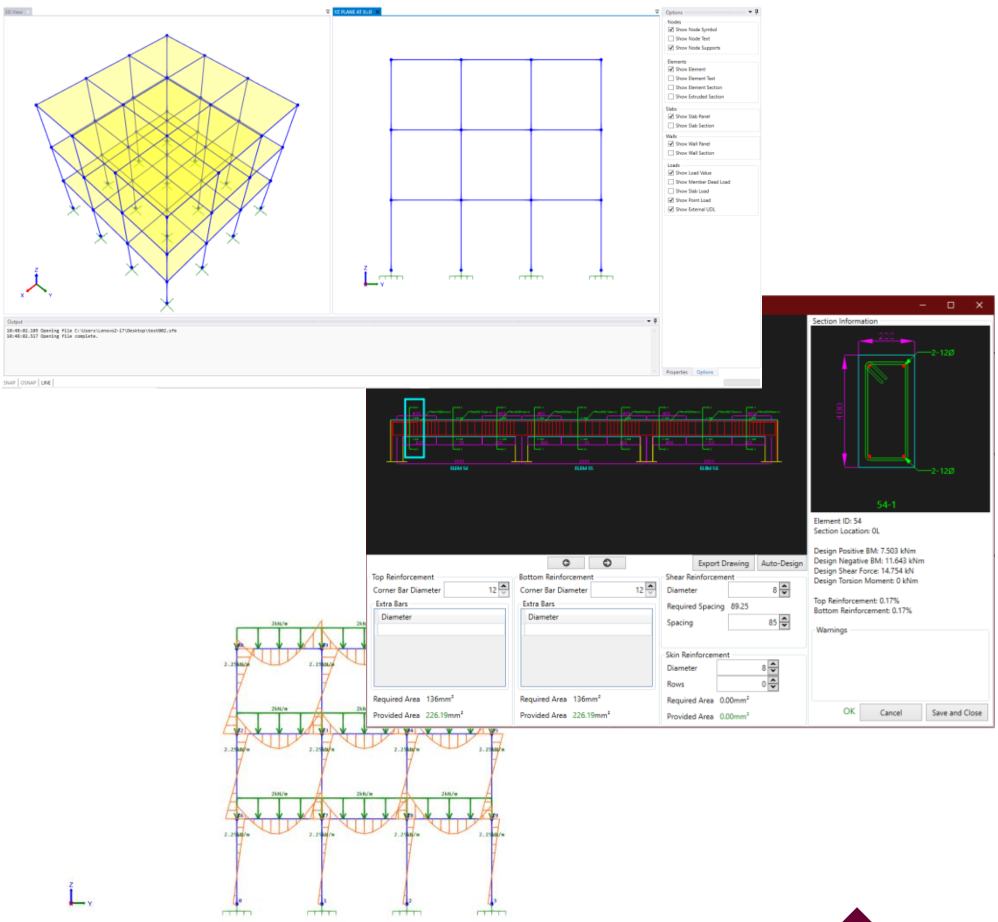
SW FEAD

Building Analysis and Design Software

Operating Manual

(Version 1.0.0)



Softwel (P) Ltd.

254 Shree Ekata Marga, New Baneshwor,
Kathmandu-31, Kathmandu, Nepal
Phone: +977-1-4566627, 4566629
Fax: +977-1-4566629
Website: https://softwel.com.np
Email: support@softwel.com.np

About SW FEAD

SW FEAD (Finite Element Analysis and Design) is a free structural analysis and design software for the analysis and design of buildings. The software can be used for three-dimensional static and dynamic analysis of buildings for earthquake loading based on the NBC 105:2020 and IS 1893:2016 standards. The software is also capable of design of reinforced concrete beams and columns, as well as drawing generation.

Softwel (P) Ltd.
254 Shree Ekata Marga, New Baneshwor,
Kathmandu-31, Kathmandu, Nepal
Phone: +977-1-4566627, 4566629
Fax: +977-1-4566629
Website: https://softwel.com.np
Email: support@softwel.com.np

User Manual Version 1
For Application Version 1.0.3

Features

Structural Modeling

- Structures can be modeled using frame, slab, masonry infill walls and concrete wall elements
- Frame elements can be modelled based on Euler-Bernoulli or Timoshenko beam theories
- Masonry infill walls modeled according to IS 1893:2016 provisions
- Elements can be drawn graphically in 3D or section views
- Building frame structures may be generated based on user defined size and number of bays and storeys
- Different support conditions can be assigned to nodes
- Reinforced concrete and steel sections are supported. Automatically calculates all required geometrical properties based on section dimensions.
- Support for flexural and shear stiffness modifiers

Load Combinations

- Pre-defined load cases for dead load, live load, storage live load and earthquake load (X and Y direction)
- Load combinations can be automatically generated based on NBC 105:2020 and IS 1893:2016 standards.
- Concentrated and distributed loads may be assigned to frame elements, and uniform area loading may be applied to slabs.

Earthquake Loading and Analysis

- Automatically computes earthquake loads based on the NBC 105:2020 and IS 1893:2016 standards.
- Seismic analysis can be performed using the Seismic Coefficient Method or the 3D Modal Response Spectrum method
- Display axial force, shear force and bending moment diagrams, as well as deformations.
- Mode shape display and animation

Reinforced Concrete Design and Drawing

- Automatically performs the limit state design of reinforced concrete beams and columns for axial, shear, bending and torsion.
- Performs ductile design based on the NBC 105:2020 or IS 13920:2016 standards. Design based on the IS 456:2000 standard is also supported.
- Computes and displays the required rebar percentage in all reinforced concrete members
- An interactive design mode to assign reinforcement bars to members
- Automatic detailing feature to auto-assign rebar number and spacing based on user-specified bar sizes

Drawing Generation

- Generates L-Section and X-Section drawings of beams and columns
- Drawings are exported to the AutoCad DXF format and can be edited or printed from any compatible CAD application

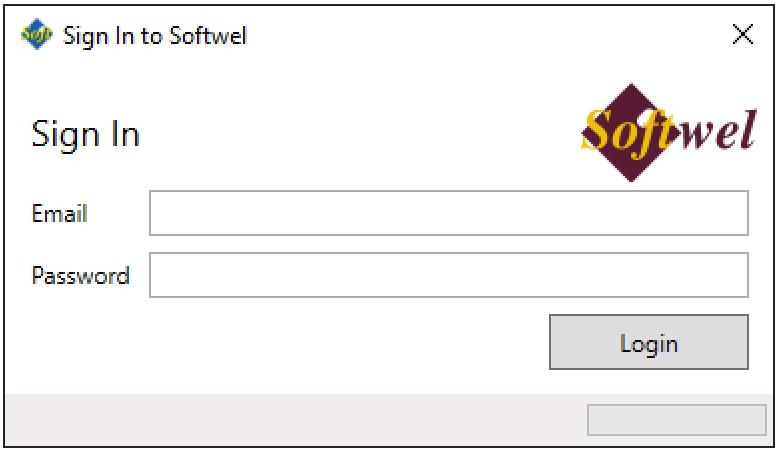
System Requirements

- Operating System: 64-Bit Windows 7 or later (Windows 10 Recommended)
- Processor: Dual Core 2.5 GHz (Quad Core 3 GHz+ recommended)
- Memory: 4 GB (8GB recommended)
- Disk space: 1.0 GB.
- Graphics: DirectX 10 Capable

Installation

To install SW FEAD, follow the instructions below.

- Register an account with Softwel. You can register an account from the Softwel official page.
- Once you register, an e-mail will be sent to you containing the activation link. Click on the link to sign in and activate your Softwel account.
- Go to the Downloads page of Softwel.
- Download the SW FEAD Setup File and run it.
- Once installed, you can start SW FEAD from your desktop or the start menu. You will be required to sign in on the first run. Once signed in, you can run the software offline.

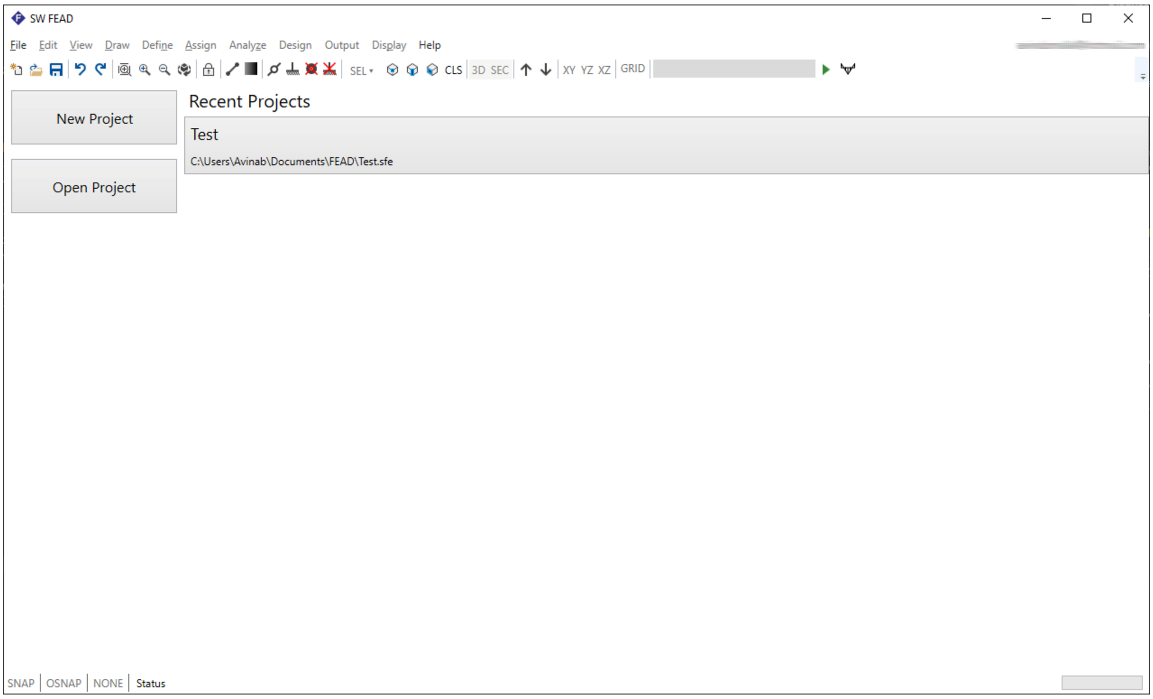


Contents

1 User Interface

1.1 The Startup Screen

When SW FEAD is first opened, the following startup screen is displayed.

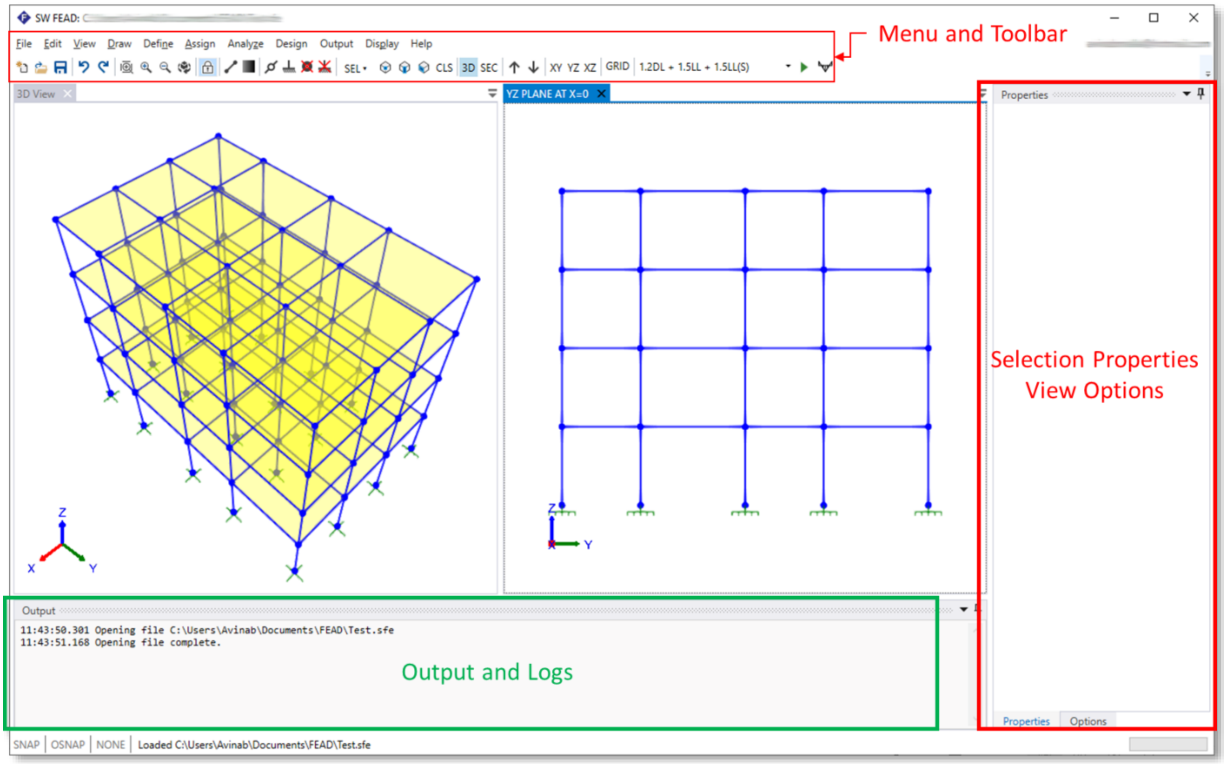


The startup screen shows a list of recent projects, and buttons for creating a new project or opening an existing project. Once a project is created or opened, the structure model view is shown and the disabled menus and tools are available.

1.2 Main Window

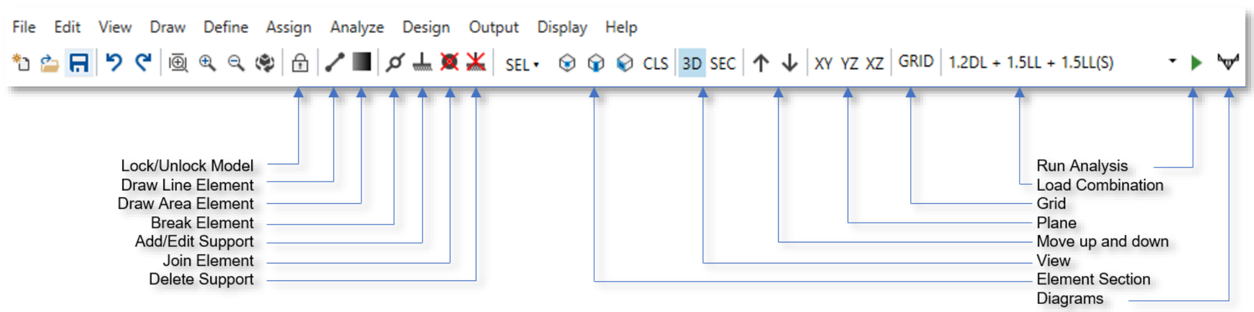
The main window is shown after a project is loaded. The window is divided into the following components.

- Menus and Toolbars
- View Windows
- Properties and Options Panel
- Output Panel
- Status Bar
- User Login Information



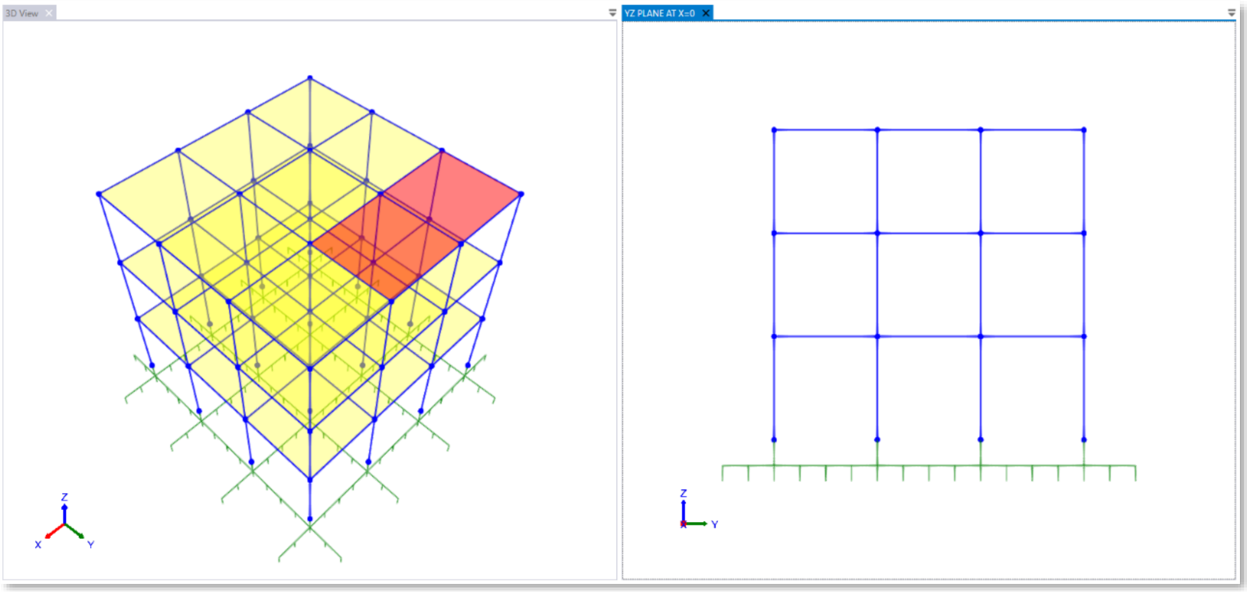
1.2.1 Main Toolbar

The main toolbar contains various tool buttons which have their respective functions as show in figure below.



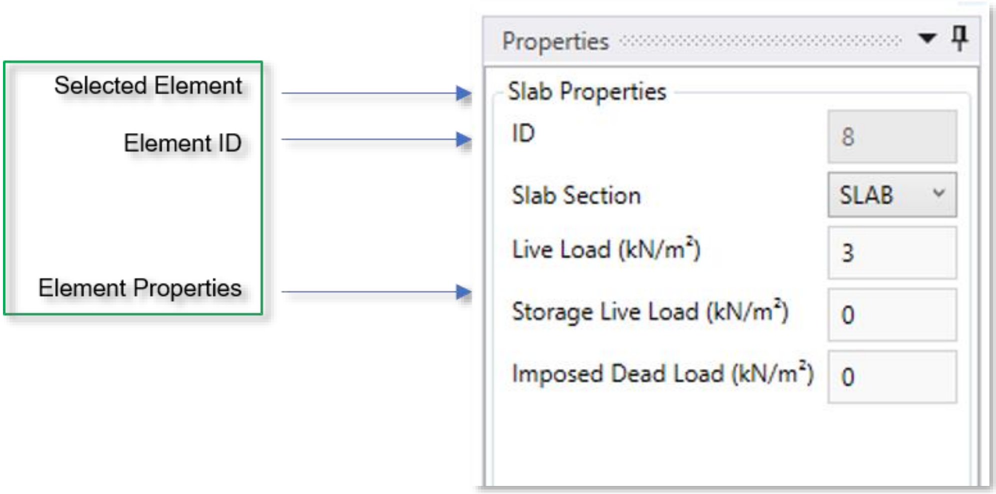
1.2.2 View Windows

These windows are used to view the model in 3D view and different section planes (XY, YZ, ZX).



1.2.3 Properties

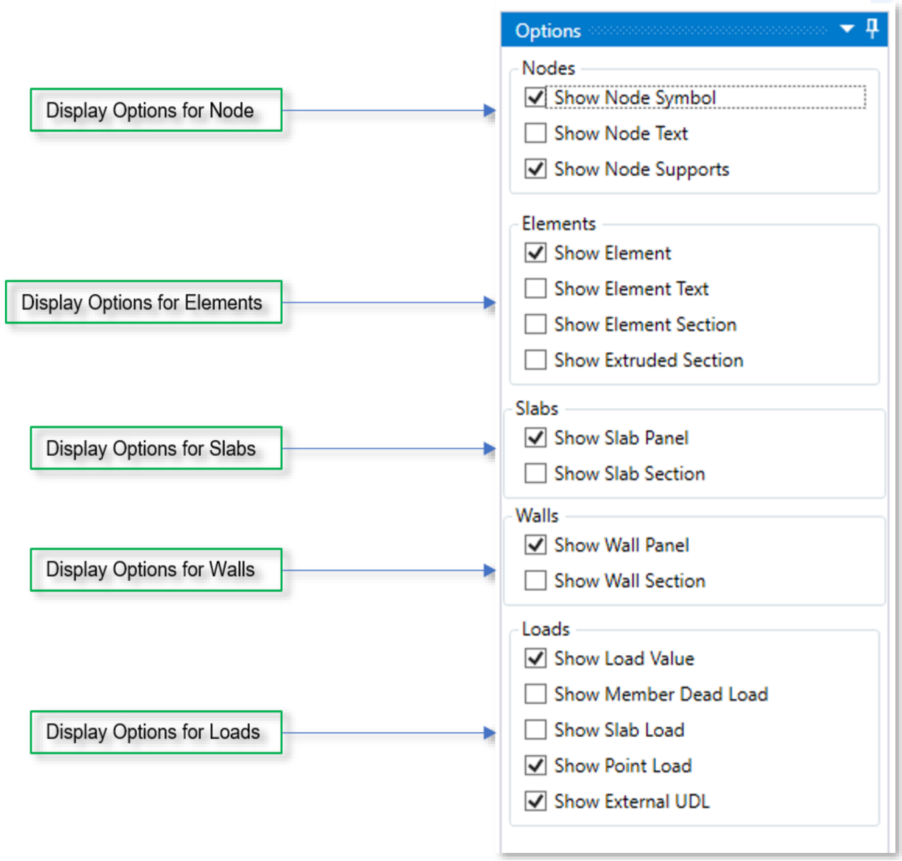
It is used to display the properties of the selected items (Node, Line, Area). The properties of selected items can be modified here. When multiple items of the same type are selected, this panel shows the common properties of the selected items, and any change is applied to all the selected items.



1.2.4 Options

The option panel controls what is displayed on the viewports. It allows turning on and off the visibility of the model's components and other information.

The visibility of some components depends on the visibility of other components. For example, node texts may not be shown if node symbols are invisible. Similarly, element text and section information cannot be shown if elements are not shown.

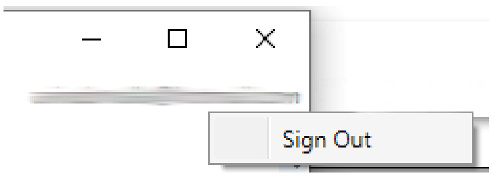


1.2.5 Status Bar

The status bar shows the current snapping and selection mode, as well as the number of elements selected. When running an analysis, the status bar shows the progress of the analysis. When updating the application, the update progress is also shown here.

1.2.6 User Login Information

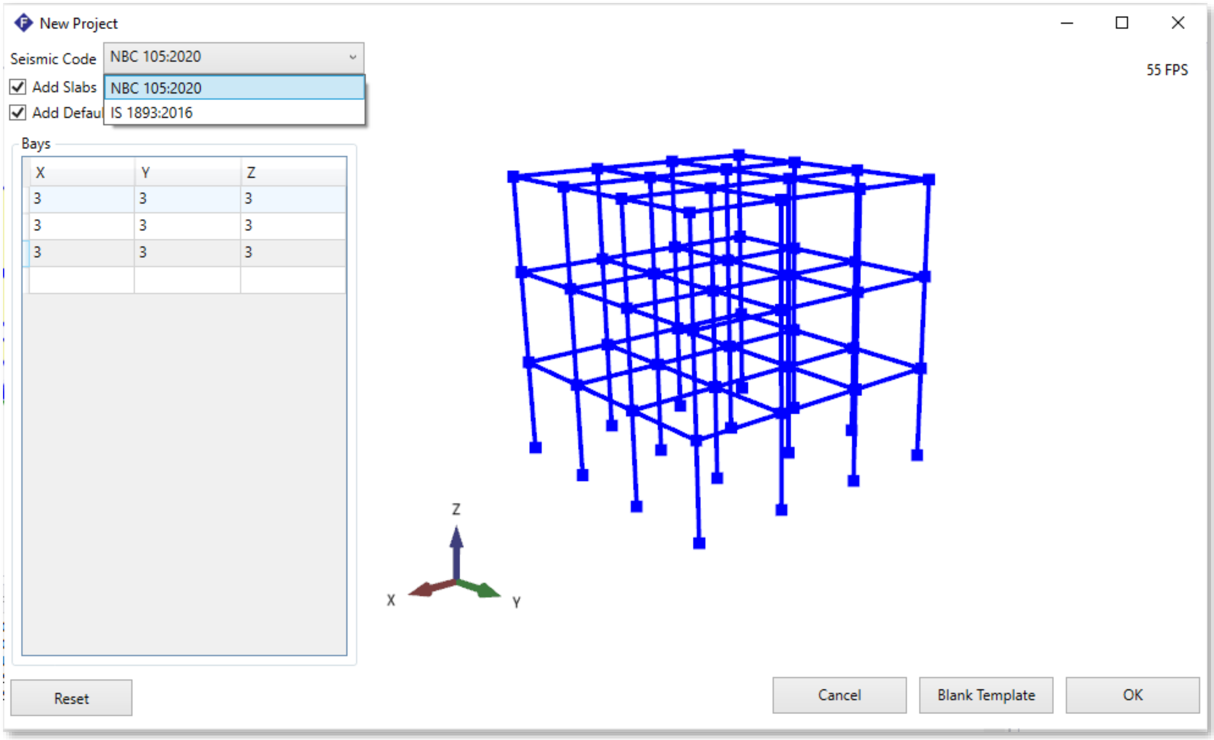
The current user login information is shown on the top right corner of the window. To sign out and sign in as a new user, you may click the email address and on the dropdown that follows, click "Sign Out".



1.3 Creating a New Project

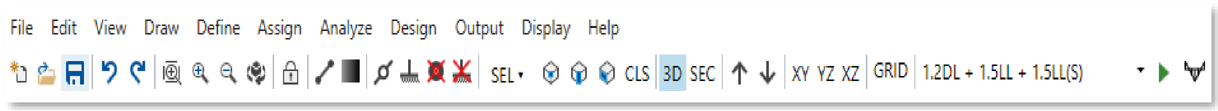
To create a new project, follow the steps below.

1. Open the New Project Window. This can be done from the startup screen, the toolbar or the **File** menu.
2. Select the seismic analysis building code to be used for the model.
3. Select whether you wish to automatically add the floor slabs and the default load combination.
4. Enter the size of bays, in meters, in the respective columns in the grid. For example, if the building has two bays in the X direction having size 3m and 4m, enter 3 and 4 in two rows in the X column. Alternatively, you may also click **Blank Template** to create a model without any elements.
5. Press OK and select the path to save the new project.



2 Main Menu

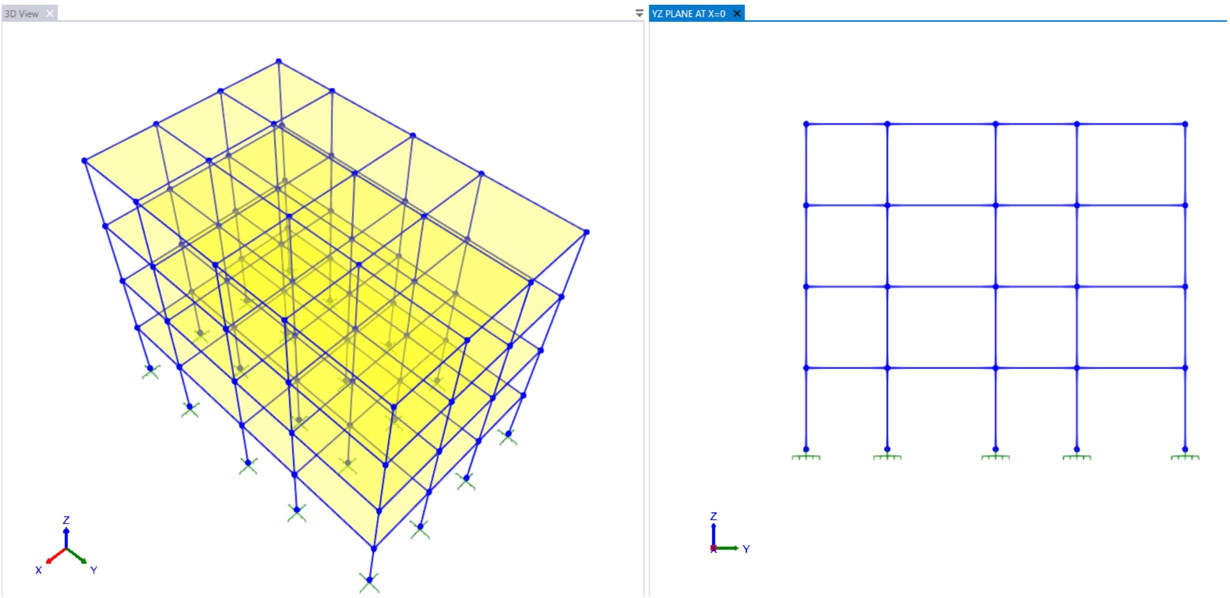
The entire system is grouped into menus based on their similarity of functions. The following table provides the summary of the menus and the subsequent section provides details of each of the menus and the sub-menus.



Menu	Description
File	Allows users to create new project, open project, save and save as project, close project and exit.
Edit	Edit data related to nodes and elements; delete, join and replicate elements; allows selection of all elements.
View	Handling of zoom options in selected View Window; Selection of 3D View window or Sectional View Window; Element Properties Viewer
Draw	Draw Node elements by inserting Coordinates or breaking existing element; Draw Line elements (beam or column) by connecting two nodes; Draw supports and slab panels.
Define	This section allows to define materials, sections, loading and diaphragm constraints.
Assign	Helps to assign selected element either as column or beam or slab element.
Analyze	Checks the error in the model and performs analysis.
Design	Design of reinforced concrete beams and columns.
Output	Gives output of different types of loads and members in table and drawing form.
Display	Options for displaying the results of analysis such as reactions, shear forces, bending moments, deformed shapes, mode shapesetc.
Help	Provides info about the system and available updates.

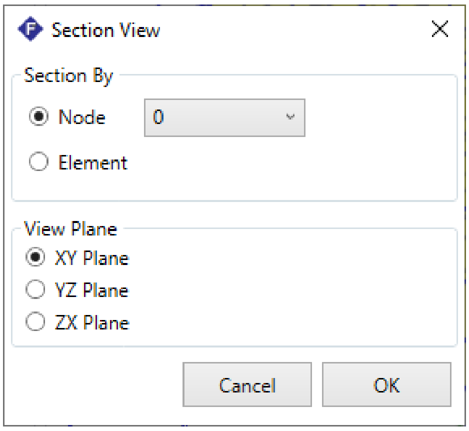
3 The SW FEAD Viewport

The viewport shows the 3D view or a sectional view of the currently open model and allows the user to interact with it. It also shows the analysis and design results.



3.1 Switching 3D/2D views

The active viewport may be switched to the 2D section view by pressing one of **XY**, **YZ** or **XZ** buttons on the toolbar. To load a section with a specific node or element, press the **SEC** button. When in section view, click the Up and Down arrow buttons on the toolbar to move the view plane forward and back.



3.2 Pan, Rotate and Zoom

To pan the view, hold the middle mouse button (Scroll wheel) and then move the mouse.

To rotate the 3D view, you may use one of the following three options.

1. Double left-click at the anchor point of rotation, then move the mouse.
2. Hold Shift and the left mouse button, then move the mouse.
3. Click the **Rotate** tool on the toolbar to activate rotate mode, then move the mouse holding the left mouse button.

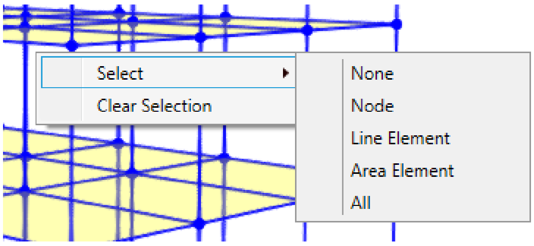
To zoom in/out, you may use the mouse wheel or the toolbar buttons. Double-clicking the middle mouse button will zoom the view to its extents.

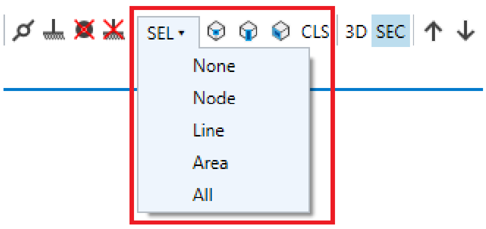
3.3 Selection

SW FEAD has 5 selection modes available.

1. None: Nothing can be selected
2. Node: Node points can be selected
3. Line: Line Elements (such as beams and columns) can be selected
4. Area: Area elements (slabs, infill and RC walls) can be selected
5. All: Everything can be selected

Selection mode can be changed from the right click context menu of the viewport, or the toolbar.





When in a selection mode, the selectable items may be selected or unselected by left clicking on them. To select multiple items with a box selection, hold the **Ctrl key** and then drag while holding the left mouse button.

You may clear selection by pressing the **Esc** key. Pressing **Esc** when nothing is selected changes the selection mode to **None**.

3.4 Adding/Removing a Viewport

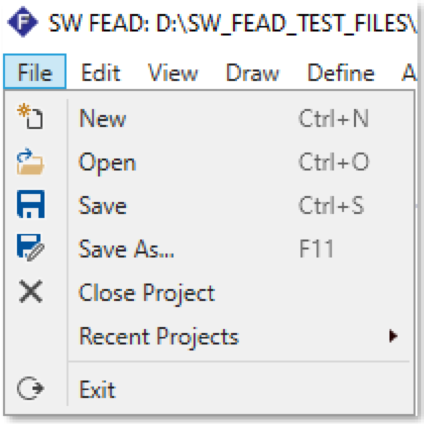
When SW FEAD is started, the current model is shown in two viewports.

A viewport may be added by using the **View->Add Viewport** menu. To remove a viewport, press the **X** icon next to the viewport title.

Viewports may be dragged around and undocked from the main window by dragging the title bar.

4 The File Menu

File menu has the following options available.



4.1 New

Opens the window for creating a new project.

4.2 Open

Open an existing project.

4.3 Save

Save the currently open project.

4.4 Save As

Save the currently open project in a new file and continue in it.

4.5 Close

Closes the current project.

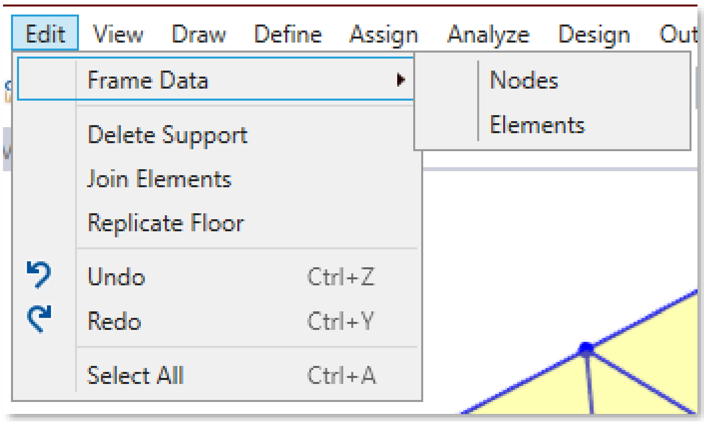
4.6 Recent Projects

Display the list of recently opened projects for quick opening of the project file.

4.7 Exit

Exits the application.

5 Edit

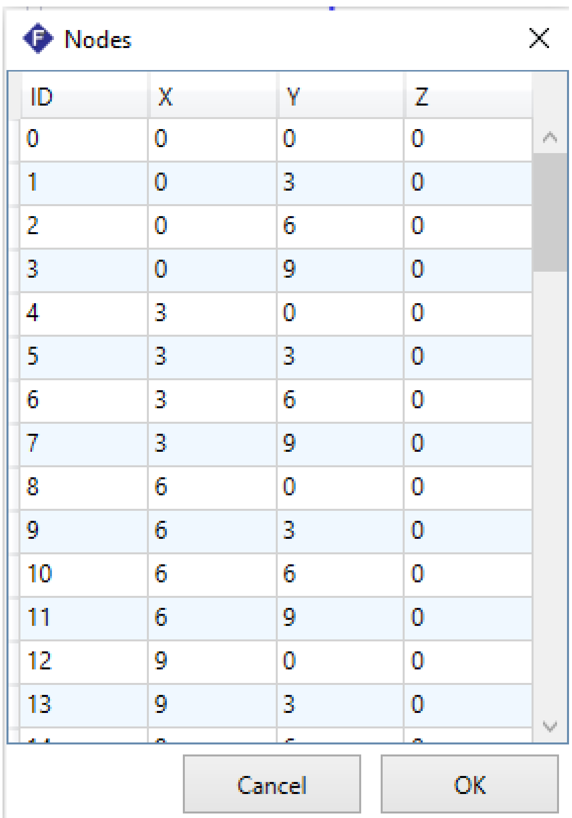


5.1 Frame Data

The Edit Frame data menu allows editing the nodes and line elements in the model.

5.1.1 Nodes

Edit nodes present in the model. Here the coordinates of every node present in the model are displayed in increasing order of their id. The node coordinates can be edited here.

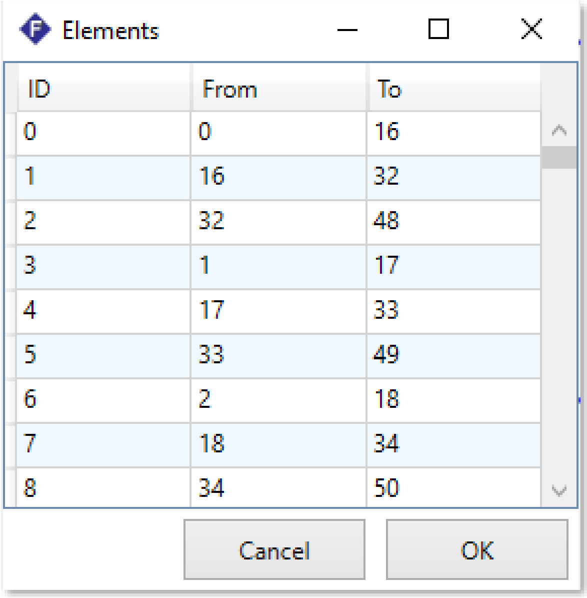

Nodes

ID	X	Y	Z
0	0	0	0
1	0	3	0
2	0	6	0
3	0	9	0
4	3	0	0
5	3	3	0
6	3	6	0
7	3	9	0
8	6	0	0
9	6	3	0
10	6	6	0
11	6	9	0
12	9	0	0
13	9	3	0

Cancel OK

5.1.2 Line Elements

This menu displays a table that allows you to view and edit the connectivity of the line elements in the model.



5.2 Delete Support

Clicking this menu will delete supports from the nodes that have been selected in the model.

5.3 Join Elements

This sub-menu is used to join elements of the model by deleting nodes. To join two elements connected by a node, press this menu or the join icon on the toolbar and then click the node common between the two elements to be joined.

5.4 Replicate Floor

This command copies the property from one floor element to another floor.

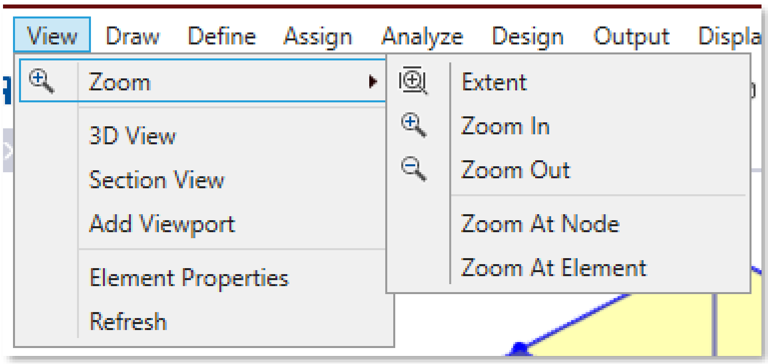
5.5 Undo and Redo

The undo command reverses the last command while redo command repeats the last un-done command.

5.6 Select All

This selects all the items that can be selected in the current selection mode.

6 View



6.1 Zoom

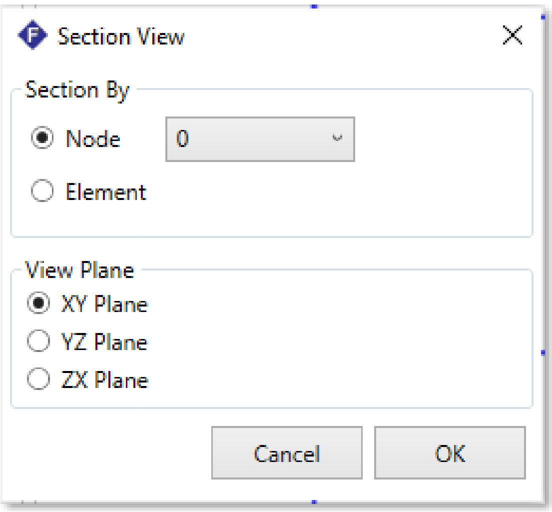
There are various zoom related commands under this menu which allows you to perform a particular zoom action in the selected view window such as zoom to extent, zoom in, zoom out, zoom at node, zoom at element.

6.2 3D View

Clicking this menu will load the 3D view in the current viewport.

6.3 Section View

This window lets the user display either XY, YZ or ZX plane in the currently selected viewport.

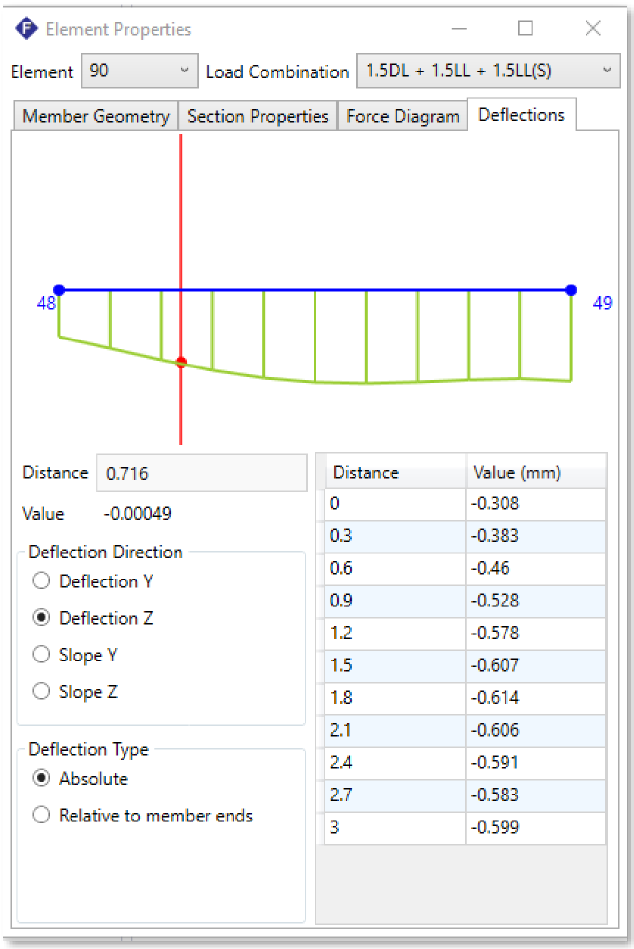


6.4 Add Viewport

This command adds a new viewport in the main window.

6.5 Element Properties

The following shows element properties dialog box:

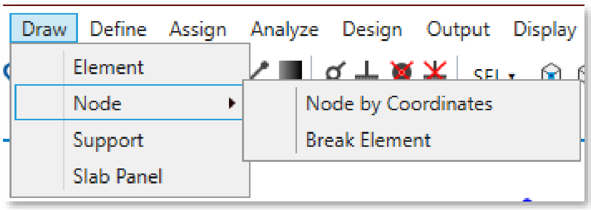


Various properties of the selected element such as geometrical, sectional is displayed as well as force diagram and deflections as per loading are displayed.

6.6 Refresh

This menu will refresh and redraw all the viewports.

7 Draw



7.1 Node

This sub menu has two options for drawing nodes:

7.1.1 By Coordinates

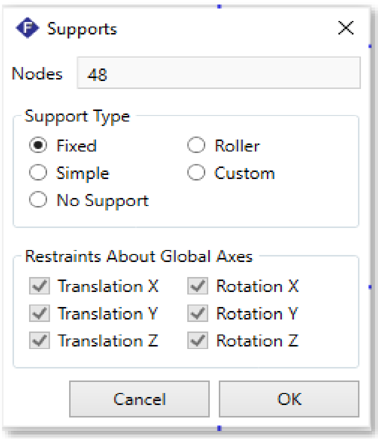
By manual entry of coordinates of the node in the window shown below, one can draw a node.

7.1.2 Break Element

This command inserts a new node between two existing nodes of a line element.

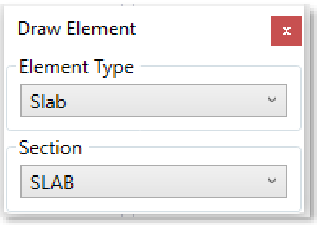
7.2 Support

This sub-menu is for drawing support. Here the node which was selected to insert support is displayed. The support type and restraints must be selected prior to inserting support at the selected node.



7.3 Line Element

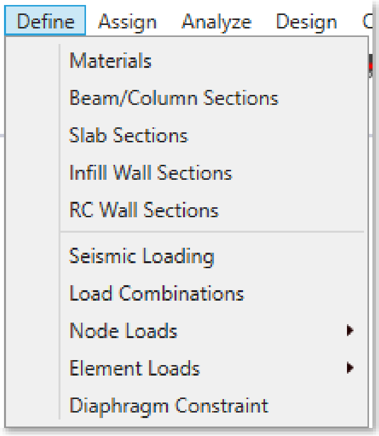
This sub-menu is a tool to draw a line element by selecting any two nodes in the current view window.



7.4 Area Element

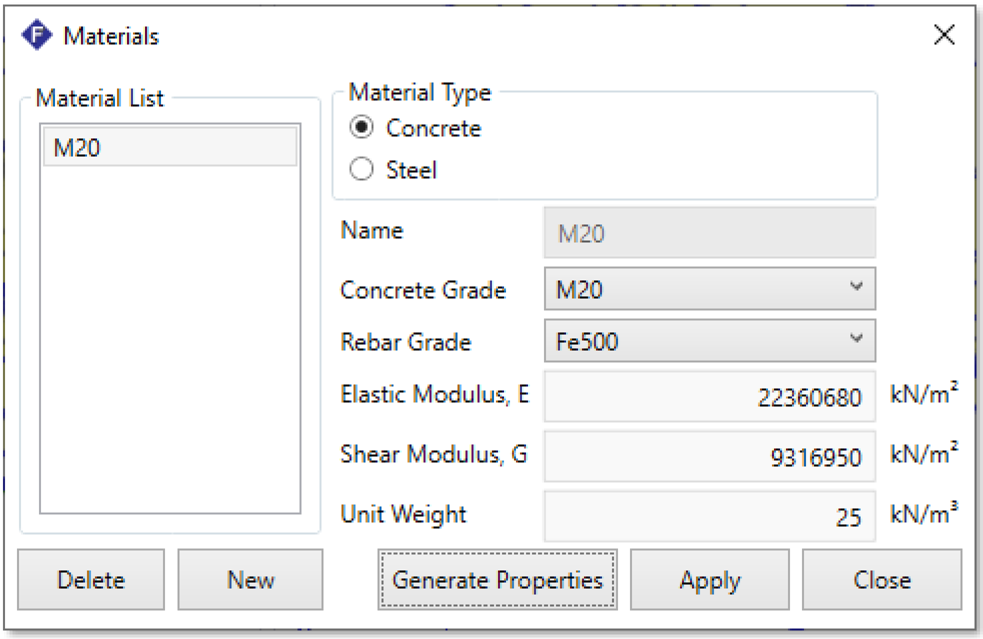
This sub-menu is for the drawing of area elements (slabs, infill walls and RC walls). After selecting the section as defined in the **Define** menu, nodes in the current window view must be clicked to draw the element.

8 Define



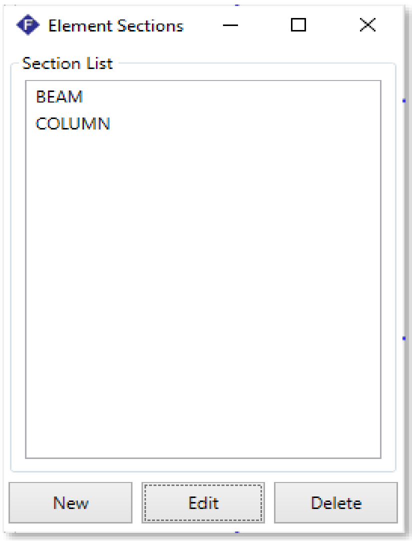
8.1 Material

This tool is used to define a material. The material may be either concrete or steel, and the engineering properties such as elastic properties, shear modulus and unit weight may be assigned or generated automatically based on the grade of the material. The materials defined here has to be assigned to a beam/column section, slab section or RC wall section to be used in the model.

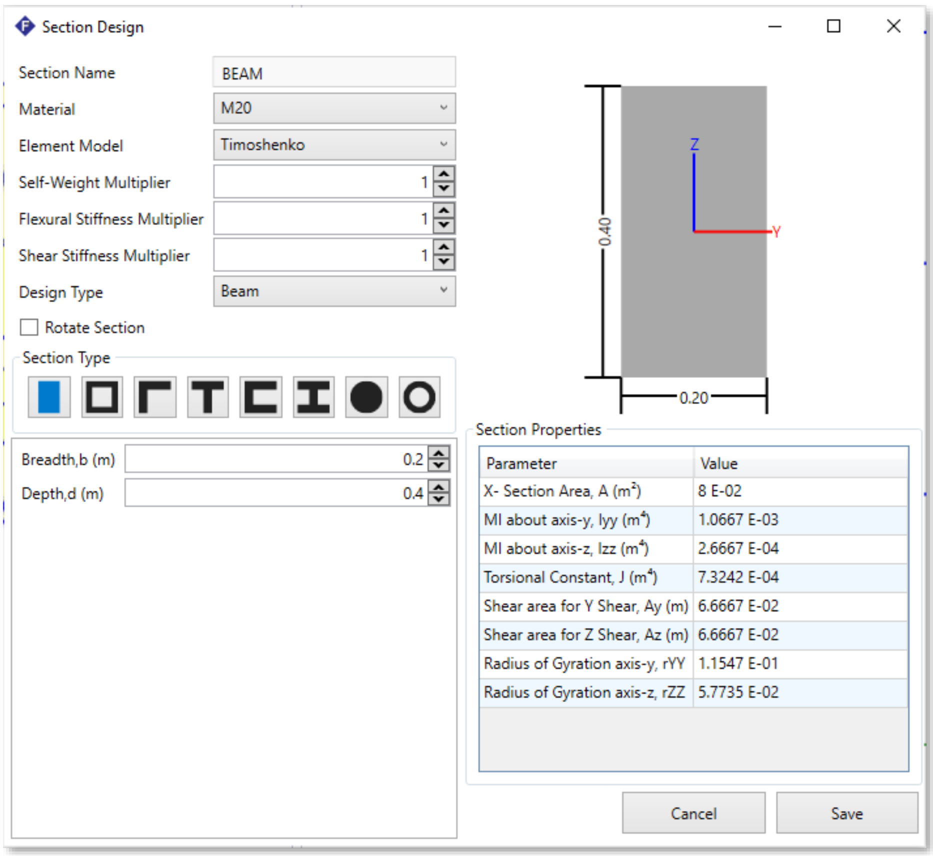


8.2 Beam/Column Sections

This menu opens a list of currently defined section and allows you to add new sections and edit/delete the existing sections.

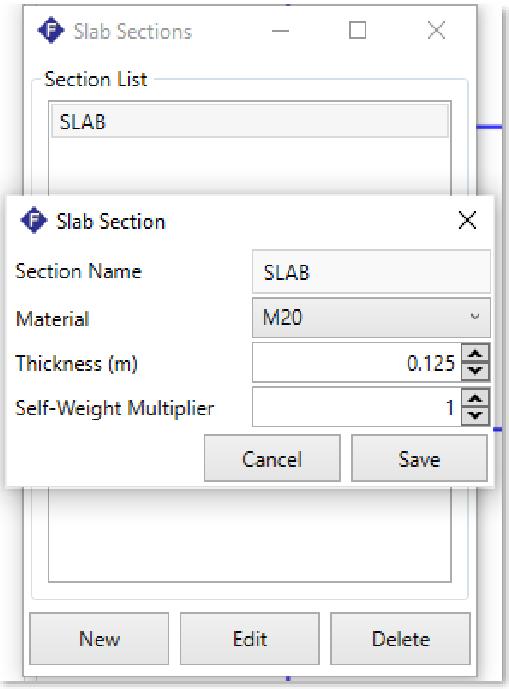


Clicking new or edit will open a new window that allows defining different types of beam or column section and assigning properties such as material, geometry, self-weight factor, flexural stiffness factor and shear stiffness factor. The design type can also be set here and it defines whether the elements with this section are designed as a beam or as a column.



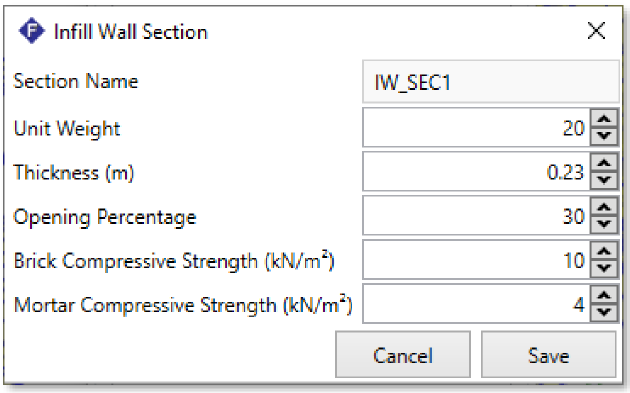
8.3 Slab Sections

This tool allows creating, editing and deleting the slab sections defined in the current project. Slab sections having different material, thickness and self-weight multipliers may be added.



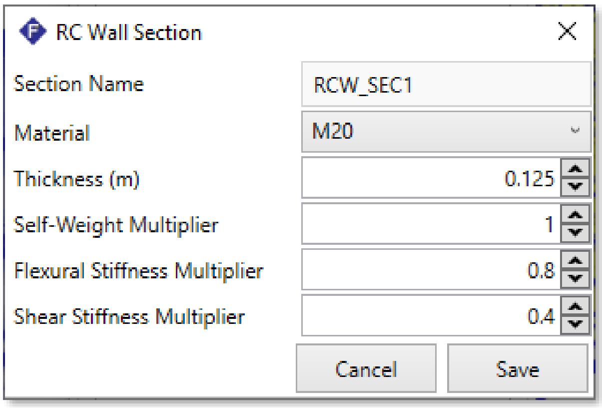
8.4 Infill Wall Sections

This tool allows creating, editing and deleting the infill wall sections defined in the current project. Infill wall sections having different material unit weight, thickness, opening percentage, and brick/mortar compressive strengths may be added.



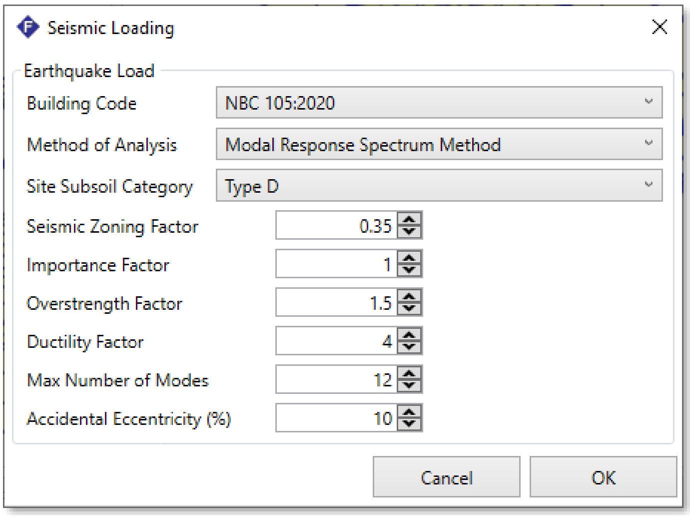
8.5 RC Wall Sections

This tool allows creating, editing and deleting the reinforced concrete wall sections defined in the current project. RC wall sections having different material, thickness and self-weight multipliers and stiffness multipliers may be added.



8.6 Seismic Loading

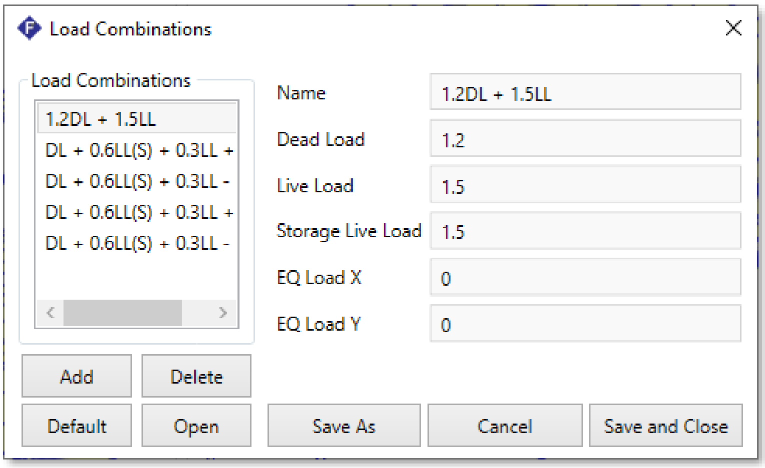
This window lets the user select the method of earthquake analysis, seismic design code its various parameters, accidental eccentricity and the number of modes (for response spectrum analysis).



8.7 Load Combinations

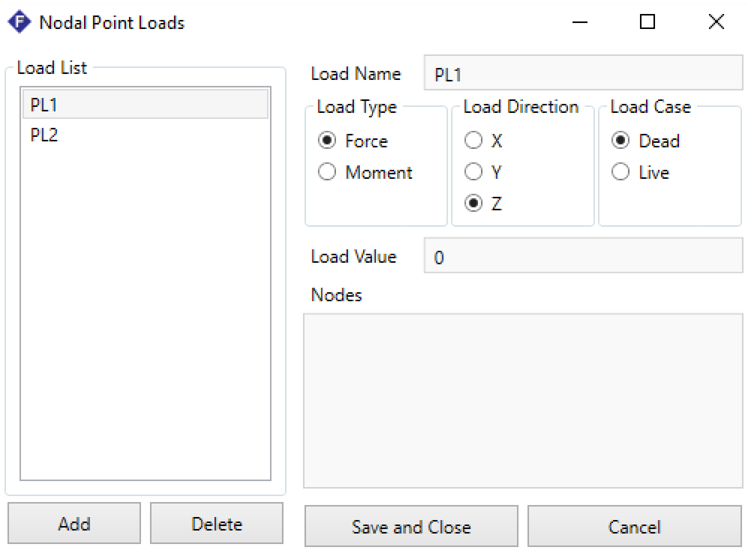
This window allows the user to select the load combinations for which the analysis is to be performed. Default load combinations for the currently selected analysis code can be added using the **Default** button.

Each load combination may have 5 different load factors: dead load, live load, storage live load, earthquake load X and earthquake load Y.



8.8 Node Loads

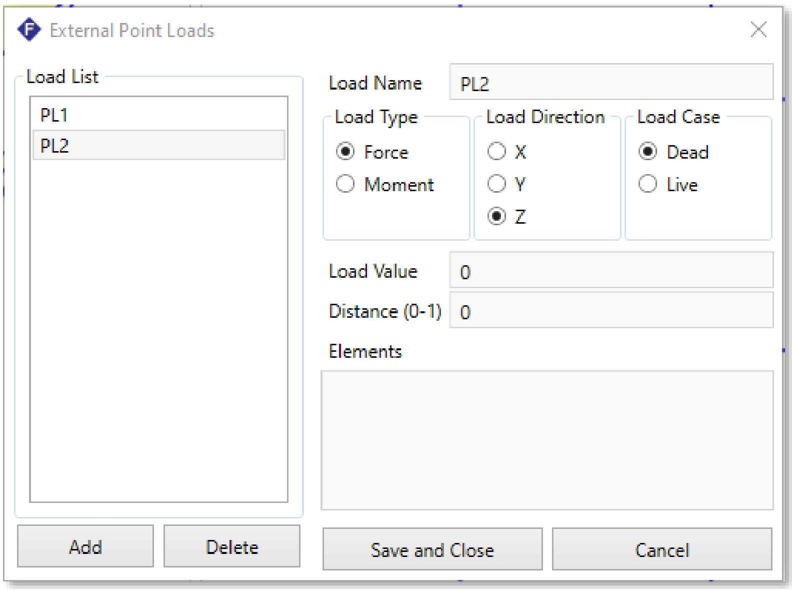
In this window, users have the option to create point load that is to be applied directly to the nodes. The force or moment may be applied in the global X, Y or Z direction, and it may be dead load or live load. The IDs of the nodes to be added has to be entered, separated by commas, in this window. The nodes may also be selected graphically and the list of selected nodes will be shown here.



8.9 Element Loads

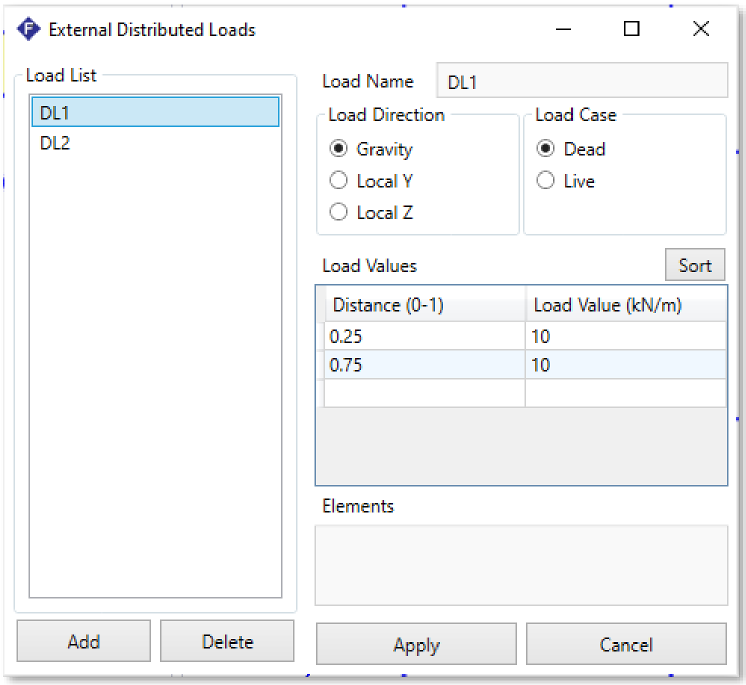
8.9.1 Point Loads

Point loads can be defined with different magnitude, direction of application and load case (dead or live). The distance is a factor between 0 and 1 where 0 is the start node and 1 is the end node.



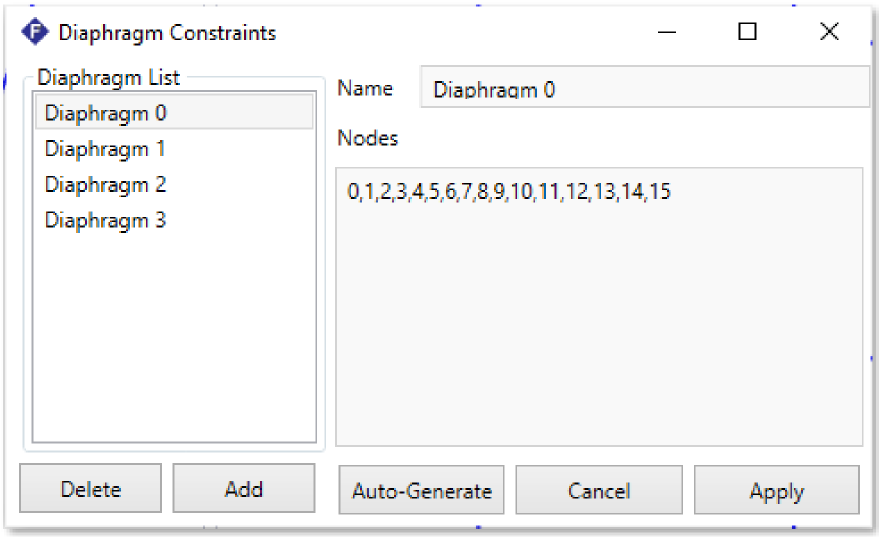
8.9.2 Distributed Loads

Distributed loads can be defined with different magnitude, direction of application and load case (dead or live). The distance is a factor between 0 and 1 where 0 is the start node and 1 is the end node. Load value at each distance point should be mentioned.



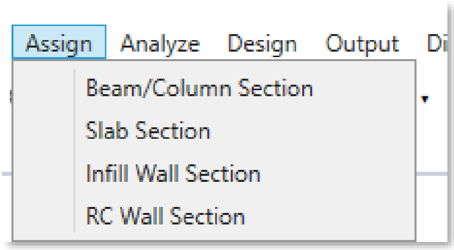
8.10 Diaphragm Constant

This window allows you to add a rigid diaphragm constraint. All the nodes in the diaphragm should have the same Z coordinate. Diaphragms may be auto generated such that all the nodes at the same level are placed in the same diaphragm.



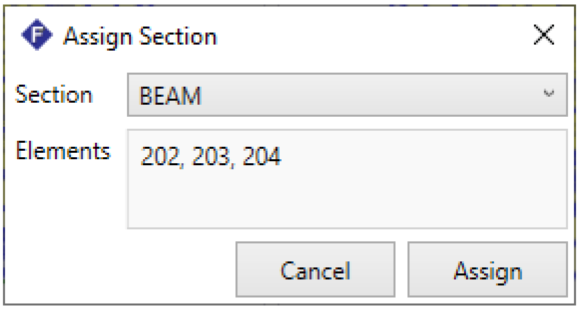
9 Assign

This menu is used to assign sections to the structure elements. The sections must be defined first using the **Define** menu. Following structure elements can be assigned from this menu.



9.1 Beam/Column Section

This command assigns a beam/column section to the selected line elements.



9.2 Slab Section

This command assigns slab sections to selected slab elements.

9.3 Infill Wall Section

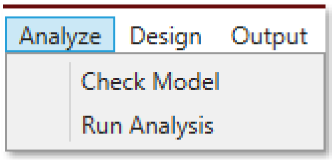
This command assigns infill wall sections to selected infill wall elements.

9.4 RC Wall Section

This command assigns RC wall sections to selected reinforced concrete wall elements.

10 Analyze

After completion of the model formation, the model can be checked for errors and analyzed based on selected code.



10.1 Check Model

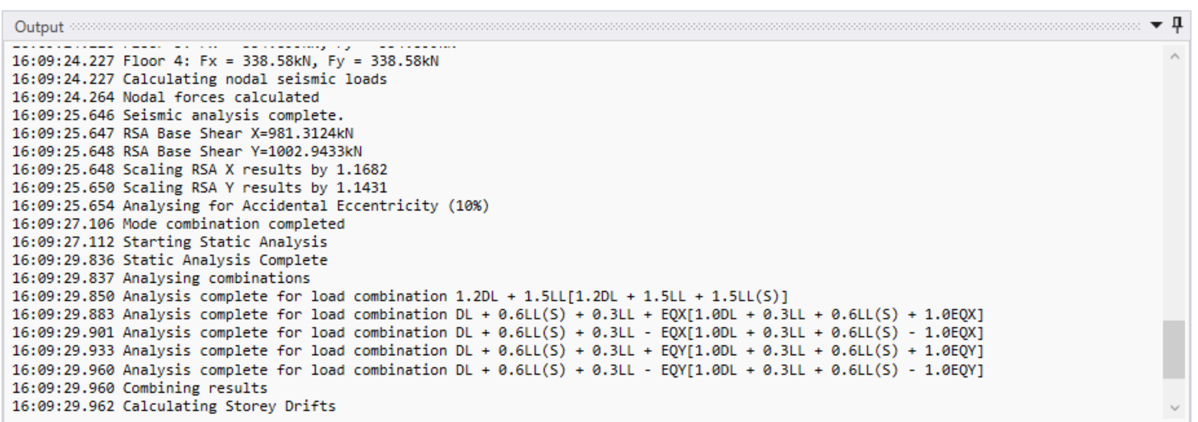
It checks the model and reports any errors that may prevent the analysis from successful completion. The following errors are checked.

1. The number of nodes in the model must be greater than zero.
2. The total number of line elements and RC Walls must be greater than 1.
3. The number of supports must be greater than 1.
4. All infill walls must have 4 nodes that lie on the same plane.
5. All slabs must have at least 3 nodes, and all nodes must lie on the same plane.
6. All slabs must have at least one edge shared with a line element for load distribution.
7. At least one load combination must be present.

10.2 Run Analysis

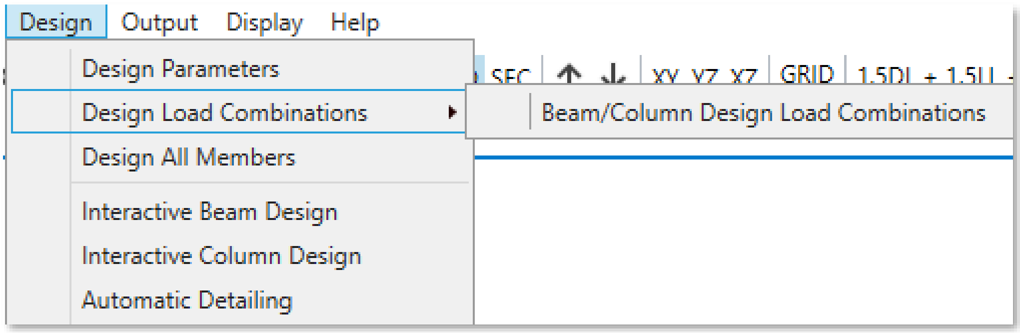
If no errors are found in the model, this command runs an analysis for the model.

The progress and output of analysis is shown in the **Output** panel as the analysis is carried out. Any errors that occur during the analysis is also reported in the **Output** panel.



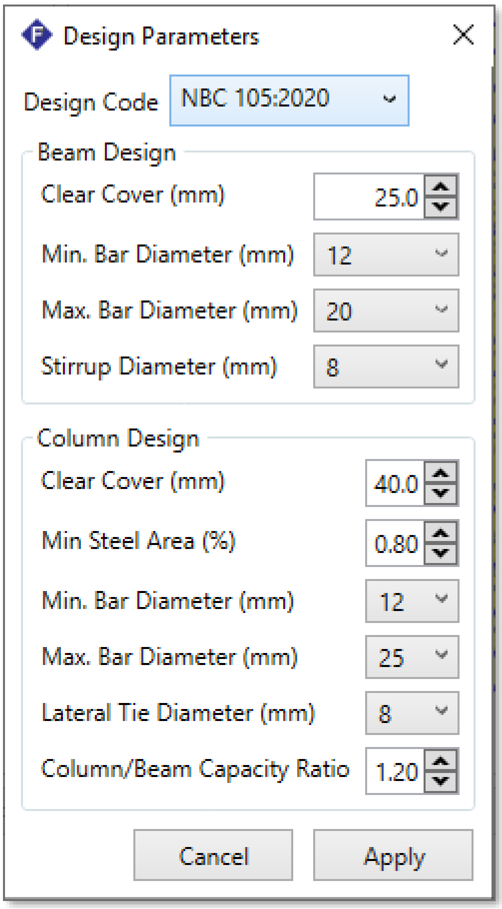
After analysis, the model is locked so that it cannot be modified. To unlock the model, press the lock icon in the main toolbar. Unlocking will delete all analysis results.

11 Design



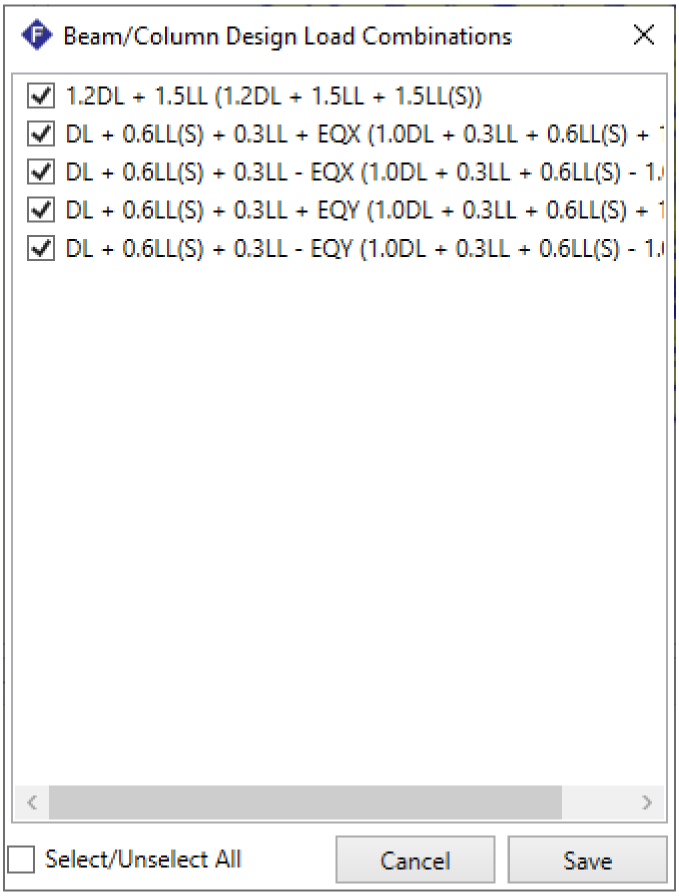
11.1 Design Parameters

In this window, all the parameters for the design of beams and columns can be defined.



11.2 Design Load Combinations

In this window, the load combinations to be used for the design of beams and columns can be selected.



11.3 Design All Members

This software carries out design of all reinforced concrete beams and columns present in the model and computes the reinforcement required.

This step may only be carried out after analysis is successfully performed. Results of the design can be viewed in the **Interactive Design** windows, and the longitudinal rebar percentage can be shown using the **Display->Rebar Percentage** menu.

11.4 Interactive Beam Design

The interactive beam design allows the user to perform the design and detailing of beams while interacting with the drawings.

To enter interactive design, select one beam element on the viewport and open **Design->Interactive Design**. Alternatively, you may also click **Interactive Design** on the context menu that opens after right clicking on the viewport.

The central portion of the window contains the longitudinal section of the full beam, and the current cross section is highlighted by a blue box. The current section may be changed by using the left and right arrow buttons below the L-section view. The cross section view on the right shows the active cross section.

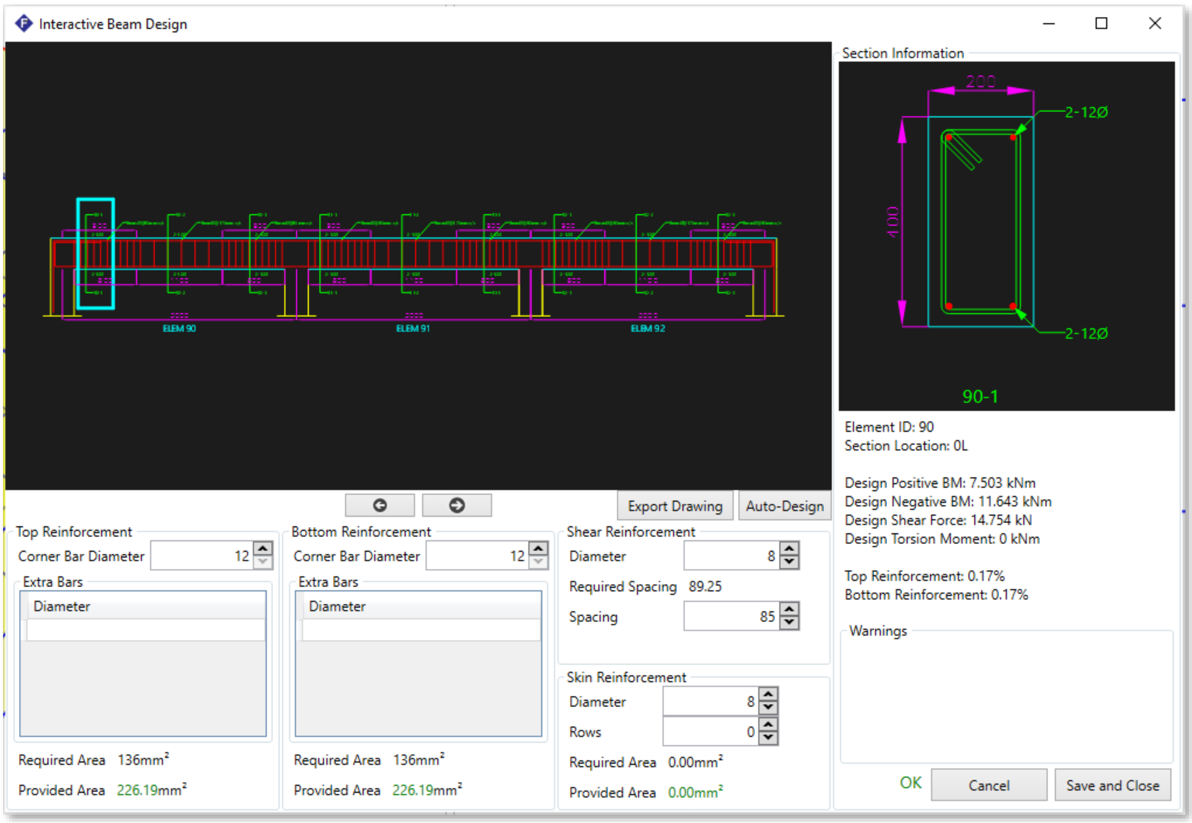
In the lower left part of the window, the user can change the corner bar diameter and add extra bars for top and bottom reinforcement.

The shear reinforcement diameter can be changed, which changes the minimum required spacing. The spacing can then be provided.

The skin reinforcement diameter and number of rows can also be set here.

The changes made to the different parameters are reflected in the drawing instantaneously. Also, if any parameter renders the section unsafe it is highlighted in the warning section. The drawings can be exported to DXF by **Export Drawing** button.

The **Auto-Design** button recalculates all the parameters (bar diameters, number and spacing) such that the design requirements are satisfied. After the design is complete, press **Save and Close**.

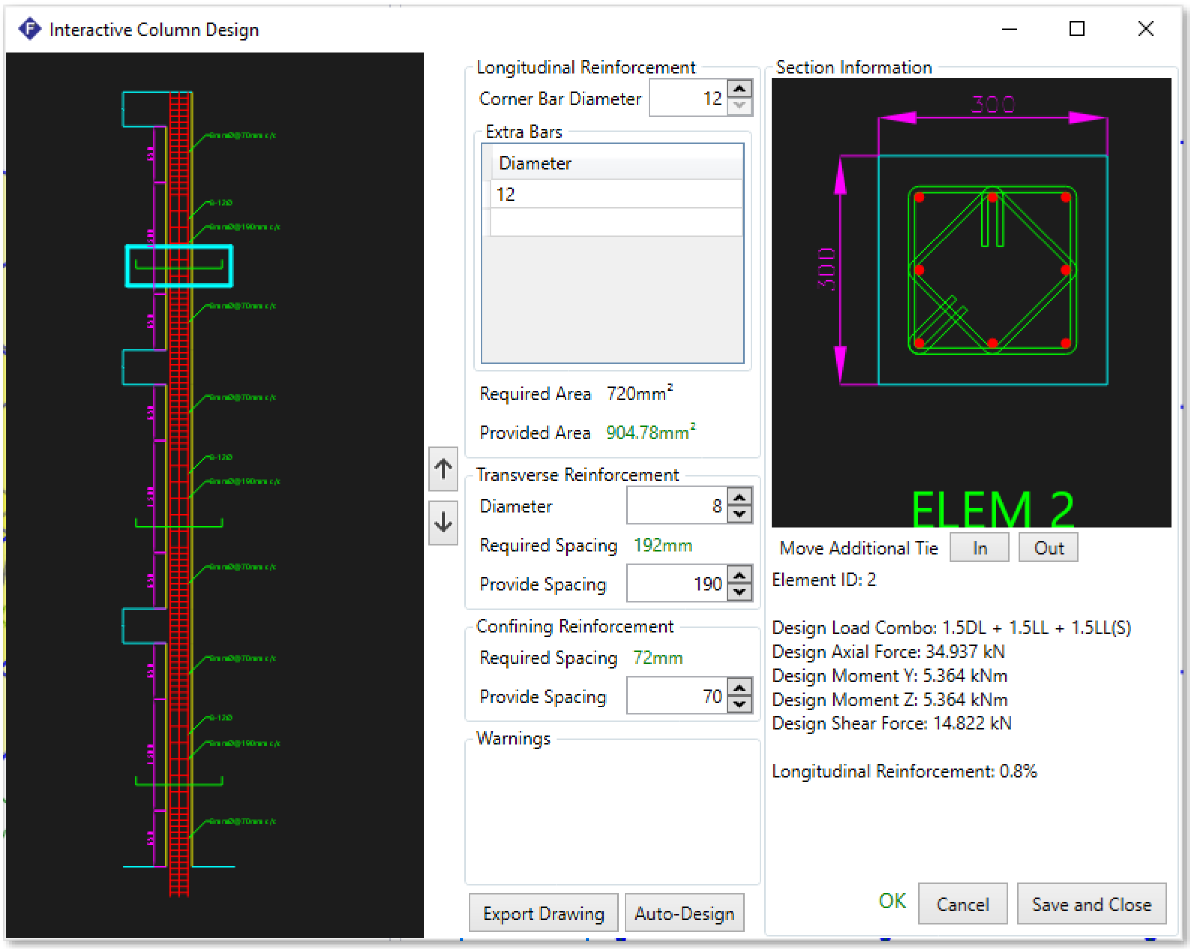


11.5 Interactive Column Design

Similar to the interactive beam design, this window allows you to perform the design and detailing of RC columns while interacting with the drawings.

On the left side of the window, the longitudinal section of the column is shown, with the current section highlighted by a blue box. On the right, the current cross section is shown along with the design loads. The additional tie can be moved using the **In** and **Out** buttons below the cross-section view.

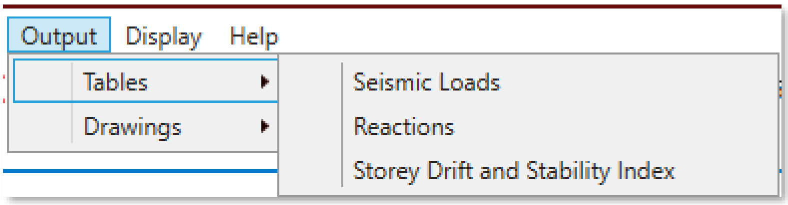
The reinforcement parameters can be changed in the center portion of the window. If any of the parameters do not satisfy the design requirement, warnings are shown. The changes made here are reflected in the drawing automatically. Drawings can be saved to DXF using the **Export Drawing** button. After the design is complete, press **Save and Close**.

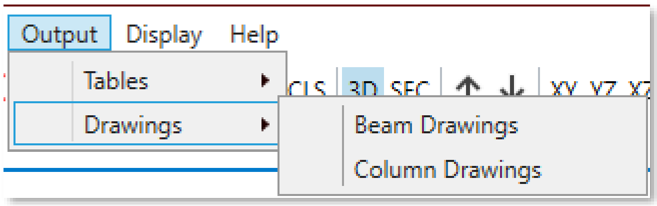


11.6 Automatic Detailing

This command is for calculating all the necessary reinforcement required for each of the elements present in the model with a single click. After this command is executed, the user can export all the beam and column drawings from the **Output > Drawing** menu.

12 Output



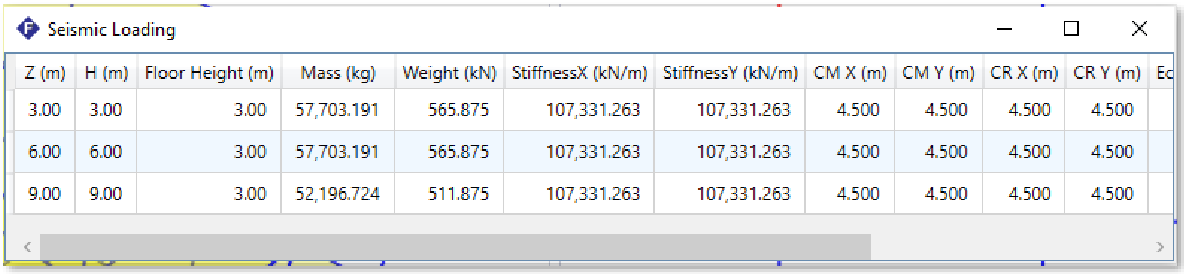


From this menu, the design output can be displayed in table format and drawings can be exported to DXF which can be opened and edited in other CAD applications.

12.1 Tables

12.1.1 Seismic Load

This table shows information about the seismic load that was calculated during the analysis.



Seismic Loading

Z (m)	H (m)	Floor Height (m)	Mass (kg)	Weight (kN)	StiffnessX (kN/m)	StiffnessY (kN/m)	CM X (m)	CM Y (m)	CR X (m)	CR Y (m)	Ec
3.00	3.00	3.00	57,703.191	565.875	107,331.263	107,331.263	4.500	4.500	4.500	4.500	
6.00	6.00	3.00	57,703.191	565.875	107,331.263	107,331.263	4.500	4.500	4.500	4.500	
9.00	9.00	3.00	52,196.724	511.875	107,331.263	107,331.263	4.500	4.500	4.500	4.500	

12.1.2 Reactions

This table displays the reactions generated in all the supports of the model, for all load combinations.

Reactions

Load Combination: 1.5DL + 1.5LL + 1.5LL(S)

Node	Rx (kN)	Ry (kN)	Rz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
0	1.505	1.505	109.484	-1.483	1.483	0.000
1	1.505	0.502	198.253	-0.494	1.483	0.000
2	1.505	-0.502	198.253	0.494	1.483	0.000
3	1.505	-1.505	109.484	1.483	1.483	0.000
4	0.502	1.505	198.253	-1.483	0.494	0.000
5	0.502	0.502	335.651	-0.494	0.494	0.000
6	0.502	-0.502	335.651	0.494	0.494	0.000
7	0.502	-1.505	198.253	1.483	0.494	0.000
8	-0.502	1.505	198.253	-1.483	-0.494	0.000
9	-0.502	0.502	335.651	-0.494	-0.494	0.000
10	-0.502	-0.502	335.651	0.494	-0.494	0.000
11	-0.502	-1.505	198.253	1.483	-0.494	0.000
12	-1.505	1.505	109.484	-1.483	-1.483	0.000

12.1.3 Storey Drift

This table shows the drift occurring in each storey of the model, and also shows whether the drift is permissible according to the selected seismic design code.

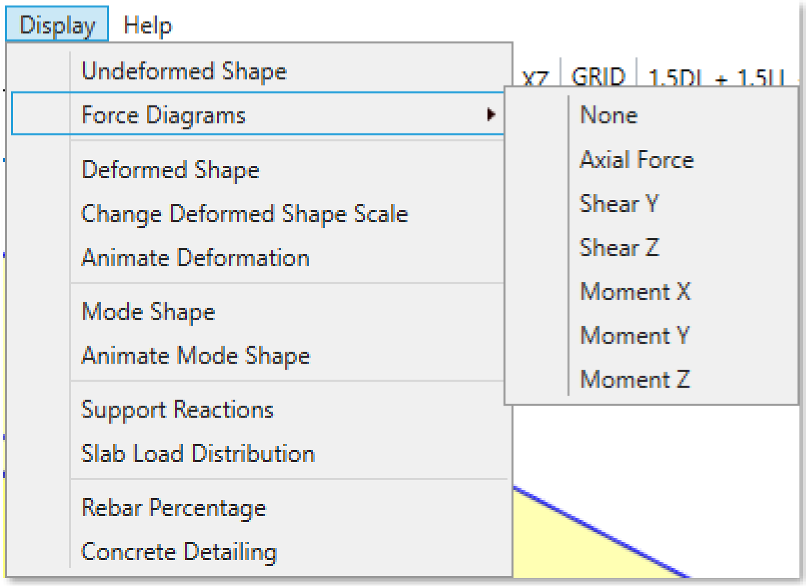
Storey Drift

Z (m)	H (m)	Floor Height (m)	Absolute Drift X (m)	Absolute Drift Y (m)	Relative Drift X (m)	Relative Drift Y (m)	Drift Permissible?
3.00	3.00	3.00	0.002957	0.002957	0.002957	0.002957	✓
6.00	6.00	3.00	0.006468	0.006468	0.003511	0.003511	✓
9.00	9.00	3.00	0.008844	0.008844	0.002376	0.002376	✓

12.2 Drawings

With this command, the user can export all the beam drawings and the column drawing of the model in DXF format.

13 Display

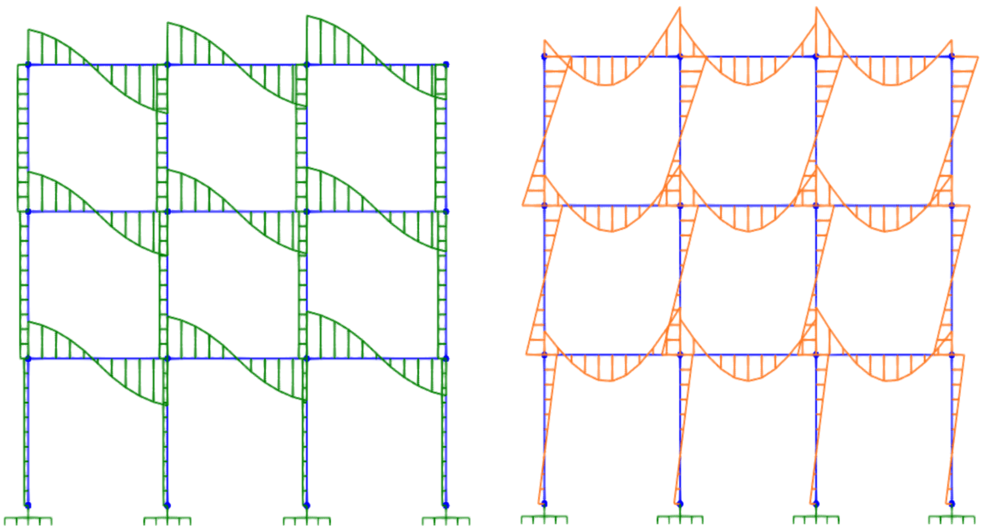


13.1 Undeformed Shape

This sub menu clears all the parameters show in the view window and only shows the undeformed shape of the model.

13.2 Force Diagram

This menu has commands to display axial force, shear in Y direction and Z direction and moments in X, Y, Z directions of each structural elements of the model.



13.3 Deformed Shape

Base on the deformed shape scale, this sub menu displays the deformed shape of the model in the view window.

13.4 Change Deformed Shape Scale

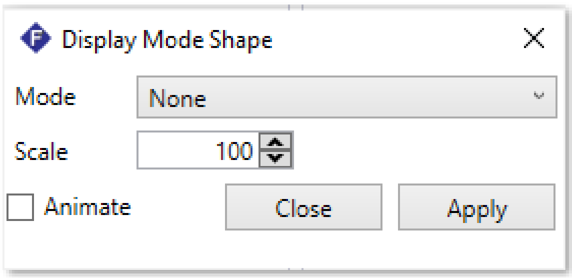
This is a tool to change the deformed shape scale. The displacements shown in the viewport will be scaled by this value for visualization.

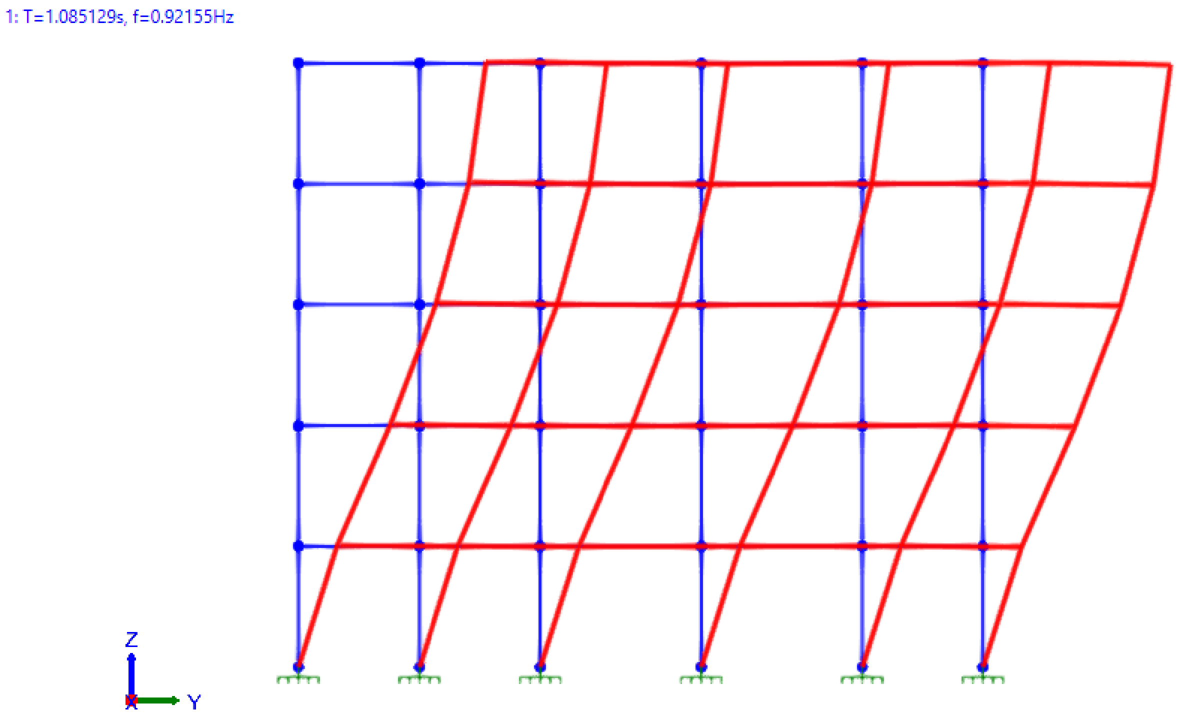
13.5 Animate Deformation

This tool allows the user to animate the deformed shape. If the deformed shape is not shown, this command does nothing. If the deformed shape is being animated, this command will stop the animation.

13.6 Mode Shape

This command will show the mode shapes computed using modal analysis. Mode shapes are available only after response spectrum analysis. Select the mode number and scale and then press **Apply** to show the mode shape. Check the **Animate** checkbox to animate the shape.



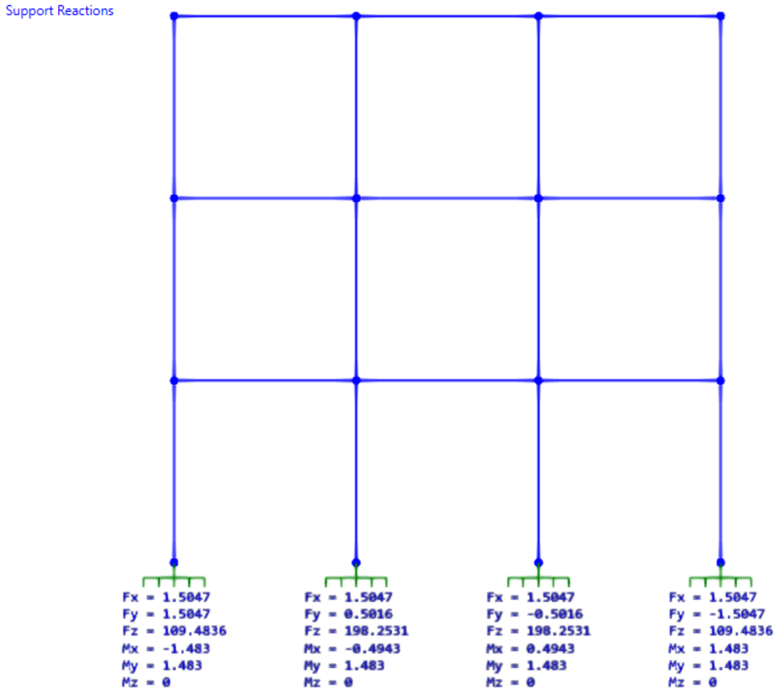


13.7 Animate Mode Shape

Animate mode shape command animates the currently shown mode shape. If no mode shape is shown, this command does nothing. If a mode shape is being animated, this command will stop the animation.

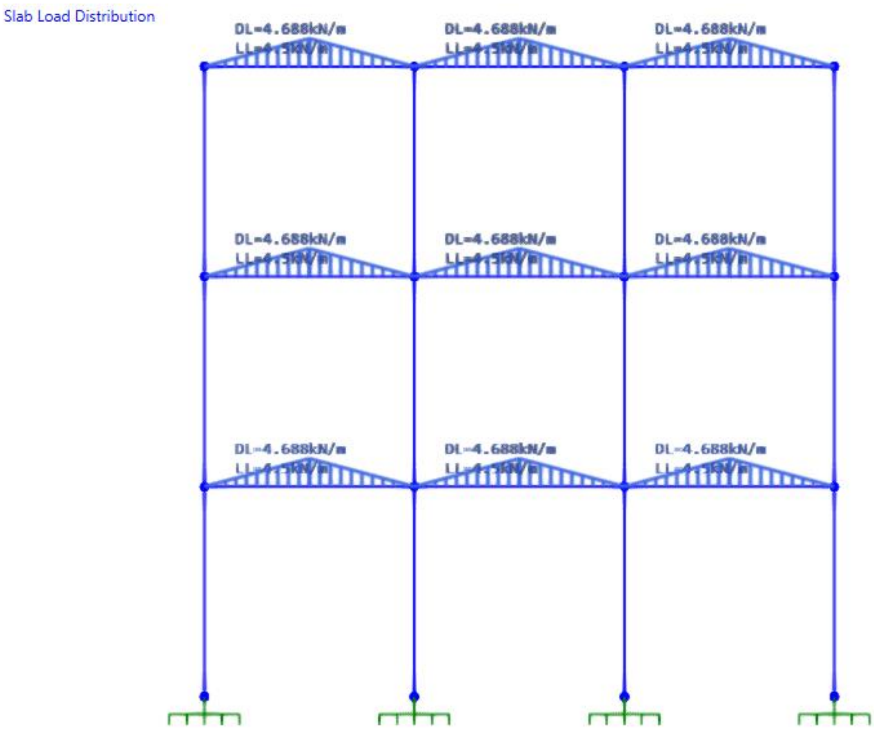
13.8 Support Reactions

This command displays the computed reactions for the current load combination in all supports.



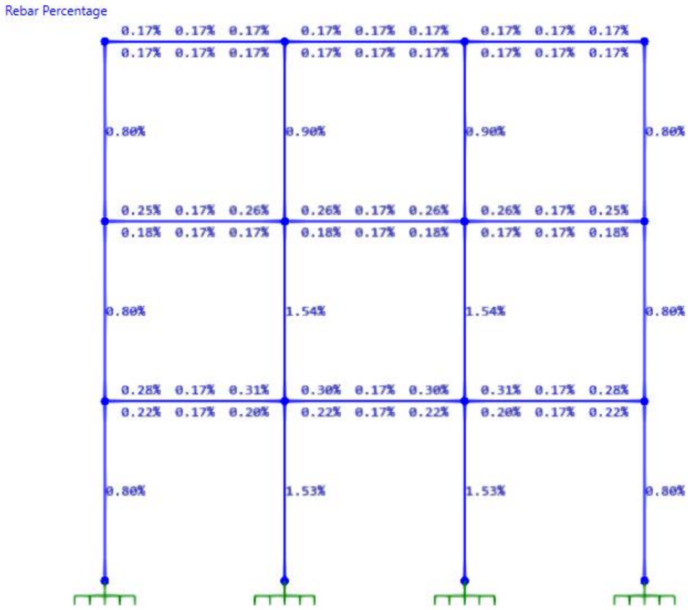
13.9 Slab Load Distribution

This command shows the distribution pattern of the slab loads (dead and live) to the beams.



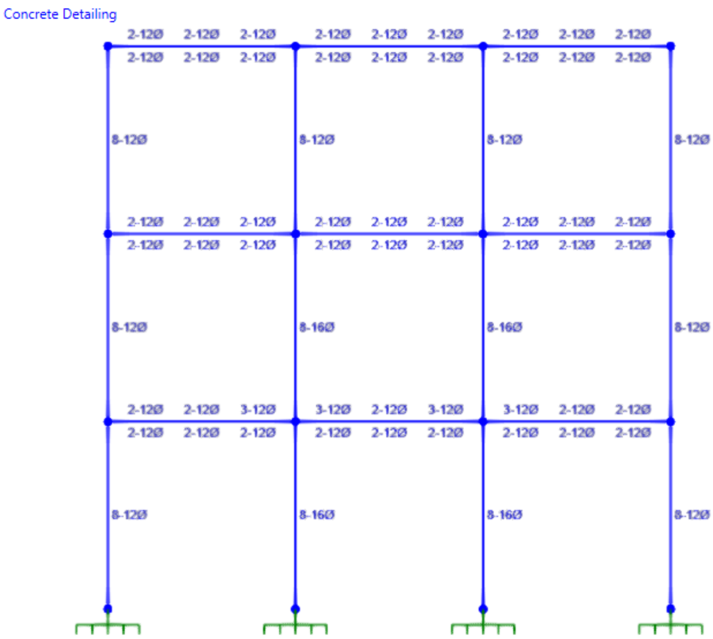
13.10 Rebar Percentage

This command displays the required longitudinal rebar percentage for all designed reinforced concrete beam and column sections.

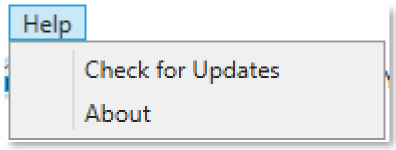


13.11 Concrete Detailing

This command shows the reinforcement assigned to each element in the model. Reinforcement may be assigned through **Interactive Design** or **Auto Detailing**.



14 Help



14.1 Check for Update

It checks whether a new version of SW FEAD is available for download. If update is available, the user can log-in using their Softwel Account and download the updates.

14.2 About

It displays information about the software, and the currently installed version.