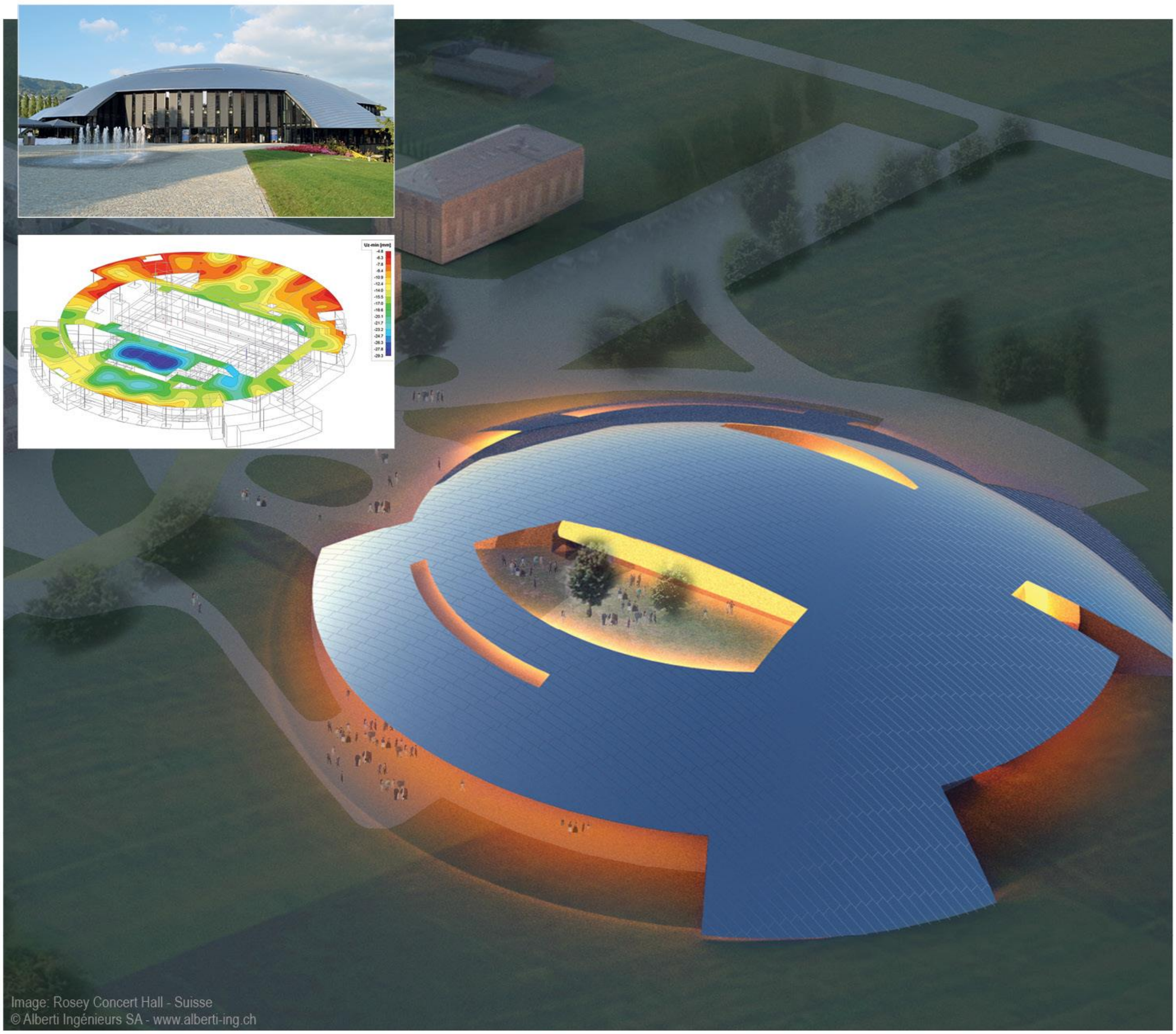
SCIA ENGINEER



Tutorial

Steel Building – Modeling

Release: SCIA Engineer 16
Revision: 06/2016

Nemetschek SCIA Offices

Latest version is available on: https://www.scia.net/en/contact/offices

Belgium Headquarters
Nemetschek SCIA nv
Industrieweg 1007
B-3540 Herk-de-Stad
Tel.: +32 13 55 17 75
Fax: +32 13 55 41 75
E-mail: info@scia.net

Support Phone
CAE (SCIA Engineer)
Tel: +32 13 55 09 90

CAD (Allplan)
Tel: +32 13 55 09 80

Support e-mail:
support@scia.net

Austria
SCIA Datenservice Ges.m.b.H
Dresdnerstrasse 68/2/6/9
A-1200 Wien
Tel.: +43 1 7433232-11
Fax: +43 1 7433232-20
info@SCIA.at

Support
Tel: +43 1 7433232-12
E-mail: support@SCIA-online.com

Brazil
Nemetschek SCIA do Brasil
Rua Dr. Luiz Migliano, 1896 – sala 702, CEP
SP -05711-001 São Paulo
Brasil
Tel: +55 11 4314-5880
E-mail: brasil@SCIA-online.com

Czech Republic
Nemetschek SCIA s.r.o.
Evropská 2591/33d
160 00 Praha 6
Tel.: +420 226 205 600
Fax: +420 226 201 673
E-mail: info.praha@SCIA.cz

Nemetschek SCIA s.r.o.
Slavickova 827/1a
638 00 Brno
Tel.: +420 530 501 570
Fax: +420 226 201 673
E-mail: info.brno@SCIA.cz

France
Nemetschek SCIA sarl
Centre d’Affaires
29, Grand’ Rue
FR-59100 Roubaix
France
Tel.: +33 3.28.33.28.67
Fax: +33 3.28.33.28.69
france@SCIA-online.com

Germany
SCIA Software GmbH
Emil-Figge-Strasse 76-80
D-44227 Dortmund
Tel.: +49 231/9742586
Fax: +49 231/9742587
info@SCIA.de

Netherlands
Nemetschek SCIA bv
Wassenaarweg 40
NL- 6843 NW Arnhem
Tel.: +31 26 320 12 30
Fax: +31 26 320 12 39
info@SCIA.nl

Slovakia
Nemetschek SCIA s.r.o.
Murgašova 1298/16
SK - 010 01 Žilina
Tel.: +421 415 003 070-1
Fax: +421 415 003 072
info@SCIA.sk

Switzerland
Nemetschek SCIA Switzerland
Dürenbergstr. 24
CH-3212 Gurmels
Tel.: +41 26 341 74 11
Fax: +41 26 341 74 13
info@SCIA.ch

USA
SCIA Inc. North America
7150 Riverwood Drive
Columbia, MD (USA)
Tel.: +1 443-671-1431
Fax: +1 410-290-8050
usa@scia.net

Table of Contents

Introduction

This tutorial aims at introducing the user to the SCIA Engineer functionalities which are used in the design of a commercial steel building structure. The functionality in this tutorial are consistent with those found in the Fundamentals Edition of SCIA Engineer. Additionally, in order to properly understand the tutorial, the user must be familiar with the basic principles of the Finite Element Method. For more detailed information on the Finite Element Method and how it is utilized in SCIA Engineer, refer to the [FEM Training Manual.](#)

In all, this tutorial consists of 7 parts including:

Part 1 – Modeling

Part 2 – Loading

Part 3 – Analysis, Meshing & Results

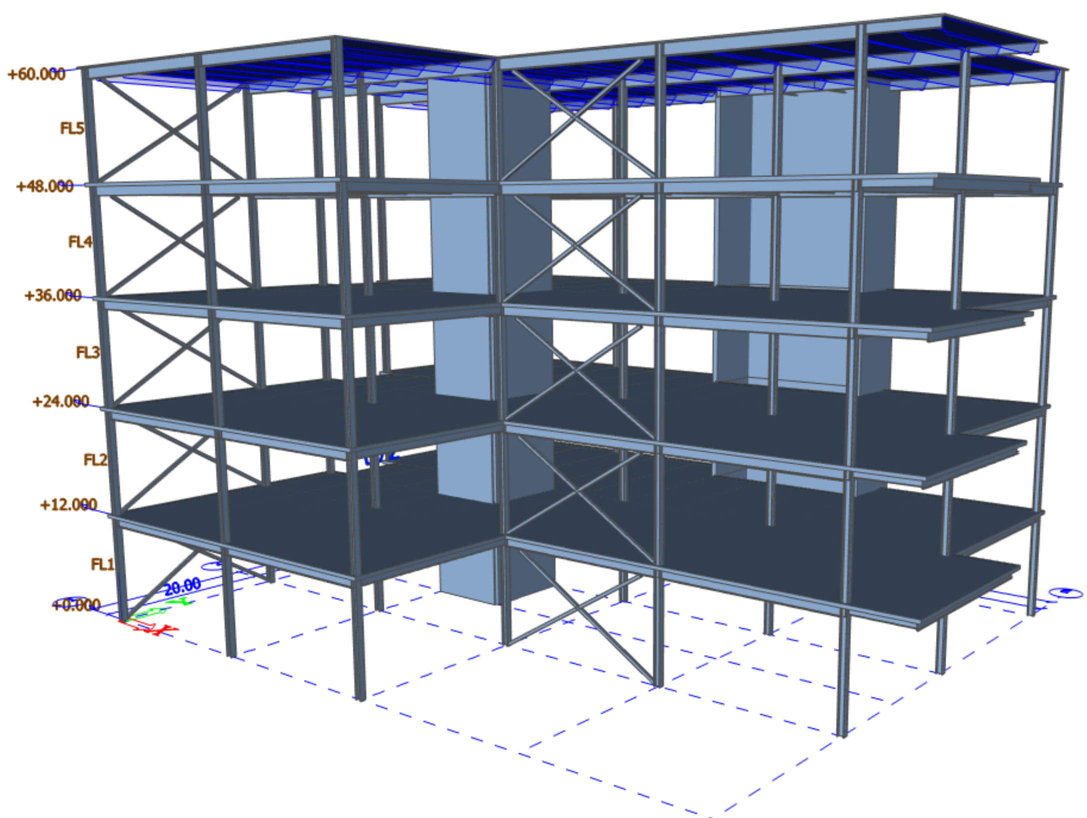
Part 4 – Steel Design (Columns, Braces, Joists)

Part 5 – Composite Steel Design

Part 6 – Concrete Shear Wall Design

Part 7 – Engineering Report

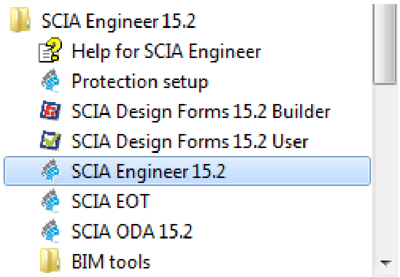
In Part 1 of the Commercial Steel Building tutorial, the functionality and features that are used to start a project and create the model of the structure will be discussed.



Getting Started

Launching the Program

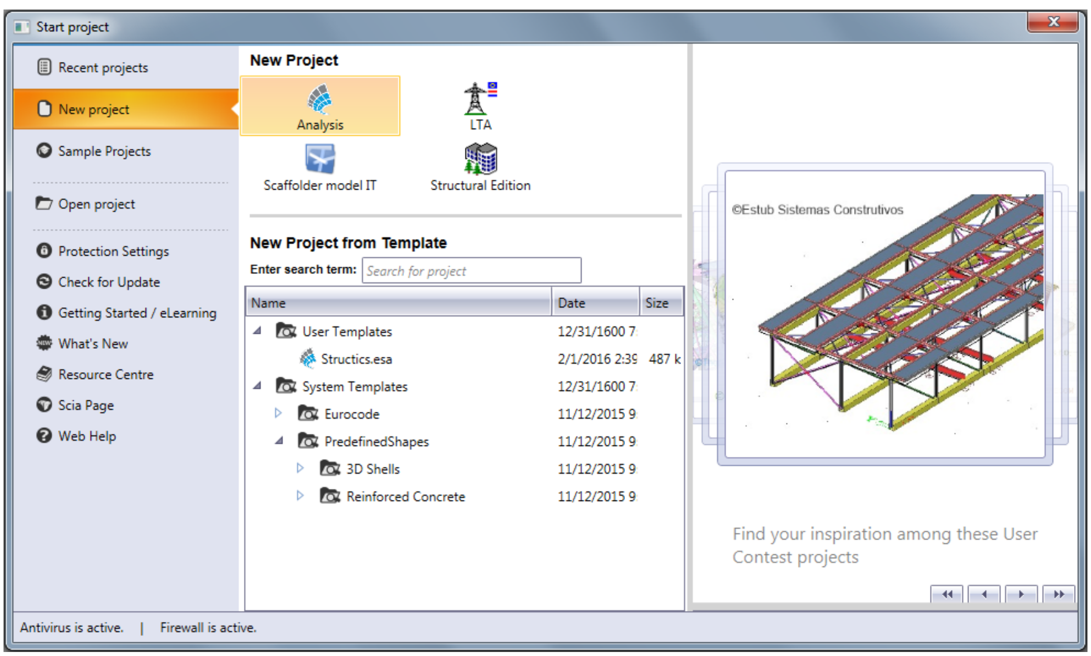
To begin, open SCIA Engineer by double clicking on the desktop icon or by navigating to **Start > All Programs > SCIA Engineer**.



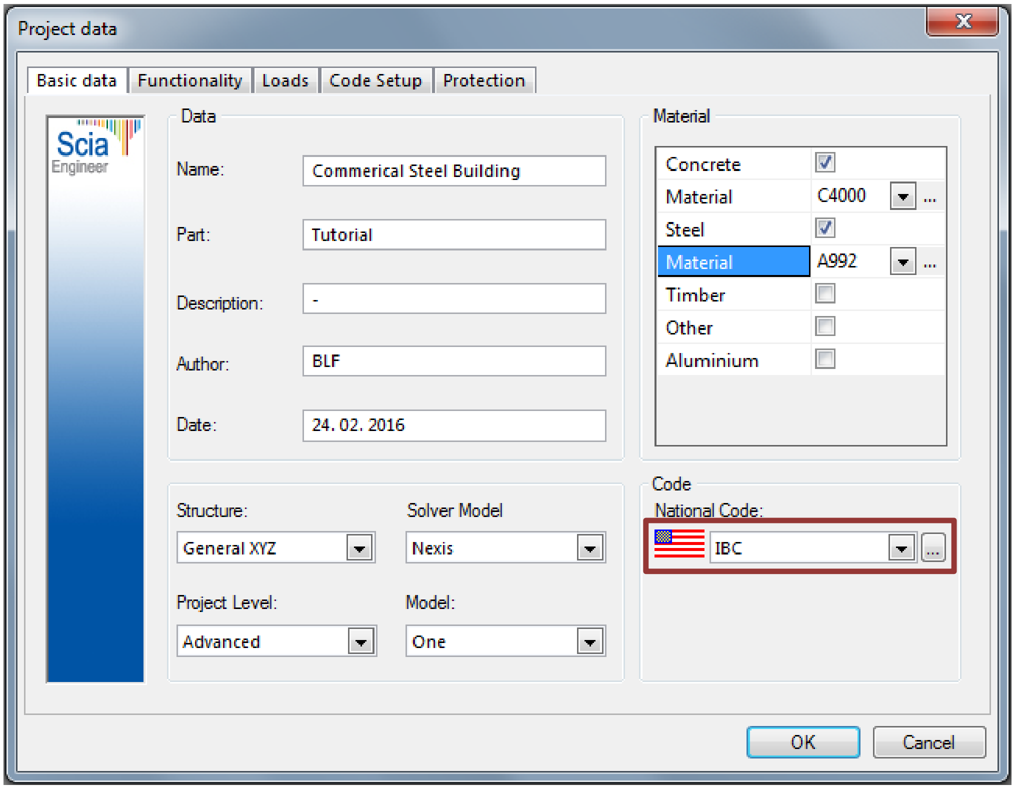
<u>Note:</u> If no protection is found, a dialogue will appear with the limitations of the try-out/demo version. For more information on the protection and licensing of SCIA Engineer, visit the **Documents** section of the [SCIA Engineer 15 downloads page.]()

Starting a New Project

Once SCIA Engineer has been opened, the **Start Project** dialogue box will appear. Select the **New Project** tab and then double click on **Analysis** to start the project.

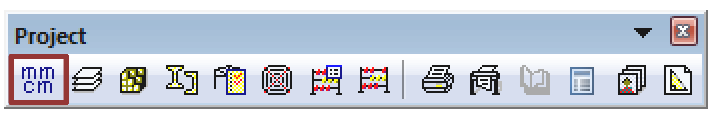


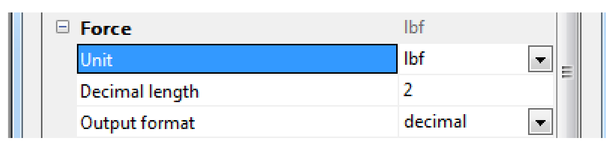
The **Project Data** window will now appear, for this project enter the options as shown in the picture below. Changes to the materials can be made by using the drop down arrows below the checkboxes, however by default all types of concrete and steel will be available in the project. Additionally, if the **National Code** is different than IBC, it can be modified by selecting the ellipsis button (3 dots) next to the current active code. Once the inputs are complete, click **OK** to start the project.



Dot Grid Settings & Units

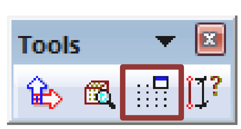
Once the project data has been set and the graphical interface is open, the user can manipulate both the units for the project and dot grid spacing. The units are accessed through the drop down menus, Setup > Units or by using the button on the **Project toolbar.** For this project, we will use units of **"ft"** for length and **"lbf"** for loading.





<u>Note:</u> If the Unit button does not show up on the **Project** toolbar, you will need to switch the current style of toolbars. To do so, navigate to Setup > Options > Current Style of Toolbars > Full Toolbars. You will have to restart SCIA Engineer in order for the change to take effect.

To access the dot grid, select the button located on the **Tools** toolbar or use the drop down menus, Tools > Dot Grid and Tracking Settings. Within the dot grid settings, change the grid spacing to be **1 ft** and click **OK.**

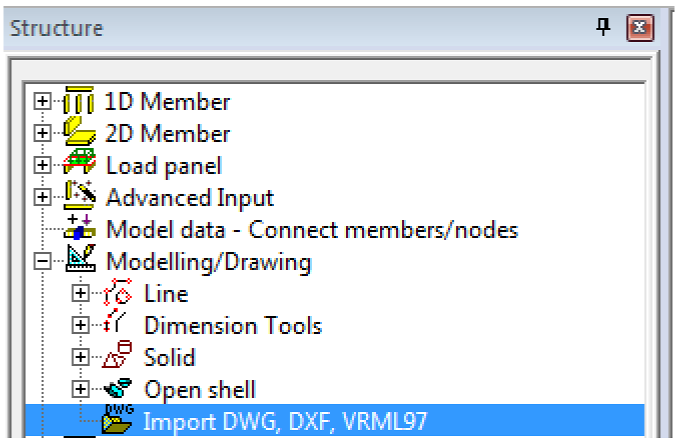


Import drawing in Graphical Window

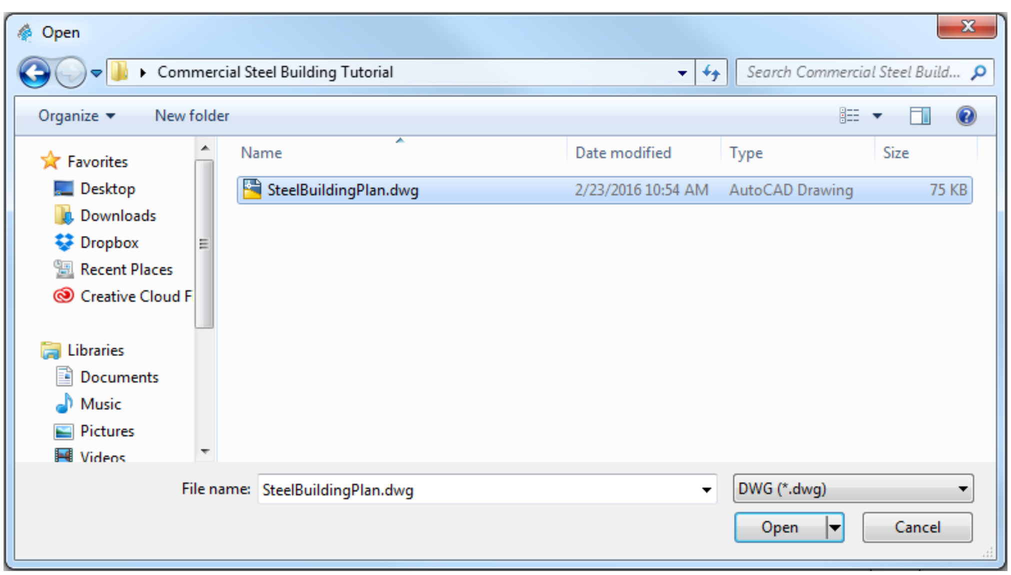
In many cases, the structural engineer has the opportunity to leverage external model data that was created by the architect in order to more efficiently create the analysis model. This can be done using a variety of model exchanges (Revit, IFC, Rhino, etc.). In this example, a file from AutoCAD will be used as the basis of the model.

Import CAD Drawing (.dwg) for modeling

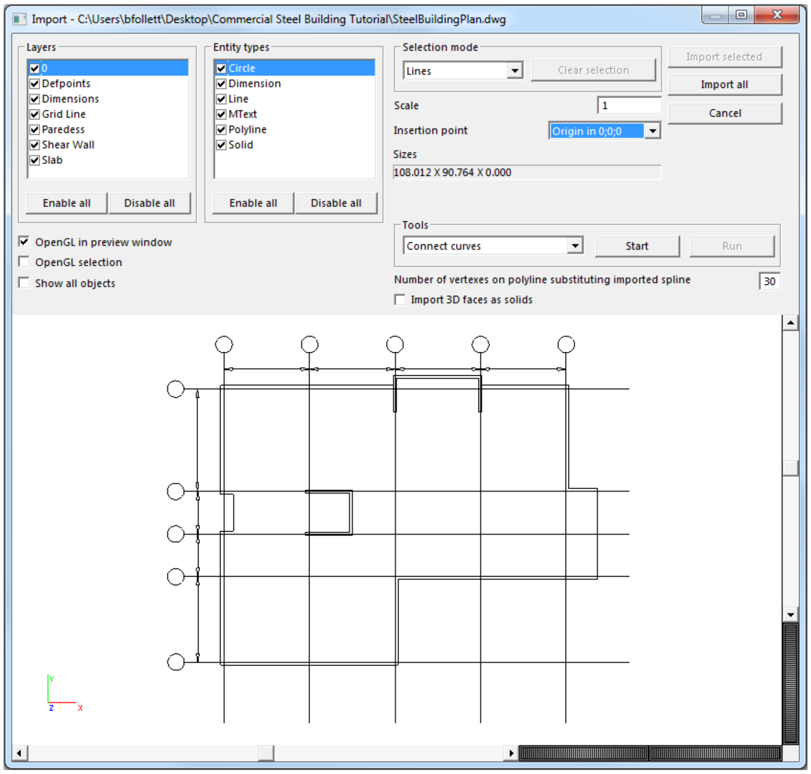
In the **Main** service tree, double click Structure to begin the process of modeling. To choose a .dwg to import into the project, double click on **Import DWG, DXF, VRML97** within the **Structure** tree.



The **Open** window appears and it is necessary to navigate to the location of the **SteelBuildingPlan.dwg** file on your local system. This .dwg is supplied as part of the tutorial. After the drawing is selected, click **Open**.



The Import window will be displayed and some of the import parameters can be adapted before importing the drawing.



Layers:

- Contains all layers that were defined in the original .dwg file
- All layers are selected by default

Entity Types:

- Contains available entity types (lines, surfaces, solids, etc.)
- All entities are selected by default

Selection mode:

- The selection mode of this project will be **Lines**
- Other selection modes are based on the various entity types

Scale:

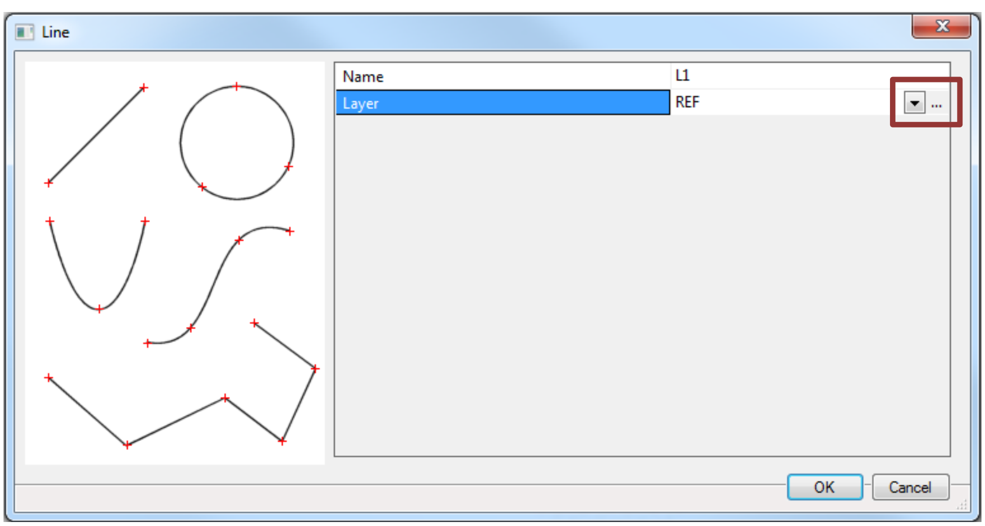
- The scale used for this import is **1:1**
- Based on the current units of a .dwg and SCIA, the scale factor may need to be changed

Insertion Point:

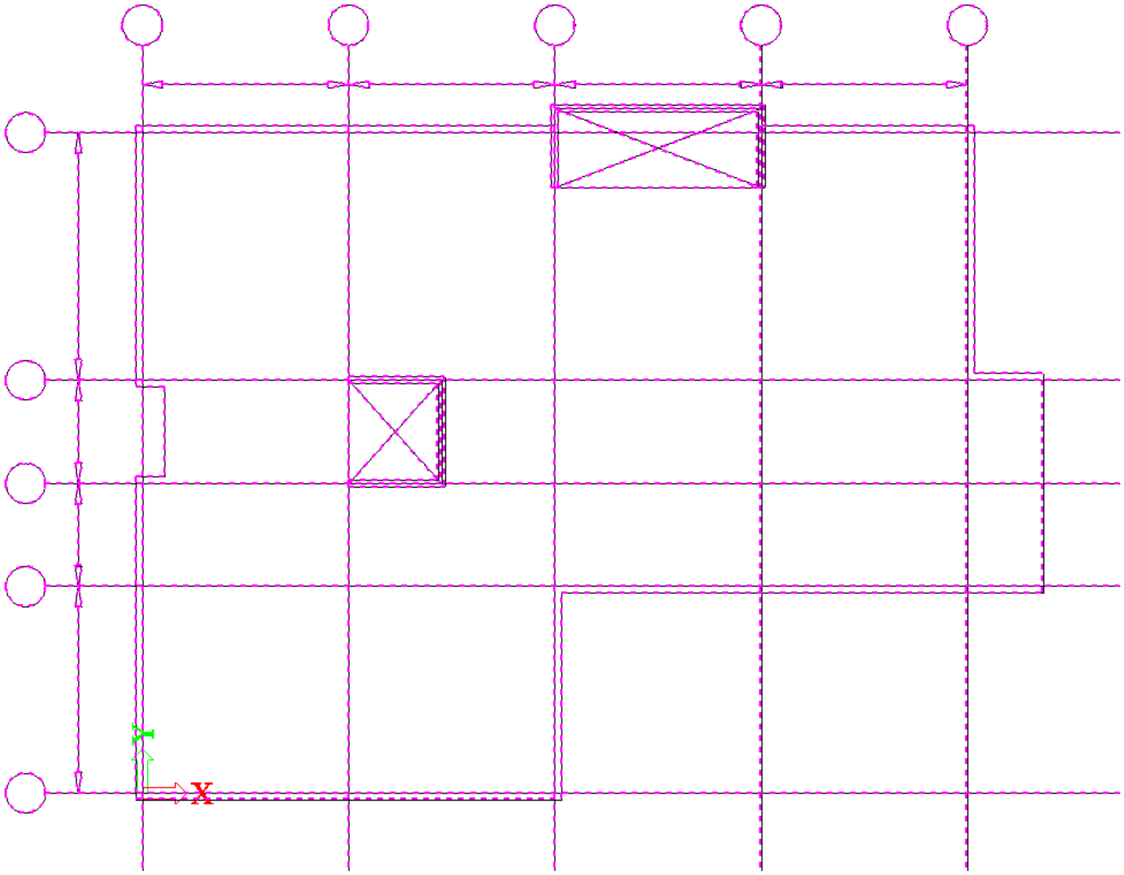
- Options include Centre (based on manual selection of point), Origin and Origin (0;0;0).
- For this tutorial, the drawing will be imported at the origin (0;0;0)

With the properties of the Import set as shown in the picture above, click **Import All.**

Note: For more information concerning the import of geometry from CAD software, refer to the SCIA Engineer Online Help, Import into the Graphical Window.



Upon import, the drawing can be placed on a specific layer. In this tutorial, click on the ellipsis button (3 dots) in order to rename **Layer 1** as **REF** and then click **OK.**



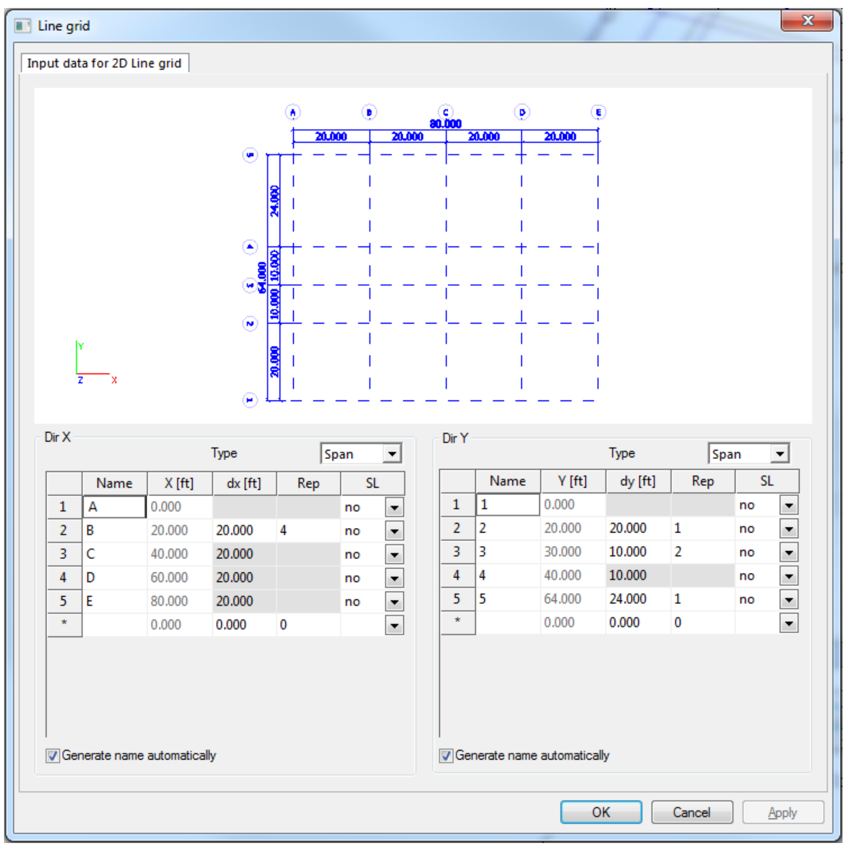
The imported CAD drawing can now be used as a basis for the model. Before, continuing it would be wise to save the file, using **File > Save As.**

Line grids & stories

Before the structure is created, Line grids & stories should be added to the project as a means to organize data. Additionally, Stories will be used to quickly view only elements on a specific level as well as reporting results for lateral loading. To create these, open the functionality by doubling clicking on Line grid and storeys in the main service tree.

Line Grids

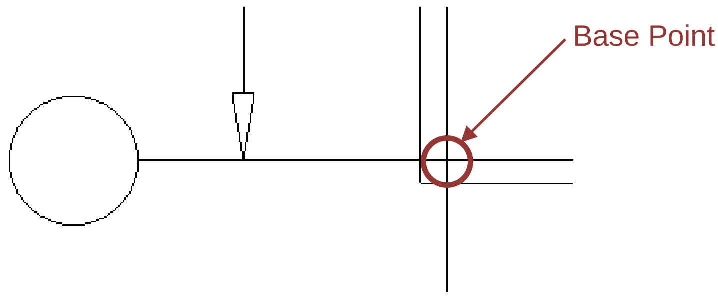
For this example, a rectangular **2D Line grid** will be used. Click on the [+] to expand the 2D Line grid field and then double click on **Rectangular grid.** Once the rectangular line grid dialogue appears, input the grid dimensions as shown in the picture below and click **OK.**



Once the grid dimensions have been added, additional properties concerning the grids labels and dimensions can be manipulated as shown below. When the changes have been made click **OK.**

Direction X	End
Label shift X [ft]	3.000
Direction Y	End
Label shift Y [ft]	3.000
Draw label in	Circle
Dimension	
Direction X	End
Direction Y	End

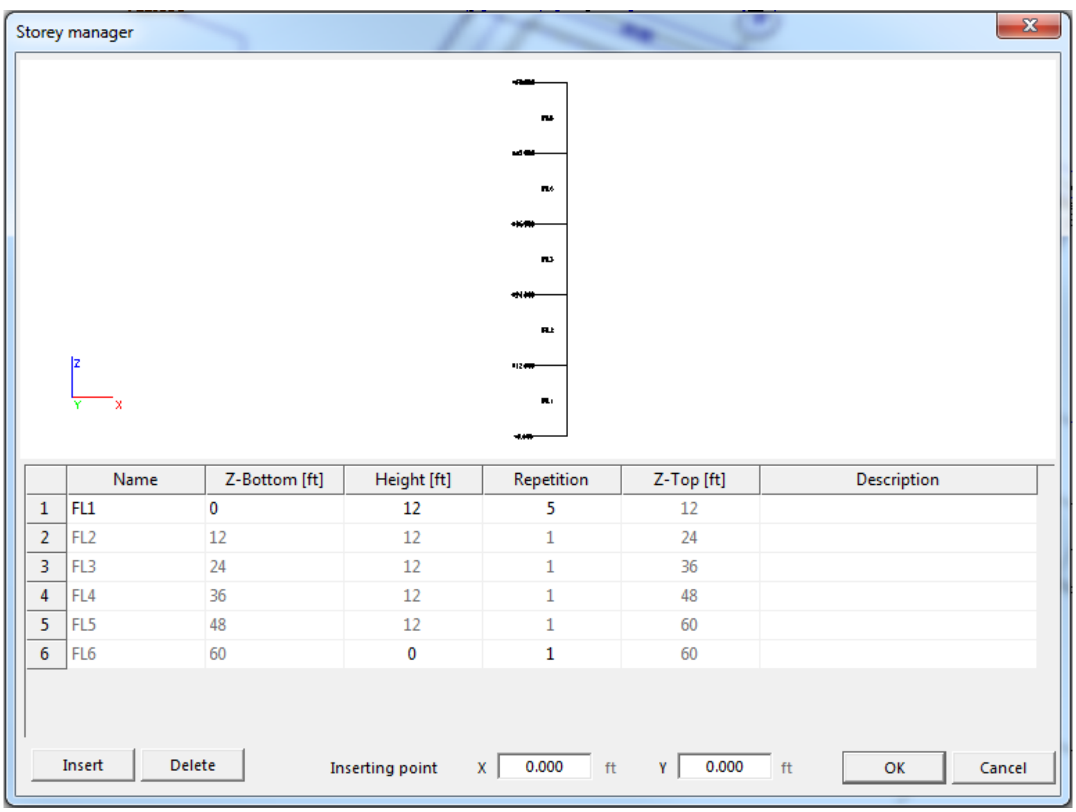
The input location of the rectangular grid is based on the intersection point of Grid Lines A & 1 located in the bottom left corner of the structure. Select that point as the base point to complete to input of the **2D Rectangular Line grid.**



<u>Note:</u> The same rectangular line grid can also be created using the **free grid lines** functionality. Free grid lines allow the user to input grid lines anywhere in the graphical window and can be used to trace over grid lines that exist from an imported CAD file.

Stories

Add stories by double clicking on Storeys in the Line grid and stories service. The building in this tutorial will consist of **5 stories, all 12ft high** for a total building height of **60ft**. Create the stories as shown in the picture below, making sure to use the correct inserting point **(0, 0)** and then click **OK.**



	Name	Z-Bottom [ft]	Height [ft]	Repetition	Z-Top [ft]	Description
1	FL1	0	12	5	12	
2	FL2	12	12	1	24	
3	FL3	24	12	1	36	
4	FL4	36	12	1	48	
5	FL5	48	12	1	60	
6	FL6	60	0	1	60	

Once the stories have been added to the graphical window, click the **Close** button at the bottom of the Main Tree window to exit the Line grids and stories functionality.

Input a Floor Plate

For this tutorial, the floor plate will consist of a concrete slab on top of a metal deck. To input the proper extents of the floor plate, the imported .dwg drawing can be used.

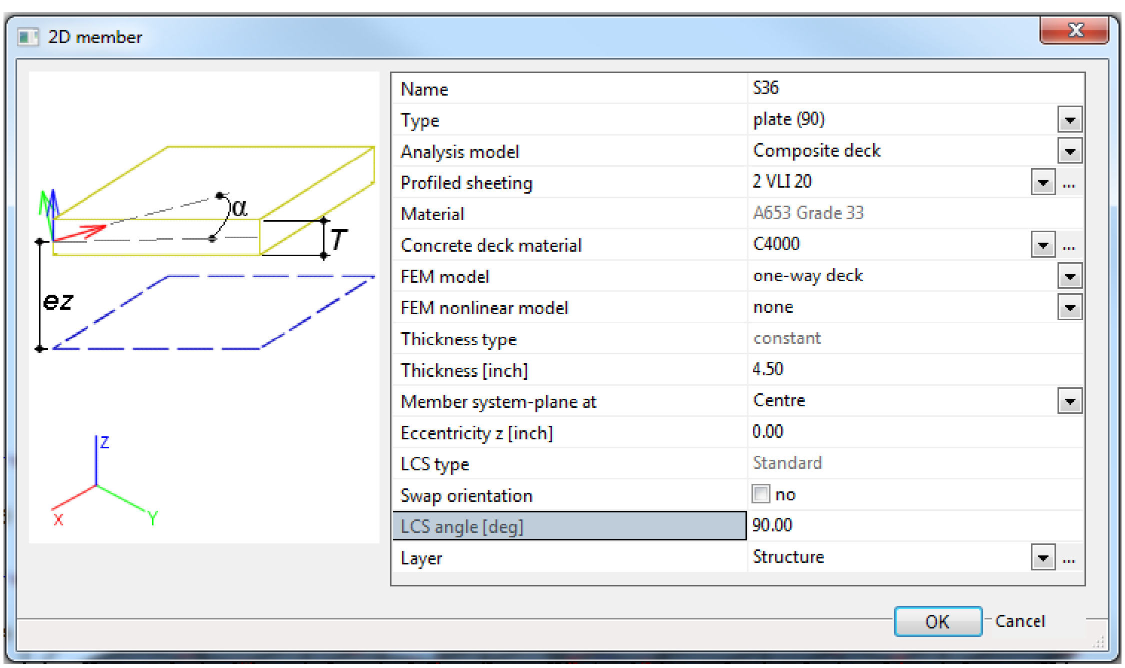
To do so, double click on the Structure service and then click on the [+] to expand the 2D Member field. Next, double click on Plate to open the plate input dialogue. Once the dialogue appears, various properties of the plate can be manipulated.

First, change the **Analysis model** type to **Composite deck**. Doing so will launch the profiled sheeting library. Click **Cancel** in order to close the dialogue. In order to access the decking library, select the system database button on the top toolbar. Once the database is open, navigate the database to find **2 VLI 20** which is a 2" Vulcraft composite metal deck profile. Also add the **Vulcraft 1.5 B 20** metal deck which will be used later in the tutorial for the roof. Click **Copy to Project** to add the profiles and then click **Close** and **OK**.

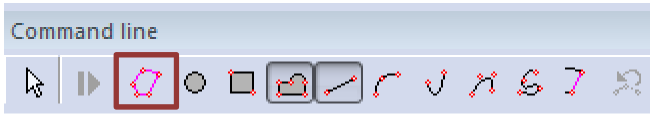
Specify the remainder of the 2D member properties as follows:

- Concrete deck material = C4000
- Thickness = 4.50" (this is the total slab thickness)
- Layer = Floor (create the new layer by clicking on the ellipsis button – 3 dots and then clicking **New** to add a new layer when the Layer dialogue opens)
    - With the layer dialogue open it is also possible to create additional layers that will be used later in the project. Create the following layers; Walls, Beams, Columns, Braces, Joists, Panels
- LCS Angle [deg] = 90°

Once the plate has been defined properly, click **OK** to continue the graphical input.



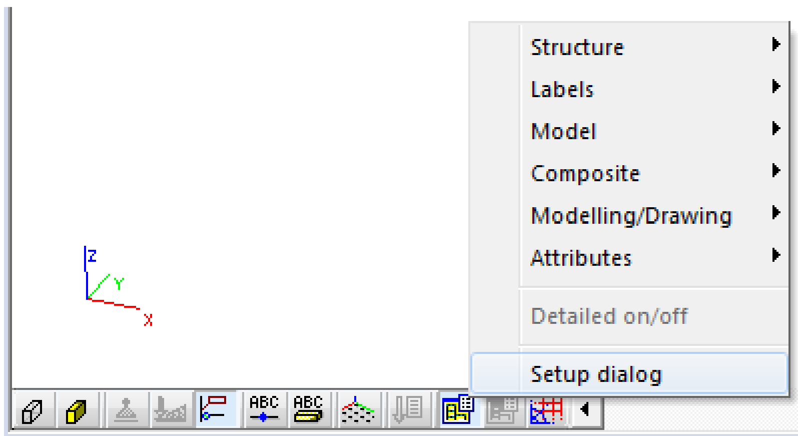
By default, when entering a 2D member, the input drawing options are available in the command line. These options include modeling tools used to enter an arbitrary slab (by sketching lines or curves) or inputing a polygon. For this tutorial, the edge of slab in the .dwg file is drawn as a polygon and therefore can be selected when using the **Existing Polygon** tool.

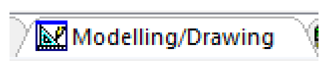


Once the polygon has been selected right click in the graphical window and click **End** or simply press the **ESC** key to complete the command.

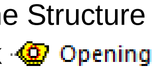
Add Opening in Floor Plate

After the floor plate has been added, it is necessary to create openings in the 2D member that correspond to the location of the elevator shaft and stairwell. Before the openings are added, it is best to modify the model's **View Parameters.** To do this, click on the **Fast adjustment of View Parameters** button located on the **View Parameters** toolbar above the **Command Line** and then select **Setup dialog.**



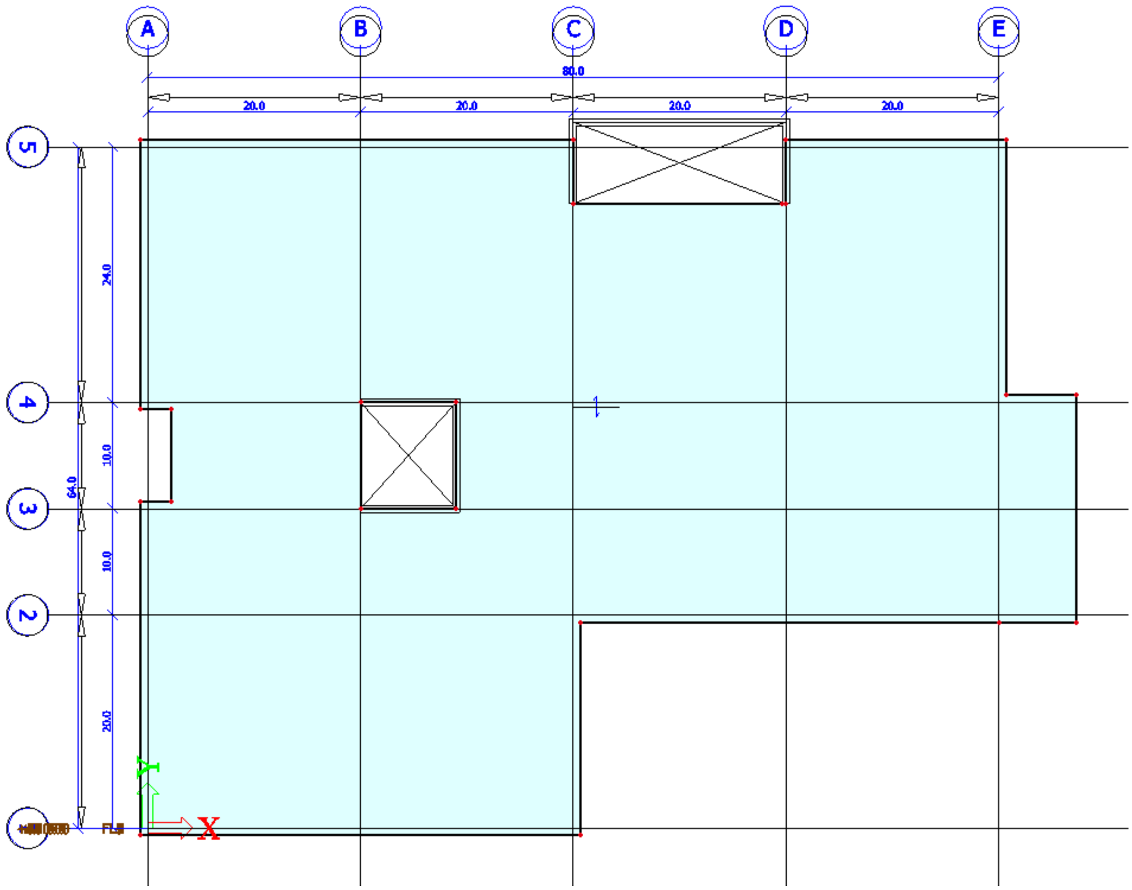
Enabled the checkbox for **Show names in tab** and then navigate to the 

tab. Next, disable the checkbox for **Grid Projection** and click **OK.** Then enable the rendering of the slab by selecting the button  on the toolbar.

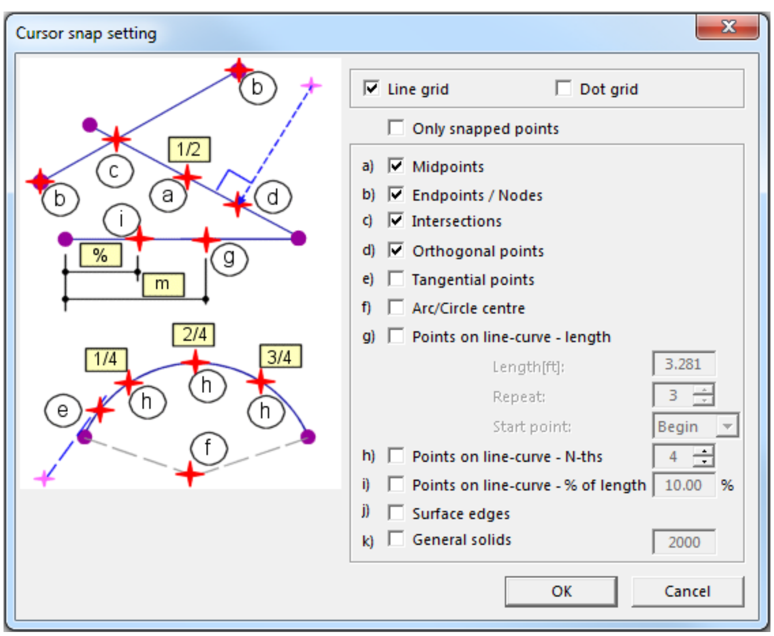
In the Structure service, expand the 2D member components field by clicking the [+] and then double click 

to begin to add the opening. Click **OK** to close the dialogue. The sketching tools for creating an opening are the same as the tools for creating a slab. By default, the **New polygon** & **straight line** input buttons are selected. Use this method of input to create the openings as shown in the picture below. End the input of the opening by pressing the **ESC** key.

Note: The extents of the opening lies on the centerline of the wall.



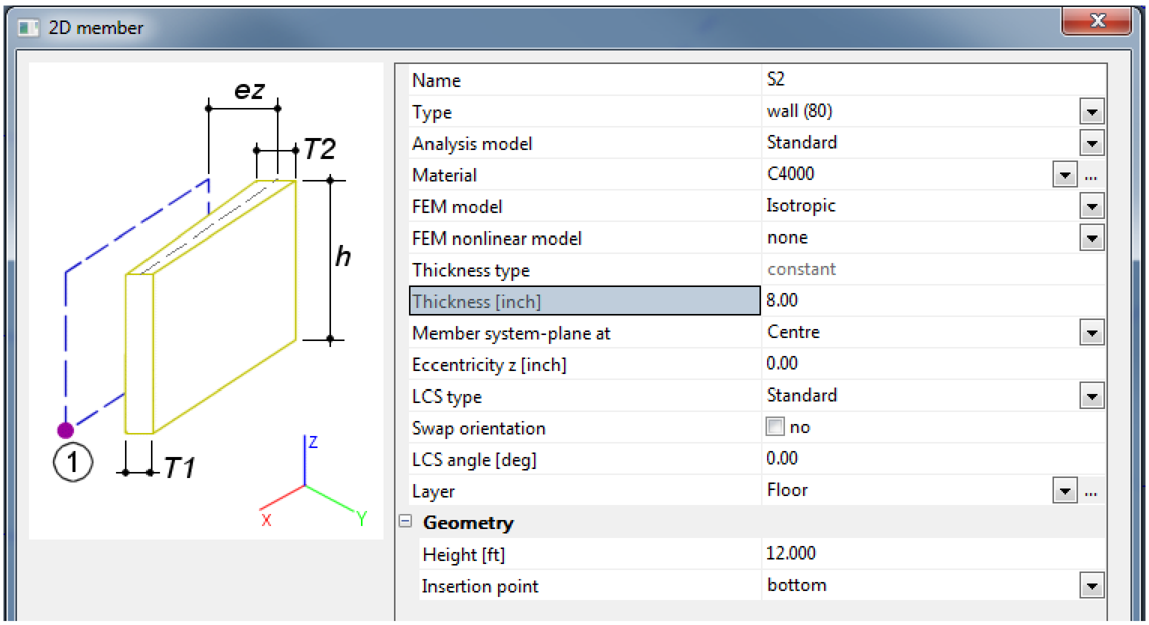
Note: The cursor snap settings (similar to object snap in AutoCAD) are used to easily select the endpoints and intersections that define the slab openings. These snap settings can be modified using the cursor snap settings button above the command line. For this tutorial, the snaps for midpoints, end points, intersections, orthogonal points as well as the ability to snap to the line grid will be used.



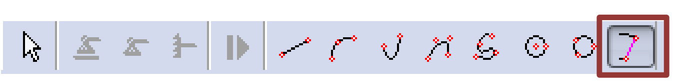
Input Concrete Walls

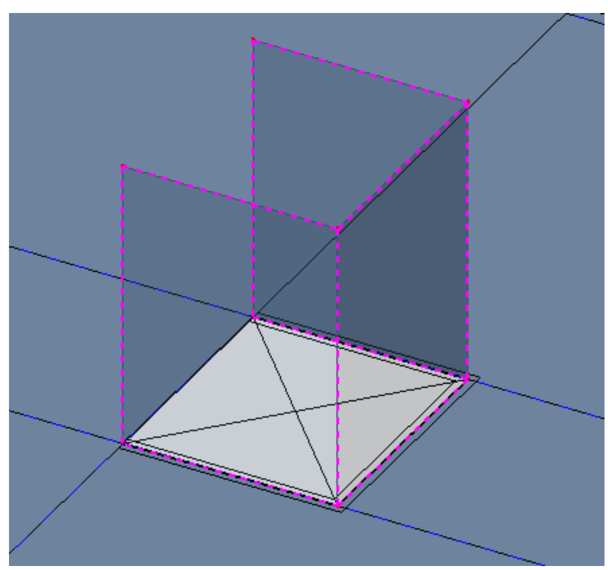
In this tutorial, there will be two concrete "core" wall elements which function as shear walls within the overall building lateral system.

To create the walls, double click on the Structure service and then click on the [+] to expand the 2D Member field. Next, double click on Wall to open the wall input dialogue. Once the dialogue appears, various properties of the wall can be manipulated. Specify the properties as shown in the picture below and then click **OK.**

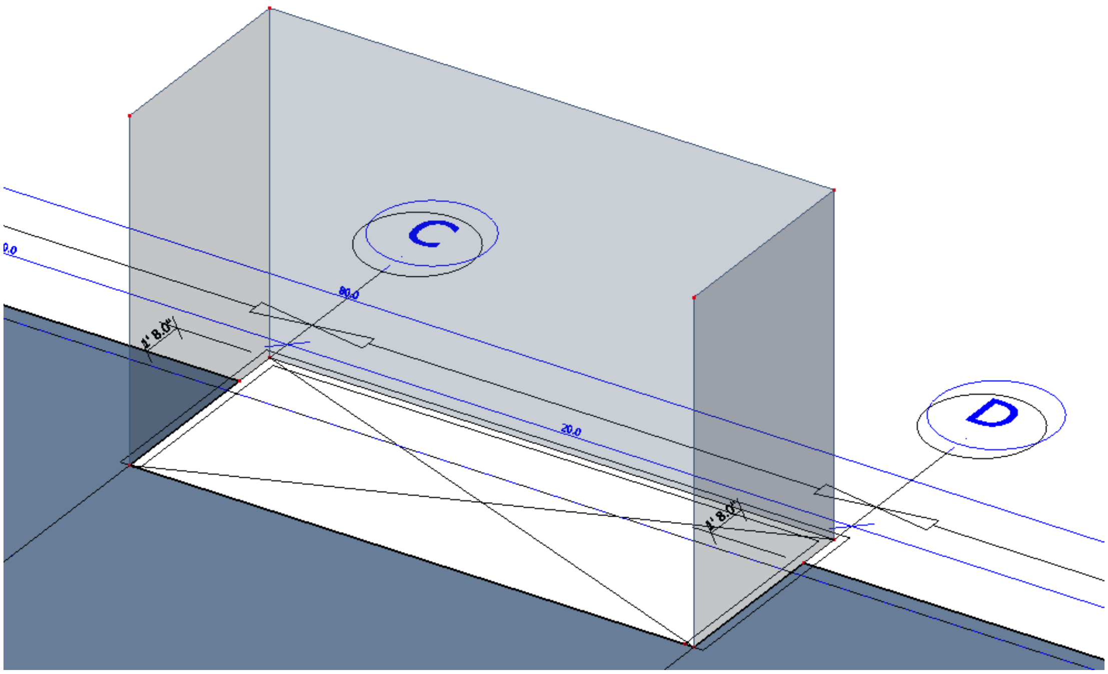


When adding the walls, be sure to enable the **REF** layer so that the .dwg drawing that was added earlier is visible. The wall systems are placed in two separate locations. First, a "C" shaped core wall can be input near Grid Line B such that the walls follow the edges of the opening (as shown in the picture below). Use the **select line** input to directly select the necessary opening edges for wall input.





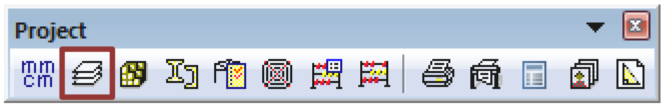
The second “C” shaped core wall is input between Grid Lines C & D. Again, the walls should follow the opening edge (along Grid Lines C & D) and along these lines should extend 1’-8” beyond the edge of the slab. The .dwg drawing can also be used to properly add the walls as shown below.



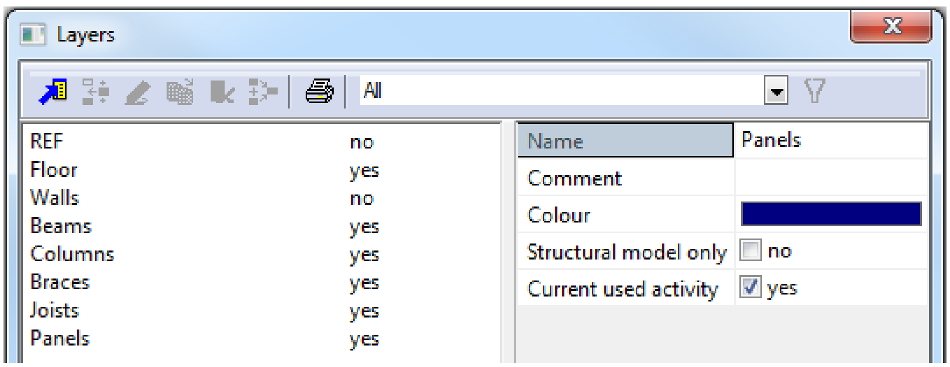
When the input is finished, press **ESC** to finish the command.

Input 1D members (columns, beams & joists)

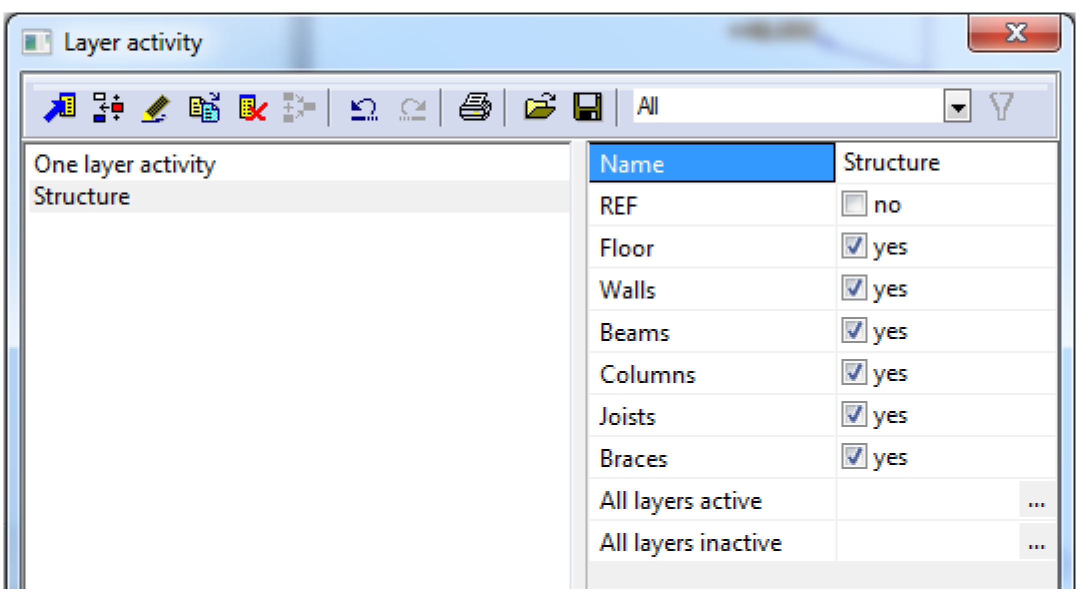
Before adding 1D members, the creation of additional layers that can be used to filter newly added elements (columns, beams, joists, braces, walls, etc.) can be created. To do this, select the **Layers** button on the Project toolbar.



Within the layers dialogue, create the following new layers:



With layers created, **Activities** can be used to filter the model. To enable the **Activity by Layers,** click the button found on the Activities toolbar. With the layer activity dialogue open, disable the check box next to the **REF** layer so that the current activity has only structural objects enabled. Click **Close** to return to the graphical window.



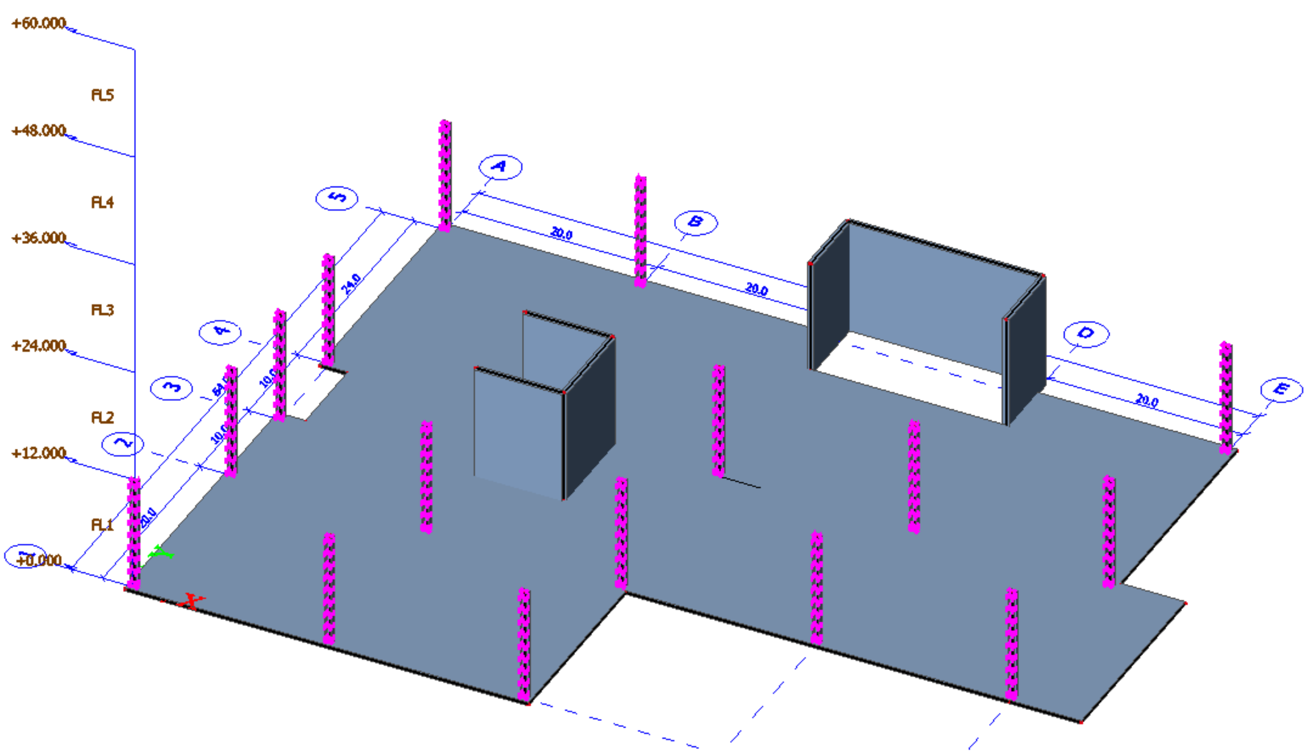
Adding Columns

From the **Structure** service, click on the [+] to expand the 1D member field and then double click on Column. Since there are currently no cross sections added to the project, the **New cross-section** dialogue will appear. Within the new cross-section dialogue, various types of cross sections (steel, concrete, thin-walled, etc.) can be added from within the available groups. For this tutorial, the **Profile Library** group will be used to add all necessary steel cross sections.

To add the wide flange cross section, select the profile shape, , group **W(Imp)** and size **8x31** and then click the **Add** or → button. The remainder of the cross sections for the tutorial can be added either manually or more efficiently by importing a database file (.db4). To import the database file, click the **Open** button in the **Cross-Sections** dialogue.

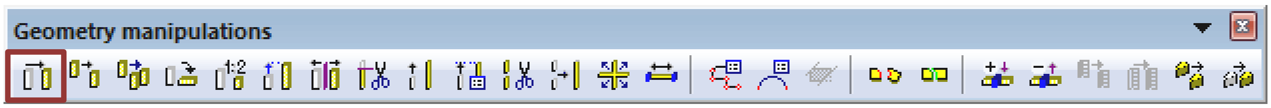
Next, navigate to the folder that contains the database file that was included with this tutorial, select the file and click **Open**. The **Read from database** window will appear and all members can be added to the project by clicking the << Copy all button. Click **Close** to finish the import. With the import finished, the cross section library can be closed to continue the column input. To finalize the input, select **Col1 – W8x31** as the desire column cross section and also set the Length to **12ft.** Using the line grid intersections as the insertion points, add the columns (16 in total) in the graphical interface as shown in the picture below. When the input is complete, press **ESC.**

<u>Note:</u> Columns are not added at C3, D3 or E3



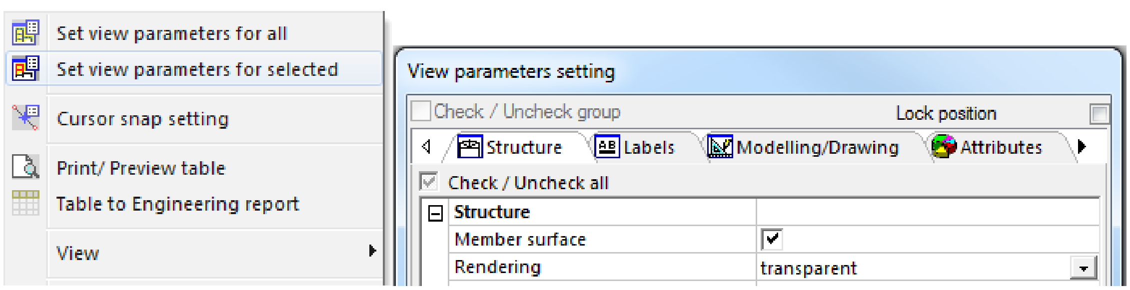
Adding Beams

Before adding beams, it is necessary to move the composite floor to the appropriate elevation. To do this, select the **Move** button on the **Geometric Manipulations** toolbar. After clicking the move button, select the entities to be moved, in this case, the composite floor plate. Finish the selection by pressing **ESC**. Next, select the base of any column as the **Start** point for the move and the top of the same column as the **End** point of the command. The composite floor plate should now be at the elevation which corresponds to the top of the columns and walls (12ft).



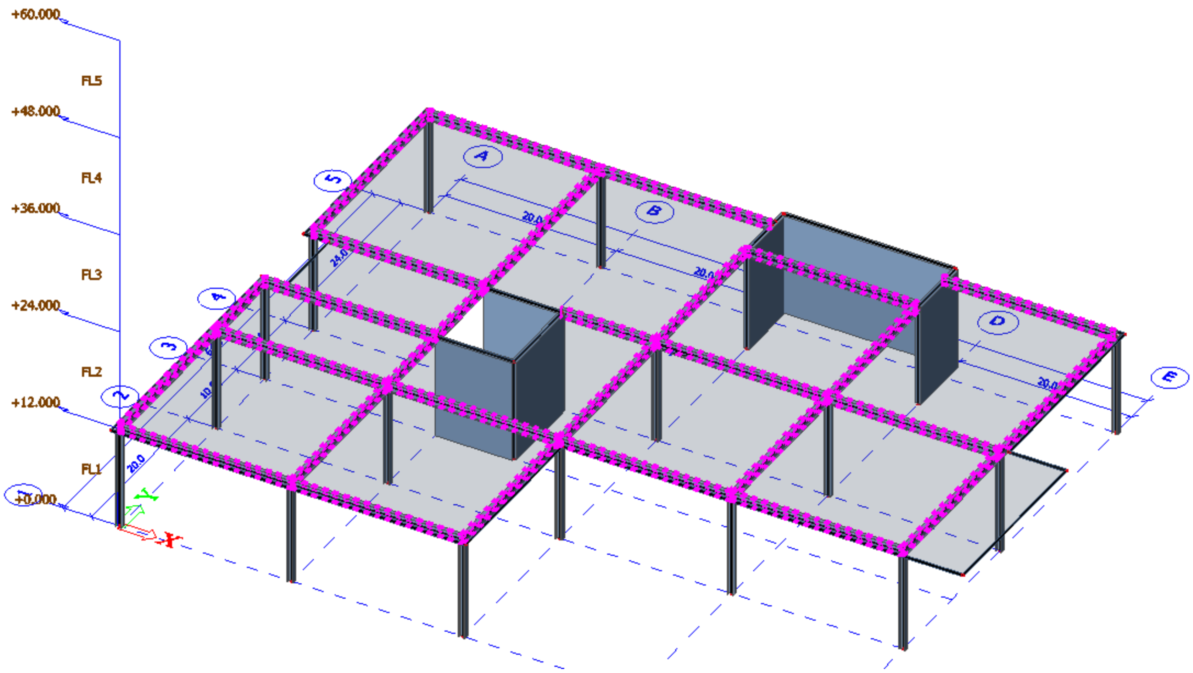
Drawing beam elements in the graphical window can be accomplished using two 1D member input options. When using the Beam functionality, the length and direction (X or Y) of the member must be defined from within the dialogue. Alternatively, modeling with the Member functionality allows all input to be done within the graphical window.

For this tutorial, it makes sense to use both input functionalities. To make input easier, select the composite floor plate and then **right click** in the graphical window. In the drop down menu that appears select **Set view parameters for selected.** Then under the **Structure** tab change the setting for **rendering** to **transparent**.

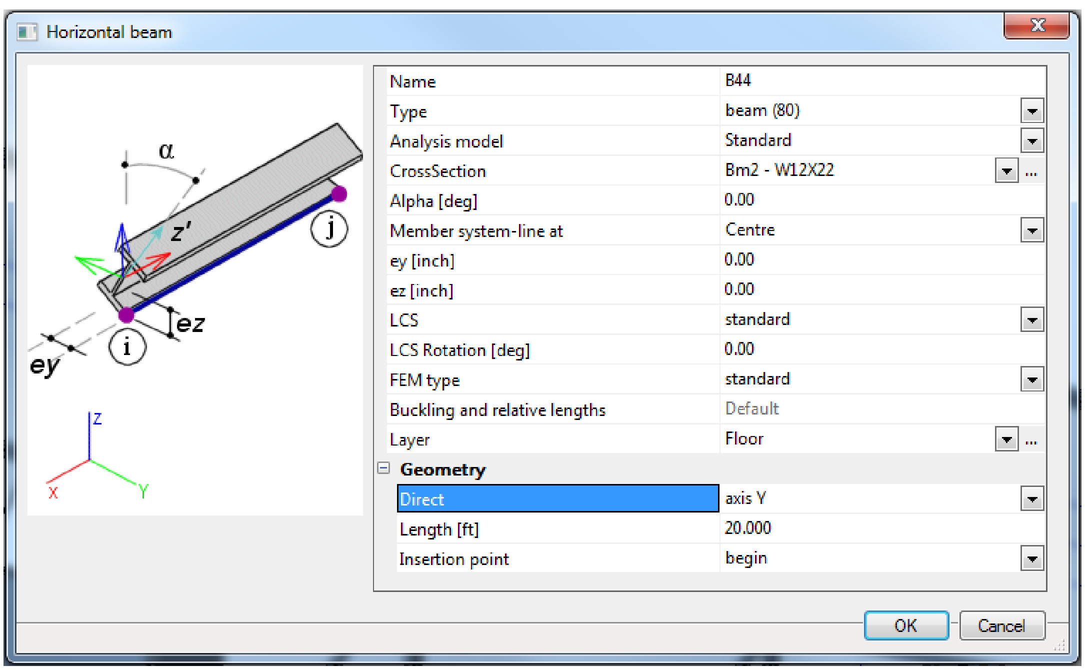


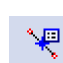
To begin the input of beams, use the Member input to connect the top nodes of all columns using cross section **Bm3 – W14x26**. Additionally, some beams will connect to the core walls at either **endpoints** or **orthogonal points** (at C5 and D5).

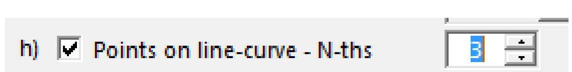
When the input is complete, there should be 28 beam members created as shown in the picture below.



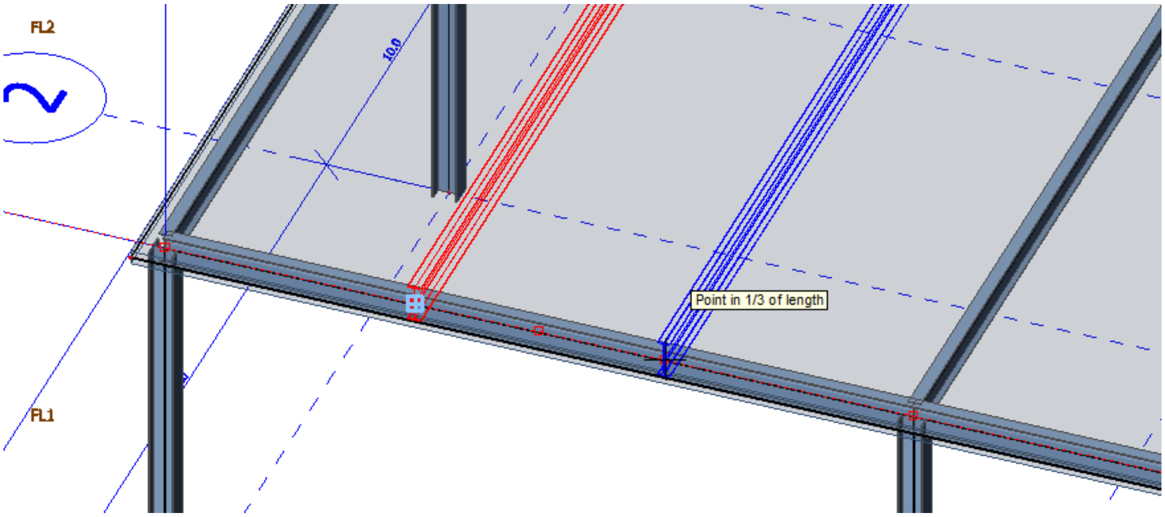
The Beam input can now be used to input the majority of the secondary framing members. With the **horizontal beam** dialogue open, set the properties as shown in the picture below and click **OK.**



Before input, access the cursor snap settings by clicking the button  on the command line and enable **the points on line-curve – N-ths snap setting**



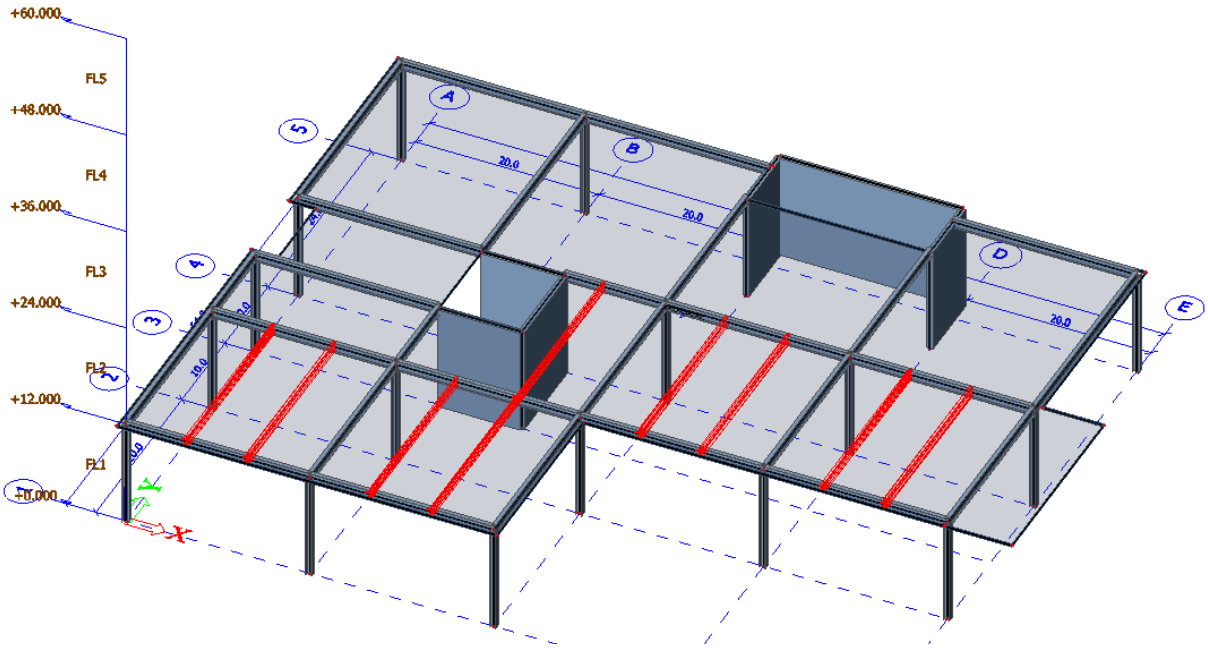
This snap setting creates 1/3 points along each member so that the secondary framing members can be easily created.



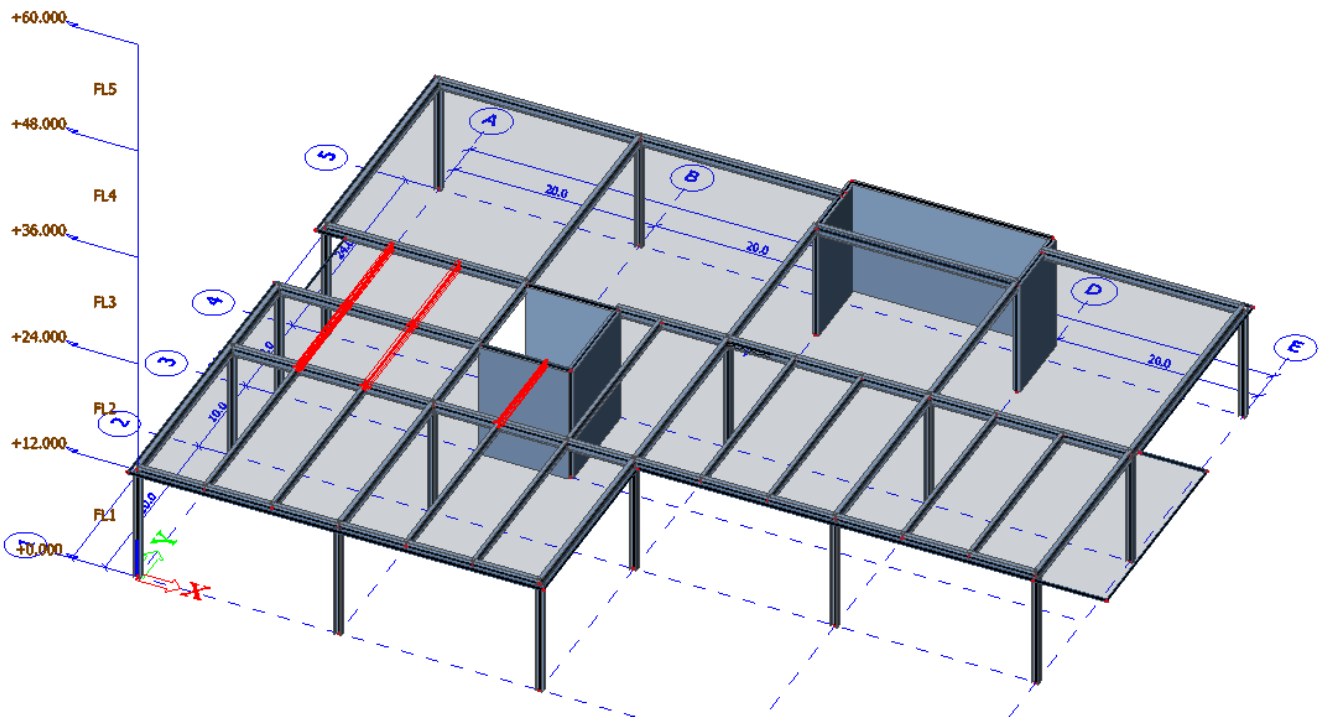
Note: Since the beam input functionality requires a length to be specified in the dialogue, the input using the beam functionality is different from the input using the member functionality in that only the **start point** of the element needs to be specified in the graphical window.

Follow the steps between to input the remainder of the floor framing.

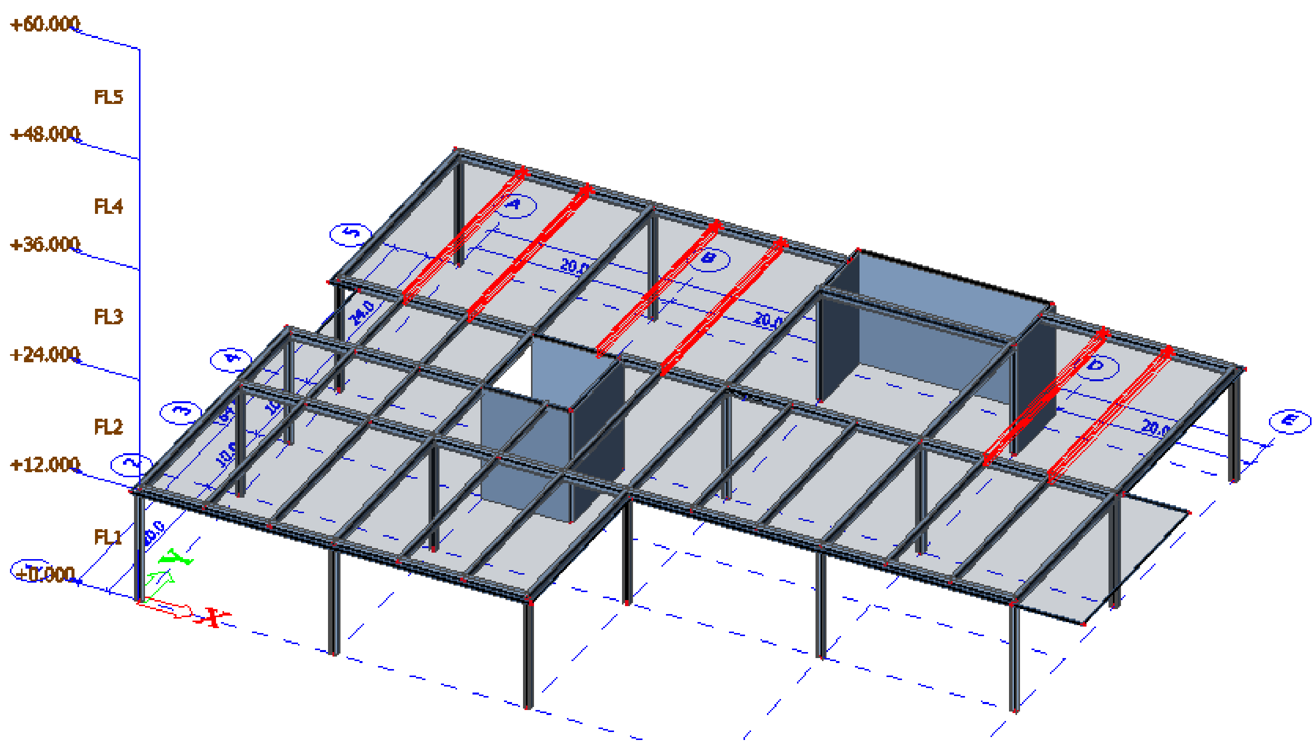
Input #1: Beam Input, Bm9 – W12x26, Y direction, 20ft, begin insertion point (9 members)



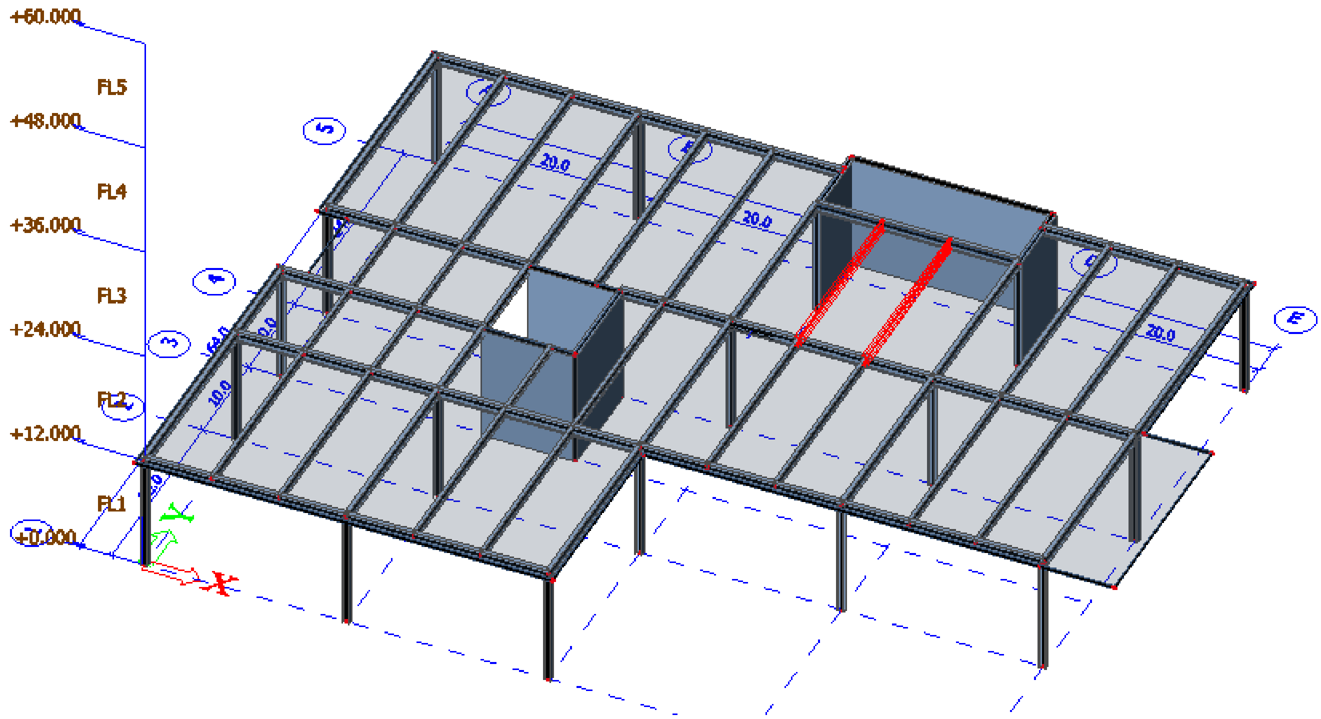
Input #2: Beam Input, Bm1 – W10x19, Y direction, 10ft, begin insertion point (5 members)



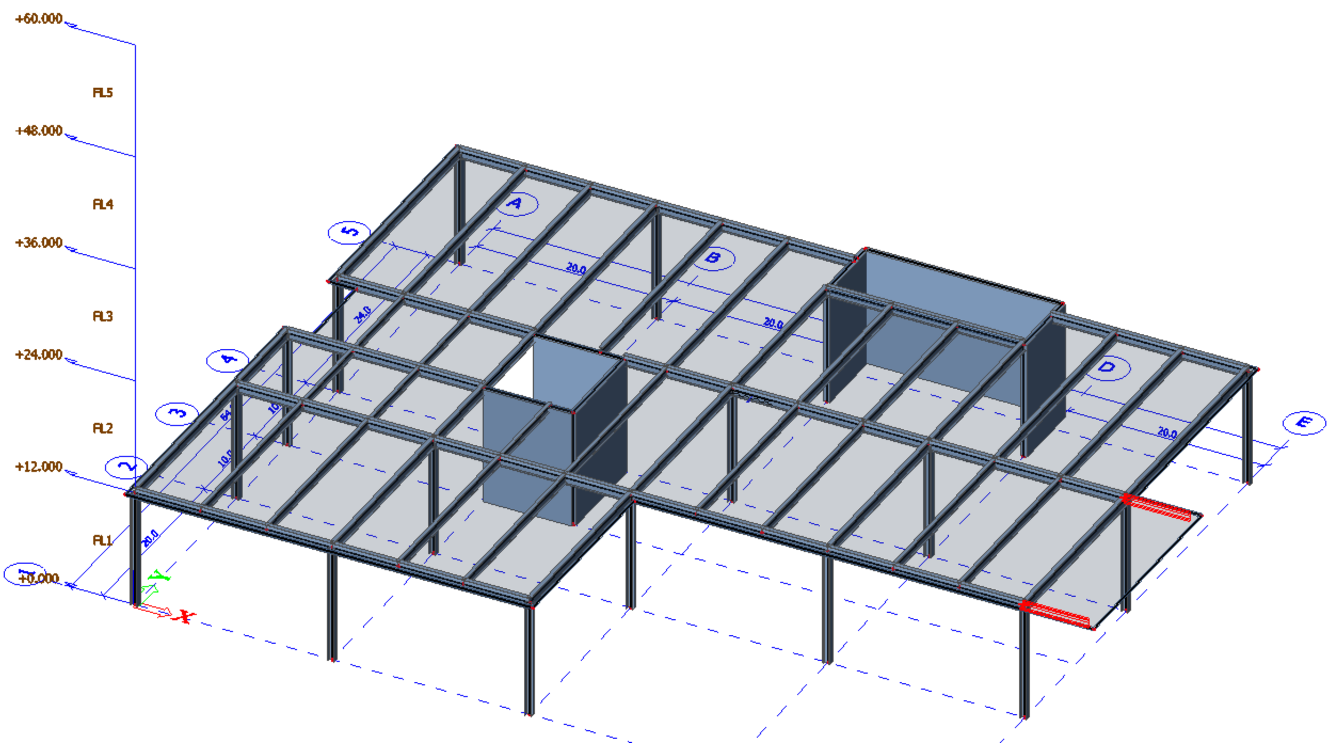
Input #3: Beam Input, Bm4 – W14x22, Y direction, 24ft, end insertion point (6 members)



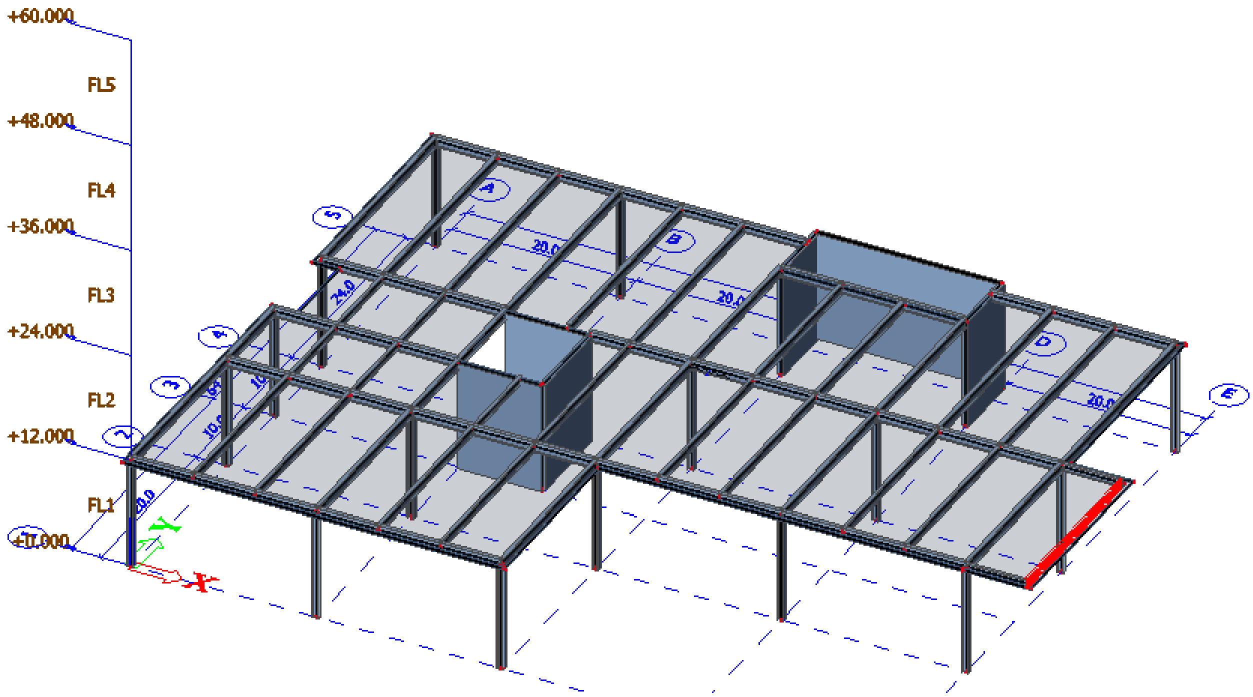
Input #4: Member Input, Bm9 – W12x26, snap from beam end point to wall orthogonal point (2 members)



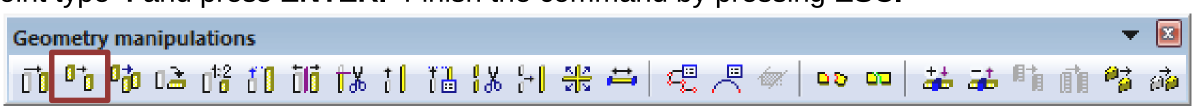
Input #5: Beam Input, Bm6 – W10x17, X direction, 6.667ft, begin insertion point (2 members)

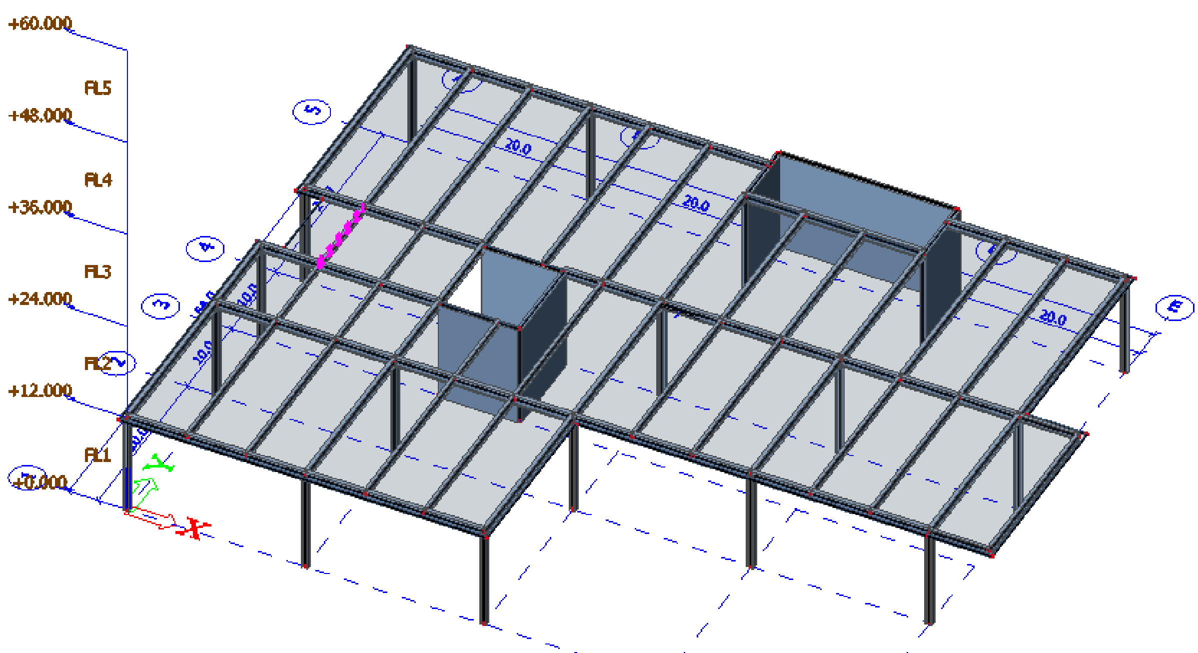


<u>Input #6:</u> Member Input, Bm2 – W12x22, snap from end point to end point (1 member)

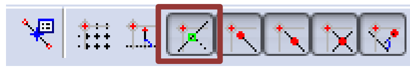


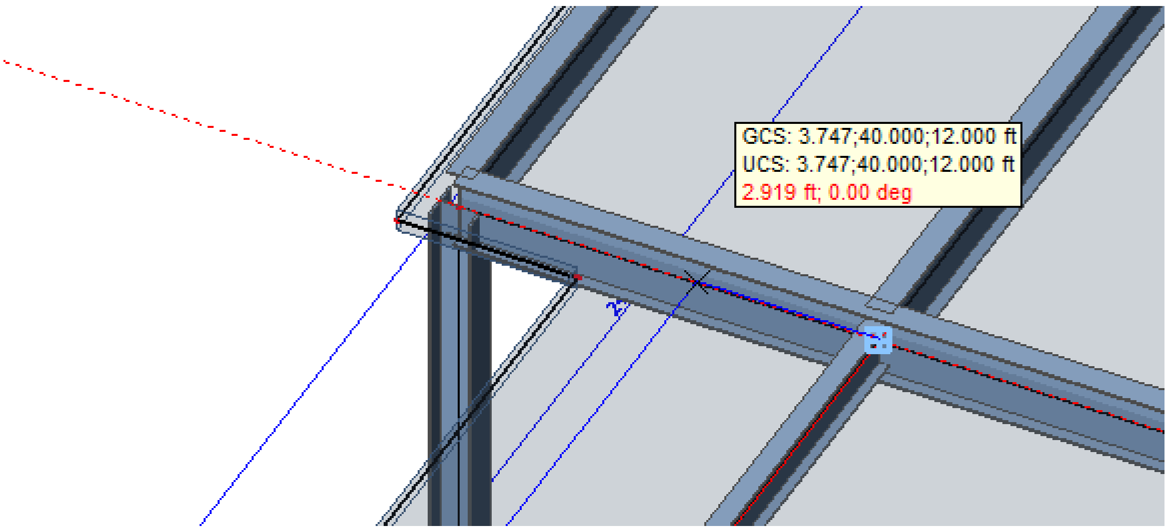
For the final input, use the **copy** command on the toolbar and select the member shown in the picture below. With the member selected, select the start point as one of the nodes of the beam and for the end point type **4** and press **ENTER.** Finish the command by pressing **ESC.**





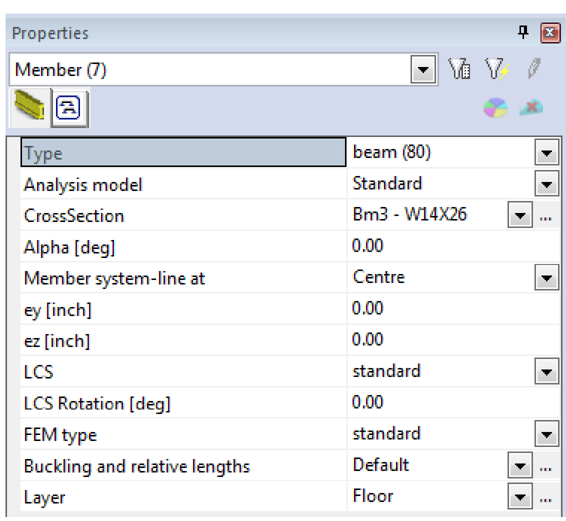
Note: When using the copy command, make sure the tracking mode for the graphical window is enabled. This will enable the tracking lines when using modeling commands. As shown in the picture below, the red tracking line represents the x direction, meaning that typing in the distance for the copy while the tracking line is present will place the new member on that same line.



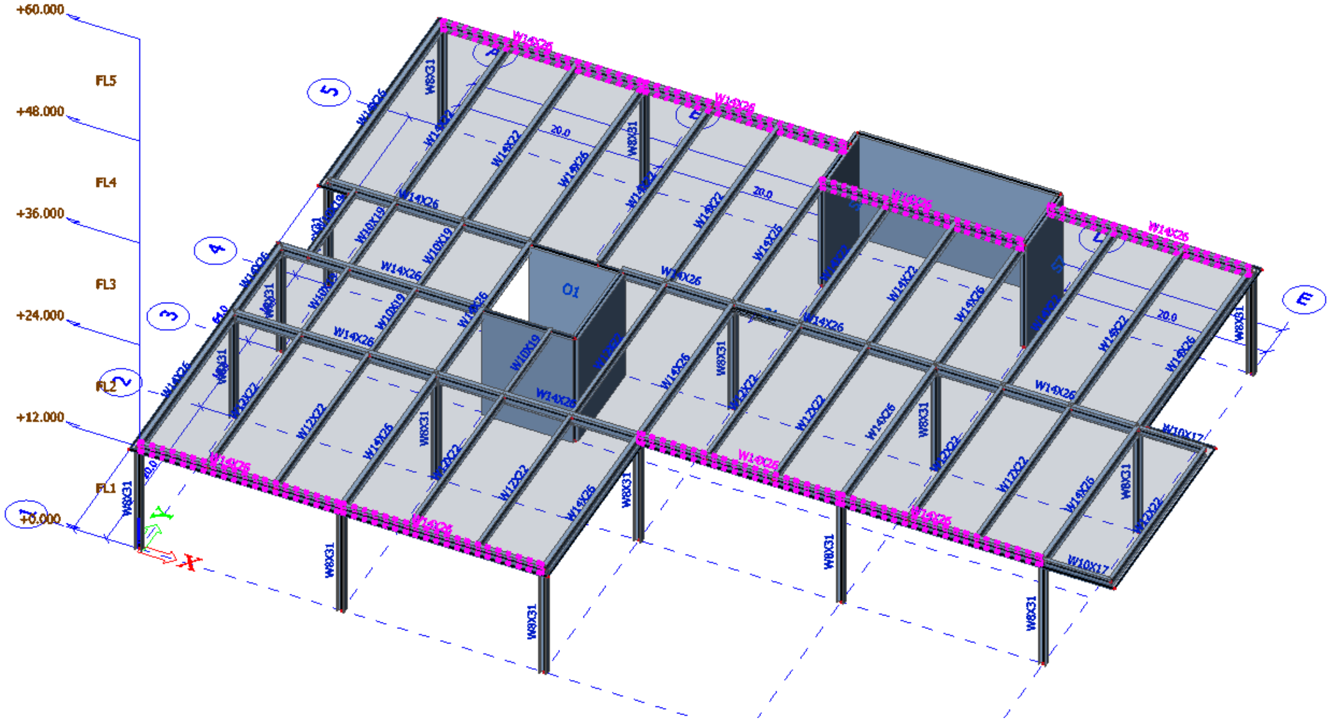


With the modeling of the floor members complete, enable the member labels by selecting the button on the **view parameters** toolbar above the **command line.**

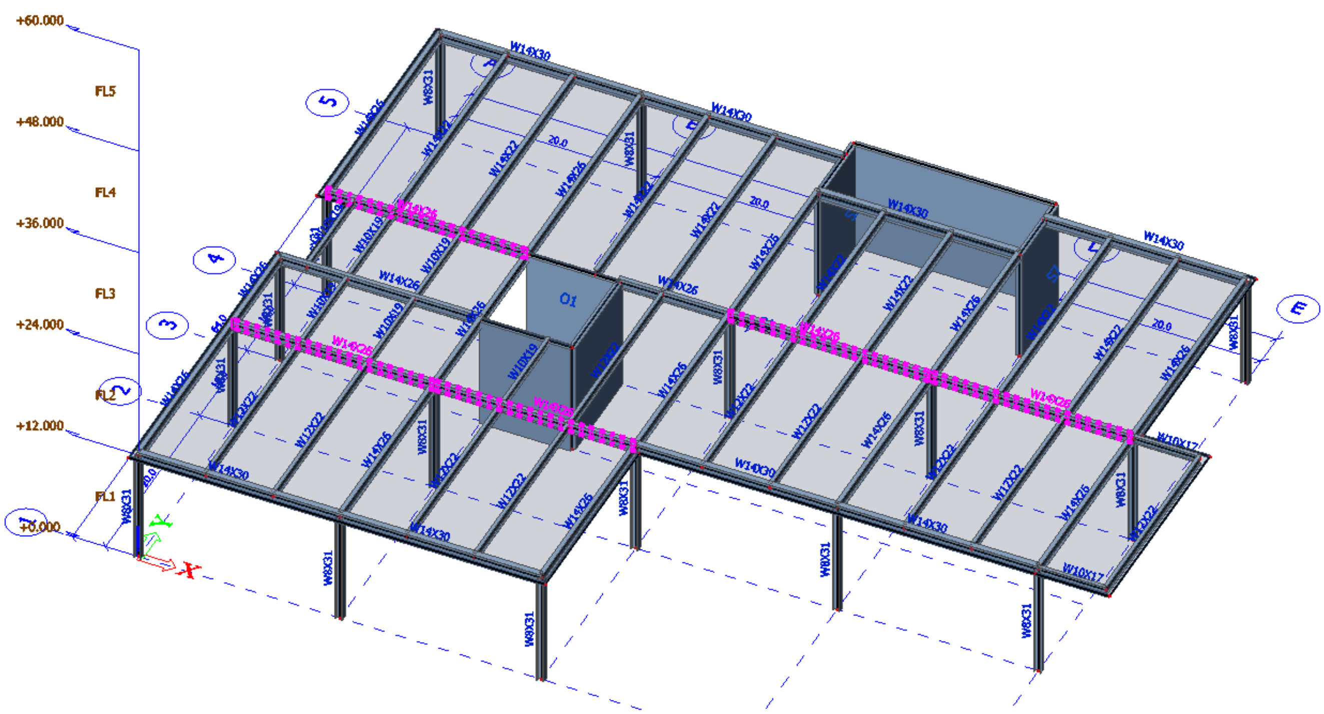
To switch the labels from the beam number to the cross section type, **right click** in the graphical window and select **set view parameters for all.** Next, choose the **Labels** tab and change the active beam labels checkbox from **name** to **cross section type**. With the labels active it is easy to make member cross section changes from within the **Property** window.



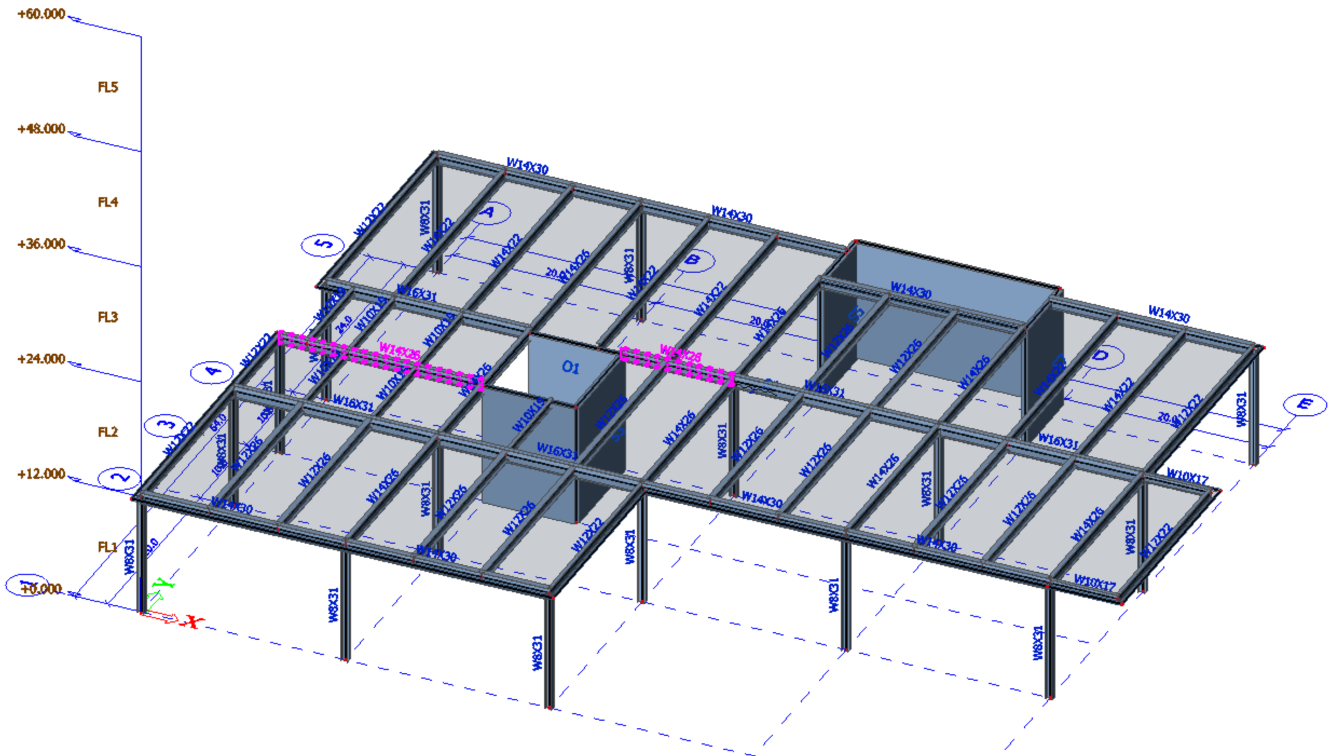
First, select the 8 exterior girder members and change the **CrossSection** from Bm3 – W14x26 to Bm7 – W14x30.



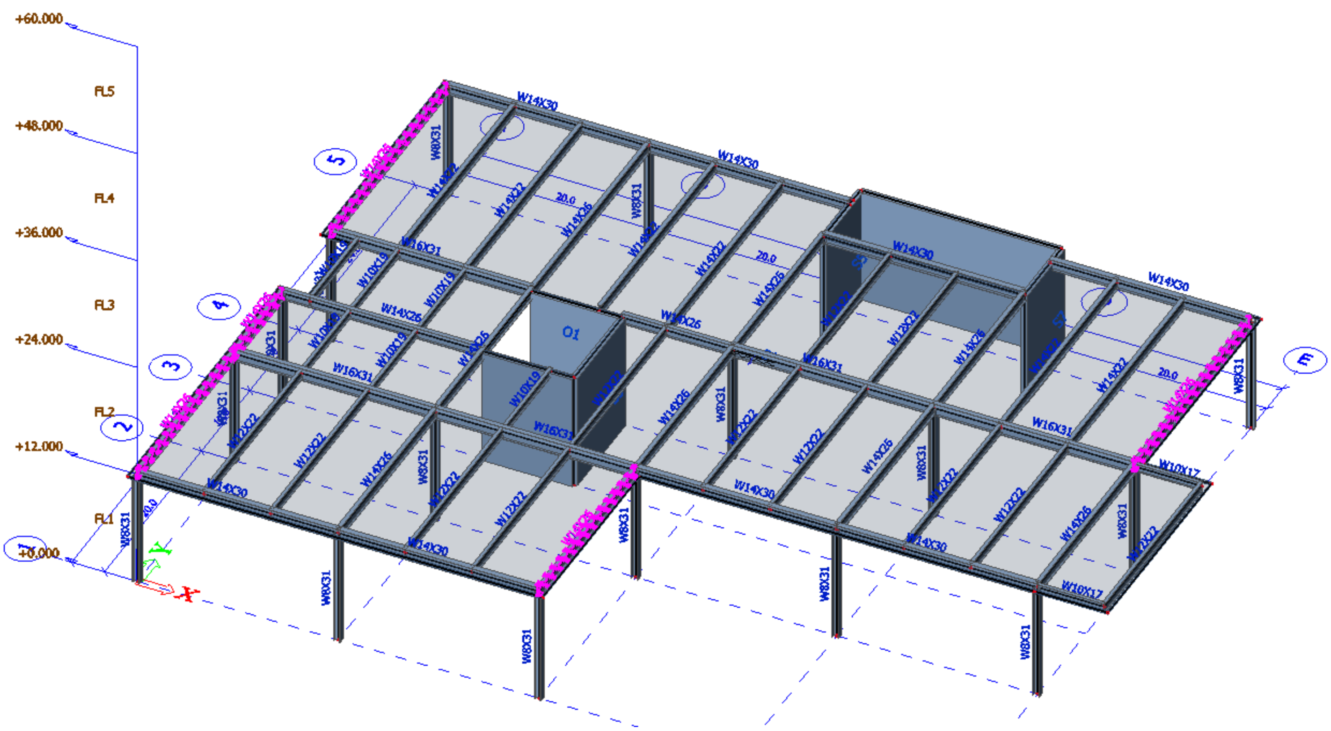
Next, select the 5 interior girder members and change the **CrossSection** from Bm3 – W14x26 to Bm8 – W16x31.



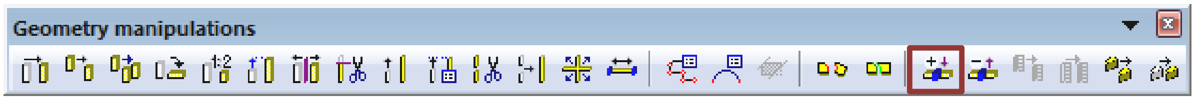
Next, select the 2 interior girder members and change the **CrossSection** from Bm3 – W14x26 to Bm5 – W14x22.



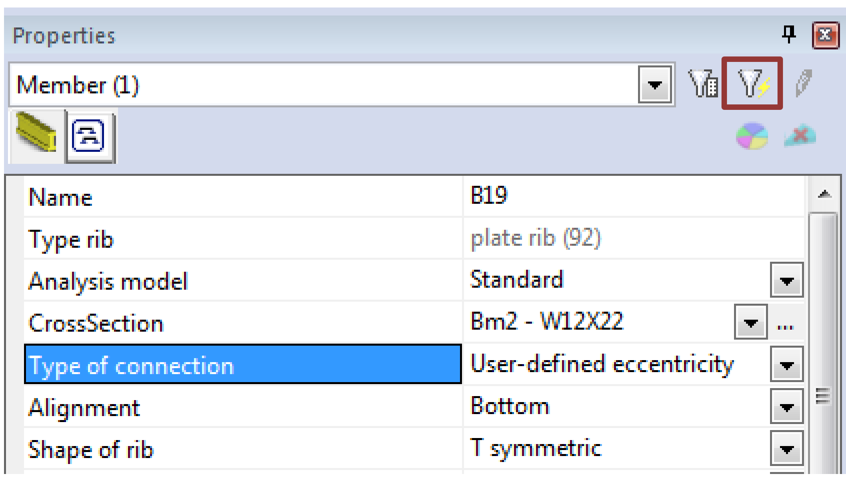
Select the 5 exterior beam members that frame into columns and change the **CrossSection** from Bm3 – W14x26 to Bm2 – W12x22.



With the modification of all the member sizes complete, the beams can now be connected compositely to the floor plate. To do this, select the **Connect Member/Nodes** functionality on the **geometric manipulation** toolbar and click **OK** to proceed with all entities.



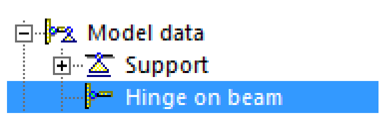
Within the setup for the connection of elements, enable the checkbox for **Connect 1D members as ribs** and then click **OK.** The beams are then switched to member type, **plate rib** and are connected to the composite floor plate. After the connection, modifications should be made to the properties of all composite members. To make changes for all members at the same time, select any plate rib member and then click on the **Type of connection** property (highlighted below in blue below). With the property selected, click on the **select elements by property** button, which is defined by a lightning bolt as seen below. This will create a selection that includes 54 beam elements.



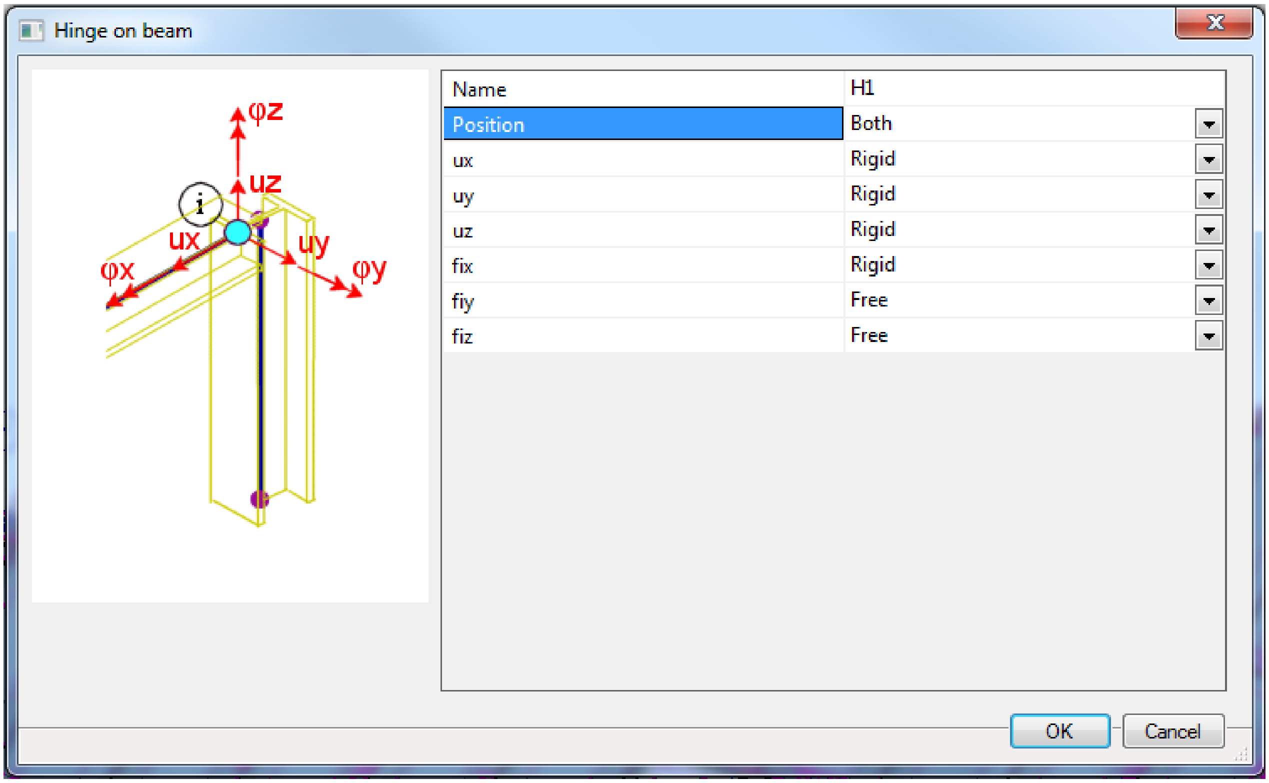
With all 54 beam elements selected change the **type of connection** from **user-defined eccentricity** to **with standard composite action**. With the selection still active, change the property for **shape of rib** from **T symmetric** to **automatic.** When the modifications are complete, press the **ESC** key to end the command.

<u>Note</u>: The setting for automatic shape of rib will automatically calculate the appropriate effective width of each beam element. The setting for standard composite action is used to eliminate axial forces caused by eccentricity between the floor plate and connected beam element.

With the composite members added, it is also necessary to add hinges on the beams to eliminate any moment transfer at the connection points. To do this, first make sure all 54 beam elements are selected, then navigate to the **Hinge on beam** functionality found in the **Structure** service under **Model data**.



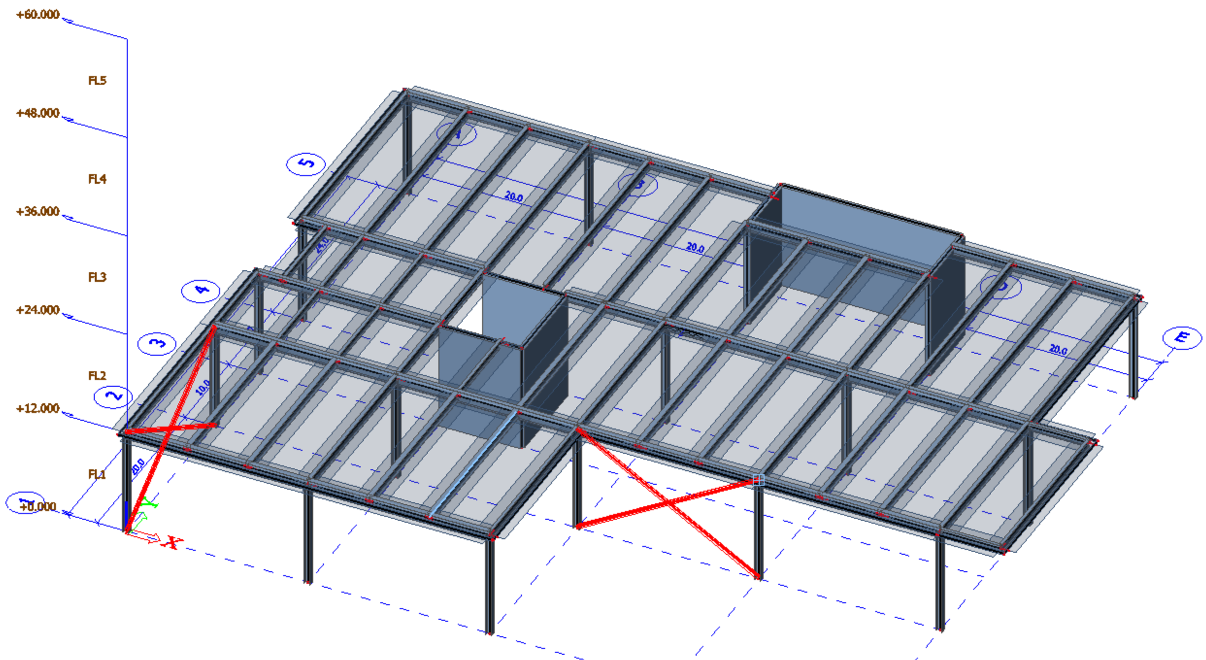
Once the hinge on beam dialogue opens, set the properties as shown in the picture below and click **OK** to add the hinges.



Note: Hinges can be viewed graphically by enabling **Show/Hide other model data** from the view parameters toolbar.

Adding Lateral Braces

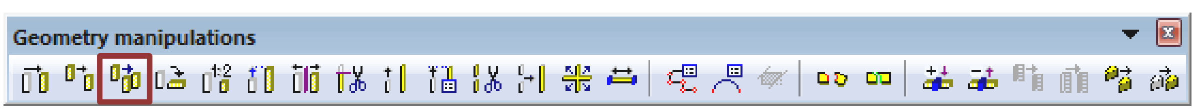
In addition to shear walls, the lateral system will include braced frames on all floors. The braced frames are modelled using the Member functionality and cross section Bm10 – HSS4x4x1/4. Add the two x-braces along Grid Line A (between 1 & 2) and along Grid Line 2 (between C & D).



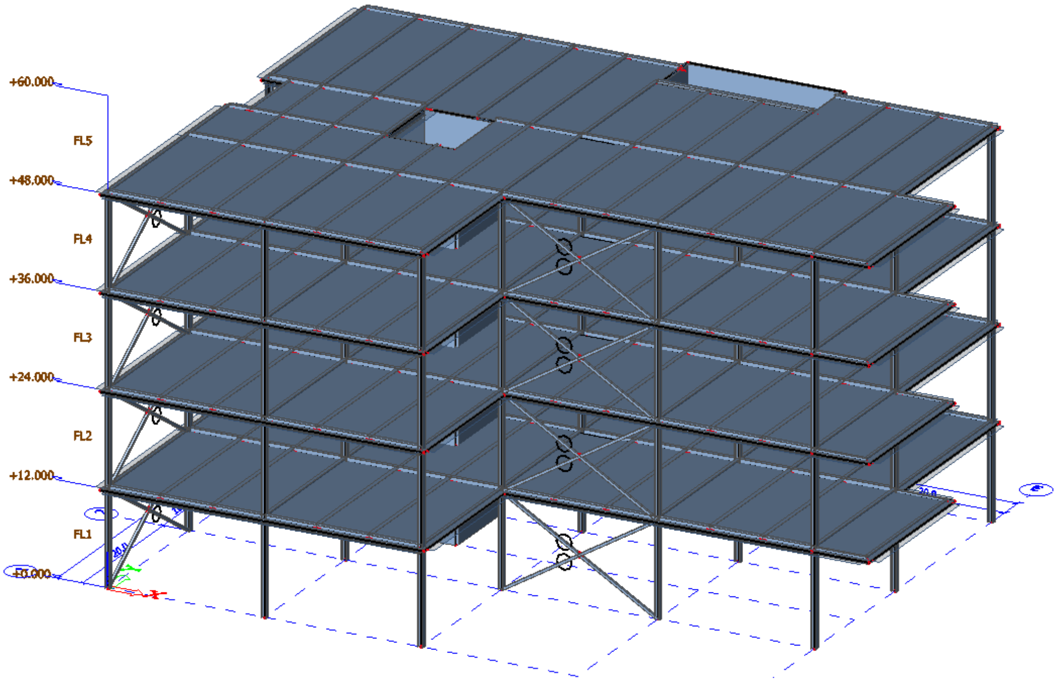
Additionally, with both braces still selected double click on Cross-link found in the **Structure** service under **Model data**. Cross-links are added in order to create a rigid, pinned or coupled connection between two bracing members. Once the cross link dialogue opens, change the **Type** to **hinged** and click **OK.**

Creating elements on additional stories

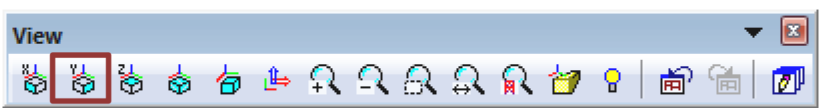
Once the walls, column, beams and braces are all added to the model, the elements can be copied to the other floors. To do this, select all the elements in the model and click the **multi-copy** button on the **geometric manipulation** toolbar.

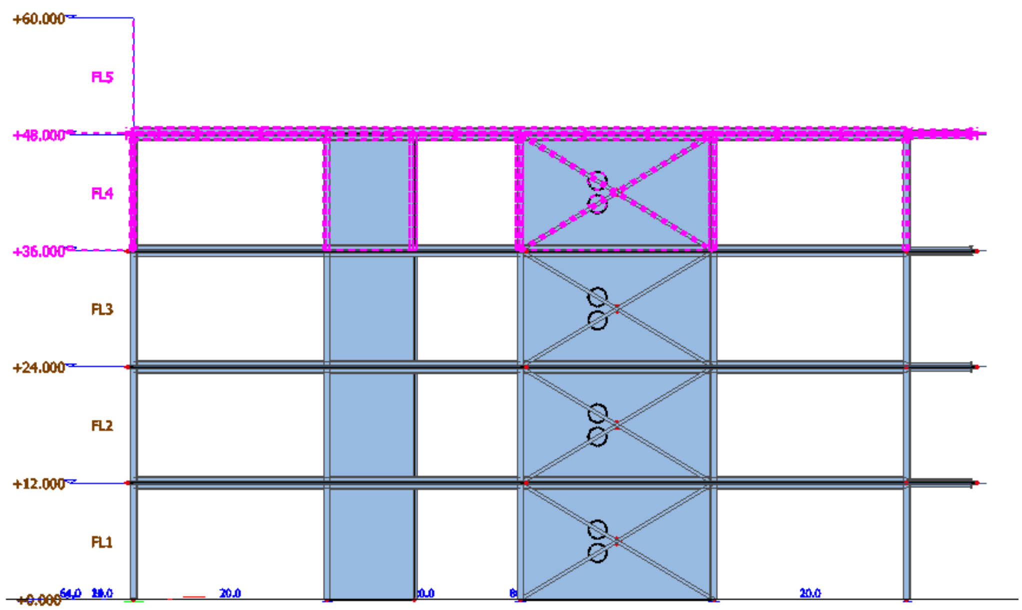


With the multi-copy dialogue open, set the **number of copies** to **3**, disable the define **distance by cursor** checkbox so that the value for **z** can be set to **12ft**. When the properties of the multi-copy are set, click **OK**.

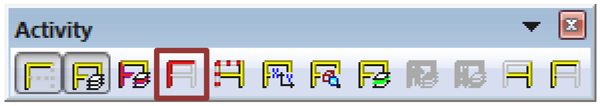


Next, click the **View in direction Y** button on the View toolbar to change the 3D view of the model and select all column, brace, wall and floor elements on the 3rd level as shown in the picture below. Do not select the floor beams, they will be individually selected in the next step.

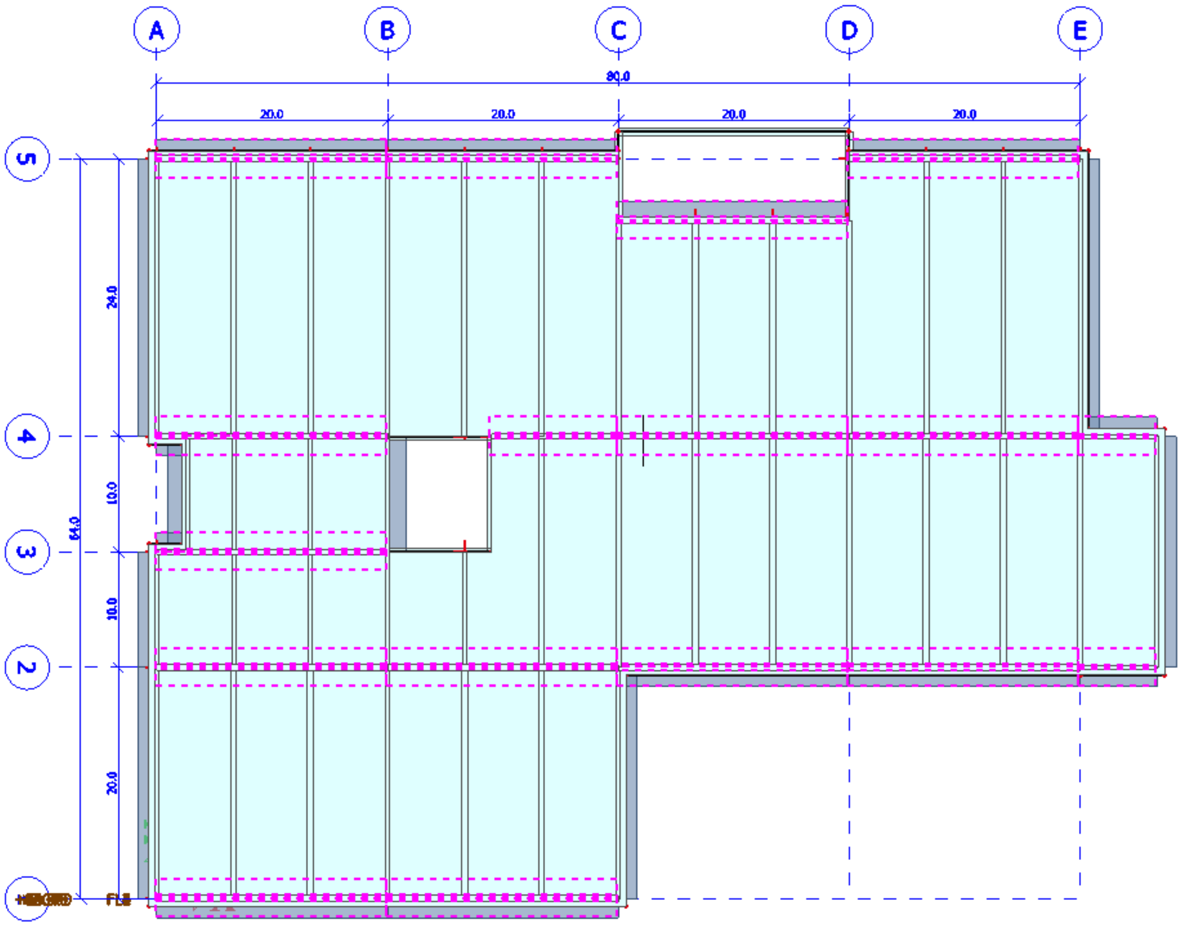




Use the **activity by selection** functionality, found on the **Activity** toolbar to filter the selection so only the currently selected elements are visible in the graphical window.



Select the following beam members (17 total), in addition to all columns, braces, walls and the floor plate visible.



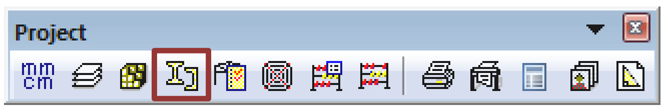
Use the multi-copy command and set the **number of copies** to **1**, disable the define **distance by cursor** checkbox so that the value for **z** can be set to **12ft**. When the properties of the multi-copy are set, click **OK**.

When the copy is complete, use the **activity by selection** functionality to filter the selection so that only the elements on the 5th floor are visible.

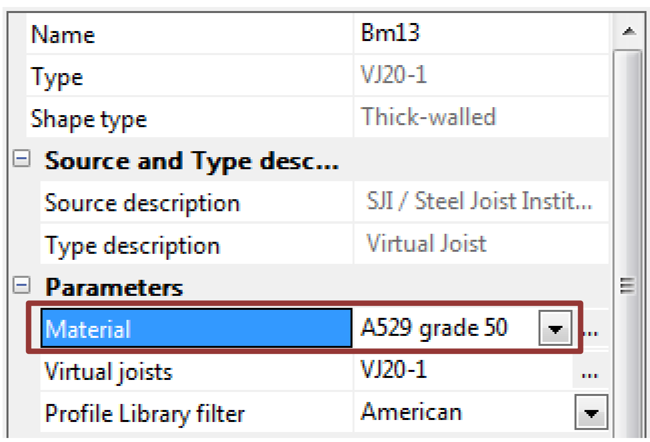
Adding roof joists

Most composite steel buildings utilize metal roof deck and steel bar joists as the main structural system for the roof. In this tutorial, **virtual joists** will be added to complete the roof framing. Virtual joists are 1D member elements created by the Steel Joist Institute for use in finite element analysis software. For more information concerning virtual joists and their implementation within SCIA Engineer, refer to the [Virtual Joist Manual.]

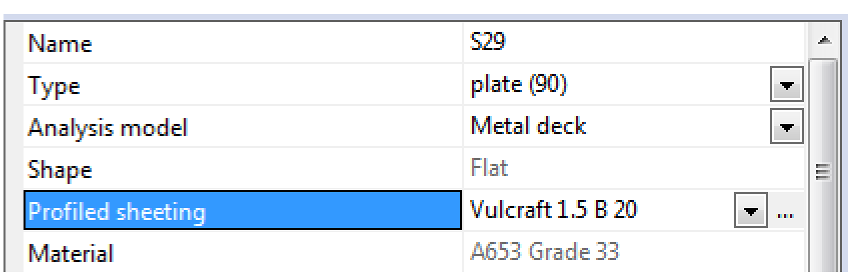
Before modeling the joists, new **virtual joist** cross sections need to be added to the library. To do this, click the **cross section library** button on the **Project** toolbar. When the dialogue of active cross sections appears, click **New**.



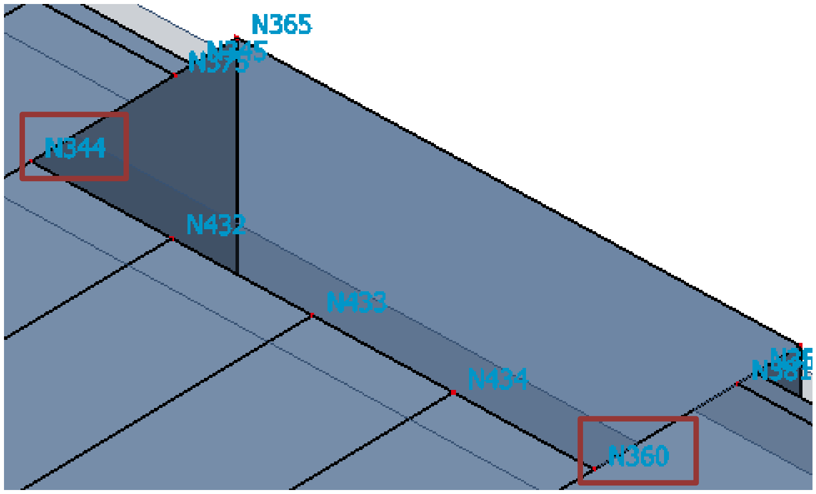
At the bottom of the list of available groups select Virtual joists and then click the [+] button to expand the **VJ** group of items. The items in the VJ list are separated by joist depth. For this tutorial add three separate VJ members: **VJ10-1, VJ16-1 and VJ20-1**. When each joist cross section is added, make sure to switch the base steel material to **A529 Grade 50**.



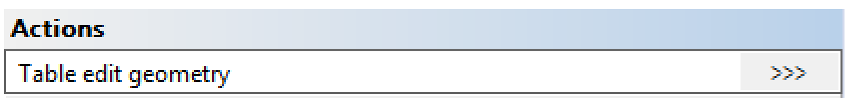
Before the joists and other roof framing members can be added the decking type and extents of the plate need to be modified. First, select the plate that was copied to the roof level and change the analysis model type and profiled sheeting to match the picture below.



Next, the opening for the stairwell needs to be eliminated. This can be done by adding a second plate or by manipulating the geometry of the slab directly. Before choosing to manipulate the slab geometry, it is important to understand which nodes need to be modified. This can easily be done by enabling the **node labels** from the **view parameters** toolbar.



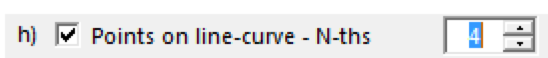
Once the nodes are identified, it is possible to select the roof plate and click the **Table Edit Geometry** button found in the **Actions** list at the bottom of the **Properties** window.

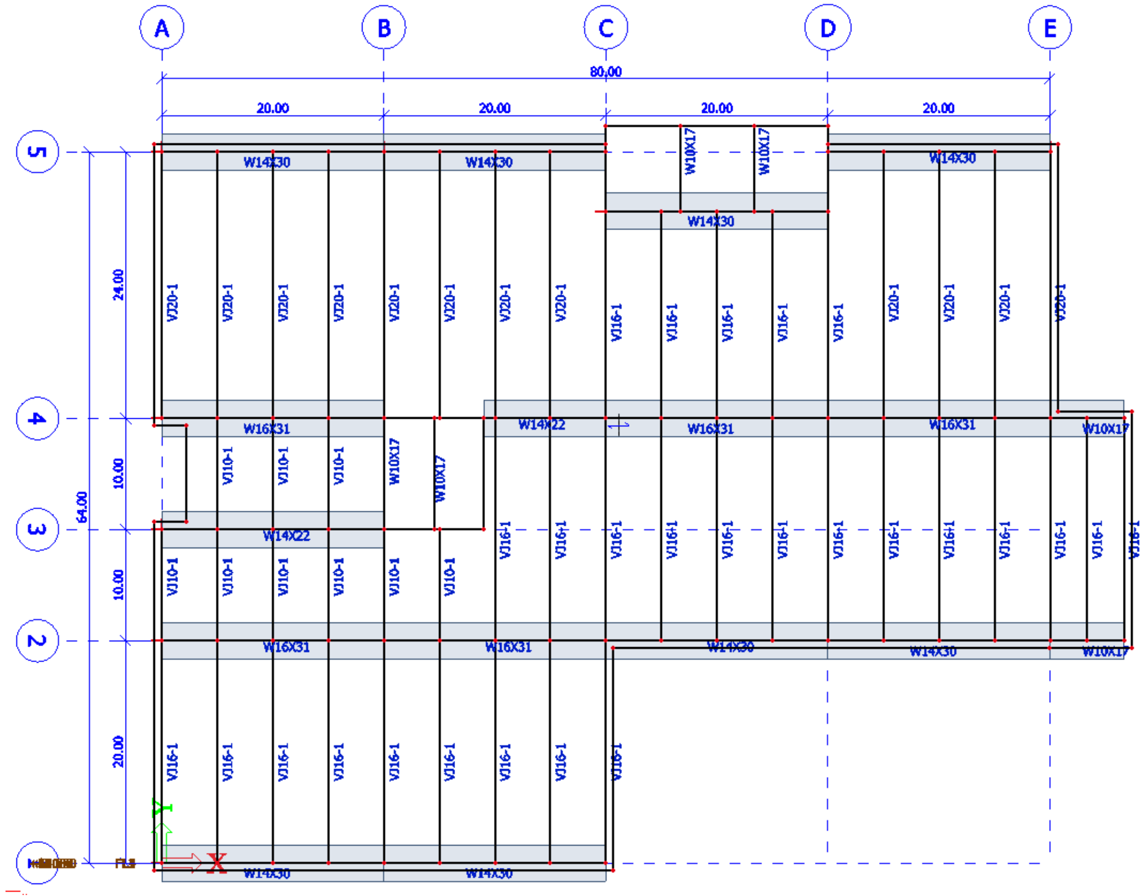


The two nodes that need modified can be easily found by sorting the table for editing geometry by clicking the **Y [ft]** column**.** Both nodes have a Y value of **58.6667 ft**, which needs to be changed to **66.3333 ft.** When the change has been made, click the **Apply** button and then click **OK**.

Once the roof plate has been modified the necessary roof framing members (joists and secondary beams) can be added by using either the **Member** or **Beam** input functionality found in the **1D Member** service. Add all additional framing members as shown in the picture on the next page.

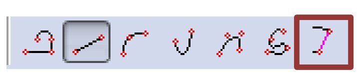
<u>Note:</u> All virtual joist members are placed at a spacing of 5' on center. To achieve this easily utilize the **cursor snap setting, points on line-curve – N-ths.** Additional beam members are also added near the elevator and stairwall as shown in the picture on the next page.





Before the input of the framing can be complete, the composite beams that were copied from the floors below need to be changed to **non-composite members.** To do this, select all (17 members) and change the **type of connection** to **without composite action**. This will eliminate the effective width of the members and make them act non-compositely.

To finish the modeling of the roof structure, the framing members need to be properly connected to the roof plate. To do this, select the plate and then select the Internal edge functionality from **Structure > 2D member > 2D member components.** Then using the **select line** input tool select all virtual joist and newly added roof beam members (52 in total).



This will add an internal edge at each member, thus creating the appropriate finite element connection.

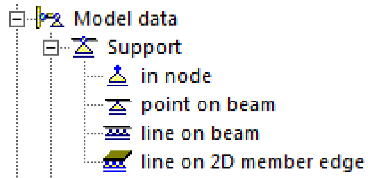
Finally, add hinges on the beams to all members, joists and beams (69 in total) with the following settings modified:

- Position = Both
- fiy and fiz = Free

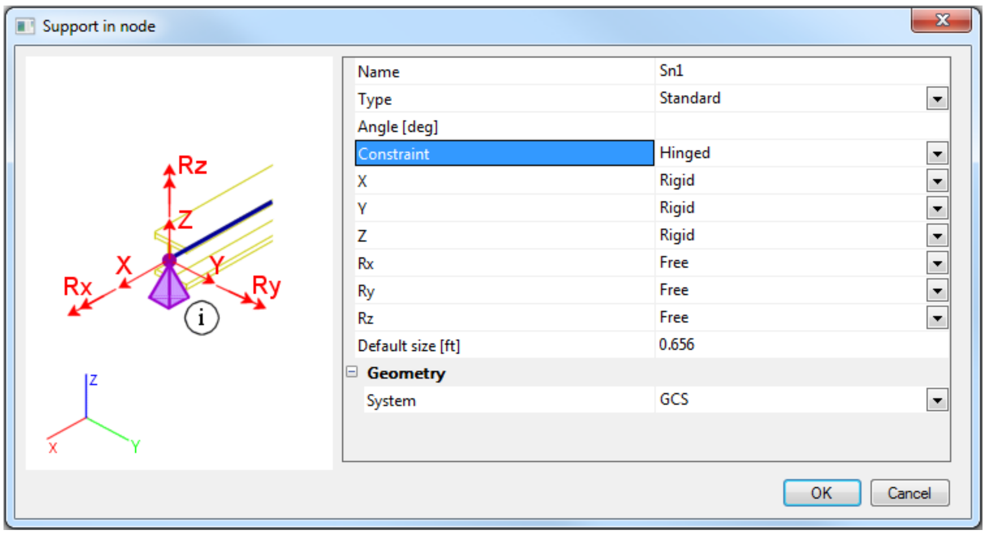
<u>Note:</u> The layer settings of each member can be modified as well. Set the layer for members (beams, columns, walls, floors, braces and joists) accordingly.

Input Supports

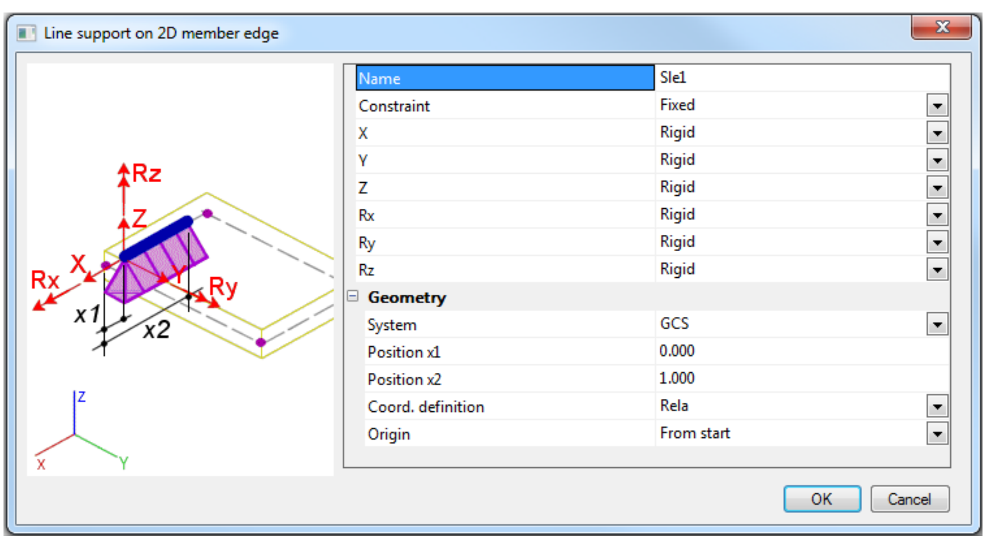
Once the structure has been modelled, support conditions can be added to the base of the columns and walls. Adding supports is possible by navigating to **Structure > Model data > Support.**



In this tutorial, **supports in node** will be used at the base of all 1st floor columns. To add these supports, double click on **in node** and configure the supports as shown below. Once the properties are set, click **OK** and select the nodes at the base of the column elements (16 in total).



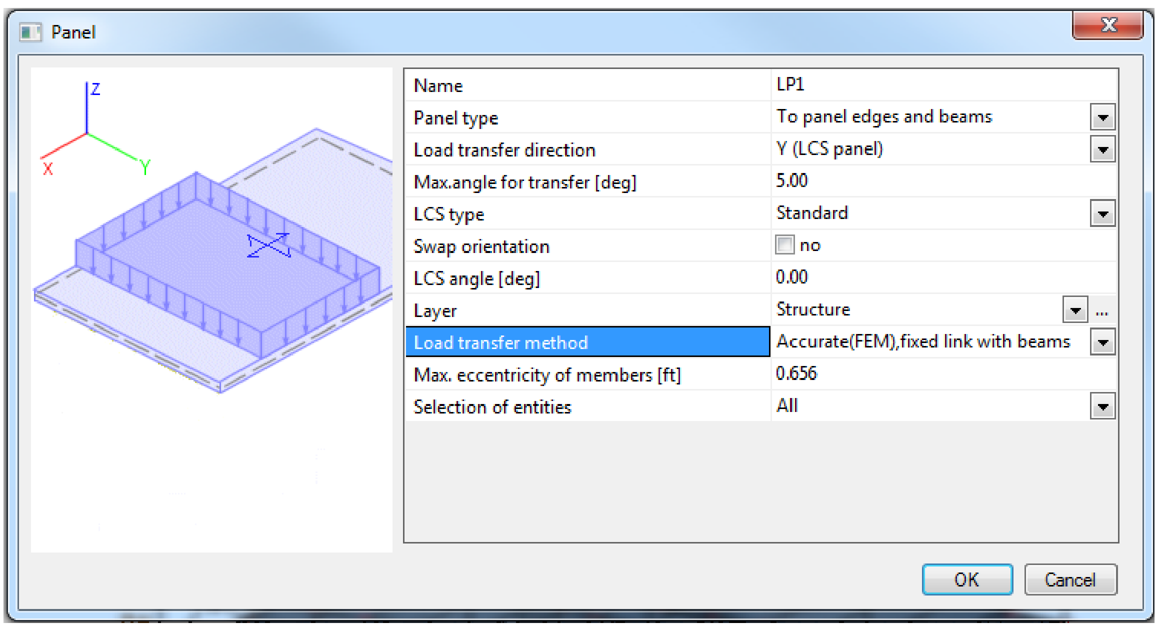
In addition to supports at the base of the columns, line supports at the base of the elevator core and stairwell are required as well. For these elements, use the line on 2D member edge support type and set the properties for the support as shown below. Once the properties are set, apply the line supports along the base of all concrete walls (6 in total).



Load Panels

Part 2 of the tutorial will focus on loading, including the application of wind loading through the use of SCIA Engineer's 3D Wind Load Generator. In order for the wind load generator to work properly, the structure must include **load panels** which wrap the exterior of the building. Load panels have no weight, no in-plane stiffness and function solely as a 2D member which distributes load to connecting members.

To begin the input, expand the Load panel functionality from within the **Structure** service. In this tutorial, Load to panel edges and beams will be used so that the applied loads are distributed properly to specific members, primarily the edges of the floor plates (diaphragms). After the proper type of load panel is selected, the panel dialogue will open and the properties of the panel can be set as shown below.



Load panels are required to be planar, therefore a panel should be modelled on each exterior frame line such that there are no gaps. This will allow the 3D wind to be properly applied to the structure. In total there should be **13 load panels** created (with all panels having a **load transfer direction, Y (LCS)**. The list below will help define the extents of each panel in addition to any modeling tips.

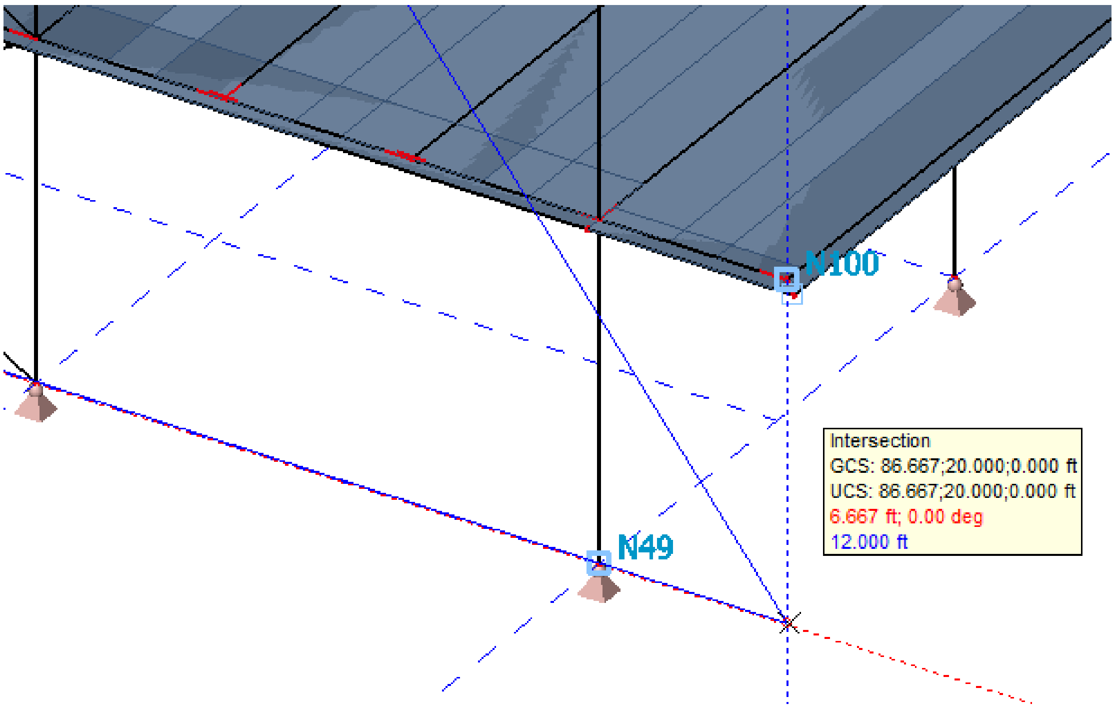
<u>Note:</u> Panels are created along the grid lines such that there is a small edge of the floor plates which would be considered "outside" of the structure. For this tutorial, this is acceptable.

Input Load Panels

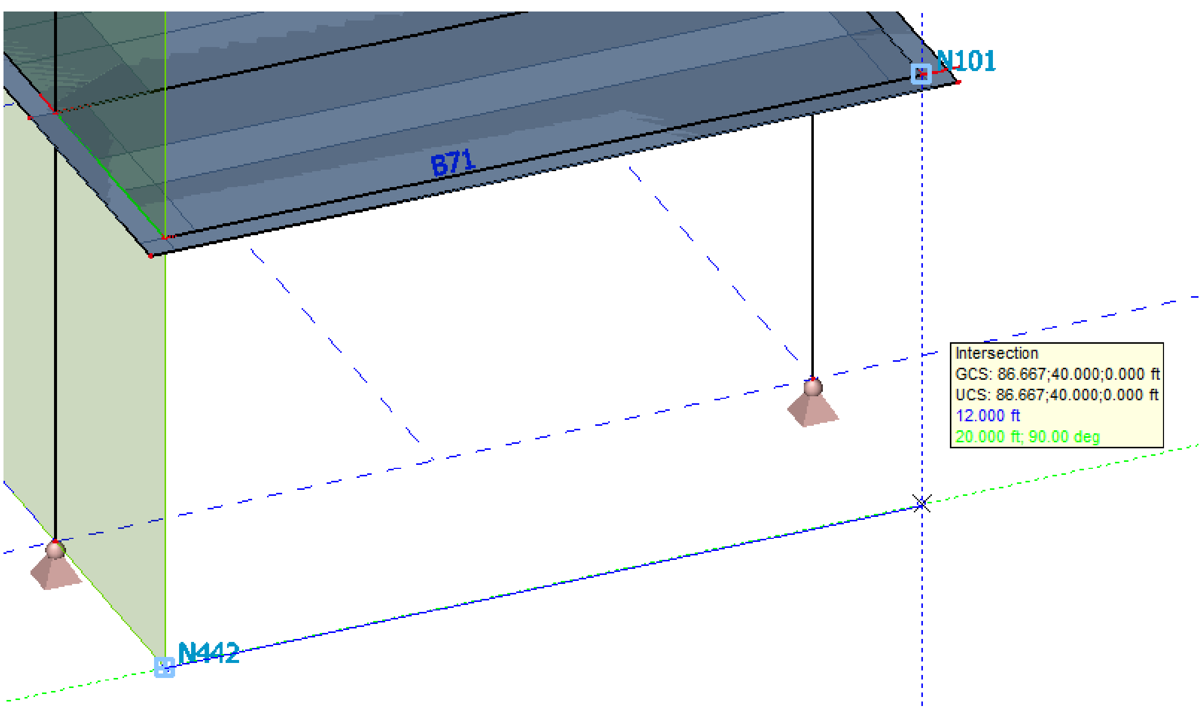
LP1: Along Grid Line 1 (between Grid Lines A & C)
LP2: Along Grid Line C (between Grid Lines 1 & 2)

LP3: Along Grid Line 2 (between Grid Line C & intersection of N49 & N100)

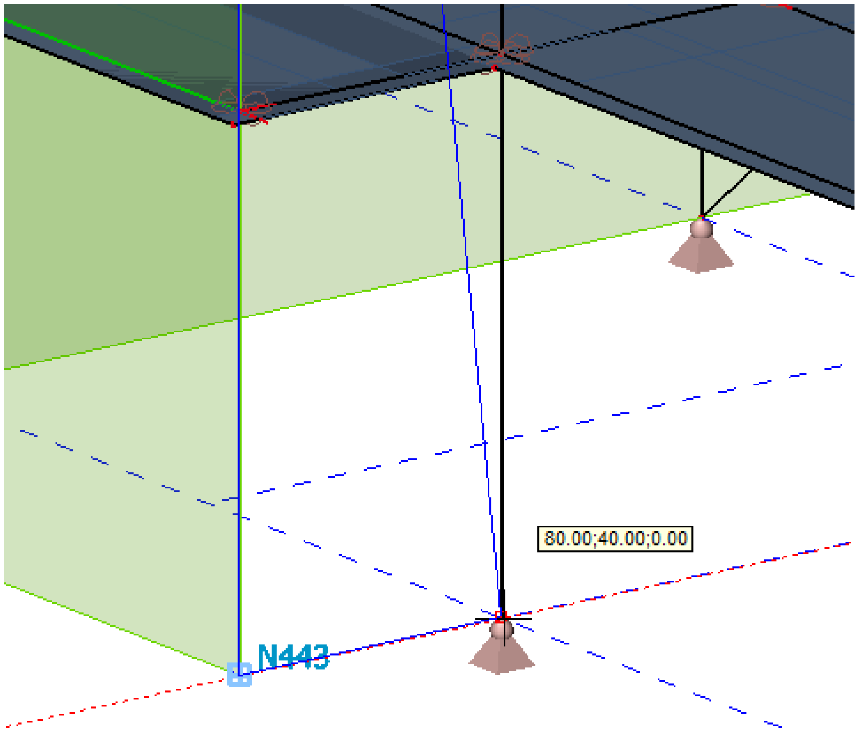
Note: To identify the intersection which defines the lower right corner of the load panel, temporary tracking points (nodes with light blue boxes around them) can be created by **holding SHIFT and clicking on the node**. Once both tracking points are created the intersection between the two will be identified and the point can be selected as shown in the picture below.



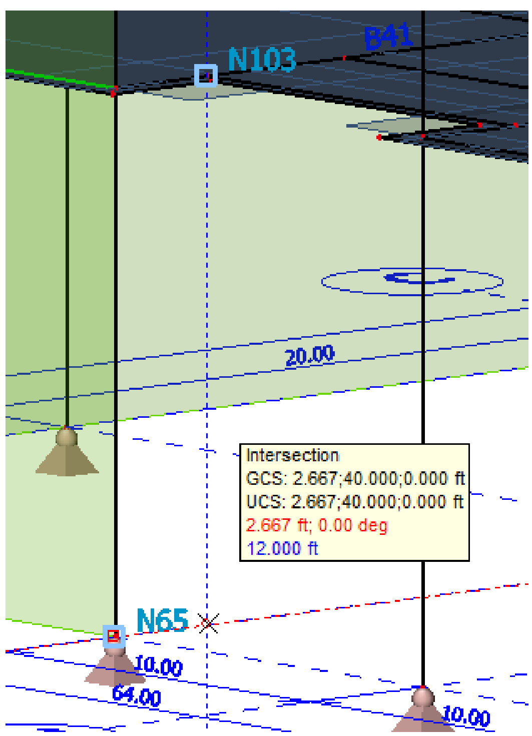
LP4: In the same plane as beam B71 (between Grid Lines 2 & 4)



LP5: Along Grid Line 3 (between LP4 and Grid Line E)

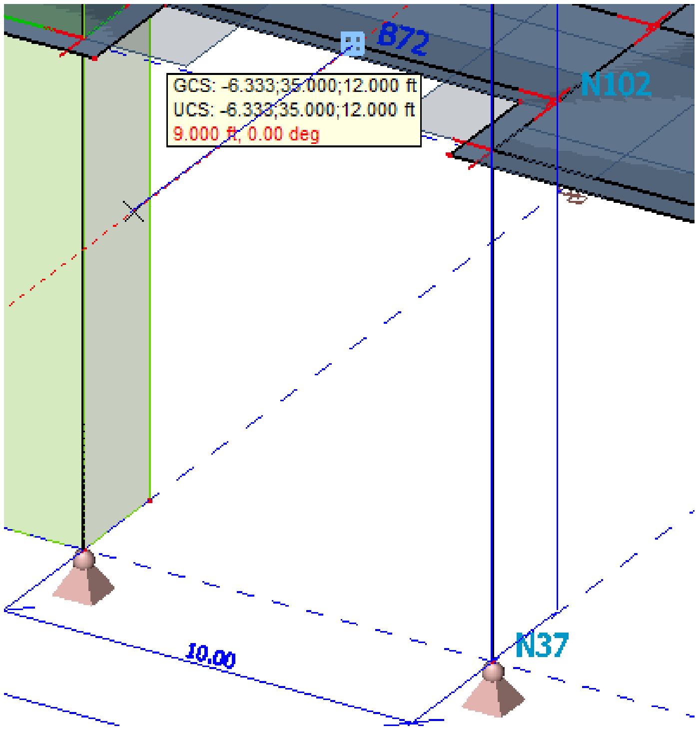


LP6: Along Grid Line E (between Grid Lines 4 & 5)
LP7: Along Grid Line 5 (between Grid Lines D & E)
LP8: Along Grid Line 5 (between Grid Lines A & C)
LP9: Along Grid Line A (between Grid Lines 4 & 5)
LP10: Along Grid Line 4 (between Grid Line A & intersection of N65 & N103)

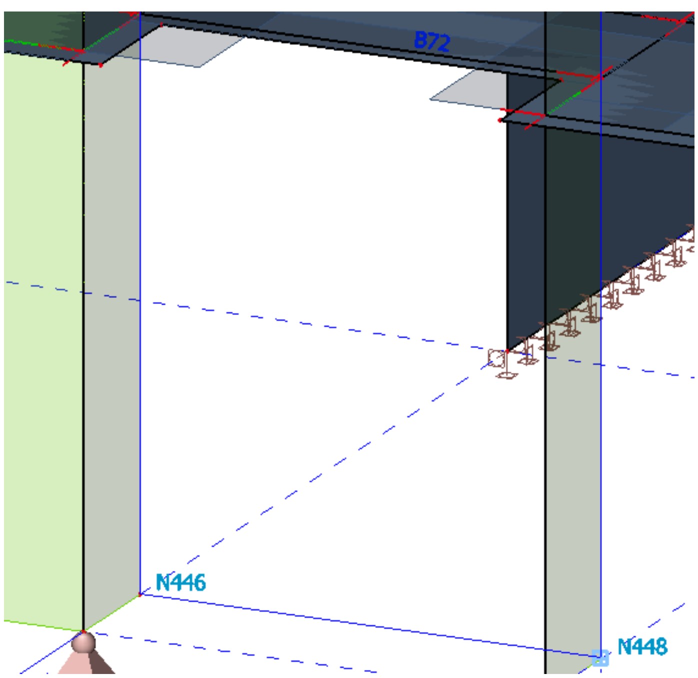


LP11: Along Grid Line 3 (between Grid Line A & intersection of N37 & N102)

Note: LP11 should be created using the **mirror** function. To complete this process, select the previously created **LP10** and then press the mirror button. Next, select the midpoint of **beam B72** as the start point of the mirroring plane. Then select any point along the X axis vector line shown in the picture below to define the end of the mirroring plane. Once the plane is defined, the command will ask if the original element should be removed, press **No** to end the command.

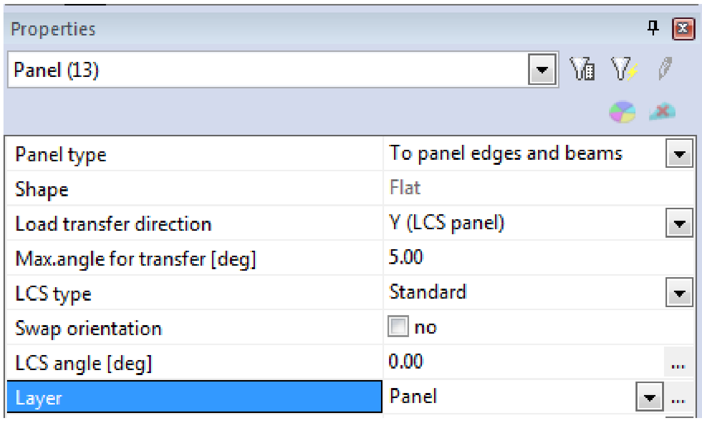


LP12: In the same plane as beam B72 (between LP10 & LP11)

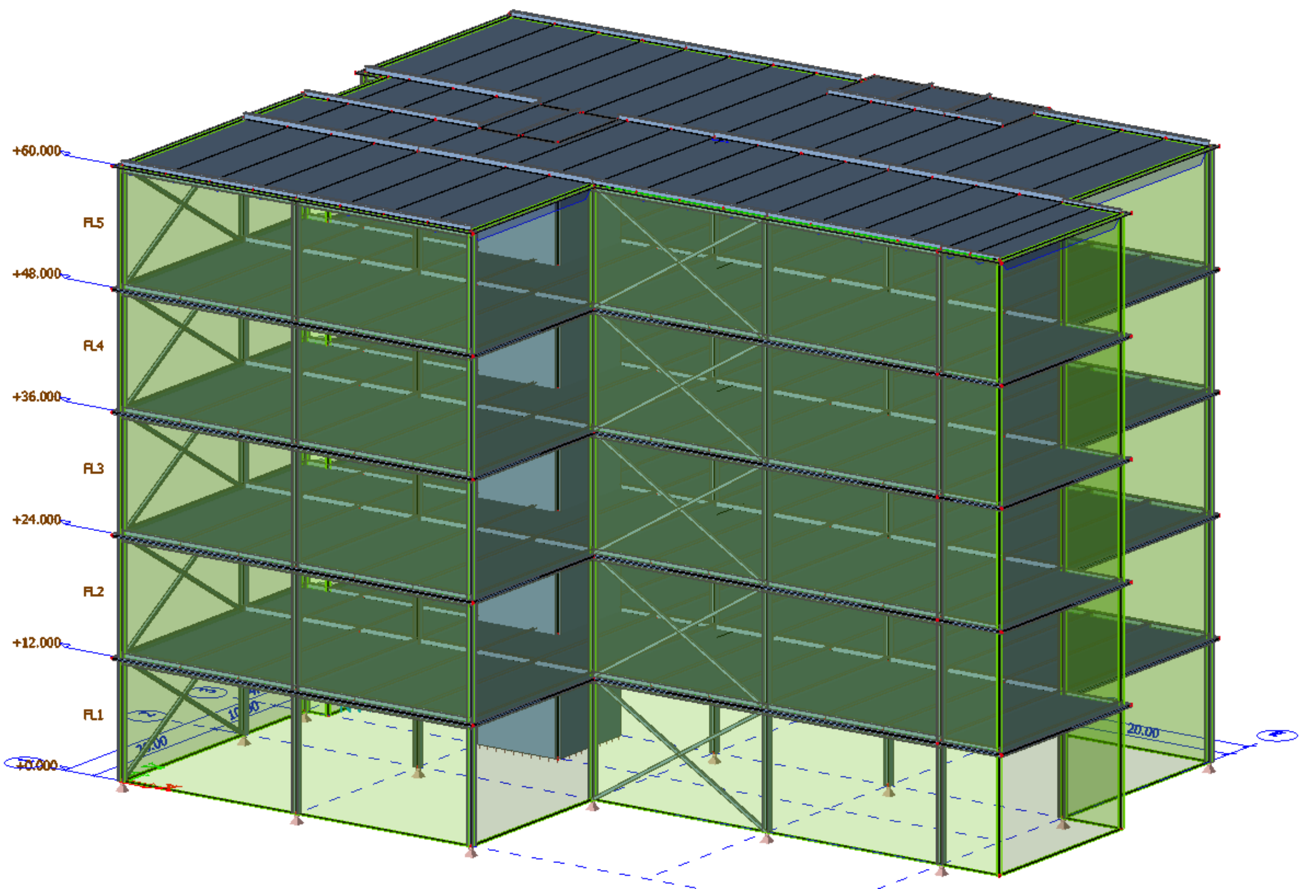


LP13: Along Grid Line A (between Grid Lines 1 & 3)

Finalize the input by selecting all of the load panels (13 in total) and changing the property for **Layer** to **Panel.**



When the input of all load panels is complete, the structure should look as shown in the picture below.



The input of load panels concludes the modeling portion of the tutorial. Save the model (**File > Save**) and then proceed to **Part 2 – Loading** of the tutorial by visiting the Manuals and Tutorials section of the website to download the PDF.