SCIAENGINEER

Open BIM

Integrating the Architectural and Structural Engineering workflows

Belgium Headquarters SCIA nv Industrieweg 1007 B-3540 Herk-de-Stad Tel: +32 13 55 17 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	France SCIA France sarl Centre d'Affaires 29, Grand' Rue FR-59100 Roubaix Tel.: +33 3.28.33.28.67 Fax: +33 3.28.33.28.69 E-mail: france@scia.net Agence commerciale 8, Place des vins de france FR-75012 Paris Tel.: +33 3.28.33.28.67 Fax: +33 3.28.33.28.69 E-mail: france@scia.net
Brazil SCIA do Brasil Software Ltda Rua Dr. Luiz Migliano, 1986 - sala 702 , CEP SP 05711-001 São Paulo Tel.: +55 11 4314-5880 E-mail: brasil@scia.net	USA SCIA North America 7150 Riverwood Drive 21046 Columbia, MD Tel.; +1 443-542-0638 Fax.:+1 410-290-8050 E-mail: usa@scia.net
Netherlands SCIA Nederland B.V. Wassenaarweg 40 NL-6843 NW ARNHEM Tel.:+31 26 320 12 30 Fax.: +31 26 320 12 39 E-mail: info@scia.net	Switzerland SCIA Swiss Office Dürenbergstrasse 24 CH-3212 Gurmels Tel.: +41 26 341 74 11 Fax: +41 26 341 74 13 E-mail: info@scia.ch
Czech Republic SCIA CZ s.r.o. Praha Evropská 2591/33d 160 00 Praha 6 Tel.: +420 226 205 600 Fax: +420 226 201 673 E-mail: info.praha@scia.cz SCIA CZ s.r.o. Brno Slavickova 827/1a 638 00 Brno Tel.: +420 530 501 570 Fax: +420 226 201 673 E-mail: info.brno@scia.cz	Slovakia SCIA SK s.r.o. Murgašova 1298/16 SK-010 01 Žilina Tel.: +421 415 003 070 Fax: +421 415 003 072 E-mail: info@scia.sk
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 E-mail: info@scia.at Support Tel.: +43 1 7433232-12 E-mail: support@scia.net	Germany SCIA Software GmbH Technologie Zentrum Dortmund, Emil-Figge-Strasse 76-80 D-44227 Dortmund Tel.: +49 231/9742586 Fax: +49 231/9742587 E-mail: info@scia.de

Table of Contents

GENERAL INFORMATION

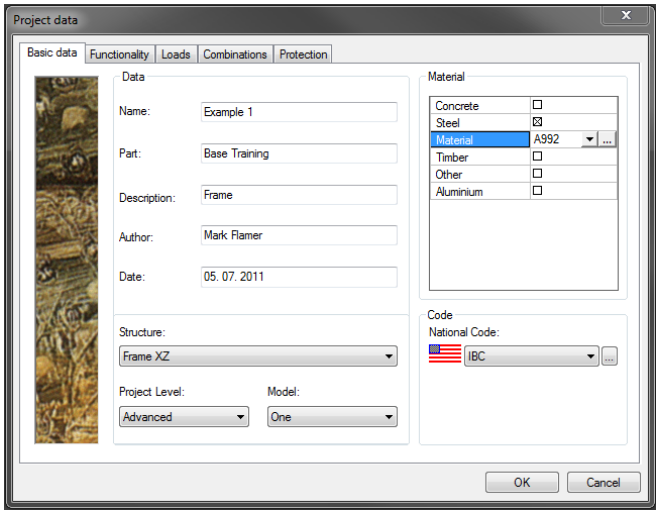
SCIA Engineer Support

Via the SCIA Support website: https://www.scia.net/en/support-downloads

SCIA Engineer – General Environment

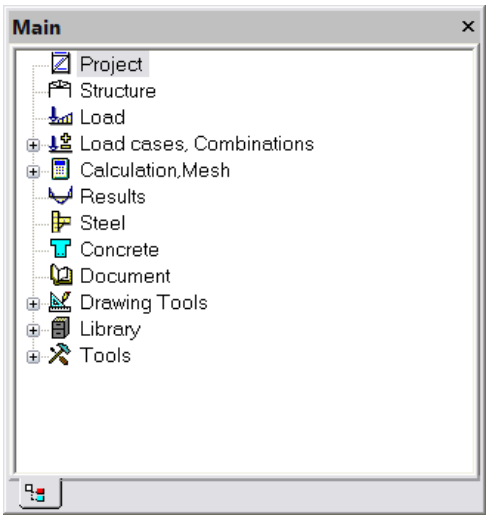
Setup > Options 
Help > Contents > Reference guide

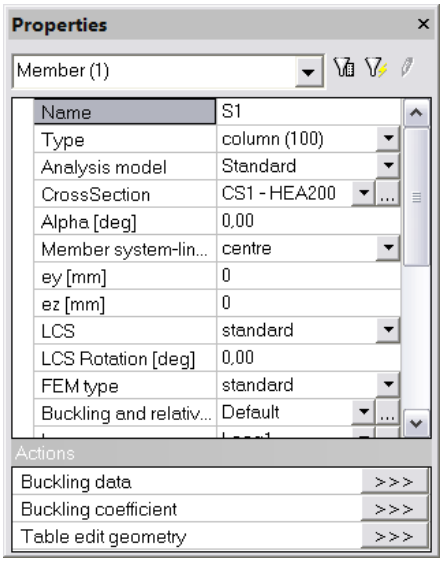
File > New  > Project data



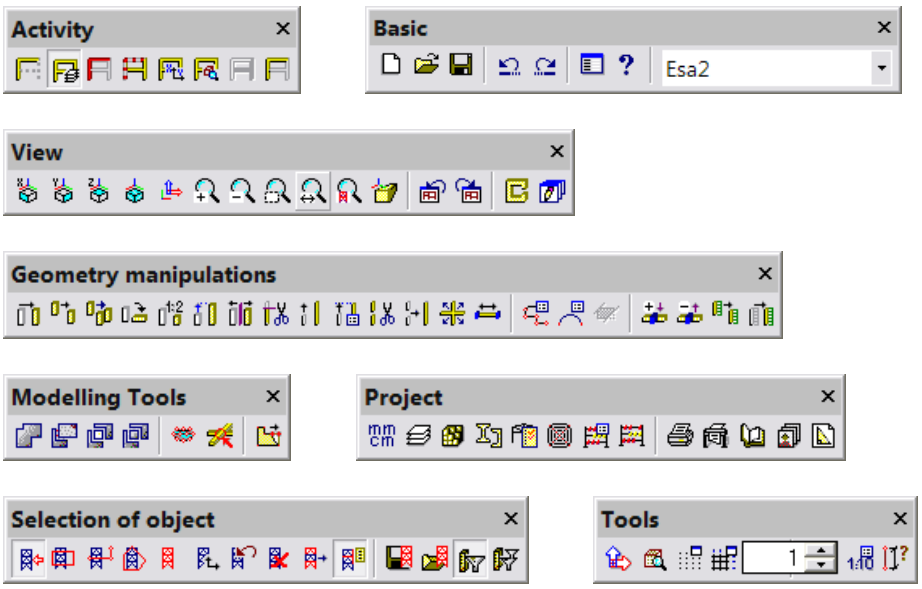
Overview of the Menus

Main Menu & Properties Menu + Actions





Overview of the Toolbars



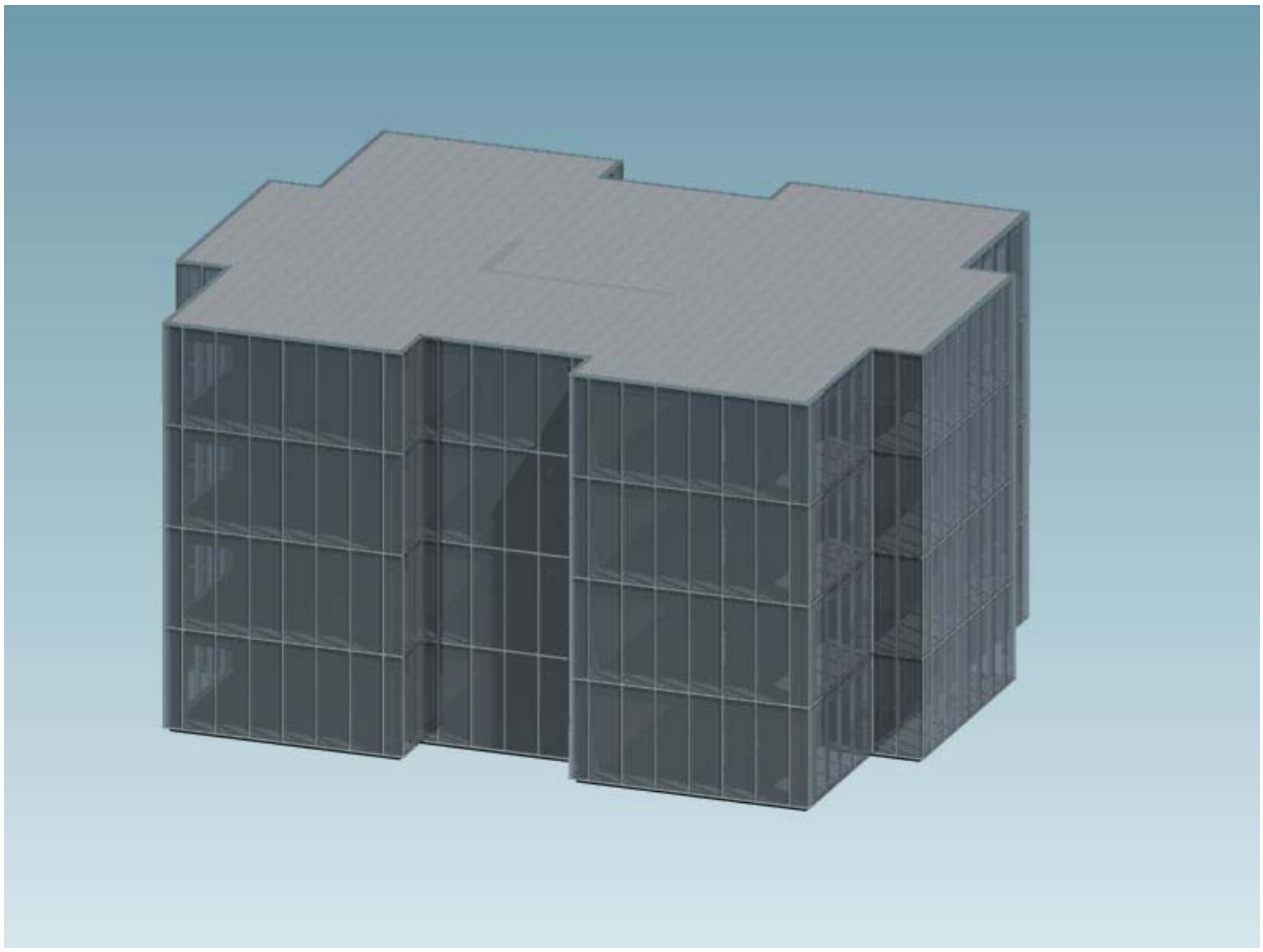
Command line

REMARK: If a menu or toolbar has been (accidentally) removed from the project, it can be re-activated via View > Toolbars.

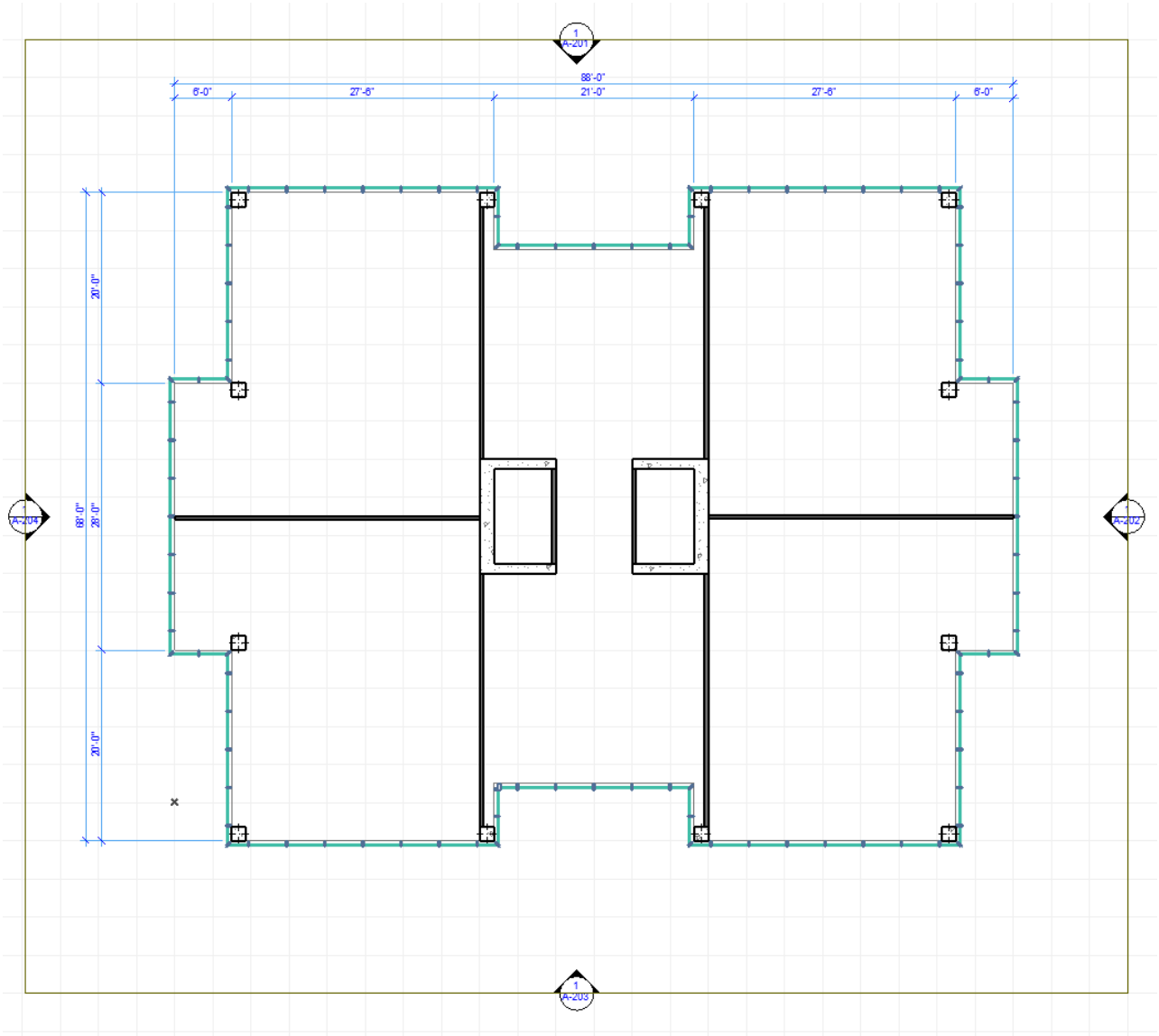
INTRODUCTION – OPEN BIM FOR THE STRUCTURAL ENGINEER

This exercise is designed to walk a student or new user of the BIM process through a small example project highlighting the benefits and pitfalls of model based collaboration using open file formats. The project is a 4 story reinforced concrete building with a non-structural curtain wall cladding system. The students play the part of a structural engineering consultant who has just received an LOD200 BIM model from the project architect in the schematic design stage. We begin the exercise by verifying the model in a third party IFC viewing application. Once verification is complete, we import the model into our structural analysis application where we leverage the efforts of the architect into a simplified finite element model, apply loading conditions and run a simple linear analysis. We will then make a few small changes to the model, export back out as IFC and once again verify in an IFC viewer prior to sending it back to the architect or downstream to detailers or fabrication consultants. We conclude the exercise by extracting a calculation report and some simple drawings from our model.

This tutorial is designed to take between 4 and 6 hours to complete. A folder with all the necessary files has been provided with this document. In this folder is a subfolder titled “Step By Step Files”. Each of these files has all the previous steps completed. If you get stuck and are short on time, skip ahead to the next section using one of these files.



Rendering of Architectural Model

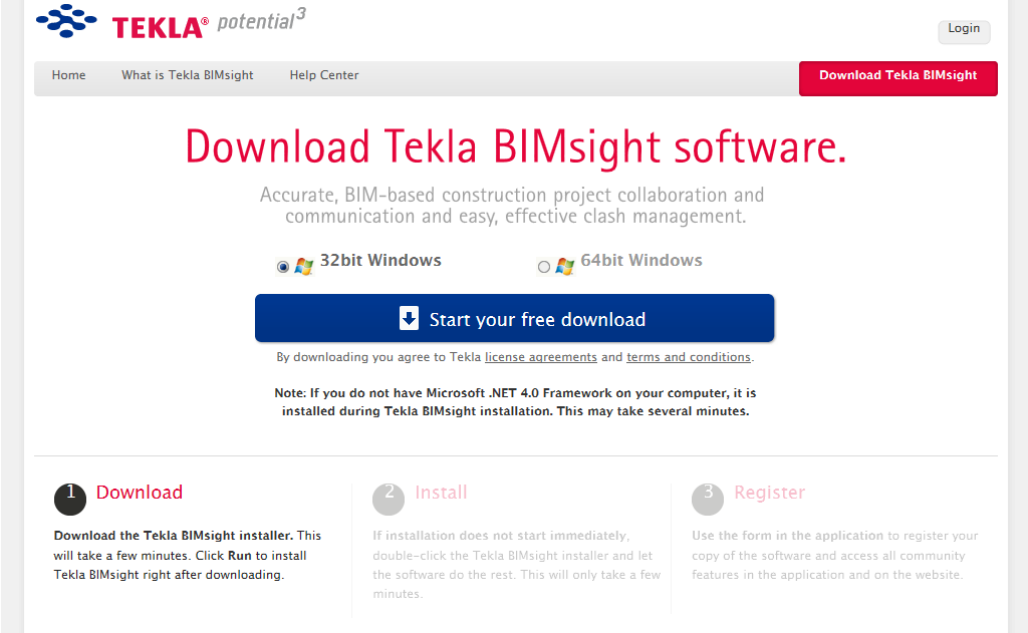


First Floor Plan of Building

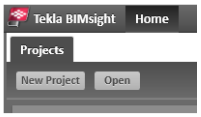
PART 1 – VERIFY BIM MODEL AND IMPORT INTO SCIA ENGINEER

3rd Party neutral IFC viewer:

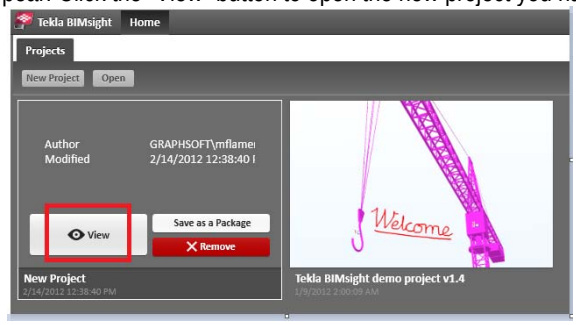
- Tekla BIMsight is a free IFC viewing and mark-up program that can be freely downloaded from the Tekla website.
  - http://www.teklabimsight.com/downloads.jsp



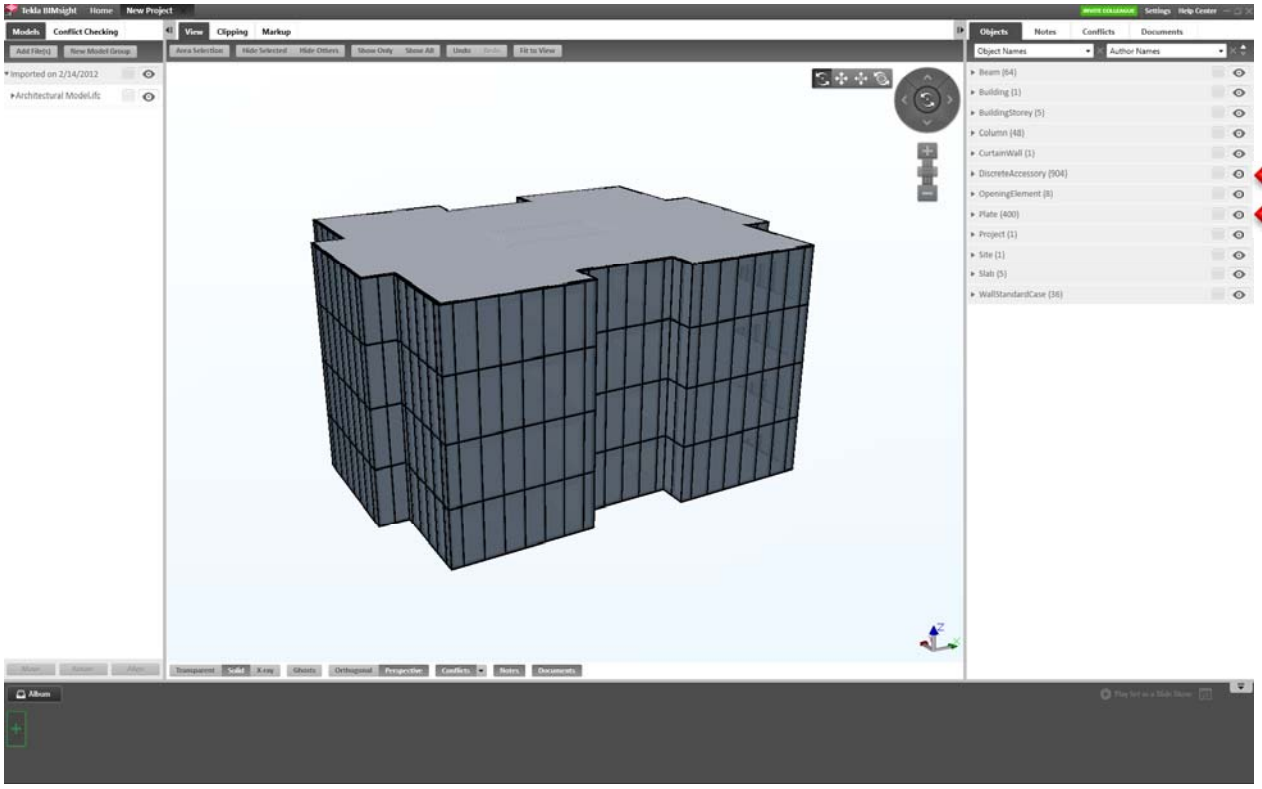
- Because IFC is an open, vendor neutral file format it is always a good idea to validate any models received from another party prior to incorporating them into your engineering process. In this exercise we will use Tekla BIMsight for this purpose.
  - Download, install and open BIMsight.
  - Click the “New Project” button.



  - A new project panel will appear. Click the “View” button to open the new project you have created.



  - Navigate to the file “Architectural Model.ifc” provided in the project folder.
  - After a few moments the model should appear.
  - Orbit the model by holding the left mouse button and dragging in the model window.
  - Make the cladding objects invisible by clicking the eye buttons on the right side of the window.
  - Verify that the model has transfered correctly.



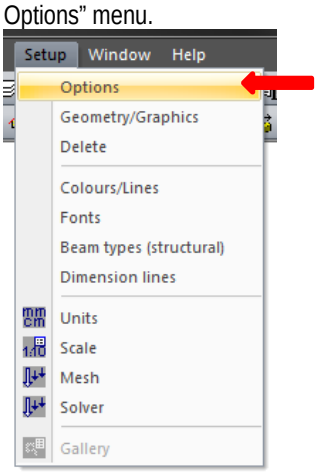
- You may now close BIM sight. The project file will be saved automatically.

Import into SCIA Engineer:

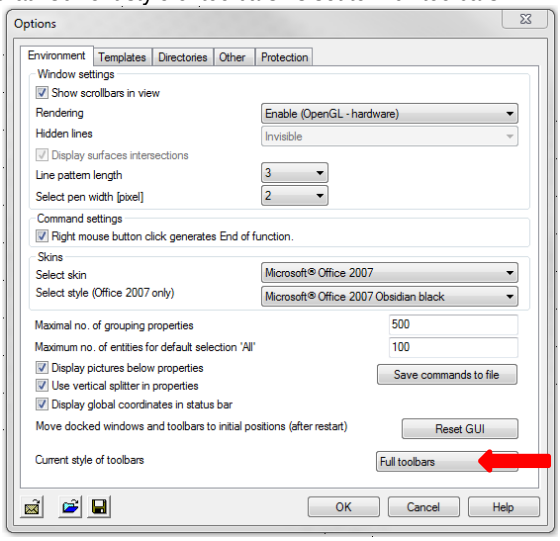
1 Download and Install

- The free student version of SCIA Engineer is available at www.scia-campus.com.

- A new installation of SCIA Engineer may have a simplified UI by default. To complete this exercise, we need access to the complete set of commands. Open the “Setup – Options” menu.



- In the “Options” dialogue, verify that “Current style of toolbars” is set to “Full toolbars”

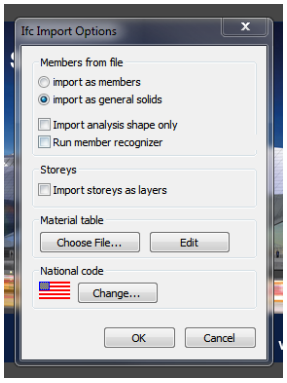


2 Import IFC

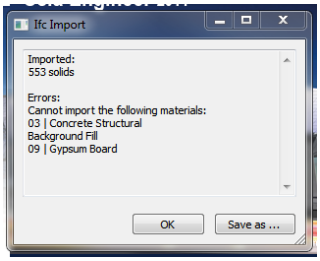
When importing an IFC file, SCIA Engineer creates a new file as opposed to placing the IFC into an existing file or template. For this example, the instructor has already done some preparation in a template file to speed up the exercise, so it will be necessary to import the IFC model and then transfer it into the template.

- Open SCIA Engineer. If prompted to select a file to open, click cancel so SCIA Engineer starts with no file currently open. Your screen should look like this. Note: SCIA Engineer offers different UI “skins” so the coloring may be different (light blue).

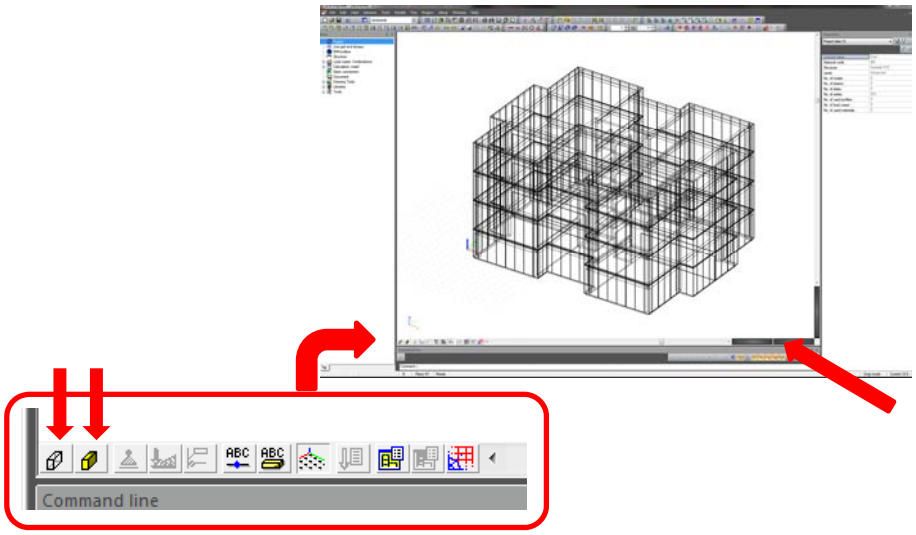
- In the “File” menu select “Import” – “IFC 2x3”. Navigate to the IFC file “Architectural Model.ifc” provided in the project folder.
- You will be prompted with some options regarding the method of import. Select “import as general solids” as shown below and leave the rest as default. SCIA Engineer allows you to import an IFC file as either native analytical members (beams, columns, plates, etc…) or as “General Solids”. General solids are neutral geometry that have no effect on analysis, but can be used as a “Reference Model”. Because this model came from an architect, chances are it contains a lot of geometry which is unnecessary or unwanted in our analysis. Things like door knobs and light fixtures. Importing as general solids allows us to ignore this additional geometry and offers us more control.



- When this process has completed successfully, you should see the following dialog. Click “OK” to complete the import.

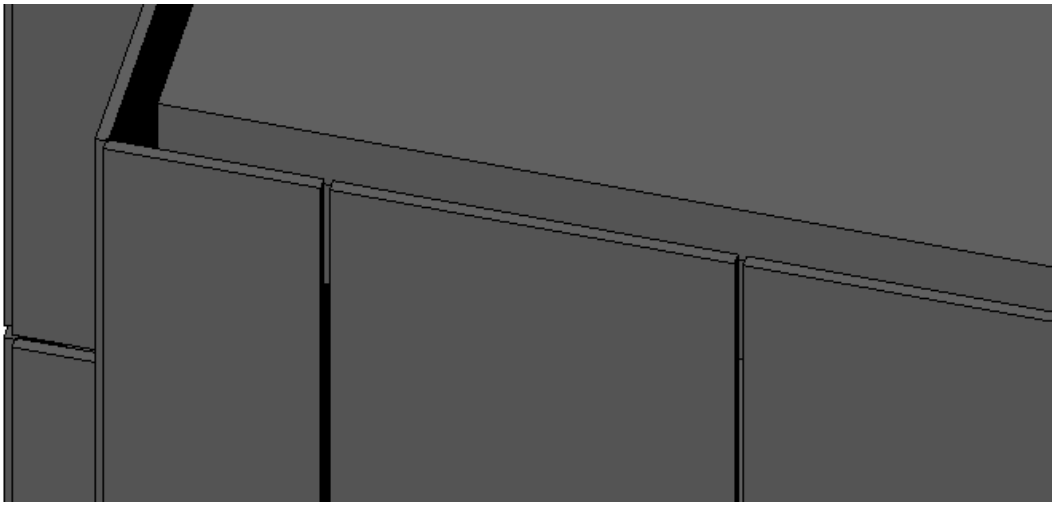


- The view can be set to render filled and volumetric with the following 2 buttons. This tool bar is for quick adjustment of view and visibility settings.



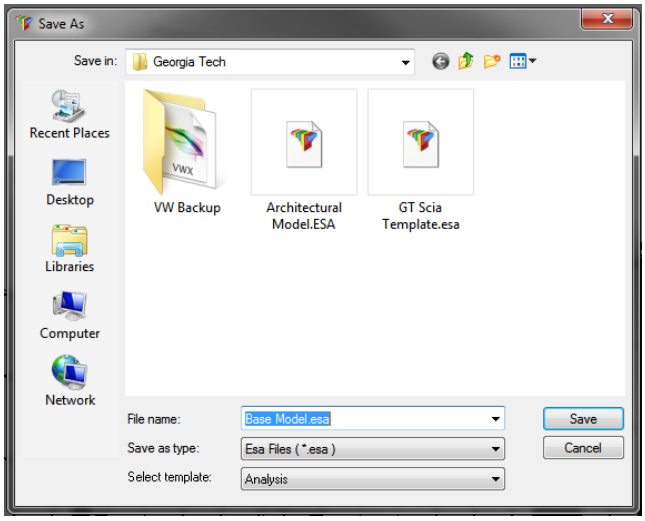
- Zoom with the mouse wheel or the 3 navigation bars on the lower right corner of the window.

- Orbit by holding down the “Ctrl” button and the right mouse button while moving the mouse. Or, by using the 3 navigation bars.
- The strength of an open file format like IFC is the ability to exchange BIM models between software packages regardless of what their native file format is. This offers freedom in the choice of software instead of being locked in to using software from a specific vendor. The drawback is that each software vendor reads and writes the IFC format a little differently and inevitably discrepancies arise. This situation is getting better as more users demand consistent IFC model exchange from the software companies.
- You will notice an example of this in the imported model. The window mullions were not imported into SCIA Engineer. This discrepancy is of no significance for our structural model or analysis but, shows the value in validating the model in a neutral viewing application prior to import.

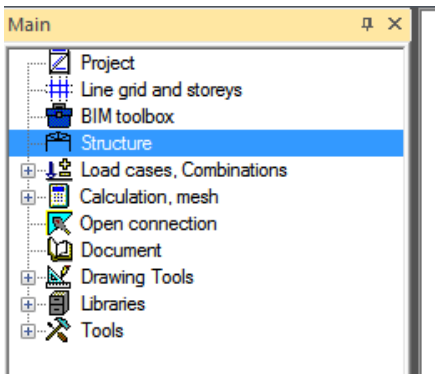


3 Transfer Model to Provided Template File

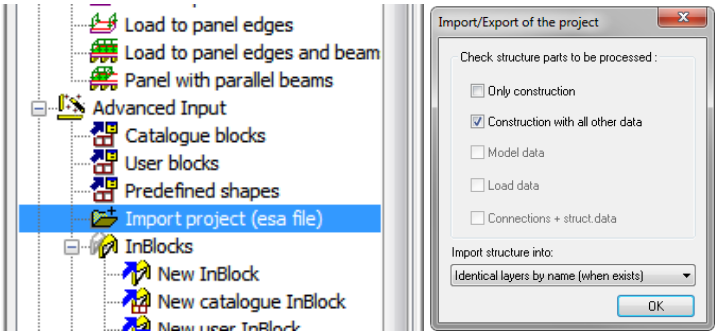
- In the “File” menu select “Save As”. Navigate to the project folder and name the file “Base Model” as shown below.



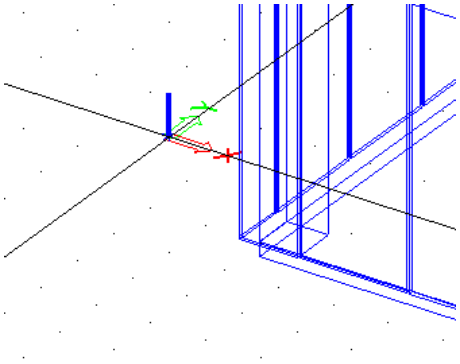
- Close the file.
- Open the file “GT SCIA Template.esa” using the “File” menu or the  button on the tool bar.
- Double click the “Structure” service in the “Service Tree” as shown below. The Service Tree is the large UI panel on the left side of the SCIA Engineer window in this tutorial. The majority of the tools in SCIA Engineer are presented in the various “Services” available in this “Tree”.



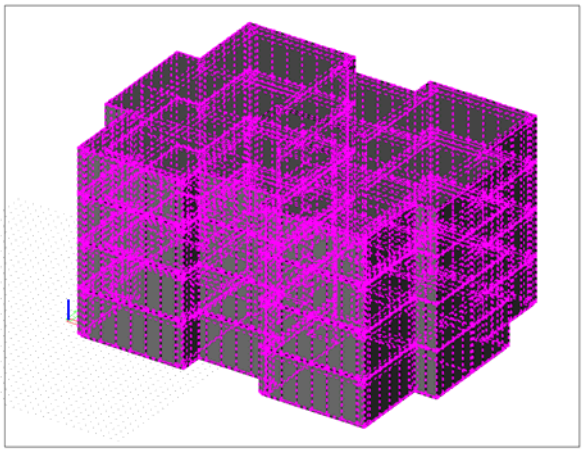
- Double click “Import Project (esa file)” and navigate to the “Base Model.esa” file in the project folder we saved in an earlier step. Click open and accept the default “Construction with all other data” in the window presented.

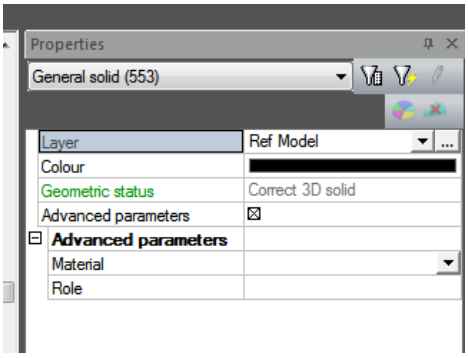


- You will notice your cursor dragging the imported model in the 3D workspace. Click at the origin to place the model. Click “Esc” to end the command.

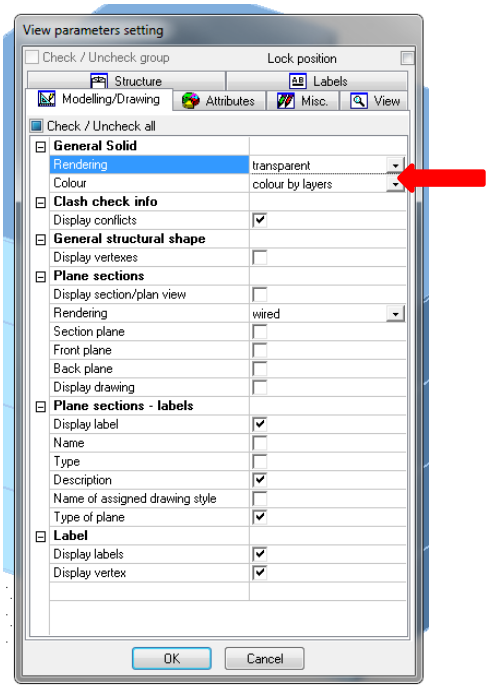


- Zoom out so you can see the entire model. Left click, hold and drag a window to select the entire model. With the model selected click the pencil button in the “Properties” palette which is on the right side of the SCIA Engineer window throughout this tutorial. This will refresh the properties display due to the large number of objects selected. Like most CAD or modelling programs, SCIA Engineer has the concept of layers as a way of separating and organizing complex files. Change the layer for the selected objects to “Ref Model” in the properties palette. As our structural model develops, this will allow us to always refer back to the original geometry and turn it off when convenient.





- Now let's make our reference model transparent so that we can begin to create a structural system within it. Right click with the mouse anywhere in the empty space of the 3D window, and select "Set view parameters for all". In the "Modelling/Drawing" tab, set the rendering to "transparent"

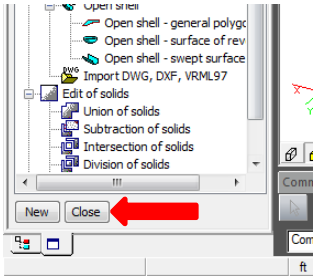


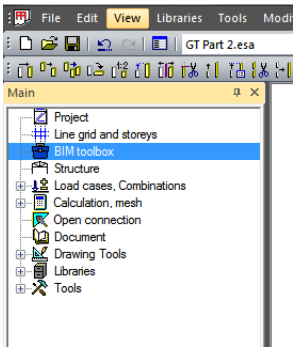
- Save your file and continue on to part 2.

PART 2 – EXTRACT STRUCTURAL SYSTEM FROM MODEL

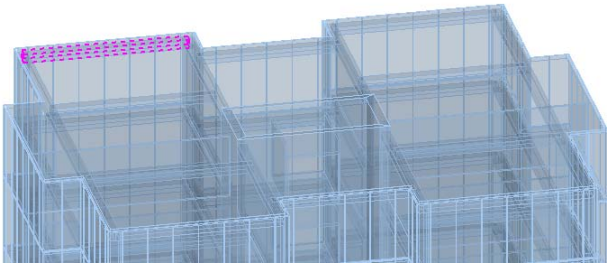
1 BIM Toolbox

- One of the strengths of SCIA Engineer is its ability to leverage model data from many sources by letting you convert general solids into native objects with analytical properties.
- Open either the file you saved at the end of Part 1 or the file provided in the “Step by Step Files” folder.
- Navigate to the “BIM Toolbox” by double clicking in the Service Tree. If you are already in another “Service” as a result of the last chapter, click “Close” at the bottom of the Service Tree Palette.

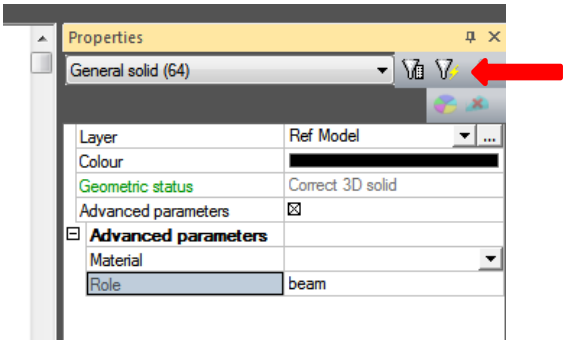


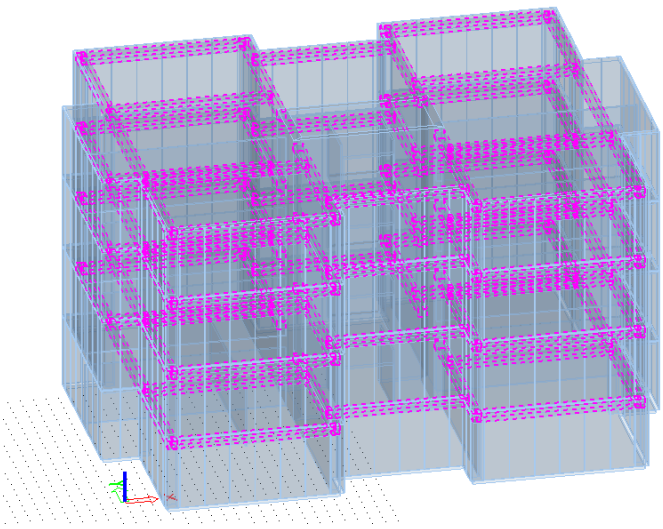


- Select a beam element with the mouse in the 3D model.

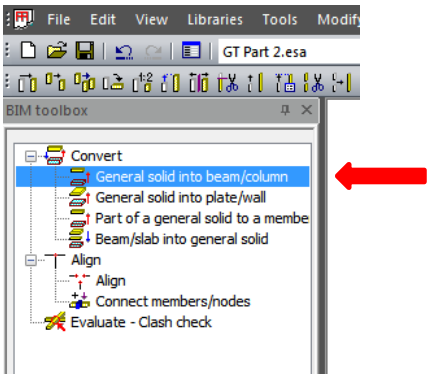


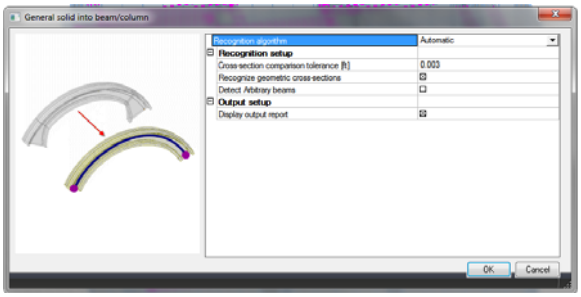
- Highlight the “Role” property field in the properties palette and then click the Funnel/Lightning Bolt button at the top of the palette as shown below. This is a quick way to select all elements with a matching property in SCIA Engineer. In this case all general solids with the role of “beam”. This “role” property is an additional data field in the IFC file.



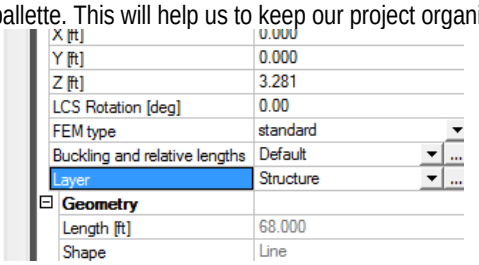


- You should now have 64 general solids selected. Convert them into native SCIA Engineer analytical 1D members by double clicking the “General solid into beam/column” tool in the Service Tree. SCIA Engineer provides options for extraction of multiple cross sections along the length of tapered or non-prismatic members, but for this simple example the defaults are adequate.

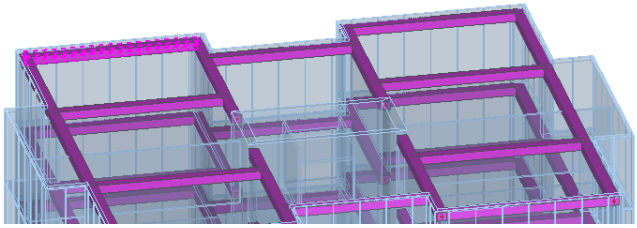


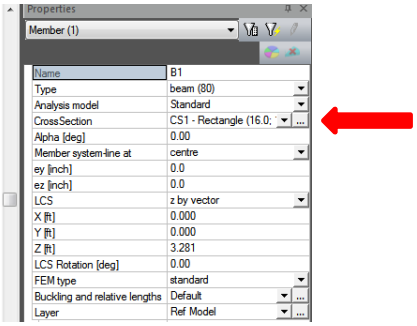


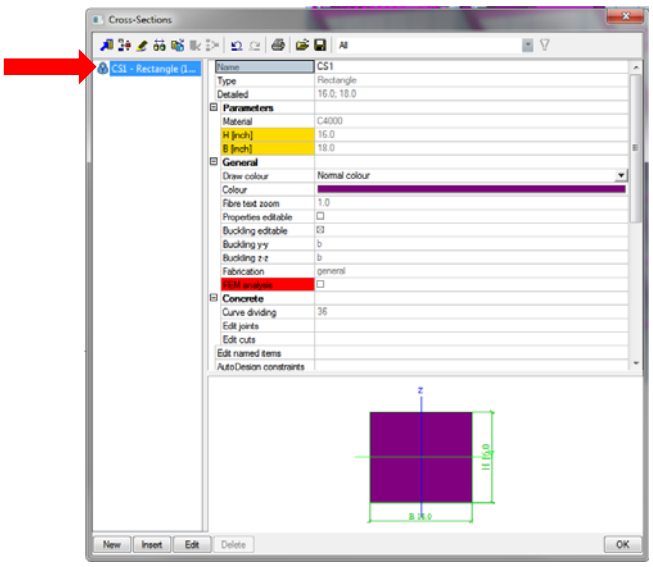
- You should notice that the beams are no longer transparent and are now a “pink” color. With the beams still selected, change the layer to “Structure” in the properties pallette. This will help us to keep our project organized.



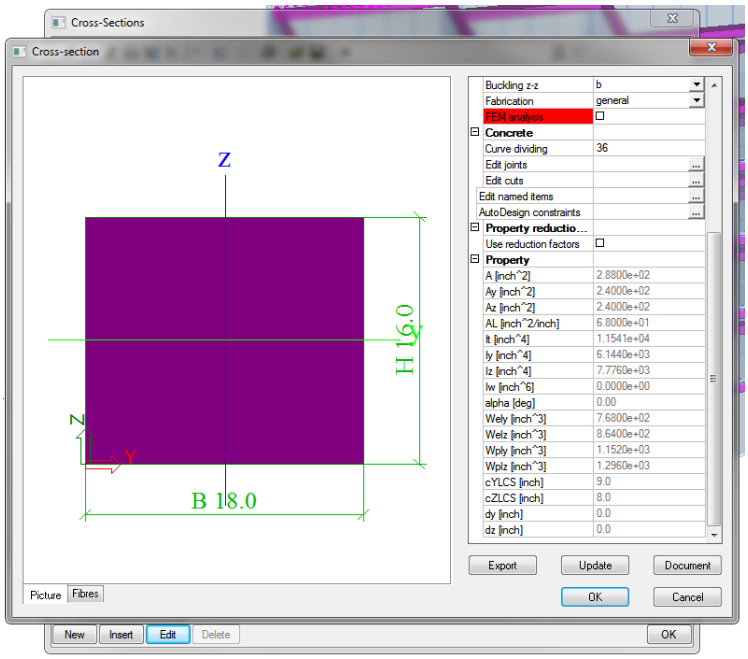
- Select one of the beams and in the properties palette click the small elipsis button to the right of the “Cross Section” property. This will take you to the cross section database where we can verify the section properties.



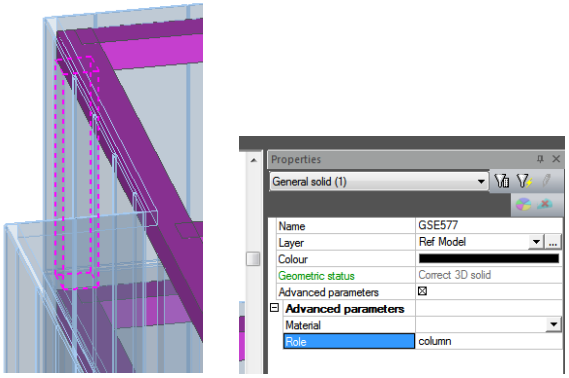




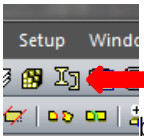
- Select “CS1 – Rectangle” in the database list and click “Edit”.



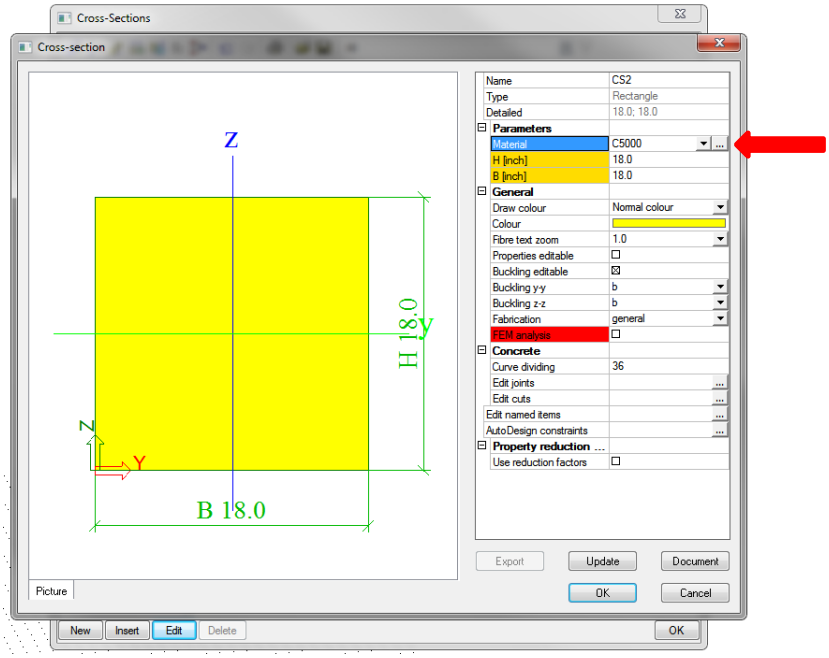
- Notice the cross sectional properties (Area, Moment of Inertia, etc. …)
- Click “Cancel” then “OK” to exit back to the work space.
- De-select any members by clicking “Esc” on the keyboard.
- Select all general solids with the role “column” using the same process as described above for beams.



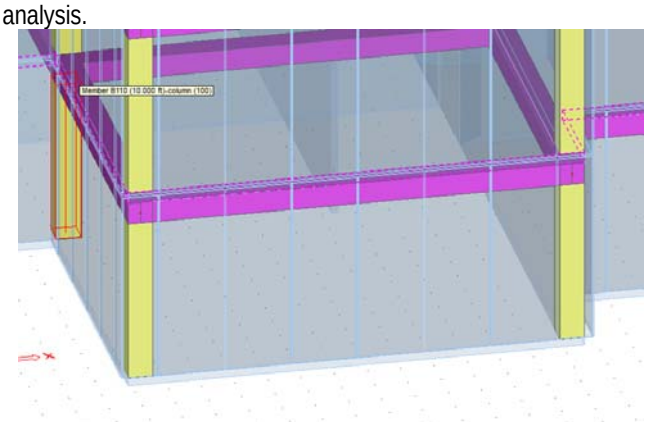
- Use “General solid into beam/column” to convert 48 members and set the layer to “Structure” as described above.

- Enter the Cross Section database using the method described above or using this 

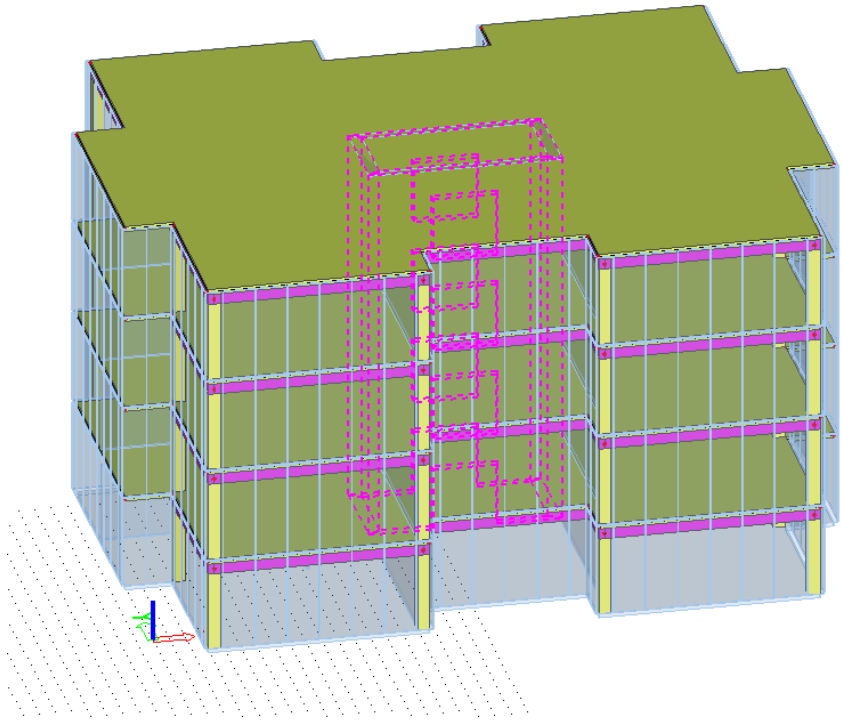
button in the tool bar. Set the material to C5000 (5000psi concrete).



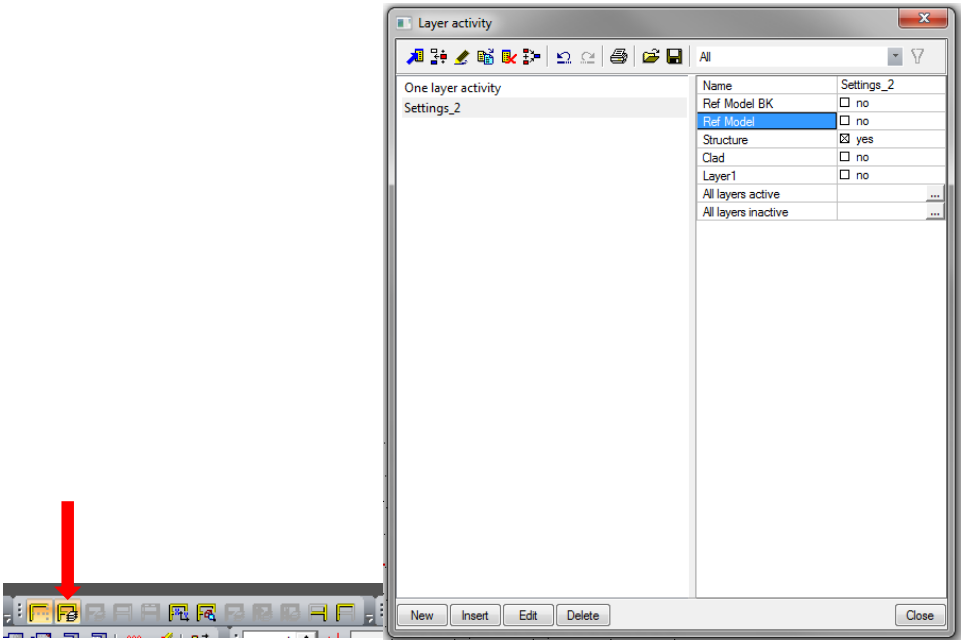
- Repeat the process above selecting all general solids with the role “slab”.
- Holding “Shift”, left mouse click on the ground floor slab to de-select it. This would be a slab on grade in this project and we do not want to include it in our analysis.



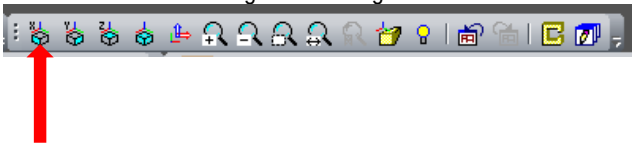
- Convert the 4 selected solids into slabs using “General solid into plate/wall” accepting all defaults. Your model should now look like this. Set the layer of the slabs to “Structure”.



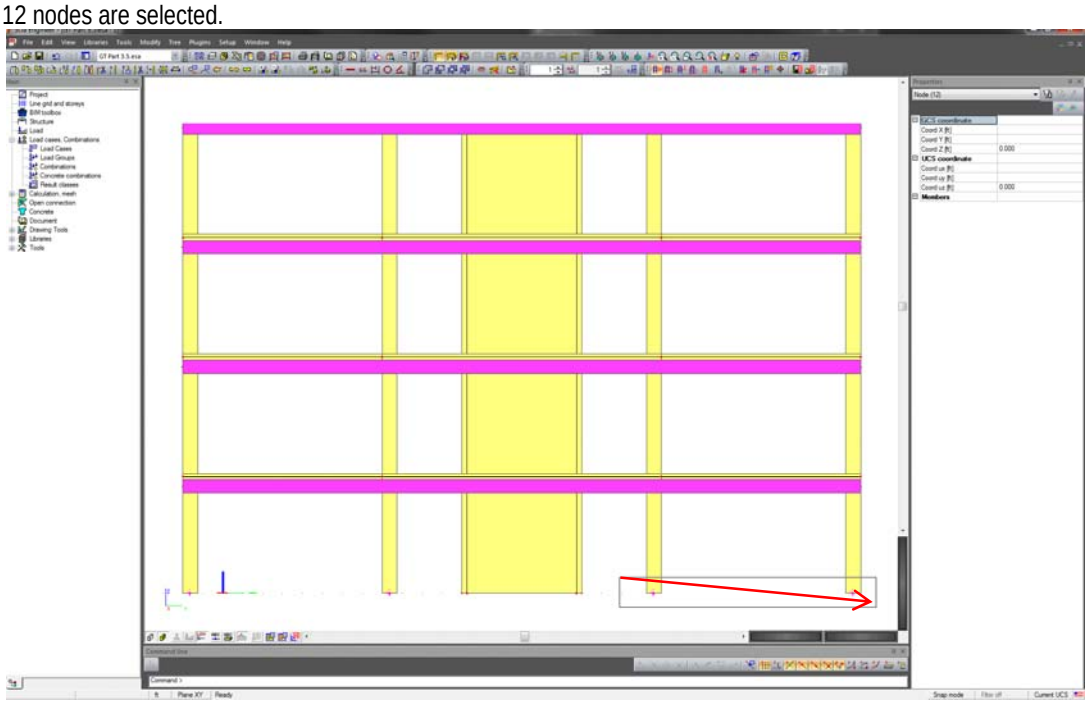
- Manually select the 4 solids representing the core walls as shown above and convert into walls using “General solid into plate/wall”. Set the layer to “Structure”.
- We are done with the reference model for now so we will turn off the visibility of the layer it is in. SCIA Engineer has several methods of organizing the model for visibility. These are called “Activities” and are manipulated through the “Activities” toolbar shown below. Select “Activity by Layers” and turn off the layer “Ref Model”. This will make it easier to work on our analysis model and see results.



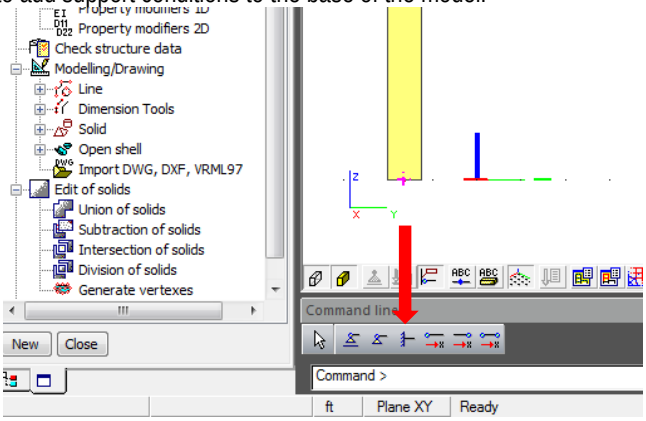
- Set the 3D window’s view to be from the X direction using the following button on the “View” toolbar.



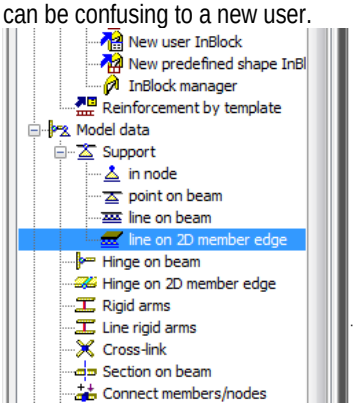
- Click-Drag twice to select all the nodes at the base of the columns as shown below. The top of the properties palette should indicate that 12 nodes are selected.



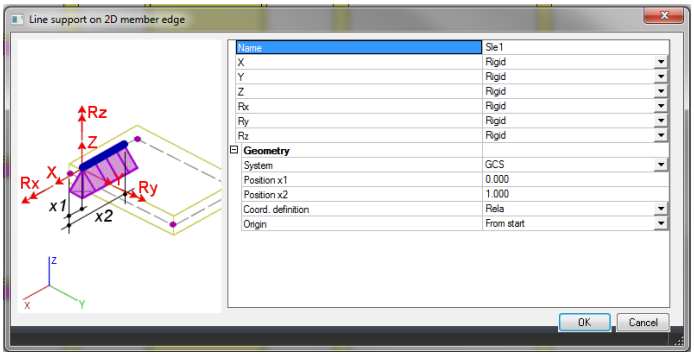
- Double click “Structure” in the Service Tree to gain access to the structure tools. With the 12 nodes still selected, click the “Fixed Support” Quick input button to add support conditions to the base of the model.



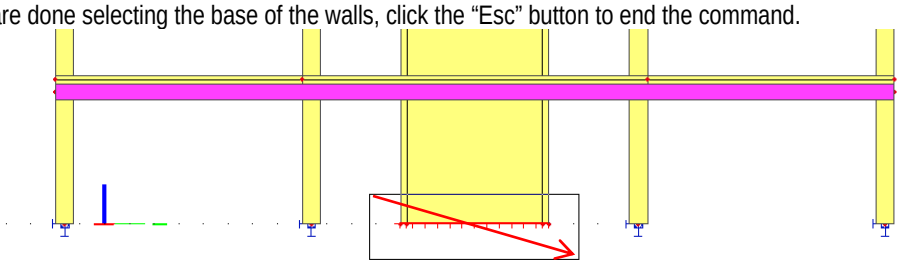
- Each Service in SCIA Engineer provides a set of “Quick Input” buttons just above the command line. These allow easy access to commonly needed commands. All of these commands and more are also available in the service tree and menus.
- Double click “line on 2D member edge” in the service tree. Pay close attention to the hierarchy of the tree in the image below. If one of these sub-sections is closed, finding a tool can be confusing to a new user.



- Accept the defaults in the dialogue by clicking “OK”.



- Click-Drag to select the base of all the core walls as shown below. This will apply a fixed support condition at the base of the walls. When you are done selecting the base of the walls, click the “Esc” button to end the command.

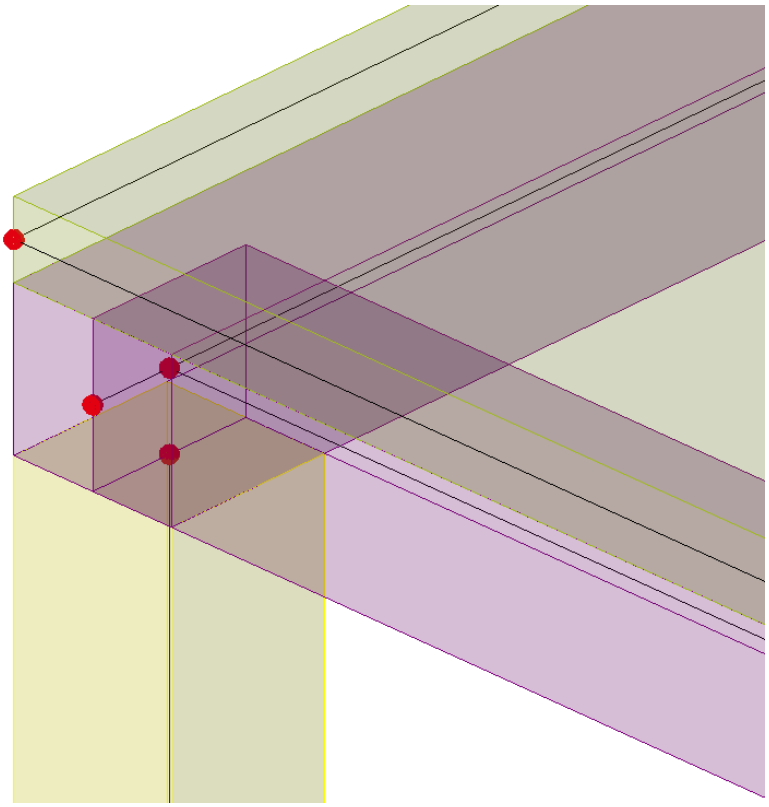


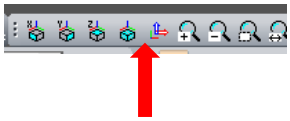
- Save your file and continue on to Part 3.

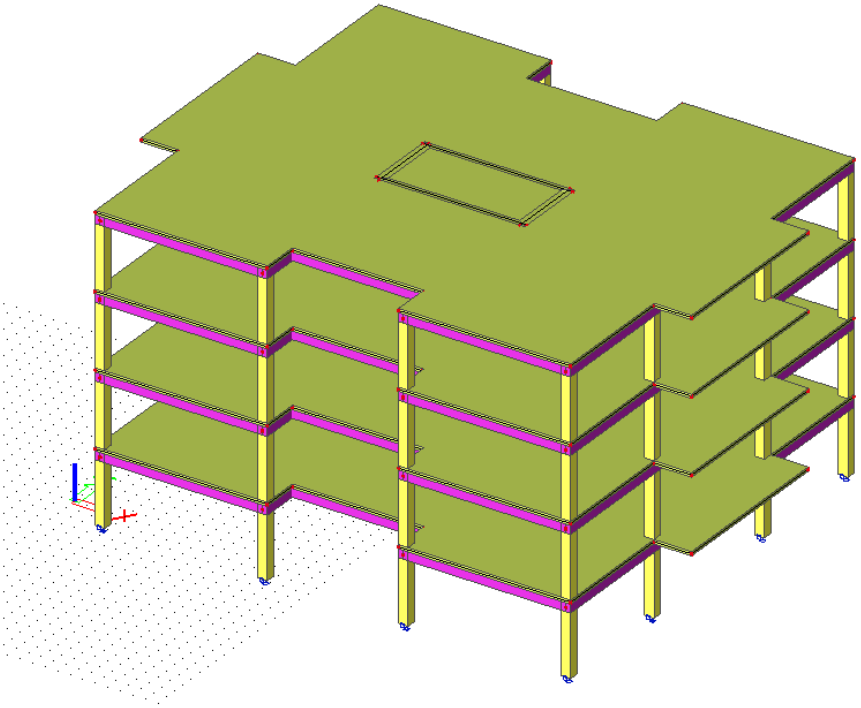
PART 3 – ALIGN AND CONNECT MODEL

1 Align

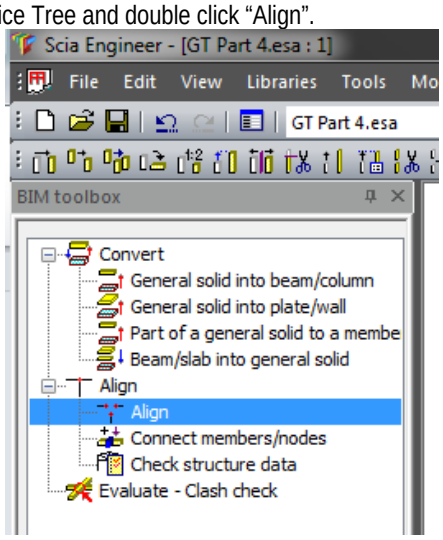
- A model created by an architect or other consultant most likely was not assembled with structural analysis in mind. In a finite element model, objects such as beams, columns and slabs are represented by their centerline or neutral axis and a stiffness matrix. Any other, more complex representation is for visualization purposes only. Our conversion from a volumetric architectural model to a simplified finite element model requires 1 more step. We need to make slight adjustments to the locations of the members and nodes to achieve transfer of forces and compatibility of deformation. This situation is illustrated in the image below.



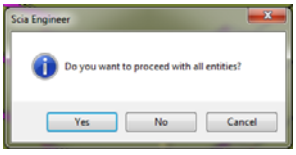
- Fortunately, SCIA Engineer's BIM toolbox has a system to help with this process.
- Click the  "AXO" button in the "View" toolbar to orient the workspace view.



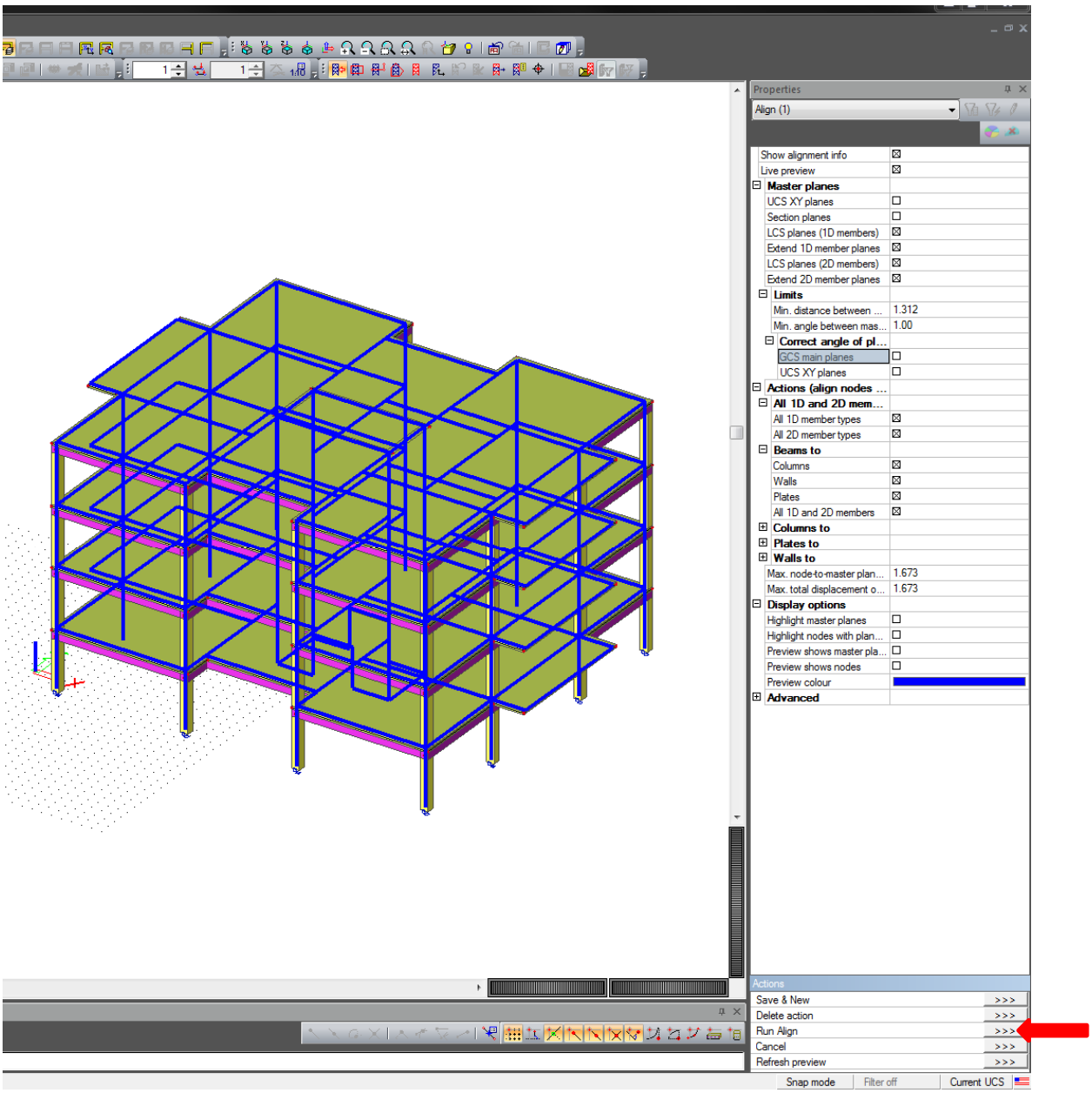
- Open the BIM toolbox in the Service Tree and double click “Align”.



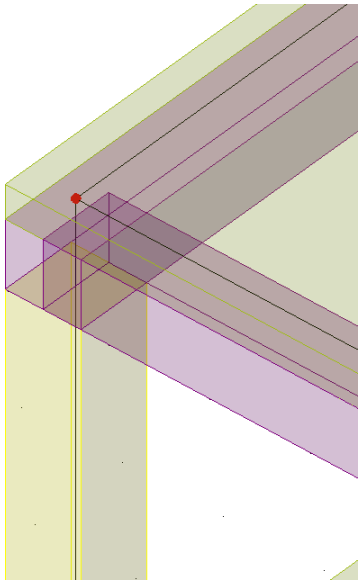
- When prompted to “Proceed with all entities” click “Yes”.



- The Properties Palette will now change and display preferences for the align command which we are about to perform. Set the preferences exactly as shown in the image below. You should see the bold blue preview of the aligned structure also.

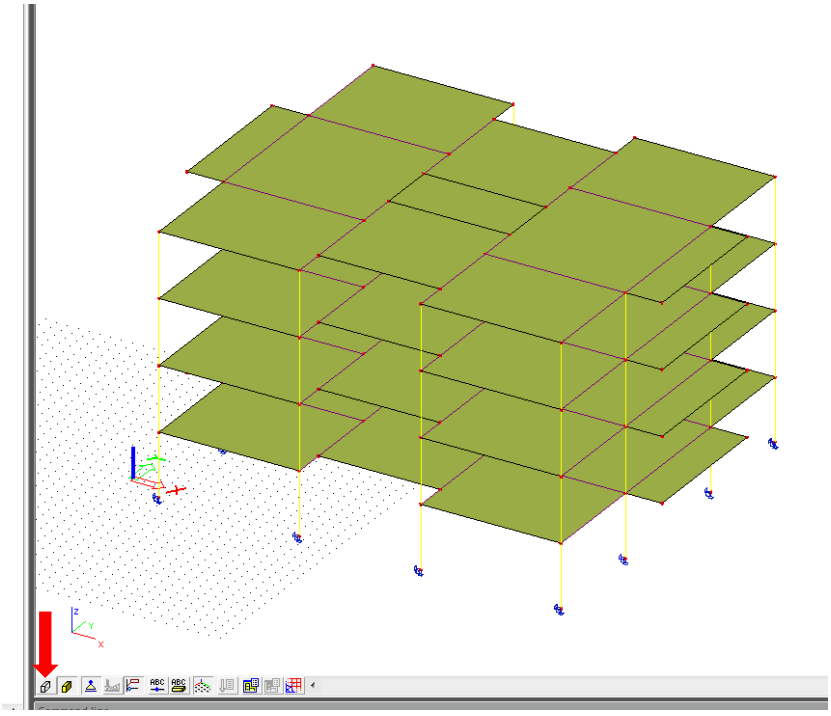


- Click the “Run Align” button in the “Actions” at bottom right corner of the SCIA Engineer window. The centrelines of the objects have now been aligned. Notice the difference in the transparent image below. The 2 beams, column and slab all join in the corner at a single node.

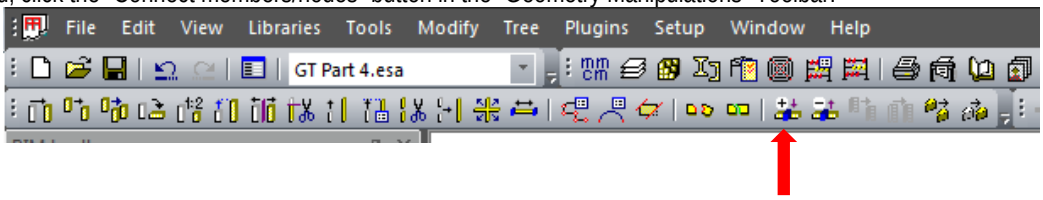


2 Connect

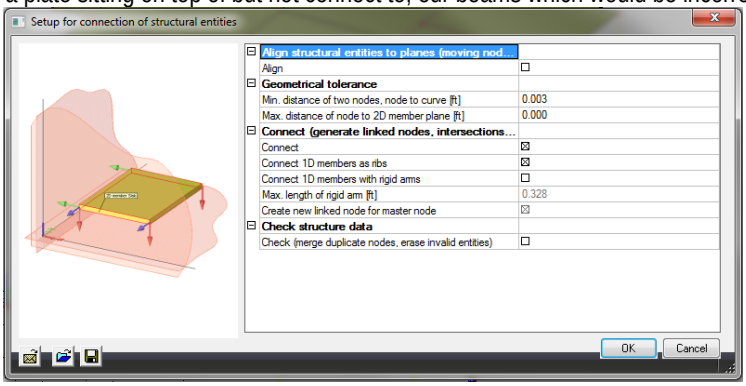
- Turn off the volumetric display of the model using the button in the quick view settings tool bar at the bottom left of the work space.



- Before we proceed with loading and analysis, we need to tell SCIA Engineer to connect our model. With the entire model de-selected, click the “Connect members/nodes” button in the “Geometry Manipulations” Toolbar.



- Click “Yes” to proceed with all entities.
- Make sure the “Connect 1D members as ribs” box is checked and click “OK”. This will connect our floor and roof beams to the floor slab. This is how a monolithically poured, concrete slab and beam system would behave (shear transfer). Otherwise our slab would behave like a plate sitting on top of but not connect to, our beams which would be incorrect.

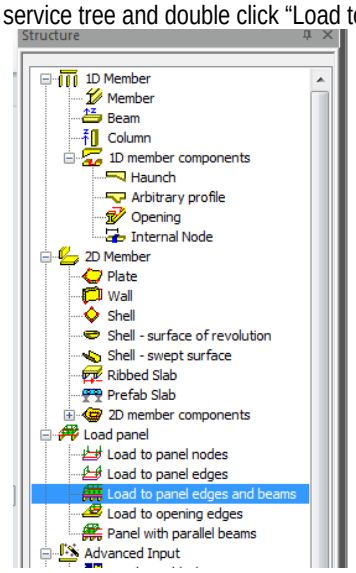


- Save your model and continue on to Part 4.

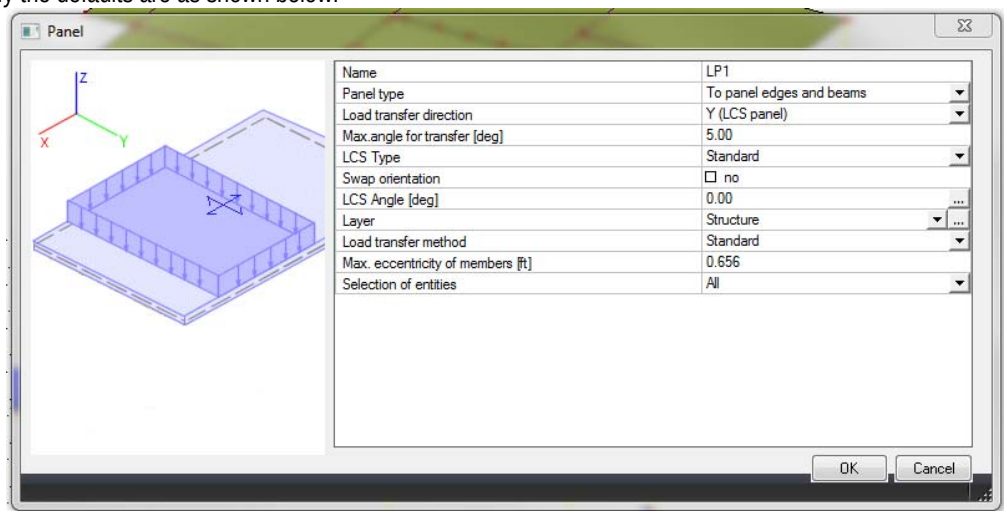
PART 4 – APPLY LOADING CONDITIONS

1 Load panels

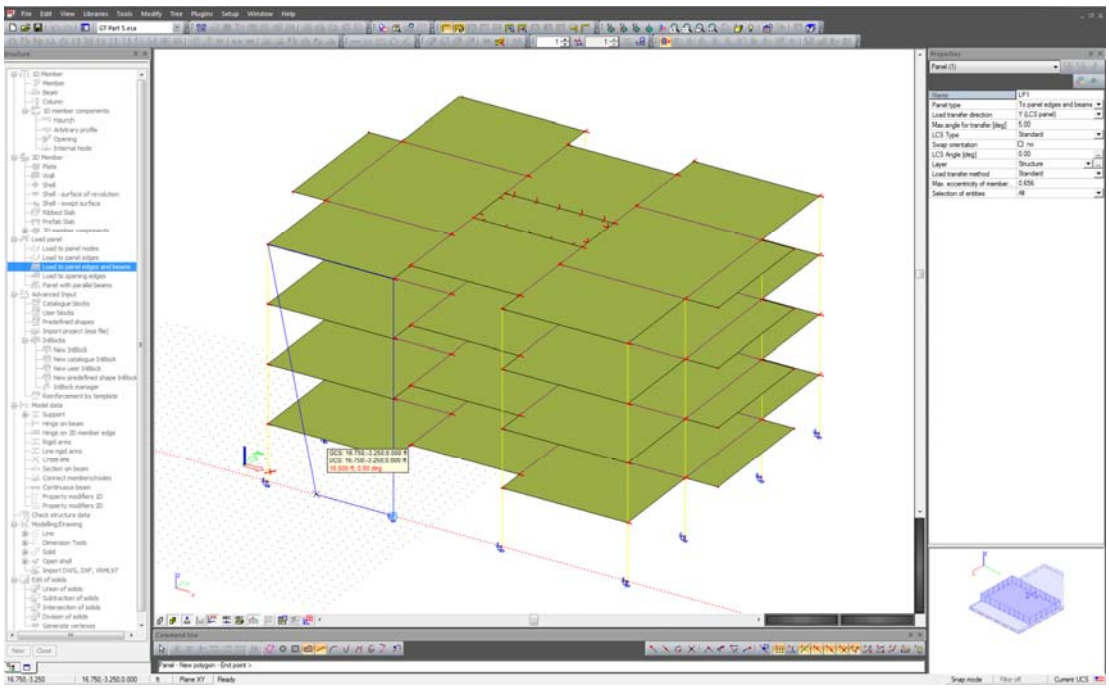
- The curtain wall cladding in this project will be designed by an outside consultant, so we will use a simplification of these elements in our analysis. SCIA Engineer provides a type of 2D object known as a “Load Panel” for this purpose. Load Panels distribute surface loads such as wind to other members, but do not influence the stiffness of the structure.
- Enter the “Structure” service from the service tree and double click “Load to panel edges and beams”.



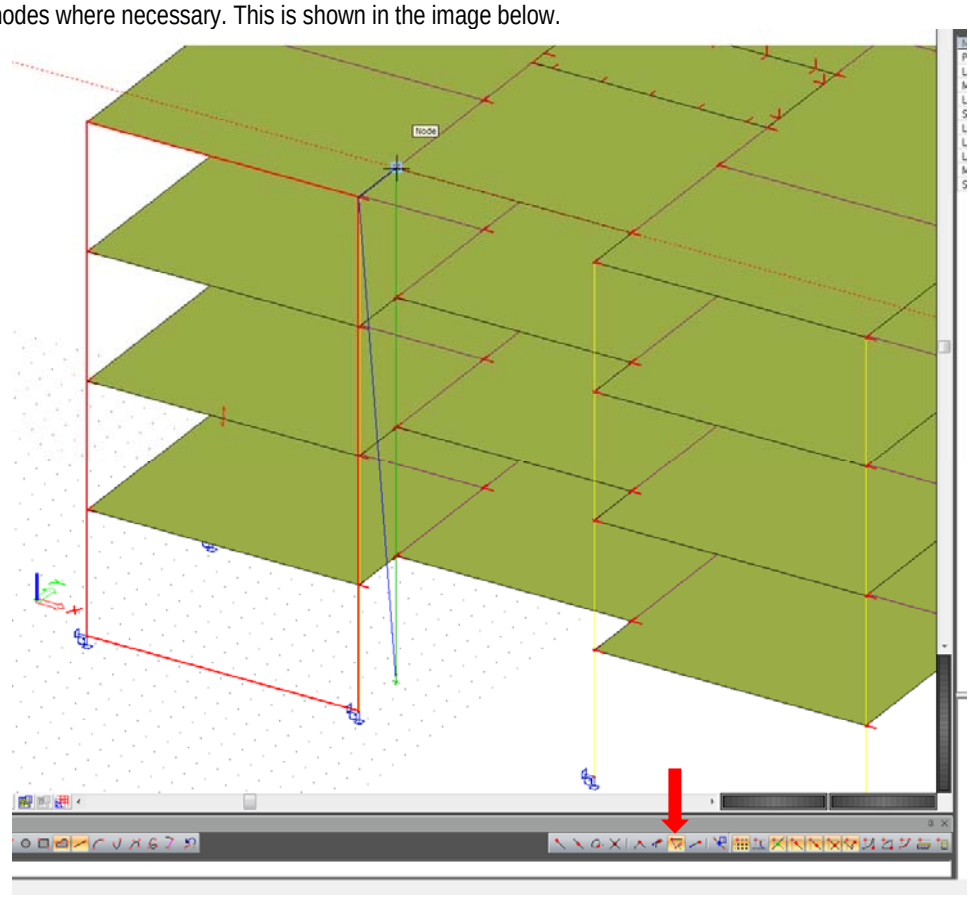
- Verify the defaults are as shown below.



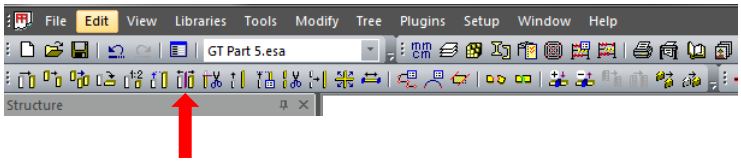
- You will now be in a SCIA Engineer “Sketching Mode”. Cover the exterior of the building with panels by clicking on the 4 corners of each planar panel as shown below. Pay close attention to “snapping” at each node. End each panel input (4 clicks) by clicking the right mouse button.



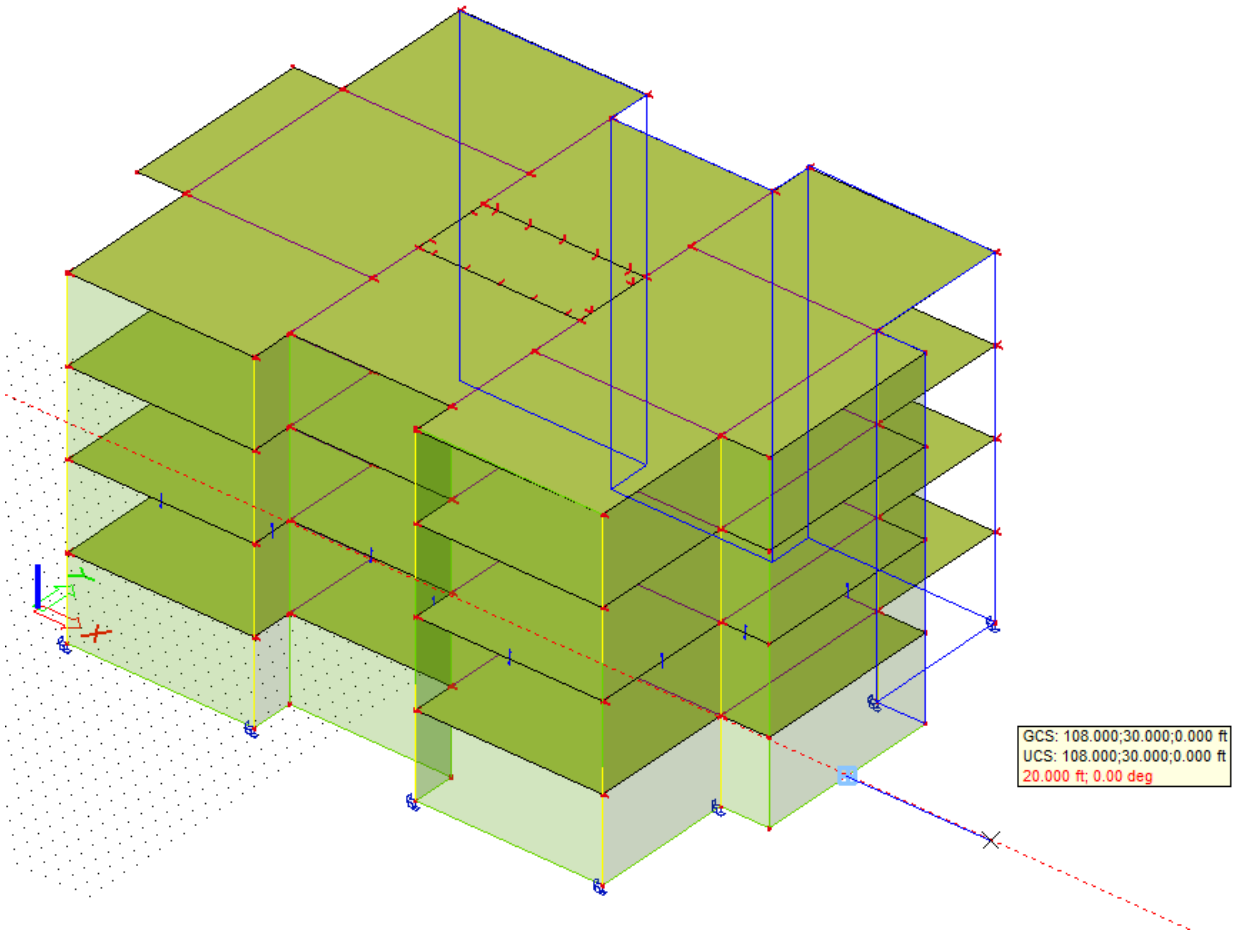
- Use the “Move to working plane” mode on the “Snapping Toolbar” to project down to the ground level from the roof nodes where necessary. This is shown in the image below.



- The “Mirror” tool is also useful in streamlining this process. Input half the panels and then click the “Mirror” button on the “Geometry manipulations” toolbar.



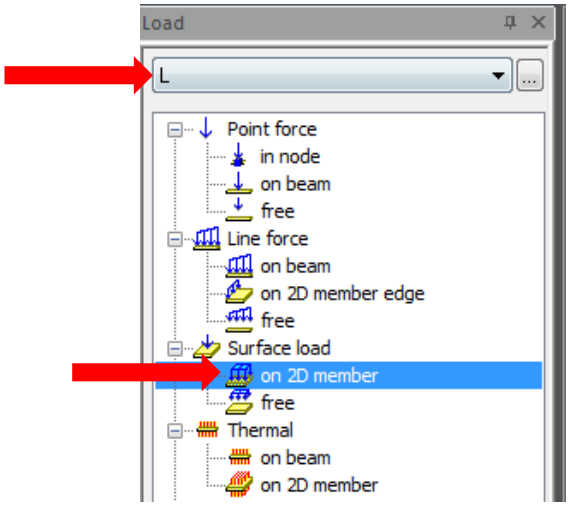
- Click to define the axis of reflection (the midpoint snapping mode will help with this and is enabled by default). A second click completes the action.



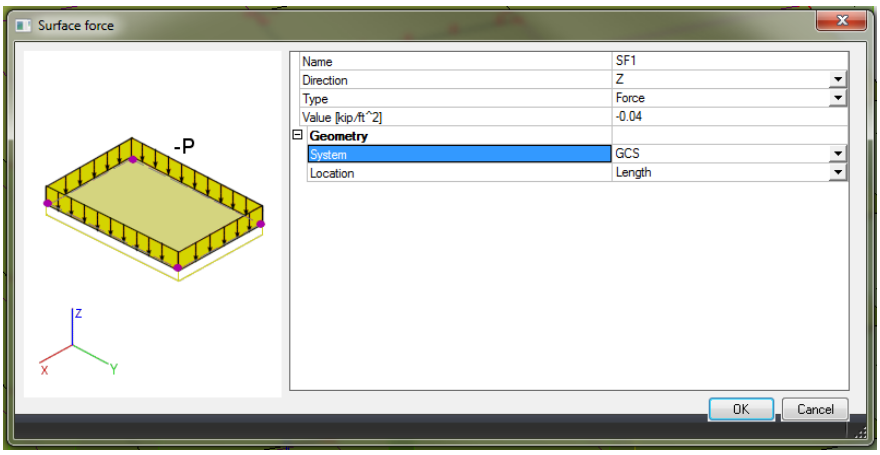
- Verify that all the walls of your building are clad with load panels.

2 Floor loading

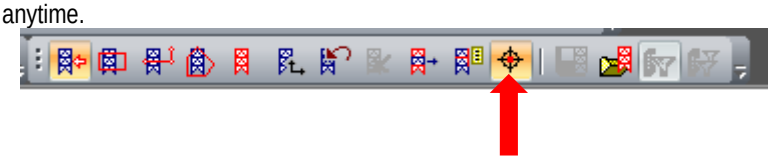
- Enter the “Load” service through the service tree.
- Load cases and combinations have already been prepared and provided in the template file you started with. Set the active load case to “L” (Live) as shown below.



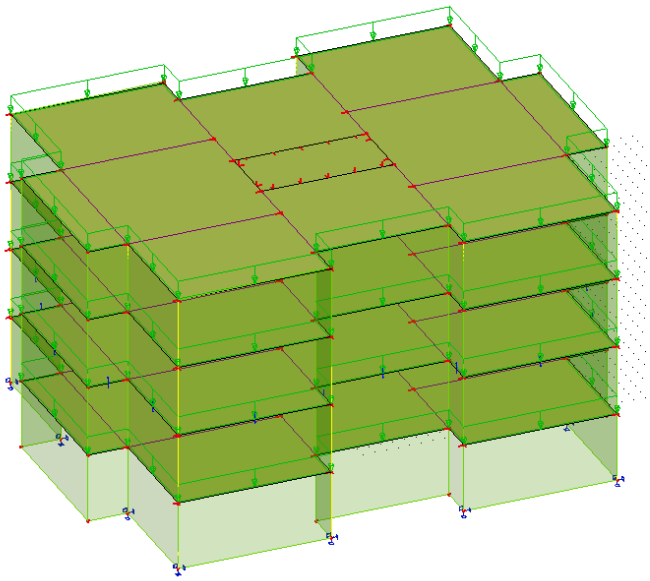
- Double click the “Surface load – on 2D member” button in the list of load types and set the options as shown below.



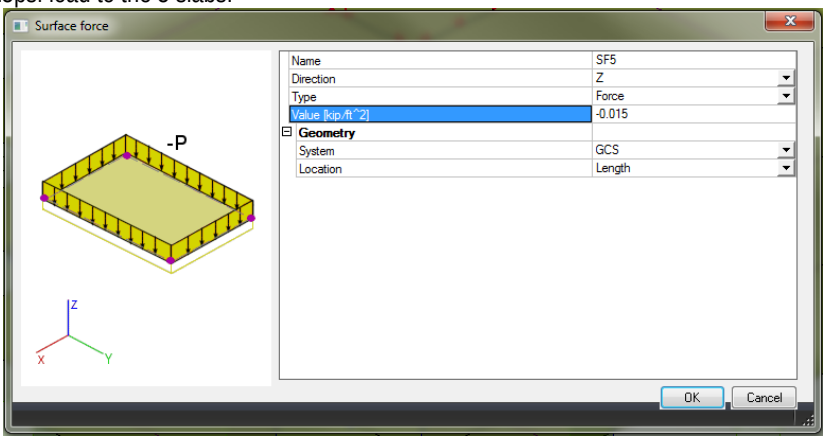
- Enable the “Visibility selection mode” in the “Selections” toolbar. This will make selection of 2D elements easier. This mode can be turned on and off anytime.



- Pick the 4 slabs to apply the -40psf live loading. Complete the action with the “Esc” button on the keyboard.

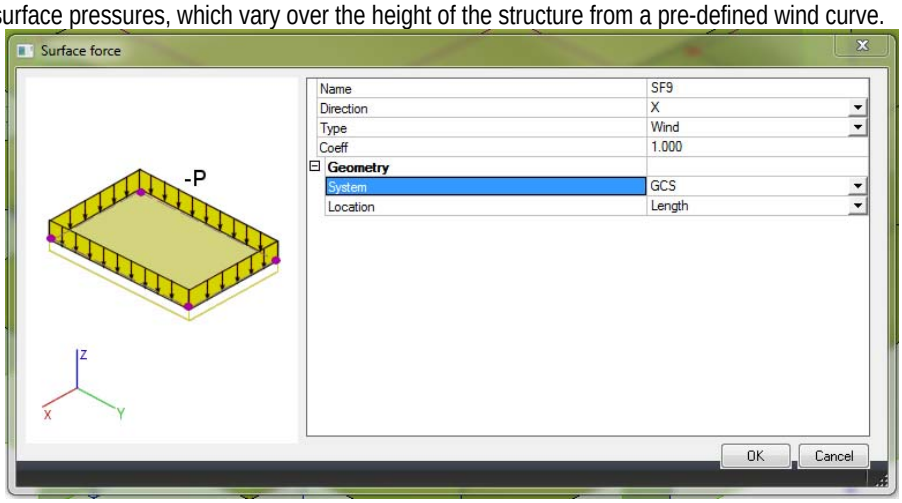


- Set the active load case to “Dsup”. This load case represents superimposed dead load for additional partitions, mechanical components, etc… The self-weight of the structural members is already included in the load case “Dself”. Repeat the process above applying a 15psf load to the 5 slabs.

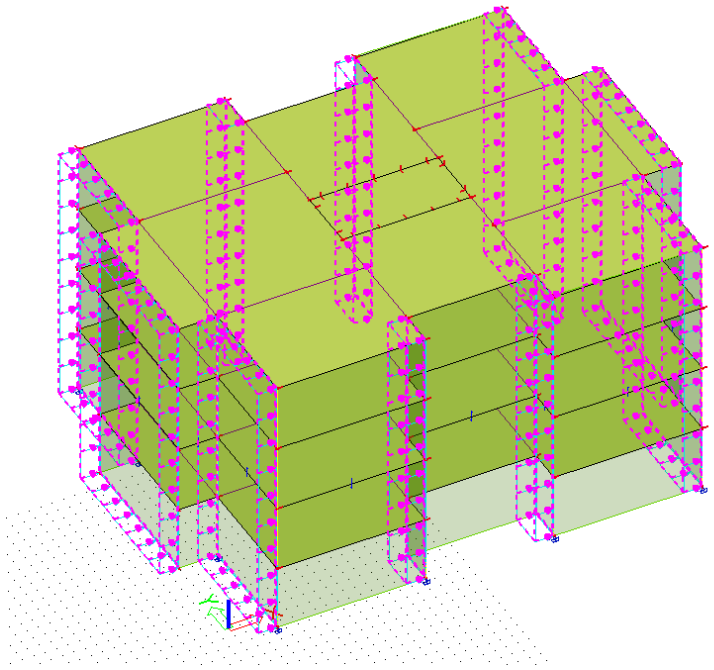


3 Wind Loading

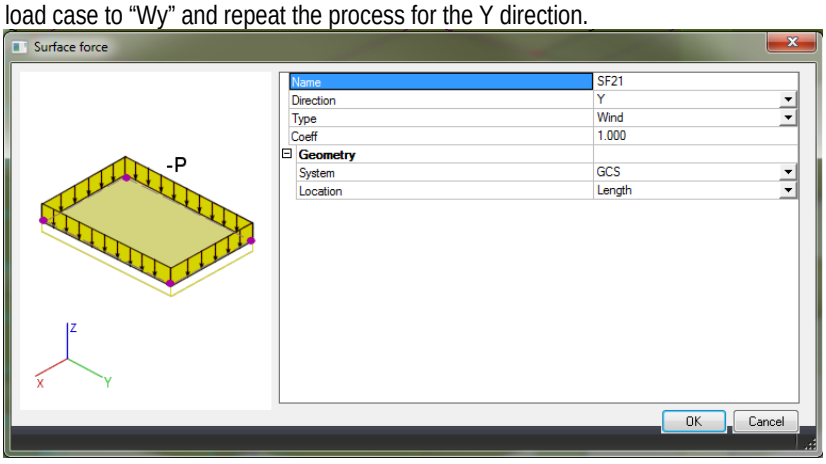
- Set the active load case to “Wx”. Double click “Surface load – on 2D member” again this time applying the settings shown below. Instead of specifying a force as we did for the live and dead loads we are going to set the “Type” of load to “Wind”. This will pull the surface pressures, which vary over the height of the structure from a pre-defined wind curve.

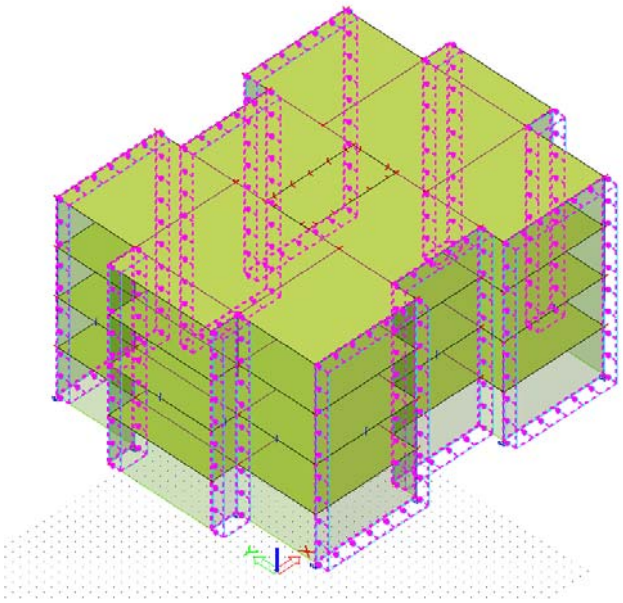


- Apply this loading to all the load panels which are perpendicular to the wind direction (X). It may be easier to complete this action with the “Visibility selection mode” disabled, due to the panels being transparent.



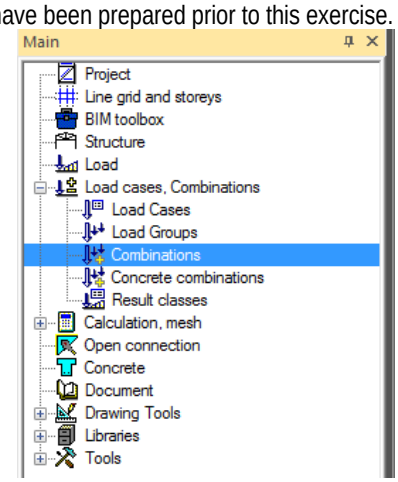
- Set the active load case to “Wy” and repeat the process for the Y direction.

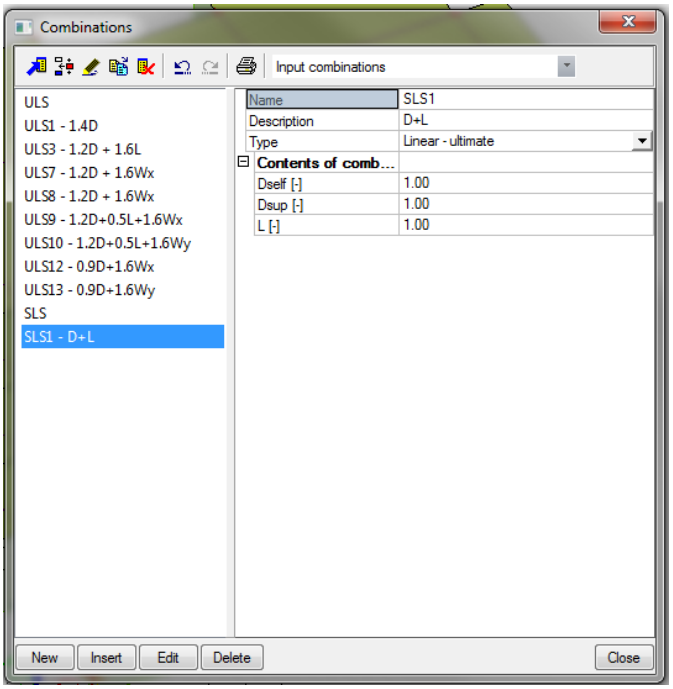




4 Load Combinations

- Click “Close” at the bottom of the service tree palette to return to the main tree. Now enter the “Combinations” service and review the load combinations that have been prepared prior to this exercise.



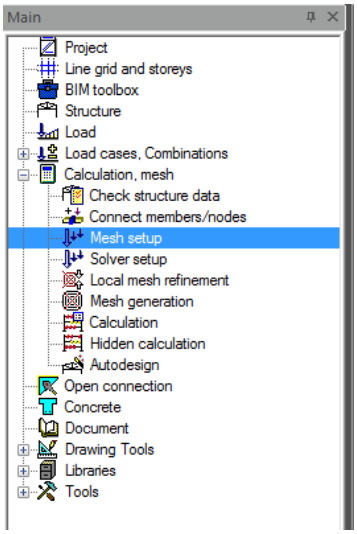


- ULS combinations are strength (factored) load combinations and envelopes.
- SLS combinations are serviceability (un-factored) load combinations and envelopes.
- Save your file and proceed to Part 5.

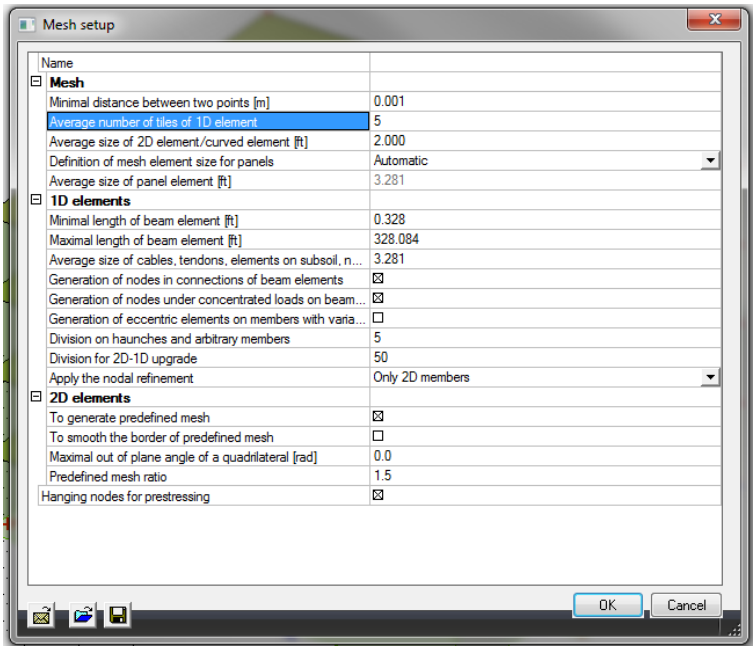
PART 5 – ANALYSIS AND RESULTS

1 Finite Element Mesh

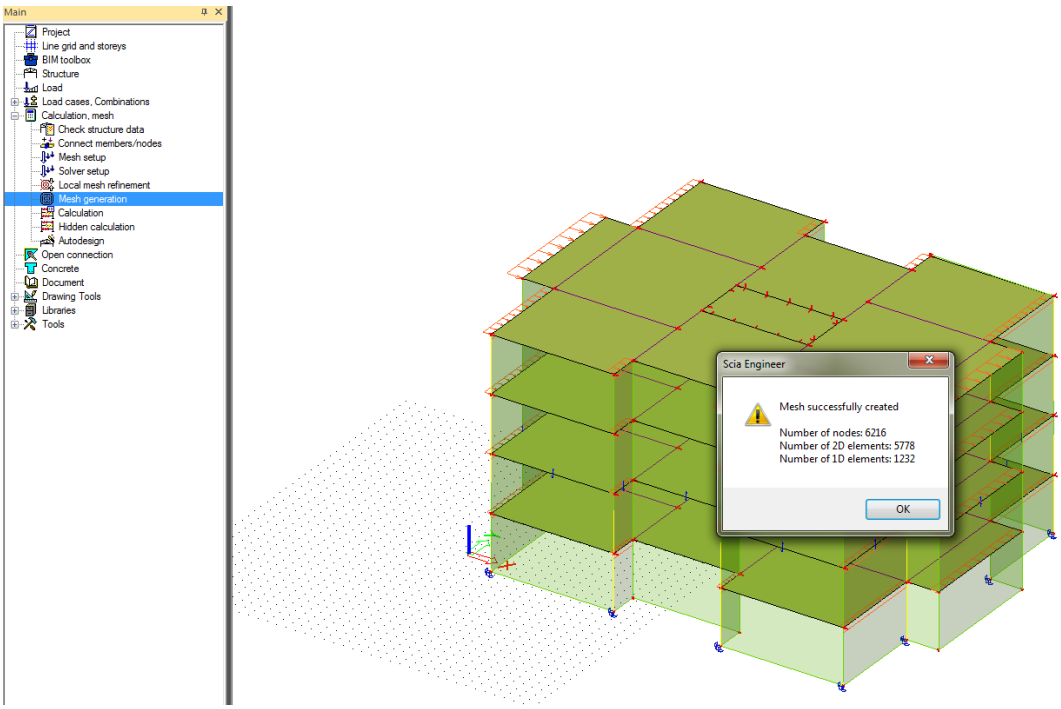
- The finite element mesh in SCIA Engineer is automatically generated prior to running the solver. This is a significant advantage over the more manual meshing process encountered in other software. The choice of mesh size and refinement can have an impact on the validity of results when using the finite element method, therefore it is always important to visually verify the mesh before accepting any results.
- Enter the “Mesh Settings” from the service tree.



- Set the number of 1D mesh elements to 5 and the 2D mesh size to 2ft as shown below.

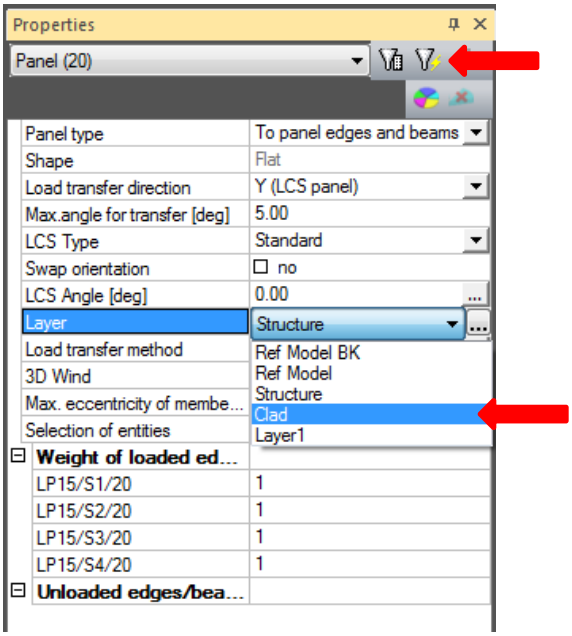


- Click “OK” to apply the settings.
- Click “Yes” if you receive a warning about changing the settings.
- Double click “Mesh generation” in the service tree. You should be presented with a dialog displaying the results of the meshing process.

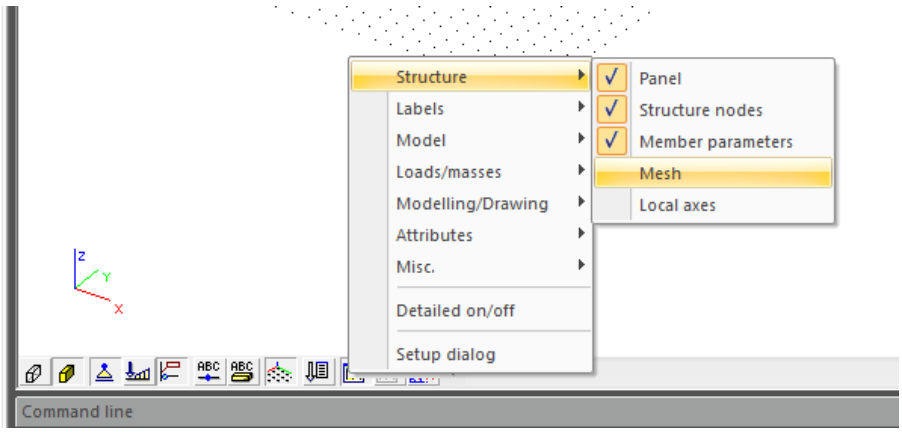


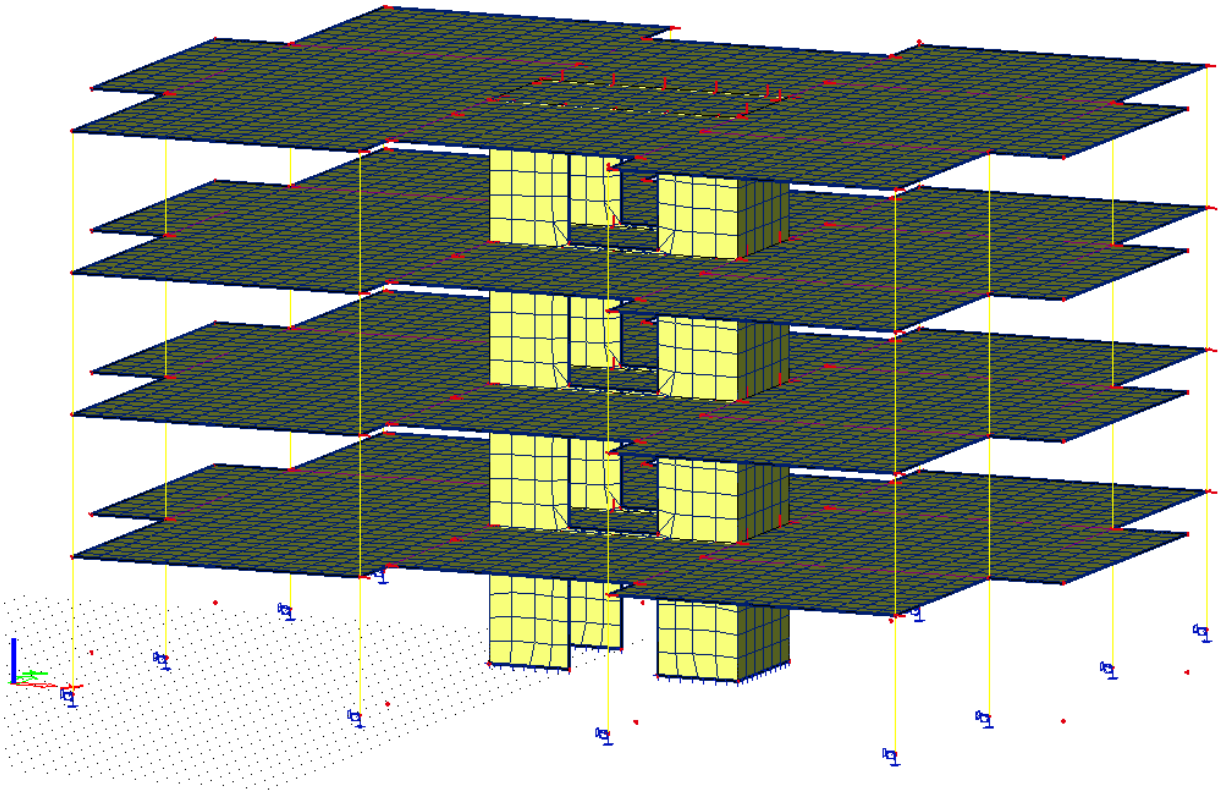
- Click “OK” to accept.
- Turn off the visibility of loads in the quick view settings tool bar.

- Select all the load panels using the “Select Elements by Property” button and a matching property.
- Set the layer of the load panels to “Clad”



- Enable the mesh visibility in the quick visibility settings.

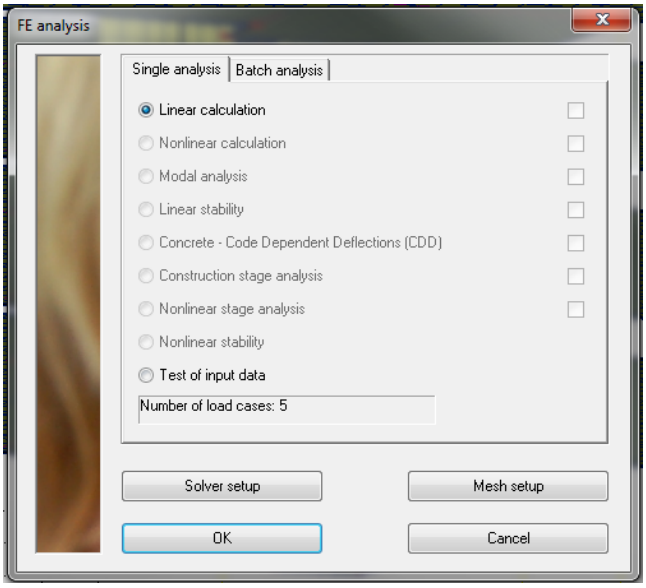




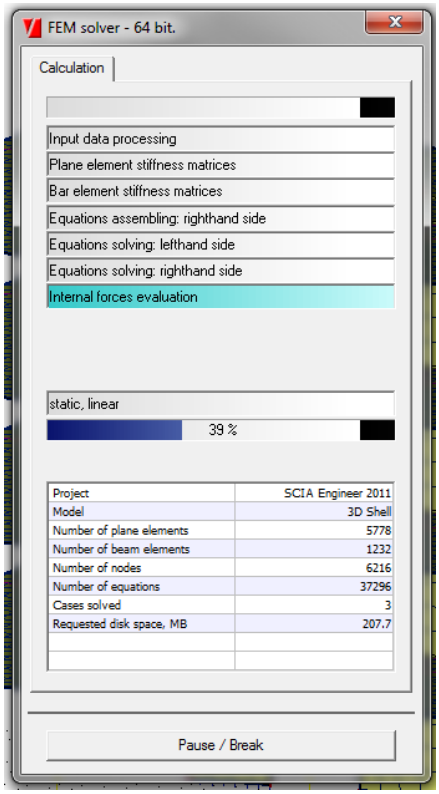
- Verify that the generated mesh is acceptable.

2 Static Linear Analysis

- Double click “Calculation” in the service tree
- Select “Linear Calculation” and click “OK”



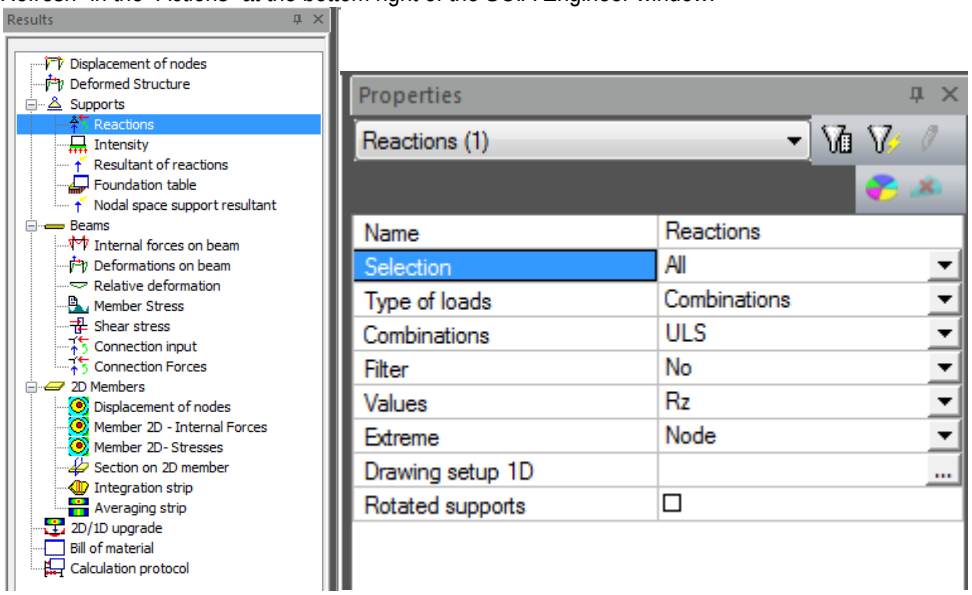
- During the calculation process you should see a solver status window.

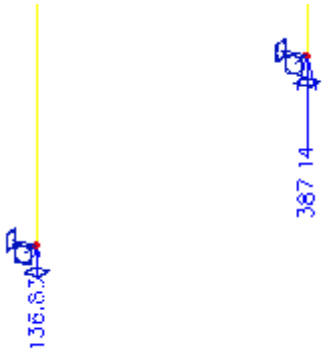


3 Results

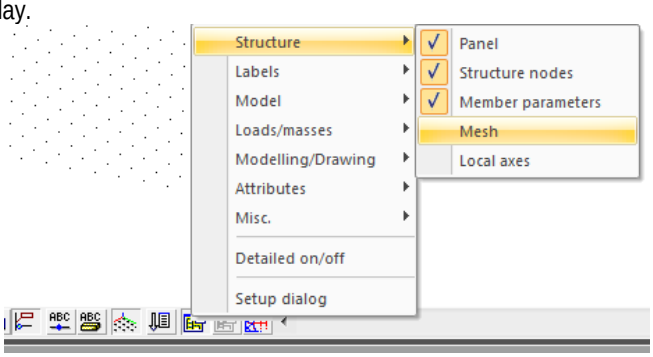
- When the solver is complete you will see that a new service “Results” has appeared in the tree.

- Enter this “Results” service, select “Reactions” and set the following properties in the properties palette.
- Click “Refresh” in the “Actions” at the bottom right of the SCIA Engineer window.

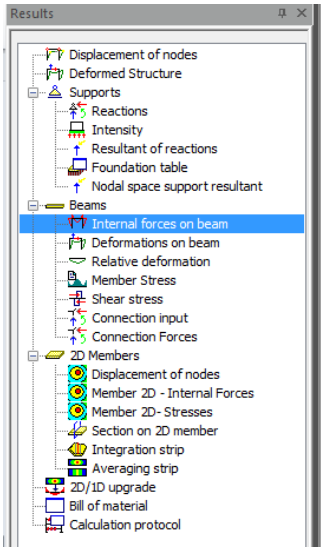


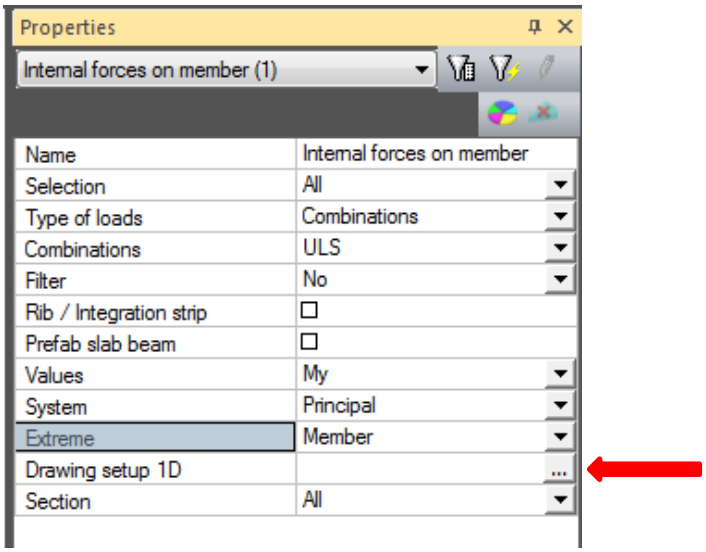


- These are the critical support reactions for the ULS envelope.
- Turn off the mesh display.

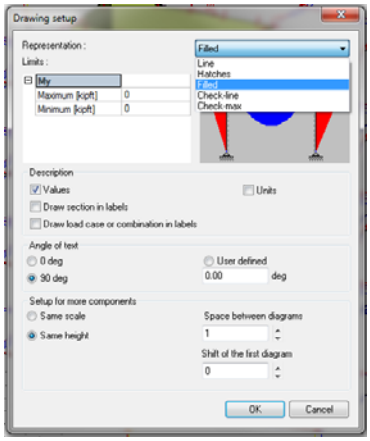


- Select “Internal Forces on Beam” in the service tree. Set the properties as shown below.

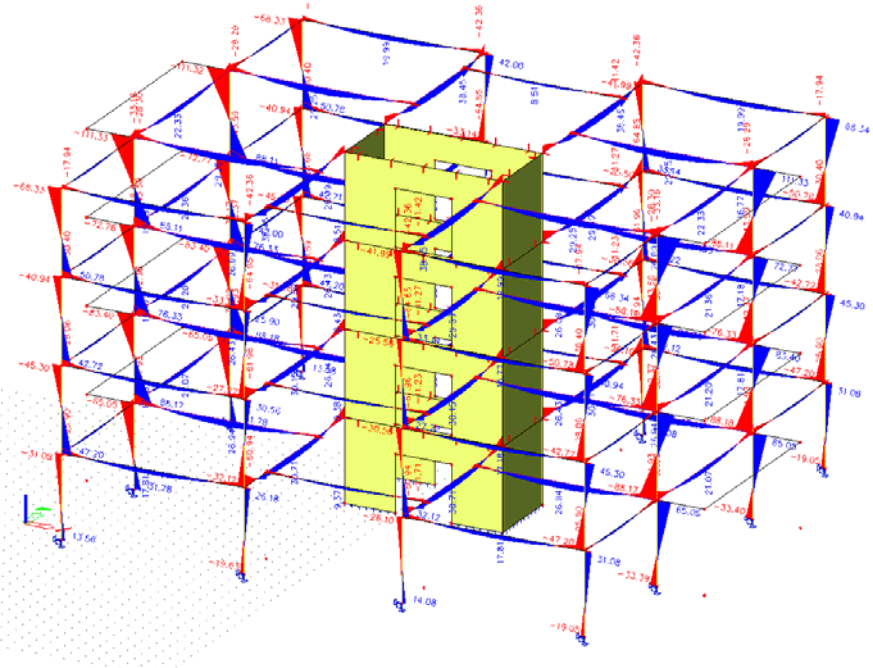




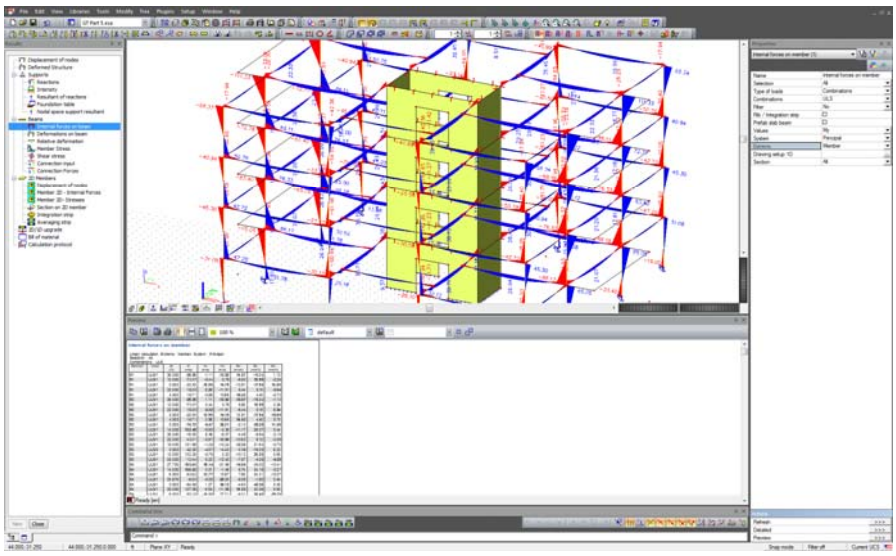
- Click the ellipsis button next to “Drawing setup 1D” in the properties palette.
- Set the “Representation” to “Filled”



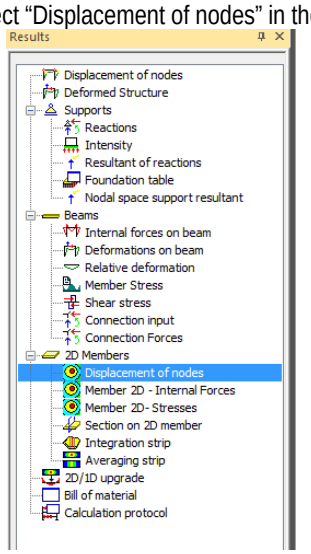
- Refresh the results display. You should see the strength level Y axis moment envelopes for the beams and columns.

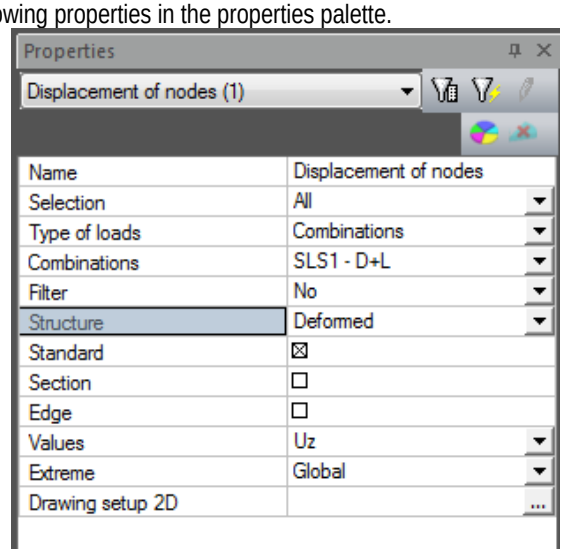


- Click “Preview” in the “Actions” at the bottom right corner of the SCIA Engineer window.

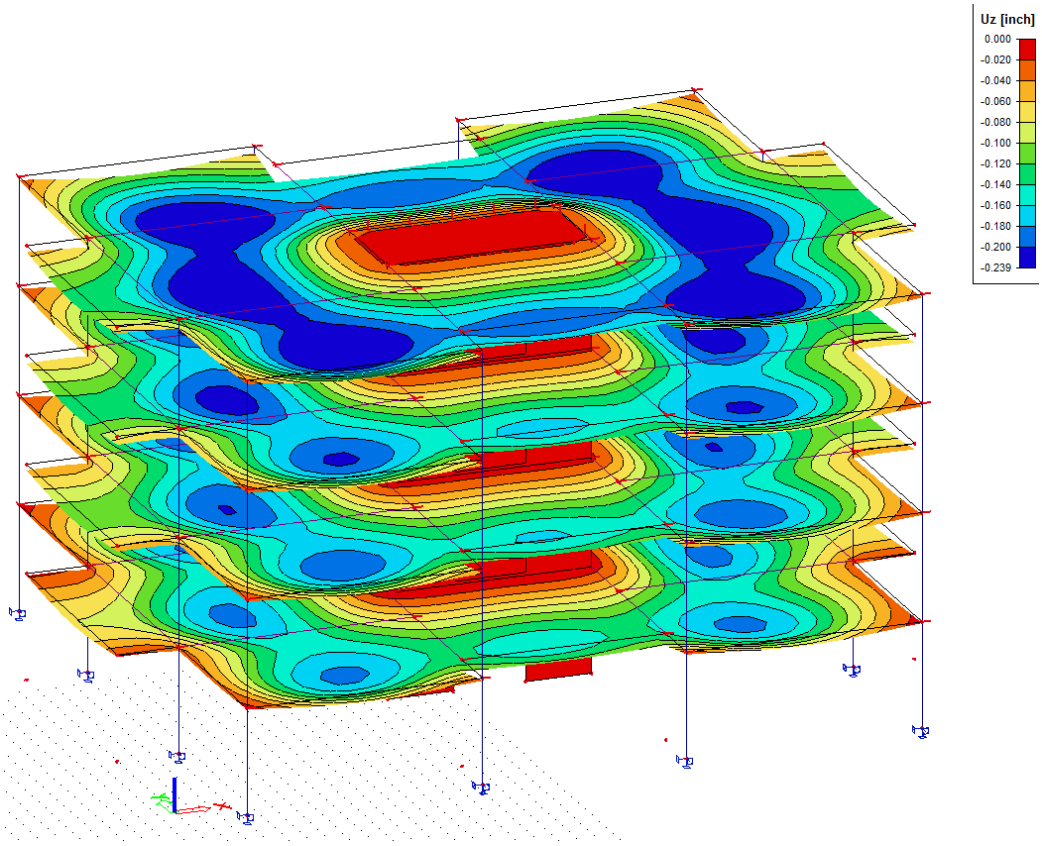


- The “Preview” window allows you to display results and other data in tabular form. In the current table the critical results for each 1D member are displayed. Close the preview window by clicking the “X” button at the top right of the table.
- Select “Displacement of nodes” in the tree and set the following properties in the properties palette.

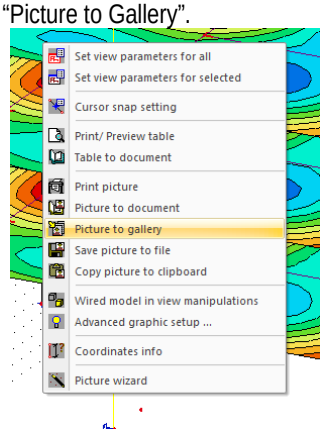




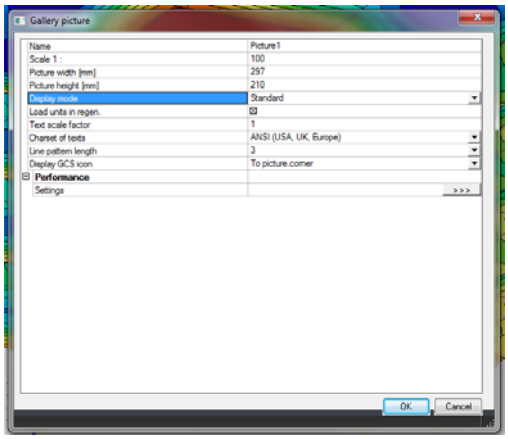
- Refresh the display. This is the vertical deformation under service level loads. This is for elastic behaviour only. In a real project we would need to account for cracking, creep and other non-linear and time dependant effects.



- Right click in the 3D window and select “Picture to Gallery”.



- Set the “Display Mode” to “Standard” and click “OK”. This will save this view of the model in our “Picture Gallery”. We will use this in Part 6.



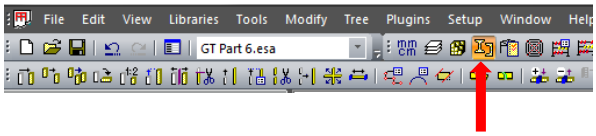
- As you can see there are many ways to display the analysis results in SCIA Engineer.
- Save your file and proceed to Part 6.

PART 6 – STRUCTURAL MODEL EXPORT

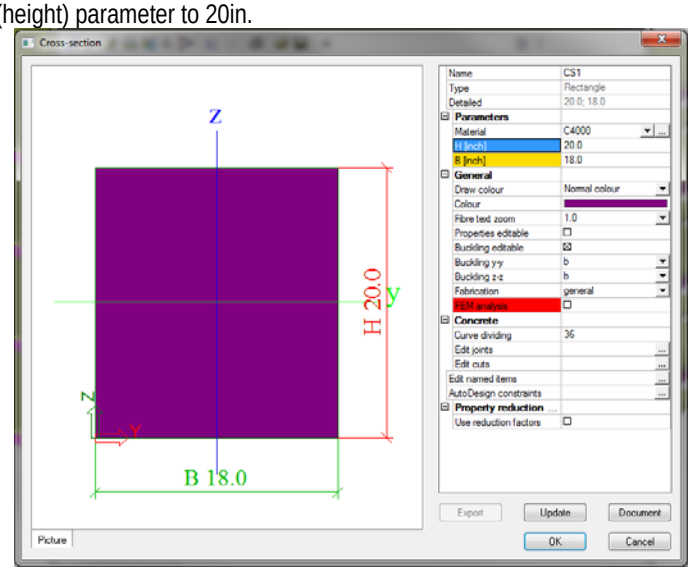
One of the features that differentiate SCIA Engineer from traditional Finite Element Analysis software is that it maintains 2 versions of the model in parallel. SCIA Engineer has the traditional "Analysis Model" which consists of bars, nodes, shells etc. … This model is what you are actually analysing, but is not very useful for communicating design intent or for exchange with another consultant. The second model representation is the "Structural Model" which is dimensionally accurate and can be used to extract drawings, generate renderings and be exported for collaboration.

It would be very unlikely that the member sizes defined by the architect would be correct from an engineering point of view. In this exercise we have determined that the beam sizes are inadequate and we need to propose a change and communicate this back to the architect through a revised model.

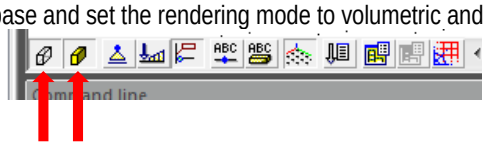
- Open the Cross-Sections Database.



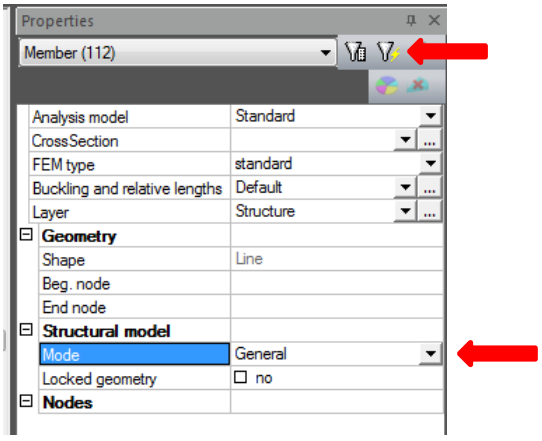
- Select "CS1" and click "Edit"
- Change the "H"(height) parameter to 20in.



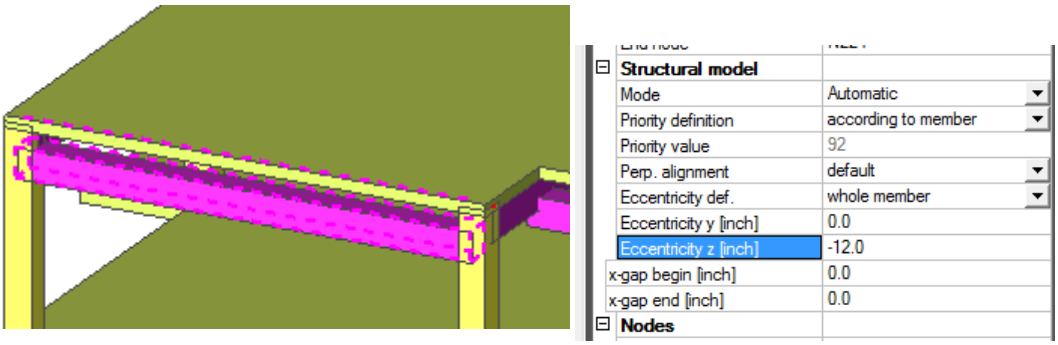
- Exit the cross section database and set the rendering mode to volumetric and filled.



- Select all 1D members with the property "Structural model – Mode" set to "General. If executed correctly you should have 112 members selected.

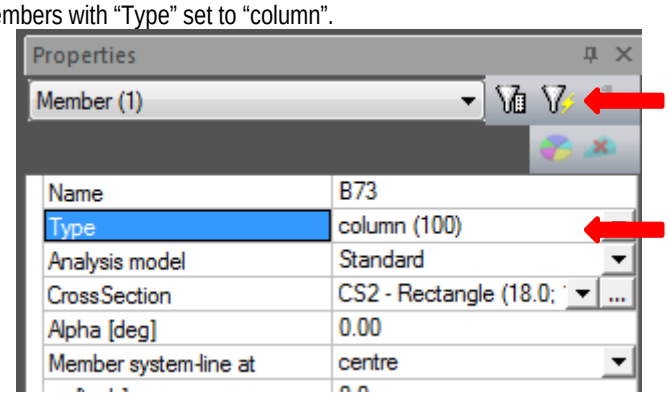


- Set “Structural model – Mode” to “Automatic”. These modes determine how much flexibility the user has in manually adjusting the “Structural Model” in relation to the “Analytical Model”. In our case we want SCIA Engineer to take care of most of this automatically.
- Select all the beams which have the “Structural model – Eccentricity z” set to -12in. Set this parameter to 0in.

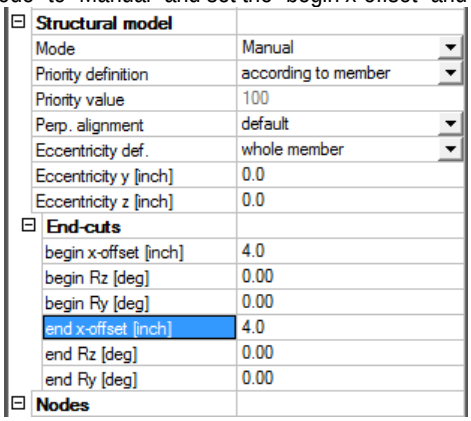


- Click the “Generate structural model” button to refresh the view.

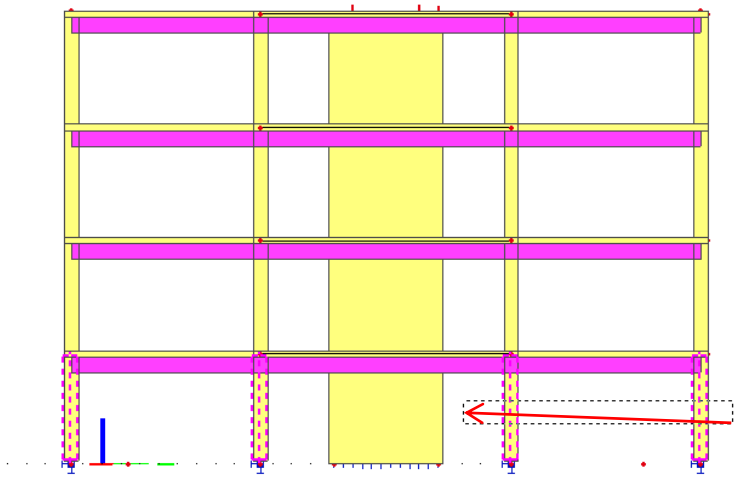
- Select all the 1D members with “Type” set to “column”.



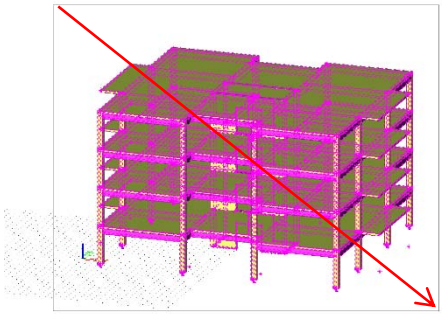
- Set the “Structural model – Mode” to “Manual” and set the “begin x-offset” and “end x-offset” parameters to 4in.



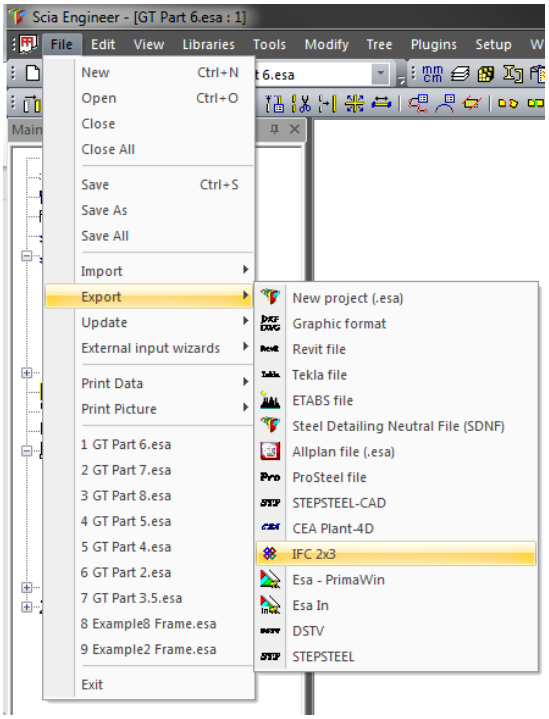
- Set your view to be from the X direction  . Drag a selection box across the first floor columns from right to left. When you select from right to left all objects crossed by the selection box are selected. When you select from left to right only objects completely inside of the selection box are selected.



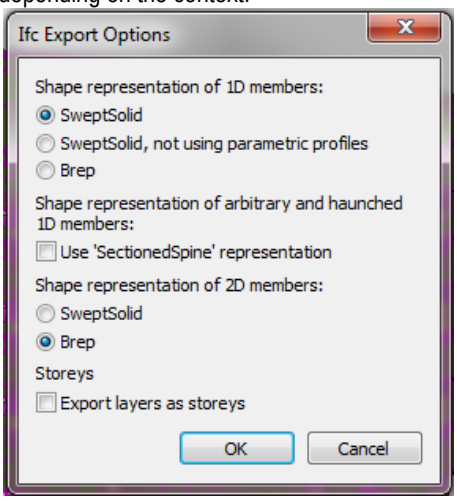
- Set the “begin x-offset” property to 0in.
- Refresh the structural model.
- Drag a selection box around the entire model from left to right. When we export our revised model to IFC, we don’t need to include the reference model layers. An easy way to filter the geometry to export is through selection.



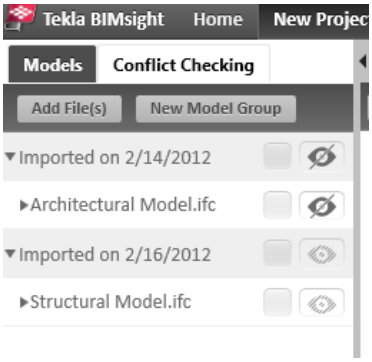
- Select “File – Export – IFC 2X3” from the menu.



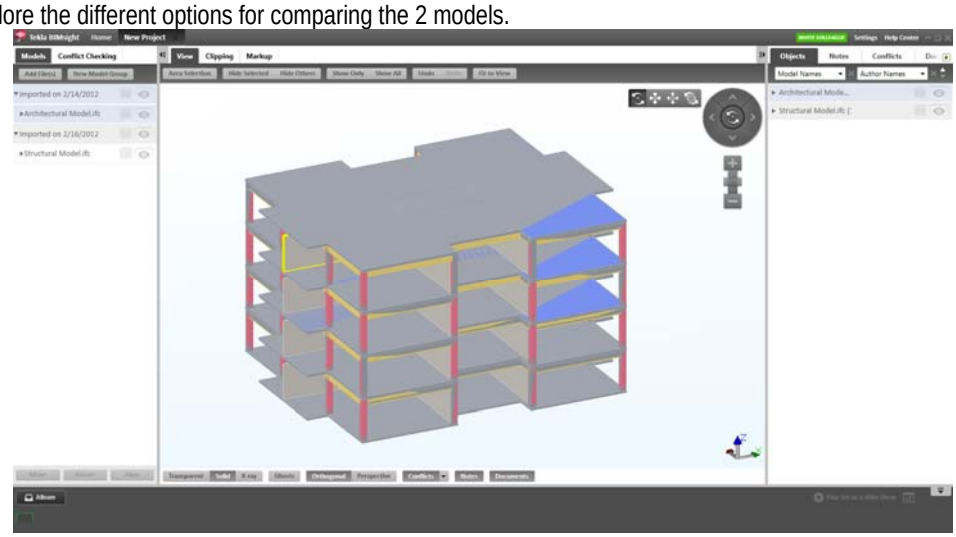
- Select a suitable name and location for the exported file.
- When prompted with options for the geometrical representation, set the parameters as shown below. Different software packages expect geometry to be represented using different data structures. “SweptSolid” is a simple extrusion with a profile and path. Most software can support this geometry type and will convert it into a native object of some type. “Brep” is short for boundary representation. This data structure supports more detail than swept solids, but typically does not get converted into a native object type in the target application. There are benefits and shortcomings to each method depending on the context.



- Open the Tekla BIMsight project you created in Part 1. Click “Add File” and select the file you just exported from SCIA Engineer.



- Explore the different options for comparing the 2 models.

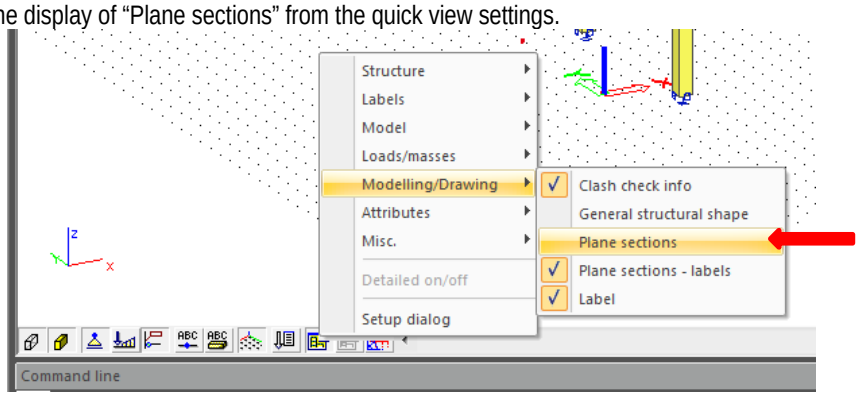


- Save your SCIA Engineer file and proceed to Part 7.

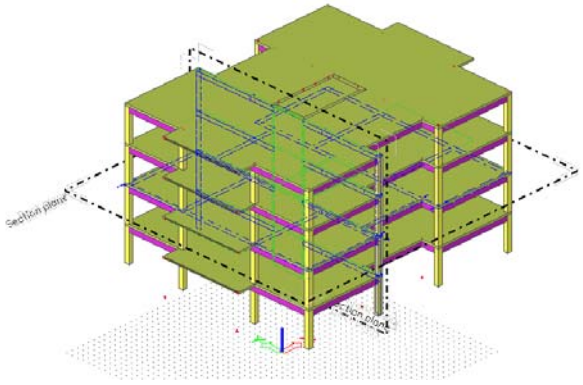
PART 7 – DRAWINGS

One of the immediate benefits of constructing a BIM model as opposed to traditional CAD drawings is coordinated documentation. In the past when a change was made on a 2D drawing, any other related drawings or layout pages also had to be manually reviewed and revised. This is a major source of potential errors. When documents are derived from a 3D BIM model, much of this manual coordination can be automated. SCIA Engineer offers a system for extracting drawing views from your model. These drawings are generated based on styles that the user can predefine to company standards and can be placed on layout pages for printing.

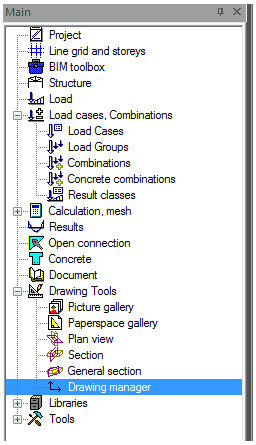
- Enable the display of “Plane sections” from the quick view settings.

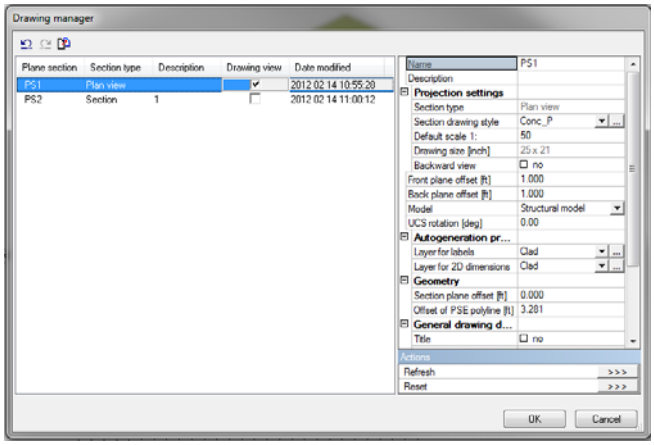


- You should see 2 section planes. 1 plan view and 1 section view. These drawing sections were predefined in the template file to save time during this exercise.

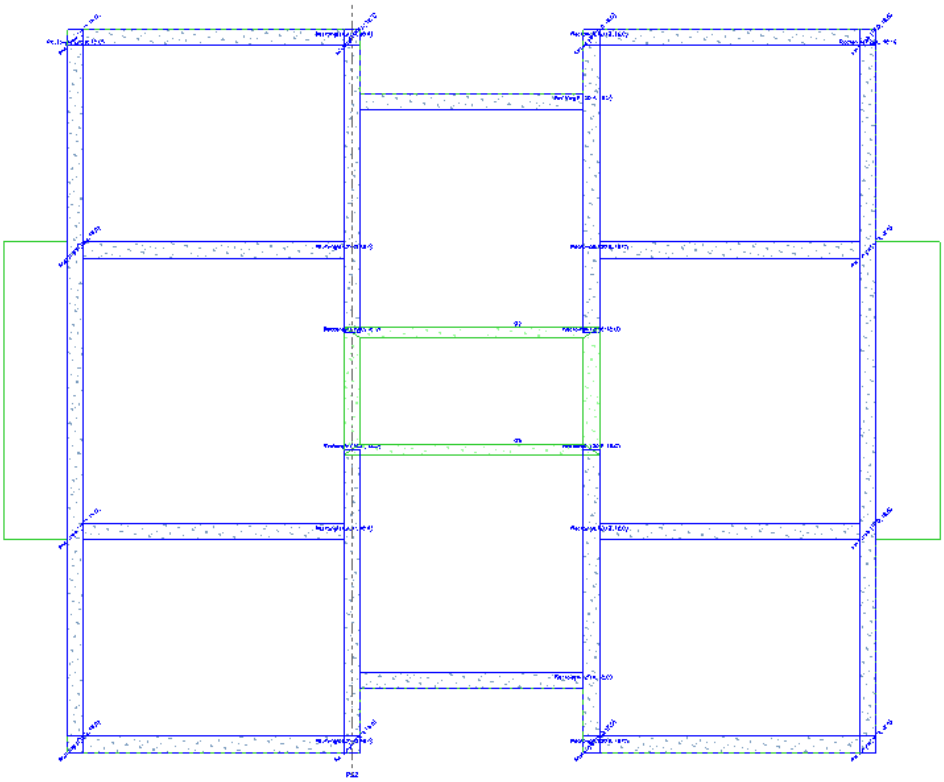


- Enter the drawing manger from the Service Tree. In the “Drawing Manger” mark the checkbox next to PS1 and click “OK”.

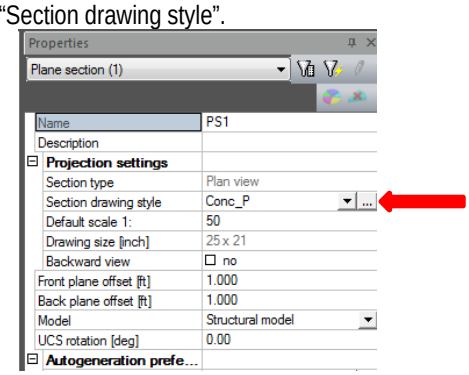




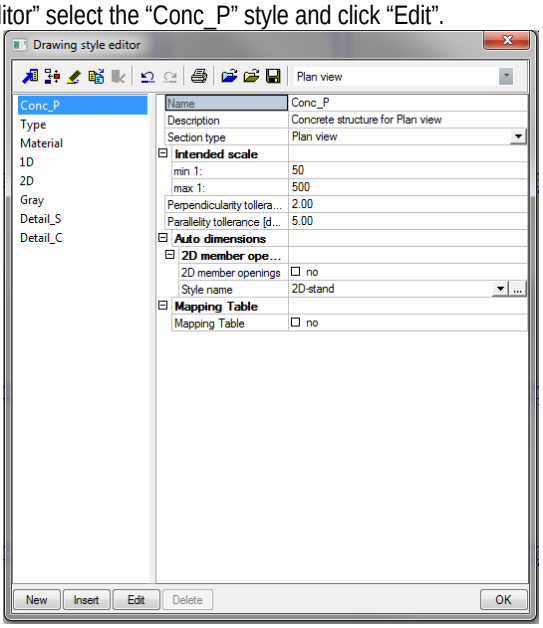
- You should now be in the SCIA Engineer 2D Drawing environment. The drawing generated is based on a set of criteria and associated line types, hatches, labels etc…



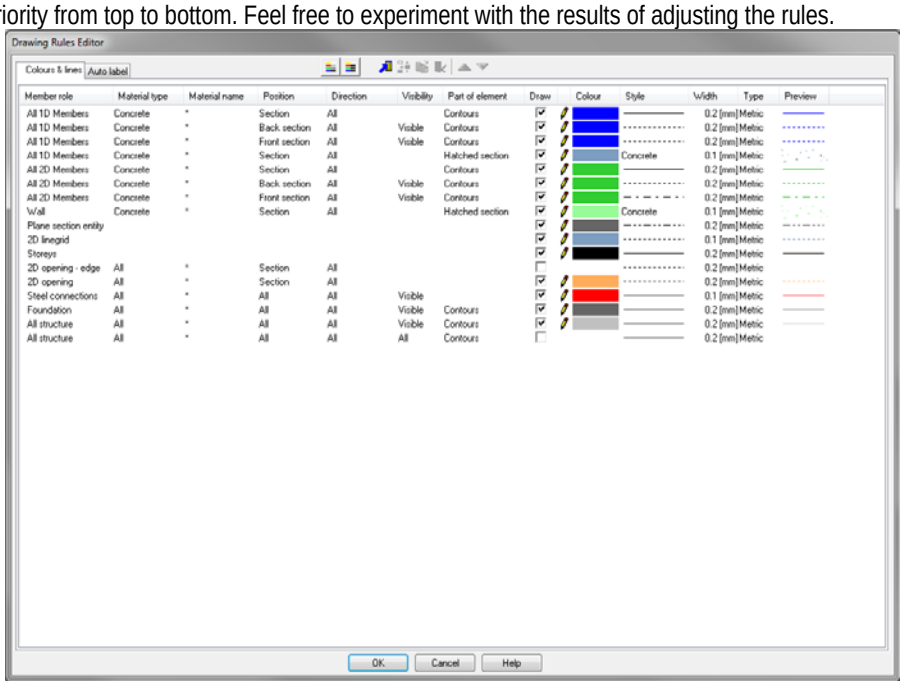
- Click the ellipsis button next to “Section drawing style”.



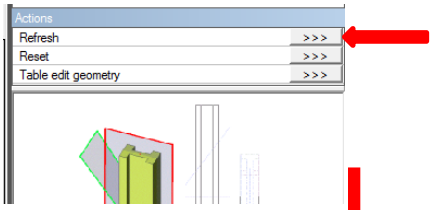
- In the “Drawing style editor” select the “Conc_P” style and click “Edit”.



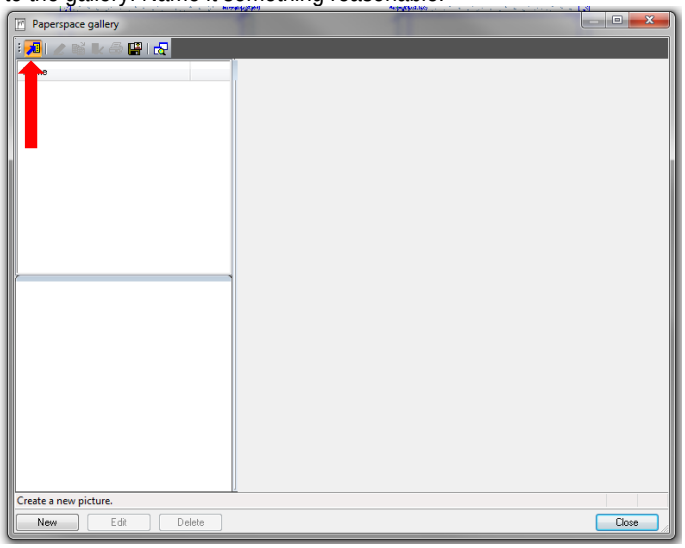
- This is the drawing “style” definition. The general idea is that members and objects that meet criteria in the list on the left are drawn using attributes chosen on the right. Additional “rules” can be added to the list and they are organised by priority from top to bottom. Feel free to experiment with the results of adjusting the rules.



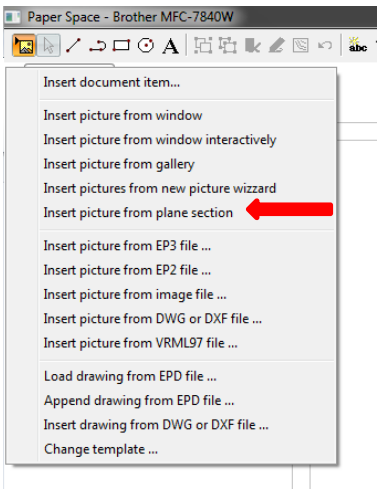
- Return to the drawing environment and click “Refresh” in the “Actions” to refresh the drawing.



- Click “Paperspace gallery” in the tool bars. 
- Add a new page to the gallery. Name it something reasonable.



- SCIA Engineer provides some default title blocks and you are also able to construct or add your own by importing .dwg or .dxf files.
- Select “Insert picture from plane section” from the drawing menu.



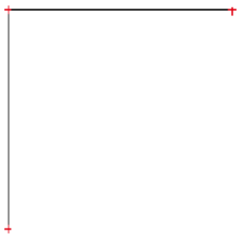
- Choose a drawing from the drawing manager and place it on the layout.
- There are many other items that can be placed on the layout pages including 3D views from the picture gallery, tables from the calculation reports and drawings from other CAD programs.

Conclusion

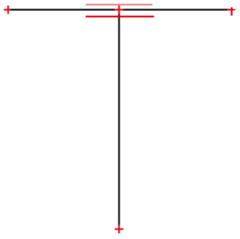
In this exercise we explored how members of the design team can share BIM models from various sources and authoring tools using open file formats like IFC. We also walked through the steps necessary to utilize this data in performing typical analysis and preparation of construction documentation. I hope this exercise has been interesting and informative. Please send any questions or comments to the author.

ANNEXES

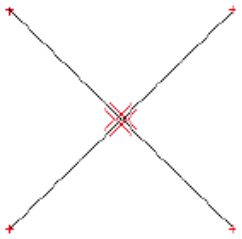
Annex 1: Connection of entities



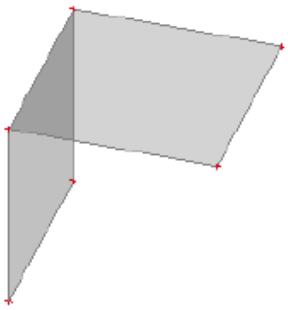
OK
Beams are connected



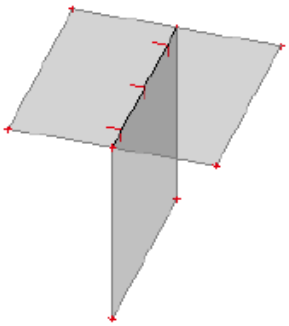
Structure menu
> Model data
> Connect members/ nodes



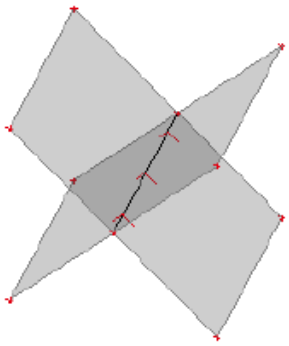
Structure menu
> Model data
> Cross-link



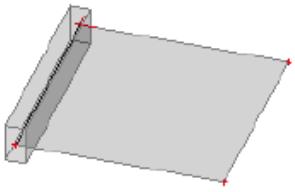
OK
Edges are connected



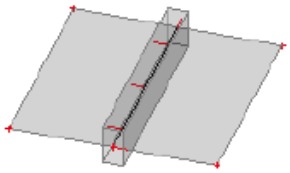
Structure menu
> Model data
> Connect members/ nodes



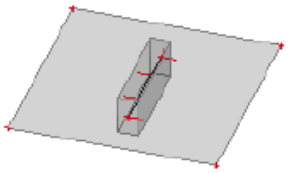
Structure menu
> Model data
> Connect members/ nodes



OK
Edge and beam are connected



*Connect members/ nodes: only nodes are connected
*Structure menu
> 2D Member
> 2D Member components
> Internal edge: entire beam connected



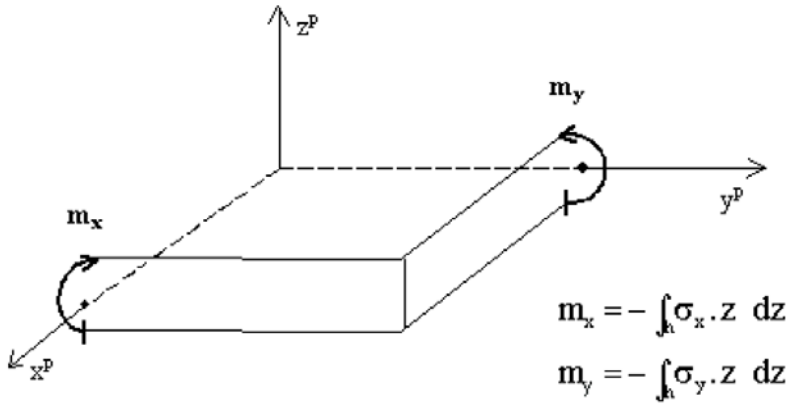
*Connect members/ nodes: only nodes are connected
*Structure menu
> 2D Member
> 2D Member components
> Internal edge: entire beam connected

Annex 2: Conventions for the results on 2D members

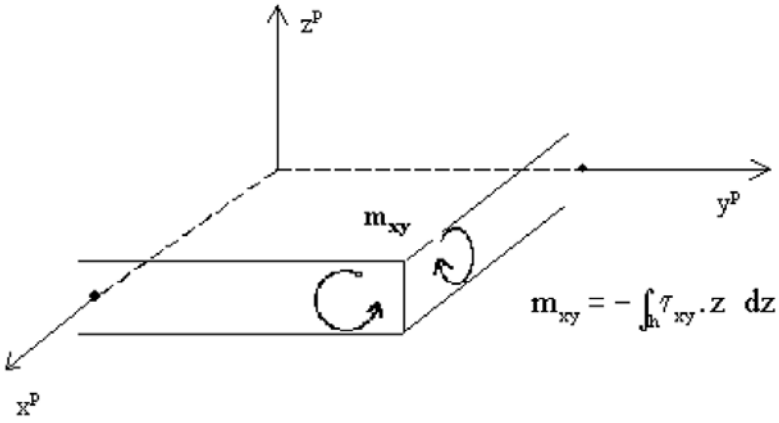
1 Basic magnitudes = Characteristic values

Bending (plates, shells)

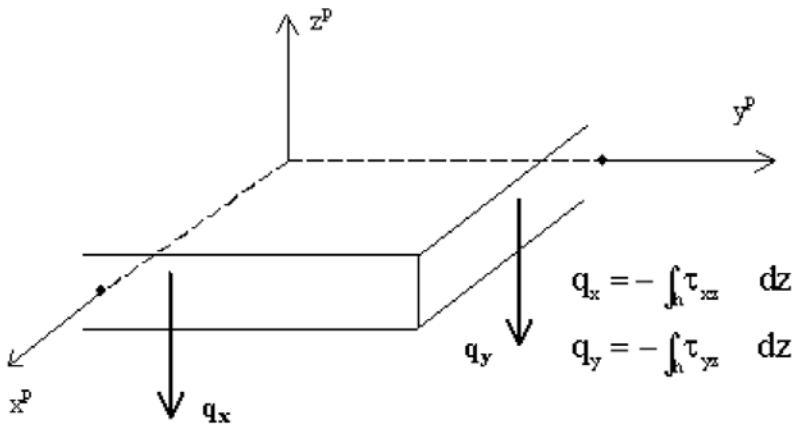
- Bending moments mx, my



- Torsion moment mxy

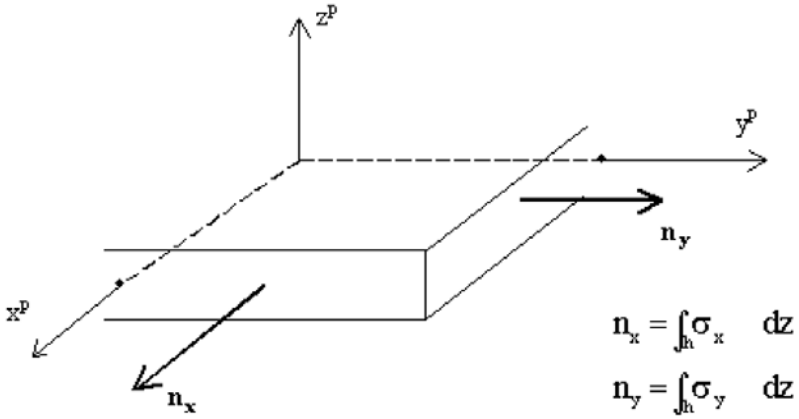


- Shear forces qx, qy (=vx, vy)

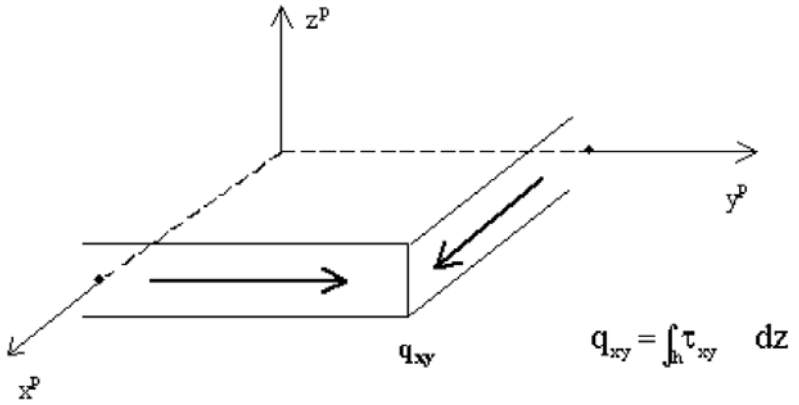


Membrane effects (walls, shells)

- Membrane forces nx, ny



- Shear forces qxy (=nxy)



2 Principal magnitudes

The principal magnitudes give the results according to the axes of the directions of the largest stresses (principal directions). These directions are defined with the help of the circle of Mohr.

`

3 Dimensional magnitudes = Design values

To derive the dimensional magnitudes from the basic magnitudes, formulas from the Eurocode are used.

See also Help > Contents > Reference guide for these formulas.

Annex 3: Results in Mesh elements and Mesh nodes > 4 Locations

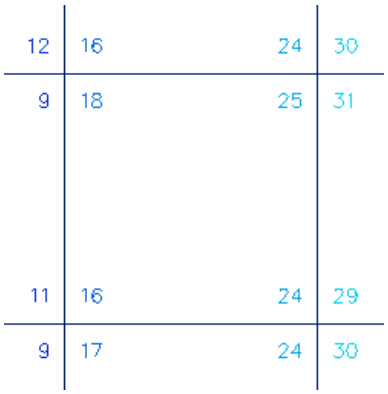
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always less accurate.

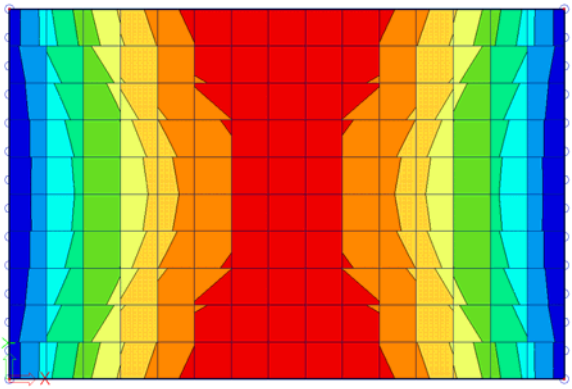
The Finite Elements Mesh in SCIA Engineer exists of linear 3- and/or 4-angular elements. Per mesh element 3 or 4 results are calculated, one in each node. When asking the results on 2D members, the option 'Location' in the Properties window gives the possibility to display these results in 4 ways.

1 In nodes, no average

All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.

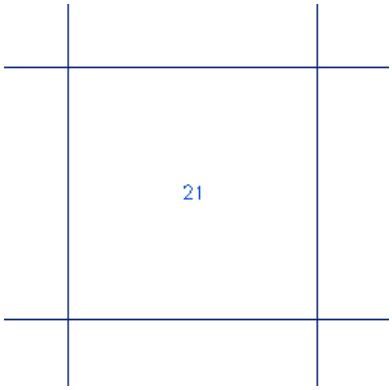
This display of results therefore gives a good idea of the discretisation error in the calculation model.

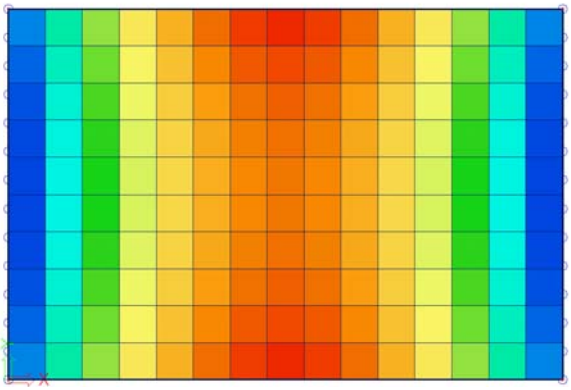




2 In centres

Per finite element, the mean value of the results in the nodes of that element is calculated. Since there is only 1 result per element, the display of isobands becomes a mosaic. The course over a section is a curve with a constant step per mesh element.





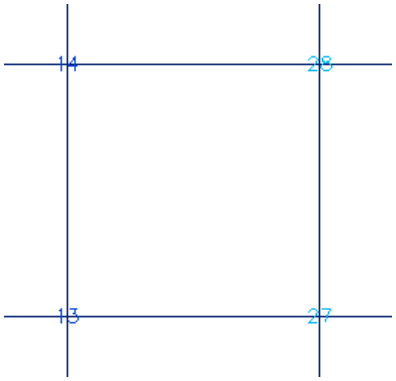
3 In nodes, average

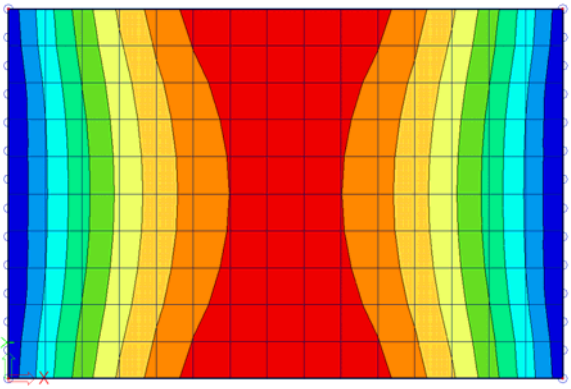
The values of the results of adjacent finite elements are averaged in the common node. Because of this, the graphical display is a smooth course of isobands.

In certain cases, it is not permissible to average the values of the results in the common node:

- At the transition between 2D members (plates, walls, shells) with different local axes.

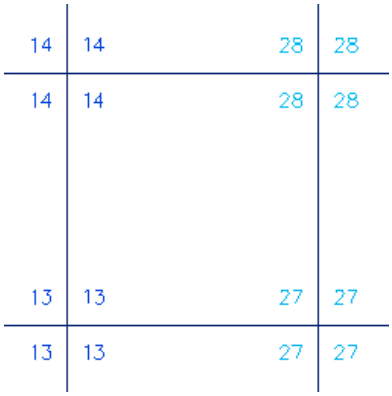
- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.

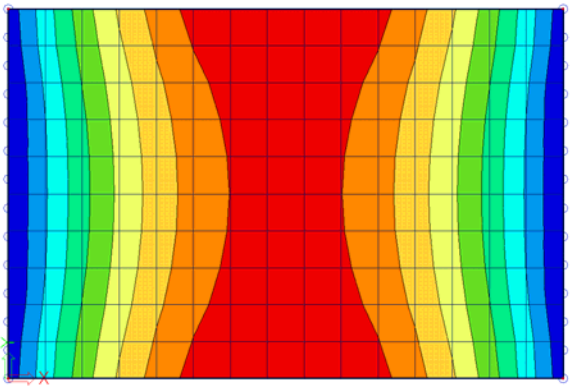




4 In nodes, average on macro

The values of the results are averaged per node *only* over mesh elements which belong to the same 2D member and which have the same directions of their local axes. This resolves the problems mentioned at the option 'In nodes, average'.





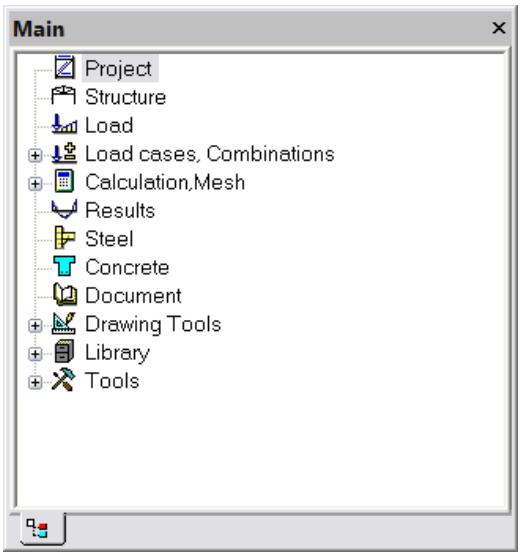
Accuracy of the results

If the results according to the 4 locations differ a lot, then the results are inaccurate and the mesh has to be refined. A basic rule for a good size of the mesh elements, is to take 1 to 2 times the thickness of the plate.

Annex 5: Overview of the Icons in Windows & Toolbars

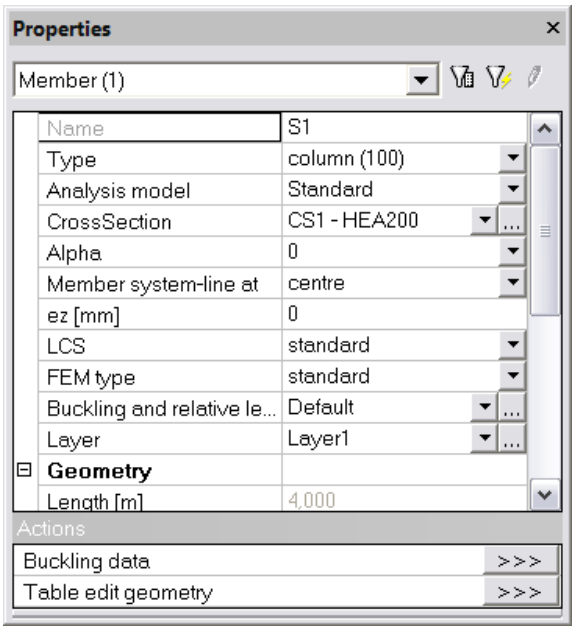
1 Main Window

In the Main window one can find the links to the most used menus and actions. Some of those links are only activated when they can be effectively used in the project: e.g. the link to the Results menu is only shown after a calculation has been performed, and the Steel and/or Concrete menus are visible depending on the materials applied in the project.



2 Properties Window

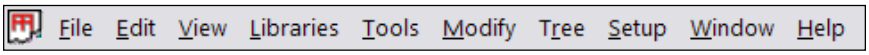
The Properties window gives information about selected objects and/or actions. Moreover it is possible to adjust the properties of each object directly via this menu. When several kinds of objects or actions are selected at the same time, it is possible to switch between their properties by means of the little arrow behind the object name. The number between brackets behind the object name represents the number of objects of this kind that is selected at this moment. If the number is larger than 1, only the corresponding properties are shown.



 Select elements by more properties

 Select elements by property

3 Menu Bar



These 'written' menus group all actions by subject. A large number of these actions is also available in the Main menu and/or as icon in the toolbars.

4 Standard Toolbars

The 'Activity' toolbar provides options for the visibility / invisibility of specific parts of the structure, which increase the ease of working and the readability of the project.

 Activity toggle

 Activity by layers

 Activity by selection (Selected members On)

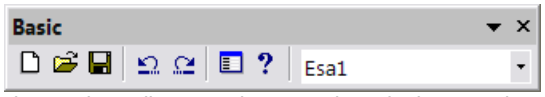
 Activity by selection (Selected members Off)

 Activity by working plane

 Activity by clipping box

 Invert current activity

 Draw inactive members



The 'Basic' toolbar contains a number of primary actions with regard to the current project and allows to modify the basic settings of the program (Setup Options).

 New (Ctrl+N)

 Open (Ctrl+O)

 Save (Ctrl+S)

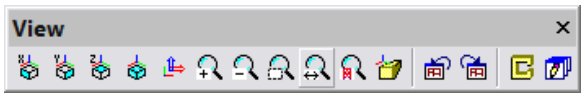
 Undo

 Redo

 Setup

 About SCIA Engineer

Name of the opened *.esa file



The 'View' toolbar allows for the execution of a whole lot of simple view manipulations.

 View in direction X

 View in direction Y

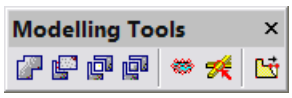
 View in direction Z

 View in direction AXO

View perpendicular to working plane
Zoom in
Zoom out
Zoom by cut out
Zoom all
Zoom selection
Perspective view
Undo view change
Redo view change
Generate structural model
Regenerate view

In the ‘Geometry manipulations’ toolbar one can find manipulations with basic entities (nodes, 1D members, 2D members), as well as with additional data.

Move
Copy
Multicopy
Rotate
Scale
Stretch
Mirror
Trim
Extend
Enlarge by defined length
Break in defined points
Join
Break in intersections
Reverse orientation
Polyline edit
Curves edit
Calculate member end-cuts
Connect members/nodes
Disconnect linked nodes
Copy additional data
Move additional data



The ‘Modelling tools’ toolbar provides for manipulations with general solids.

 Union of solids

 Subtraction of solids

 Intersection of solids

 Division of solids

 Generate vertexes

 Clash check of solids

 Move vertexes/points

The ‘Project’ toolbar collects various actions, from the definition of databases (layers, materials, cross-sections) for the project, to several options for the output.

 Units

 Layers

 Materials

 Cross-sections

 Check structure data

 Mesh generation

 Calculation

 Hidden calculation

 Print data

 Print picture

 Picture gallery

 Paperspace gallery

The ‘Selection of object’ toolbar contains different possibilities to select a specific part of the structure. A selection can also be saved and loaded again later on.

 Selection by mouse

 Selection by cut out

 Selection by intersecting line

 Selection by polygonal cut out

 Select all

 Selection by working plane

Previous selection

Cancel selection

Selection mode toggle (Select or Deselect)

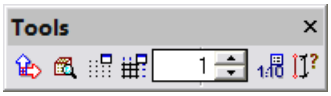
Single selection mode toggle (All found or First found)

Save selection

Load selection

Filter for selection on/off

Filter by service tree on/off



In the 'Tools' toolbar a number of clever means can be found for the input and graphical display of a structure.

Setup UCS

Clipping box

Dot grid setting

Line grid

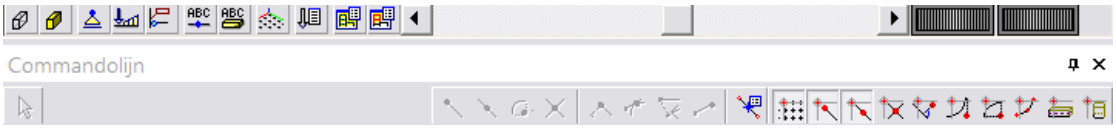
Scale factor

Scale manager

Coordinates info

5 Command line Toolbars

On the Command line itself, a number of commands for the operation of the program can be inputted. Also, during running actions, instructions on the next steps are shown.
Apart from that, quite a number of toolbars can be found here; some of them are only available during a certain action or in a specific menu.



Show/Hide surfaces

Render geometry

Show/Hide supports

Show/Hide loads

Show/Hide other model data

Show/Hide labels of nodes

Show/Hide labels of members

Show/Hide dot grid

Set load case for display

Fast adjustment of view parameters on whole model

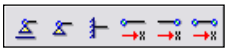
 Fast adjustment of view parameters on selection

 Cursor snap settings

 Fast adjustment of cursor snap settings

 Change insertion point, available during input of geometry

 Definition of new form, available during input of geometry

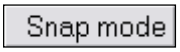
 Fast input of supports & hinges, available in Structure menu

 Fast input of loads, available in Load menu

 Fast display of results, available in Results menu

Adjust Units of length

Change Active working plane



Adjust Cursor snap settings

Adjust UCS (=User Co-ordinates System)

 Change Active code