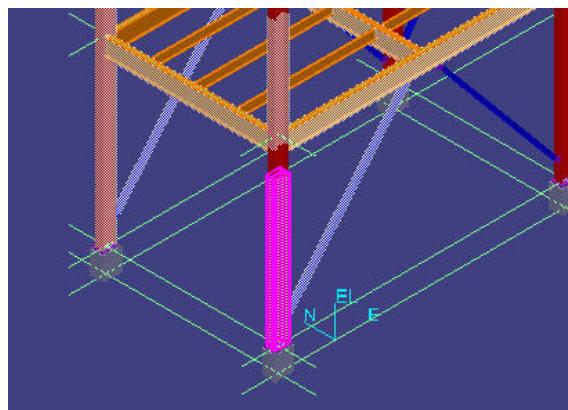


5. Your view should resemble the following graphic. Now, hit OK.
6. Select “*Place Fireproofing*” command.
7. Fence select the entire structure.
7. Select the *Condition* as “*Add*” in the toolbar.
8. Select the *Insulation Spec* as Conc\_1hr.
9. Select “*By Rule*”.
8. Hit Finish. Hit OK. Hit Cancel. Fireproofing is applied to the entire structure. Your view should resemble the following graphic.

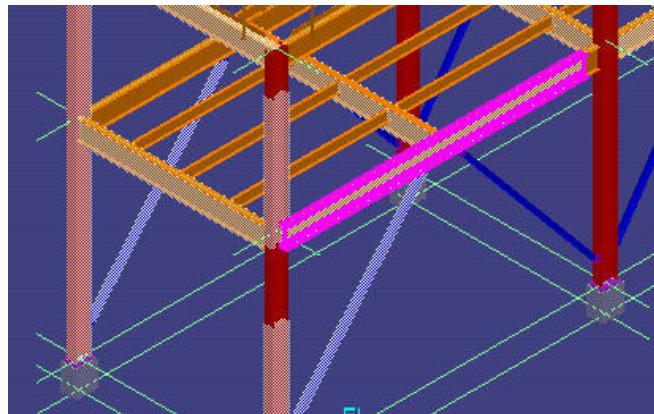


### Part G: Modification.

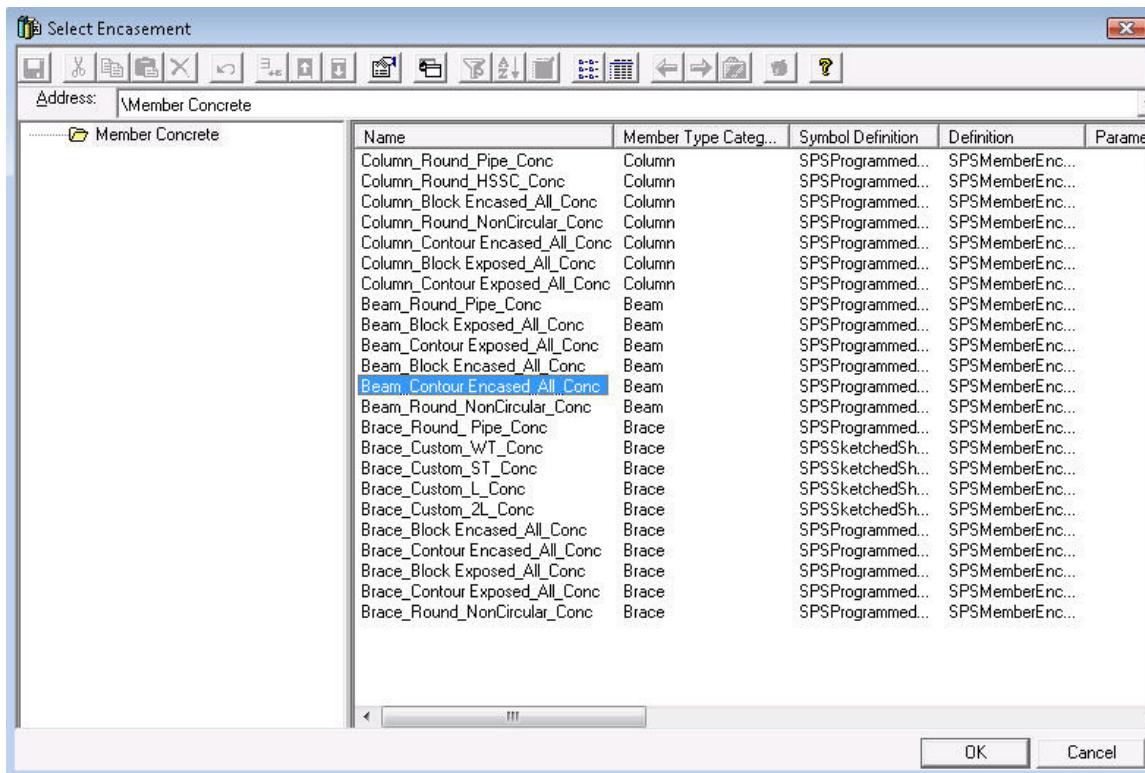
1. Define your workspace to include the A2 → U02 and CS → U02 CS system.
2. Set the Locate Filter to *Insulation*. 
3. Select the insulation as shown in the image below,



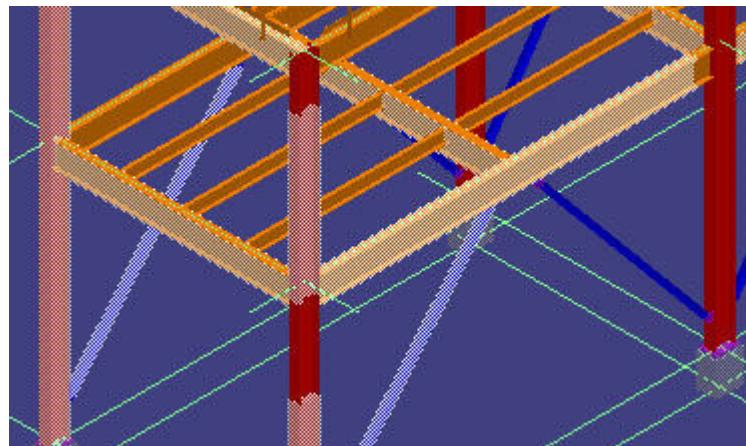
4. Change the *Part Start* distance from 0 ft to 2 ft.
5. Change the *Part End* distance from 15 ft to 18 ft.
6. Select the insulation for the beam as shown in the image below,



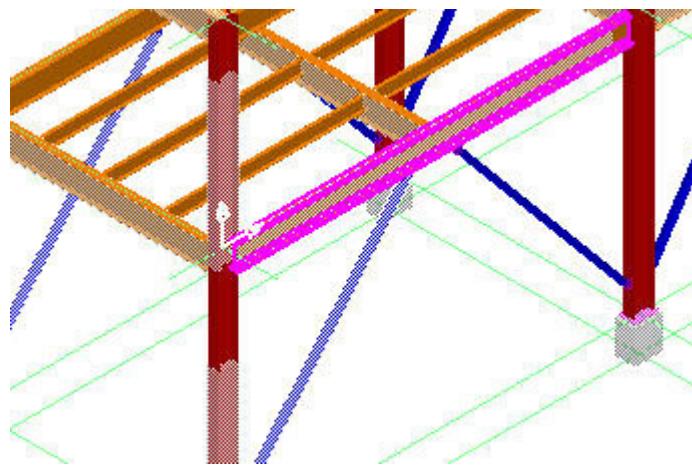
7. Go to More from the encasement pull down list. Your view should resemble the following graphic.



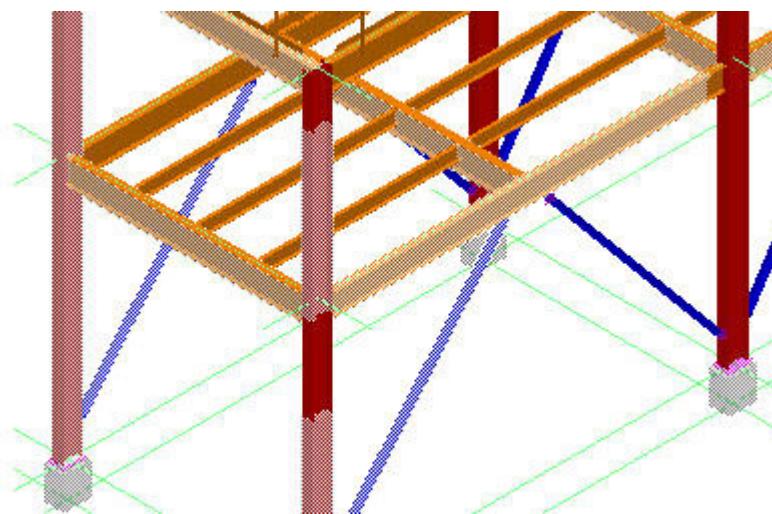
8. Change the encasement from “*Beam\_Contour\_Exposed\_All\_Conc*” to “*Beam\_Contour\_Encased\_All\_Conc*”. Hit OK.
9. The encasement has changed. Your view should resemble the following graphic.



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



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

# LAB-7: Slabs

## Objectives

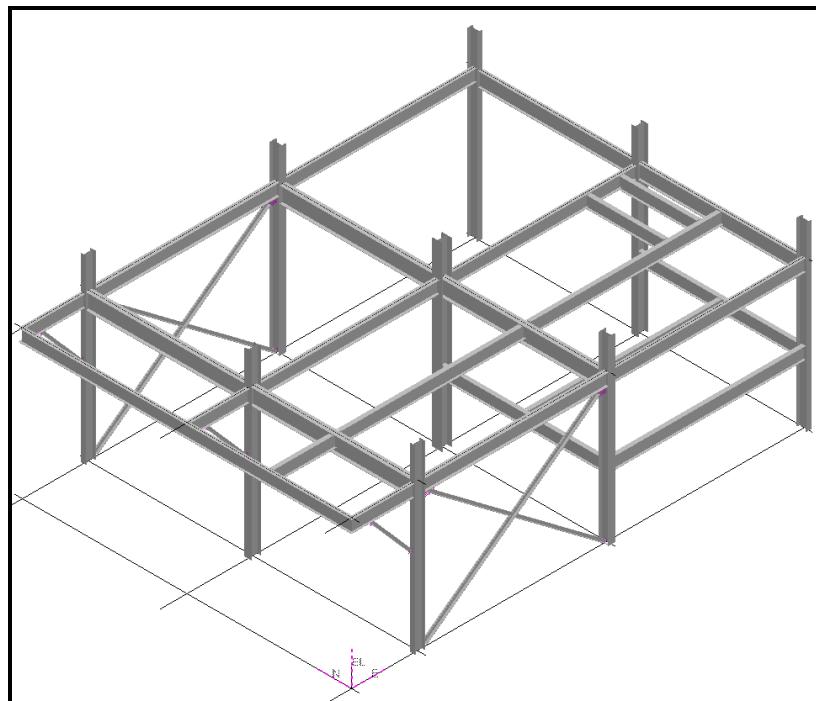
After completing this lab, you will be able to:

- 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 First Floor:**

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



5. Select Place Slab command to place one slab on the first floor.

6. Set the slab placement parameters as follows:

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

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

8. Select the boundaries as shown in Figure 1. Click "Accept" button.

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

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

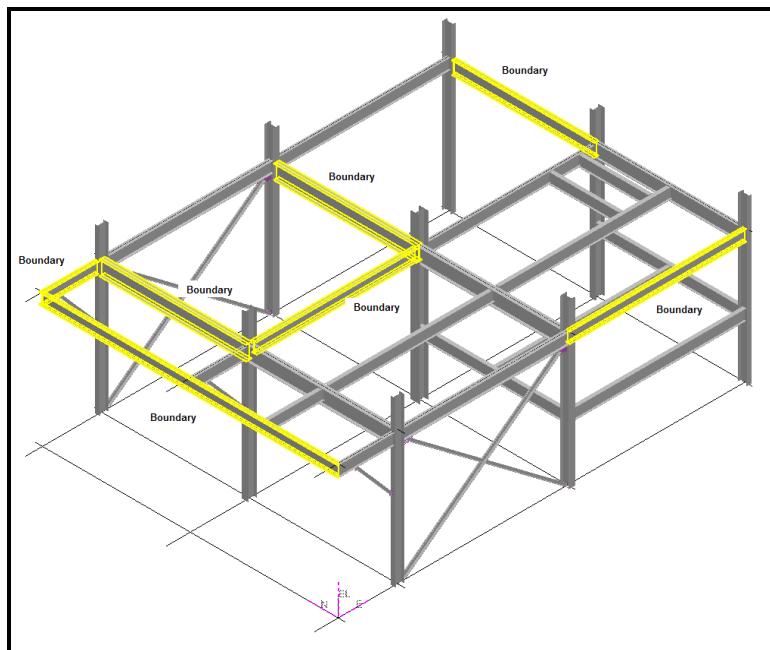


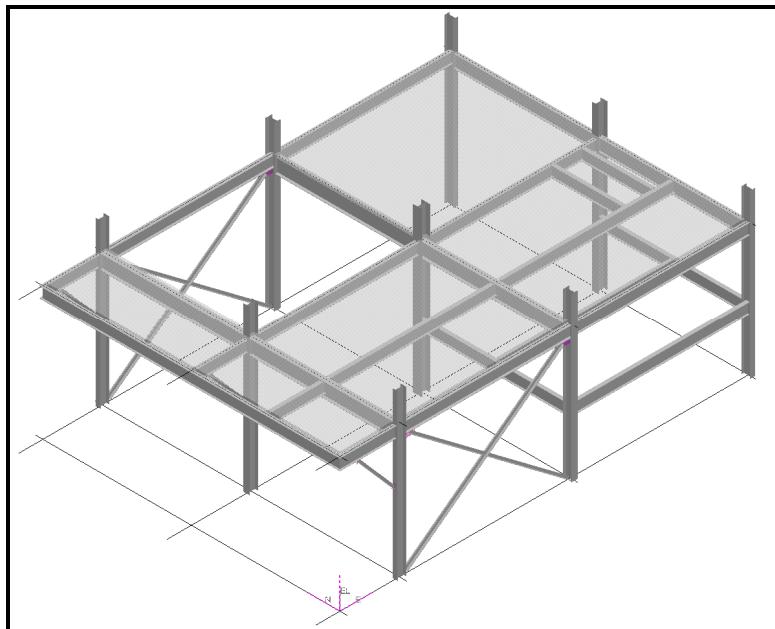
Figure 1 – ISO View of the First Floor Frame

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

10. Skip Define the Boundaries offsets step

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

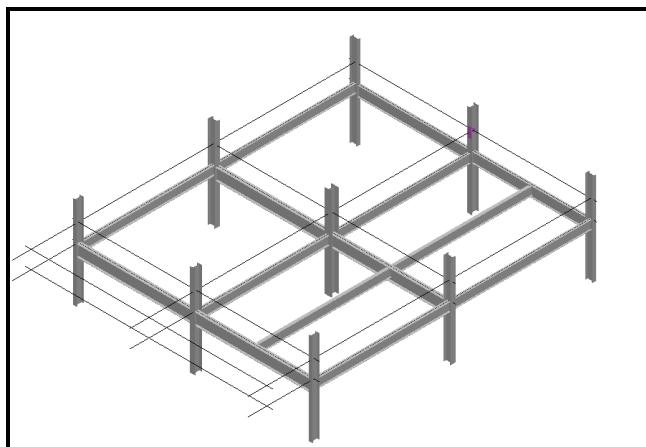
Your View should now resemble the following graphic:



#### **Part II: Place Slab on Second Floor:**

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

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



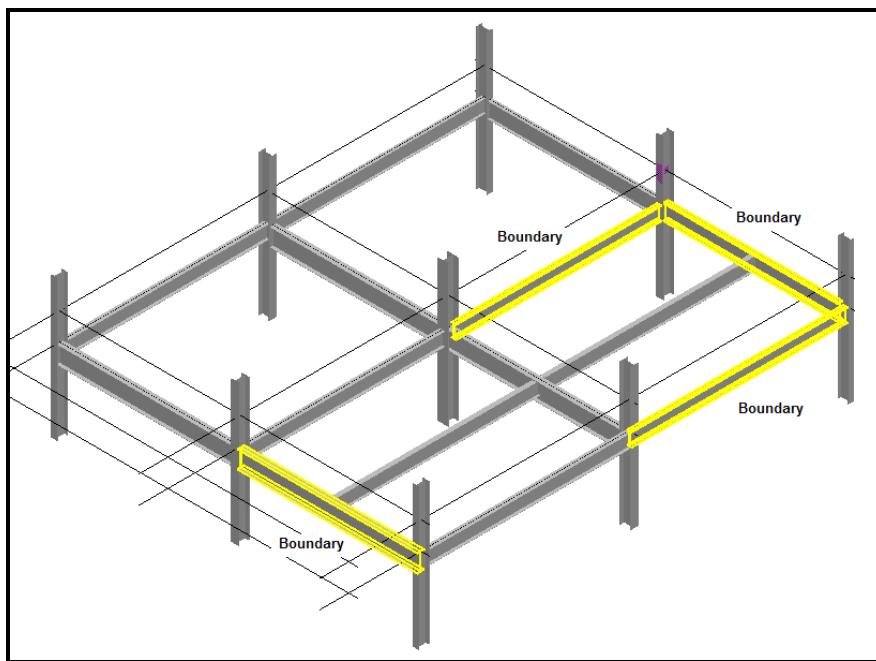
3. Select Place Slab command to place one slab on the second floor.

4. Set the slab placement parameters as follows:

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

5. Select Elevation Plane at 31'- 0" for the support plane. Click "Accept" button.

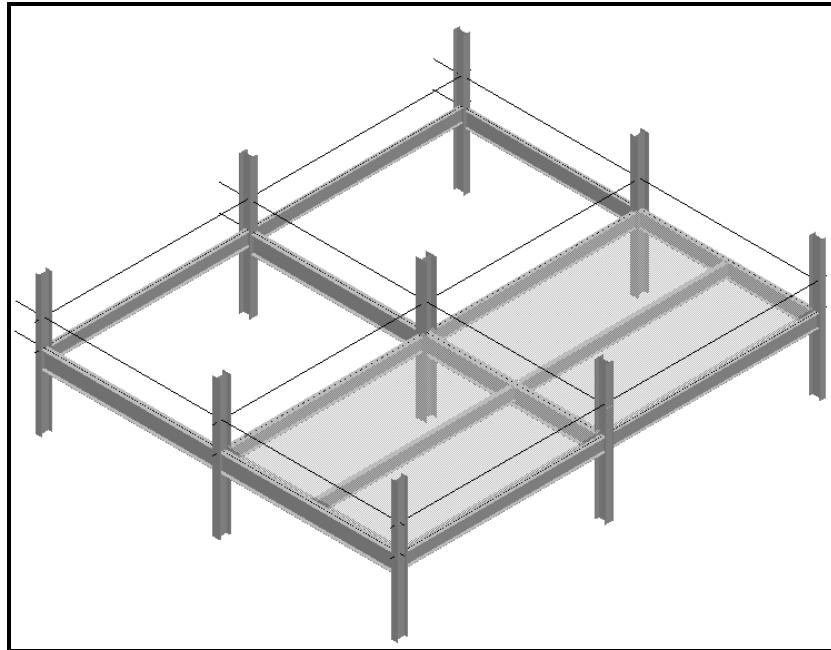
6. Select the boundaries as shown in Figure below.



7. Skip Define the Boundaries offsets step.

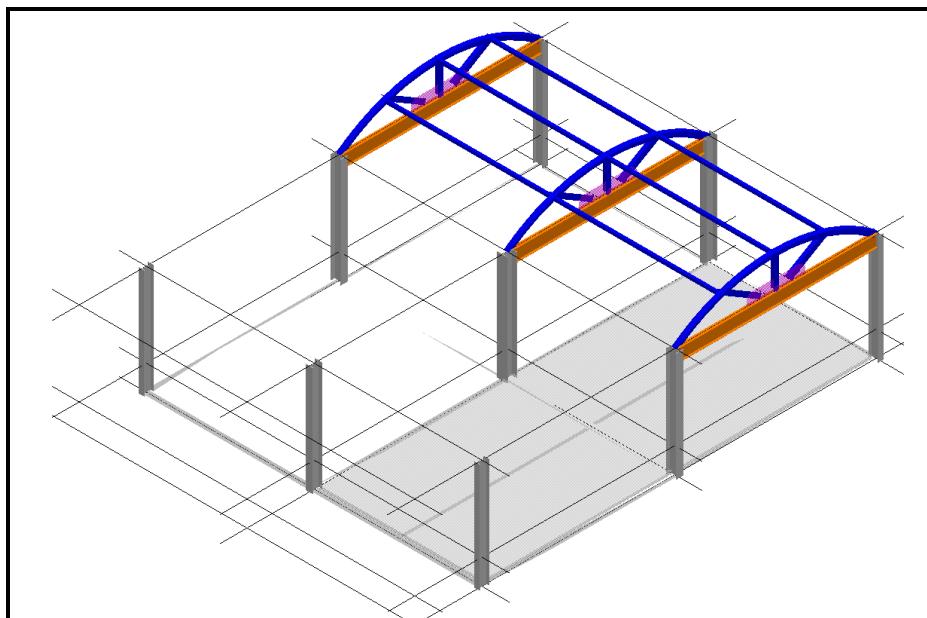
8. Hit "Finish" button to commit the transaction.

Your View should now resemble the following graphic:



### **Part III: Roof Deck**

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



3. Select Place Slab command.
4. Select Plane Method: Offset from a Plane.

5. Select Elevation Plane at 44' - 0" for the support plane using the Workspace Explorer.
6. Key in -2'-0" for offset.
7. Select "Accept" button.
8. Set the slab parameters as follows:

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

9. Select the boundaries as shown in Figure 2

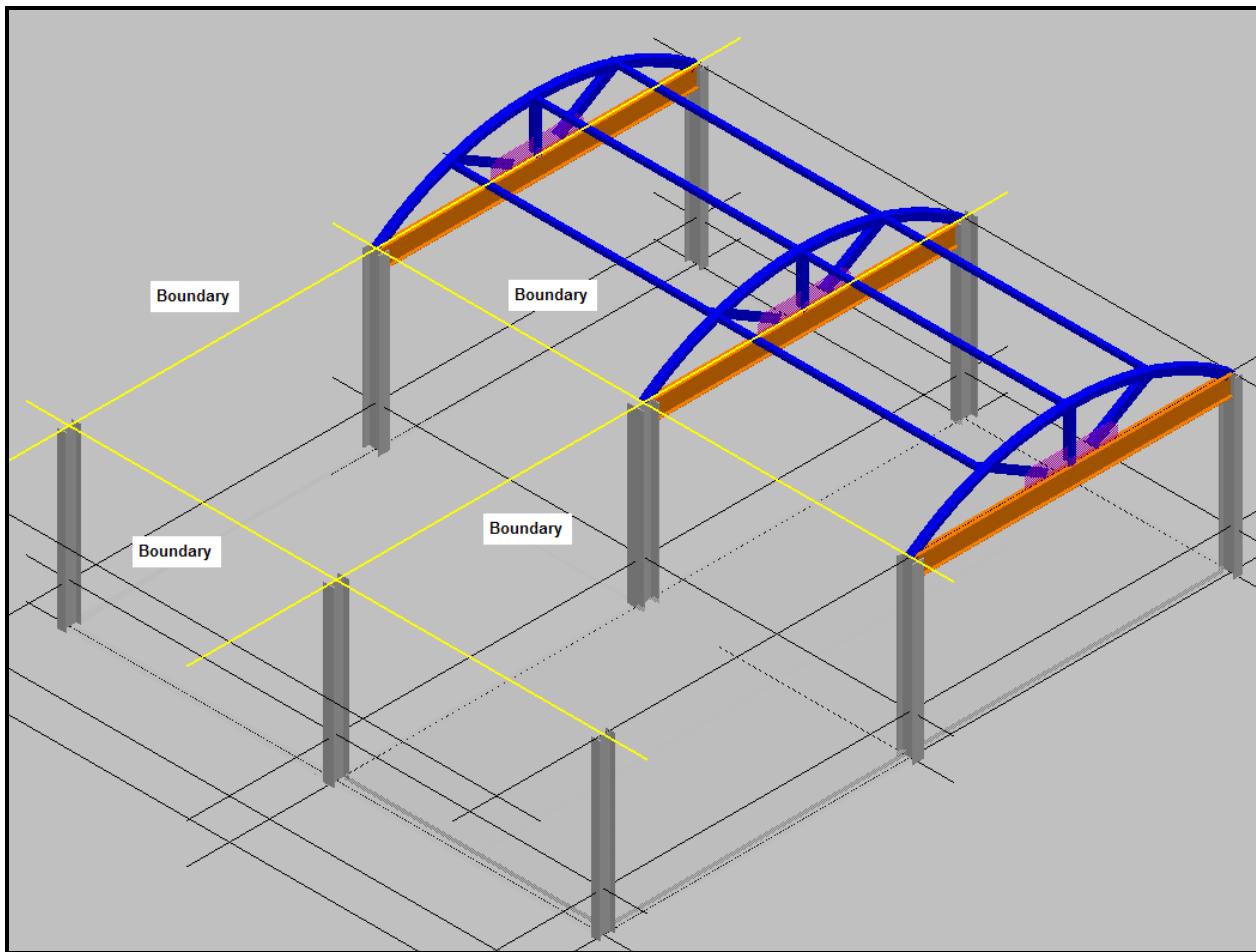
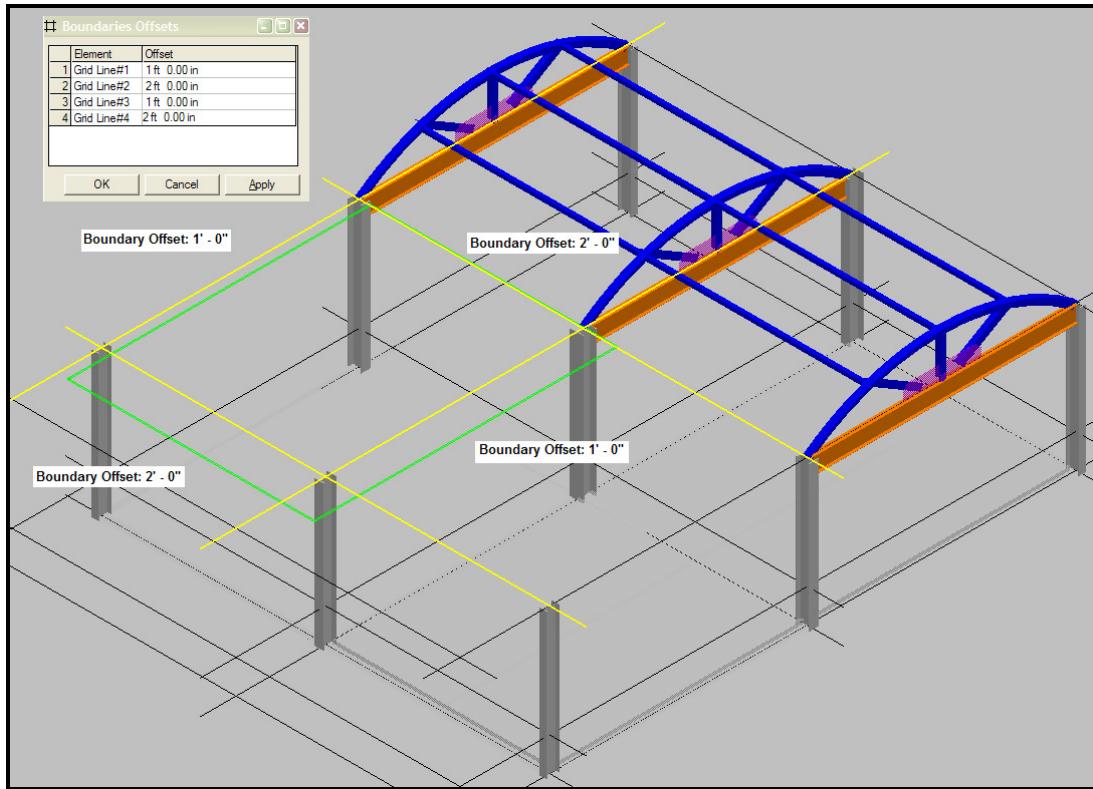


Figure 2 – ISO View of the Roof Deck

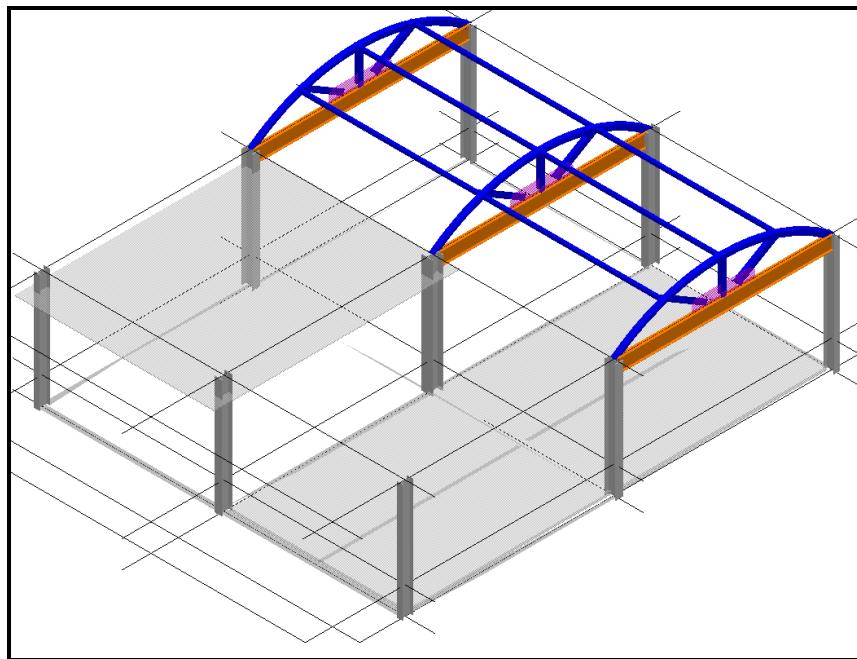
10. Select "Accept" button.

11. Define the Boundaries offsets step as shown below:



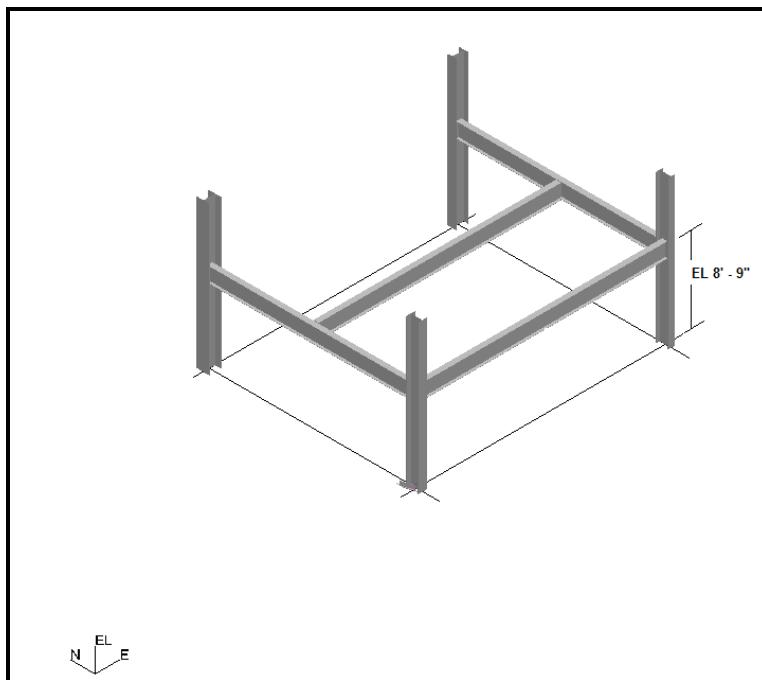
12. Hit “Finish” button to commit the transaction.

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 below:



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

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

7. Select the boundaries as shown in figure 3.

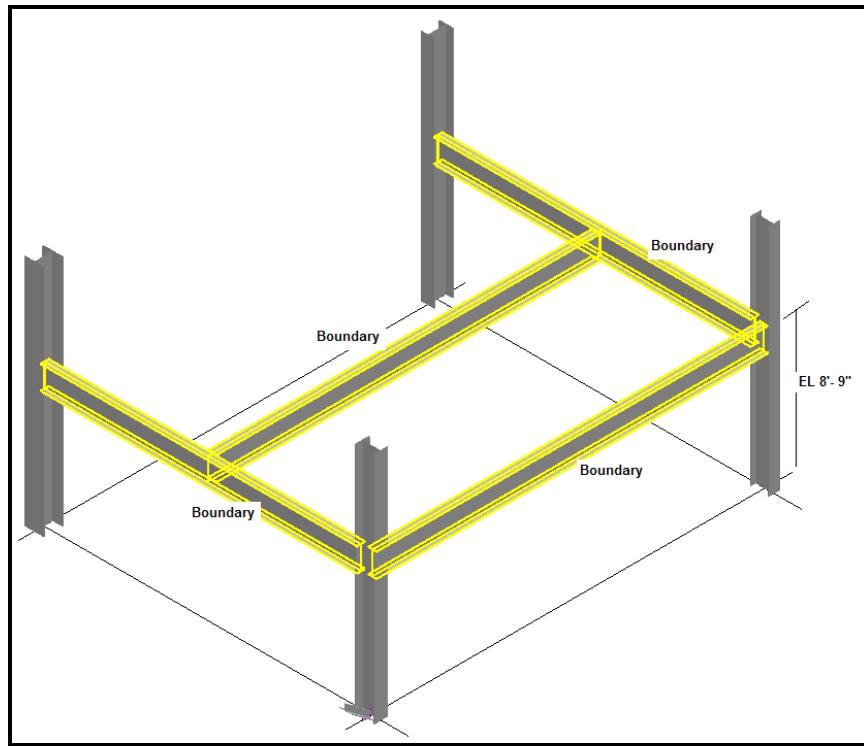
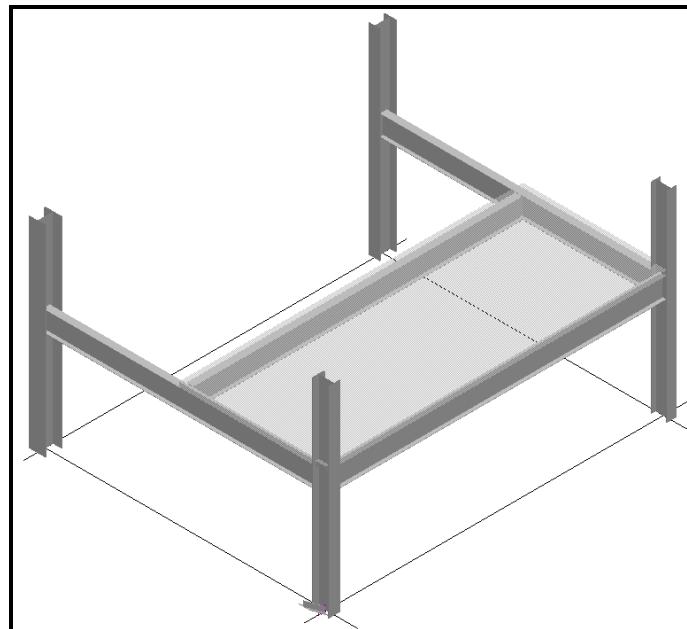


Figure 3 – ISO View of the Far-East Bay

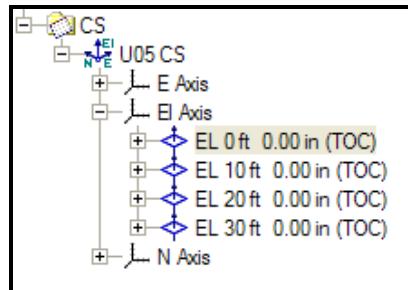
8. Select “Accept” button.
9. Hit “Finish” button to commit the transaction.

Your View should now resemble the following graphic:



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

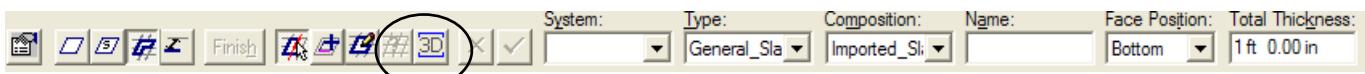
1. Select View-> Clear Clipping to remove the clipping volume.
2. Re-define your workspace to include the A2 -> U05 and CS -> U05 CS systems.
3. Select Place Slab Command.
4. Select Elevation 0'- 0" for the support plane using the Workspace Explorer.



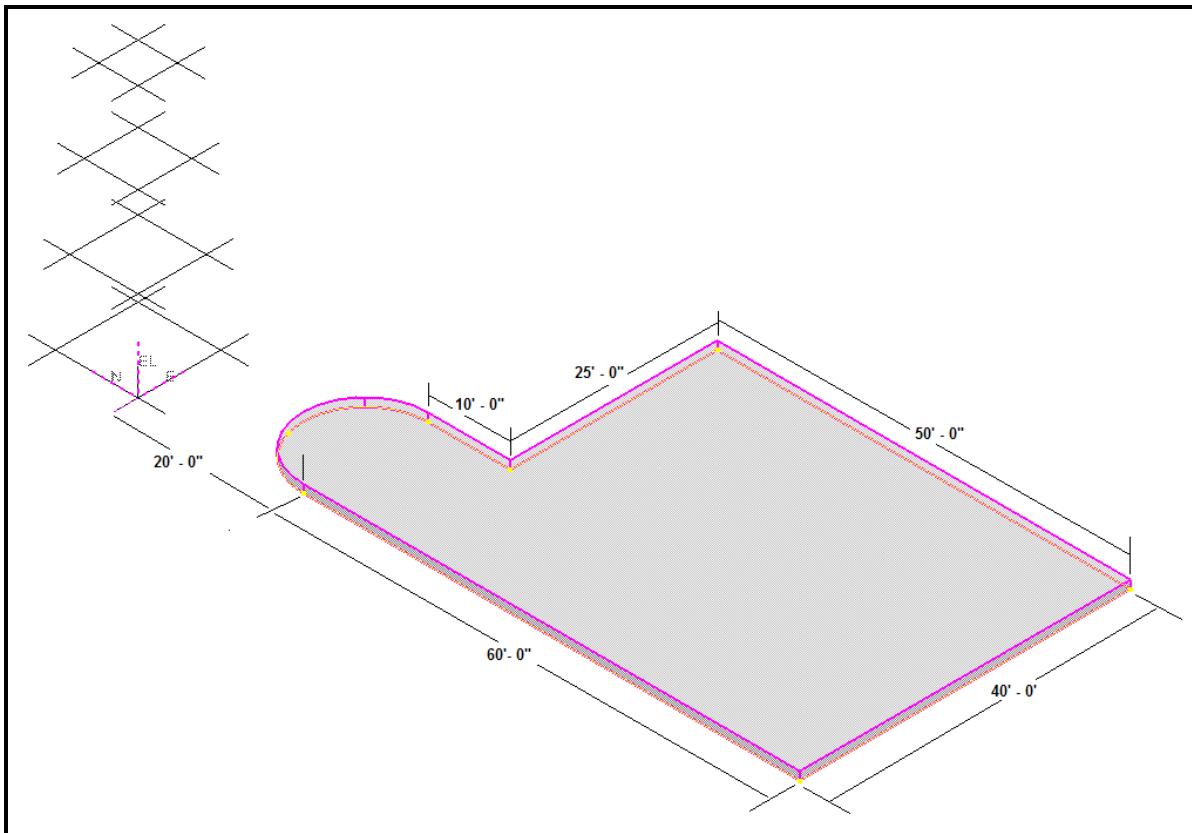
5. Select “Accept” button.
6. Set the slab placement parameters as follows:

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

7. Select the Sketch 3D step.



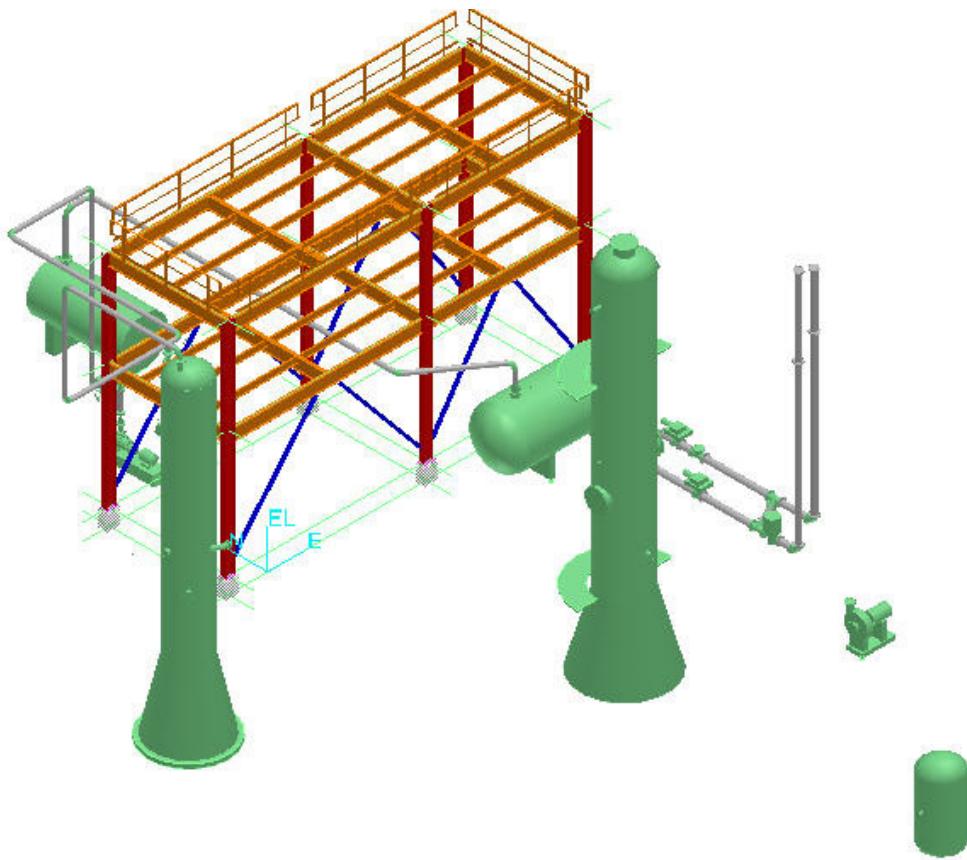
8. Sketch the boundaries of the slab as shown below:



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

#### **Part VI: Sketch Multiple Slabs**

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



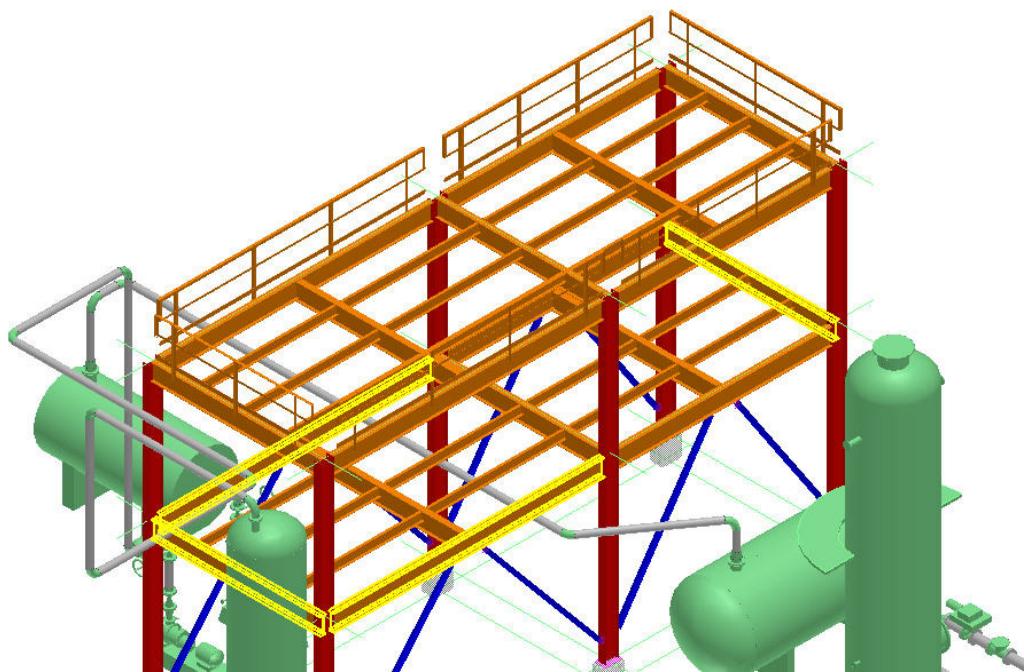
5. Select the *Slabs* command to place multiple slabs.

6. Set the slab placement parameters as follows:

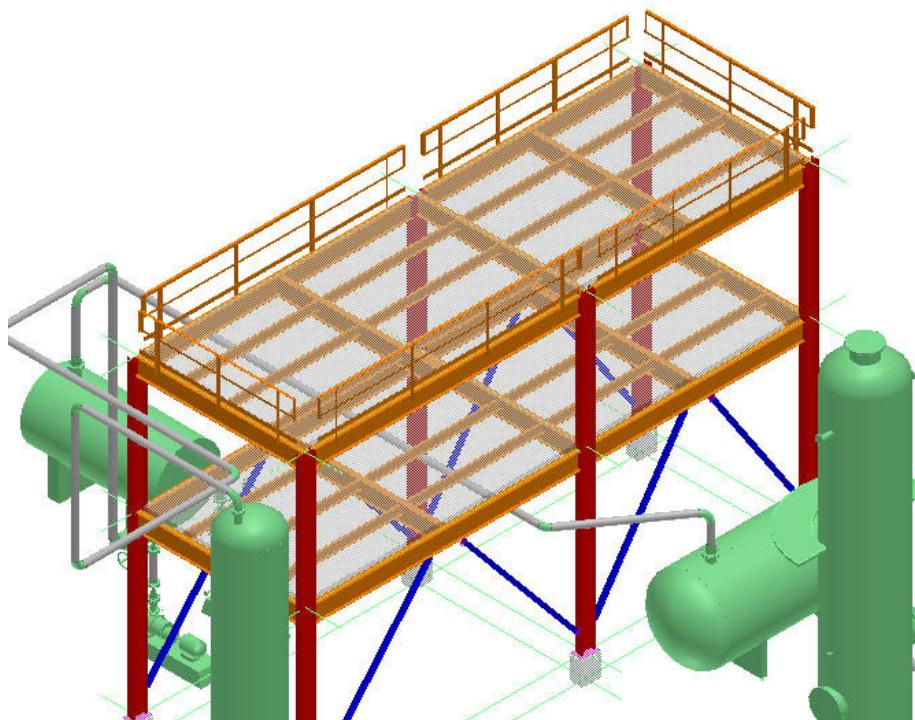
Plane Method:	Coincident
System:	A2-> U02 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4" _Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Boundary Offset Reference:	Inner port-face of a bounding member

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

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



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



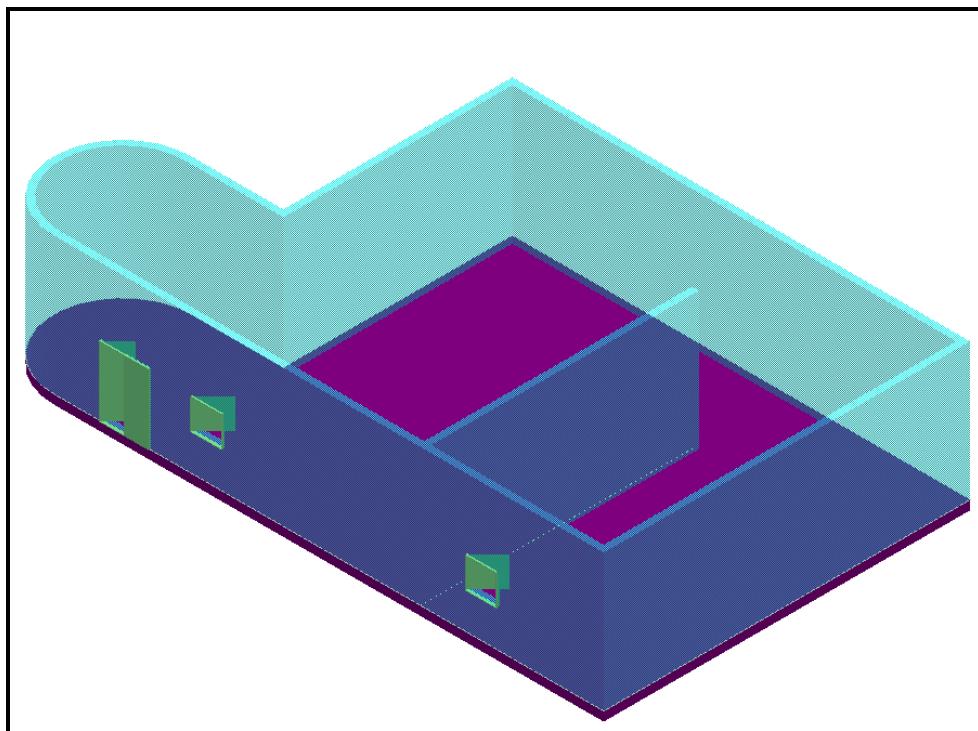
## **LAB-8: Walls**

### **Objectives**

After completing this lab, you will be able to:

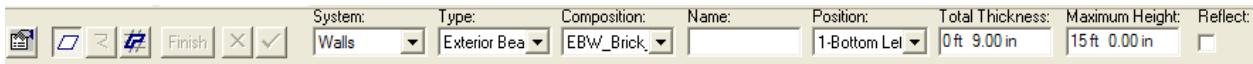
- Understand the wall entities and relationships
- Place Wall using different options
- Place Architectural objects like windows and doors in the model

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:

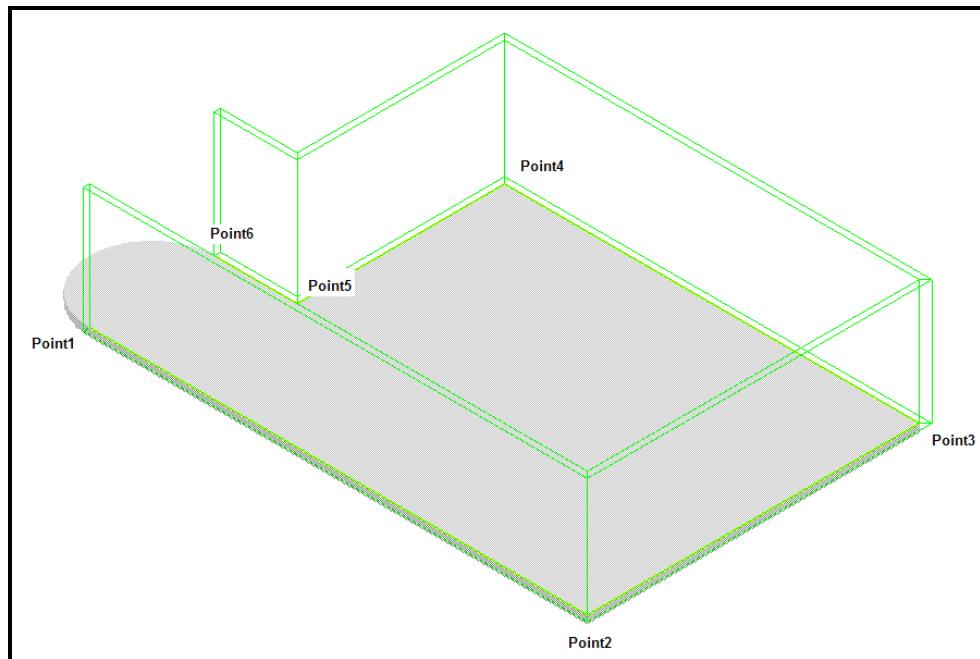
System: A2 -> U05 -> Structural -> Walls  
 Wall Type: Exterior Bearing Wall - Composite  
 Composition: EBW\_Brick\_5'\_Air\_4'\_Bath&Roll\_2'\_Siding\_Vinyl\_0.046"  
 Face Position: 3 - Bottom\_Right  
 Total Thickness: 0'-9"  
 Maximum Height: 15'-0"

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

6. Select the Sketch 3D step.

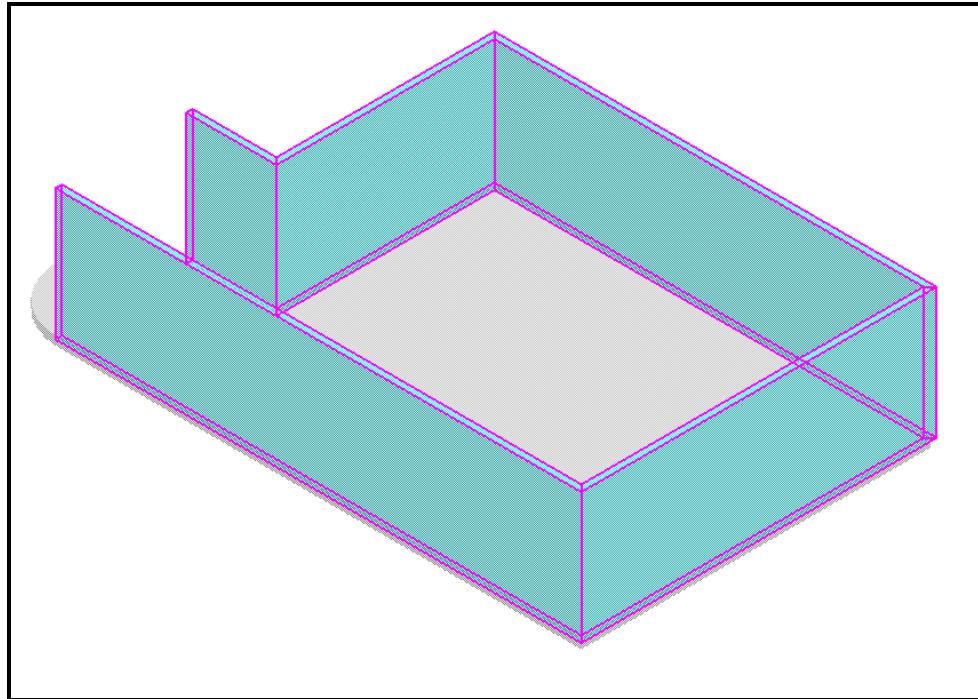


7. Sketch the boundaries of the wall as shown below:



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

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



10. Repeat the above step and place a curve wall.

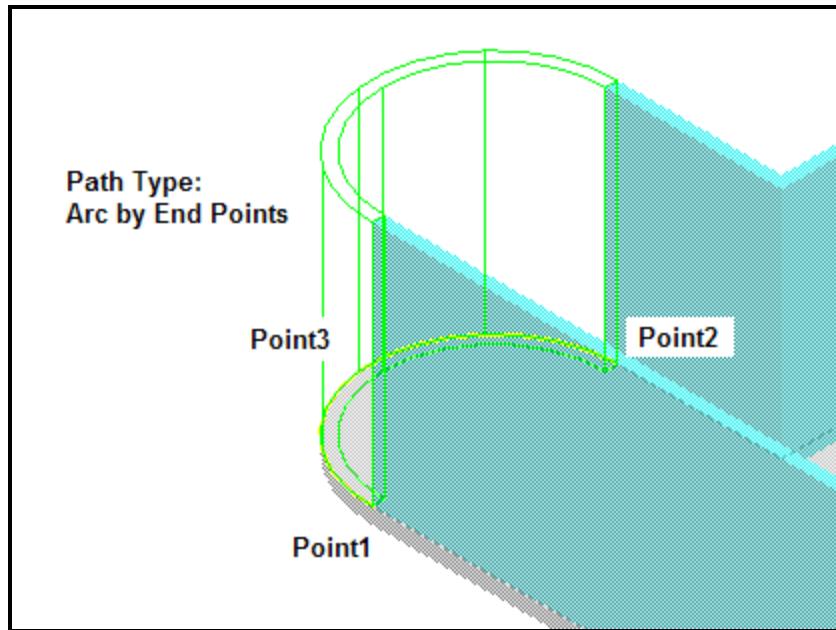
11. Set the wall parameters as follows:

System: A2 -> U05 -> Structural -> Walls  
Wall Type: Exterior Bearing Wall - Composite  
Composition: EBW\_Brick\_5" Air\_4" Batt&Roll\_2" Siding\_Vinyl\_0.046"  
Face Position: 1 Bottom\_Right  
Total Thickness: 0'-9"  
Maximum Height: 15'-0"

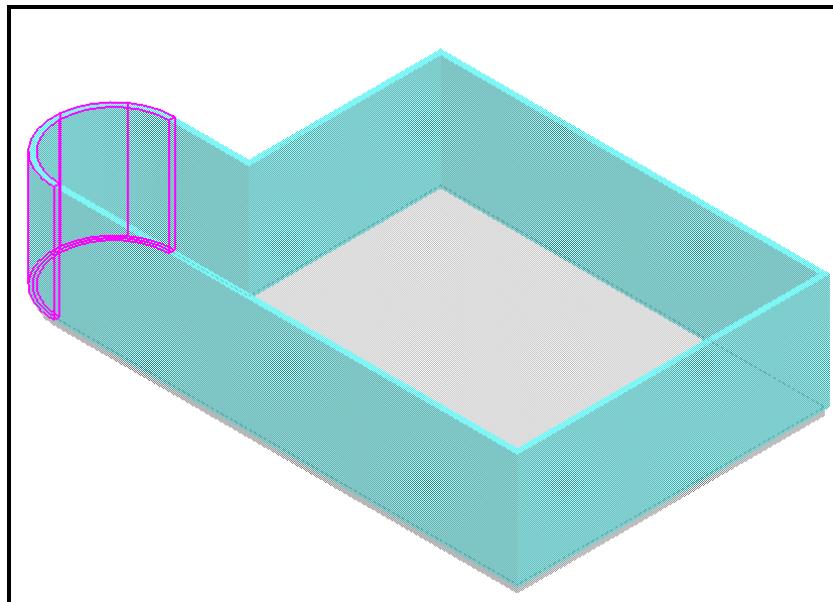
12. Select the top surface of the slab for the sketching plane.

13. During the sketch step, select the path type as Arc by End Points and define three points as shown below:

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



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

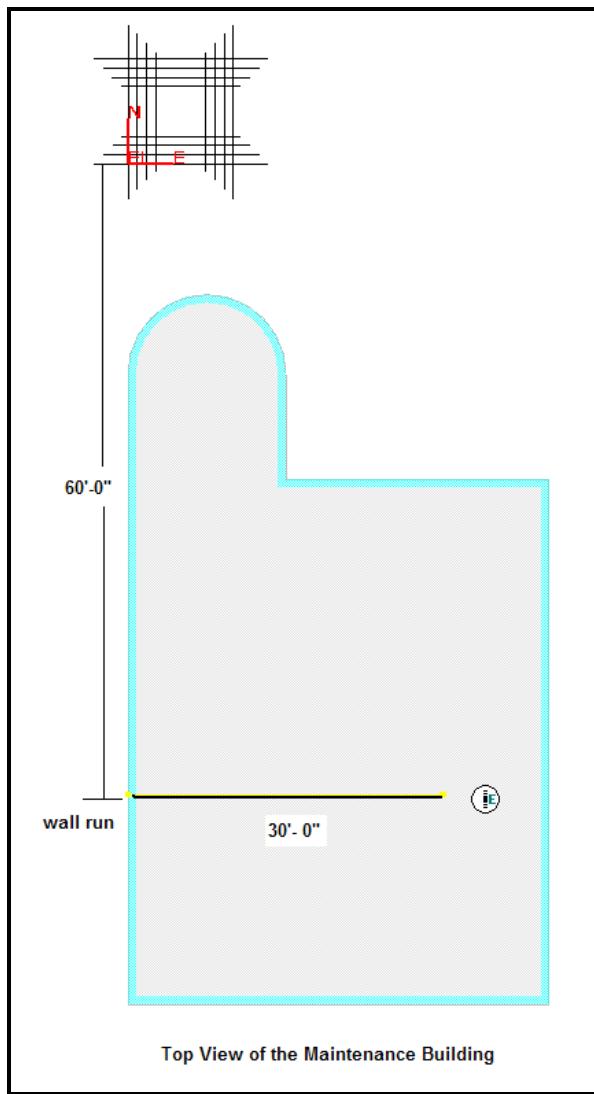


16. Repeat the above step to place a divider wall.
17. Set the wall parameters as follows:

System:	A2 -> U05 -> Structural -> Walls
Wall Type:	Interior Bearing Wall – Non-Composite
Composition:	IBW_Stone_Granite_6"

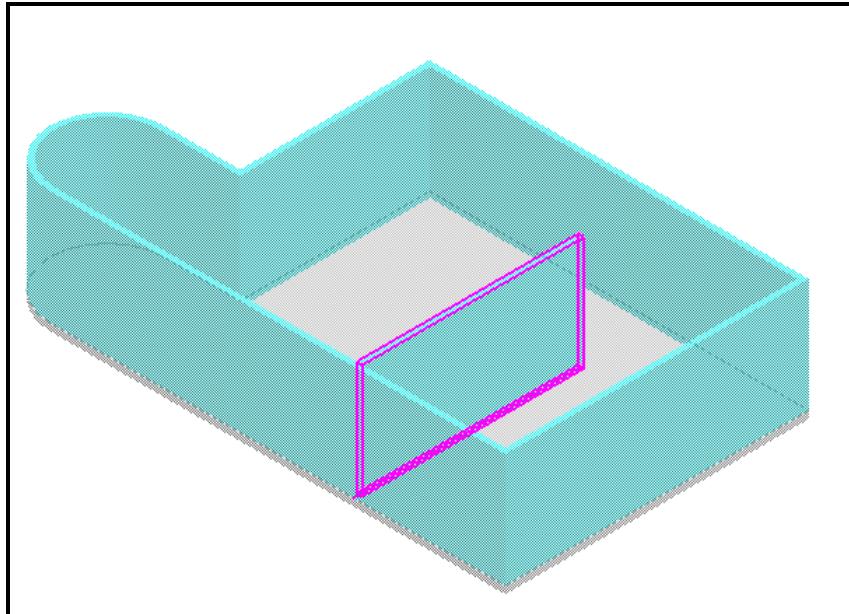
Face Position: 1 Bottom\_Center  
Total Thickness: 0'-9"  
Maximum Height: 15'-0"

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



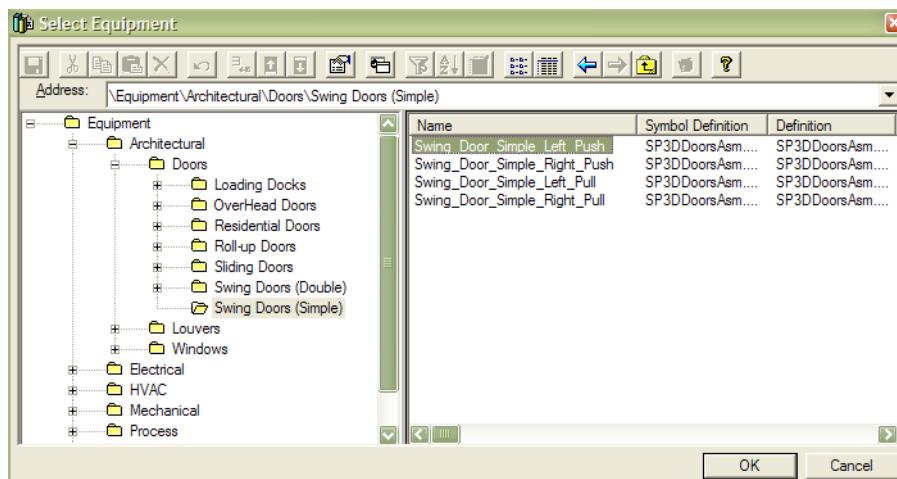
20. Select “Finish Path” button to commit the wall run (path). System returns to the SmartStep ribbon bar of the Place Wall command.
21. Select “Finish” button to persist the wall part in the model.

Your View should now resemble the following graphic:



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

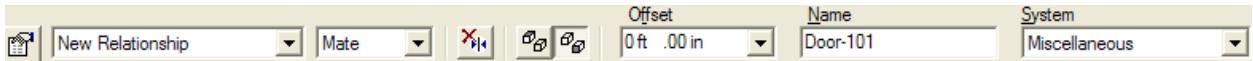
1. Make sure the Active Permission Group is set to *Structural*.
2. Select Place Equipment Command.
3. Expand the equipment hierarchy \Equipment\Architectural\Doors\Swing Doors (Simple) folder until you see the part Swing\_Door\_Simple\_Left\_Push. Select the part and click the “OK” button.



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:

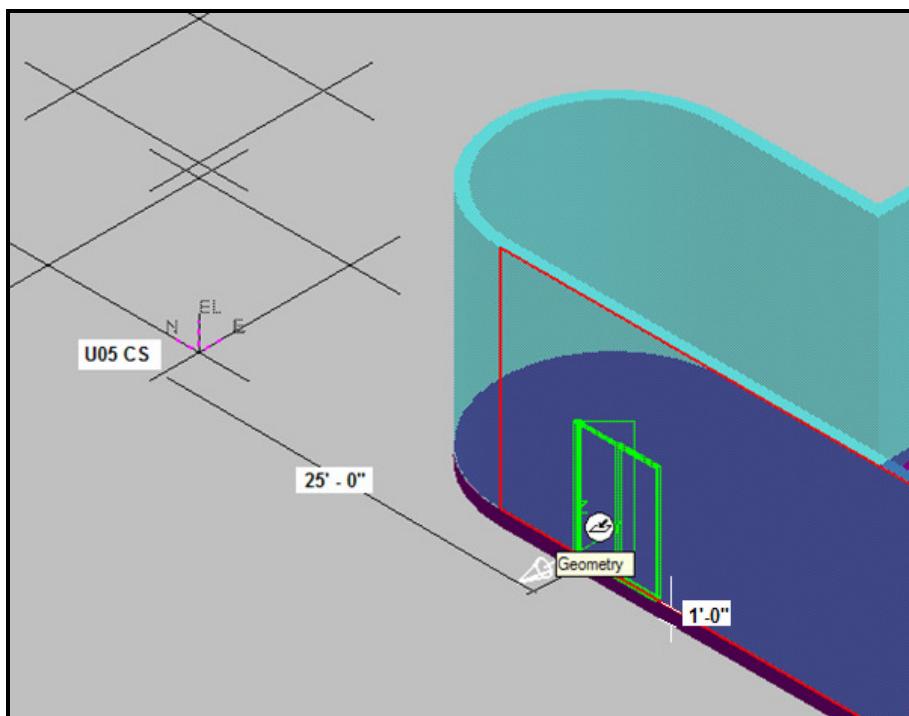
System: A2 -> U05 -> Structural -> Miscellaneous  
 Name: Door-101



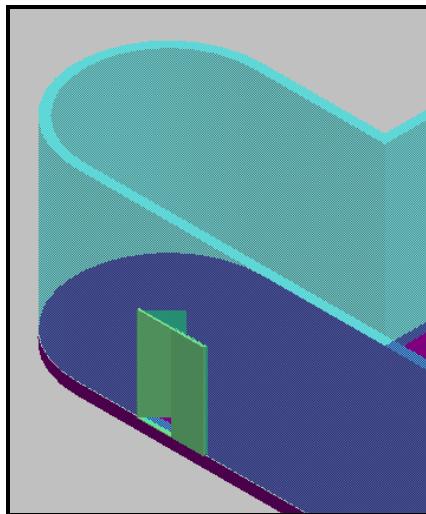
7. Go to the PinPoint ribbon bar and key in -25' for North and 1' for Elevation. Don't keyin anything or lock the Easting field.



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

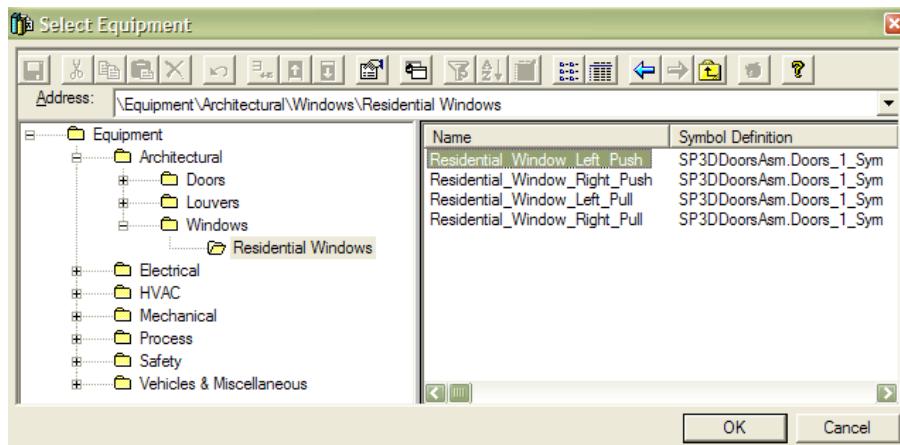


10. Left Mouse Click to place the door on the model.



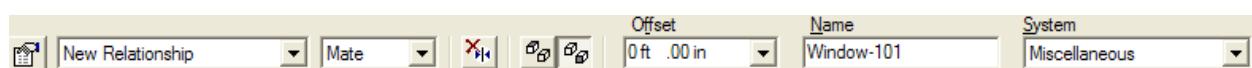
Repeat the above workflow to place the first window.

11. Select Place Equipment Command.
12. Expand the Equipment Hierarchy \Equipment\Architectural\Windows\Residential Windows folder until you see the part Residential\_Windows\_Left\_Push. Select the part and click the “OK” button.



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

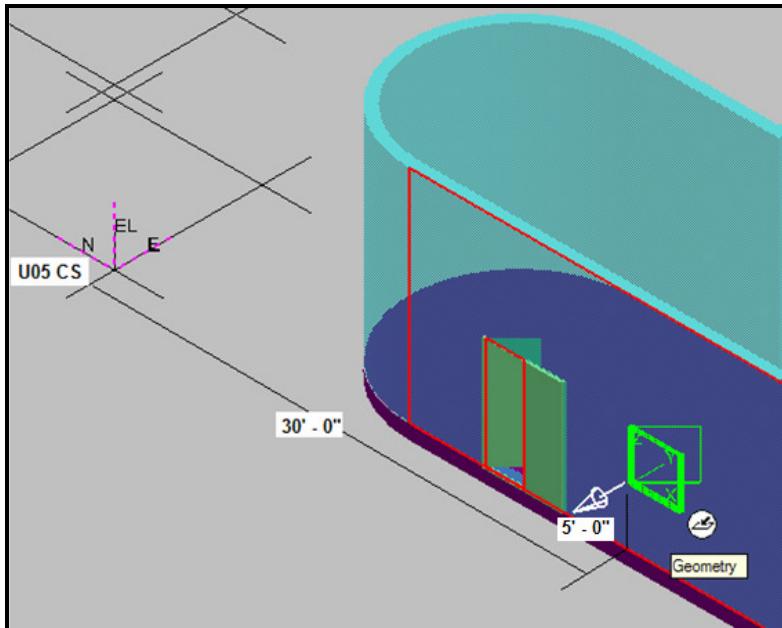
System: A2 -> U05 -> Structural -> Miscellaneous  
 Name: Window-101



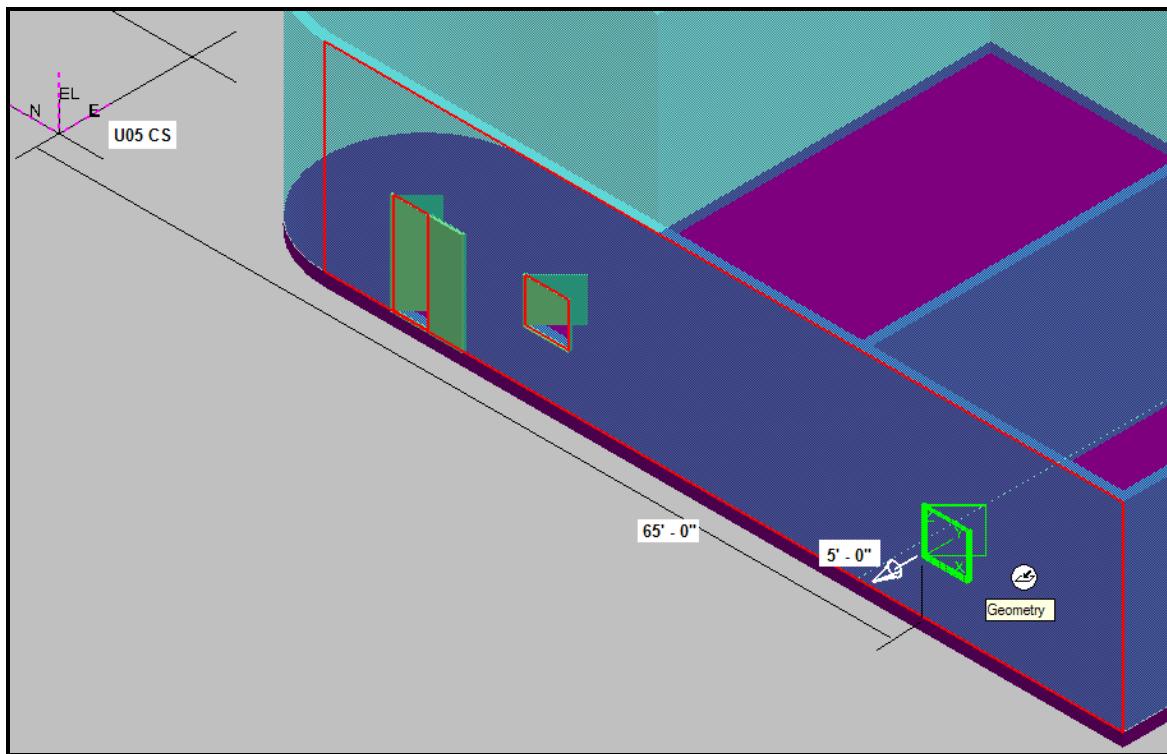
16. Go to the PinPoint ribbon bar and key in -35' for North and 5' for Elevation. Don't keyin anything or lock the Easting field.



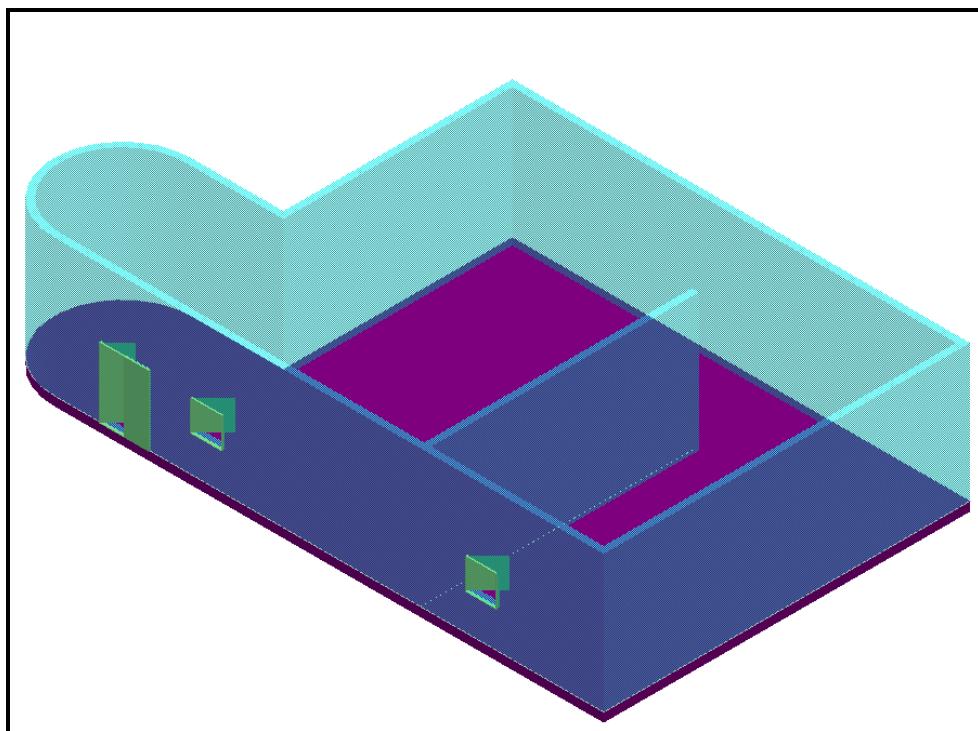
17. Use the Left Arrow key to rotate the equipment 90 deg about the z-axis.  
18. Move the cursor over the wall object as shown below:



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.

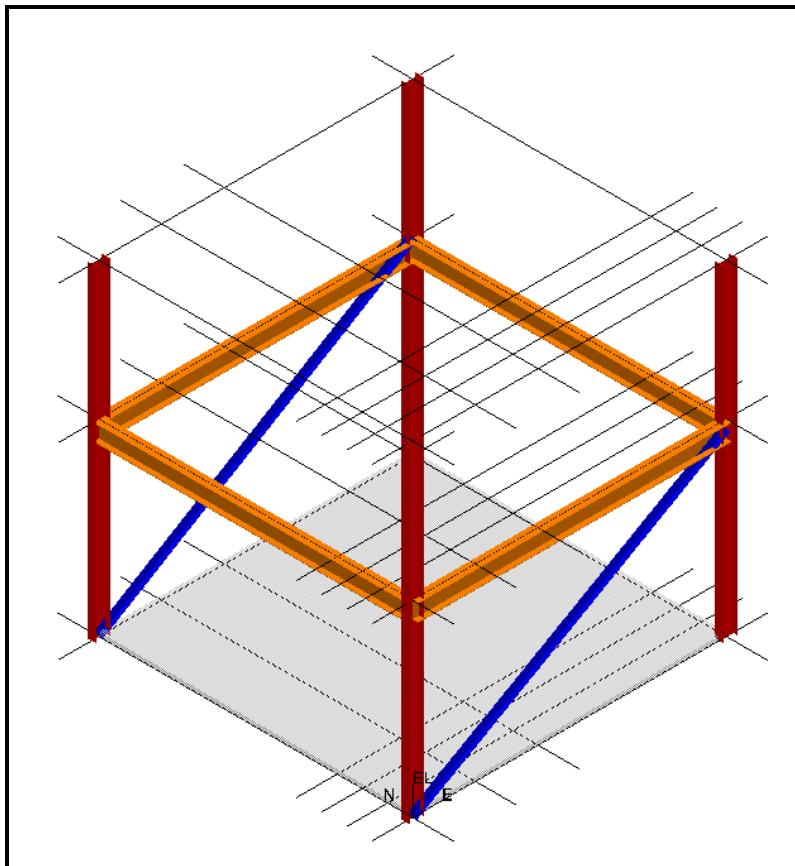


Your View should now resemble the following graphic:



**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:

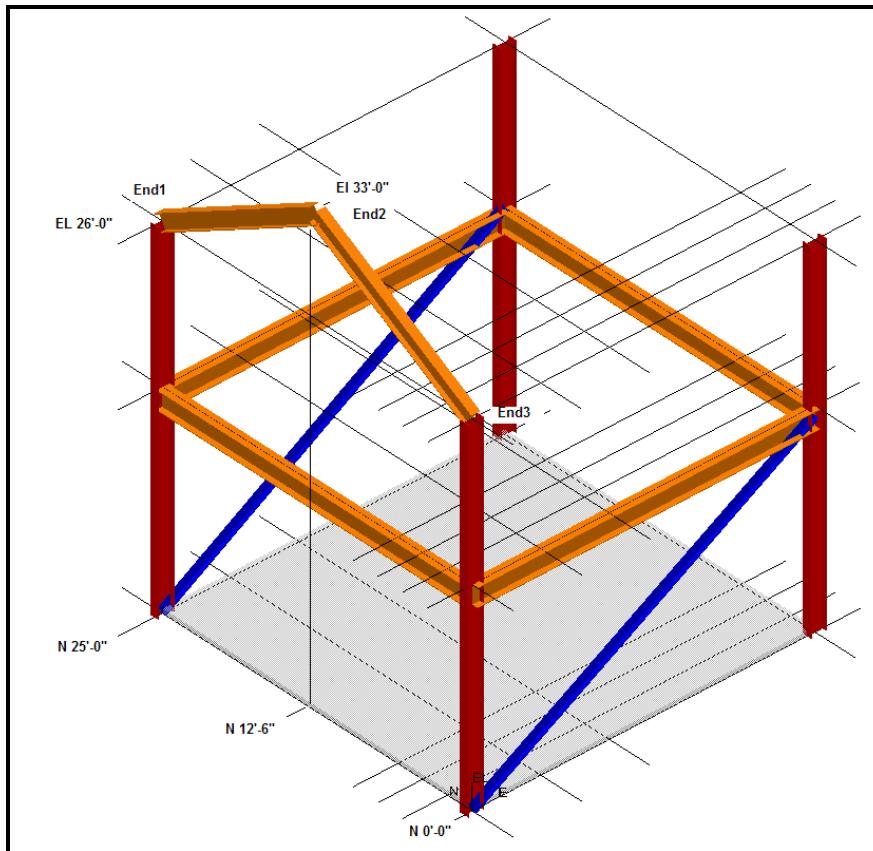


2. Select Place Linear Member System command. System displays the smart step ribbon bar.
3. Use the ribbon bar to set the active member parameters as follows:

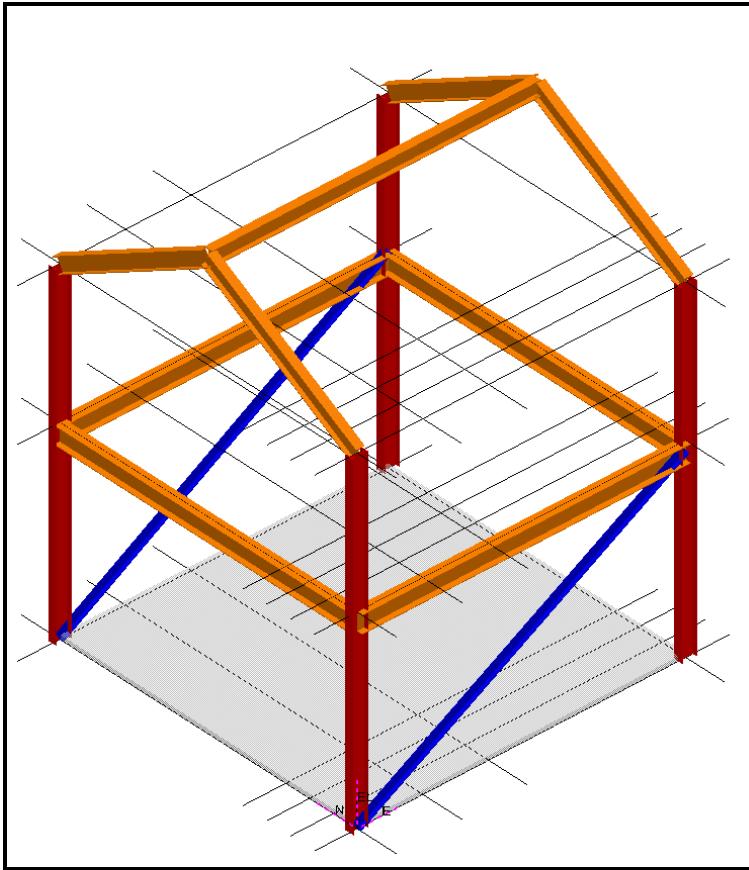
Connection:	By Rule
System:	A4 -> U04 -> Structural -> Beams
Type Category:	Beam
Type:	Beam
Section Name:	W14x53
Cardinal point:	8
Angle:	0 deg
Material:	Steel- Carbon
Grade:	A36

4. Place End 1 at the end of the column located at North Plane 25'-0" and Elevation 26'- 0" intersection.

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



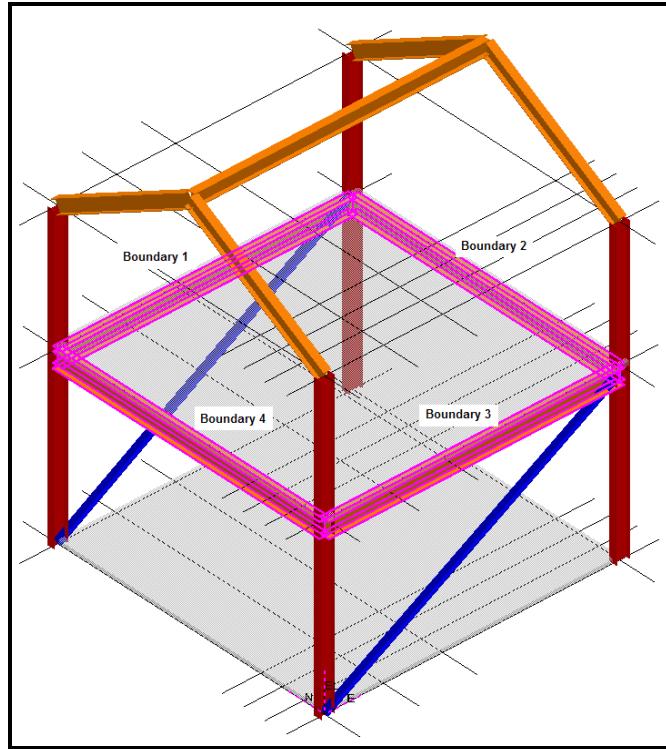
7. Repeat the above workflow to finish building the roof as shown below:



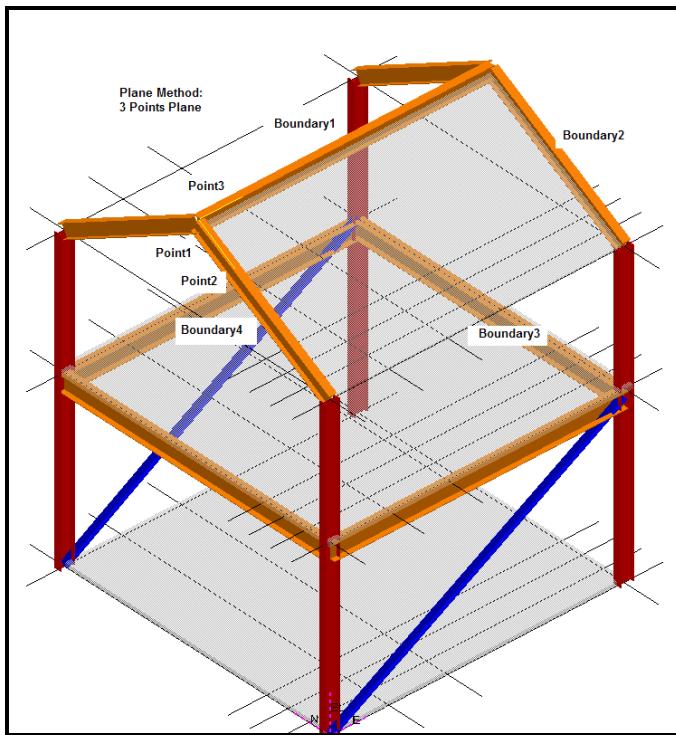
8. Select Place Slab command.
9. Select the Top surface of the beam located at Elevation 15'- 0" for the support plane.
10. Select “Accept” button.
11. Set the slab placement parameters as follows:

Plane Method:	Coincident
System:	A2-> U03 -> Structural -> Slabs
Slab Type:	4" Elevated slab - Composite
Composition:	4" _Slab, 1.5VL22
Face Position:	Bottom
Priority:	Primary
Thickness:	0'-4"
Boundary Offset Reference:	Outer port-face of a bounding member

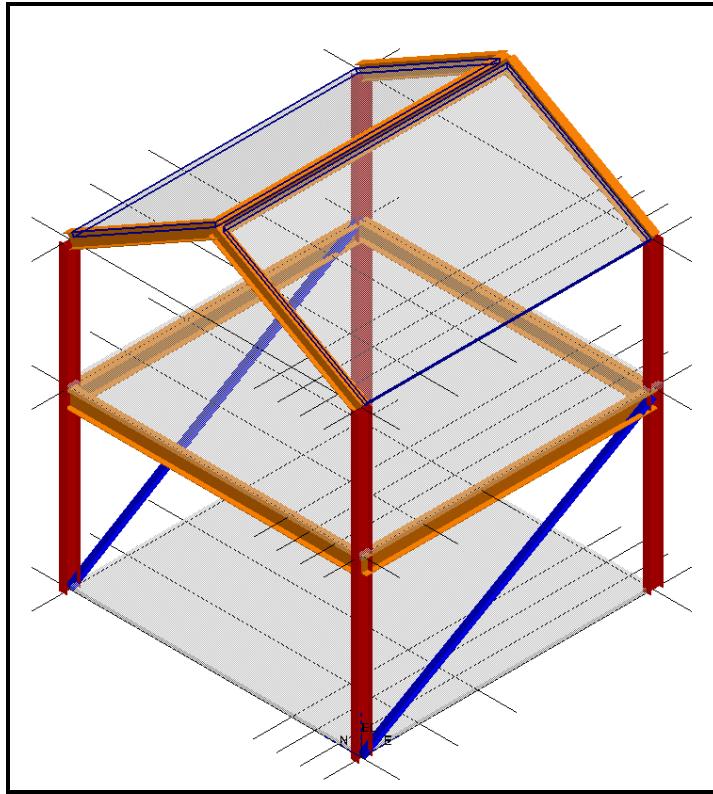
12. Select the boundaries as shown below:



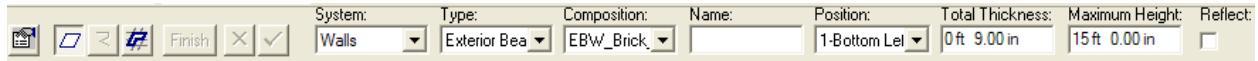
13. Select “Accept” button.
14. Hit “Finish” button to commit the transaction.
15. Repeat the Place Slab Command to place another slab as shown below:



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



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



18. Set the wall parameters as follows:

System: A2 -> U05 -> Structural -> Walls  
 Wall Type: Exterior Bearing Wall - Composite  
 Composition: EBW\_Brick\_5" Air\_4" Batt&Roll\_2" Siding\_Vinyl\_0.046"  
 Face Position: 2 - Bottom\_Center  
 Total Thickness: 0'-3"  
 Maximum Height: 17'-0"

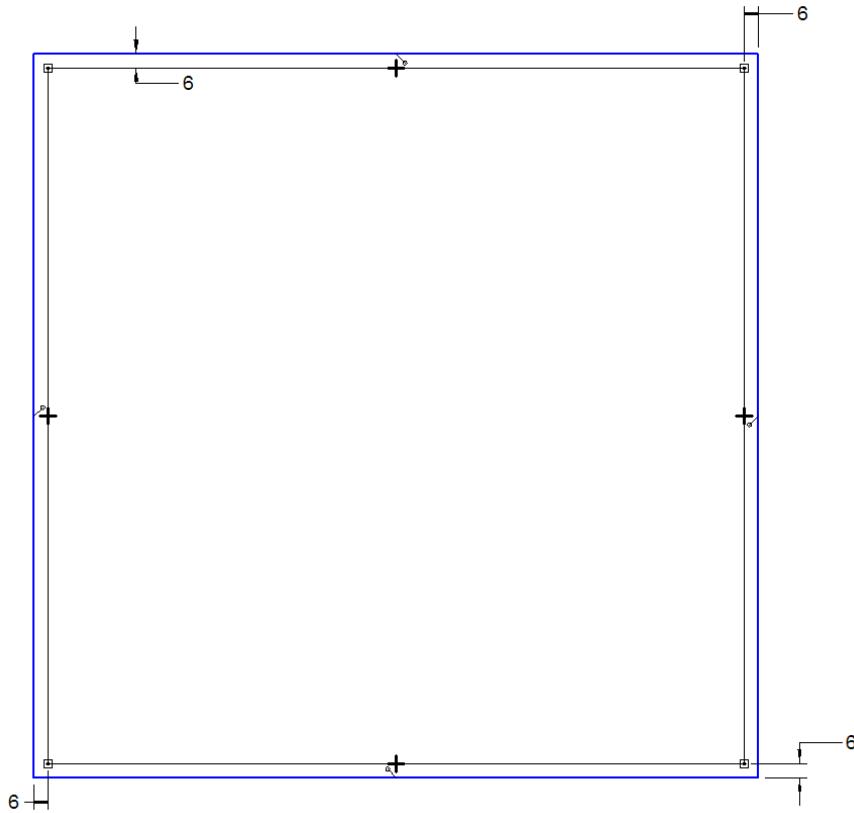
19. Select the top surface of the slab (EL 15'-0" for the sketching plane).

20. Select the Sketch 2D step.



21. Select “Draw” button to open the 2D Editor.

*Hint: Use place line and dimension commands and sketch the boundaries as follows:*



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

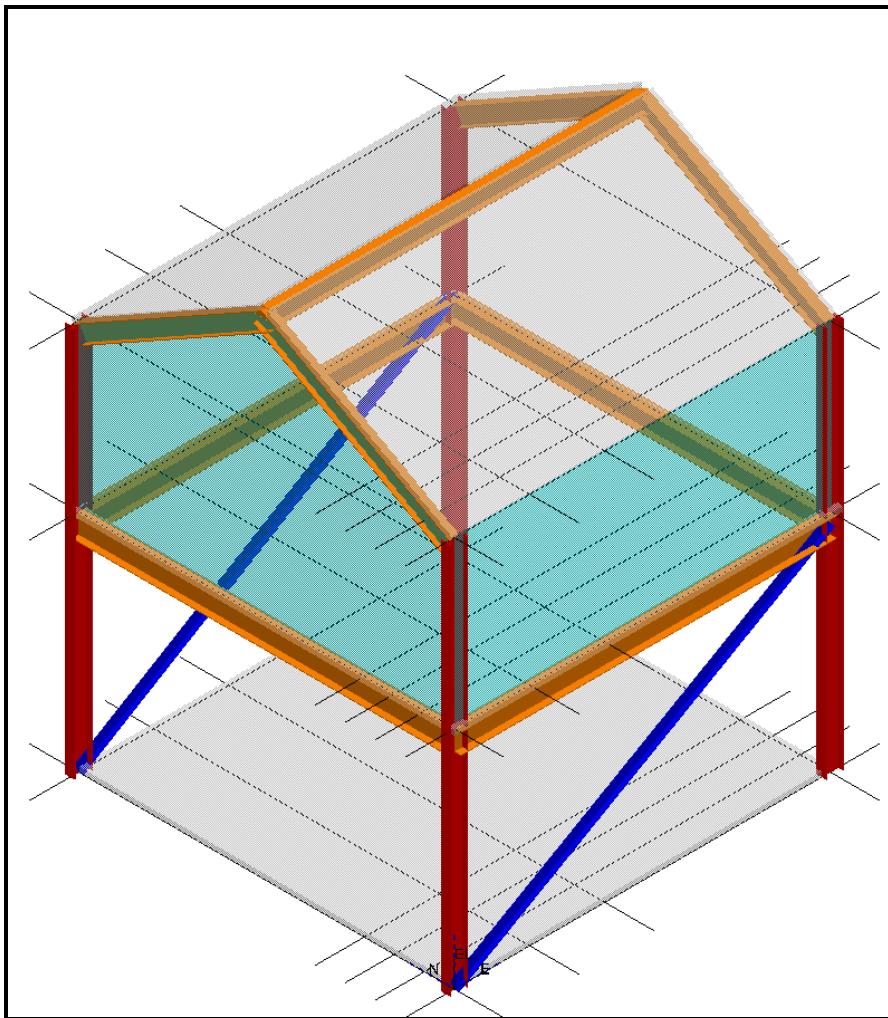


23. Select Finish button.

24. Select the boundary step and pick the two slabs located on the roof.

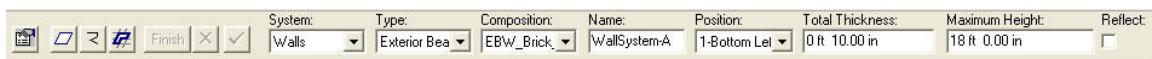
25. Hit “Finish” button.

Your View should now resemble the following graphic:



#### **Part IV: Split Walls 3-D sketch**

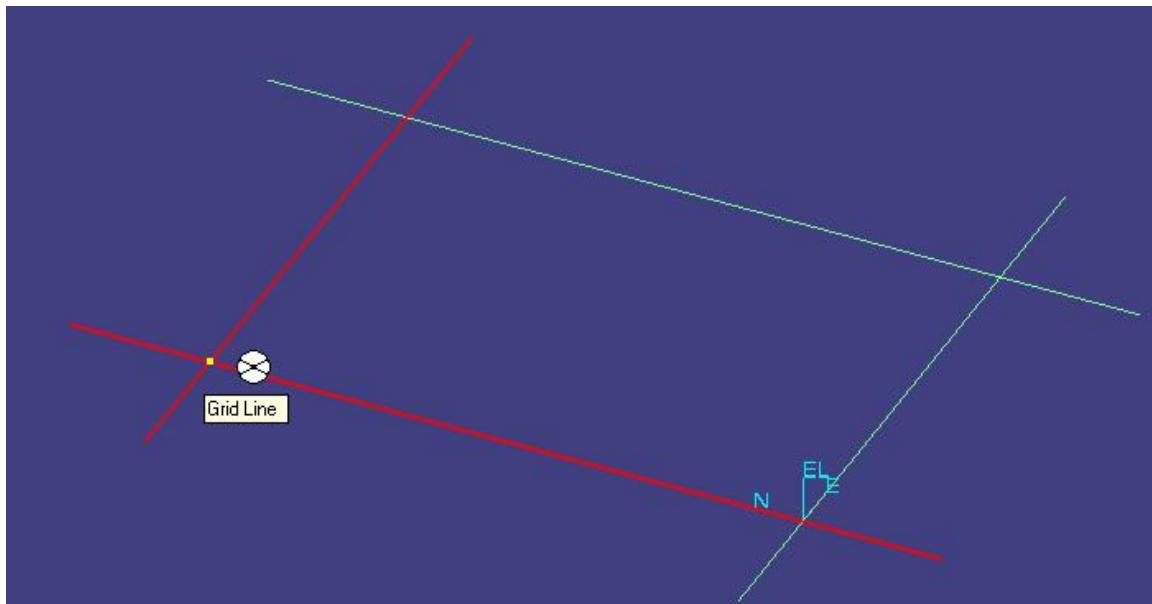
1. Re-define your workspace to include the A2 → U06 and CS → U06 CS system.
2. Select Place Wall Command to open the SmartStep ribbon bar.



3. Set the wall parameters as follows:

System: A2 -> U06 -> Structural -> Walls  
 Wall Type: Exterior Bearing Wall - Composite  
 Composition: EBW\_Brick\_5”\_Air\_4”\_Bath&Roll\_2”\_Siding\_Vinyl\_0.046”  
 Face Position: 1 - Bottom \_Left  
 Total Thickness: 0'-10"  
 Maximum Height: 18'-0"

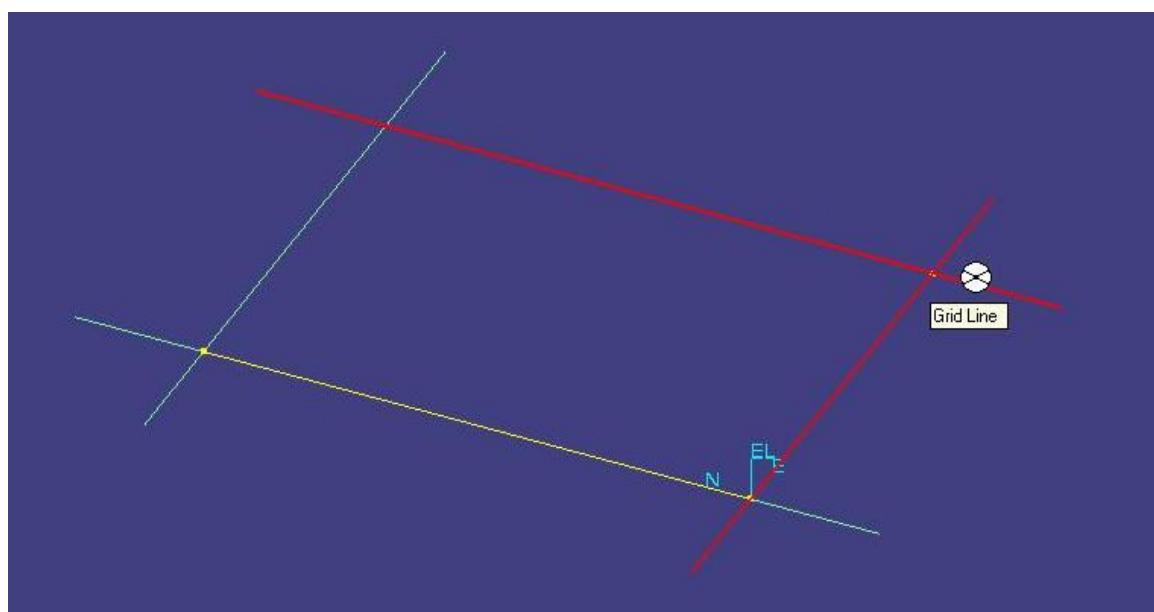
4. Select the Elevation plane, El 0ft as the sketching plane.
5. Select the Sketch 3D step.
6. Select the intersection of the grids as shown below.



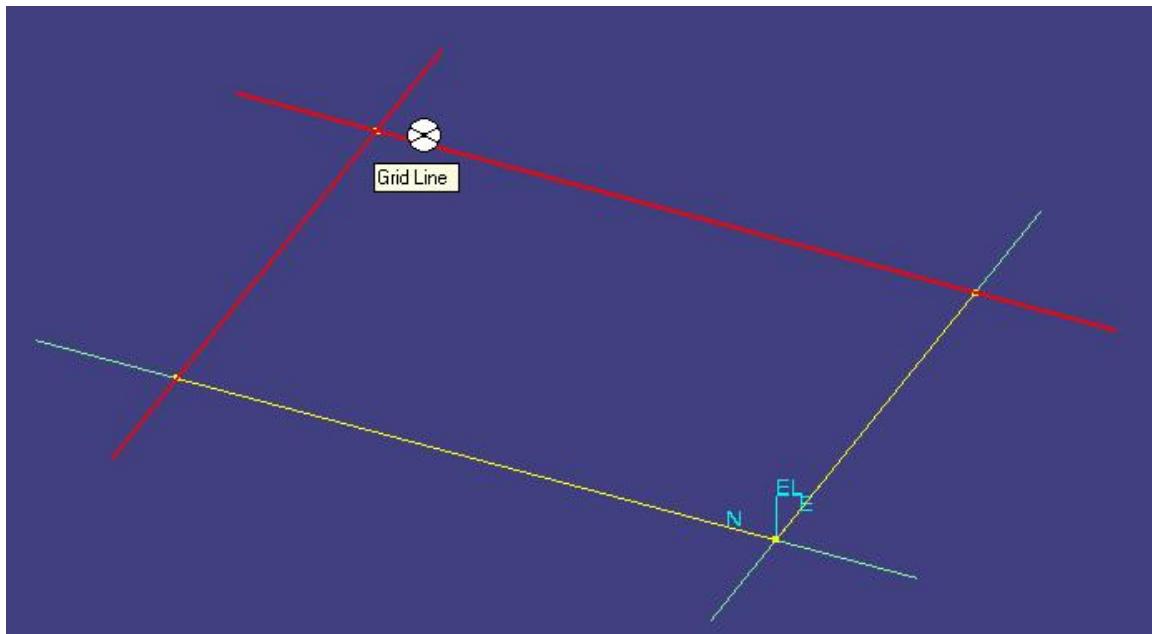
7. Select the next intersection along the North along the grid line and perform a left mouse click.
8. Go to the ribbon bar and change the Turn Type to “Split”.



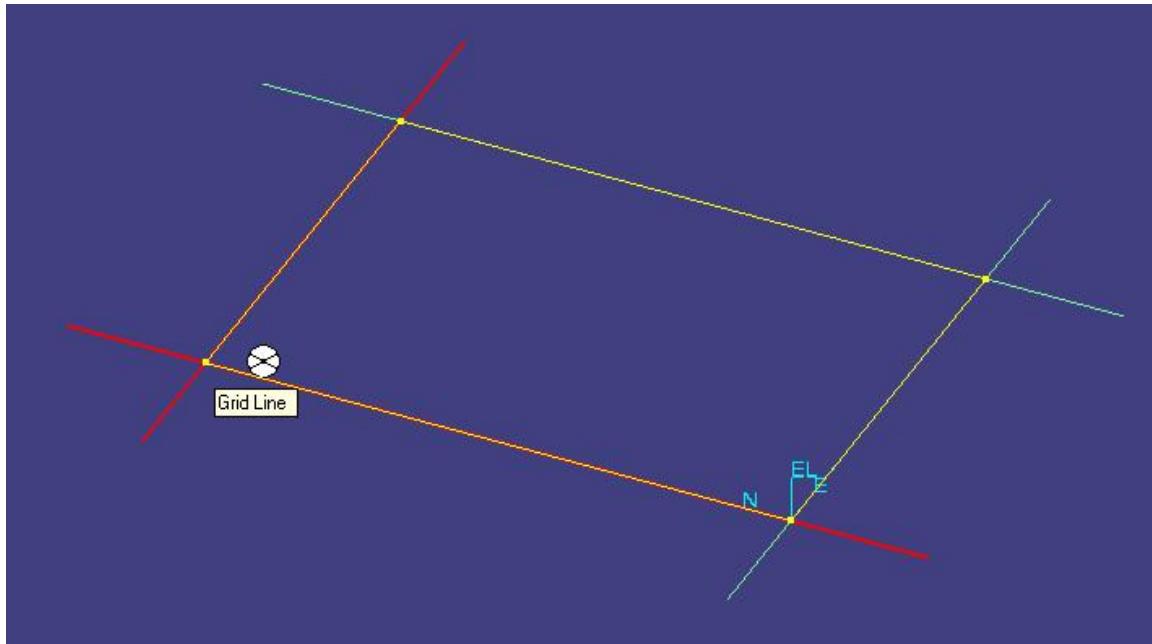
9. Now, select the intersection as shown below.



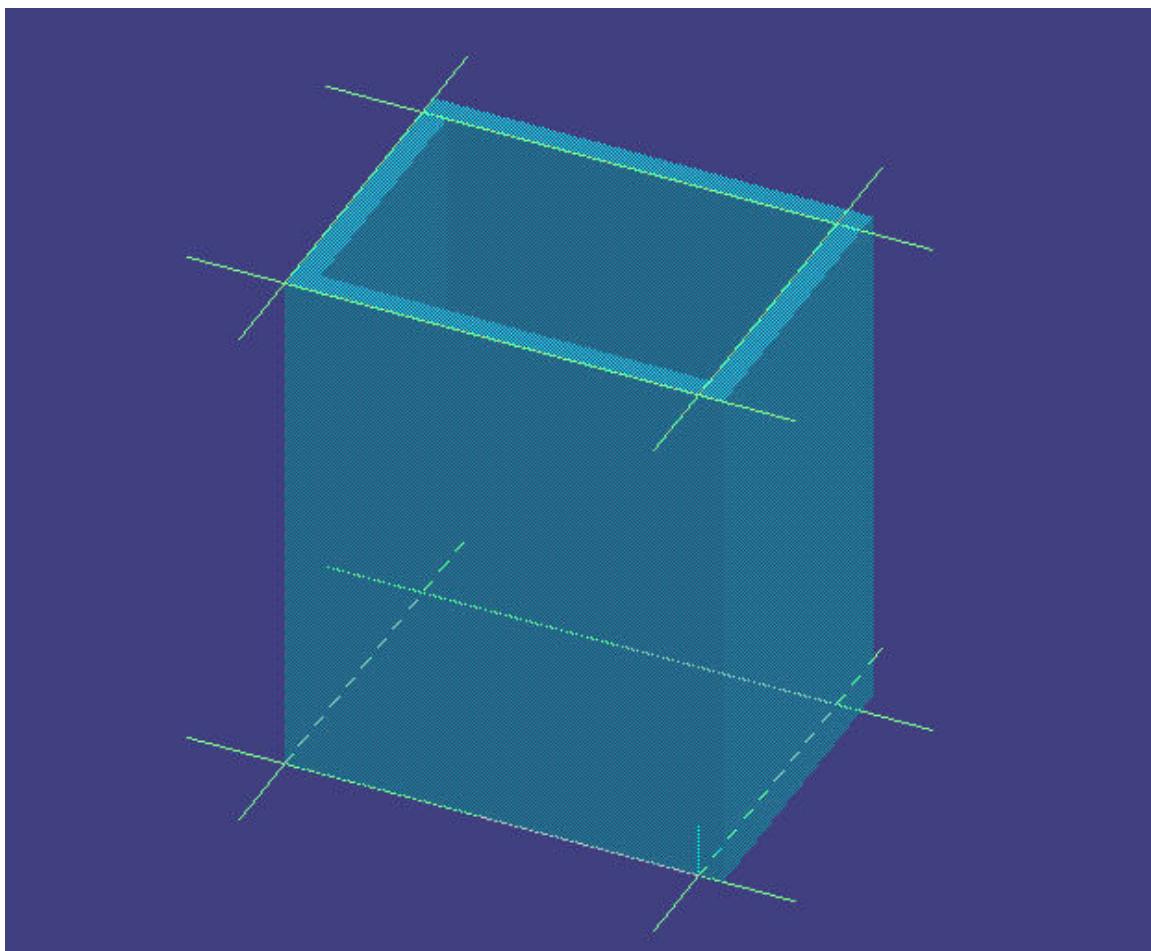
10. Change the Turn Type to “None” and then select the next intersection along the North and as shown below.



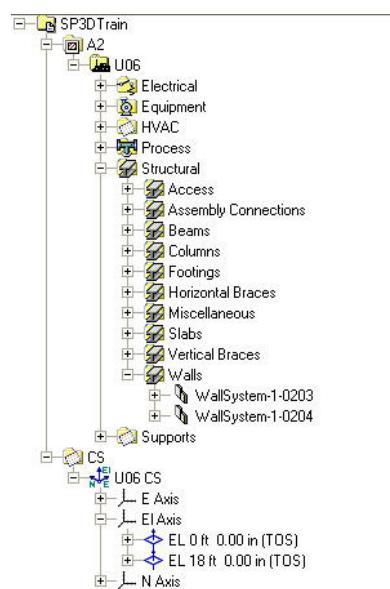
11. Change the Turn Type to Split again and select the intersection as shown below.



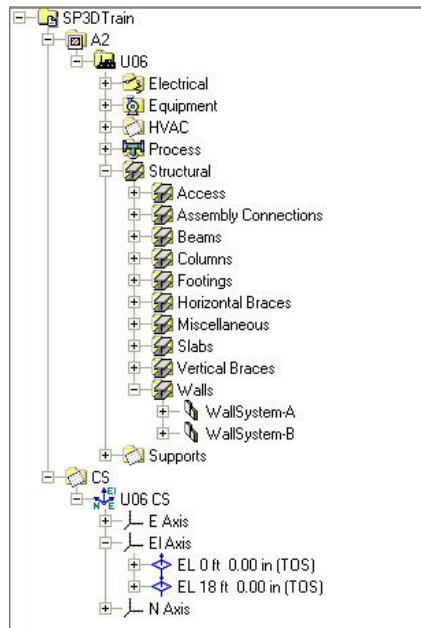
12. Hit Finish twice. Your view should resemble the following graphic.



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

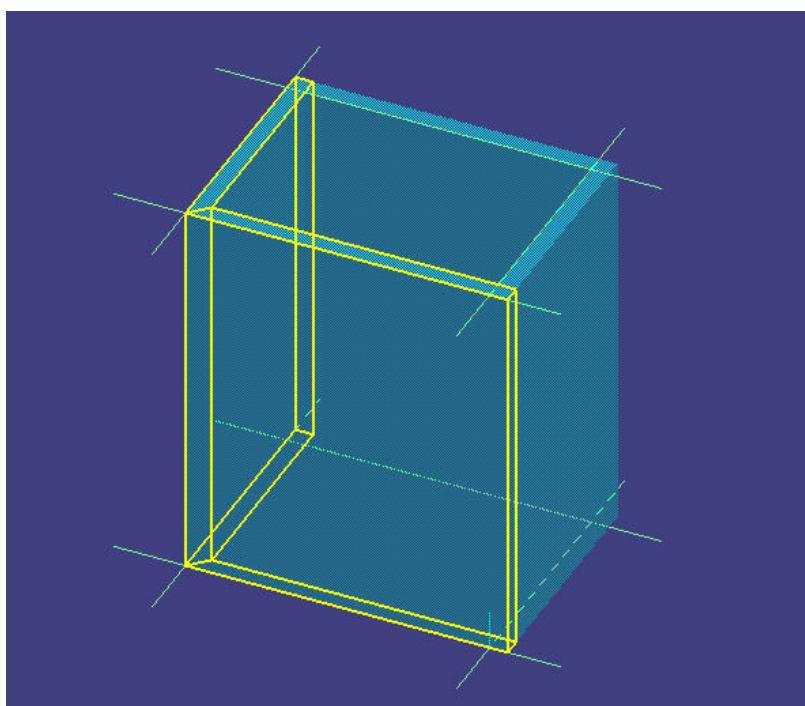


14. Rename the WallSystem-1-0203 as WallSystem-A and rename the WallSystem-1-0204 as WallSystem-B. Your workspace explorer should view as shown below.

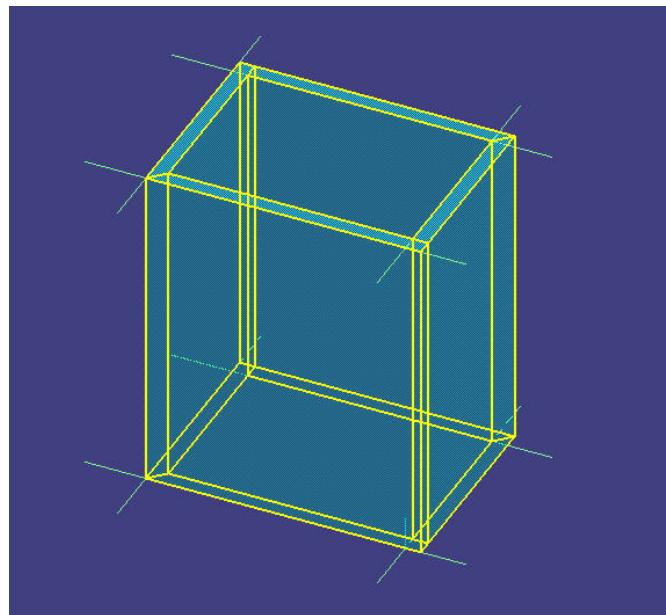


#### **Part V: Toggle Wall Corner**

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.

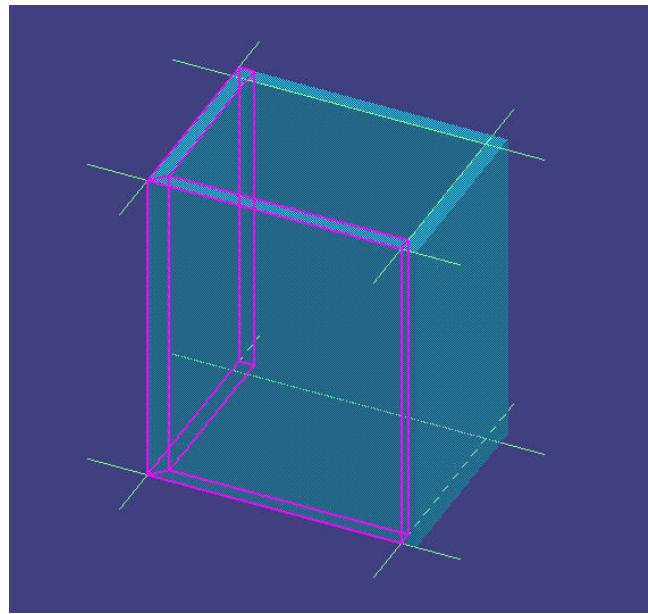


17. Select the WallSystem-B as shown below.



18. Hit Finish.

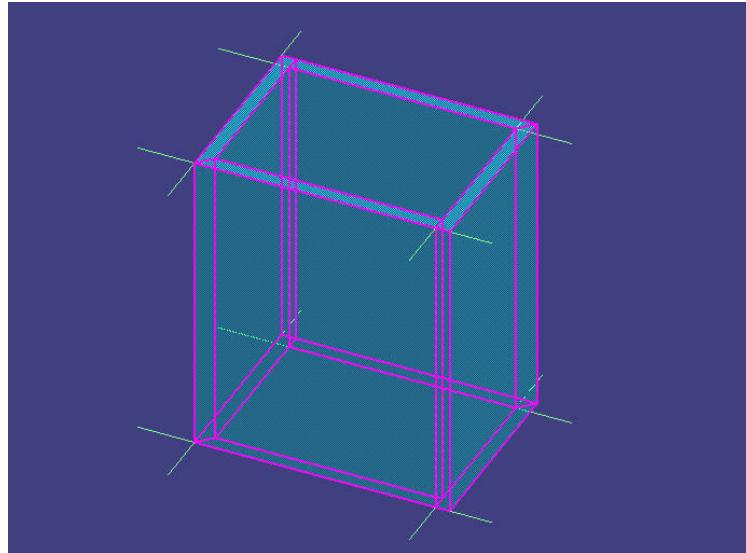
19. Select WallSystem-A as shown below and observe the changes.



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 modification**

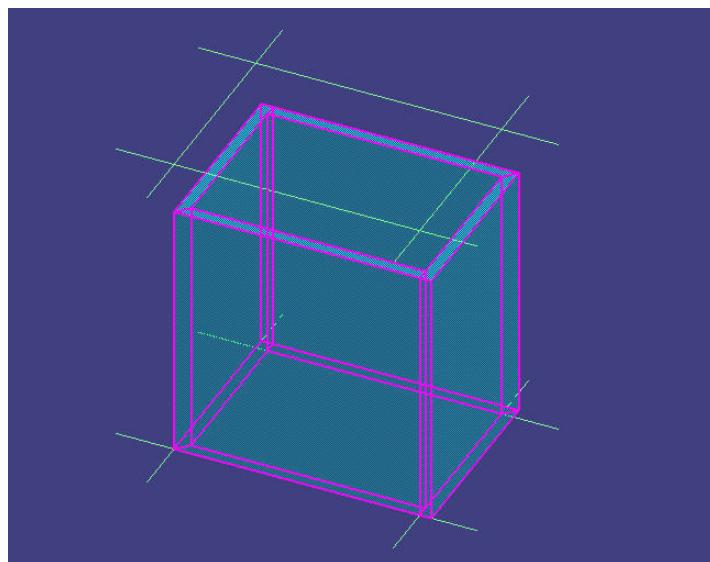
21. Multiple walls can be selected for modification. Select WallSystem-A and WallSystem-B.
22. Your view should resemble the following graphic.



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



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



# LAB-9: Openings

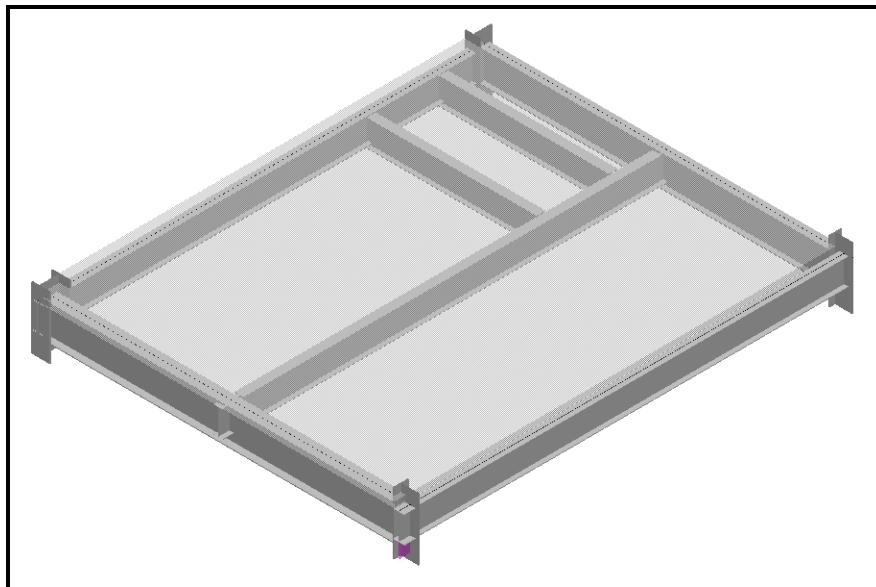
## Objectives

After completing this lab, you will be able to:

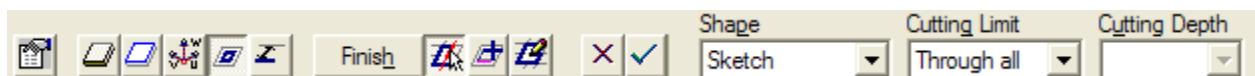
- Understand the opening entities and relationships
- Place Openings using different options

### 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 Place Opening Command.
- 4 Select the slab located in the first floor frame as shown in Figure 1.
- 5 Keep the default parameter in the smartstep ribbon bar.



- 6 Select the members as shown in Figure 1 to define the boundaries of the opening. Use quick pick service to pick the members.

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

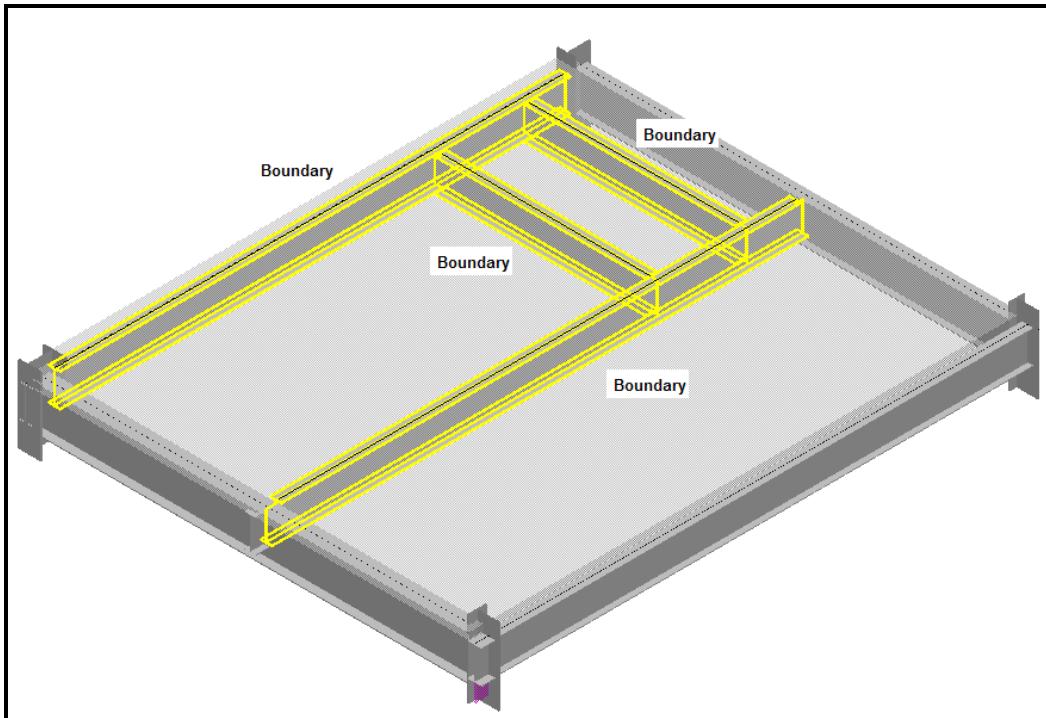
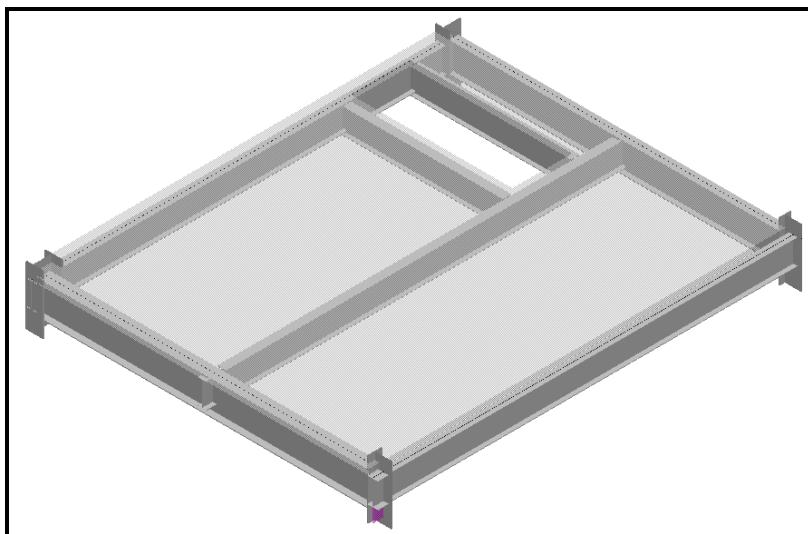


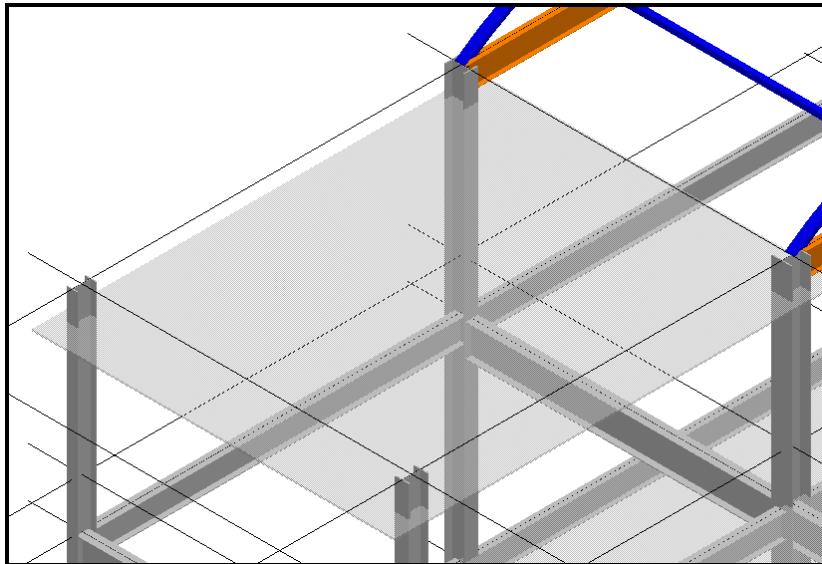
Figure 1 – ISO View of the Stairway Opening

Your View should now resemble the following graphic:

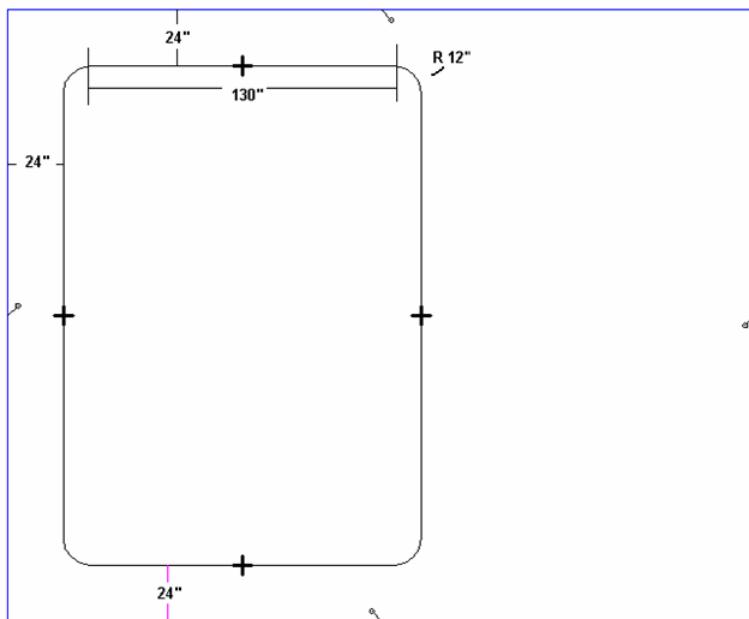


## Part II -Place Opening on the Roof Deck

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



3. Select Place Opening Command.
4. Select the roof deck located at Elevation 43'- 0" for the Smartstep1. Select the draw method button to define the boundaries of the opening. System automatically opens the 2D Editor.



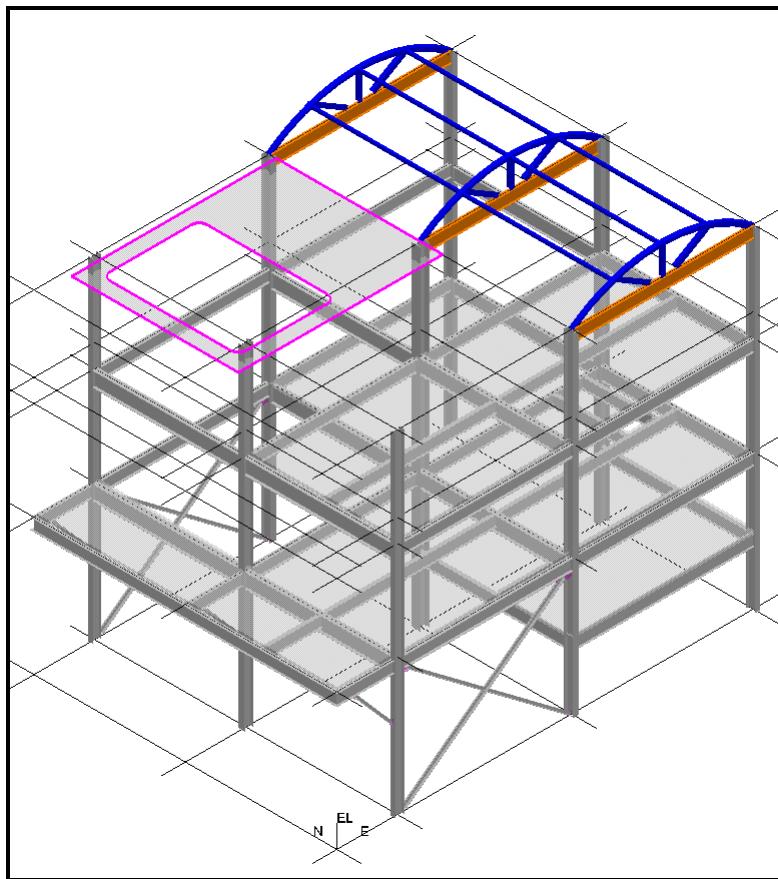
*Hint: Use place line, fillet, distance equal relationship and dimension commands*

5. Hit “Close” button to return to the SmartStep ribbon bar.
6. Keep the default parameters.



7. Hit “Finish” button.

Your View should now resemble the following graphic:



# LAB-10: Stairs / Ladders / Handrails

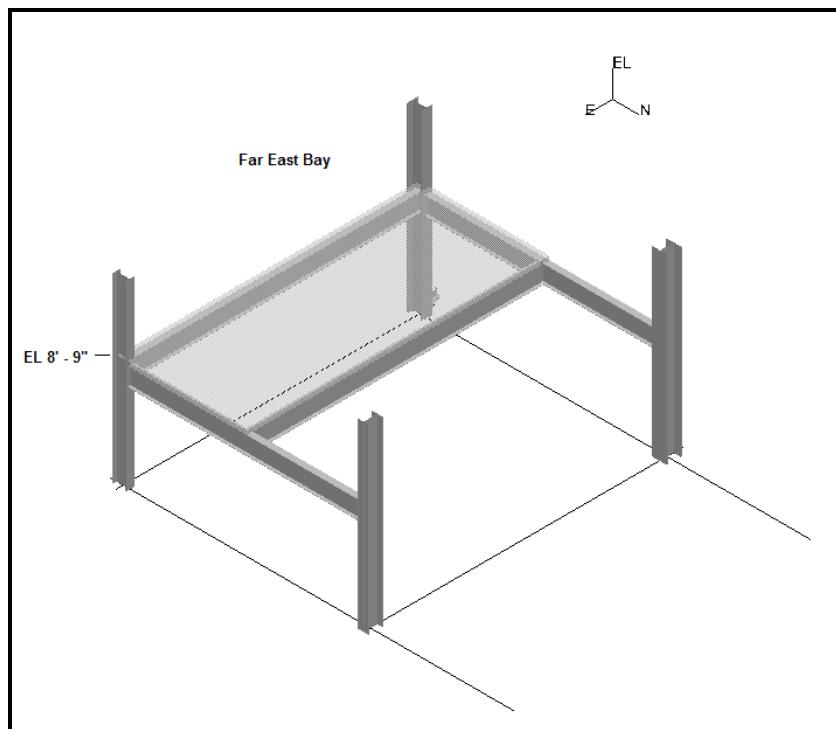
## Objectives

After completing this lab, you will be able to:

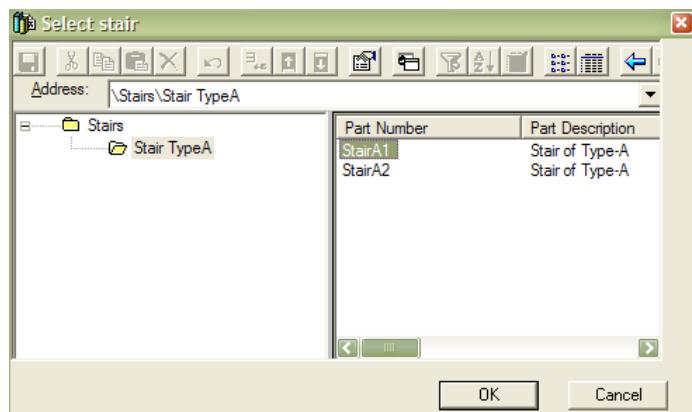
- Understand the stair, ladder and handrail entities and relationships
- Place stairs, ladders and handrails using different methods

### Part I –Place First Stair

1. Use Clip by Object command to isolate the beams and the columns for the Far-East Bay as shown below:



2. Select Place Stair Command on the vertical toolbar.
3. 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. See Figure 1.

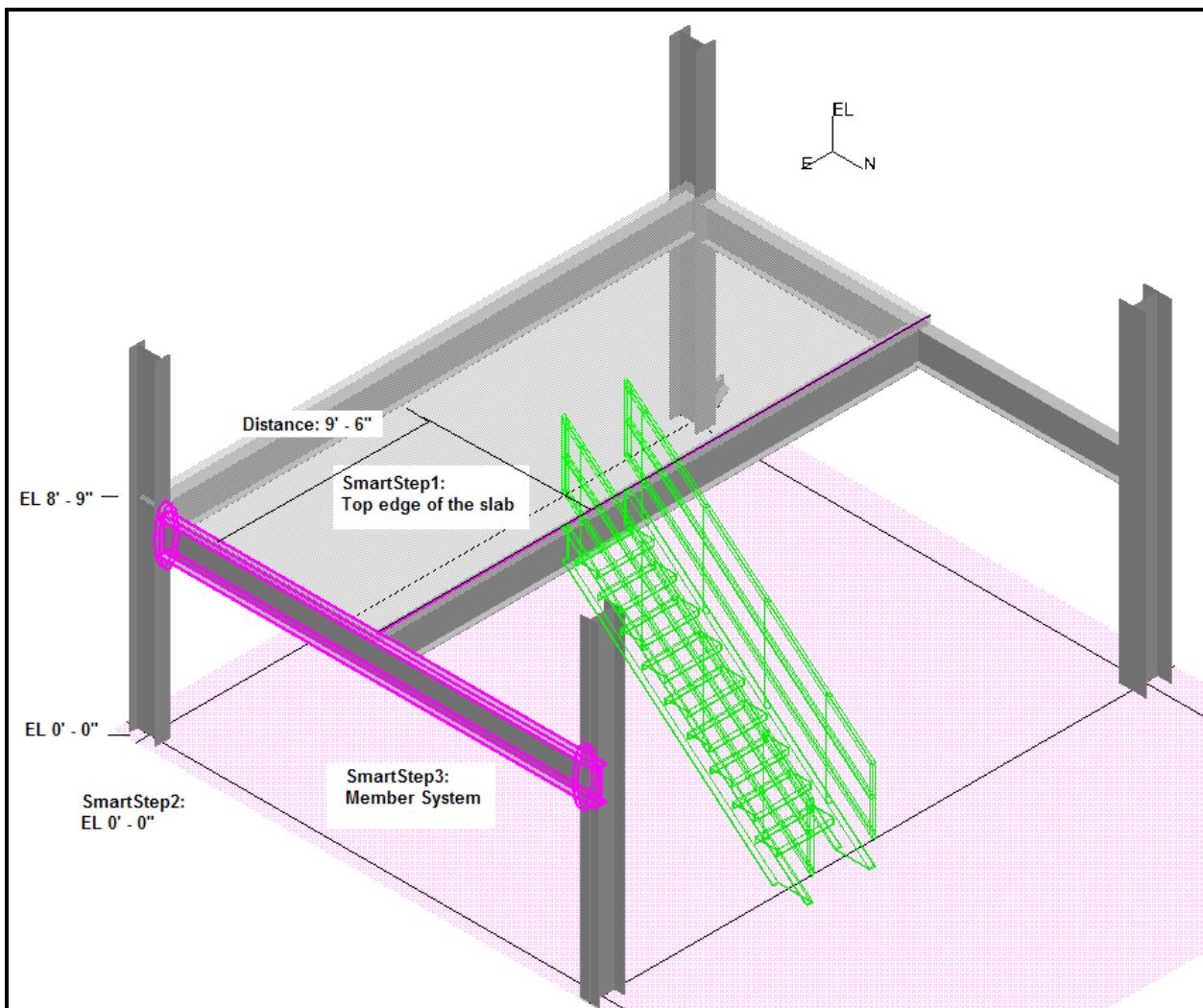


Figure 1 – ISO View of the first stair

5. Select the elevation plane at 0'- 0" for the stair. This plane defines the bottom elevation of the stair.

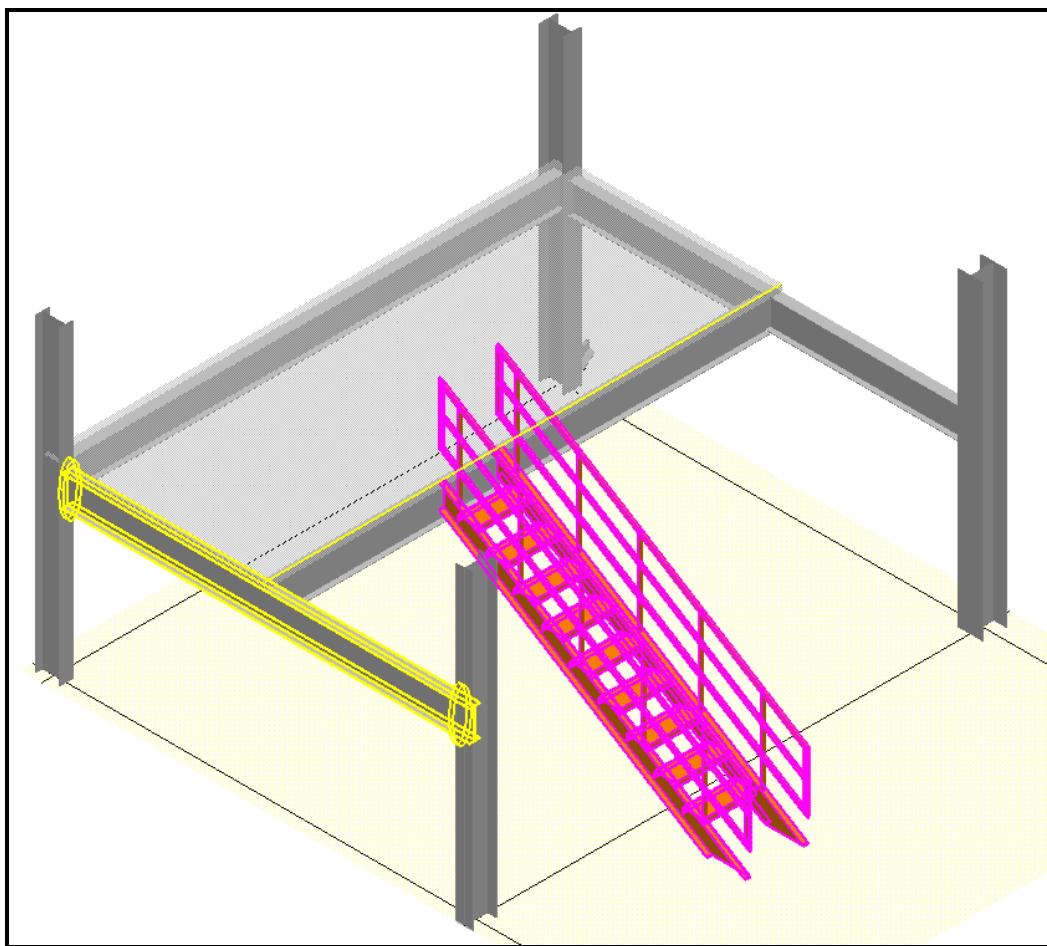
6. Select the reference edge for the stair.

7. Set the following parameters:

Width:	2' - 6"
Angle:	32 deg
Pitch:	0' -10"
Horizontal Offset:	9' - 6"
Vertical Offset:	0' - 0"
System:	A2-> U03 -> Structural -> Miscellaneous

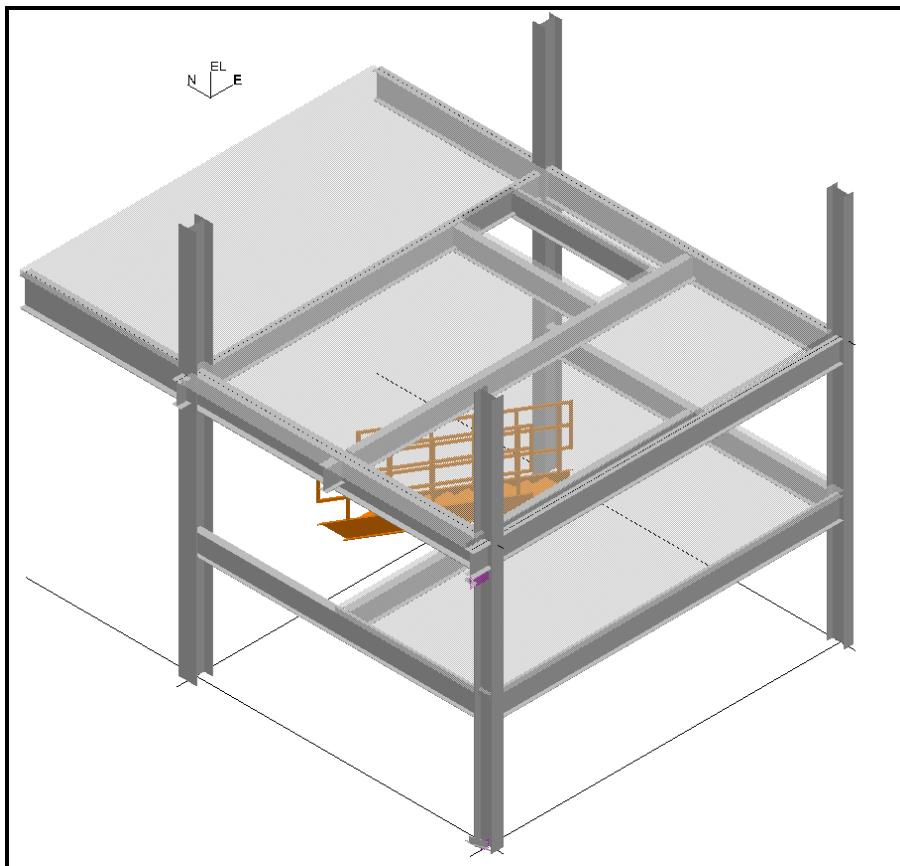
8. Click “Finish” Button.

Your View should now resemble the following graphic:



**Part II -Place the Second Stair**

9. Use Clip by Volume command to adjust the clipping volume as shown below:



10. Select Place Stair Command.
11. Select stair A1 from the catalog browser dialog box.
12. Select the top edge of the beam. This defines the top elevation of the stair. See Figure 2.

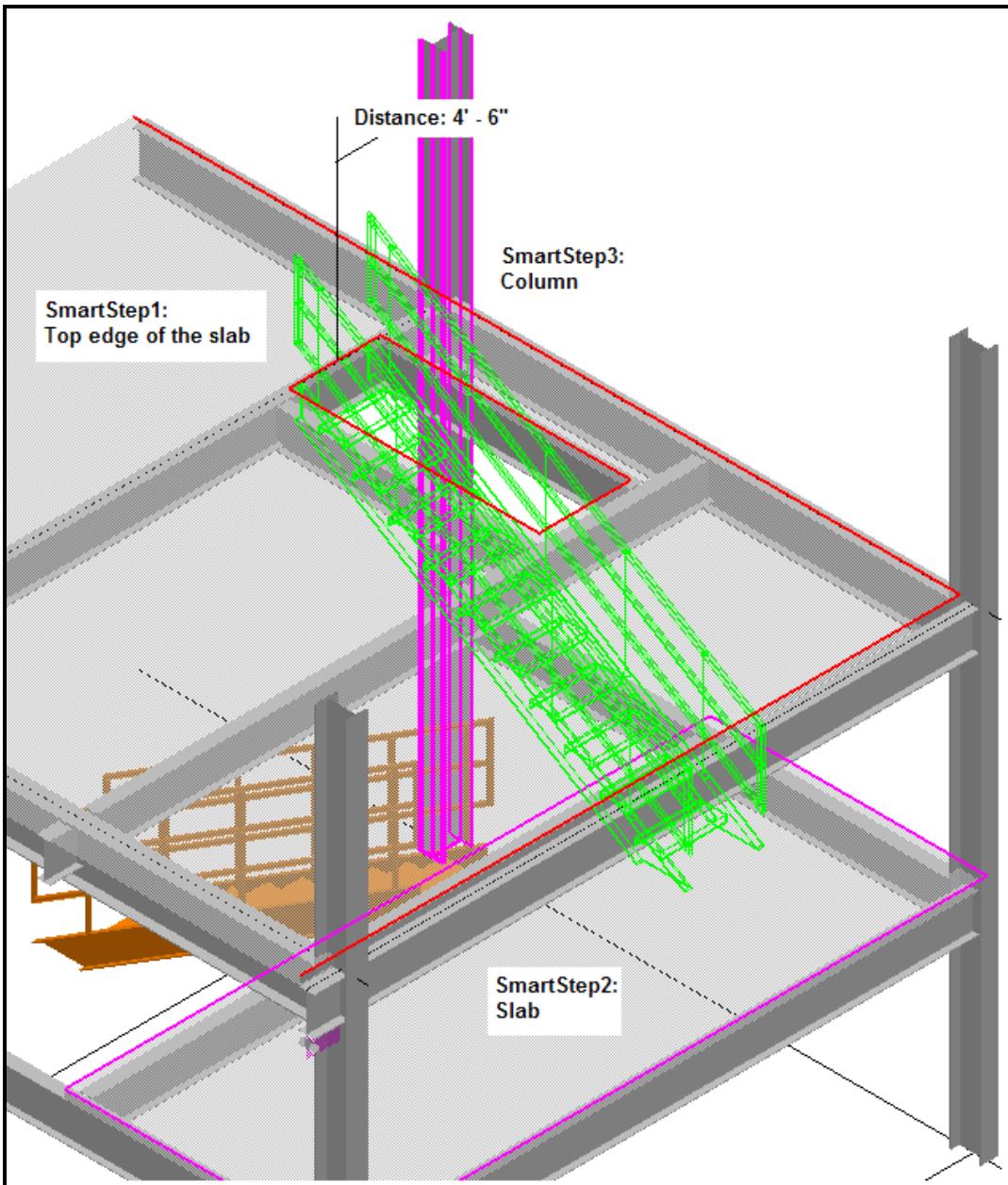


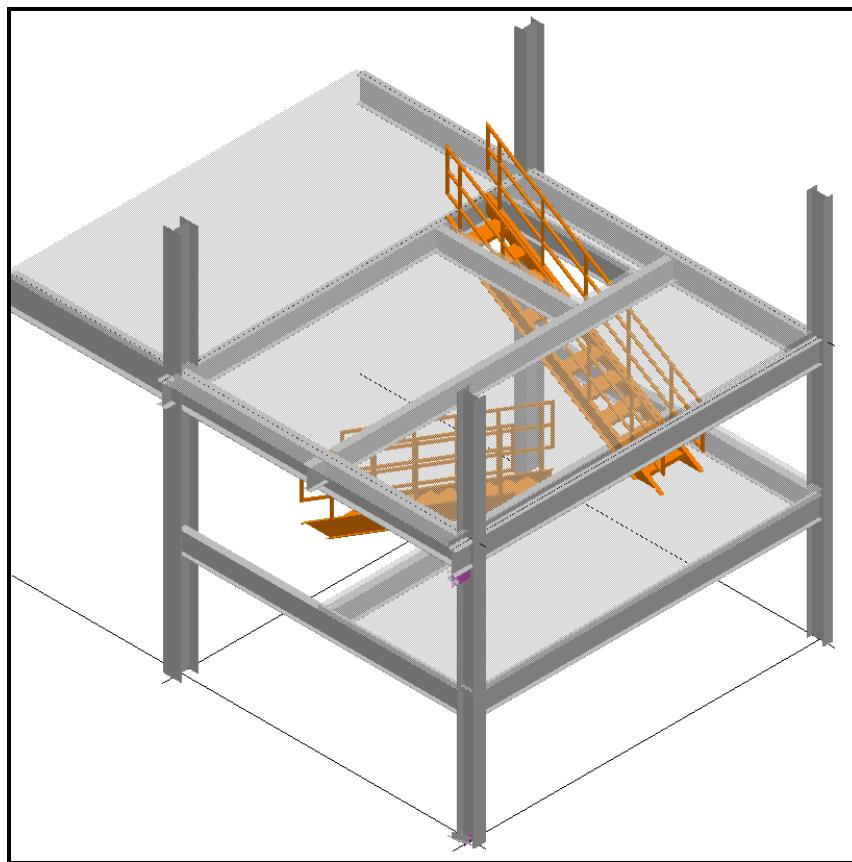
Figure 2 – ISO View of the second stair

13. Select the top surface of the slab for the stair. This plane defines the bottom elevation of the stair.
14. Select the reference edge (column) for the stair.
15. Set the following parameters:

System:	A2->U03 -> Structural -> Miscellaneous
Width:	2'- 6"
Angle:	32 deg.
Pitch:	0.83'- 0"
Horizontal Offset:	4'- 6"
Vertical Offset:	0'- 0"

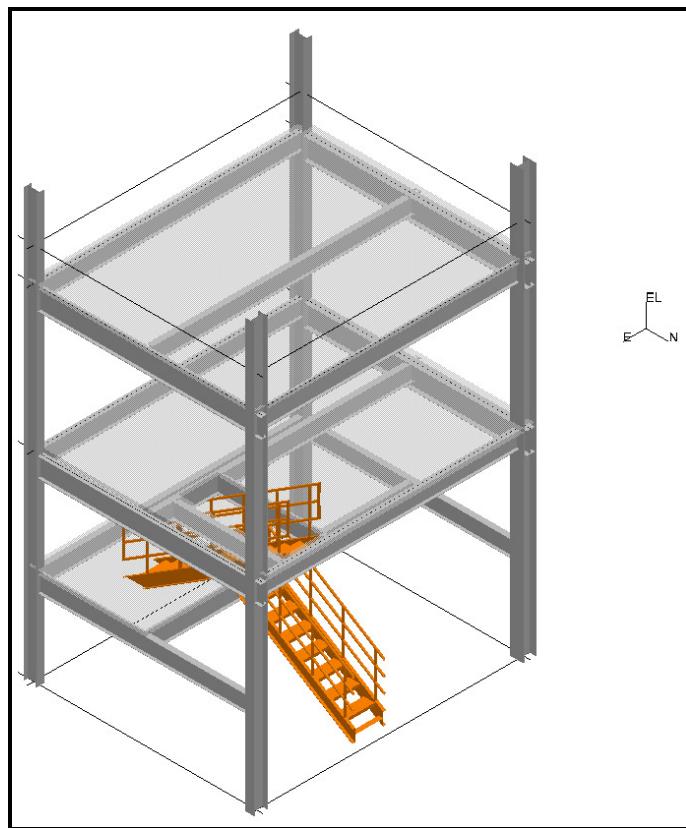
16. Click “Finish” button.

Your View should now resemble the following graphic:

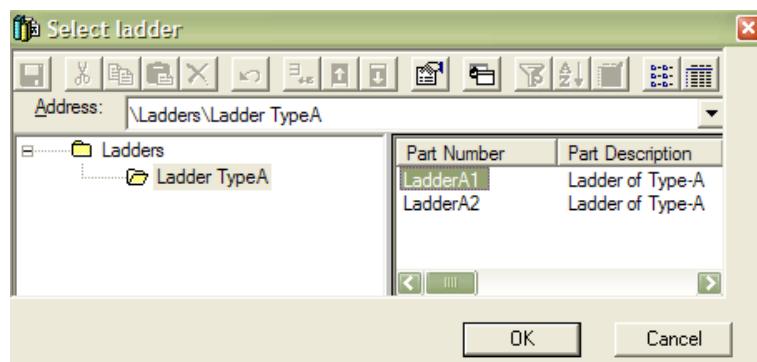


### **Part III – Place a Ladder**

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



2. Select Place Ladder Command on the vertical toolbar.
3. Select ladder A1 from the catalog browser dialog box.



4. Select the top edge of the beam. This edge defines the top elevation of the ladder.  
See Figure 3 for detail.

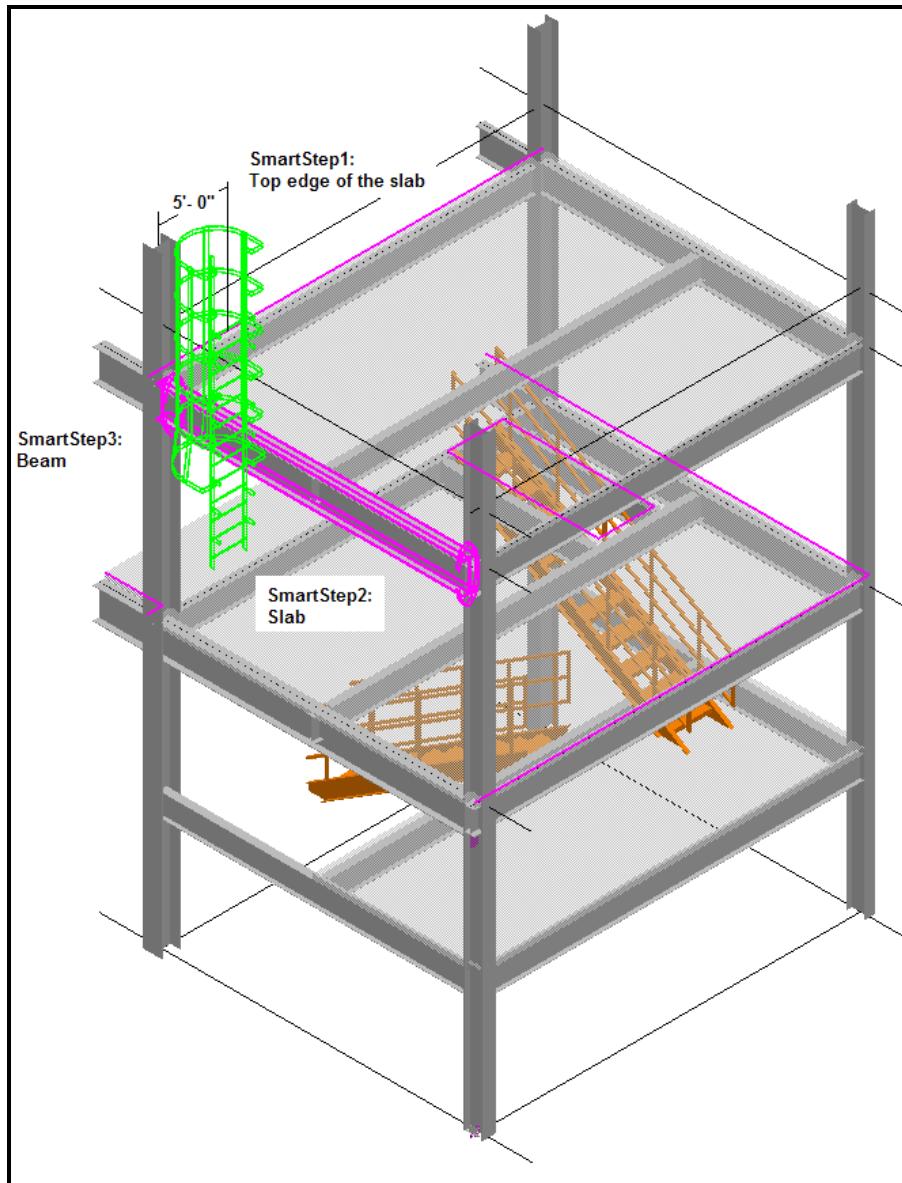


Figure 3 – ISO View of the ladder

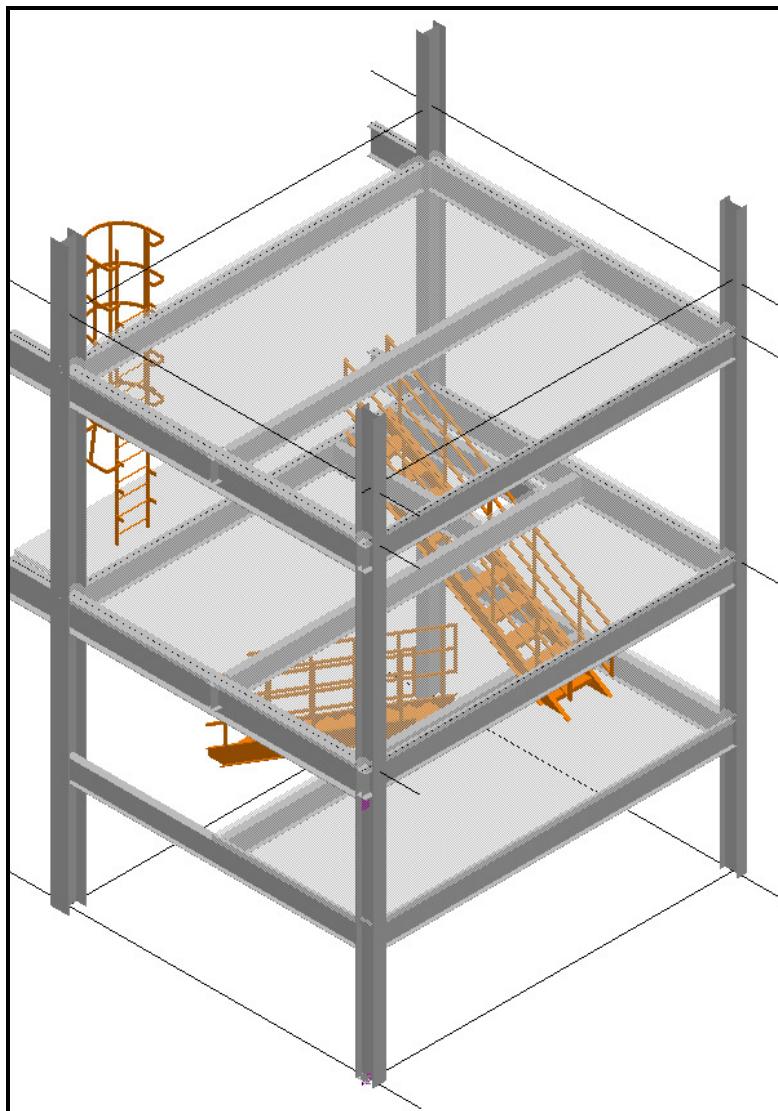
5. Select the top surface of the slab located at Elevation 18'- 0". This plane defines the bottom elevation of the ladder.
6. Select the reference edge for the ladder.
7. Set the following parameters as follows:

System: A2 -> U03 -> Structural -> Miscellaneous  
Width: 2' - 0"  
Angle: 90.0 deg.

Pitch: 1'- 0"  
Horizontal Offset: -5' - 0"  
Vertical Offset: 0'- 0"

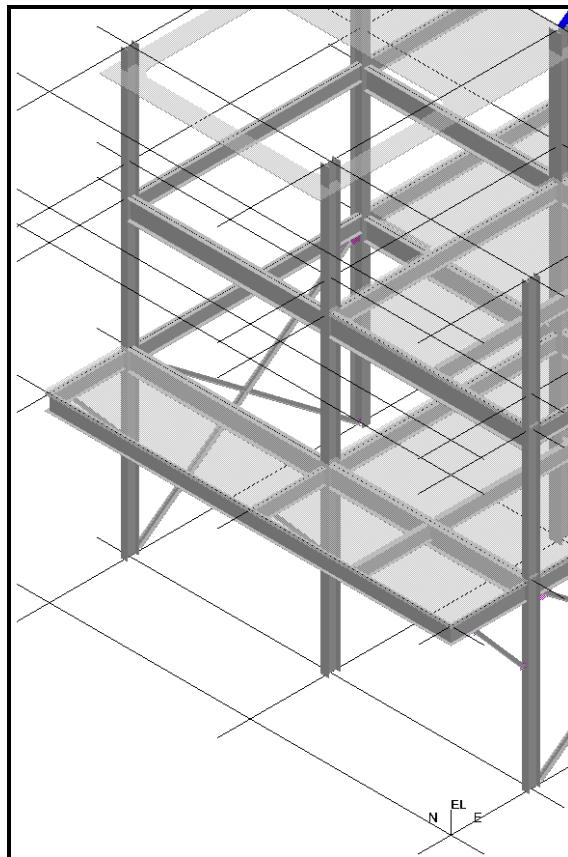
8. Click “Finish” button.

Your View should now resemble the following graphic:

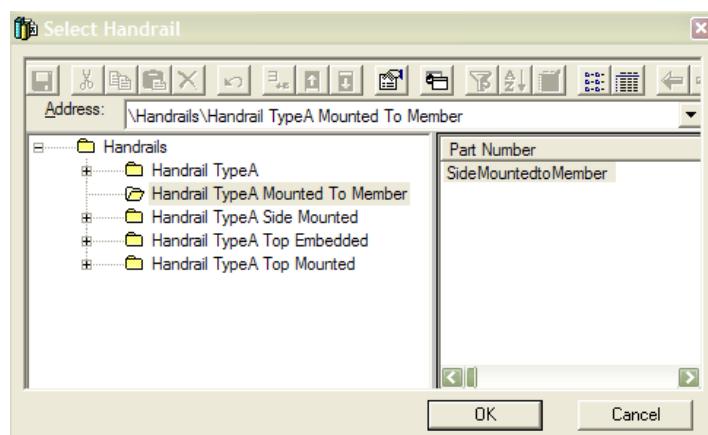


## **Part IV – Placing Handrail**

1. Select View-> Clear Clipping to remove the clipping volume.
2. Use Zoom Tool to window area the East side of the building as shown below:



3. Select Place Handrail by Member Command on the vertical toolbar.
4. Select handrail SideMountedtoMember from the catalog browser dialog box.

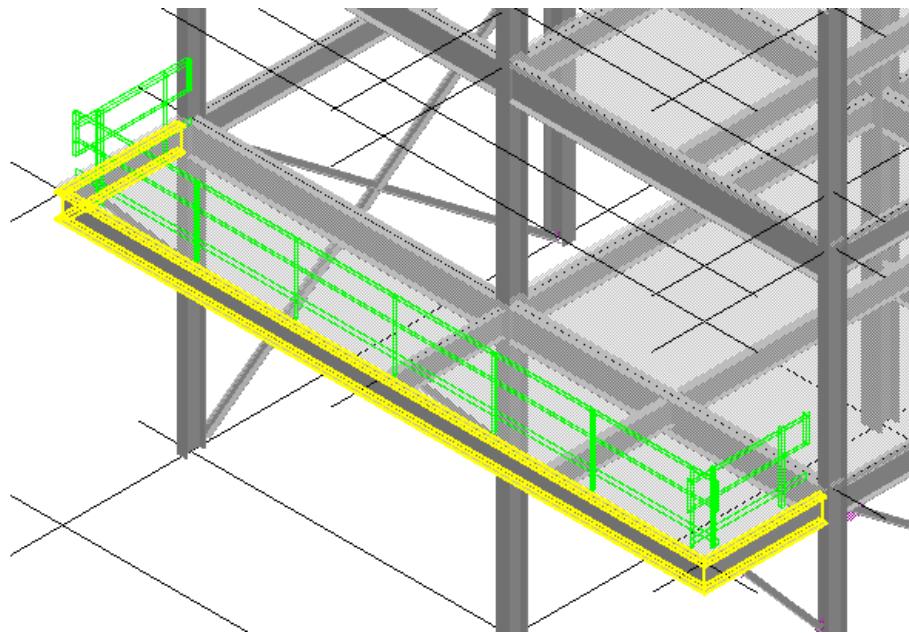


5. Set the following parameters as follows:

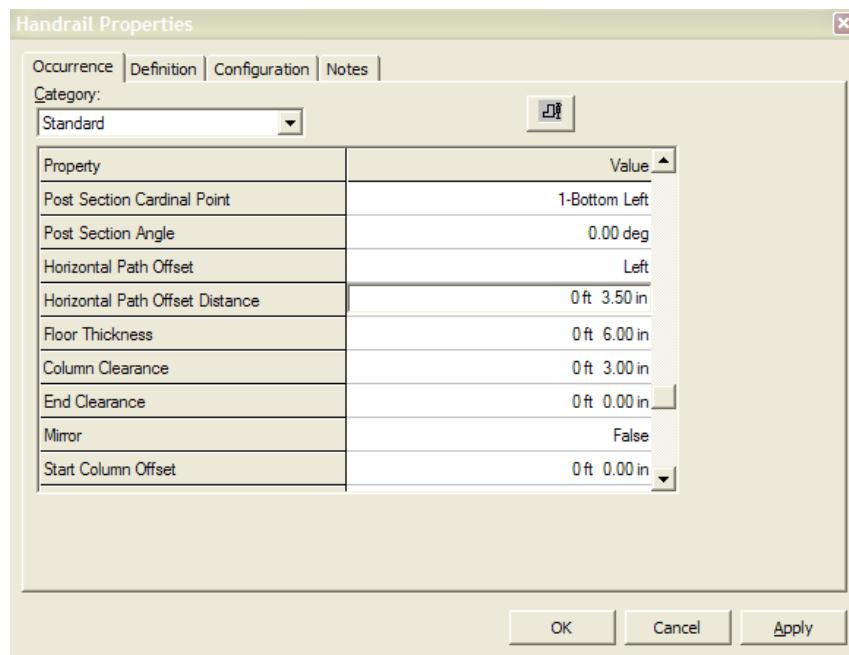
System:	A2-> U03 -> Structural ->Miscellaneous
Begin Treatment:	Rectangular
End Treatment:	Rectangular
Offset Reference:	Centerline
Column Clearance:	0'- 3"
End Clearance:	0'- 0"



6. Select three beams for the SmartStep1 and click “Accept” button.

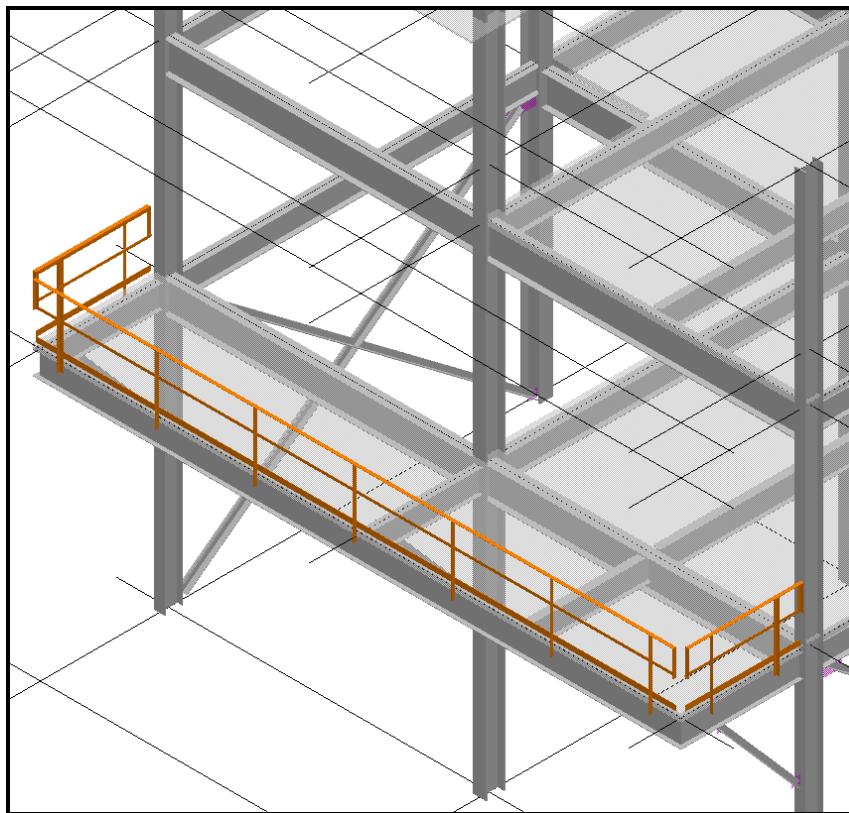


7. Select the slab as the walking surface location for SmartStep2.
8. Click “Finish” button.
9. Click “Select Command” on the vertical toolbar and select the handrails you just have placed.
10. Open the properties page and change the Horizontal Path Offset Distance to 0'- 3.5".



11. Click “OK” button.

Your View should now resemble the following graphic:



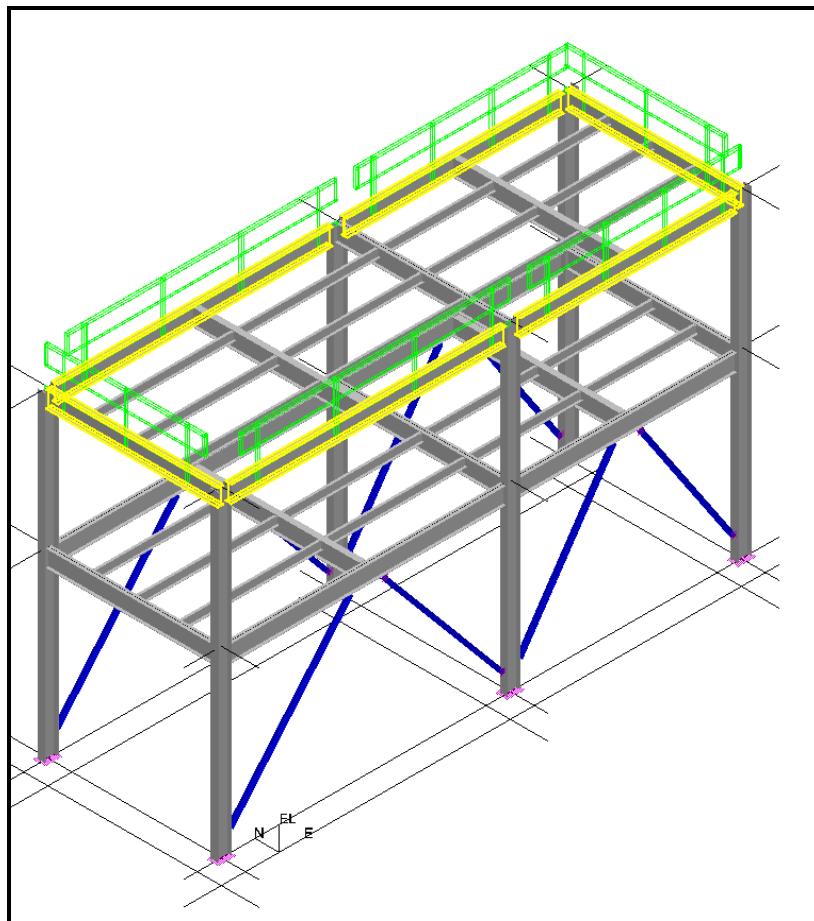
## **Part V – Placing Handrail in Structure U02**

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:

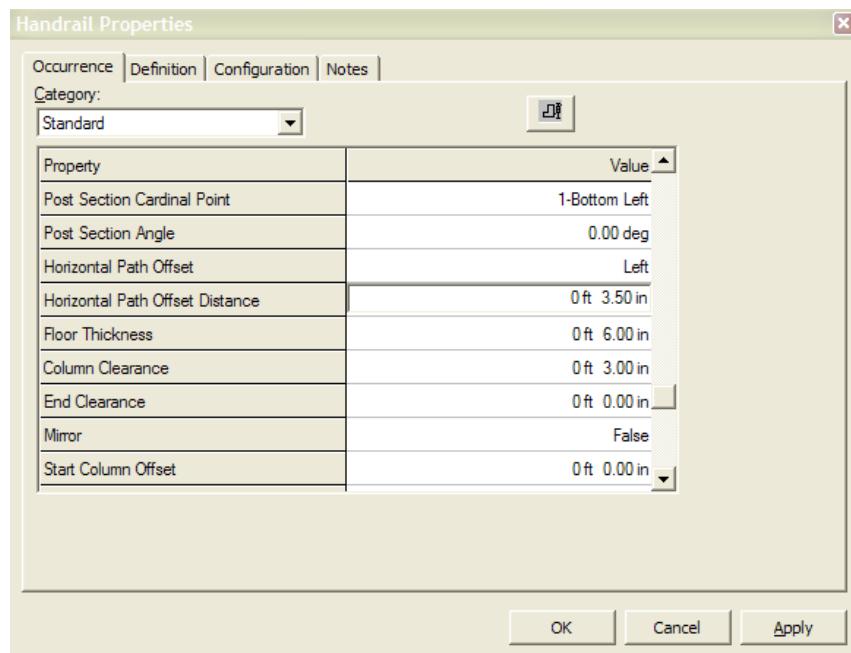
System:	A2-> U02 -> Structural ->Miscellaneous
Begin Treatment:	Rectangular
End Treatment:	Rectangular
Offset Reference:	Centerline
Column Clearance:	0'- 3"
End Clearance:	0'- 0"



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

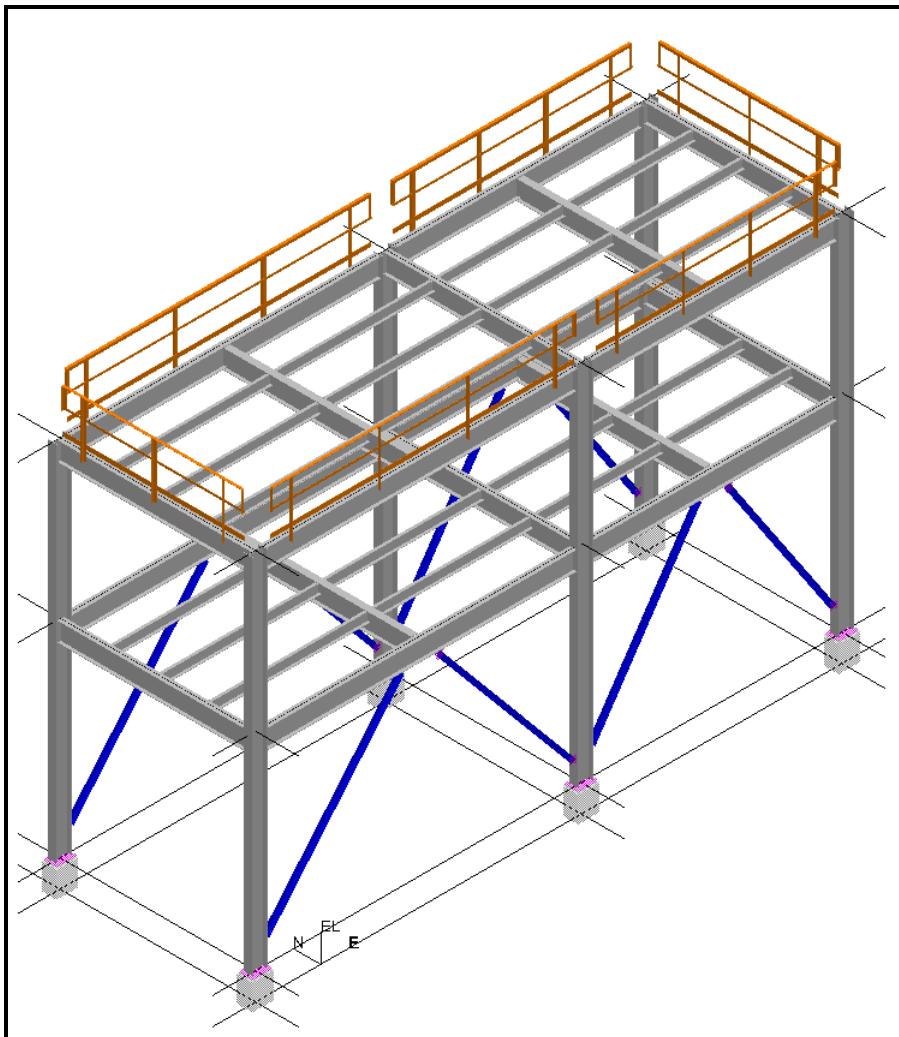


6. Select an interior beam as the walking surface location for SmartStep2.
7. Click “Finish” button.
8. Click “Select command” on the vertical toolbar and select the handrails you have just placed.
9. Open the properties page and change the Horizontal Path Offset Distance to 0'- 3.5".



10. Click “OK” button.

Your View should now resemble the following graphic:

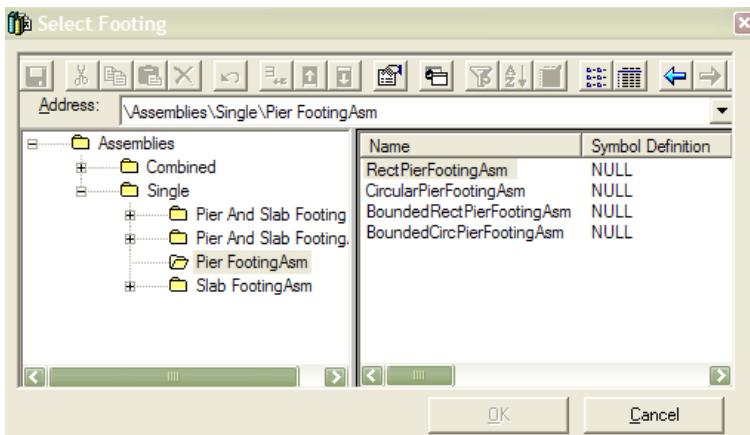


# LAB-11: Footings

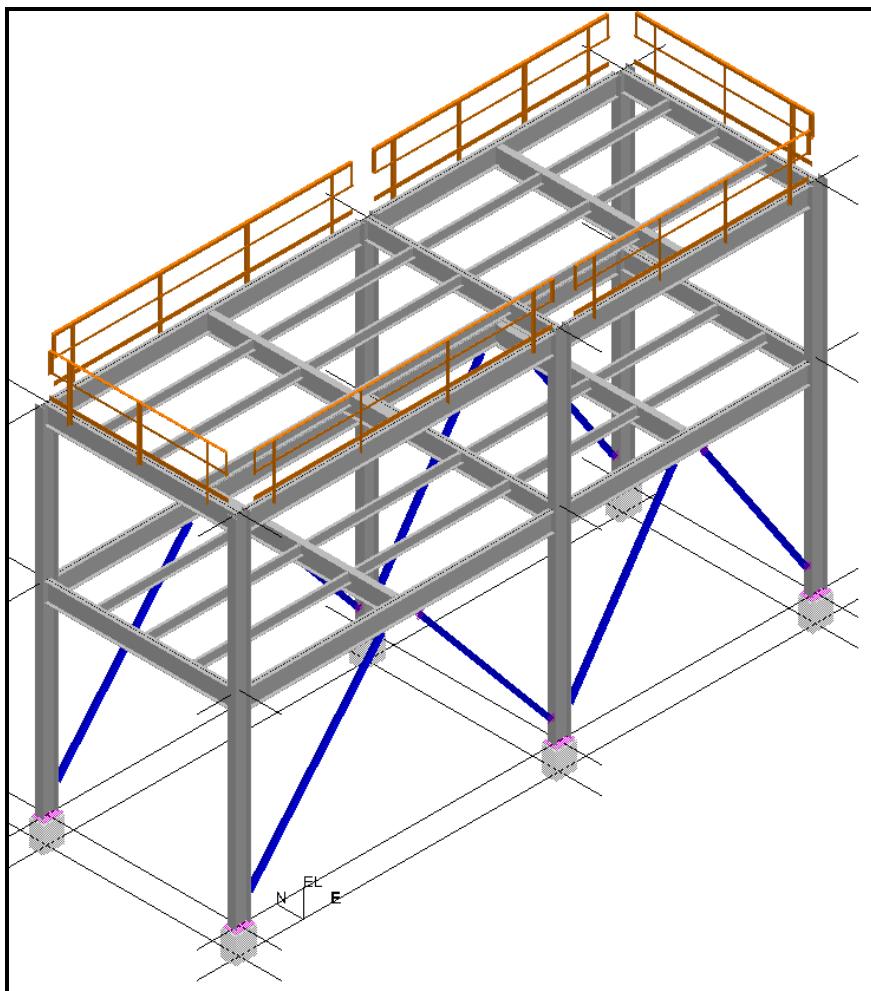
## Objectives

After completing this lab, you will be able to:

- Understand the footing entities and relationships
  - Able to Place different type of footings
1. Select Place Footing Command on the vertical toolbar.
  2. Navigate the Assemblies Hierarchy and select RectPierFootngAsm 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 “Accept” button.
5. Go to the System combo box and select A2 ->U02 -> Structural ->Footings
6. Hit the “Finish” button to place all footings in the model.



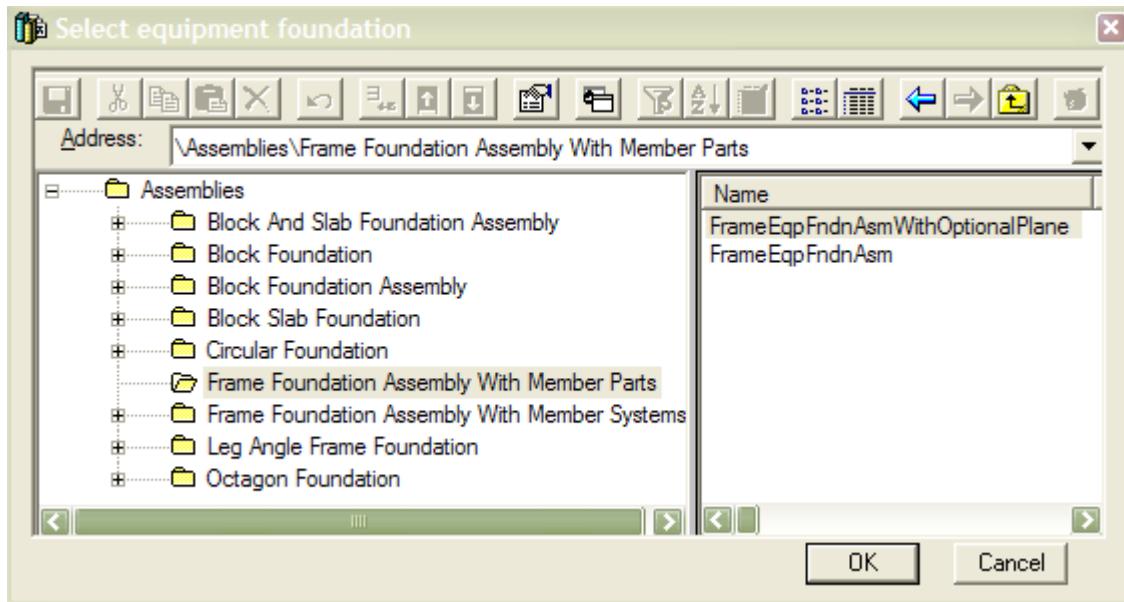
# LAB-12: Equipment Foundations

## Objectives

After completing this lab, you will be able to:

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

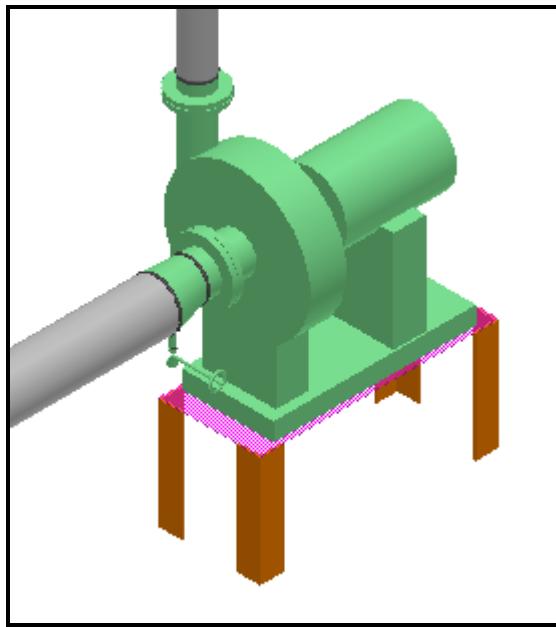
1. Set your workspace to show pump P-101 located in A2 -> U03 -> Equipment system.
2. Select Place Equipment Foundation Command on the vertical toolbar.
3. Select pump P-101 on SmartStep1.
4. Uncheck the By rule option and Select the Type combo box to open the Catalog browser.
5. Navigate the Assemblies Hierarchy and select FrameEqpFndAsmWithOptionalPlane from the catalog browser dialog box.



6. Hit “OK” button to close the catalog browser dialog box.
7. Select “Accept” button.

8. Select the slab on grade or a E1 0'- 0" plane for the support plane.
9. Go to the System combo control and select A2 -> U03 -> Structural -> Miscellaneous
10. Hit the “Finish” button to place the equipment foundation assembly in the model.

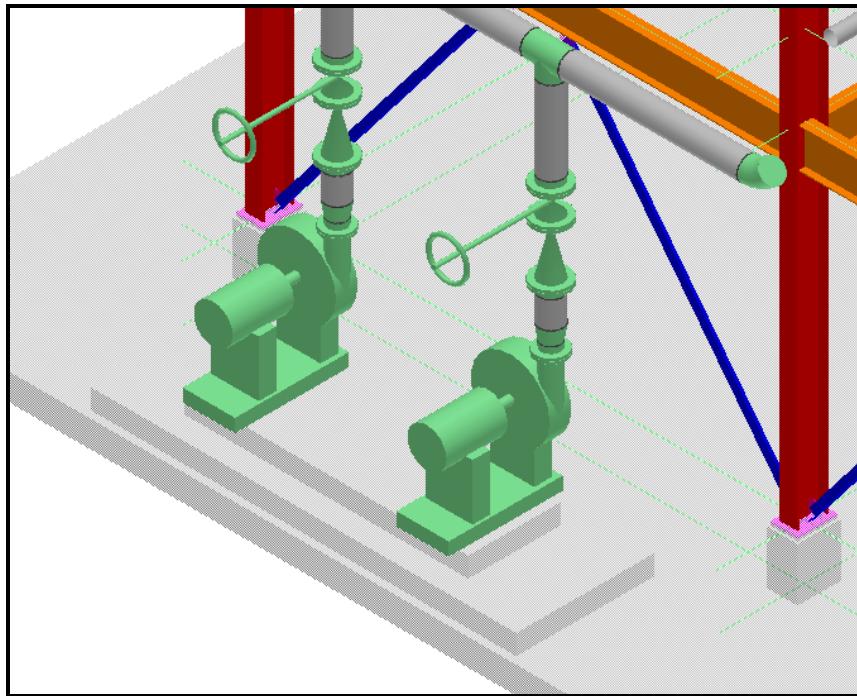
Your View should now resemble the following graphic:



11. Select Place Equipment Foundation Command on the vertical toolbar.
12. Check the By Rule Option.
13. Select Pump-001 and Pump-002 located in A2 -> U01-> Equipment in your workspace.

*Notice that the system selects the default foundation defined in the reference data*
14. Select “Accept” button.
15. Uncheck By Rule and select BlockandSlabEqpFndnAssmWithOptionalPlane.
16. Select Slab on Grade for the support plane.
17. Go to the System combo control and select A2-> U01 -> Structural -> Miscellaneous System.
18. Hit the “Finish” button to place the equipment foundation in the model.

Your View should now resemble the following graphic:

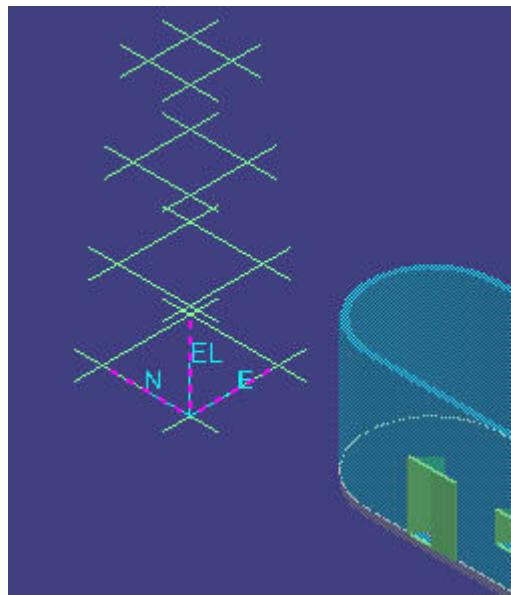


# LAB-13: Designed Solids

## Objective

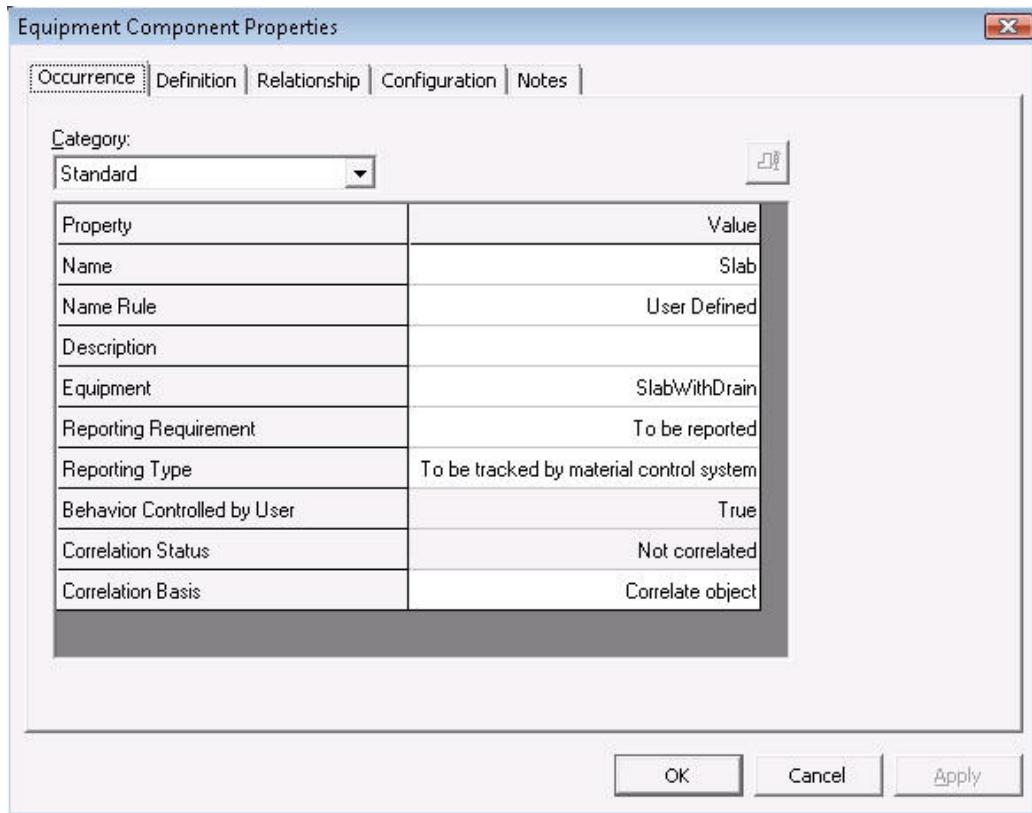
After completing this lab, you will be able to:

- 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. 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 U05CS by expanding the CS. Select the reposition target and select the highlighted U05 CS. Your view should resemble the following graphic.



4. Expand the fly-out toolbar and press the  icon on the vertical toolbar to start the Place Designed Equipment command. Expand *Equipment -> Civil* and select *Miscellaneous*. 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: -60 ft, N: -20 ft and EL: 0 ft.
7. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Equipment Component  icon.
8. Select *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\Process Components\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

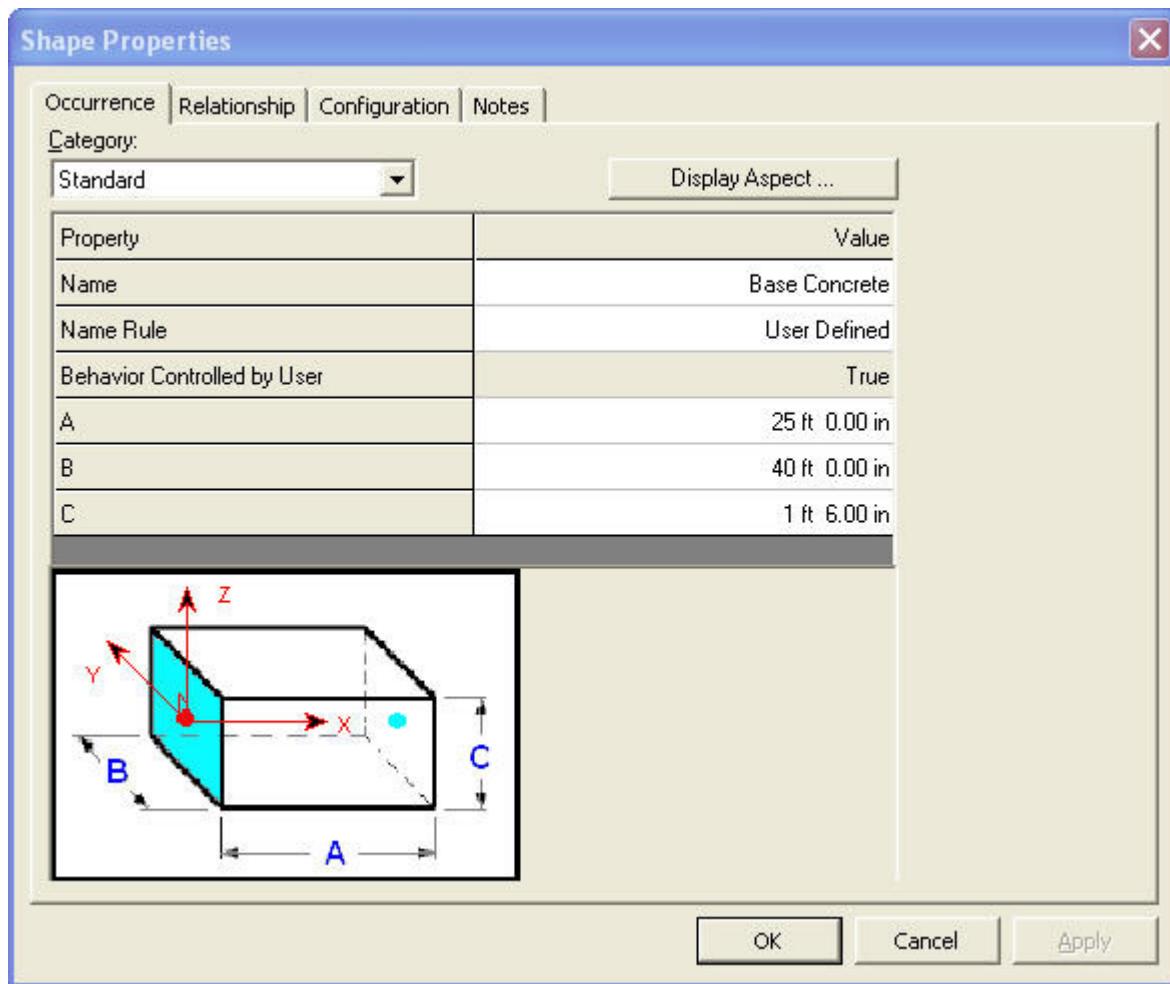


11. Place *Slab* at E: -60 ft, N: -20 ft and EL: 0 ft.
12. Expand the Fly-out by holding down the  icon on the vertical toolbar. Select the Place Designed Solid  icon.

13. Choose *Slab* as the parent. Rename the Solid to *SlopedSlab*.
14. Go to Edit → Properties. Properties window will show up. Set the Material Type and Grade to Concrete and Fc 3000 for Solids, *SlopedSlab*. Your workspace explorer should resemble the following graphic.

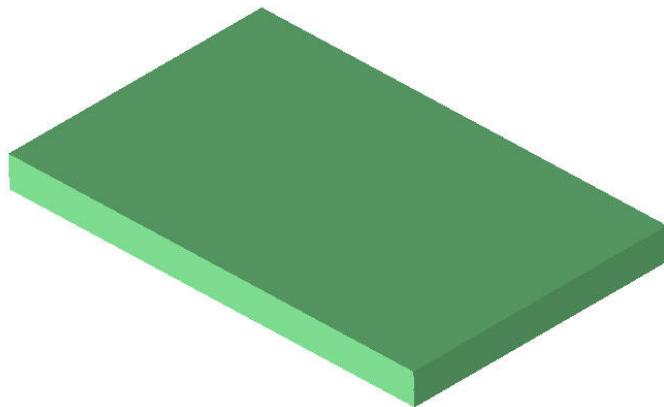


15. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid* from the Shapes dialog. Choose *SlopedSlab* as the parent of the Shape.
16. In the Shape Properties dialog, set A to 25 ft, B to 40 ft, and C to 1 ft 6in. Name the *RectangularSolid* as *Base Concrete*. Press OK.



17. Place the Shape at E: -60 ft, N: -40 ft and EL: 0 ft.

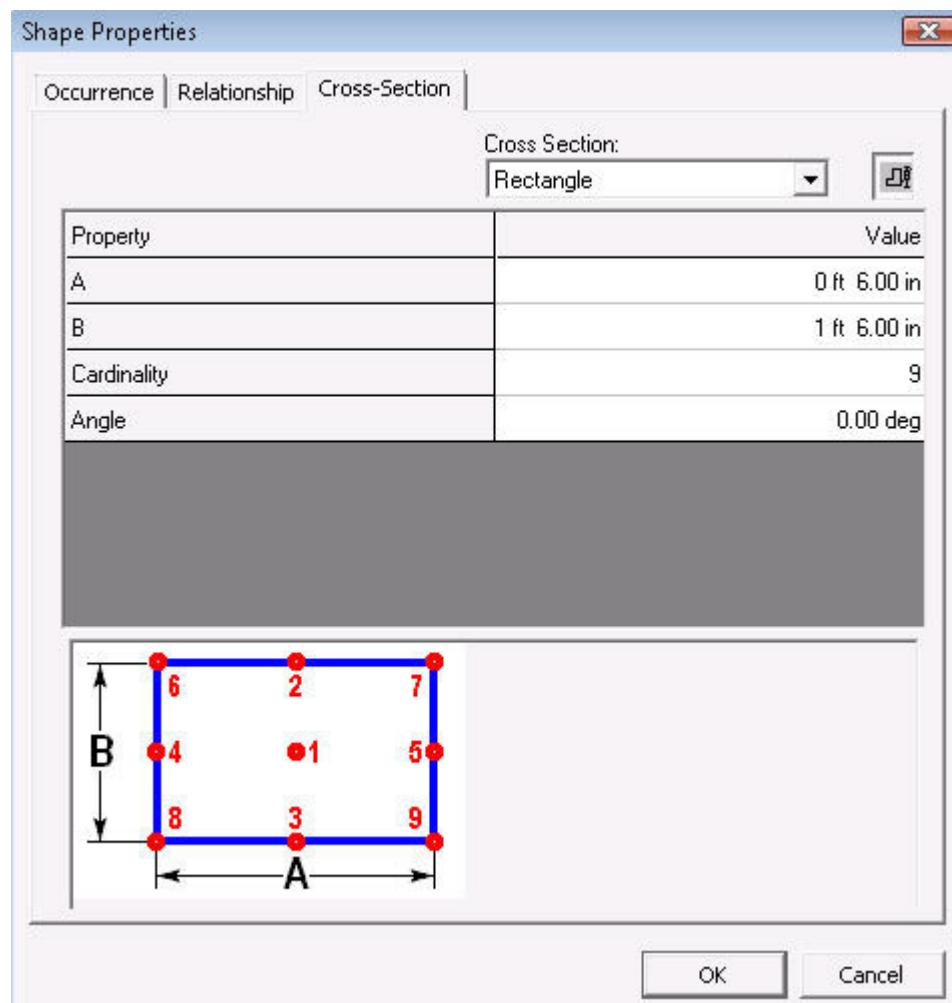
18. Select *RectangularSolid* from the Shapes dialog. Choose *SlopedSlab* as the parent of the Shape. Your view should resemble the following graphic.



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.

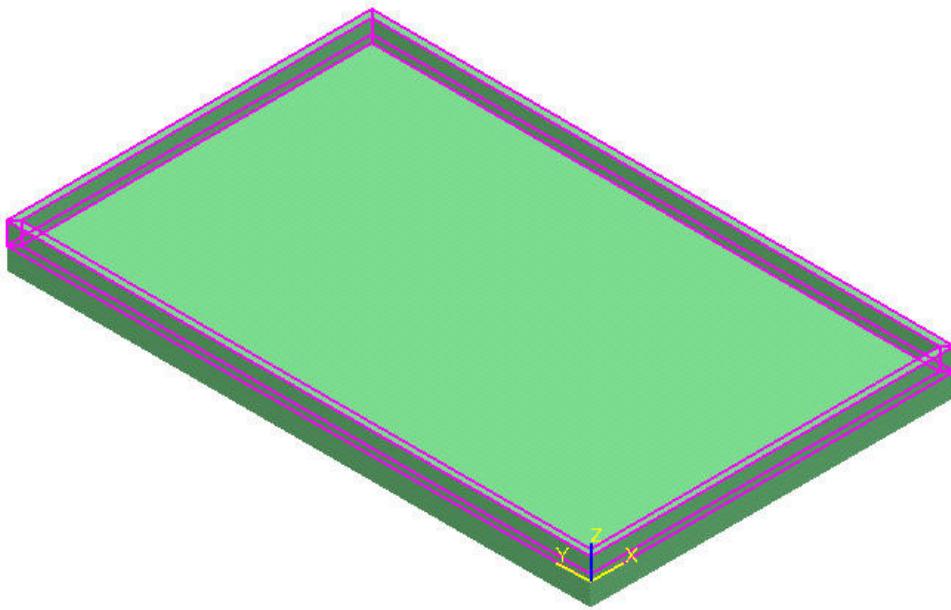


21. Choose Rectangle from the Cross Section pull down menu and enter the value for A = 6 in and B = 1 ft 6 in. Enter cardinality = 9. Your view should resemble the following graphic.



22. Select the top edges of the *Base Concrete*. Name the *Prismatic Shape* as “*Curb*” and hit the finish. Your view should now resemble the following graphic. Hit Finish.

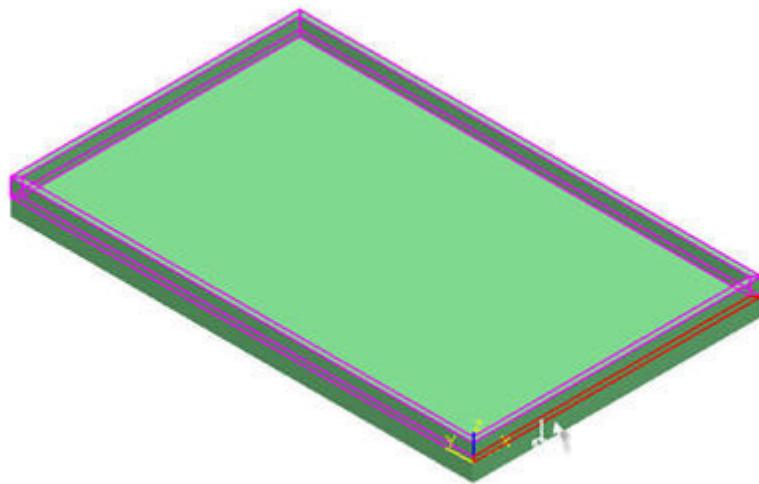




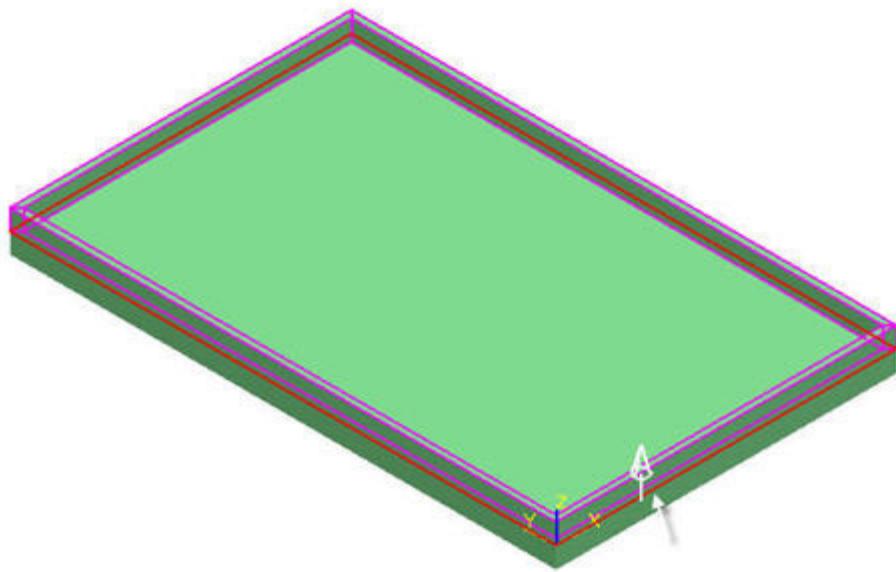
23. Select *Curb* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Mate” as the new relationship.



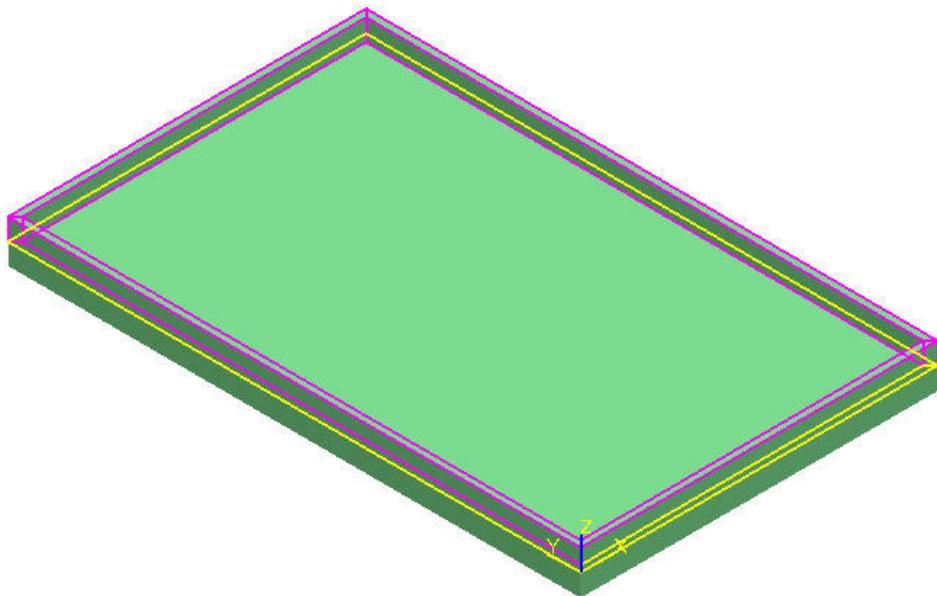
24. Select the plane as shown in the following graphic.



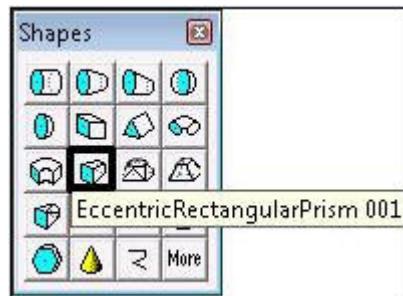
25. Select the top face of *Base Concrete* and your view should resemble as shown in the graphic below,



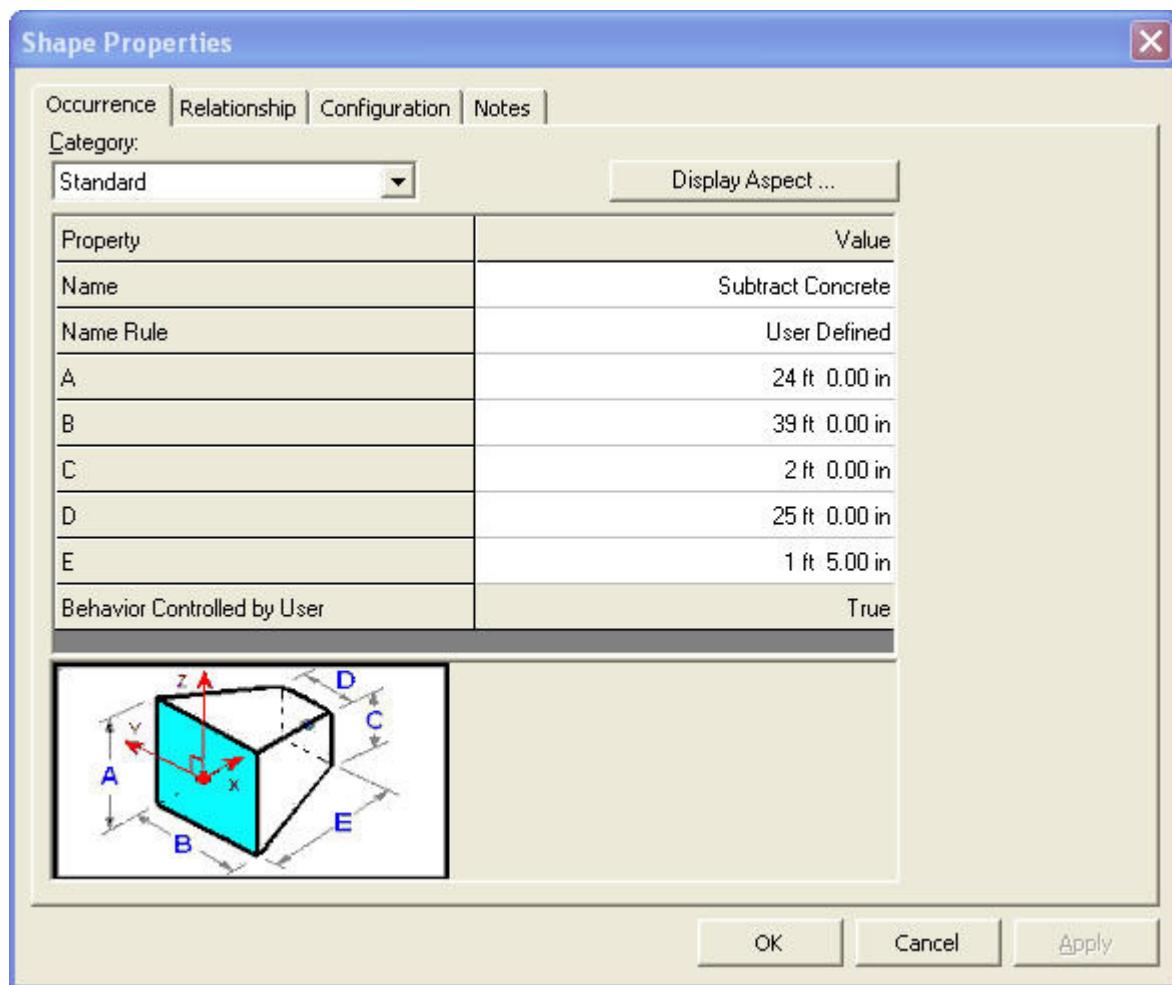
26. Your graphic should resemble as shown below.



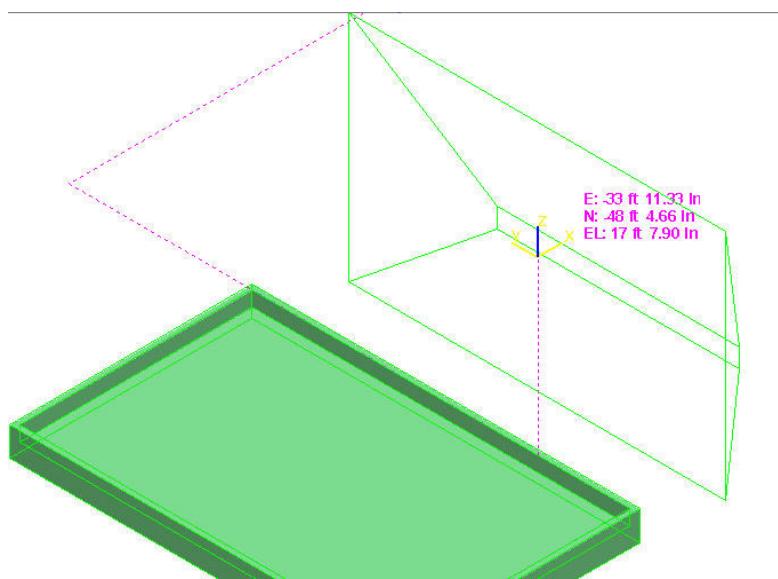
27. Select the EccentricRectangularPrism 001 from the shapes dialog box as shown below in the image.



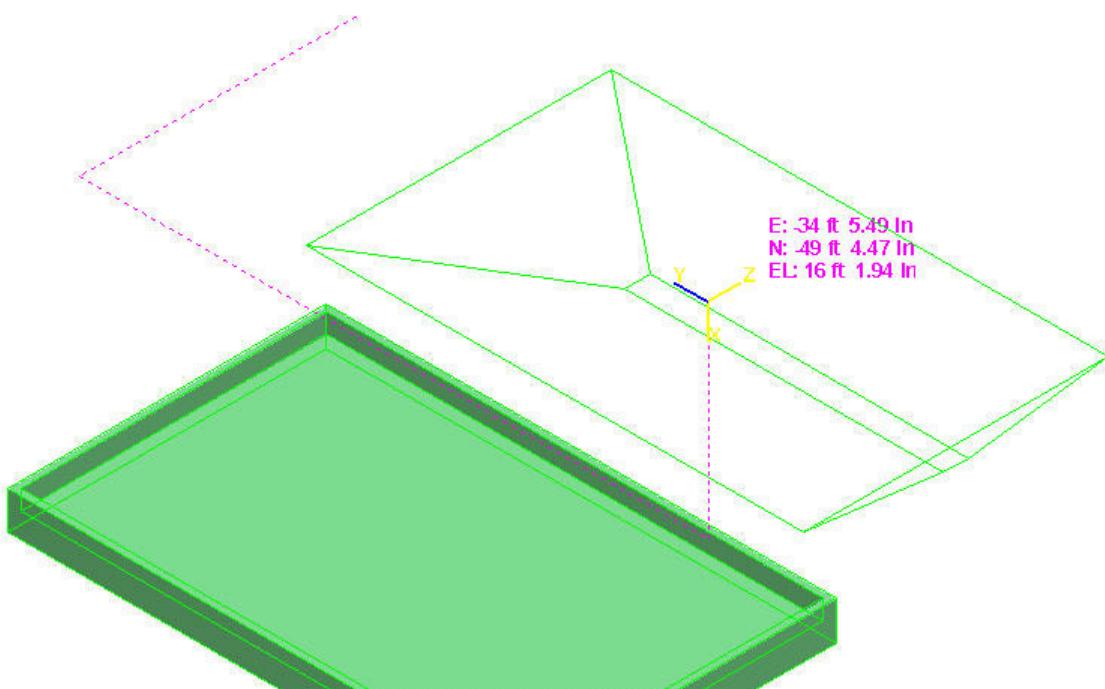
28. Choose *SlopedSlab* as the parent of the Shape.
29. In the Shape Properties dialog, set A to 24 ft, B to 39 ft, C to 2 ft, and D to 25 ft and E to 1 ft 5 in. Name the EccentricRectangularPrism 001 as “*Subtract Concrete*”. Press OK.



30. Your view should now resemble the following graphic

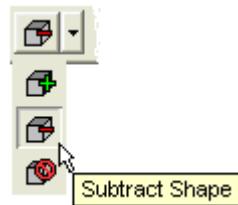


31. We need to rotate the Shape to place it correctly. This can be done before placement time using the Arrow keys on the keyboard.
32. Press Up arrow key once and then press left arrow key once. Your view should resemble the following graphic.

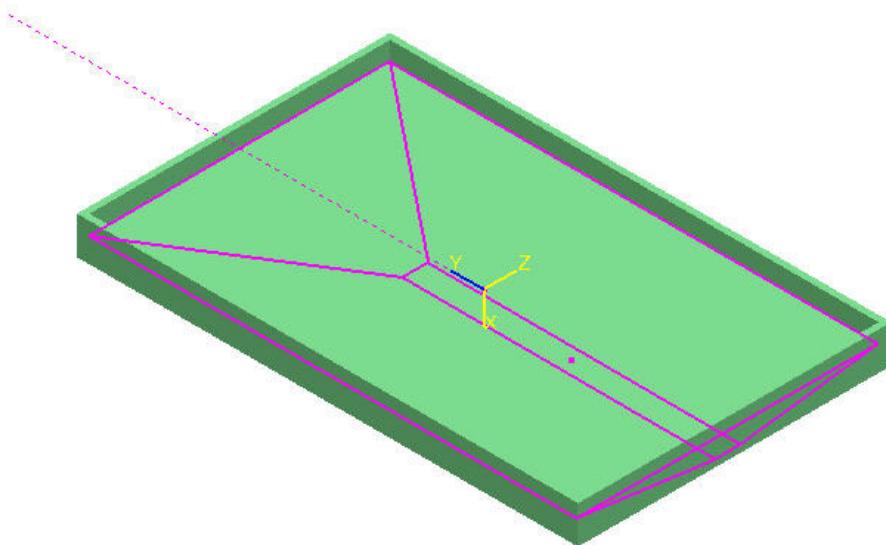


33. Place the Shape at E: -47 ft 6in, N: -40 ft and EL: 0ft 9 in.

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



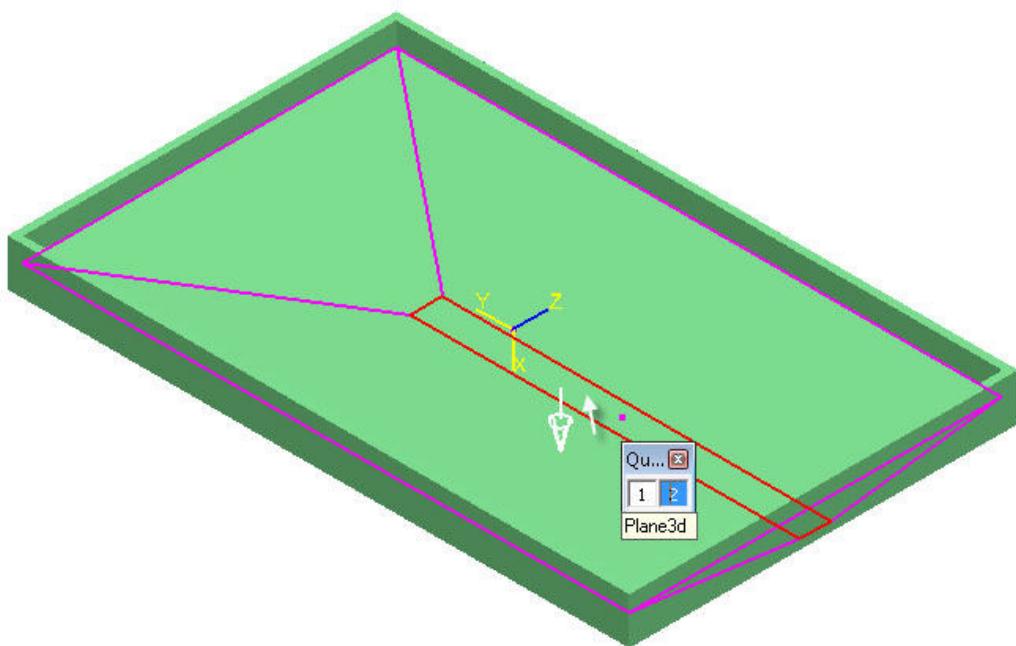
35. Your view should resemble the following graphic.



36. Select *Subtract Concrete* from the workspace explorer. From the smart toolbar choose New Relationship. Choose “Mate” as the new relationship and key in -1 ft 5 in as the offset value.

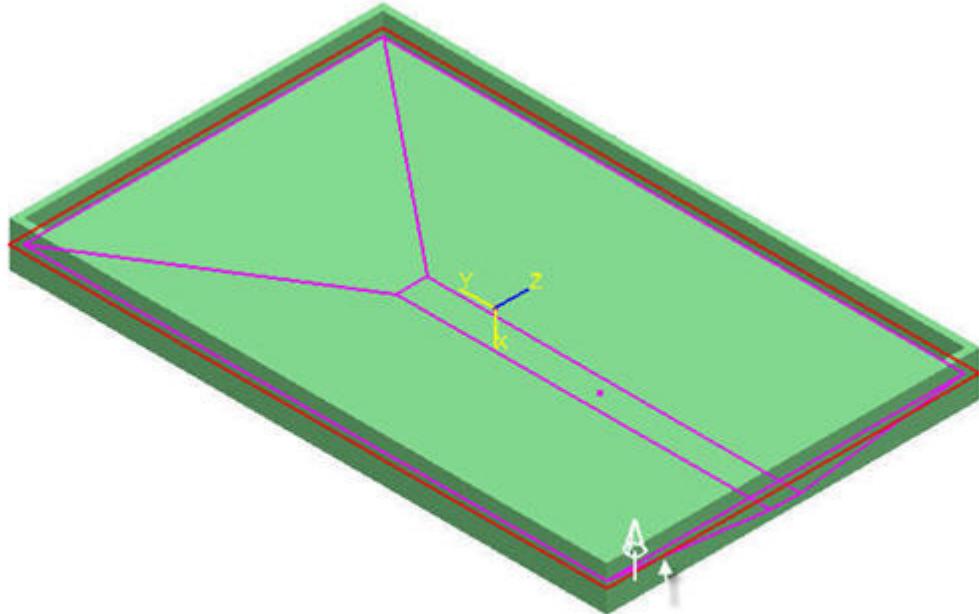


37. Move your mouse over the SlopedSlab and it should highlight *Subtract Concrete* and click the left mouse button once at the location as shown in the figure. Your view should now resemble the following graphic.

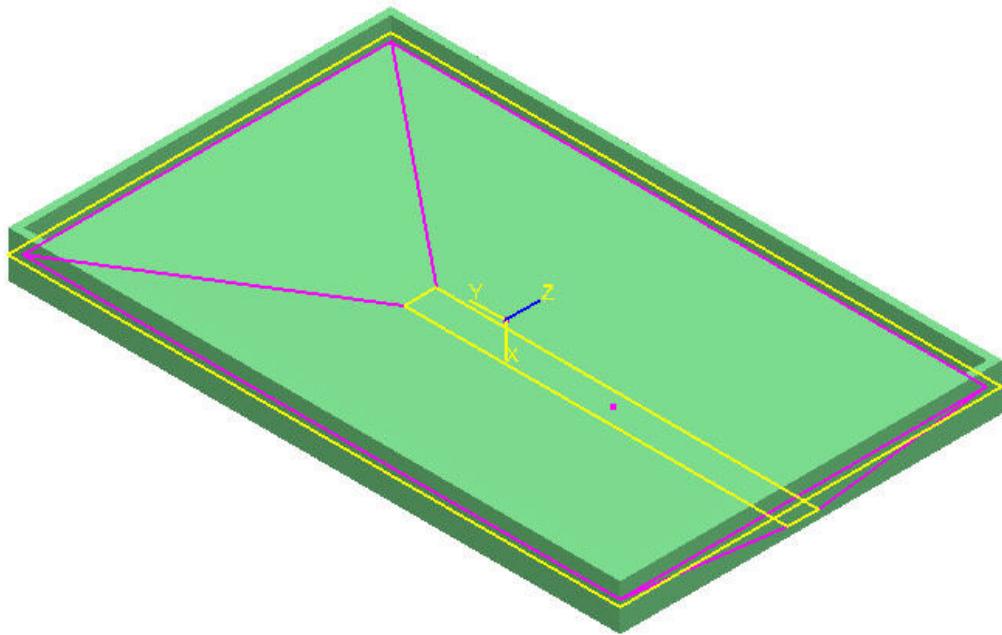


38. Select 2 from the quick pick.

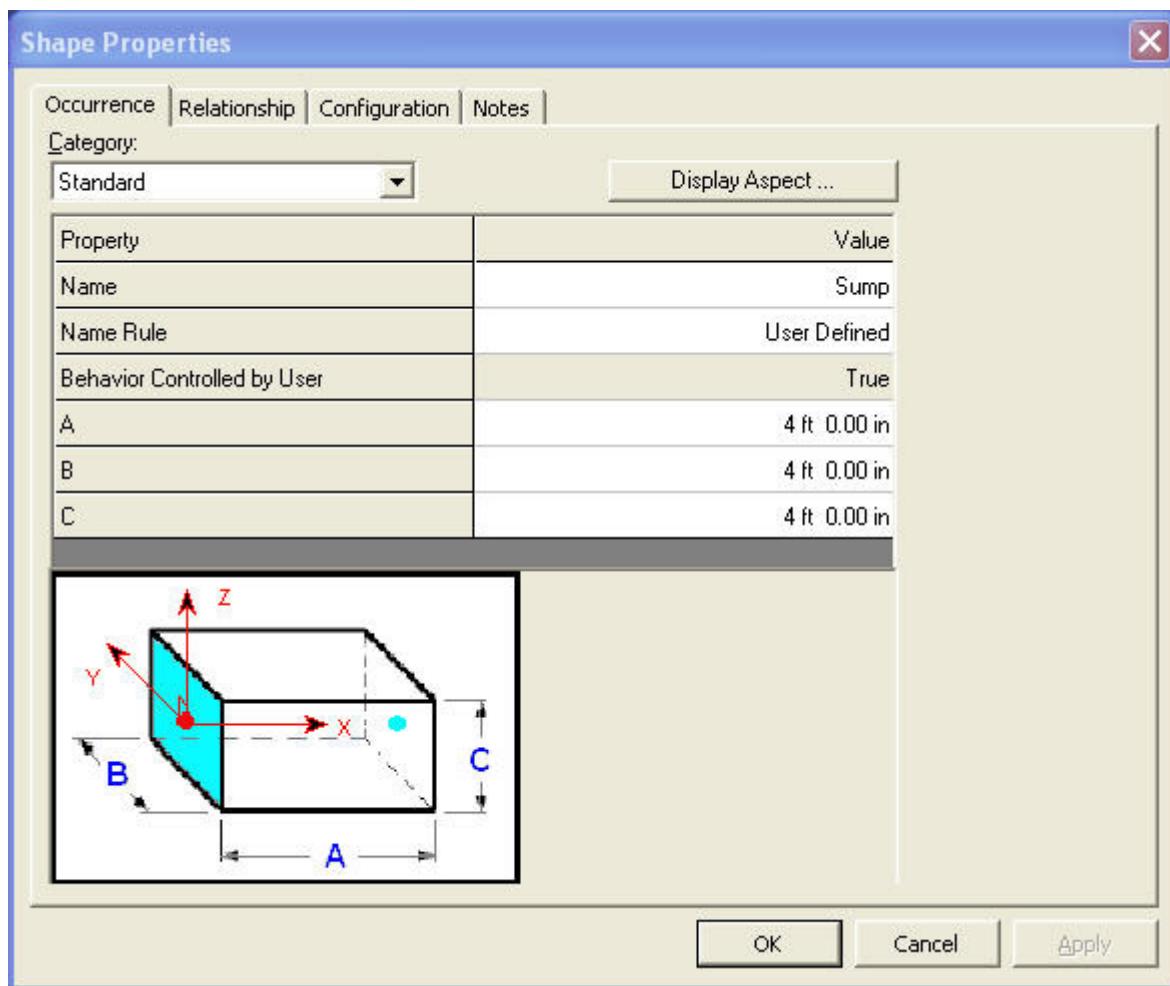
39. Select the *Base Concrete*. Your view should now resemble the following graphic.



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

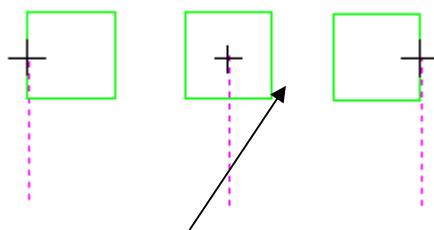


41. Hold down the Place Shape command in the vertical toolbar until the Place Shape dialog appears. Select *RectangularSolid* from the Shapes dialog.
42. Choose *SlopedSlab* as the parent of the Shape.
43. In the Shape Properties dialog, set A to 4 ft, B to 4 ft, and C to 4 ft and enter the name as “*Sump*”. Press OK

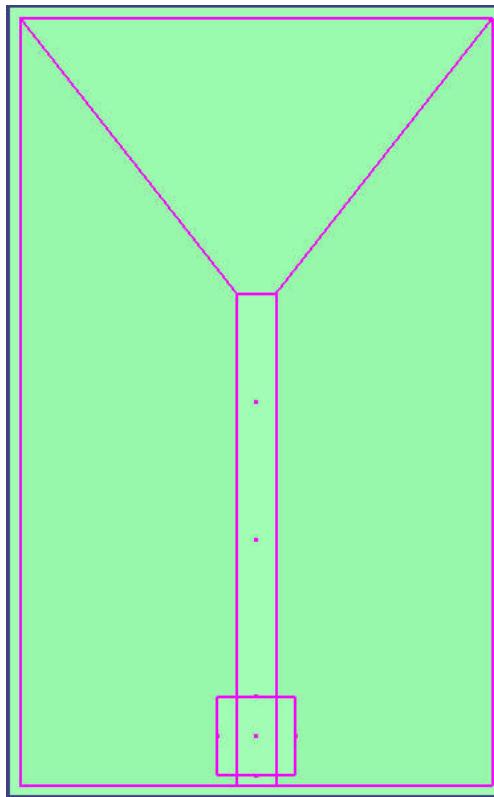


44. While looking in the Plan Plane, use the Up arrow keys (if needed) to change the axis of rotation and use the Left or Right arrow keys to rotate the shape until the point of placement is the center of the rectangle.

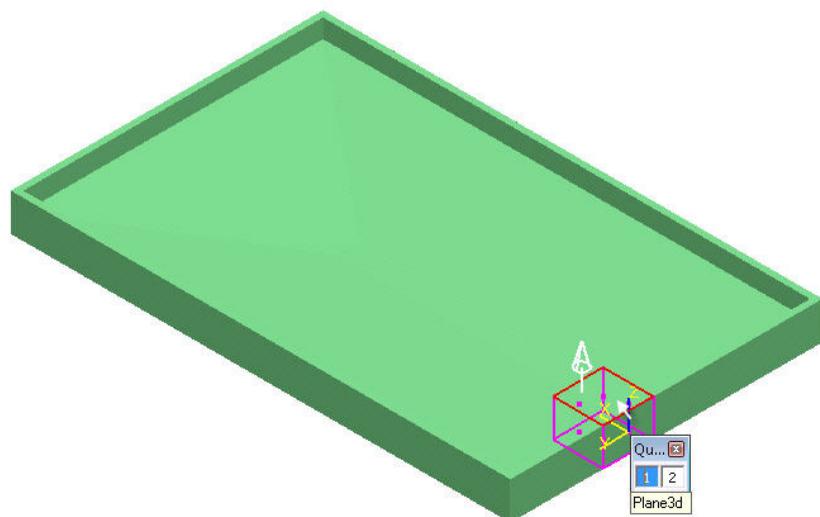
#### a. Looking Plan View



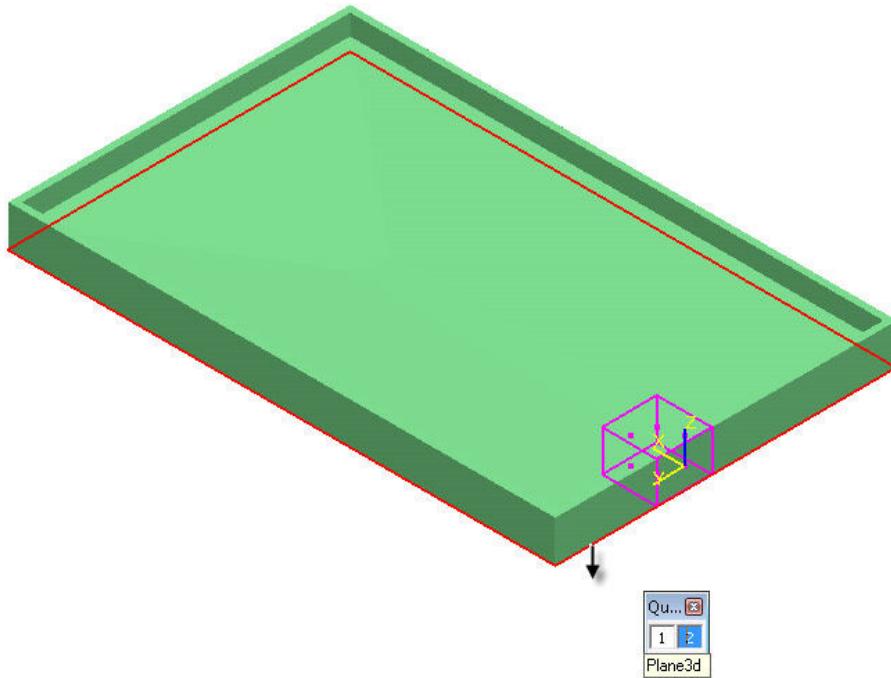
45. Place the Shape at E: -47 ft 6 in, N: -57 ft and EL: 2 ft 2 in.
46. Change back to the Looking Plan view. Create a Connect Relationship at the point shown below and click to place the Shape. Your view should resemble the following graphic.



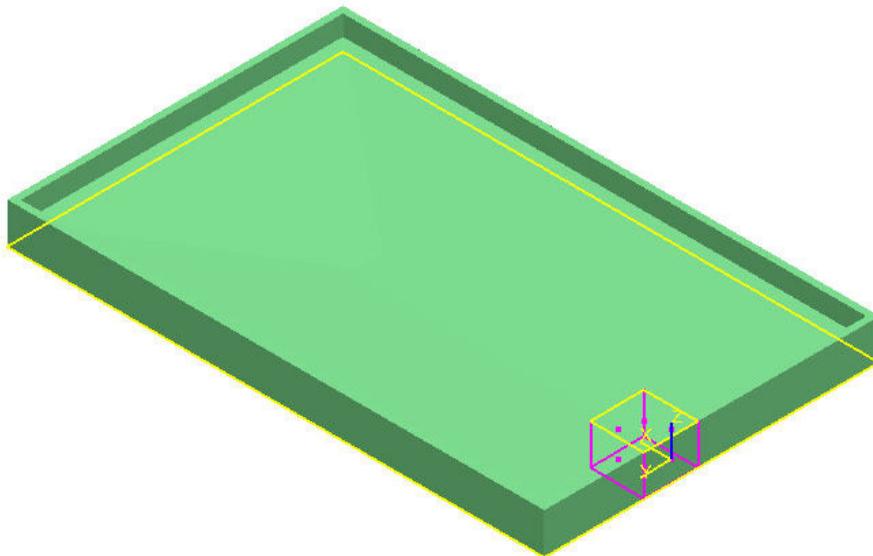
47. Change the view back to isometric. Select the *Sump* and add mate as the new relationship. Select the top face of the *Sump* and your view should resemble the following graphic.



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



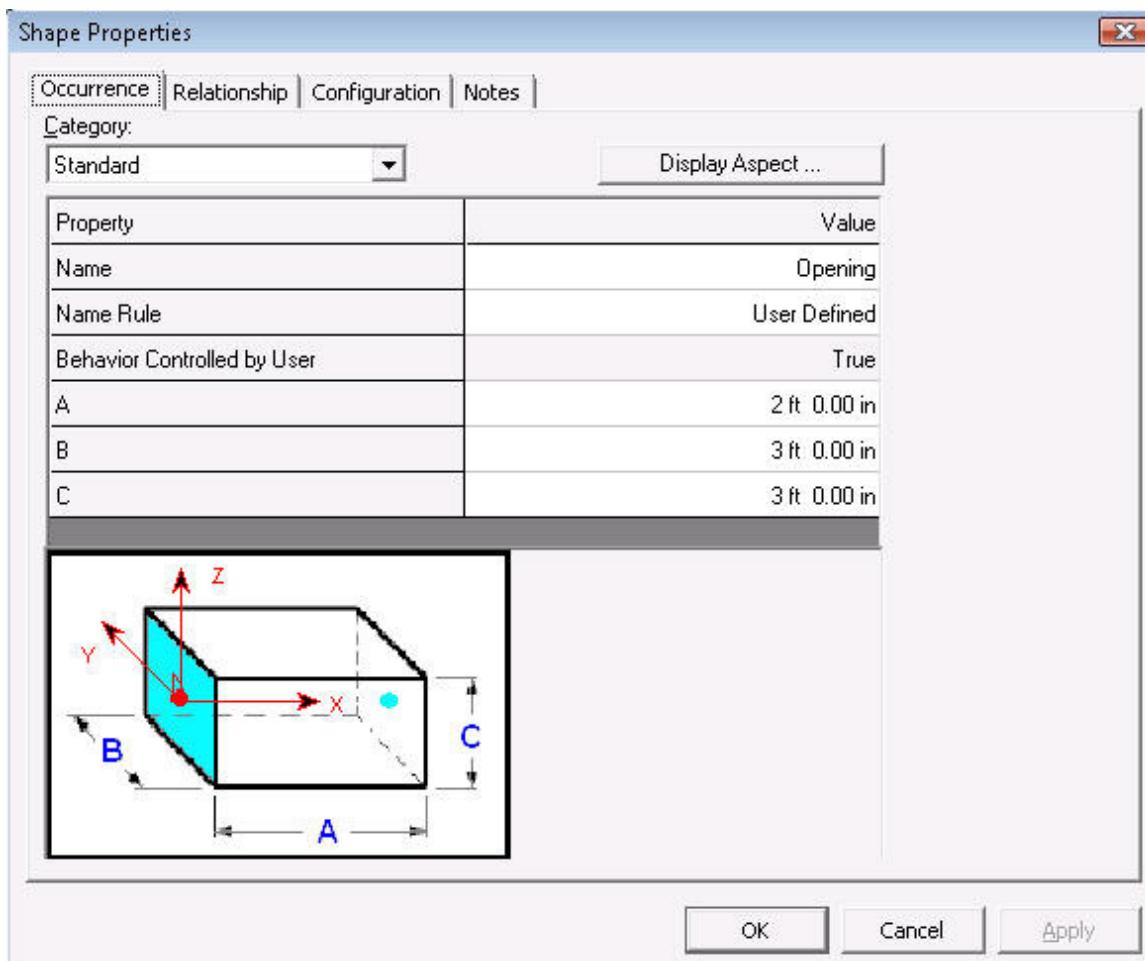
49. Once the mate relationship has been applied, your view should resemble the following graphic.



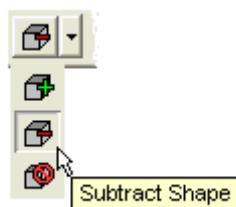
50. Change back to the Plan Plane view and start the Place Shape command again.

51. Select *RectangularSolid* from the Shapes dialog

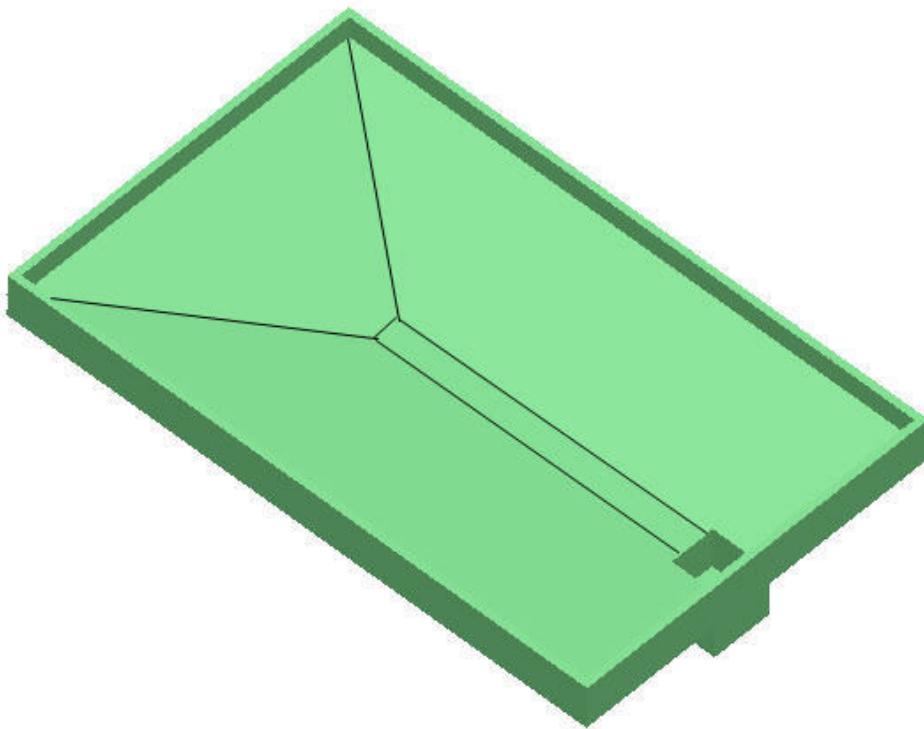
52. Choose *SlopedSlab* as the parent of the Shape and make the following additions as shown in the image below.



53. Press the up arrow key once and then left arrow key once.  
 54. Place the Shape at E: -47 ft 6 in, N: -57 ft and EL: -2 ft 8 in.  
 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.

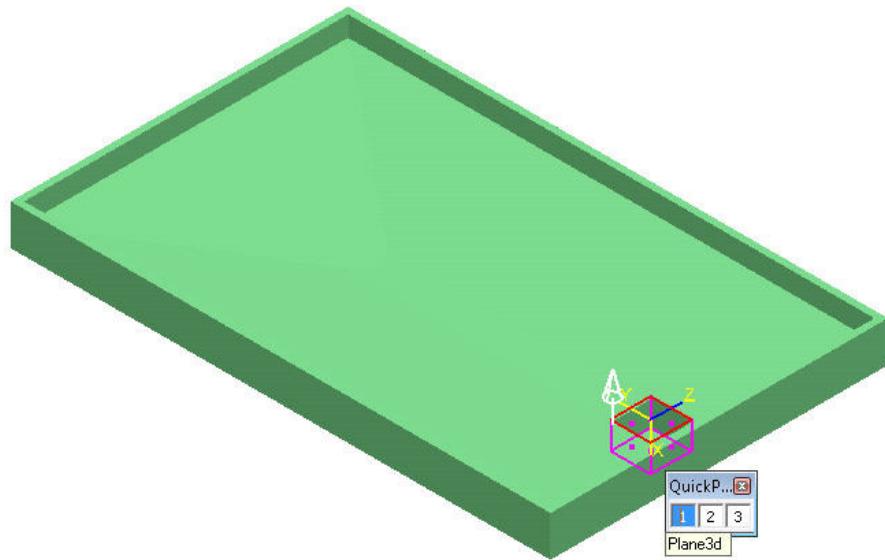


\*\* Lines shown within this image are used only for visual enhancement of the object.

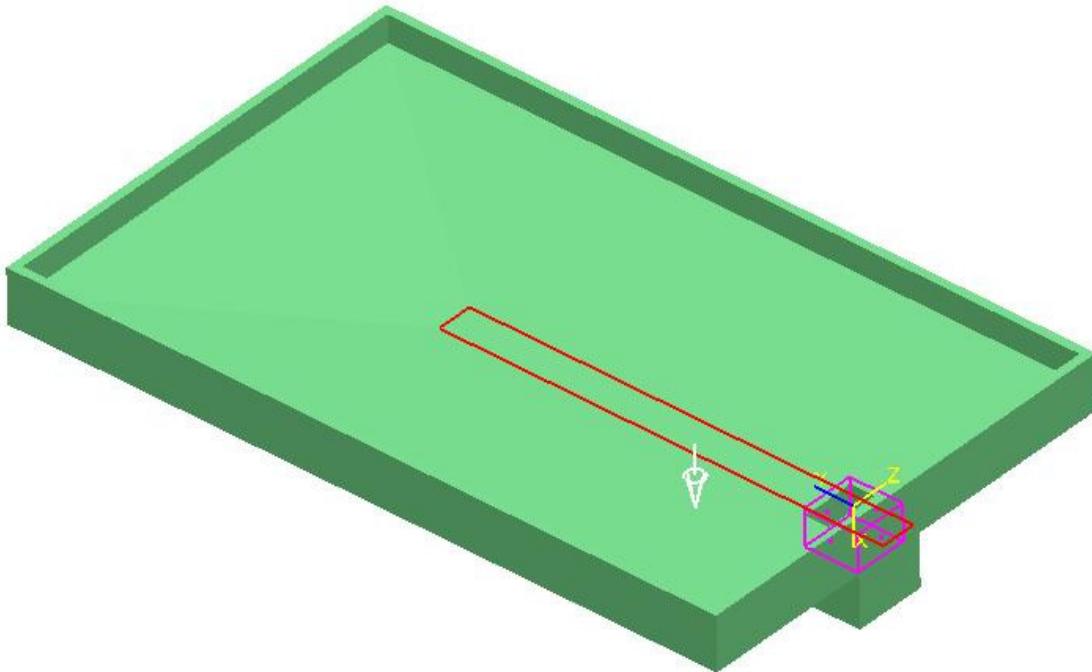
57. Select the opening and in the ribbon bar enter offset value as -0 ft 5 in.



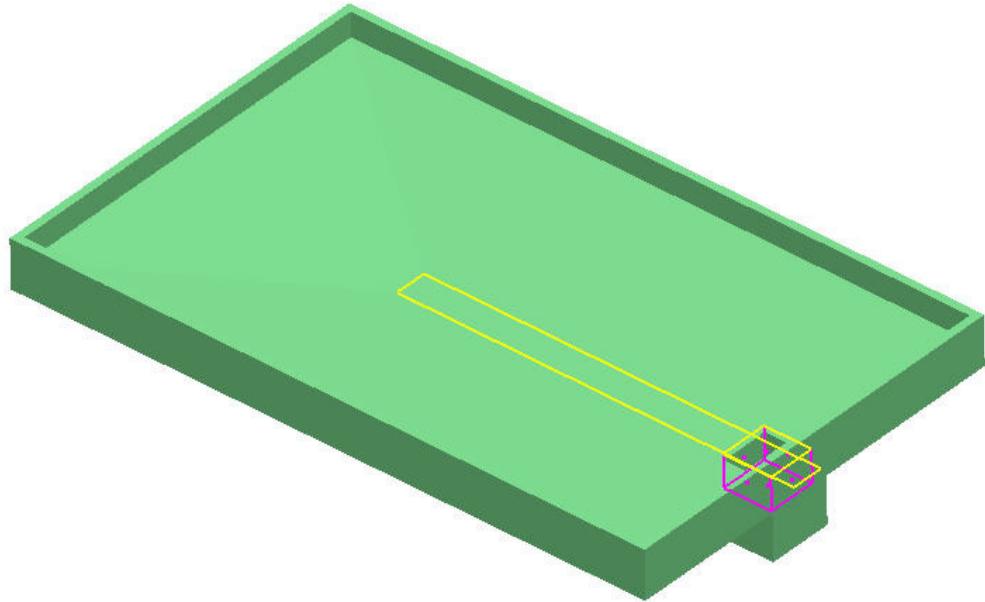
58. Select the top face of the *Opening*, select 2 from the quick pick. Your view should resemble the following graphic.



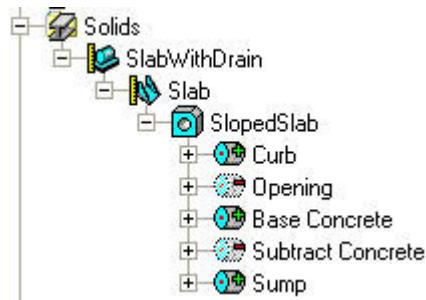
59. Select the surface of the *Subtract Concrete*. Your view should resemble the following graphic.



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

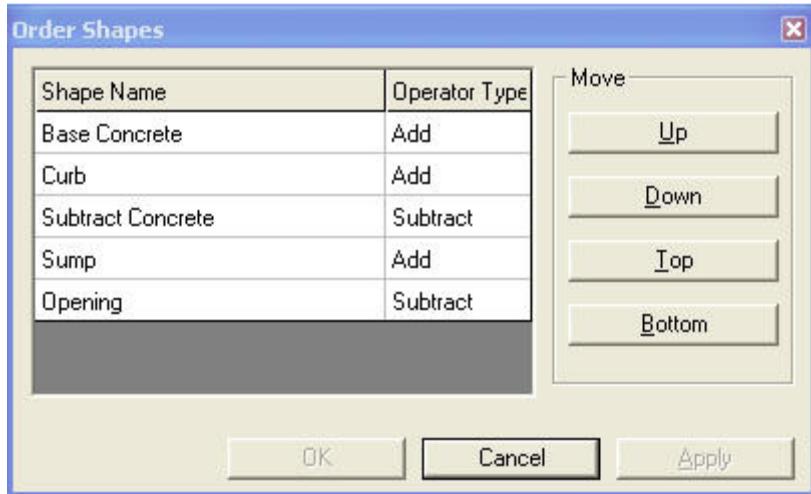


61. The finished hierarchy in Workspace Explorer should look like the following with the exception of the Shape Names.

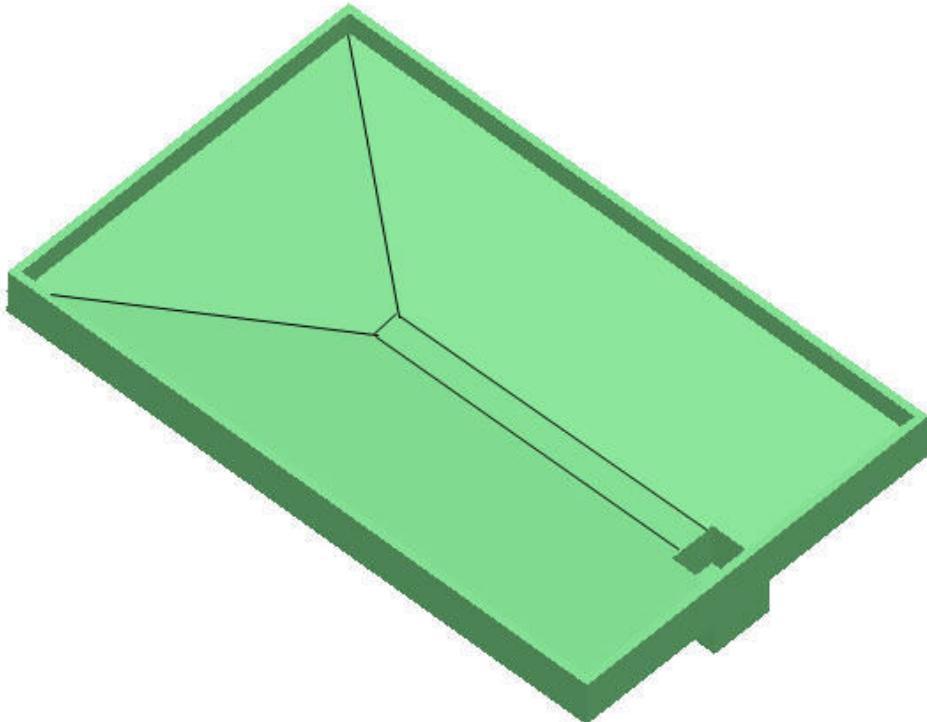


#### Order Shapes

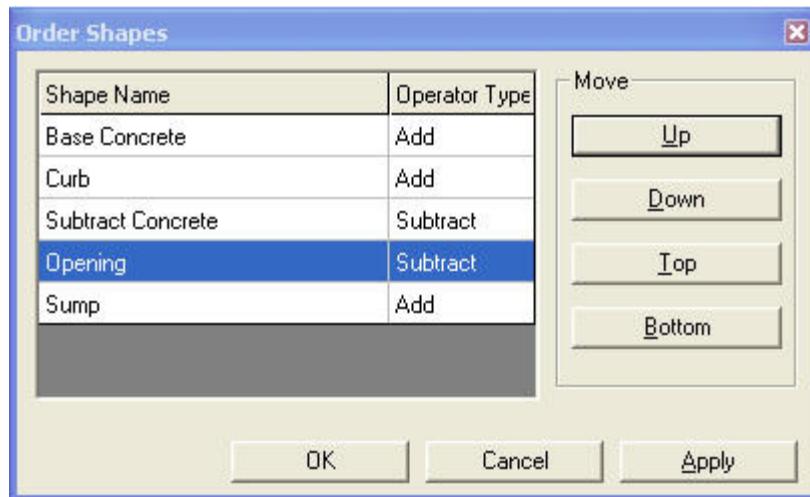
62. Order Shapes controls the order in which the designed solid's shapes are processed by the software, which can be very important when a shape that cuts material from the designed solid overlaps a shape that adds material to the designed solid. The designed solid could look very different depending 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.

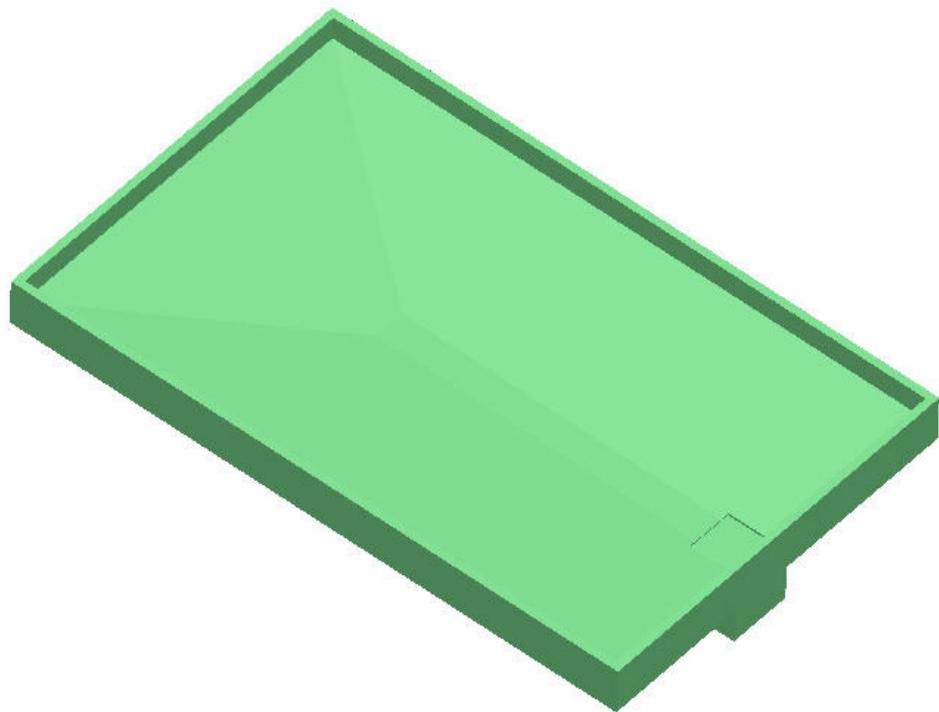


65. Select the Opening in the dialog box and hit the “Up”. Your view should resemble the following graphic.

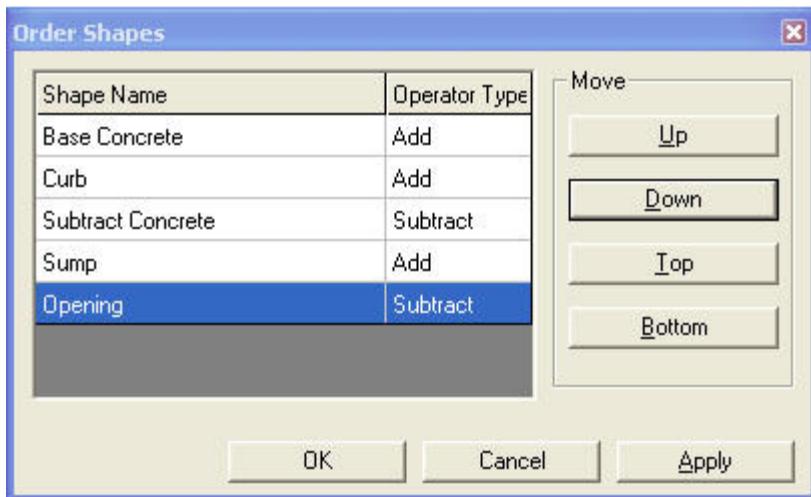


66. Hit Apply. Hit Cancel.

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*..

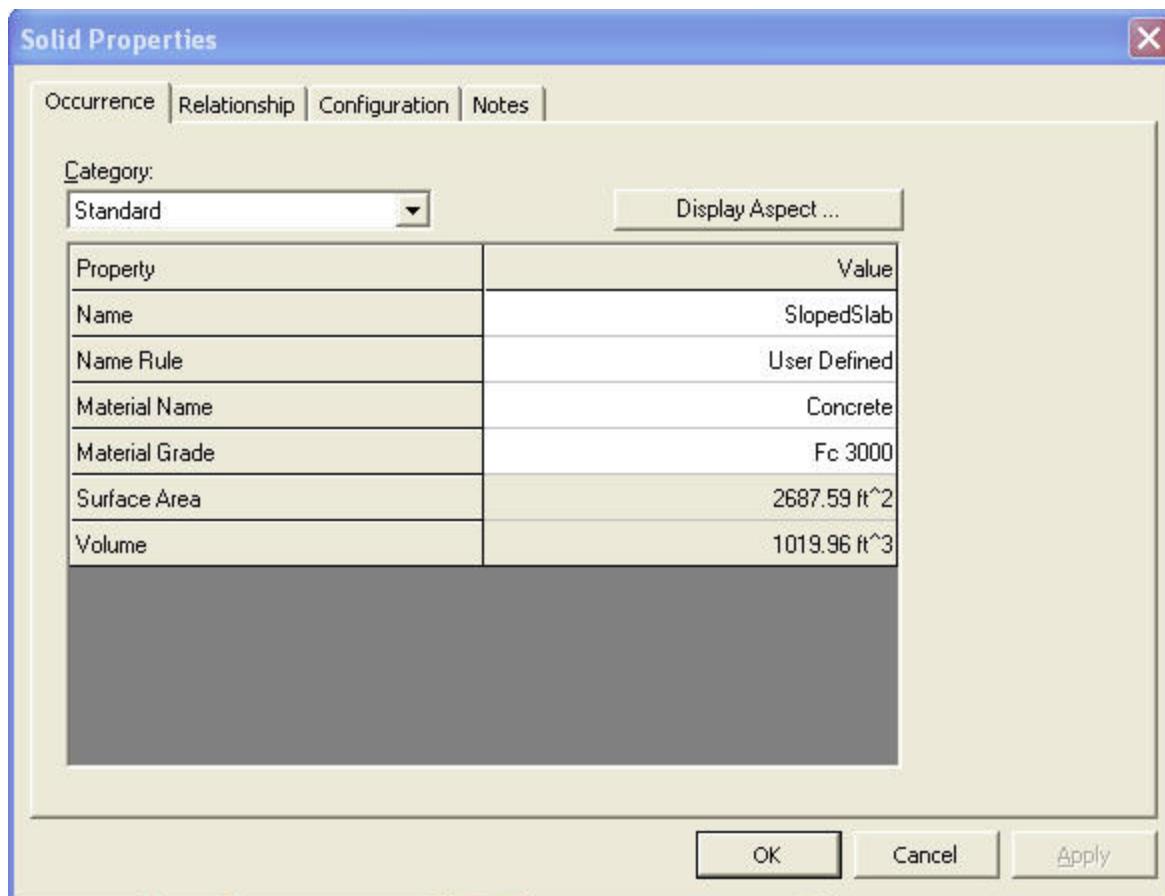
71. *SlopedSlab* with the *Sump* and without the *Sump* weight can be computed using suppress option.

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	147894.05 lbm
DryCGX	-47 ft 6.00 in
DryCGY	-39 ft 5.55 in
DryCGZ	-0 ft 2.18 in

74. Change the Category dropdown to Standard.

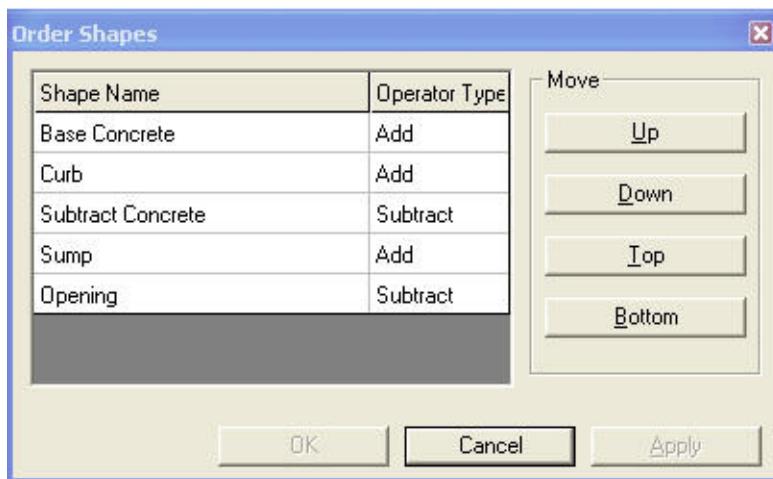


### Suppress

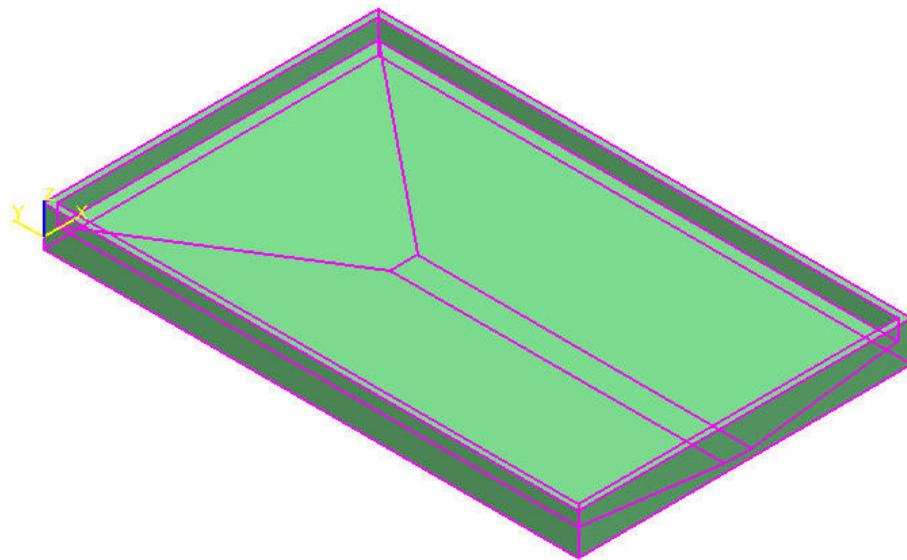
75. Select *SlopedSlab* in Workspace Explorer.

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

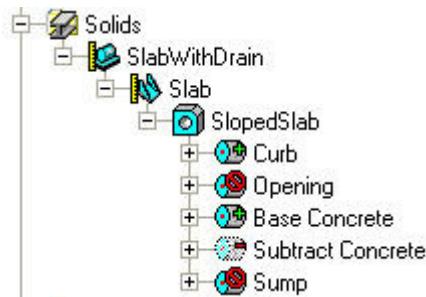
77. The Order Shapes dialog will appear. Notice that the order of the Shapes in the dialog is the same as the order in which the Shapes were placed.



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



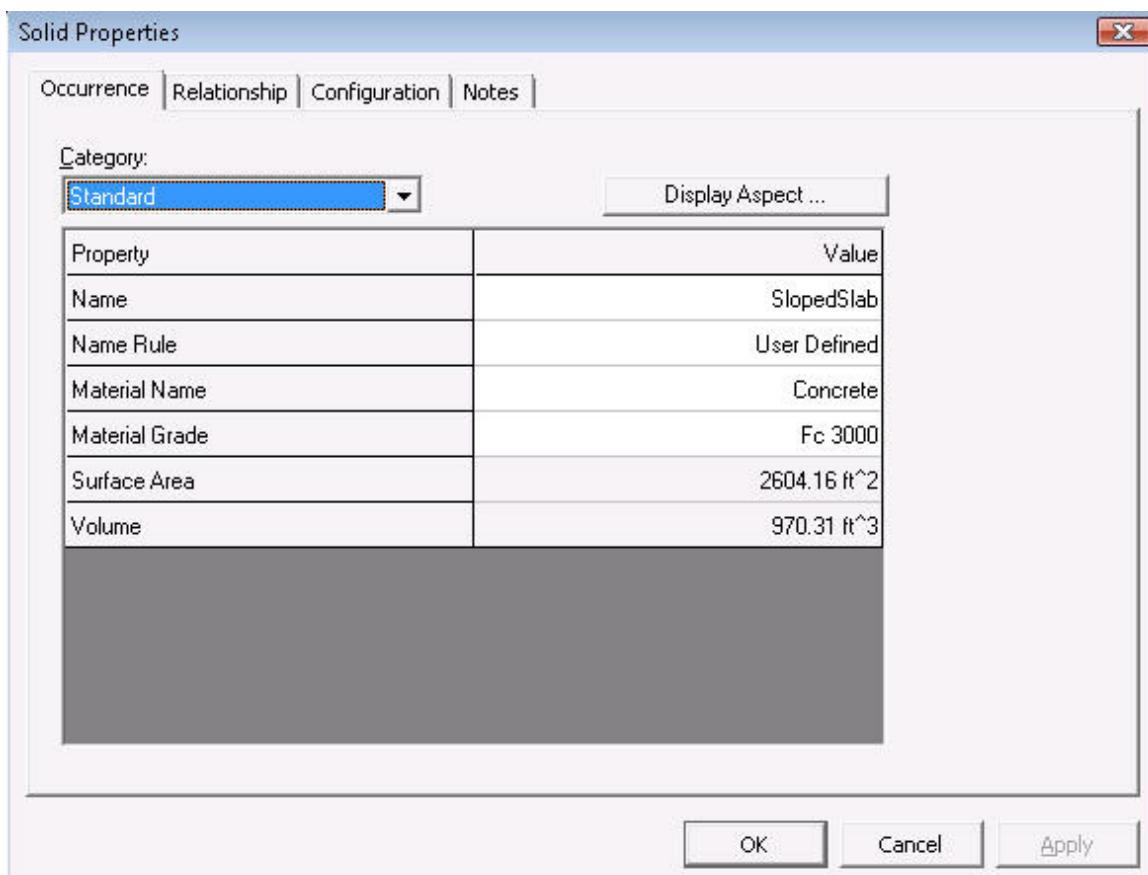
79. Workspace hierarchy should resemble the following graphic.



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	140694.31 lbm
Dry CGX	-47 ft 6.00 in
Dry CGY	-38 ft 6.74 in
Dry CGZ	-0 ft 0.37 in

82. Change the Category dropdown to Standard.

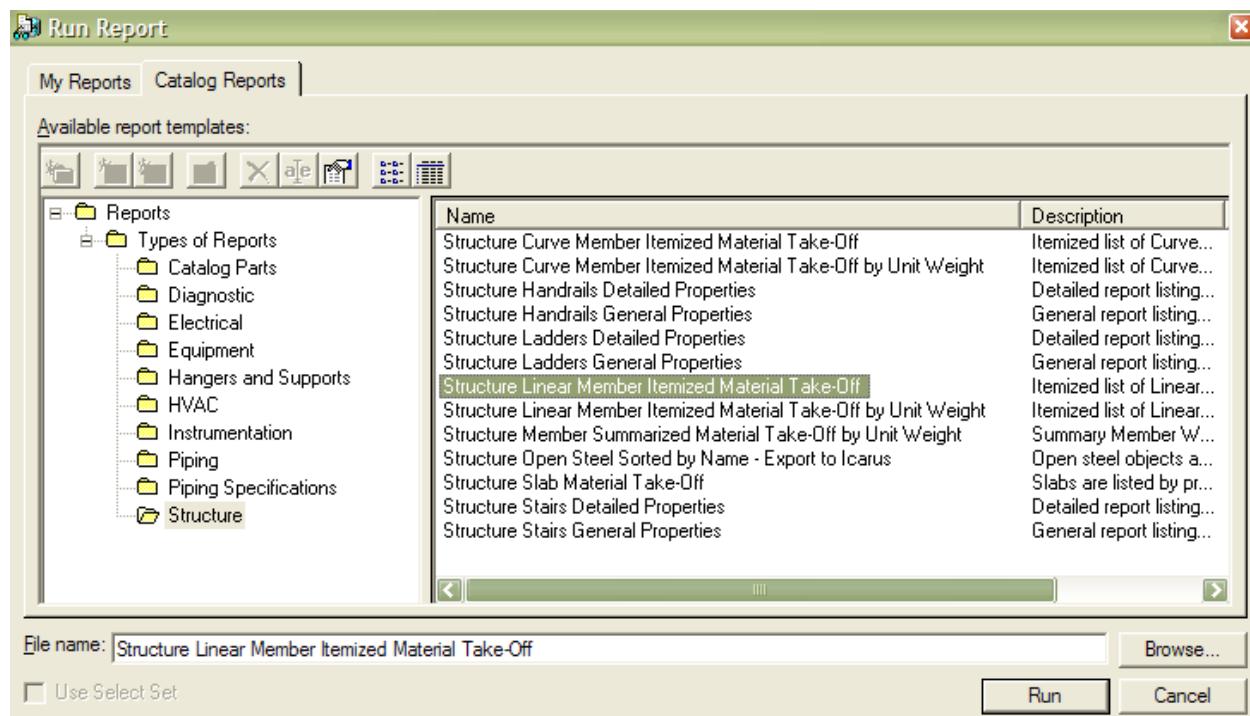


# LAB-14: Reports

Extract a Linear Member material list from the model.

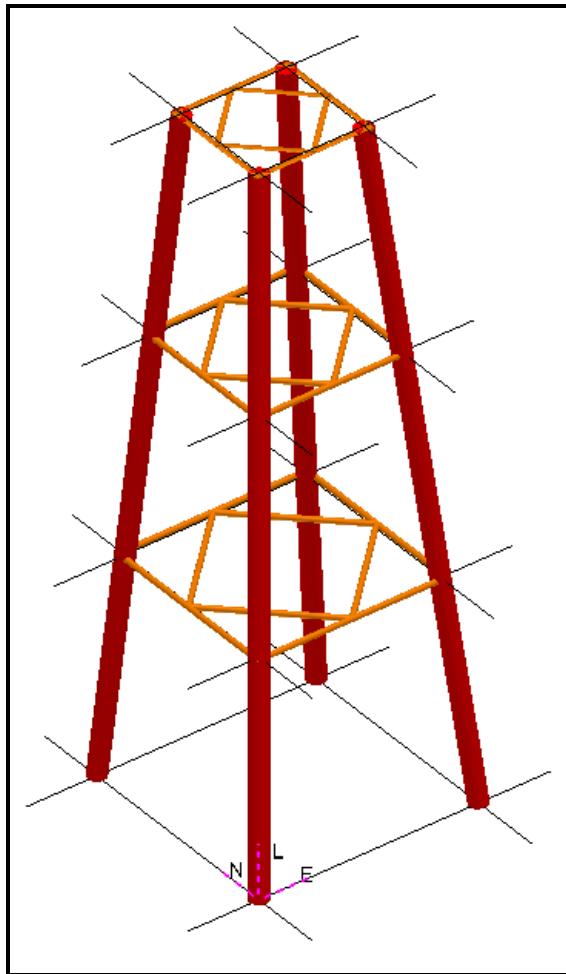
Itemized Linear Member Material Take-Off											
	Item	Name	Priority	Type Category	Type	Section	Material Type	Material Grade	Length ft	Weight lbm	Surface Area ft <sup>2</sup>
1	Beam-1-0065		Undefined	Beam	Beam	W18X35	Steel - Carbon	A36	40.50	1382.21	196.09
2	Column-1-0051		Primary	Column	Column	W14X53	Steel - Carbon	A36	34.00	1769.41	164.05
3	Column-1-0061		Primary	Column	Column	W14X53	Steel - Carbon	A36	34.00	1769.41	164.05
4	Column-1-0052		Primary	Column	Column	W14X82	Steel - Carbon	A36	34.00	2741.02	188.70
5	Column-1-0042		Primary	Column	Column	W14X74	Steel - Carbon	A36	34.00	2492.12	187.85
6	Column-1-0048		Primary	Column	Column	W14X82	Steel - Carbon	A36	34.00	2741.02	188.70
7	Column-1-0053		Primary	Column	Column	W14X53	Steel - Carbon	A36	34.00	1769.41	164.05
8	Column-1-0040		Primary	Column	Column	W14X74	Steel - Carbon	A36	34.00	2492.12	187.85
9	Column-1-0055		Primary	Column	Column	W14X53	Steel - Carbon	A36	34.00	1769.41	164.05
10	Column-1-0056		Primary	Column	Column	W14X99	Steel - Carbon	A36	34.00	3344.31	237.43
11	Vertical Brace-1-0004		Undefined	Brace	Vertical Brace	L8XX4X12	Steel - Carbon	A36	29.15	570.44	57.82
12	MemberPartPrismatic-1-0022		Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	29.00	740.08	76.36
13	MemberPartPrismatic-1-0018		Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	29.00	740.08	76.36
14	Vertical Brace-1-0003		Undefined	Brace	Vertical Brace	L8XX4X12	Steel - Carbon	A36	28.81	563.72	57.14
15	MemberPartPrismatic-1-0019		Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	28.29	721.92	74.49
16	Column-1-0079		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
17	Column-1-0080		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
18	Column-1-0081		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
19	Column-1-0074		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
20	Column-1-0083		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
21	Column-1-0075		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
22	Column-1-0063		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
23	Column-1-0077		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
24	Column-1-0072		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
25	Column-1-0082		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
26	Column-1-0085		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
27	Column-1-0076		Undefined	Column	Column	W14X53	Steel - Carbon	A36	27.83	1448.49	134.30
28	MemberPartPrismatic-1-0023		Primary	Brace	Vertical Brace	2L4X4X12X3/8	Steel - Carbon	A36	26.31	671.38	69.28
29	Column-1-0032		Undefined	Column	Column	W14X53	Steel - Carbon	A36	26.00	1353.08	125.45
30	Column-1-0028		Undefined	Column	Column	W14X53	Steel - Carbon	A36	26.00	1353.08	125.45
31	Column-1-0030		Undefined	Column	Column	W14X53	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.

# LAB-15: Structural Modeling (offshore Jacket) - Optional



## Part I: Jacket Pipes

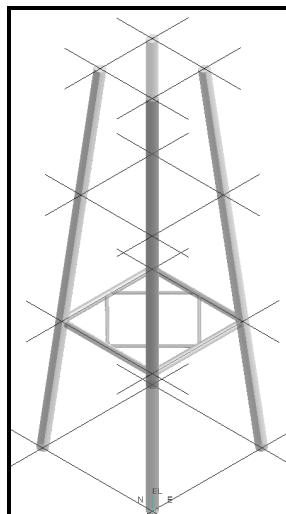
- 1 Open or create a session file and define an appropriate filter for your workspace.
- 2 Go to the Structure Task environment. Make sure the Active Permission Group is set to *Structural*.
- 3 Select Place Linear Member System Command. Use the views shown in Figure to place the appropriate support columns and beams.
- 4 Set the active member parameters as follows:

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

5. Place the first point for column end at the grid intersections Elevation 0'- 0".
6. Place the second column end at the grid intersections Elevation 30'- 0".
7. Repeat the above steps to place the other support columns. Toggle the start/end icons to set the discrete/contiguous placement method.
8. Place the Beams at elevation 10'- 0".
9. Set the active member parameters as follows:

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

Your View should now resemble the following graphic:

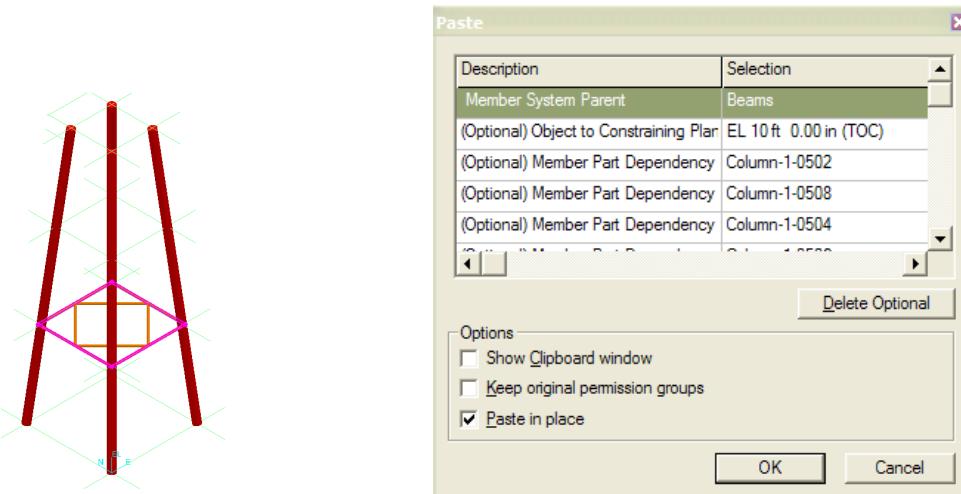


10. Select the Frame connections of the intermediate beams and set the Position Rule to Ratio.
11. Use Copy/Paste functionality to place the other steel frames on the other EL 20' and 30'.

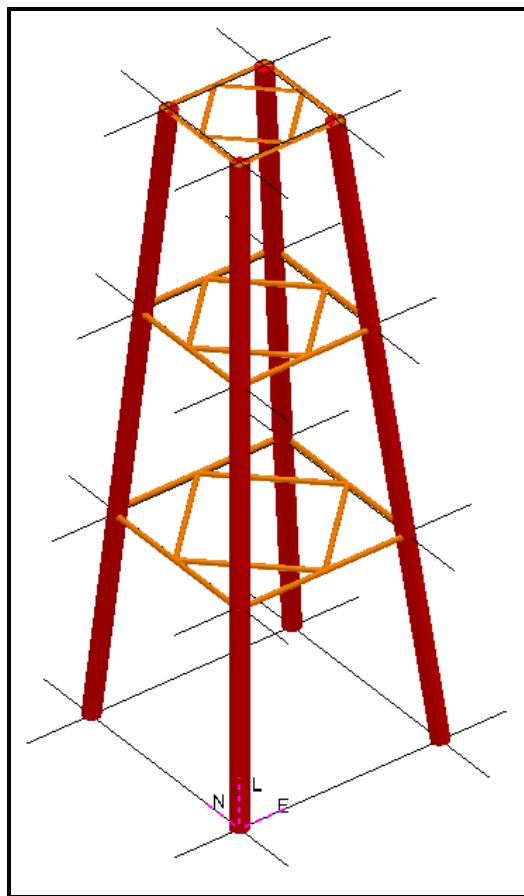
*Note: Make sure to select the appropriate objects to re-establish the connection in the Paste Dialog box.*

*Hint: Set the select filter to Member Systems to select 8 members.  
Only 1 related object is needed to re-establish the connection (EL plane).*

12. Use the Paste command three times.

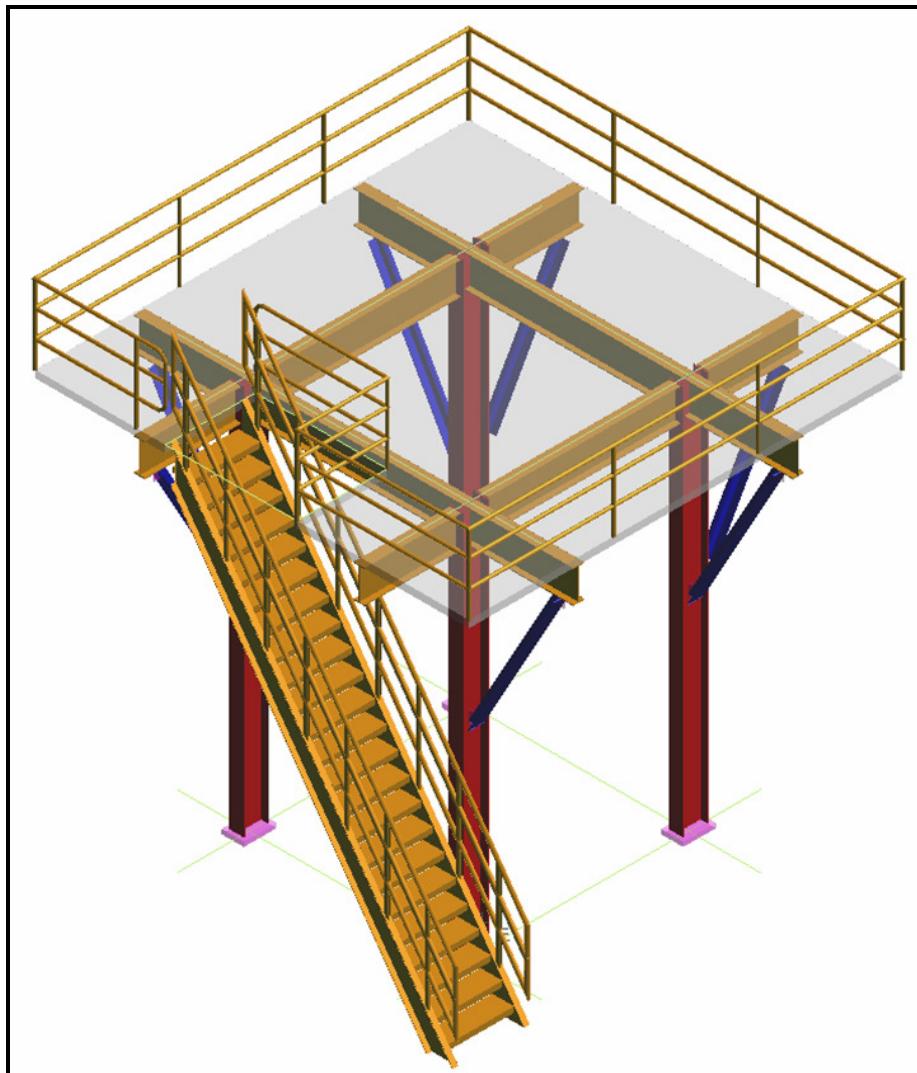


Your View should now resemble the following graphic:



**Part II: Rotated Coordinate System – Optional**

- Use the commands you have learned to model the following:
- Use Copy/Paste functionality to make a copy of the U04 Coordinate system and place it at Easting 120'-0", Northing 150'-0" Elevation 0'-0".



Iso View - U04 - B Structure

**Coordinate System:** Copy (U04 CS)

**Columns:**

Connection:	By Rule
Type Category:	Column
Type:	Column
Section Name:	W14x53
Cardinal point:	5
Angle:	20
System:	A2 -> U04 -> Structural -> Columns

**Beams:**

Connection:	By Rule
Type Category:	Beam
Type:	Beam
Section Name:	W18X40
Cardinal point:	8
Angle:	0
System:	A2 -> U04 -> Structural -> Beams

Beam Extension Length: 5' (TYP)

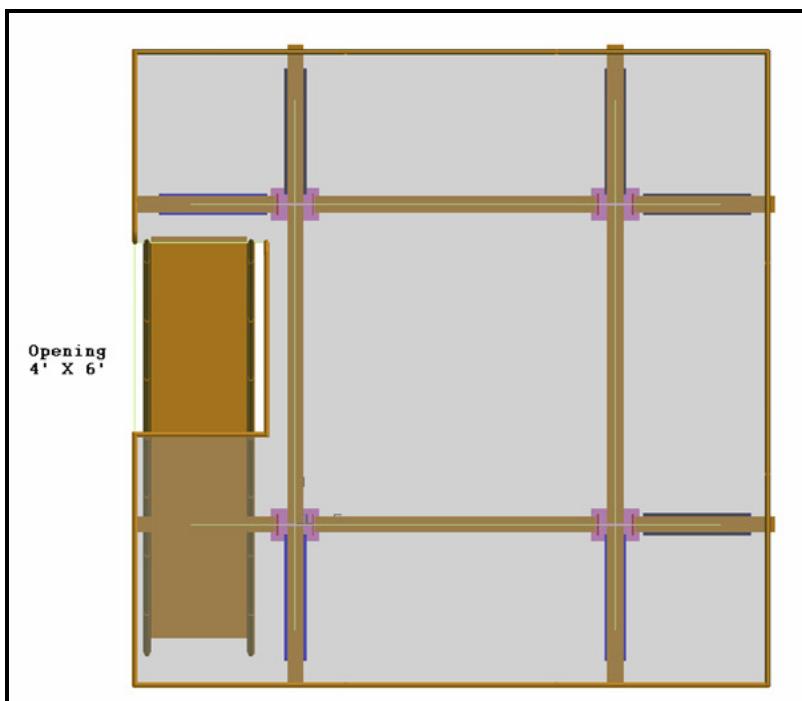
**Vertical Braces:**

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

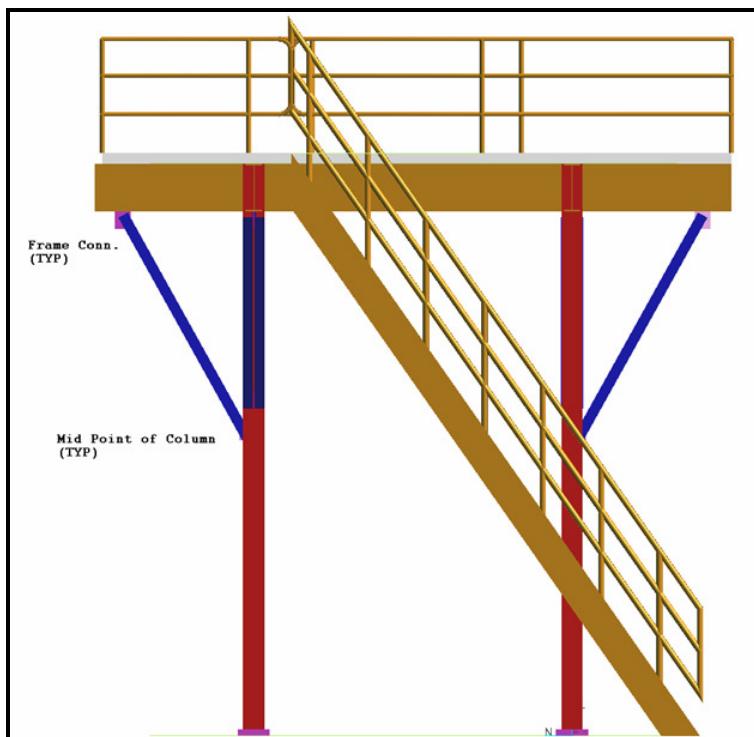
**Stair:**

Type:	StairA1
Width:	3'
Angle:	55

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



Top View - U04 - B Structure



West View - U04 - B Structure

# LAB-16: Using Building Wizard (Optional)

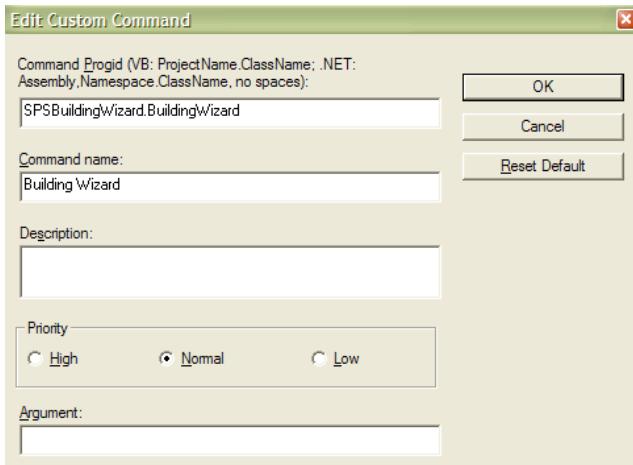
## Objective:

- Create Building\_2 at -500,10, 0 using the building wizard

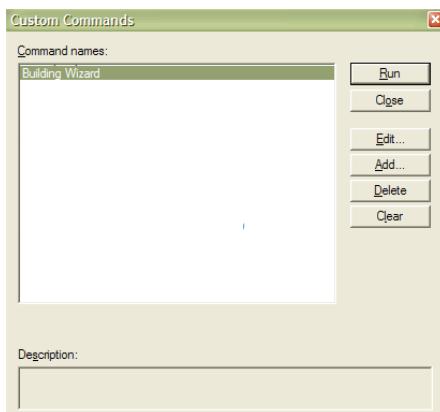
### Add custom command

Add a custom command for the building wizard as follows:

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



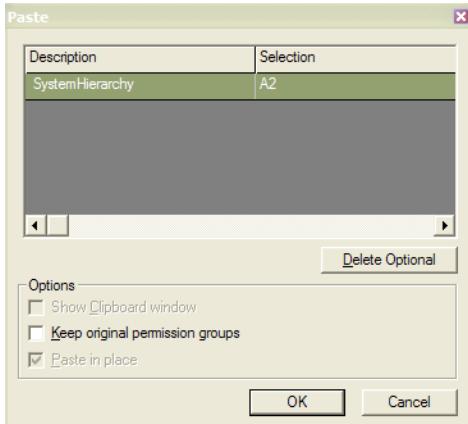
4. Click “OK” Button.



5. Click “Close” Button.

Create the systems required

1. Switch to Systems and Specifications 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.



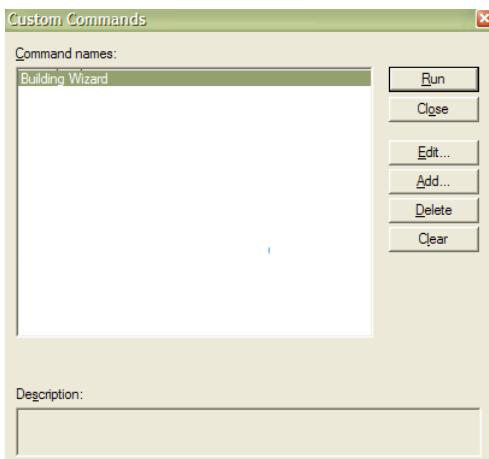
7. Select A2 system in the tree view. Select “Copy of U05” in the right window.

Name	Type
Copy of U05	Unit System
U01	Unit System
U02	Unit System
U03	Unit System
U04	Unit System
U05	Unit System

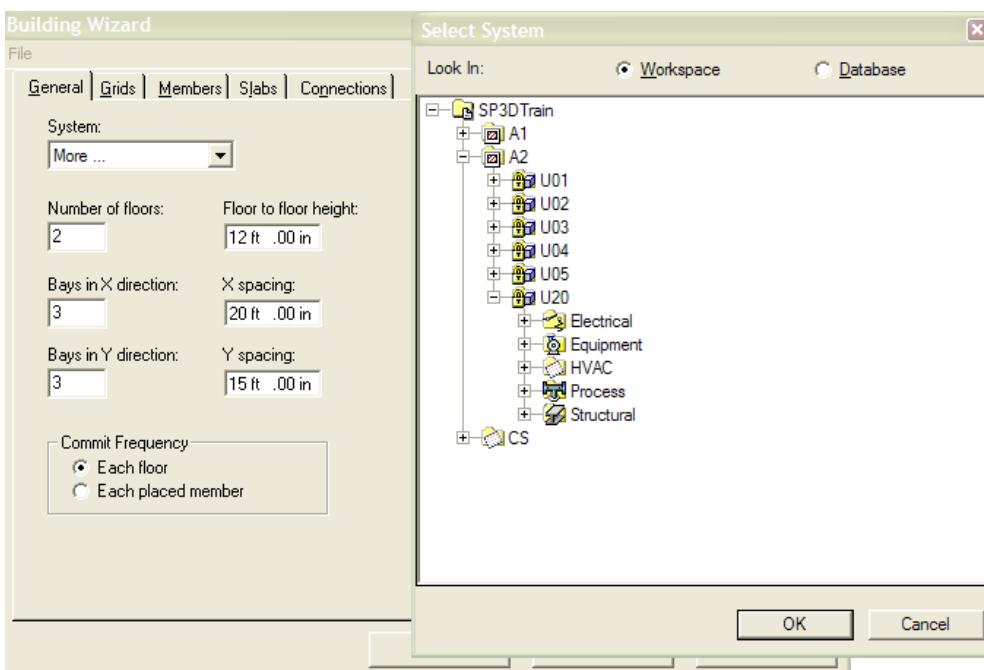
8. Select Copy of U05 and rename it to U20.

Run the building wizard

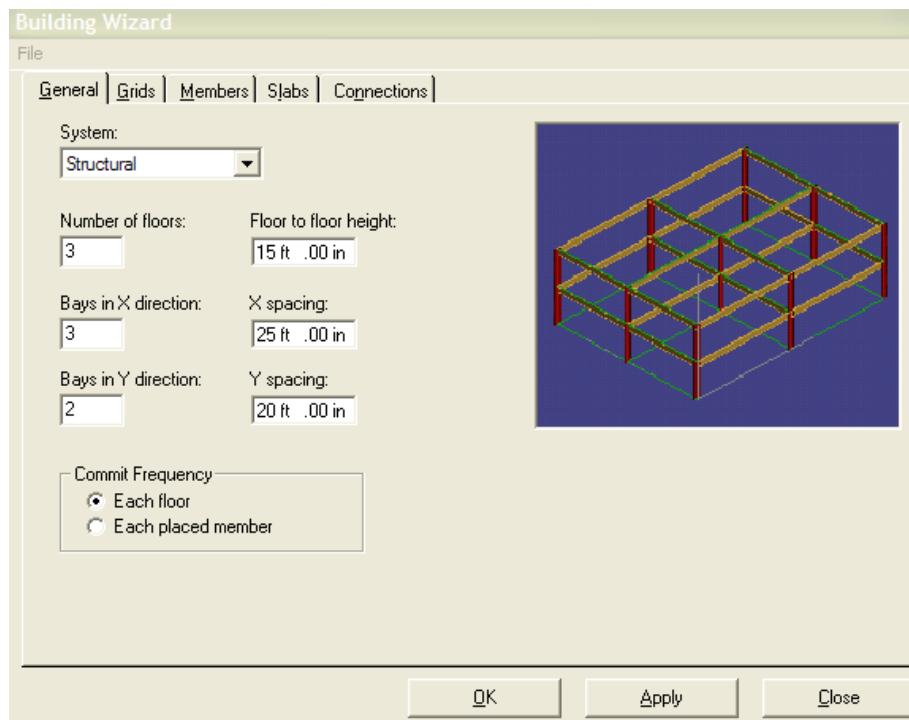
1. Switch to the Structure Task.
2. Select Tools ->Custom Commands.



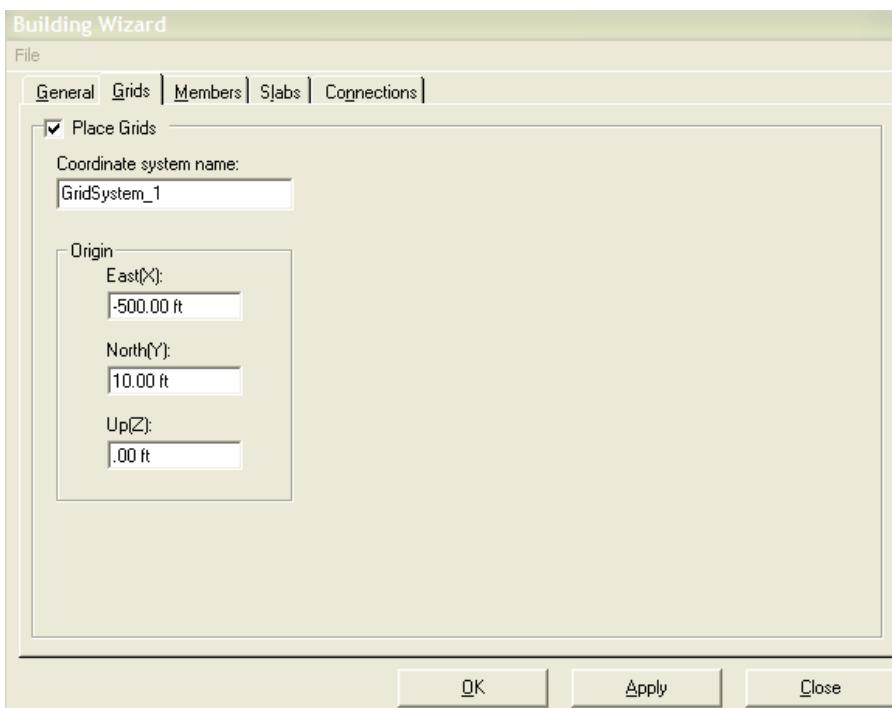
3. Select the Building Wizard.
4. Click “Run” button.  
The building wizard opens up.
5. In the System field, select More..., then pick the A2 -> U20 -> Structural



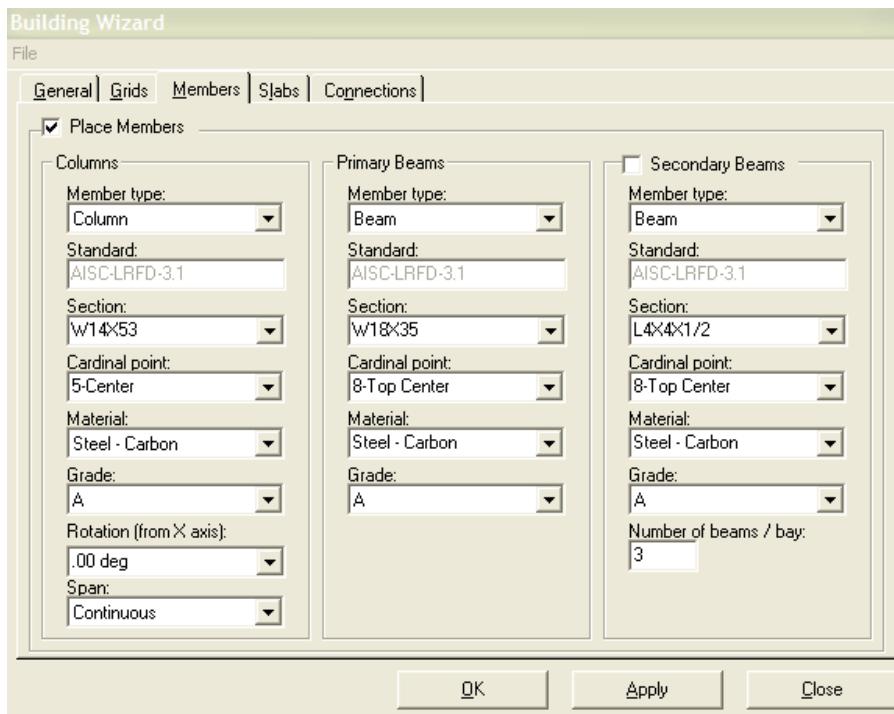
6. On the General Tab, key-ins the following:



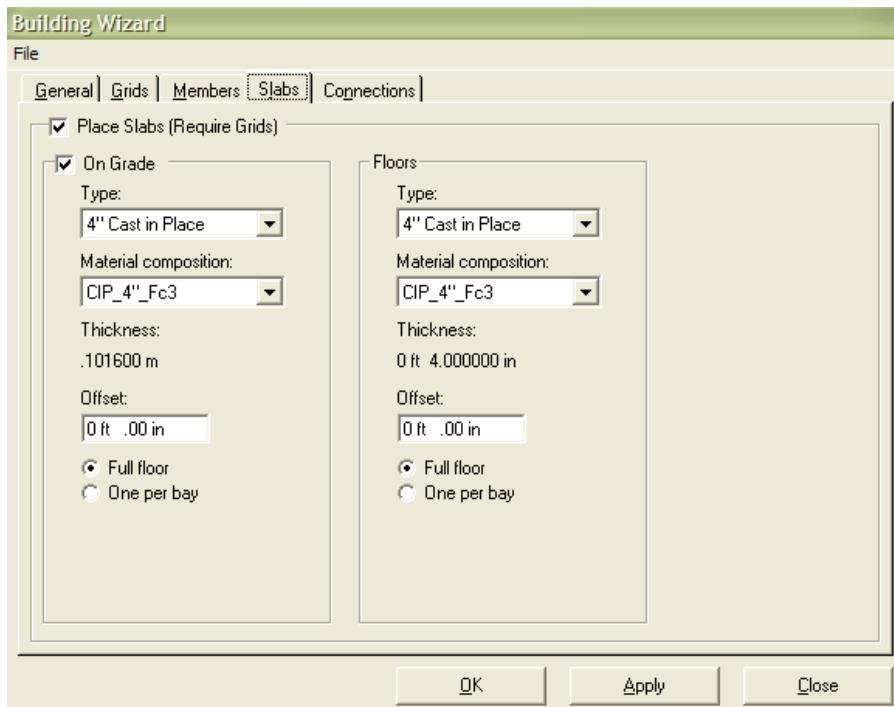
7. On the Grids tab, key-ins the followings:



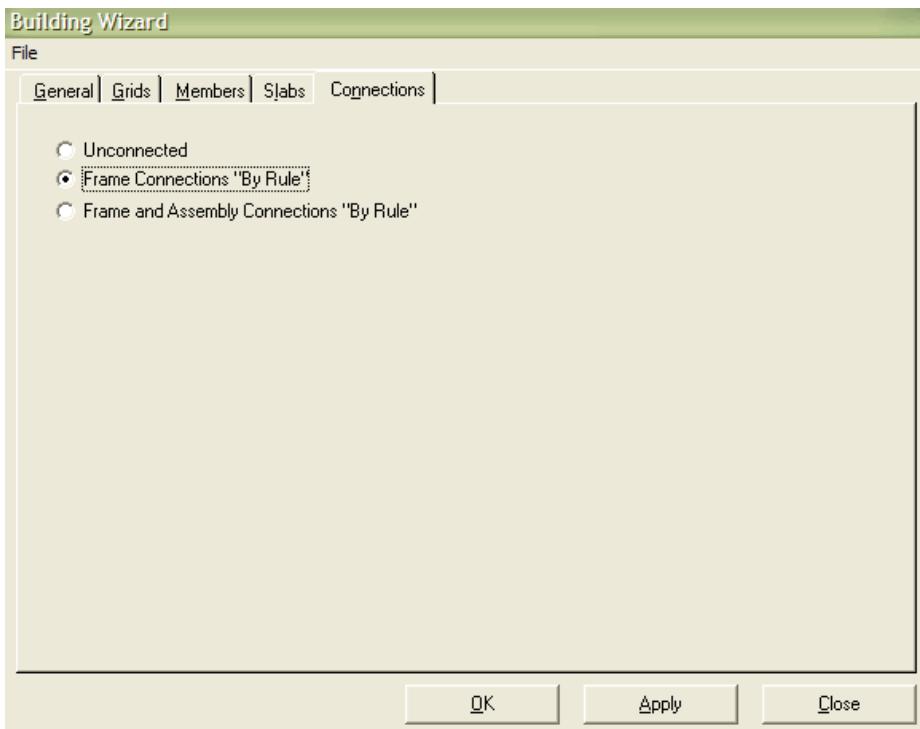
8. On the Members tab, key-ins the followings:



9. On the Slabs tab. Keep the defaults.

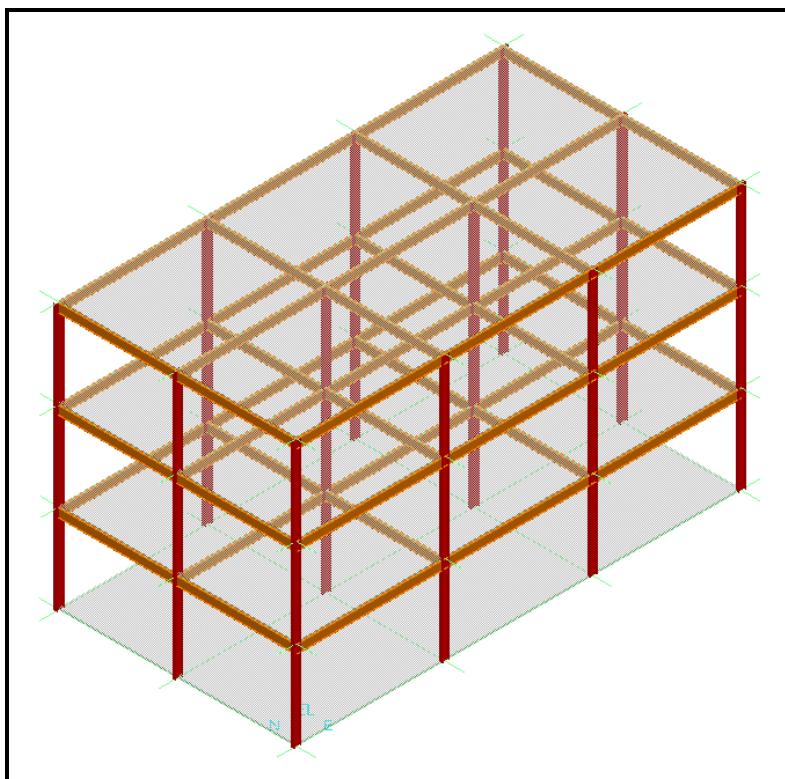


10. On the Connections tab, select 'Frame connections by rule'.



11. Click “OK” Button.

Your View should now resemble the following graphic:



# LAB-17: Data Exchange for Solids (Optional)

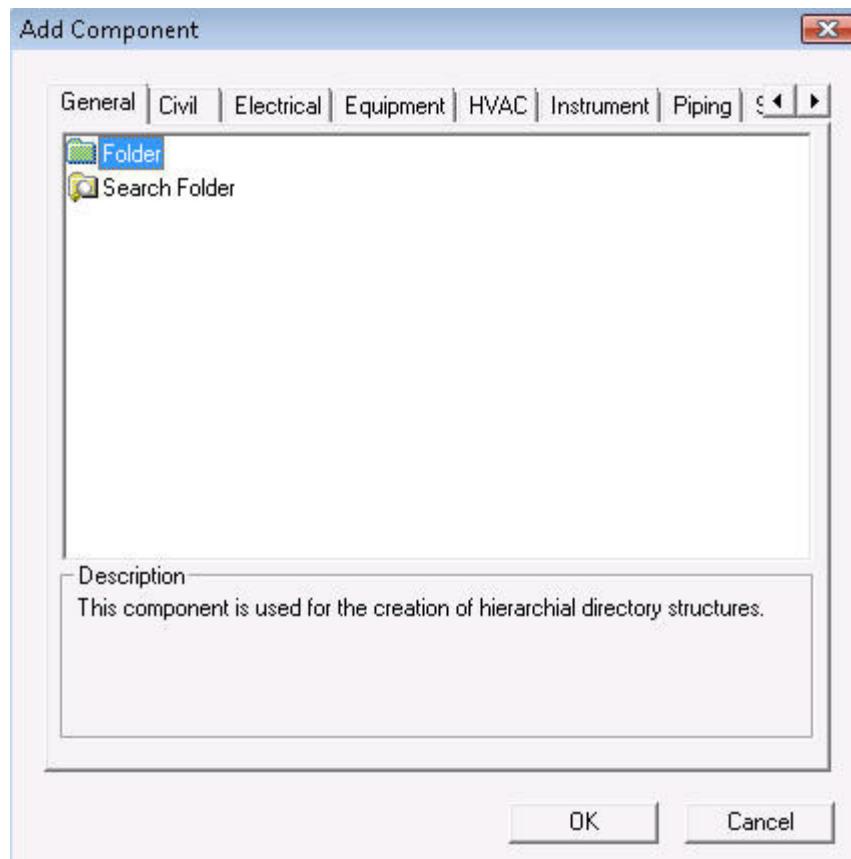
## Objective

After completing this lab you will be able to

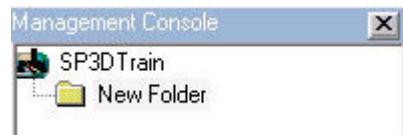
- Export solids as a .SAT file.
- Import solids as a .SAT file.

### Part A: Export

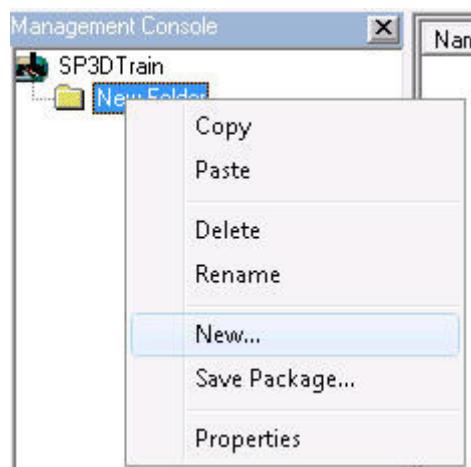
1. Open Solids.ses that was previously saved on the desktop.
2. Go to Tasks → Drawings and Reports.
3. In the Management Console window, right click on SP3D train and select New.
4. Add Component window will appear. In the General tab Select Folder. Your view should resemble the following graphic.



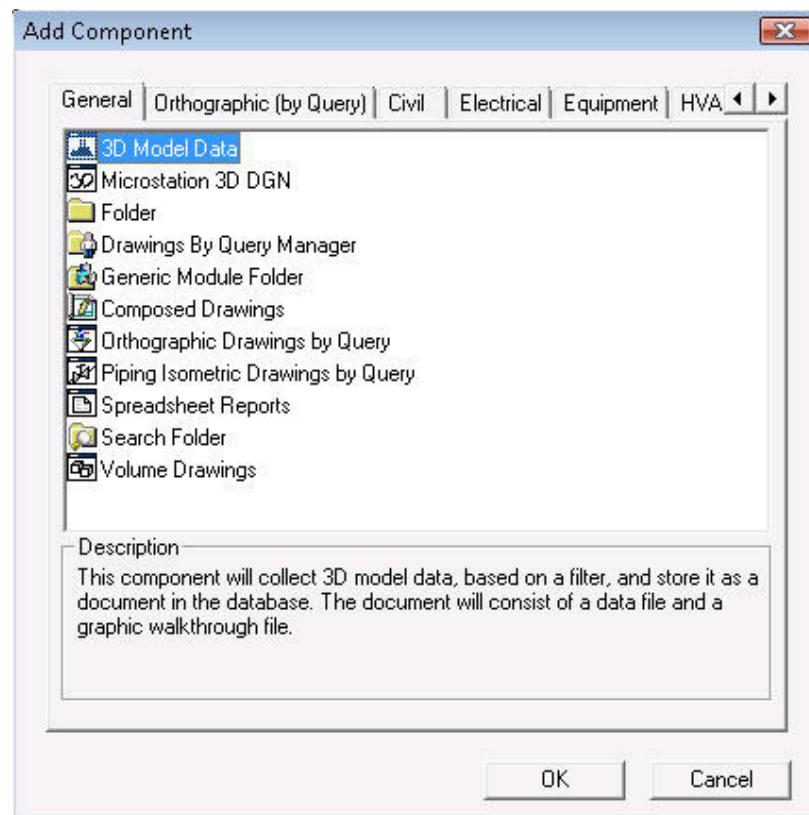
5. Hit OK. In the Management Console window, New Folder is created.



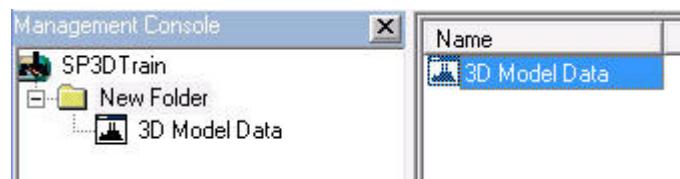
6. Right click on the New Folder, and select New as shown below,



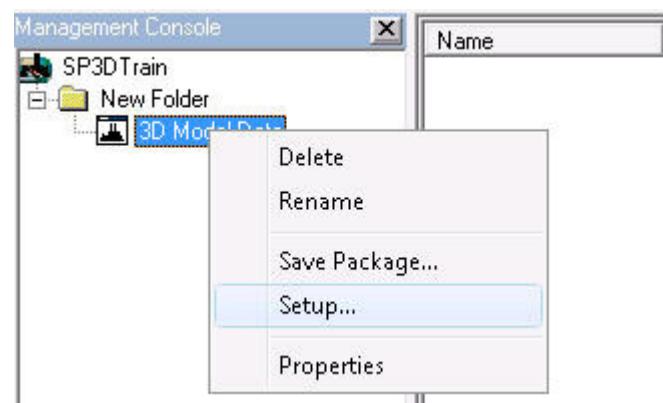
7. Choose 3D Model Data from the General tab as shown below and hit OK.



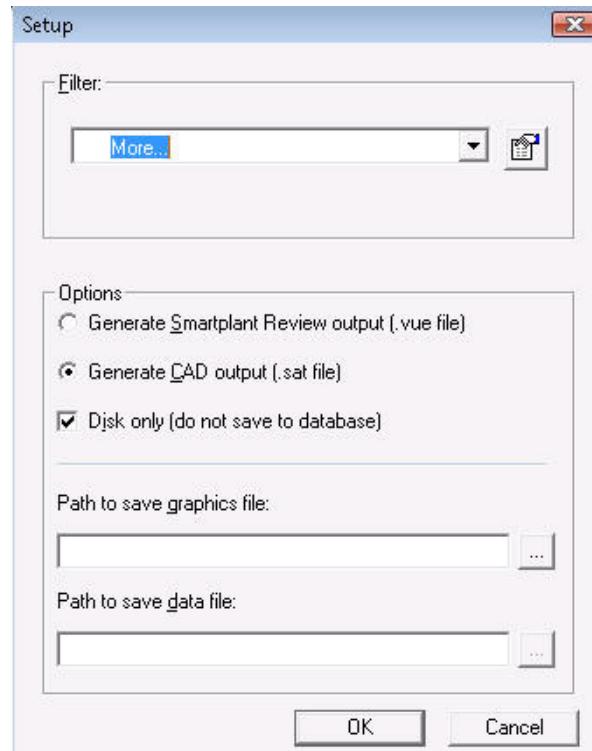
8. Your Management console window should resemble the following graphic.



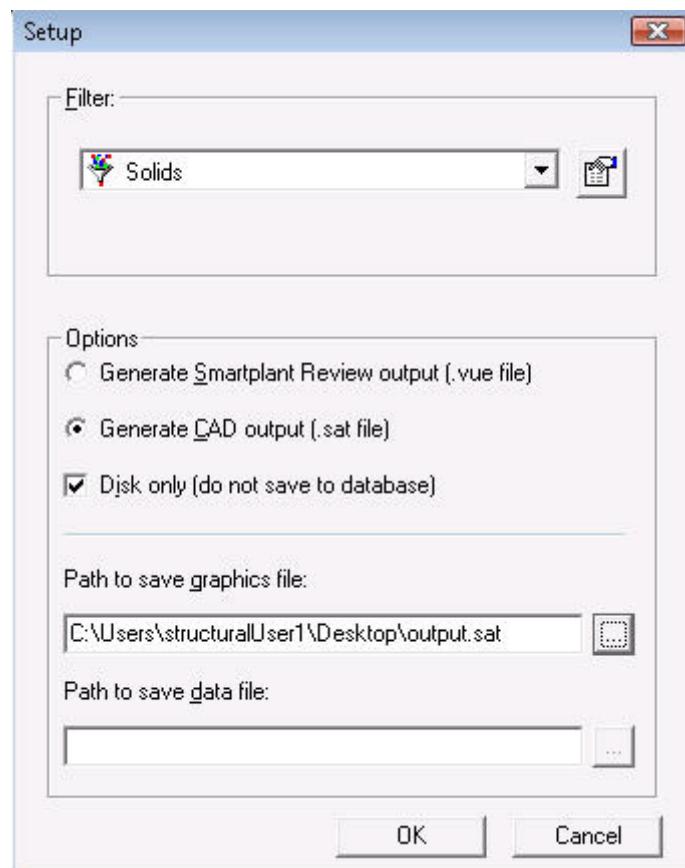
9. Right click on 3D Model Data and choose Setup...



10. Your view should resemble the following graphic,

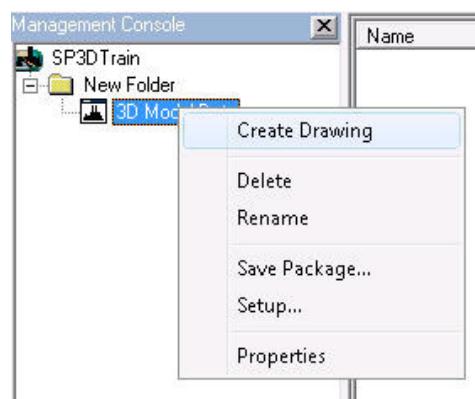


11. Go to More and select the filter that contains the *SlopedSlab*. Also provide the path to save the sat file. Your view should resemble the following graphic.

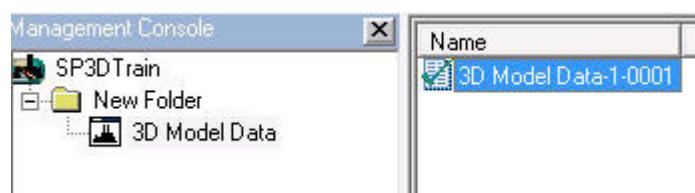


12. Hit Ok.

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



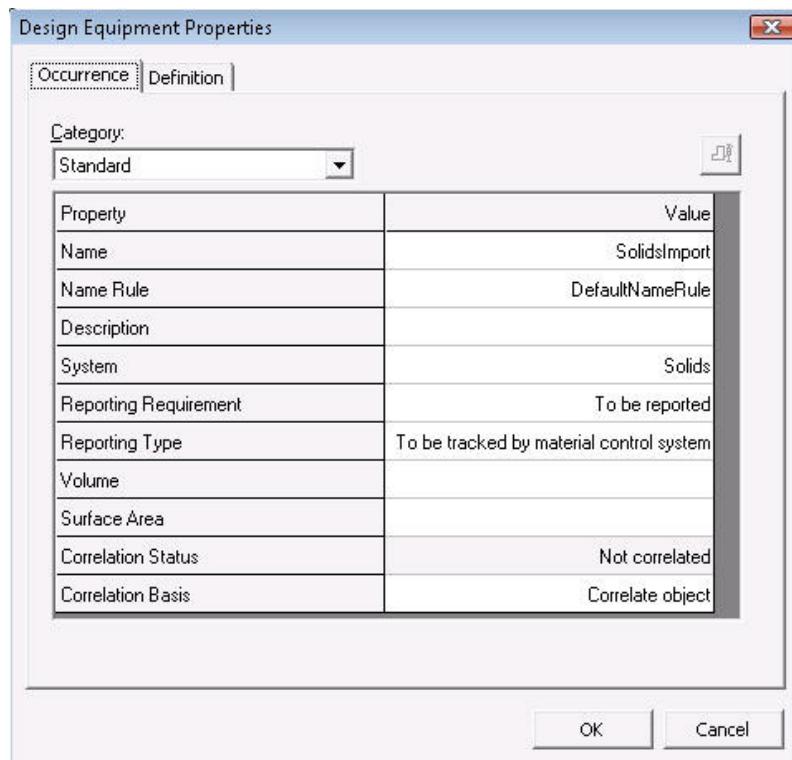
14. Right click on the 3D Model Data-1-0001 and hit Update Now. Your view should resemble the following graphic.



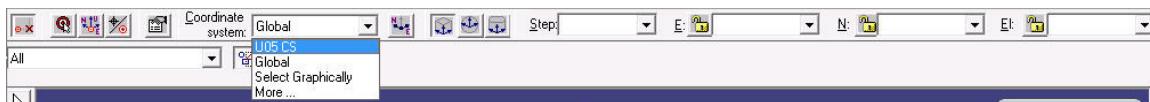
- The file has been successfully exported.

## Part B: Import

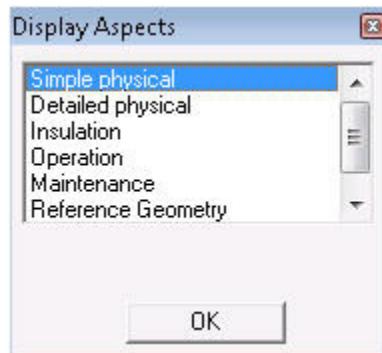
- Create a import.ses and save it on the Desktop.
- Create new filter by the name Solids-import.
- Define workspace to include A2 → U05 → Structural → Solids.
- Expand the fly-out toolbar and press the  icon on the vertical toolbar to start the Place Designed Equipment command. Expand *Equipment -> Civil* and select *Miscellaneous*. Press OK.
- In the Design Equipment Properties dialog, choose *Solids* as the System and give a User Defined name of *SolidsImport*. Also change the system to A2 → U05 → Structural → Solids & Press OK.



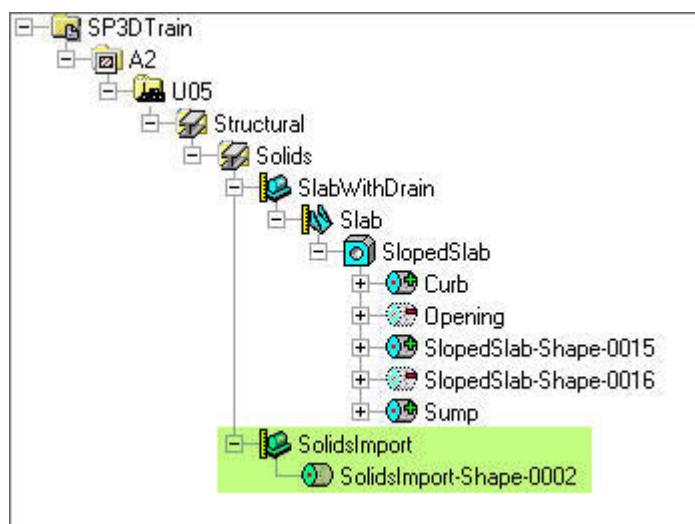
- Activate the PinPoint toolbar by selecting Tools -> PinPoint or by pressing  in the ribbon bar and change the coordinate system from Global to U05 CS in the ribbon bar.
- Change the coordinate system to U05 CS.



8. Place the Designed Equipment at E: -80 ft, N: -40 ft and EL: 0 ft.
9. Select Place Imported Shape from File command.
10. Select *SolidsImport* from the workspace explorer.
11. Select the output.sat file and hit Open.
12. Select *Simple physical* from the Display Aspects window and hit OK.



13. Key in E: -80 ft, N: -40 ft and EL: 0 ft. using the pin point values and click the left mouse button in the workspace window.
14. You view should resemble the following graphic in the workspace explorer window.



# STRUCTURAL ANALYSIS TASK

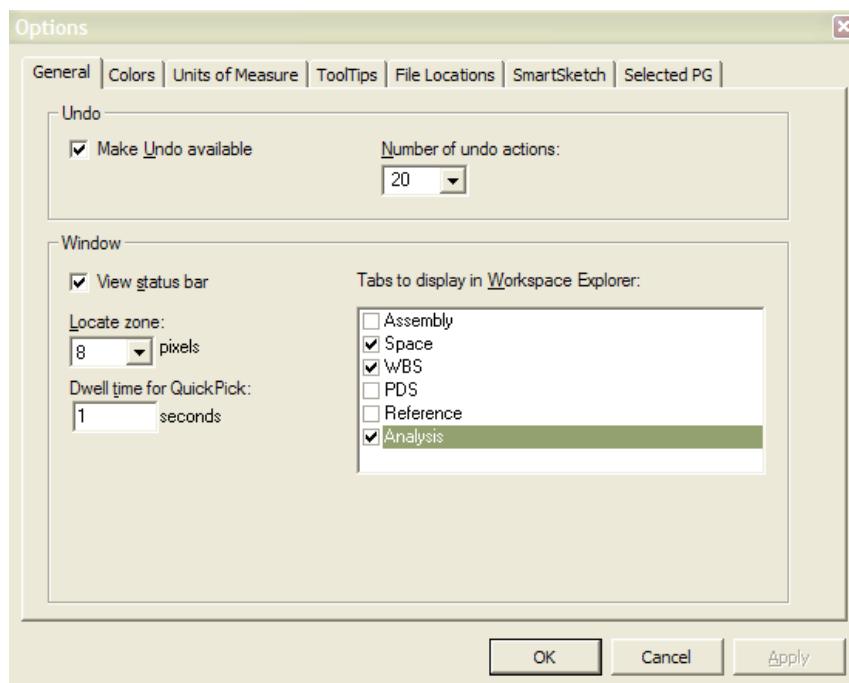
## LAB-1: Loads, Releases, Boundary Conditions and Creating a CIS file

### 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.

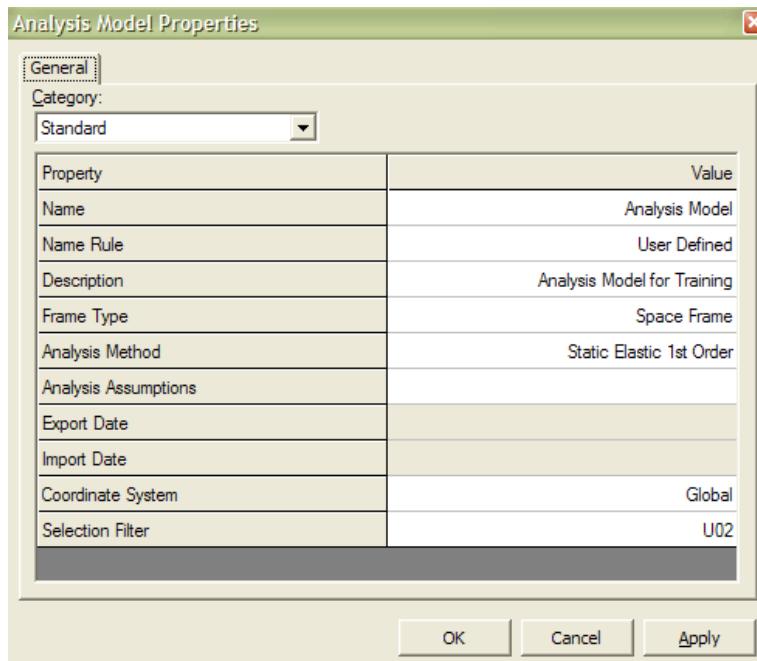
#### Part I: Create Load Cases and Combinations

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.

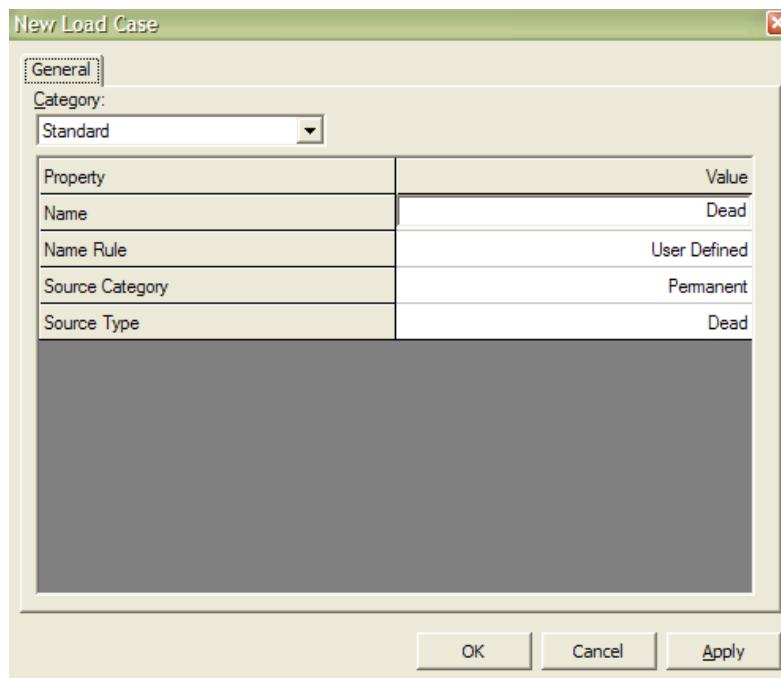


3. Hit “OK” button. Save your session.
4. Exit your session and Open it again. Notice the Analysis Tab in the Workspace Explorer.
5. Go to the Structural Analysis Task.

6. Create an Analysis Model using New Analysis Model Command 

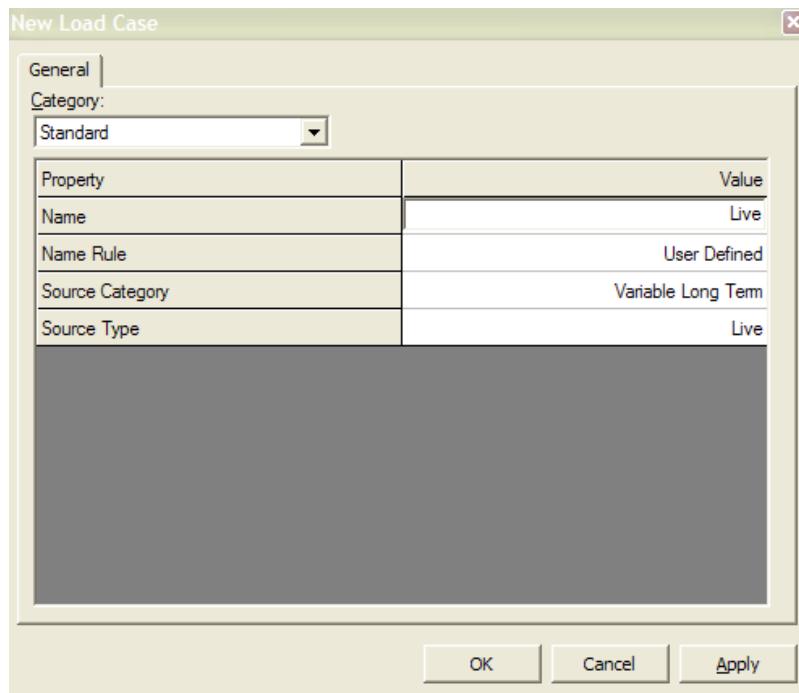


7. Hit “OK” Button.
8. Select Create New Load Case command. 
9. Create a DEAD load using New Load Case Command.



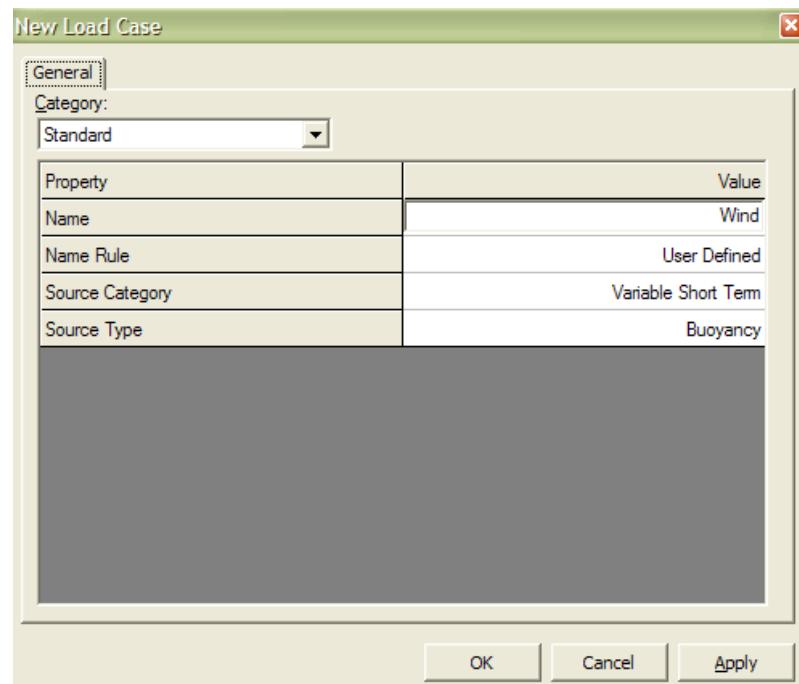
10. Hit “Apply” Button.

11. Create a LIVE load.



12. Hit “Apply” Button.

13. Create a WIND load.

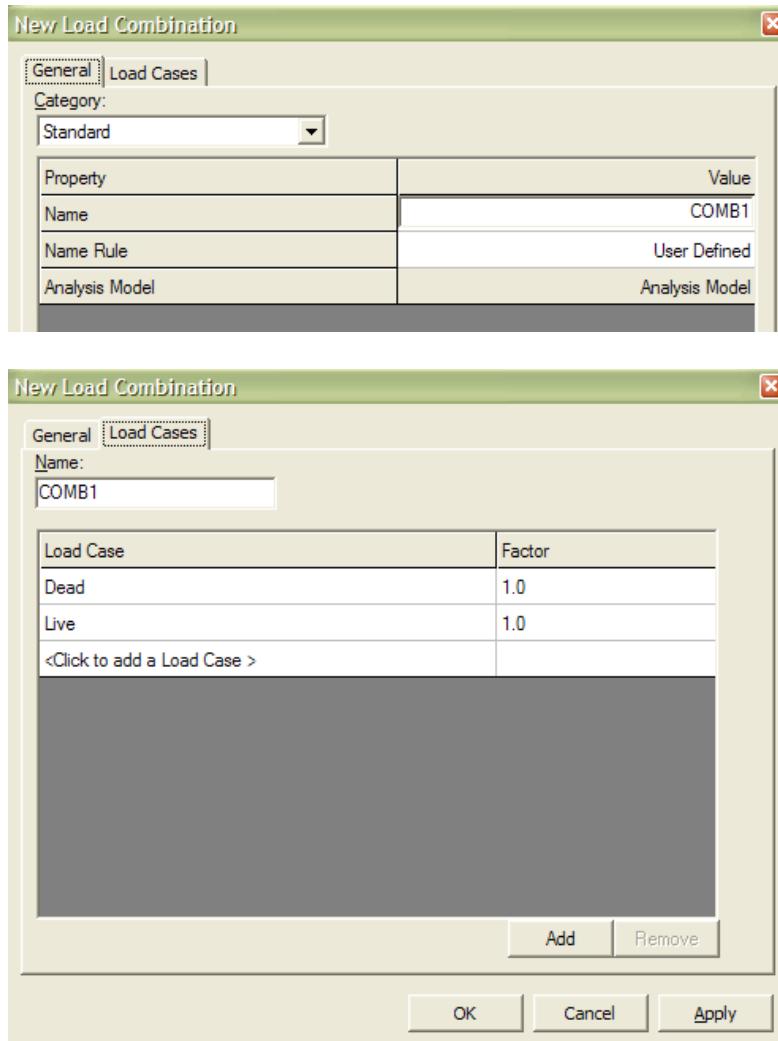


14. Hit “OK” Button.

15. Create Load Combinations using New Load Combination Command 

COMB1: Dead + Live with the Factor as 1.0

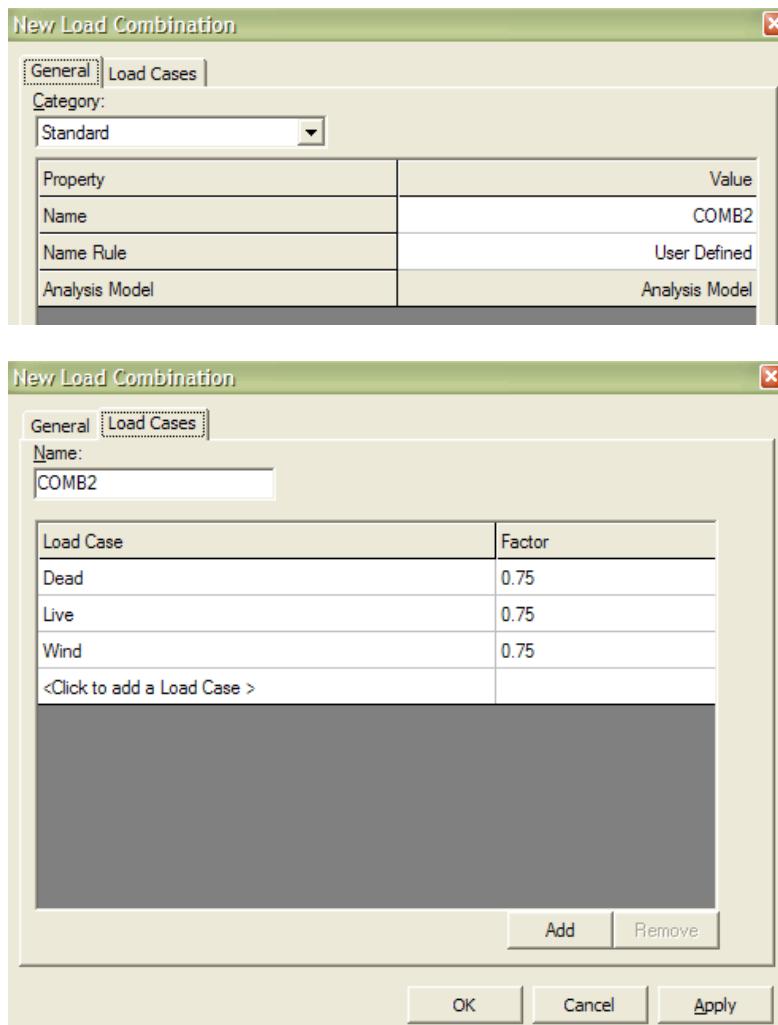
16. Set the parameter as follows:



17. Hit “Apply” Button.

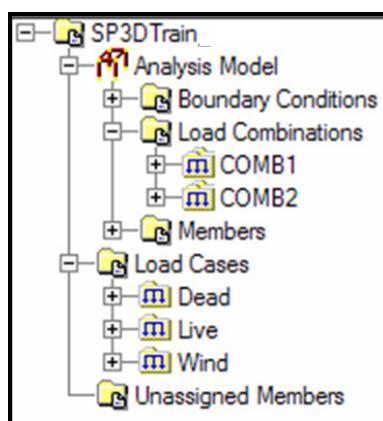
18. Create Load Combinations. 

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**

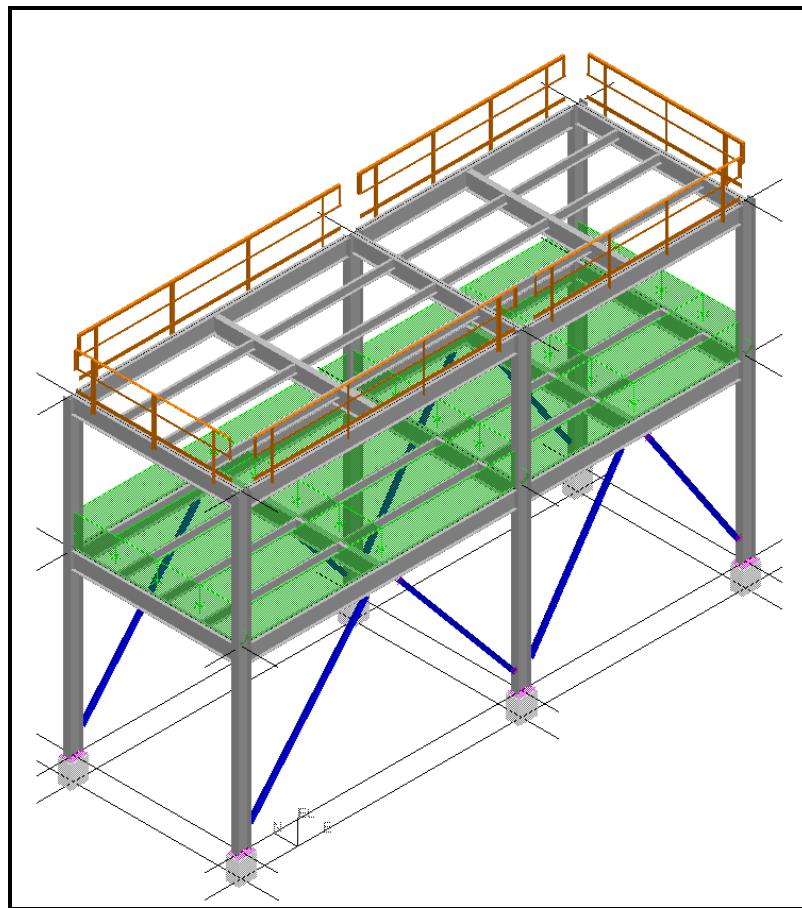
20. Select Place New Distributed Load Command. 

21. Define the following load for the beams at Elevation 18'- 0":

Load Case: DEAD  
 Load: Force Z  
 Reference: Global  
 Position as: Relative  
 Start Position: 0  
 End Position: 1  
 Start and End Magnitude: -0.65 kpf



22. Select all beams at Elevation 18'-0" and click "Finish" button. Loads will be placed as shown below:



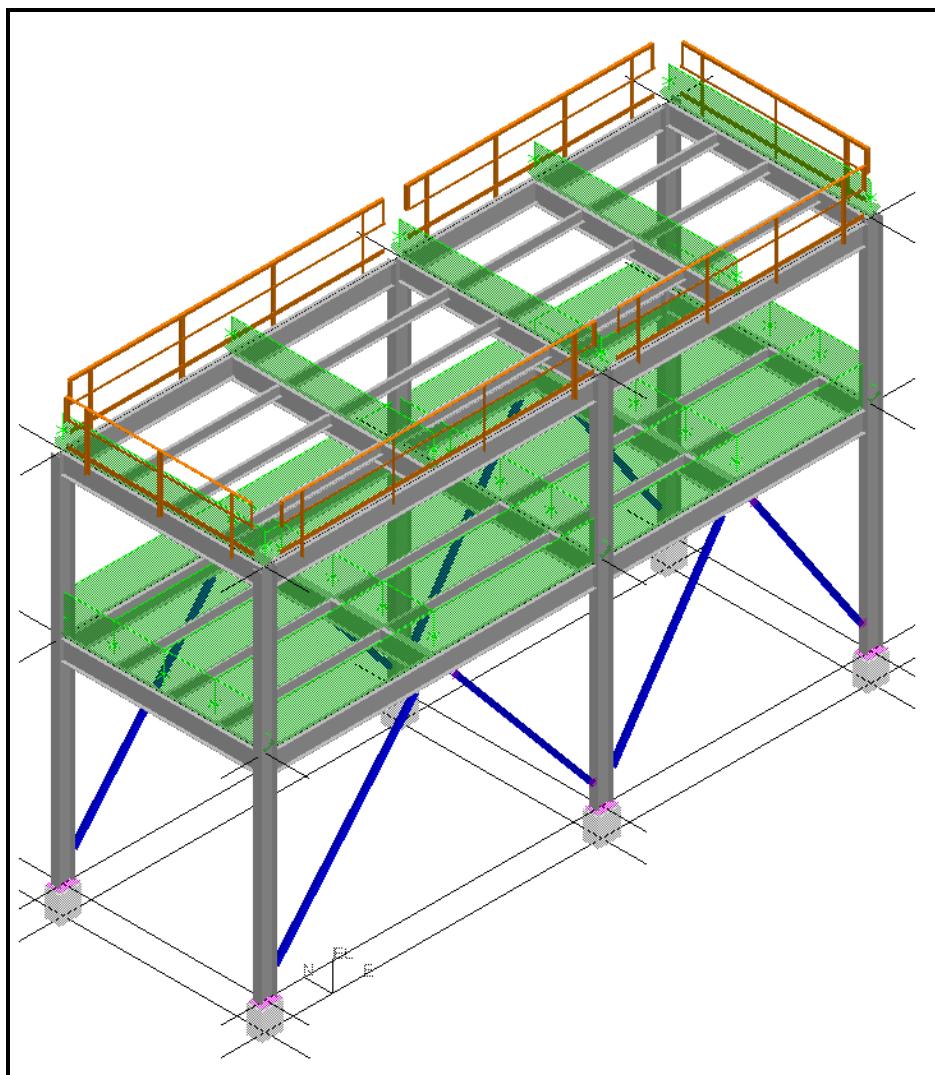
23. Define the following distributed load for the beams at Elevation 30'-0":

Load Case: DEAD  
Load: Force Z  
Reference: Global  
Position as: Relative  
Start Position: 0  
End Position: 1  
Start and End Magnitude: -0.100 kpf

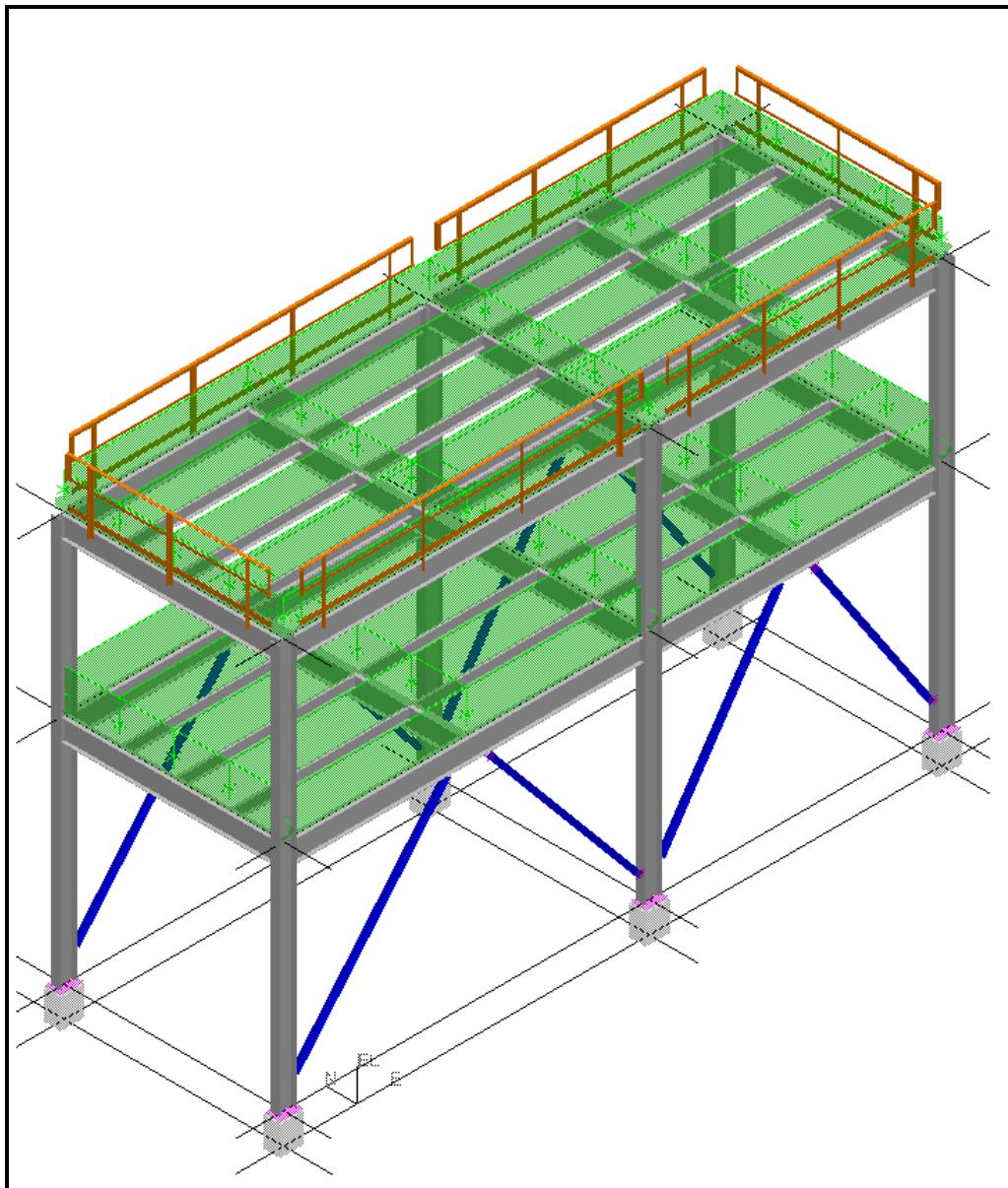


24. Select the beams running north/south at Elevation 30'-0" and click Finish.

Loads will be placed as shown below:



25. Change the Magnitude to  $-0.200 \text{ kpf}$  and apply this to the 4 primary beams running east/west at Elevation  $30'-0"$ .
26. Change the Load Case Name to Live and the Magnitude to  $.220 \text{ kpf}$  and apply the load to the interior framing members and to the 4 primary beams running east/west at Elevation  $30'-0$ .
27. Change the Magnitude to  $-0.110 \text{ kpf}$  and apply the Live Load to the 3 beams running north/south at Elevation  $30'-0"$ .



**Part III: Place Wind Loads**

28. Select Place New Concentrated Load Command. 

29. Define the load as:

Load Case: WIND

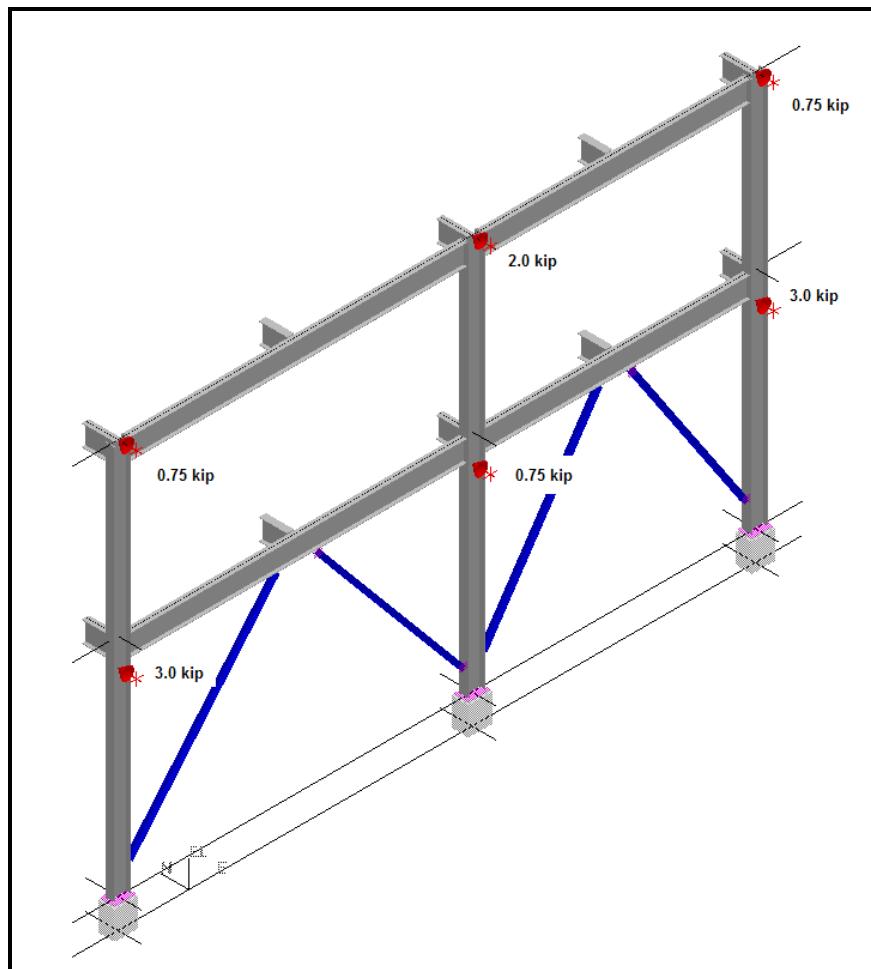
Load: Force Y

Reference: Global

Position as: Relative

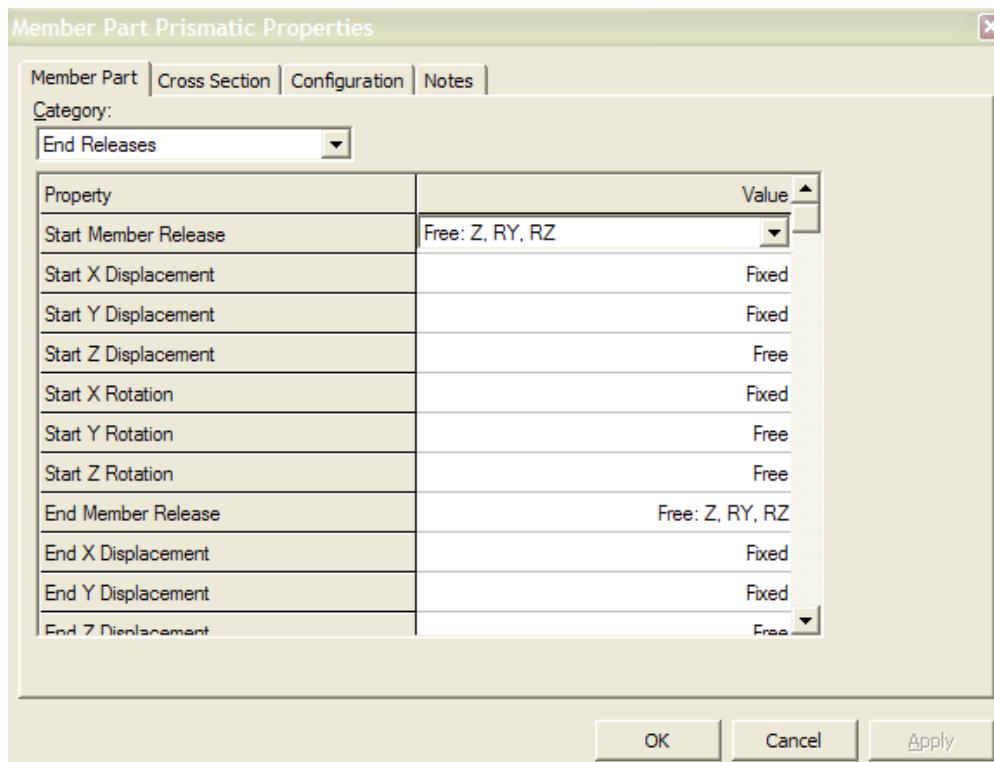


30. Place concentrated wind loads along the south side of the structure with the following magnitudes:



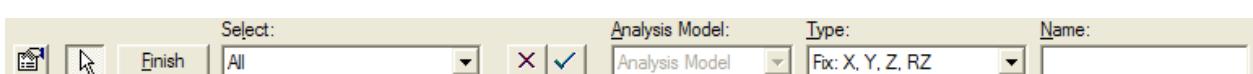
#### Part IV: Set Member Releases

31. Select all beams in U02 with the Locate Filter set to Member Parts.
32. Select Edit -> Properties in Menu to open the Properties dialog box
33. Choose the End Releases Category.
34. Set Start and End Member Release to Free: RY, RZ.



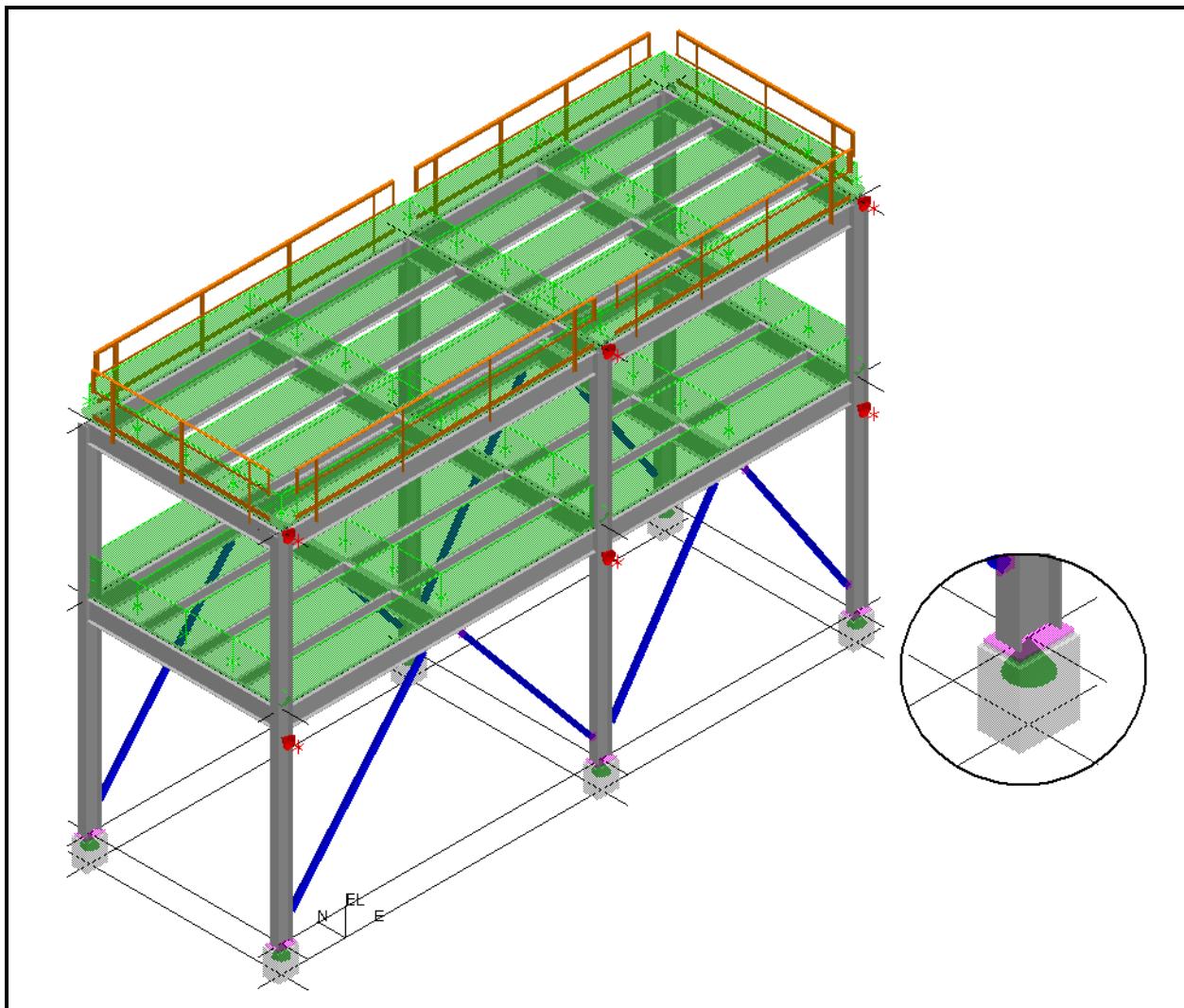
#### Part V: Place Boundary Conditions

35. 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:

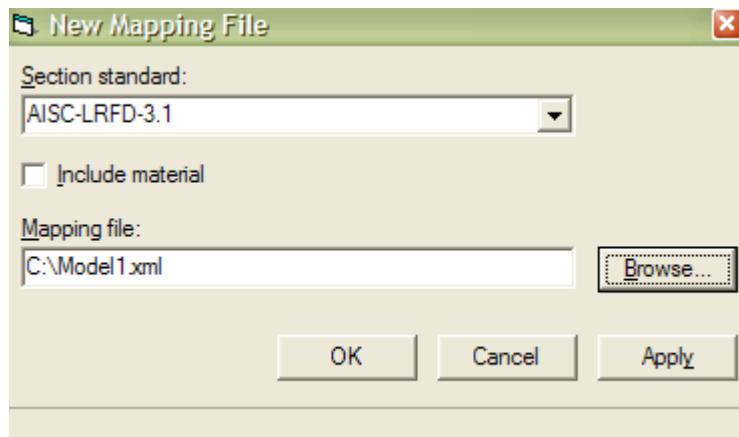


## **Part VI: Export Analytical Model and Update from Analysis**

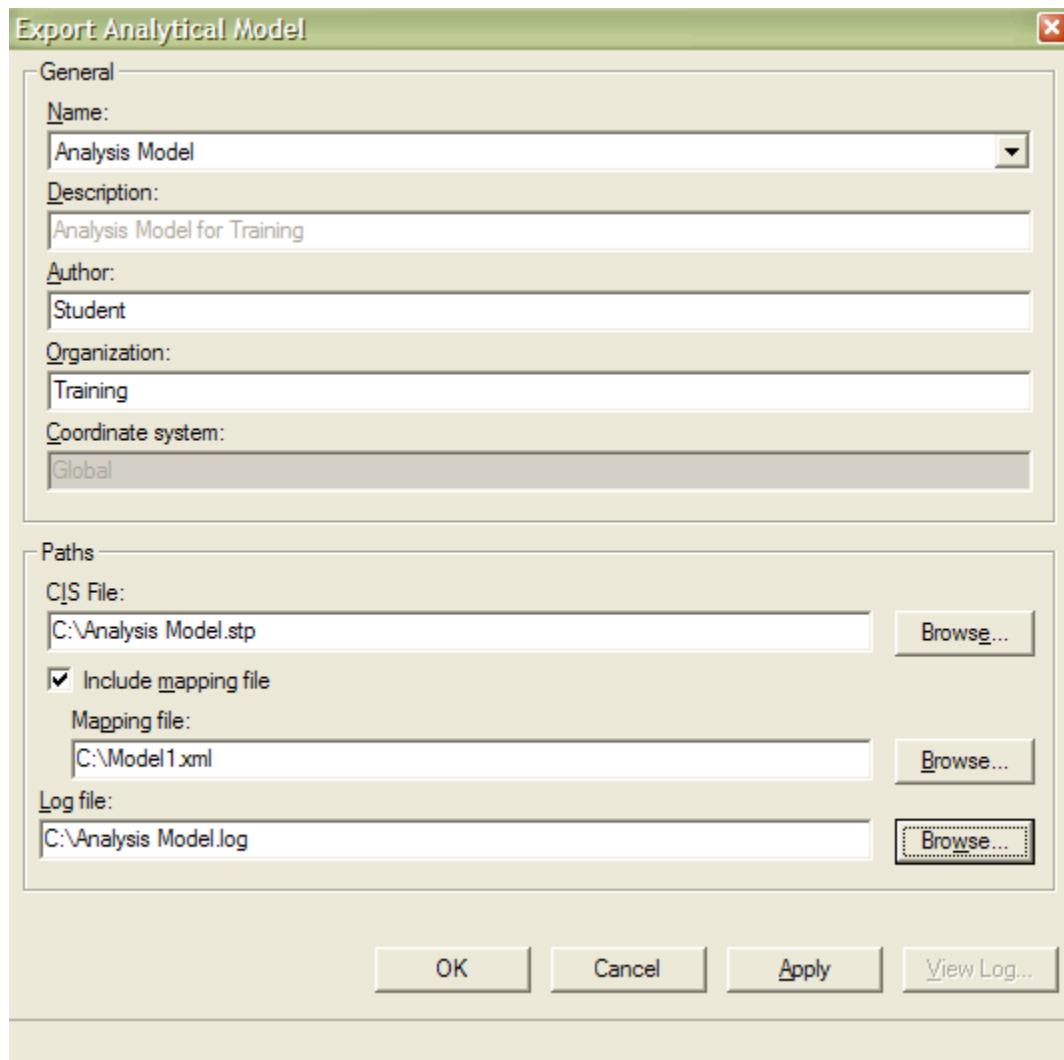
1. Select File -> New Mapping File

*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)