BUILD-PRO

USERS MANUAL

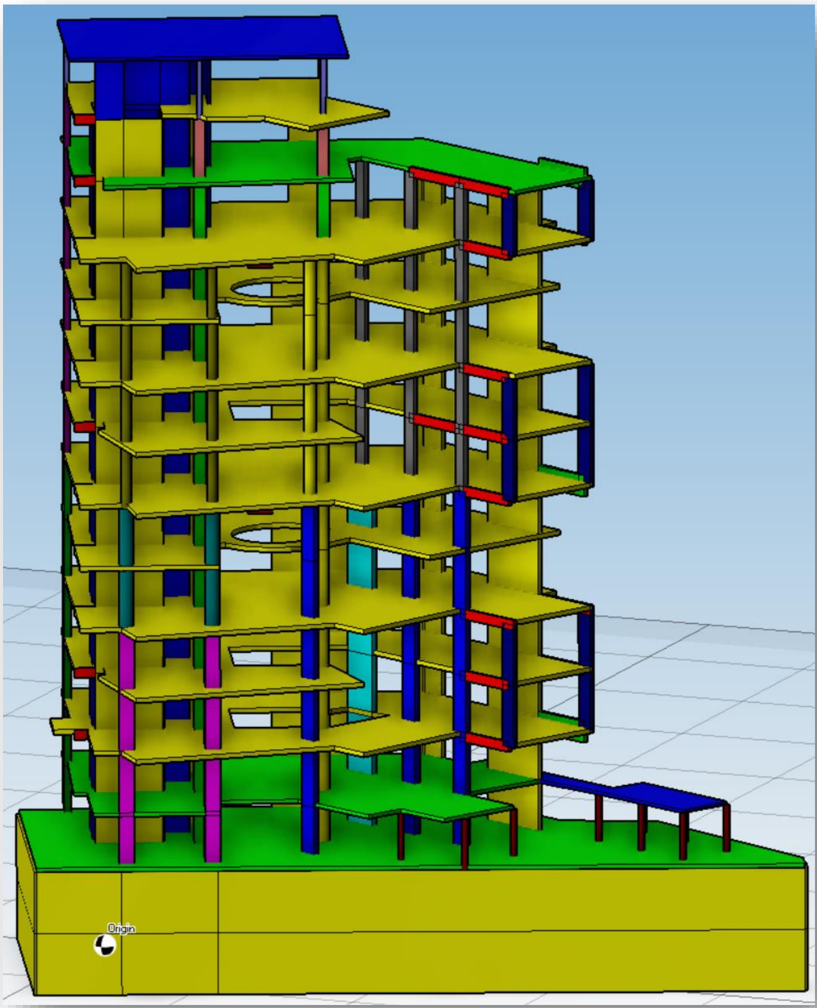


Table Of Contents

Chapter 1- Quick Start. 4
1.1 Quick modeling 5
1.2 Main Features 26
Chapter 2- Introducing the software 27
2.1 BUILD-PRO Interface 27
2.2 3D VIewer Interface 42
2.2.1 The SETTINGS Menu 43
2.2.2 Results: Diagrams 44
2.2.3 Results: Maps 45
2.2.4 Display meshes 46
2.2.5 Tabulated results 47
2.2.5.1 Reactions table 48
2.2.5.2 Modal analysis table: 49
2.2.5.3 Table of forces in floors 50
2.2.5.4 Table of P-Delta check 51
2.2.5.5 Storey drift ratios 52
2.2.5.6 Torsional irregularity 53
2.2.5.7 Geometrical centers 54
2.2.6 The ANALYSIS Menu 55
Chapter 3- Modeling insights 57
3.1 Prepare the drawing and import it 57
3.2 Basis of modeling 58
3.3 Why meshing is not generated 64
3.4 Analysis errors 68
Chapter 4- Design of reinforced concrete members 70
4.1 Sway / Nonsway 70
4.2 Design of walls 71
4.2.1 Drawings generation 73
4.3 Design of columns 74
4.3.1 Drawings generation 79
4.4 Design of slabs 80
4.4.1 Design for flexure and membrane forces 80
4.4.1.1 Drawings generation 81
4.4.2 Design for punching shear 82
4.4.2.1 Drawings generation 83
4.5 Design of beams 84
4.5.1 Design for bending, shear and torsion 84

4.5.2 Drawings generation and deflection calculation 85
4.5.2.1 Drawings generation 85
4.5.2.2 Deflection calculations 87
4.6 Design of foundations 89
4.6.1 Drawings generation 93
Chapter 5- Sample models. 94

Chapter 1- Quick Start.

Eager to start? Here are the steps you should follow.

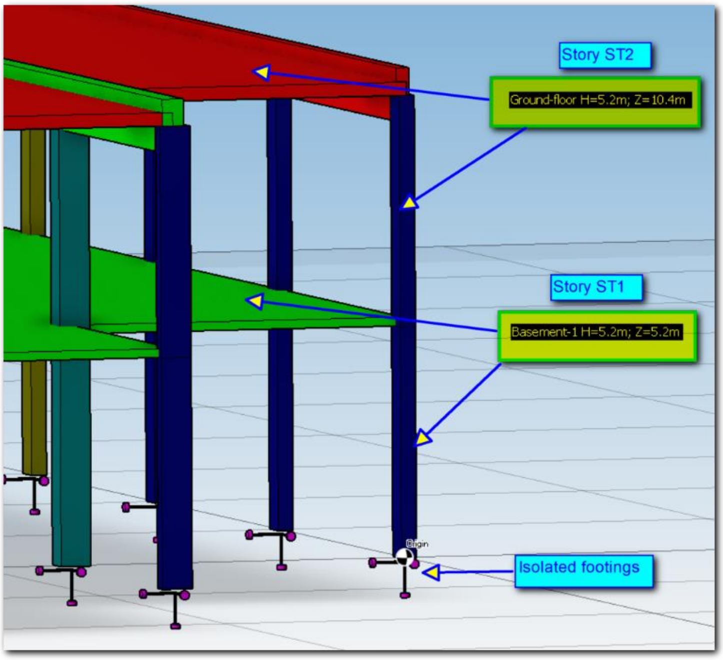
1. Prepare the AutoCAD drawing based on the structural framings
2. Open the prepared AutoCAD drawing of the building. It should be 2D, without blocks, and units should be meters. Keep in the drawings what is just needed for BUILD-PRO.
3. Save the project. A BUILD-PRO project consists of a folder; it is not a single file. Inside this folder many files will be generated, all associated with the model.
4. Adjust in the drafting settings the pointer snap and the orthogonal tracking. This will be of much help when modeling.
5. Open the General Settings from the Settings menu and specify the materials and the eventual presence of raft foundations.
6. Once finished with the General Settings, open the Stories Manager from the Stories menu and introduce the name and the height of the first story.
7. In the Layers window (opened with the Format menu) the structural layers which name begin with “ST1-…” are created automatically. BUILD-PRO associates to each layer the corresponding structural element (beam, column, slab….) that belongs to the story ST1. Each time a new story is added, the corresponding layers are created.
8. Put in the layers which name begin with “ST1-…” the corresponding structural members.
9. The “Convert To Structure” command allows to convert the content of the layers into structural elements. The user can hence affect to the structural members their sizes. Steps 8 & 9 should be repeated for each story.
10. Introduce loads, supports, meshing parameters and combinations.
11. You can see the 3D model by opening the 3D viewer from the “Building” menu.
12. Mesh all slabs at once from the “Building” menu. Each time the geometry of the floor is modified, or any column, beam, wall….is added, the slabs should be re-meshed before proceeding to the next step.
13. Check the integrity of the whole structure from the “Building” menu. It is a mandatory step before proceeding to calculations. Any modification to the model (loads, geometry, etc,…) requires integrity checking. BUILD-PRO verifies the alignment of the columns, walls, intersections at nodes, and affects surface and linear loads to the concerned members.
14. Open the 3D viewer from the “Building” menu and load the structure. Calculations can proceed.

Always remember to frequently save your model !

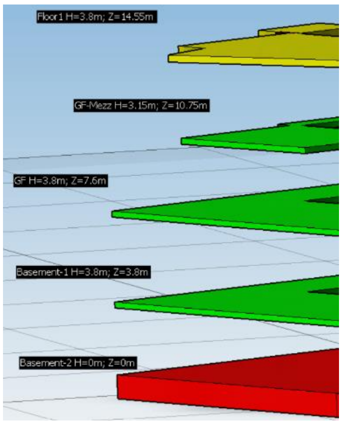
1.1 Quick modeling

Understanding the story concept

A story consists of a slab and the columns below the slab, with a specified height. For instance, the “ground floor” story comprises the columns of the ground floor and the first-floor slab. Hence, it is convenient that the structural framing for each floor, intended for modeling, to show the columns below.

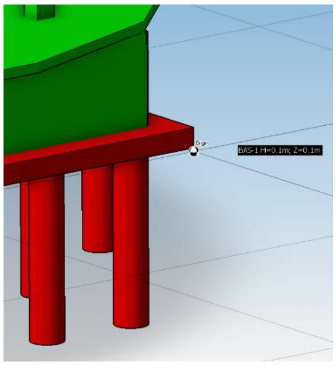


As illustrated in the figure, the first story (ST1), which corresponds to the first basement in this example, includes the columns below basement 1 slab and the slab itself. ST1 consistently represents the lowest level in the structure. The software automatically assigns the base of the column supports, which can be configured as needed. The story above is ST2, it comprises in this case the ground floor slab and the supporting columns which belong to basement 1 floor. These supporting columns and the GF slab represent story ST2.



The bottom slab of the lowest level is not modeled; it is assumed as a floating slab on grade, in the case of isolated footings system.

In the case of mat foundations, as shown in the figure at the left, the lowest story ST1 becomes the raft slab itself; it is introduced with a height equal to zero, indicating that it is supported on elastic springs and not on columns.



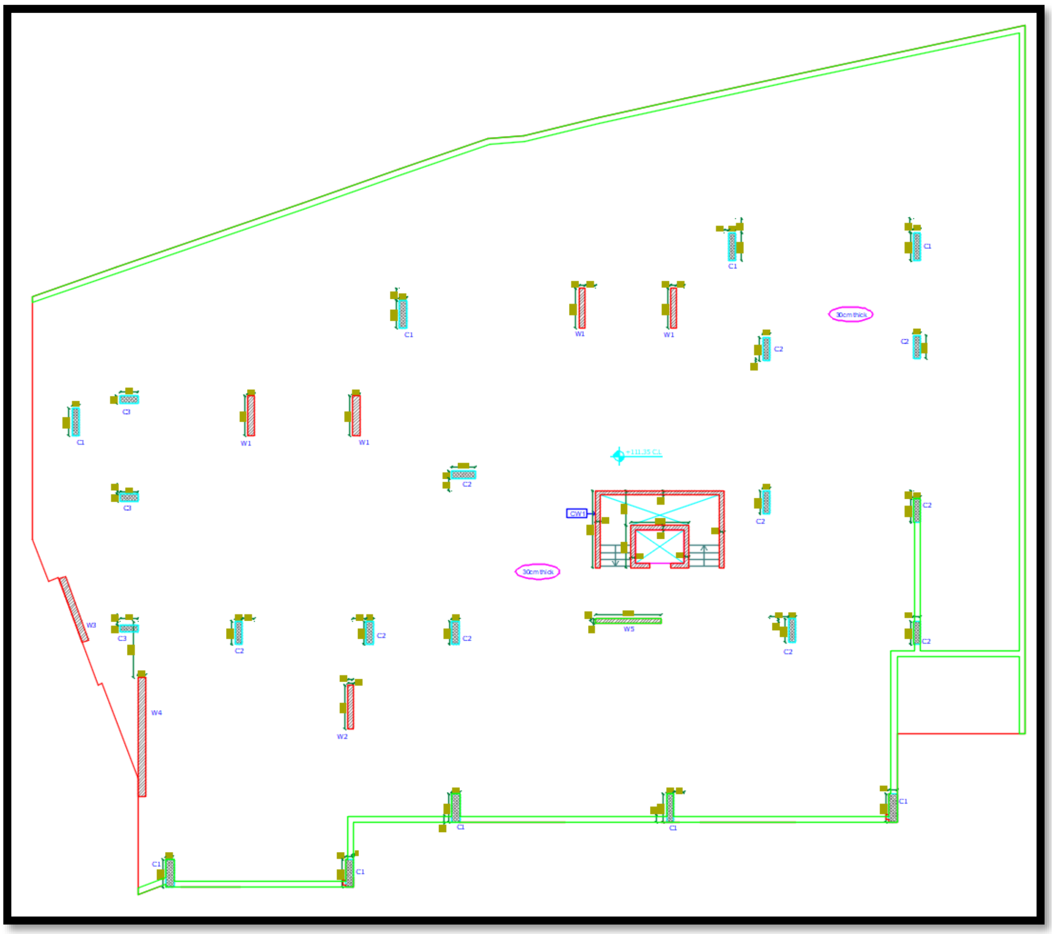
In case of raft on piles, as shown in the figure at the right, the user should affect the 0.1 value to the height of ST1, implying the presence of deep foundations system. The 0.1 value is just nominal.

STEP 1: PREPARE THE AUTOCAD DRAWING.

The following layers should be created in the AutoCAD preparation model.

LAYER NAME	FUNCTION	WHAT TO DRAW
ST-COLUMNS	Rectangular & circular columns	Rectangle (use command rectangle or circle).
ST-WALLS	Walls. Represented as line at the centerline of the wall.	Line
ST-SLABS	The main slabs	closed polyline
ST-SUBSLAB	A slab 100% inside the main slab: Could be a sunken, or a drop panel.	closed polyline
ST-CONTOURS	Lines used to put linear loads on them.	Line
ST-BEAMS	Beams. Represented as line at the centerline of the beam.	Line
ST-OPENINGS	An opening inside a slab or inside subslab	closed polyline

As with other analysis and design software, structural framing must be prepared before proceeding further. Here-below is a sample of a floor's structural framing. The units should be meters. The structural framings are the base of the preparation model, where the new layers should be created. It is mandatory that the framings of all floors to be located in the same drawing, it will be later imported to BUILD-PRO.

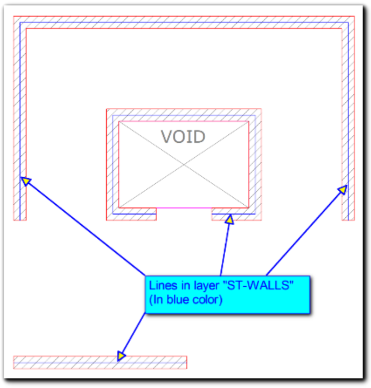


Structural framing of a slab showing walls and columns below

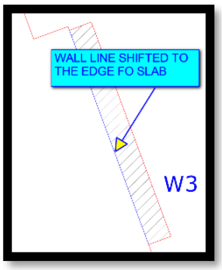
Columns:

Draw rectangles or circles in layer “ST-COLUMNS”. Only rectangular and circular columns are considered in the software. These columns are assumed to be under the concerned slab.

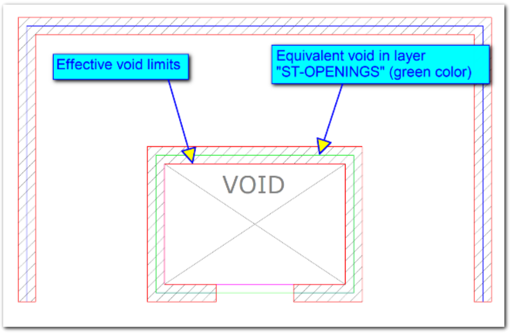
Walls:



Draw lines in layer “ST-WALLS”. These walls are assumed to be under (supporting) the concerned slab.
Typically, these lines are drawn at mid-width of the walls, although this is not mandatory, depending on the geometrical particularities. For instance, the line that corresponds to wall W3 (figure at the right) is shifted to the side of the wall, to match the slab limit. Keeping the line at mid-thickness will lead to a narrow slab strip that might prevent the slab from being meshed.

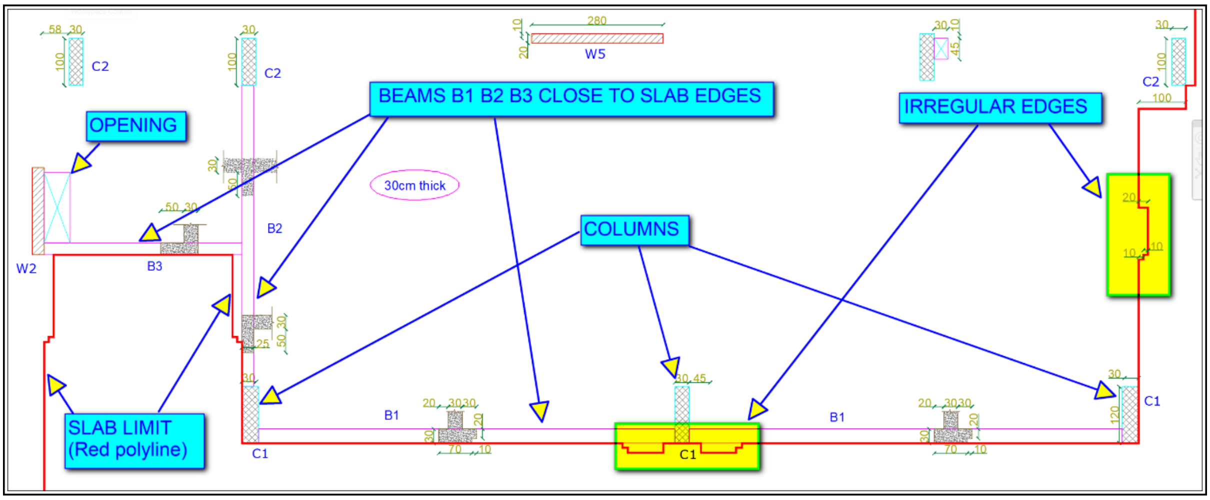


Openings in slabs:



Draw closed polylines (or rectangles) in layer “ST-OPENINGS”. Verify that in the properties window the polyline is effectively closed.
The effective lift shaft corresponds to the “X” symbol in gray color. The modeled opening (rectangle in green color) is larger. It is superposed exactly over the wall’s lines, to prevent dealing with narrow slabs strips that might inhibit successful meshing. An opening should be totally enclosed inside the slab with no common edges.

Beams and slabs:



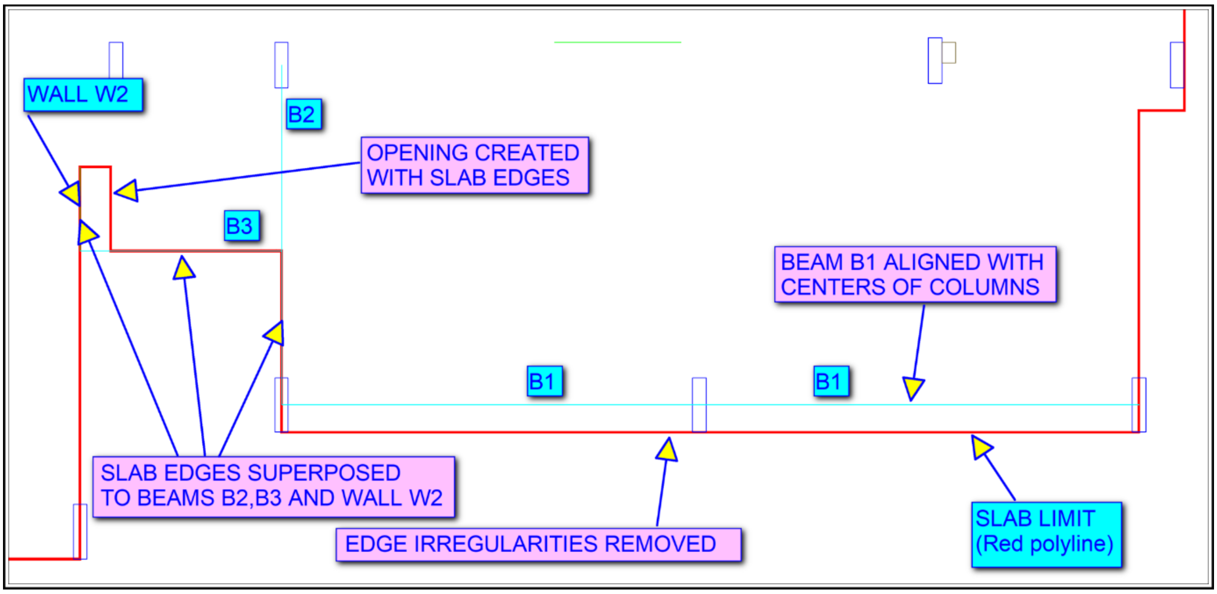
This figure illustrates a typical real structural framing of a slab, showing beams, columns and walls.

The following characteristics are noted:

- Edge beams, close to the slab limits
- Irregular slab edges related to architectural requirements
- Presence of opening at the face of wall W2

The equivalent preparation model is shown in the figure here-below.

- Beams are line objects in layer "ST-BEAMS". Similarly to the treatment of walls, these lines are typically created at mid-width of the webs. But in the present case, to avoid having narrow slabs strip with potential meshing problems, the edges of the slab are heavily modified to superpose over beams lines. In the other hand, beams should intersect with the center of the rectangles which represent the columns. That's why beam B1 shifts upward to coincide with the centers of the three columns. The strip between B1 and the slab edge below is large enough to allow a successful meshing. It is recommended to draw beams as individual segments spanning between column's center instead of one single segment (refer to the two-span beam B1).
- Slabs are modeled as closed polylines in layer "ST-SLABS". Most of the time, it is not possible to follow exactly the architectural outline. It is highly advisable to remove decorative protrusions and recesses, as they complicate meshing and greatly increase the number of meshes. At the left of the figure, there is an opening at the corner of the slab, adjacent to wall W2. A particular treatment is required. It involves aligning W2 with slab edge, and modeling the opening with the slab edge itself, without the need to create an opening by drawing a rectangle in layer "ST-OPENINGS". Beam B3 is as well aligned with the slab edge and coincides with the end node of the wall, though that is not mandatory. It is important to avoid two nodes distant less than 15cm (end nodes of: beam and wall, beam and column center, beam and slab vertex, etc…)
  It is possible to represent the entire slab area with one single closed polyline, or several ones if different thicknesses and different loads are expected.
- To model sunken slabs or drop panels, closed polylines should be used, in layer "ST-SUBSLAB". Such regions should be 100% located inside the slab, with not a single edge of a subslab superposed to any edge of the slab (total island). If the sunken or the drop panel is located at the edge of the main slab, then it should be modeled as a slab and not as a subslab.



Linear loads:

Linear loads are applied in the software on lines objects. The preparation model in AutoCAD can anticipate them by drawing lines in layer “ST-CONTOURS”. If the linear loads are to be applied on beams, there is no need to draw another line over that of the beam. To model facades loads, it is recommended to draw the line directly over the edge of the slab limit (superposed segments).
It is not recommended to have two parallel lines (that of the load and that of slab edge) with a narrow strip in-between.

AutoCAD preparation model: Final configuration.

The drawing to be imported from the software should reflect the sample provided (figure here-below), framings of all slabs included in the same drawing. It is recommended that this latter includes only the designated layers (if possible) and purged from unnecessary elements, particularly:

- Delete all layouts in the paper space from the AutoCAD file.
- Delete all blocks (sections symbols, elevations symbols, attributes…)

If there are similar typical floors, there is no need to duplicate their framing. For instance, in this sample figure, the framing of the 2nd and 3rd floors is made once.

It’s of extreme importance to ensure the vertical alignment of columns and walls along stories. The reliable way consists of copying from a fixed reference the modeled columns and walls from the completed floor to the next one. This will guarantee the perfect superposition of the vertical elements across all floors of the building. Even if the columns and walls are of different sizes and width between two floors, the center of columns should be superposed and same for the wall’s lines. It is easy to verify the absence of offsets by superposing all floors and visually inspecting the drawing.

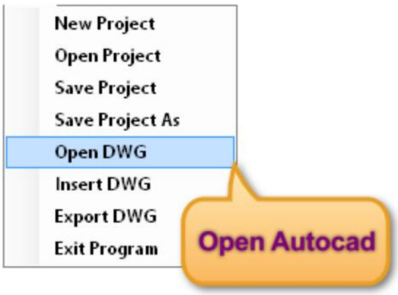
It is possible to implant columns and walls on slabs. The comment above concerns vertical elements that are presumed to be aligned.

BUILD-PRO meshing engine generates regular quadrangular finite elements and can deal with slabs of complex geometry. However, its algorithm prevents the production of distorted elements, i.e. with narrow angles or disproportionate sides. The preparation model done on AutoCAD should be free of ambiguities and simplified.

Refer to chapter 3 for more instructions and advice about modeling

STEP 2: IMPORT THE PREPARATION DRAWING.

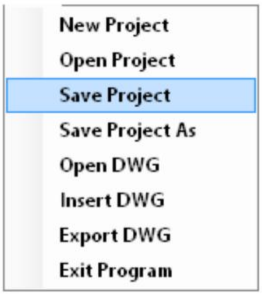
Open BUILD-PRO, go to the file menu and open the prepared drawing.



Before opening the drawing that contains all floors, you should:

- Delete all layouts in the paper space from the AutoCAD file.
- Delete all blocks (sections symbols, elevations symbols, attributes…)
- The structure in the drawing should be drawn in meters.
- Move the drawing the closest to the (0,0) origin.
- Save the drawing as AutoCAD 2013/2010 format

STEP 3: Menu FILE → Save Project.

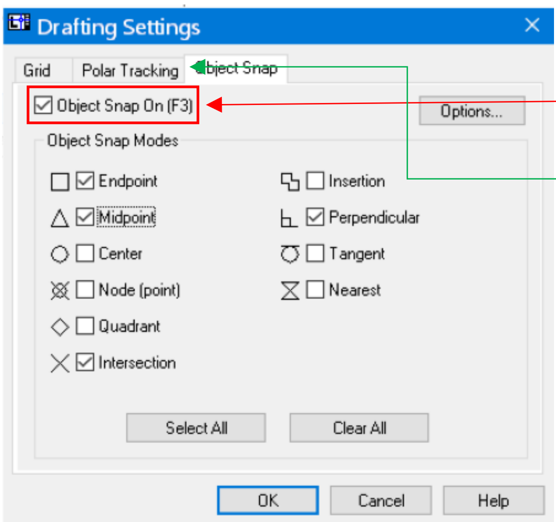


The project is a folder where the program stores its files. A new folder will be created. The name of the folder will be that of your project.

When you want to open a project, you select "Open Project" and you select the folder that you created previously.

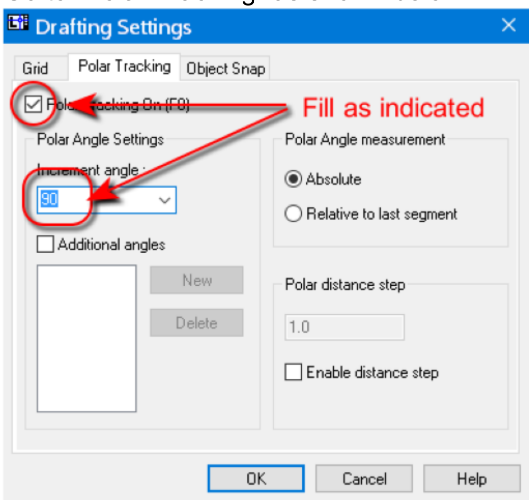
If the software is not able to save the project, it means that the AutoCAD that you opened needs to be cleaned (simplified). It is good practice to keep in the AutoCAD just the required structural layers.

STEP 4: Menu TOOLS → Drafting Settings.

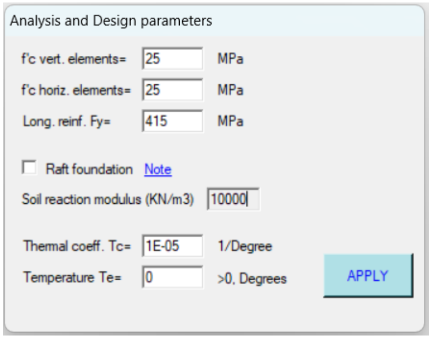


Check this and the items listed below.

Go to "Polar Tracking" as shown below:

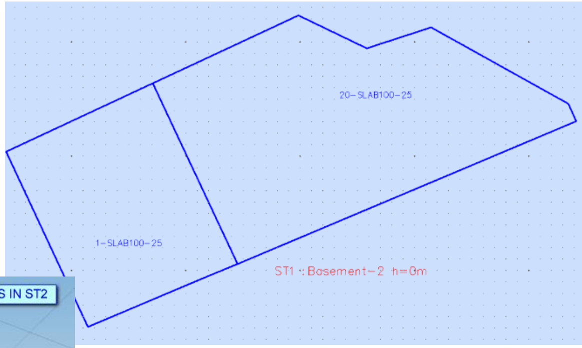


STEP 5: Menu SETTINGS → General settings.

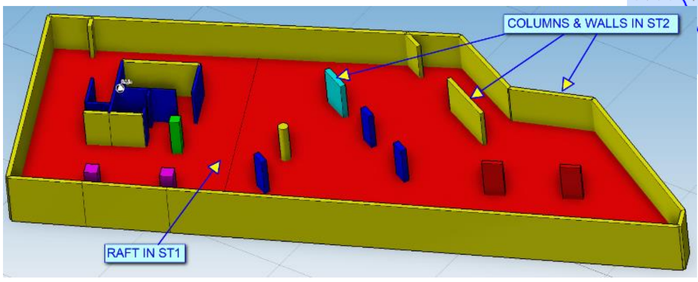


Once the project is saved, the user should define the materials and introduce eventually thermal loads (uniform temperature variation).

The main interest of the Settings at this stage is to specify the foundations type, before opening the Stories window. If raft is selected, the lowest floor ST1 corresponds to the raft foundations layout.



As shown in the right figure, the story that corresponds to the raft has null height. Columns and walls should be introduced in the story above, ST2.

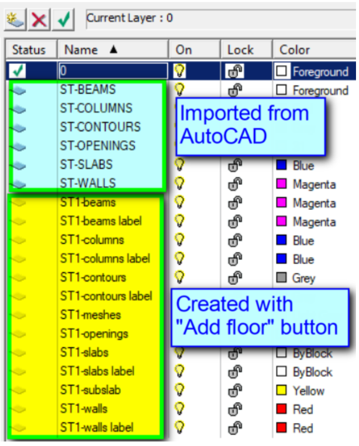


STEP 6: Menu STORIES → Stories manager.

The stories manager provides the following tasks:

- Create a new floor by entering its name, height and clicking "Add floor" button.
- Delete a floor by selecting it first and clicking "Delete floor" button
- Modify the name and height of a floor by selecting it first, entering the new information, and then clicking the "Modify floor" button.
- Display and hide the selected floor(s).

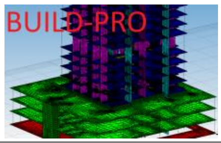
Adding a floor is equivalent to creating new empty layers which names begin with "STx" where "x" is the order of the floor.
Deleting a floor deletes the layers and their content.



Once a floor is added, the "layers" window can be opened from the "format" menu. All layers which names begin with "ST-…" are shown, they contain the structural elements that are already modeled.

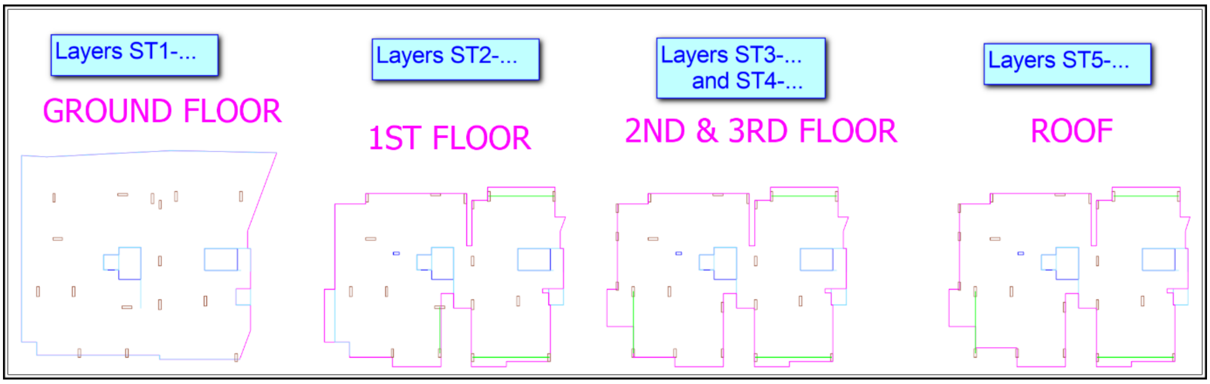
In addition, once a floor is added, the corresponding layers are automatically created. The figure at the left shows the layers of floor ST1 (the lowest one). These layers are empty. The user should select the objects inside layer "ST-BEAMS" and put them in layer "ST1-BEAMS", identically for the columns, walls, slabs, etc…Once done, floor ST1 comprises now the related structural elements.

Layers that end with the word "label" should not be manipulated by the user. They are intended to contain the labels of the beams, columns, etc… which are generated only by the software.



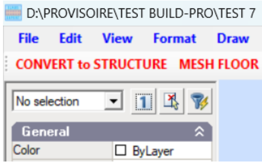
Objects originally placed in layers prefixed with "ST-," without differentiation by floor, should be reassigned to the appropriate story-specific layers: ST1, ST2, and so forth.

This process is straightforward, as BUILD-PRO's graphical interface closely resembles that of AutoCAD. Users may select objects and adjust their respective layers via the menu located on the left side of the screen. Remember that as shown in this figure, all floors should be in the same drawing.



STEP 9: Convert to structure.

Steps 7 and 8 related to the objects assignments to their respective layers are included in step 6.



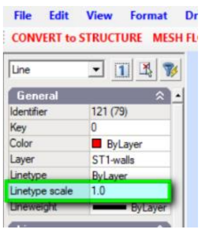
The "Convert to Structure" command is of prime importance. Once a floor is created and the corresponding layers are filled with the structural elements, this command should be prompted. It converts the lines of the beams and walls, the polylines of slabs and openings, the rectangles and circles of columns into structural elements and requests the user to introduce their sections.

The "Convert to Structure" command transforms the lines and polylines into structural elements. In addition, it verifies the integrity of the floor and might trigger error messages if singularities in the model are detected. This verification and the related messages are done to all floors that are created with the stories manager window. In other words, it will apply to the entire building if all stories are created. That's why it is strongly recommended to use it floor by floor to ease the detection of errors and confine them to a single floor. Do not create a new floor if the actual one is not converted to structure.

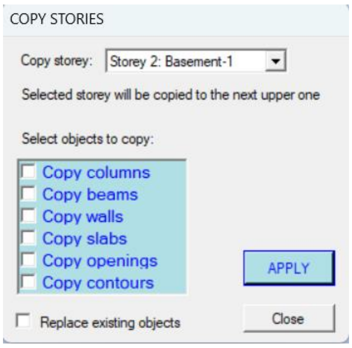
Once the lowest floor ST1 is converted to structure, and all potential errors resolved, it is possible to create a new floor, assign the objects from their original layers "ST-…" to their new corresponding layers "ST2-…", and once again use the convert to structure command. This cycle is to be repeated to the last floor.

Once converted to structure and all sections have been assigned to columns, walls, beams and slabs, it is recommended to mesh the floor. This will guarantee the correctness and convenience of the concerned slab framing. If meshing fails, see the error management chapter in this document.

- Columns sections are automatically recognized, based on the dimensions of the introduced rectangular and circular shapes.
- To assign beams sections, select menu Bars → Create & assign beams.
- To assign slabs thicknesses, select menu Surfaces → Create & assign slabs.
- To assign walls widths, graphically select the walls and enter in the "linetype scale" field the width in centimeters. The default value 1.0 indicates that the width of the selected wall is not assigned yet. Refer to the figure at the right.



When the building comprises typical identical floors (namely the 2nd and 3rd floors in the figure here-above), it is possible to accomplish the modeling of one and then copy it to the subsequent next level. Use the “Copy floor” command from the “Stories” menu to open the appropriate window, and select the objects that need to be copied from floor “n” to floor “n+1”.

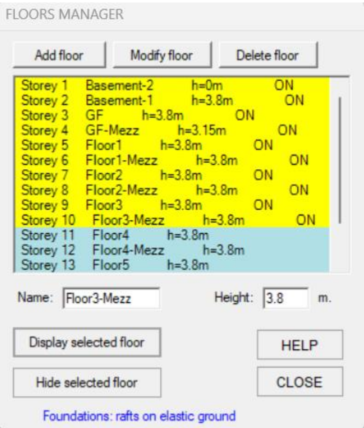


Before copying floor “n”, floor “n+1” needs to be created first from menu “STORIES”→”Stories manager”. If floor “n+1” already exists and contains objects, it is possible to override them by checking the “Replace existing objects” checkbox.

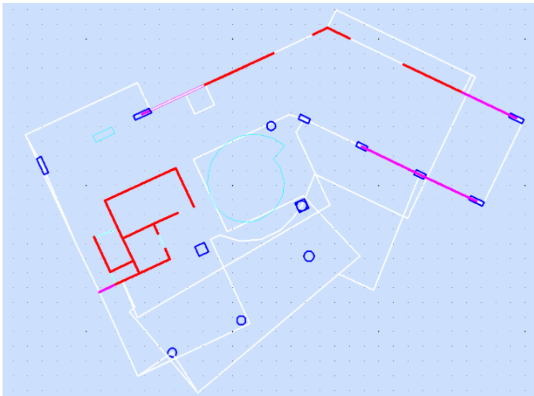
Duplicating identical floors speeds up modeling since all defined loads are copied automatically.

Superposing the floors

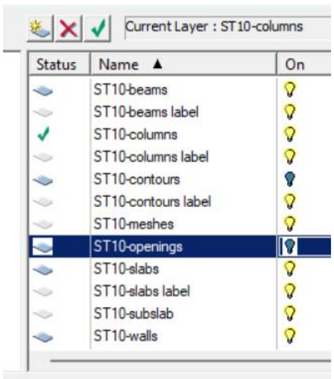
Once all floors are created and their corresponding structural elements are defined, it is necessary to superpose them based on a fixed reference point that is common to all of them. The command “Move” from the “Modify” menu is used to align each floor over the other.



It is possible to see simultaneously all floors by selecting them successively from the stories manager and clicking on the “Display selected floor” button. The displayed floors are highlighted in yellow color. Keep in mind that displaying a floor is simply equivalent to turning on its corresponding layers, which is possible to do manually (though it’s less efficient) via the layers window.



The figure on the right shows a sample of several superposed floors, all displayed. The main interest is to inspect the alignment of columns and walls, which is crucial for the integrity of the structure. One can zoom for inspection and turn off selected layers without having to hide the whole floor.



As an example, in the figure on the left, the contours and openings for floor ST10 are hidden if the user chooses not to display them.

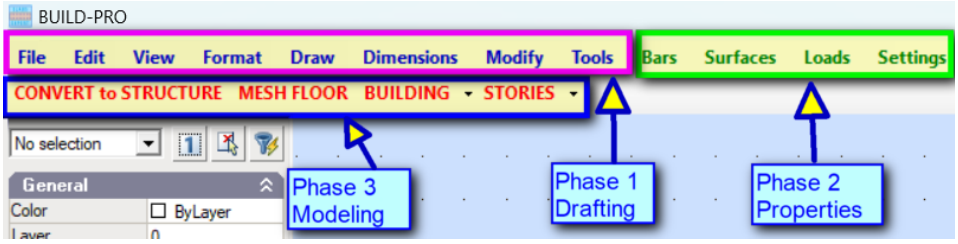
The layers window lets you choose which elements to show or hide, providing hence full control over the floor.

The stories manager always either hides or displays all layers of the selected floor(s), at once.

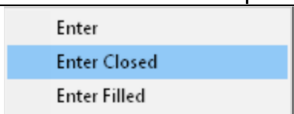
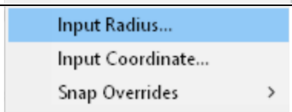
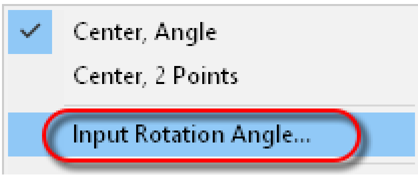
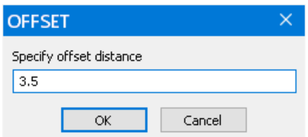
You can make new layers and add inside them objects such as text, lines, dimensions, etc... These additions won’t affect the model because BUILD-PRO only recognizes elements placed in layers labelled “STx-…”.

Drafting tools

BUILD-PRO interface is organized into three distinct sections for drafting, affecting sections and loads, and finalizing the model to be ready for analysis. The related groups of menus are shown in this figure.

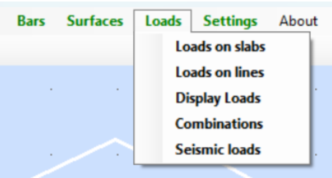


Drafting tools are user-friendly and straightforward. The table below lists the most frequently used ones:

COMMAND LINE	DESCRIPTION
"L" → Line	1- Select the beginning point. 2- For the end point you can type "@3,0"
"PL" → Polyline	 Right-click and choose "Enter Closed" to close the polyline
"C" → Circle	 Right-click to input the radius. But before you have to select the center.
Rectangle: with the menu "Draw"→ "2 Points"	1- Select the beginning point 2- For the end point you can type "@3,2"
"C0" → Copy	1- Select the beginning point 2- For the end point you can type "@3,2"
"M: → Move	1- Select the beginning point 2- For the end point you can type "@3,2"
"RO" → Rotate	1- Select the object 2- Selection the center of rotation  3- Input angle (Right click)
"T" → Draw text	Follow the instructions in the displayed window
"TR" → Trim	1- Select the cutting edge 2- Right-Click and select the part to cut
"EX" → Extend	1- Select the boundary edge 2- Right-Click and select the line to be extended
"O" → Offset	1- Select the line to offset  2- Right-Click and select "distance" 3- Click on the screen for offset direction.

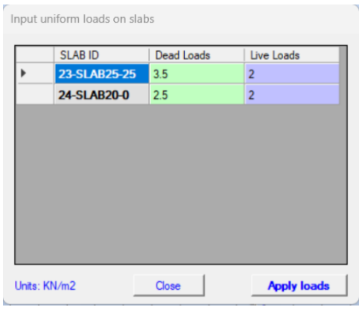
STEP 10: Introducing loads, combinations, supports and meshing parameters.

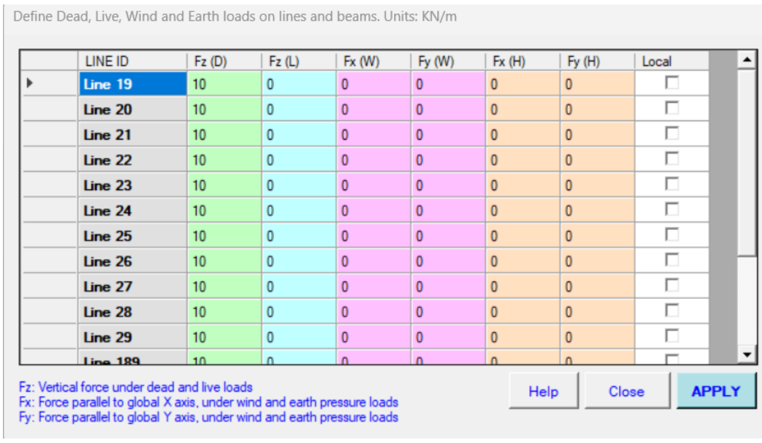
Introducing static loads



Once the sections of the structural elements are defined, and the conversion to structure is successful, it is possible to apply loads on the floor. Surface loads (KN/m^2) are applied to slabs and subslabs, linear loads (KN/m) are applied to beams and to lines located in layer "STx-CONTOURS". Sefl-weight of the structure is automatically considered.

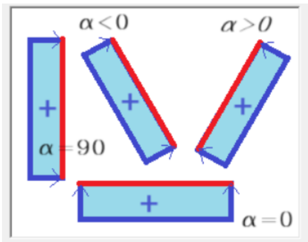
- First, the concerned floor should be displayed (use the menu "Stories" →"Stories manager").
- Second, enter imposed dead and live loads in the window (menu "Loads"→"Loads on slabs"), for each slab. These are gravity loads acting downward; enter positive values. It is not possible to introduce partial uniform loads. If required, the user should define several slabs for variable loads.





To introduce linear loads, use the menu "Loads"→"Loads on lines". Lines in layer "STx-CONTOURS" and beams in layer "STx-BEAMS" can receive dead, live, wind and earth loads. These cases are respectively denoted as "D", "L", "W" and "H".

- Dead and live loads are always vertical forces Fz, introduced with positive values.
- Wind and earth loads are always lateral forces, Fx & Fy, oriented according to the global reference system (X,Y). For lateral loads, it is possible to introduce the forces according to the local system of the lines. This option is useful when the lines are not parallel to X or Y axis and the user wishes to apply the loads perpendicularly to these lines. In this figure, α is the angle between the line and the X axis, positive counterclockwise. If the introduced force is >0, then its orientation is as depicted.

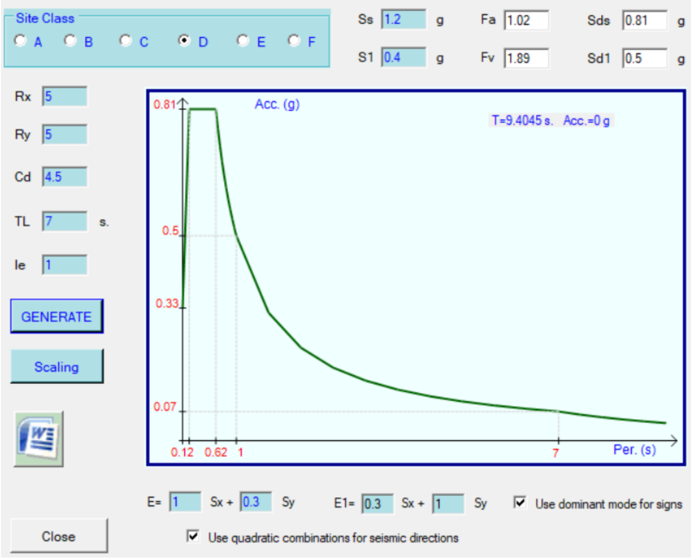


The number of load cases cannot be modified, it is restricted to the tables shown here-above, for surface and linear loads. There are: one dead load case, one live load case, two wind load cases (for X and Y directions) and two earth load cases (for X and Y directions).

Self-weight of the structure (beams, columns, slabs and walls) is always generated by the software. It is embedded in the combinations within the load case "D", i.e. added to the introduced dead loads.

Seismic loads

Menu “LOADS” → “Seismic loads” opens the following window where parameters of the spectral analysis are defined. The construction of the spectrum follows the requirements of ASCE 7-16. The main entries are S_S, S_1, site class, response factors Rx & Ry, deflection amplification factor Cd, importance factor Ie and the long-period transition T_L. Once the “Generate” button is clicked, the program calculates the spectral accelerations and displays the spectrum curve.



The software applies to the eigenvectors (obtained from the modal analysis) the corresponding spectral accelerations, and this in X and Y directions, which are the fixed coordinate system of the software. The corresponding results (forces, displacements and reactions) are denoted Sx and Sy, which are based on the quadratic combination of the individual results of the modes. ASCE 7-16 proposes a linear superposition for Sx and Sy, since earthquake actual orientation is unknown. BUILD-PRO generates two seismic cases, E and E1 which respectively correspond to the dominant actions along X and Y directions.

- E=Sx+0.3*Sy
- E1=0.3*Sx+Sy

It is possible to modify the factors of Sx and Sy. However, combining seismic directions as per E and E1 equations is in accordance with ASCE 7-16 guidelines.

X and Y directions of the fixed coordinate system are arbitrary with regards to the dynamic properties of the structure, unless the principal modes of vibrations effectively have the biggest mass participation factors along X and Y, which is not always the case. If the building is rotated about Z axis, the new base shears in X and Y might be bigger than in the original orientation. The reason is that the main modes of vibrations -which are eigenvectors- are not necessarily “bending” along X or Y axis. For a rectangular building with uniform mass distribution and symmetrical disposition of shear walls in X and Y directions, the principal modes (main flexural modes) will be aligned with this system and consequently base shears have their maximums for these same directions. In this case, if the first mode corresponds -for instance- to the general bending along X, its mass participation will be maximum in X and quasi null along Y.

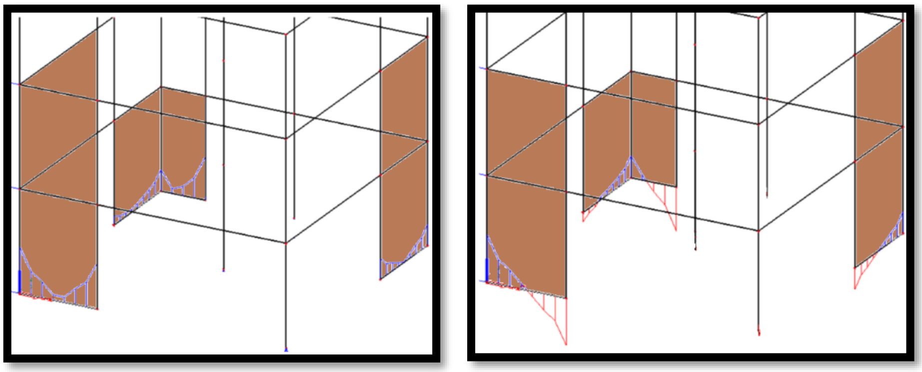
For irregular buildings (the general case) the mass participation for the main modes will be scattered among X and Y. It is recommended that the user rotates the structure about Z axis to “align” its dynamic principal bending reference with the coordinate system (X,Y), noting that depending on the torsional irregularities of the building, it is not possible to reach an “exact” orientation, but rather an optimal one. The target is to attain an orientation where the mass participation of the two major bending modes have maximal components in X and Y, and minimal ones in Y and X.

While looking for the optimal orientation, it is possible to set the 0.3 coefficient to zero in the seismic cases E and E1. Hence these cases will correspond respectively to the X and Y components of the earthquake. The user can verify that base shear in X direction under E case and that in Y direction under E1 case effectively reached their maximal values.

As an alternative to the directional combination approach, the user may adopt the quadratic combination method. Only one seismic case is created: E=$\sqrt{S_x^2 + S_y^2}$ where the factors of Sx and Sy are always 1.

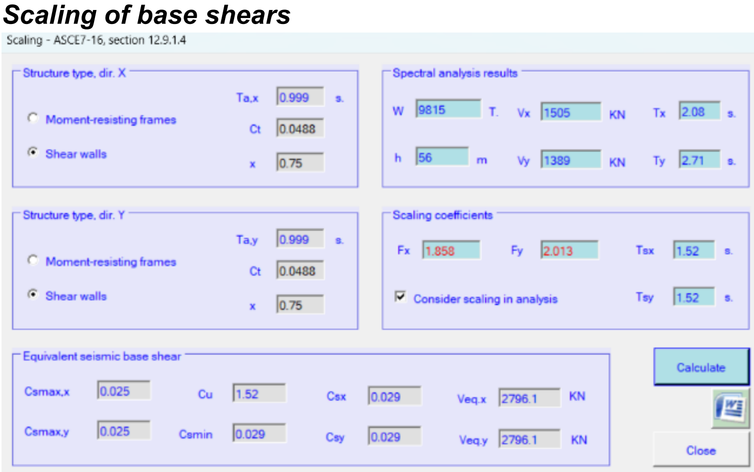
Use dominant mode for signs

During the combination of modal result vectors, the sign of the displacements and internal forces are lost and all values become positive. That's consequent to the quadratic combination of modes results (CQC) which mathematically leads to positive figures. This loss of sign leads to incoherent deformed shape of the structure when visualizing seismic displacements, and to confusion when designing shear walls since tension/compression boundary zones cannot be distinguished.
When the "Use dominant mode for signs" is activated (refer to the figure above), the program detects the dominant mode that corresponds to the result (axial force, moment, reaction, displacement….) and affects to this result the sign that it had in the mode. For example, considering a force in a finite element that belongs to a wall, its sign in the spectral analysis Sx is that it has in the mode in this direction with maximal mass participation. This same force, in seismic direction Sy, will have the sign of the dominant mode in this direction.
The major benefits consist of a more realistic computation of the global equivalent in-plane axial force and in-plane bending moment in walls, which are required in reinforced concrete design.



Membrane forces from direct output and with signs considered

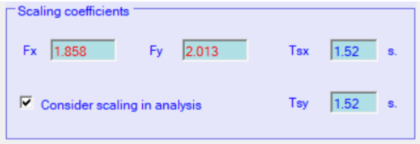
Scaling of base shears



Once the "Scaling" button is clicked, this window appears. ASCE 7-16 section 12.9.1.4 proposes a procedure aiming to amplify the seismic forces obtained from spectral analysis, based on a comparison between the base shears obtained from an equivalent static calculation and those obtained from the rigorous dynamic analysis.

The user should define:

- The lateral-resisting system in X and Y directions
- The dynamic weight (all dead loads with eventually part of live loads)
- Building height
- Dynamic base shears V_x and V_y obtained from E and E1 cases. The reactions table in the 3D Viewer module displays the base reactions under any load case or combination.
- The main modal periods in X and Y directions which correspond to modes with maximal participation factors.

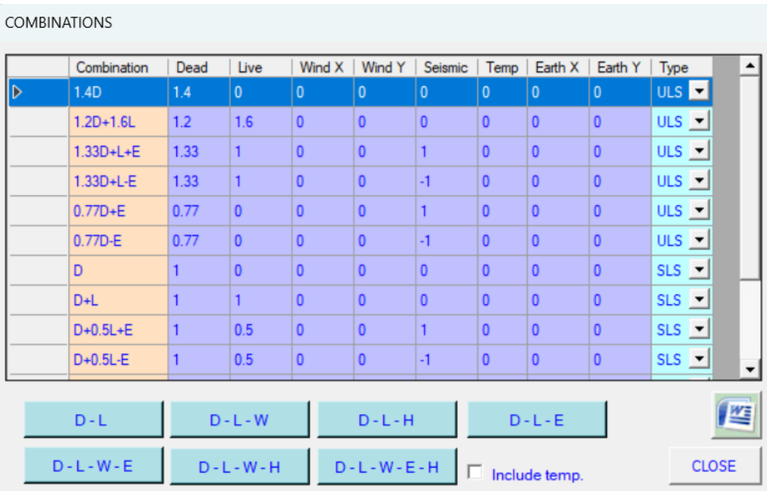


The software provides the scaling factors Fx and Fy used to automatically amplify the generated seismic results Sx and Sy. The user should activate the "Consider scaling in analysis" checkbox and run again the analysis.

The scaling factors Fx and Fy are obtained by dividing the equivalent static base shears $V_{eq,x}$ and $V_{eq,y}$ by respectively V_x and V_y. The user can override these factors by introducing different figures.

Once scaling procedure is performed, the user should re-generate the spectrum by clicking on the "generate" button in the Spectral Analysis window.

Generating combinations of load cases: Menu "LOADS"→ "Combinations"


COMBINATIONS

Combination	Dead	Live	Wind X	Wind Y	Seismic	Temp	Earth X	Earth Y	Type
1.4D	1.4	0	0	0	0	0	0	0	ULS
1.2D+1.6L	1.2	1.6	0	0	0	0	0	0	ULS
1.33D+L+E	1.33	1	0	0	1	0	0	0	ULS
1.33D+L-E	1.33	1	0	0	-1	0	0	0	ULS
0.77D+E	0.77	0	0	0	1	0	0	0	ULS
0.77D-E	0.77	0	0	0	-1	0	0	0	ULS
D	1	0	0	0	0	0	0	0	SLS
D+L	1	1	0	0	0	0	0	0	SLS
D+0.5L+E	1	0.5	0	0	1	0	0	0	SLS
D+0.5L-E	1	0.5	0	0	-1	0	0	0	SLS

This window displays the load cases available in the software. It is possible to generate the predefined combinations (according to ASCE 7-16) by clicking on the corresponding buttons. For instance, button "D-L-W-E" generates all combinations of dead, live, wind and seismic cases. It is possible to add combinations or modify those generated by intervening on the table.

The type (ULS or SLS) should be specified. When reading the analysis results in the 3D Viewer, the software performs distinct envelopes for ULS and SLS combinations. Furthermore, in reinforced concrete design modules, when ULS envelope is selected, the software considers the relevant combinations. That's why the combination's type is more than just an attribute.

When the checkbox "Include temperature" is activated, the generated combinations incorporate the case "T" that denotes thermal loads. Thermal loads are introduced in the menu "SETTINGS"→ General settings window. A positive value should be entered, corresponding to temperature variation. The corresponding expansion strains will be applied to all the structural elements. Retraction strains (negative variation of the temperature) are considered in the combinations, since the load case "T" occurs with factors (+1) and (-1).
Combination's names, when introduced by the user, should follow identical format of those automatically generated. No space is allowed in combination's name, and it is mandatory to keep the labels of the simple cases as per ASCE notation that is adopted in the software, to know: D,L,W,H,T,E. Changing the names of simple cases will confuse the program.

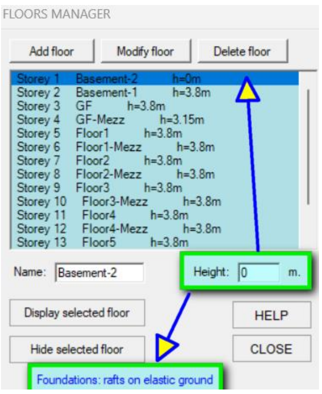
As shown in the table, only one seismic case is present, namely “E”. The software will duplicate all seismic combinations that contain this case, replacing E by E1. Hence, combination 1.33D+L+E will be replicated to 1.33D+L+E1 without intervention from the user.

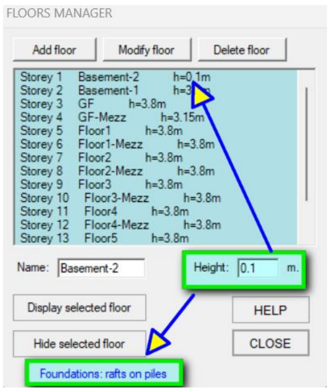
The coefficient 1.33 (in the table above) might appear odd at first sight, expecting instead 1.2. ASCE 7-16 considers the effect of the vertical component of earthquake by integrating to dead loads $\pm$ 0.2*S_{DS}.
The software identifies seismic combinations from their names, that’s why labels D,L,W,E… should be used. If quadratic directional combination is selected (SRSS{Sx,Sy}), only “E” needs to be created.

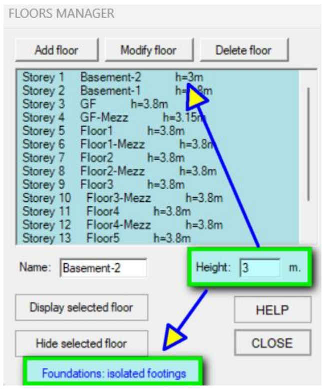
Defining supports.

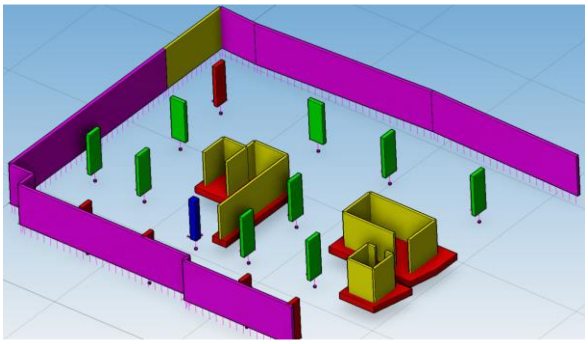
It is possible to define three types of foundations in the stories manager, by changing the height of the lowest floor ST1:

- When a normal height is introduced, the software considers isolated footings under columns and walls. The introduced supports are at the base of columns & walls of floor ST1.
- When floor height=0, rafts on elastic supports are assumed. Rafts can be partial (not covering all columns and walls of ST2) which in this case requires the use of isolated footings.
- When floor height=0.1, the software considers rafts on piles as foundations system. The 0.1m height is just nominal, not of real interest. The user should define the elastic stiffnesses of the piles, as described hereafter. The raft can be partial; columns and walls of floor ST2 can receive isolated piles at their base

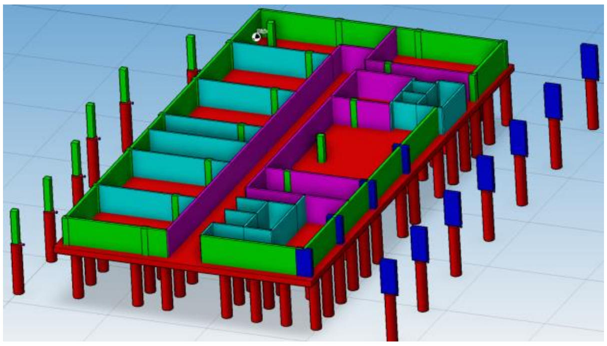








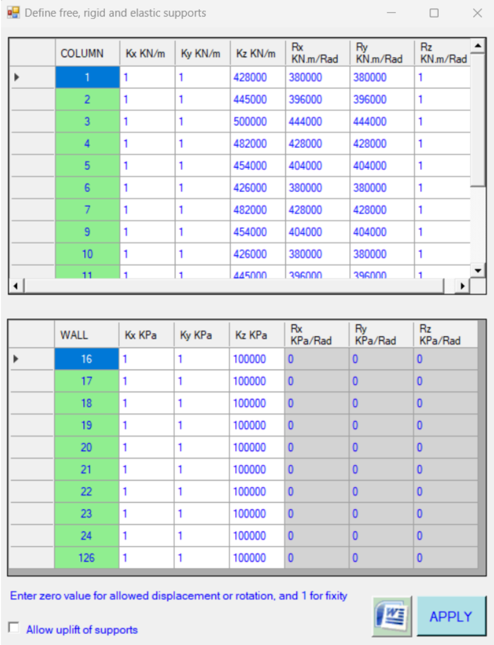
Raft on elastic ground (ST1:h=0)



Raft on piles (ST1:h=0.1)

When the foundations consist of mat plates over all columns and walls, the user needs to define the soil vertical subgrade modulus from the General Settings menu. The software internally affects horizontal supports to ensure the stability of the structure.
When mat plates are partial (refer to the figure here-above at the left), isolated supports under the columns must be defined.

In the case of raft on ground (ST1 height=0) or on piles (ST1 height=0.1) the user should not draw columns and walls in ST1.


Define free, rigid and elastic supports

	COLUMN	Kx KN/m	Ky KN/m	Kz KN/m	Rx KN.m/Rad	Ry KN.m/Rad	Rz KN.m/Rad
▸	1	1	1	428000	380000	380000	1
	2	1	1	445000	396000	396000	1
	3	1	1	500000	444000	444000	1
	4	1	1	482000	428000	428000	1
	5	1	1	454000	404000	404000	1
	6	1	1	426000	380000	380000	1
	7	1	1	482000	428000	428000	1
	9	1	1	454000	404000	404000	1
	10	1	1	426000	380000	380000	1
	11	1	1	445000	396000	396000	1

	WALL	Kx KPa	Ky KPa	Kz KPa	Rx KPa/Rad	Ry KPa/Rad	Rz KPa/Rad
▸	16	1	1	100000	0	0	0
	17	1	1	100000	0	0	0
	18	1	1	100000	0	0	0
	19	1	1	100000	0	0	0
	20	1	1	100000	0	0	0
	21	1	1	100000	0	0	0
	22	1	1	100000	0	0	0
	23	1	1	100000	0	0	0
	24	1	1	100000	0	0	0
	126	1	1	100000	0	0	0

Enter zero value for allowed displacement or rotation, and 1 for fixity

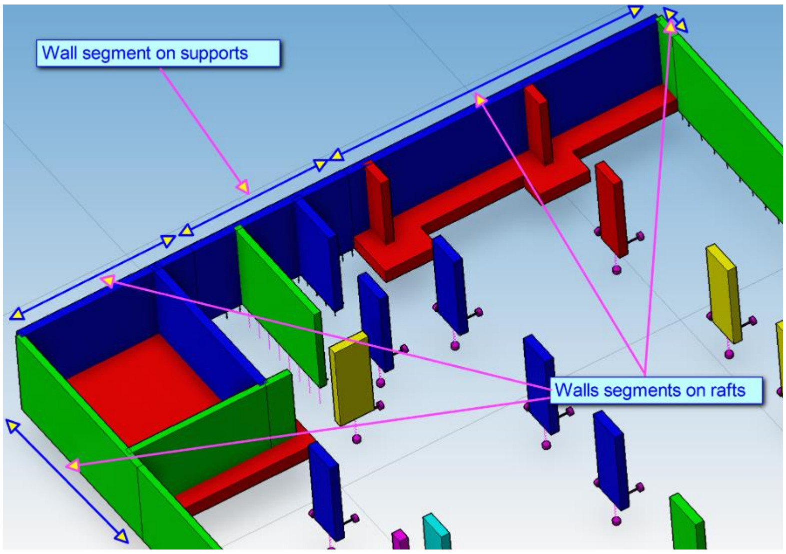
Allow uplift of supports

APPLY

Menu “SETTINGS”→ “Define supports” opens the window showing the labels of columns and walls that are not located over mat foundations. The program assigns per default rigid supports at bottom of columns and walls. Symbol “1” denotes a restrained Degree of Freedom (DoF) for displacements and rotations. Symbol “0” means a released DoF. For example, if for a column the rotational DoF’s are affected the “0” symbol and displacements ones the “1” symbol, the column becomes pinned at its base (no displacements allowed, only rotations).
For walls, rotational DoF’s are always released, assuming small rotations about the weak inertia in presence of strip footing.
Any value other than “0” or “1” is interpreted as stiffness. The footing is hence equivalent to an elastic support. In the figure at the left, vertical supports under walls and columns are springs. Horizontal supports are rigid. Rotational supports under columns, about X and Y axes, are springs as well. Torsional (Rz) DoF under columns is kept rigid.

If uplift is allowed (checkbox at the bottom of the window), the program performs an iterative analysis where supports subject to tension are virtually eliminated. This works as well for mat foundations on soil and over piles.

When columns or walls are deleted or added to the model, the supports are not directly updated. The user should first “convert to structure” and then open the “define supports” window. Once the “Apply” button is clicked, the supports are re-assigned according to the new geometry.



In the case of partial rafts, it is important for walls to be entirely located either over the mat slabs or outside them. This will help the software to correctly recognize the walls under which linear supports will be generated. In this figure, the peripheral wall in blue color is introduced by three segments; the intermediate one will receive linear supports.

Defining meshing parameters.

Meshing of slabs and walls

	Floor ID	Floor Name	Floor height (m)	Slabs (m)	Walls-Bot (0.5<s<1.5)	Walls-Top (0.5<s<1.5)	Walls-Vert (m)
	ST1	Foundations	0	0.6	1	1	0
	ST2	B1	2.6	0.6	1	1	0.43
	ST3	GF	3.5	0.6	1	1	0.58
	ST4	F1	3.1	0.6	1	1	0.52
	ST5	Attic	3.1	0.6	1	1	0.52
▸	ST6	Roof	3.5	0.6	1	1	0.58

Menu "SETTINGS"→ "Meshing parameters" provides a full control on the mesh sizes (and hence number of meshes) of walls and slabs. The program presents for each floor (slab and walls below) an average size for the meshes. The window at the left shows a sample building of 6 floors (ST1 to ST6) where the first one has a heigh of zero, which means it just comprises the mat foundations without vertical elements underneath. The average mesh size for slab is 0.6m. If the user wishes to reduce the size of the model, larger mesh dimension can be set. Slab mesh sizes typically range from 0.6m to 1m, noting that the meshing algorithm might not respect the introduced figures, for several technical reasons.

⚠ The meshing parameters control the meshing that is generated with the command "Mesh all slabs" from the "Building" menu.

Meshing of walls depends on two parameters:

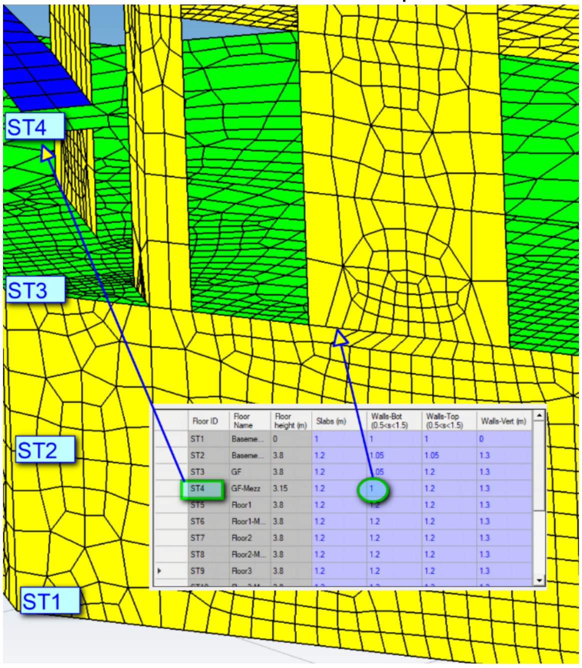
- Length of the mesh in the plane of the slab
- Height of the mesh in the vertical direction.

The height of the mesh is defined by introducing its size in meters (last column of the table).

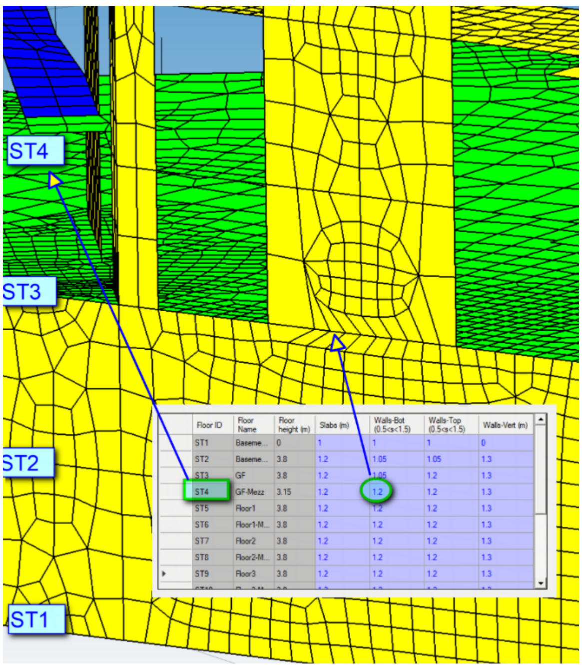
The length of the mesh in a wall cannot be controlled as for slabs, by just introducing a dimension. The coefficient "s" (0.5<s<1.5) is per default equal to 1. It controls the meshing distribution at the bottom sides and top sides of all the walls that belong to the floor in question (columns "Walls-bot" and "Walls-top" in the table), as follows:

- If s<1 then the mesh length is smaller and the <u>number of meshes along the wall will increase</u>
- If s>1 then the mesh length is bigger and the <u>number of meshes along the wall will decrease</u>

Here-below is an illustrative example where the first floor ST1 is a raft:



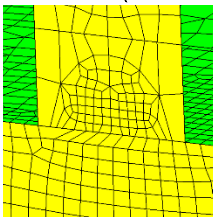
Floor ST4, Wall bottom side s=1



Floor ST4, Wall bottom side s=1.2

Let's consider the walls in floor ST4 (below the ST4 slab) and focus on one of them. In the left figure, the coefficient "s" has the default value 1 for the bottom side. The bottom side of the wall intersects ST3 slab. Its top side intersects with slab of ST4, since always a wall that belongs to a story is in fact under the slab of this story.

When this coefficient is set to 1.2, the number of meshes to be generated at the base of the wall will decrease (refer to the right figure).



If s=0.8 for the bottom sides of ST4 walls, the software generates a much denser meshing. Not all meshes are connected to those of the wall below located in ST3. Such situations may occur and are not considered critical; the program can effectively process unconnected meshes if of small number.

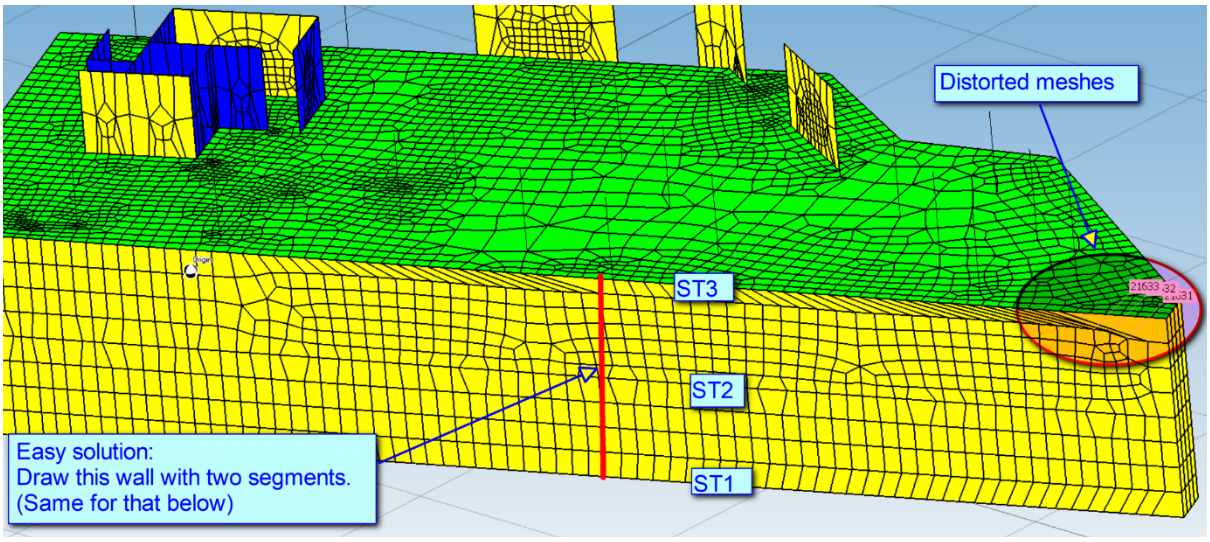
The user can improve meshing quality by controlling the number of meshes to be generated in walls, at the intersections with slabs.

> ⚠ Controlling the length and height of the meshes in walls is fundamental in case the software is unable to calculate the structure. If the run fails and displays the error "Unstable Structure", the user should check first for the presence of distorted meshes with the help of the tools available in the 3D viewer module. It is possible to fix this error by modifying the walls meshes. First attempt is to reduce or increase the mesh height. Second, slightly varying the "s" factor in the stories where the distorted meshes are detected. If the outcome of this iterative process is not satisfying, it is recommended to divide the wall into two segments or more instead of one single line.

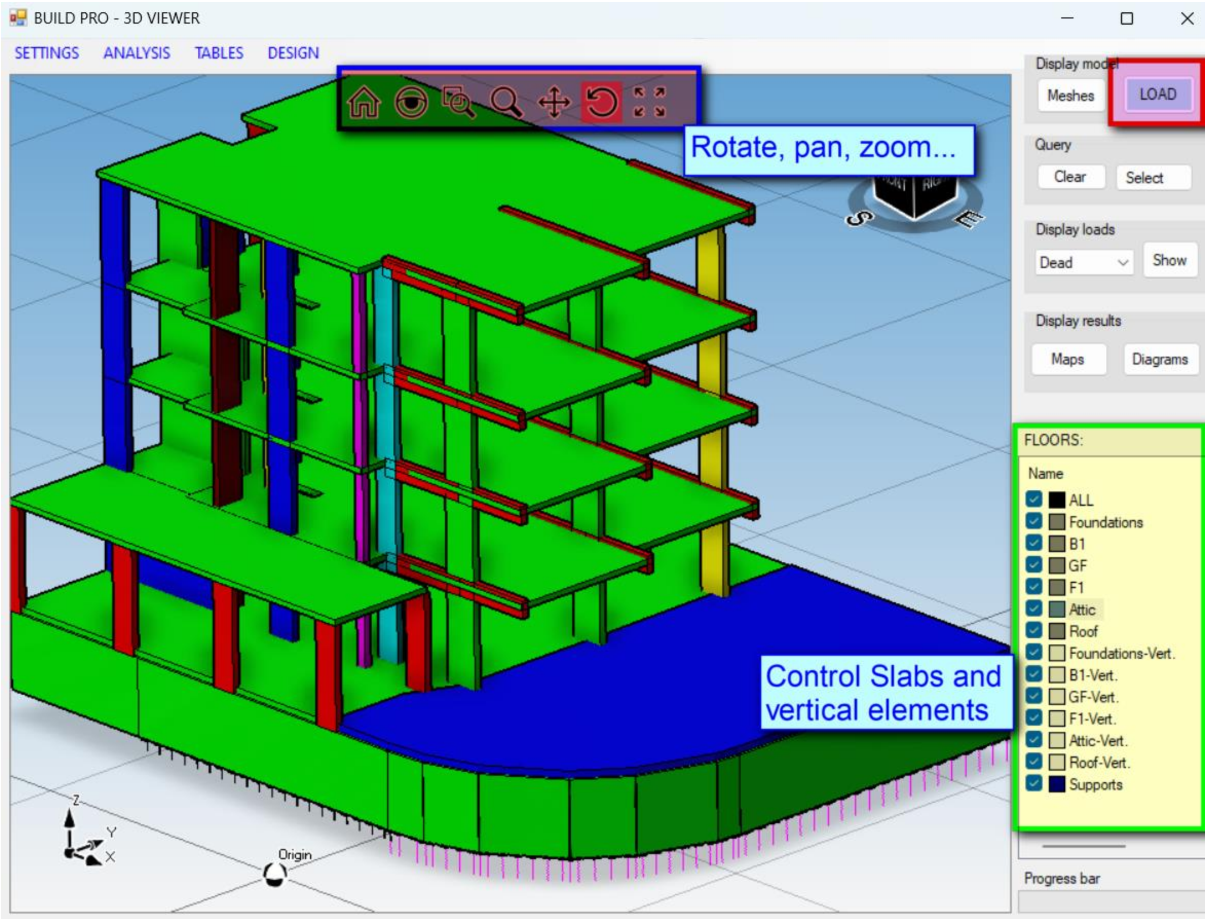
BUILD-PRO can handle complex structures. Slabs are always meshed prior to walls. When analysis fails because of incoherent meshes, it's possible to modify wall meshing and/or geometry.

In this example (here-below), the analysis failed because of distorted meshes detected in a wall that belongs to floor ST3 (the building has a raft foundation in ST1). Due to the big length of this basement wall, the meshing engine was unable to generate enough nodes on its top side. One possible approach is to lower the "s" factor on this side. However, this isn't a straightforward fix, and it could lead to issues with other walls on the same floor, as this factor affects <u>all</u> walls in ST3.

Simpler and more efficient is to draw this wall -and that right below in ST2- with two segments, instead of using one single line. Having one segment from each side of the red line will ease the meshing operation and will eliminate both the distorted meshes and the "void" area within the circle.



STEP 11: The 3D Viewer: Menu “BUILDING”→ “Open 3D Viewer”



The 3D Viewer provides the following features:

- Display the structural 3D model, including the applied loads
- Visualize the meshed model
- Detect and show distorted meshes
- Acquire information about the selected members (columns, beams, walls and slabs)
- Perform linear and non-linear static, and dynamic analysis
- Control the graphical display with the Settings menu
- Display tabulated analysis results
- Display graphical analysis results, like diagrams and maps for bars and panels
- Reinforced concrete design for all structural elements according to ACI 318-19 metric

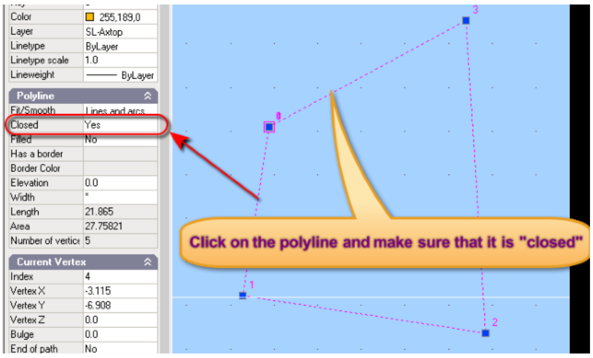
The 3D Viewer does not allow changes to the structure, such as adding or removing members, making geometric alterations, or adjusting loads. These functions are available in the BUILD-PRO 2D interface, which was described earlier in this document.

The “LOAD” button imports the model from BUILD-PRO once it is successfully converted to structure with the use of “Convert to Structure” command. If not, it is not possible to load it in the 3D Viewer. Any modification made to the model (adding or deleting members or floors, modifying sections or loads, modifying supports) is not reflected in BUILD-PRO unless the “Convert to Structure” command is triggered before loading again the structure with the “LOAD” button.

- It is possible to hide or display slabs and vertical elements with the checkboxes located at the right of the display. The "vert." text next to the name of the floors denotes the vertical members.
- The toolbox at the top of the screen contains classical graphical tools used to rotate, zoom, pan and execute other similar tasks.

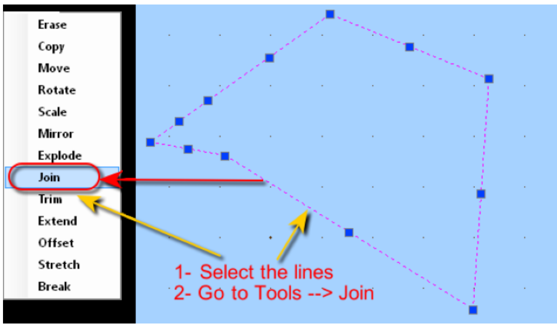
It is possible, during the modeling process in BUILD-PRO, to see in parallel the model in the 3D Viewer. However, as previously mentioned, the structure to be loaded should not have members with undefined properties. Sections of columns, walls, slabs and beams should be defined.

- Slabs, subslabs and openings should be closed polylines.



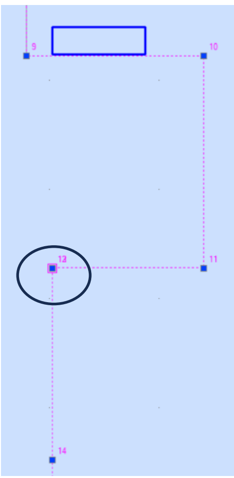
To be sure that the polyline is closed, select it and choose "yes" that is written next to the "Closed" option.

- To create a polyline, the procedure is illustrated as follows.



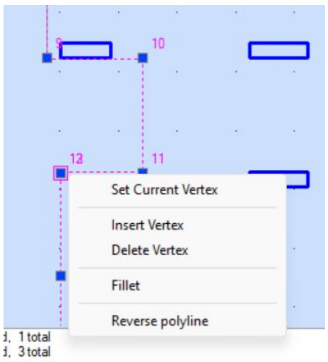
a) Draw the lines in the appropriate layer
b) Select the lines with the mouse.
c) Go to "Tools" → "Join"

- Polyline vertices are numbered in order, starting from 0 up to the last vertex. If two vertices overlap—often a result of repeated editing during drafting—the 3D Viewer will display an error upon loading the structure. Overlapping vertices need to be removed. You can spot them by visually inspecting their labels. Once the polyline is selected, vertices labels appear.



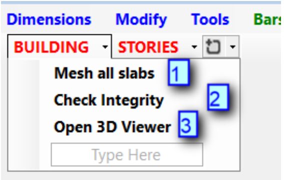
The figure at the left shows overlapped vertices of labels 12 and 13. The shown sequence of numbering is 9-10-11-12-13-14, where labels 12 and 13 are superposed.
Right-click directly on the vertex and select "Delete" from the window that appears, to resolve the issue.

It is possible to add a new vertex to modify the geometry of a polyline, which is of big help during drafting. The new vertex will be inserted between two consecutive vertices.

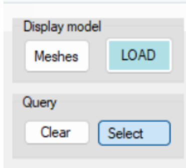


STEPS 12,13 and 14: The "BUILDING" menu

The "BUILDING" menu involves the final actions before running analysis.



First, mesh all slabs (step 1 in the left figure) based on the specified meshing parameters.
Second, integrity check should be performed. If the user just wishes to see the model in 3D, then no need to mesh slabs or to check integrity.



Integrity check is the last step before launching the 3D Viewer and loading the model with the "LOAD" button. Checking integrity performs the following tasks:

- It verifies the soundness of the full structure geometry based on internal algorithms.
- Detects implanted columns and walls and re-meshes the concerned slabs to generate nodes at their location.
- Meshes all walls in the model. The software performs this operation with respect to the nodes of the slabs meshes, which are located at the top and bottom sides of the wall.
- Consolidates all the specified attributes like supports and loads.

Mesh all slabs: Important note:

> If the geometry of a slab is modified (edges, openings, subslabs…), or beams and columns are added, the slab must be re-meshed either through the "Mesh Slab" command or by meshing all of them from the "BUILDING" menu. Once done, it is recommended to check integrity.

Check Integrity: Important note:

> Any modification to the geometry of the model (adding or deleting members) to the sections (modifying sections of columns, beams, walls or slabs) to the loads (adding, deleting or modifying loads) to the supports table (changing boundary conditions) to the meshing table (modifying mesh size) requires an integrity check. The changes will not be reflected in the 3D Viewer if the model is not re-loaded with the "LOAD" button.

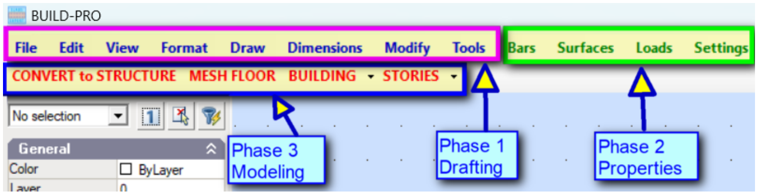
1.2 Main Features

- AutoCAD-like graphical interface allowing modeling with great ease.
- Direct import from AutoCAD of the 2D drawings of the floors.
- State-of-the-art meshing engine that generates refined quadrangular elements, offering to the user high precision results for moments, shears, etc…
- Analysis types: static, non-linear (second order and geometrical) modal and spectral.
- Automatic generation of combinations.
- The software performs structural integrity check and detects inconsistencies. It checks lines intersections, walls superposition, supports under columns and walls, preventing the user from running analysis and facing unexpected problems.
- Automatic incorporation of base shear scaling according to ASCE 7-16.
- Automatic integration of accidental torsional moments based on the position of the rigidity center toward the center of gravity, for each floor.
- Check for P-Delta requirements as per ASCE 7-16.
- Calculation of the floor drifts with limits verifications according to ASCE 7-16.
- Checking for torsional normal and extreme irregularities based on the limits of ASCE 7-16.
- High-quality 3D interface offering full control over the geometry.
- Determination of the sway/non-sway configuration of each floor as a prelude for the design of the columns.
- Detailed design of the columns according to ACI 318-19 with generation of the required and provided (real) longitudinal and transversal reinforcement. Automatic recognition of the clear height of the columns with calculations of the buckling length based on the stiffness of the slab. The critical buckling force (Euler) can be calculated according to all options provided in ACI, allowing more versatility in the design. Seismic dispositions are considered.
- Core walls can be formed by assigning the same core label to multiple walls. The software then computes the overall flexural moments, axial forces, and the shear forces for each wall within the core.
- Detailed design of walls and core walls of any complex configuration and any reinforcement arrangement. Generation of interaction diagrams and stresses maps. Design of the required horizontal reinforcement to resist shear forces.
- Design of slabs and mat foundations under ULS and SLS (crack width). Design for punching shear is based on a exclusive algorithm that handles all slab-opening geometries.
- Design of beams subject to flexure, axial forces, shear and torsion moments. Calculation of long-term and incremental deflections.
- Design of isolated footings under columns and walls, based on a unique hybrid formulation that combines analytical and finite elements calculations. Sizing and reinforcement consider both service and ultimate soil bearing capacities, backfilling over footing, height of the pier, etc…Safety factors for rotation and sliding are provided.
- Generation of design reports in Word format.
- Generation of high-quality shop drawings for beams, slabs (bending and shear reinforcement), walls, columns and foundations.

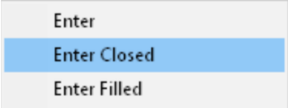
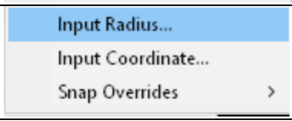
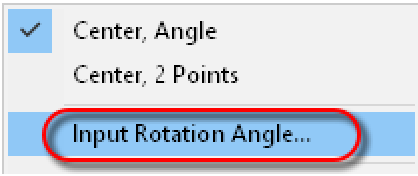
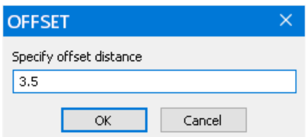
Chapter 2- Introducing the software

2.1 BUILD-PRO Interface

BUILD-PRO interface is organized into three distinct sections for drafting, affecting sections and loads, and finalizing the model to be ready for analysis. The related groups of menus are shown in this figure.

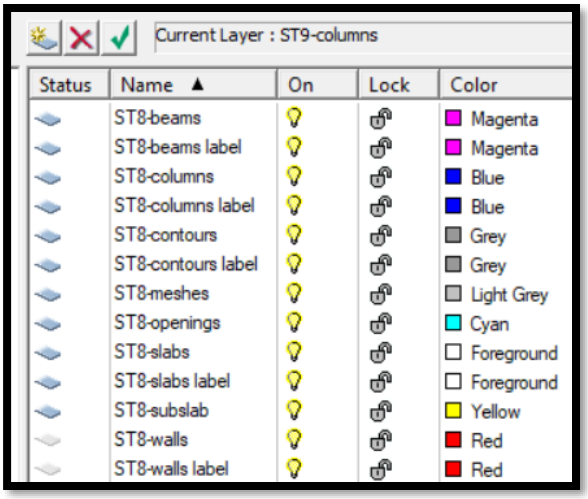


Drafting tools are user-friendly and straightforward. The table below lists the most used:

COMMAND LINE	DESCRIPTION
"L" → Line	1- Select the beginning point. 2- For the end point you can type "@3,0"
"PL" → Polyline	 Right-click and choose "Enter Closed" to close the polyline
"C" → Circle	 Right-click to input the radius. But before you have to select the center.
Rectangle: with the menu "Draw"→ "2 Points"	1- Select the beginning point 2- For the end point you can type "@3,2"
"C0" → Copy	1- Select the beginning point 2- For the end point you can type "@3,2"
"M: → Move	1- Select the beginning point 2- For the end point you can type "@3,2"
"RO" → Rotate	1- Select the object 2- Selection the center of rotation 3-  Input angle (Right click)
"T" → Draw text	Follow the instructions in the displayed window
"TR" → Trim	1- Select the cutting edge 2- Right-Click and select the part to cut
"EX" → Extend	1- Select the boundary edge 2- Right-Click and select the line to be extended
"O" → Offset	1- Select the line to offset  2- Right-Click and select "distance" 3- Click on the screen for offset direction.

Using these commands, it is possible to start from scratch a model. It's more common, however, to import from AutoCAD an already prepared drawing using menu "FILE"→"Open DWG".

Understanding layers


Current Layer : ST9-columns

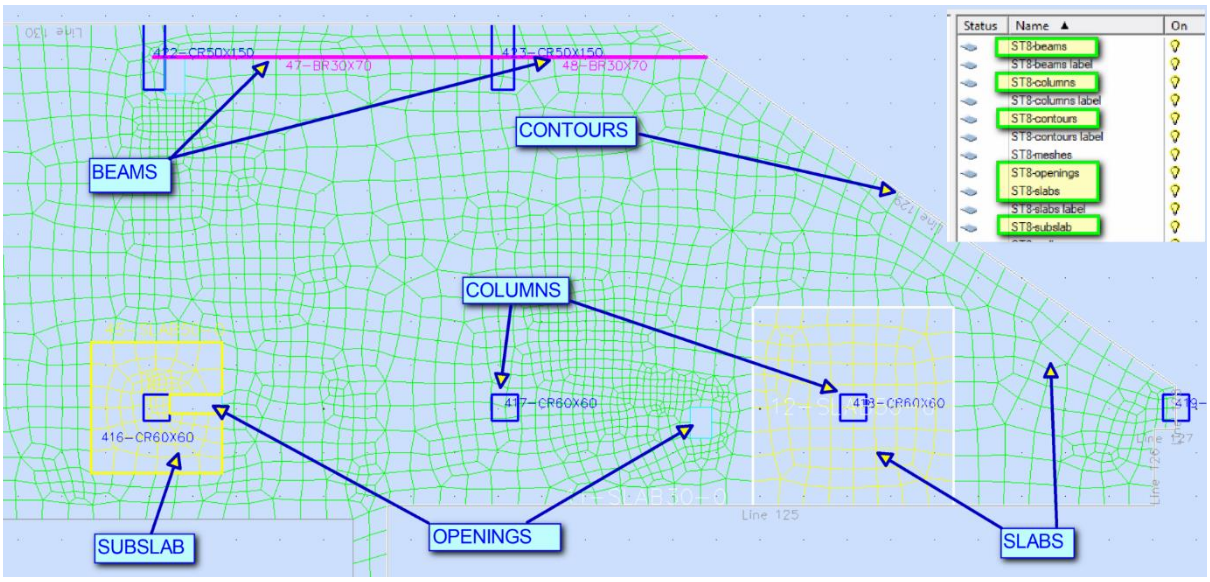
Status	Name ▲	On	Lock	Color
	ST8-beams			Magenta
	ST8-beams label			Magenta
	ST8-columns			Blue
	ST8-columns label			Blue
	ST8-contours			Grey
	ST8-contours label			Grey
	ST8-meshes			Light Grey
	ST8-openings			Cyan
	ST8-slabs			Foreground
	ST8-slabs label			Foreground
	ST8-subslab			Yellow
	ST8-walls			Red
	ST8-walls label			Red

Once a story is created by the stories manager, the software generates the corresponding layers. Menu “FORMAT”→”Layers” displays all the layers present in the model. For instance, this figure shows the layers of story ST8. Structural elements should be located inside layers of beams, columns, slabs, subslab, openings and contours. Layers ending with “label” contains only the names of the structural elements. It is possible to turn on/off any layer. If the name of a structural element is moved or deleted, that does not affect the element itself. Labels are always automatically generated by the software and cannot be modified by the user. To delete a member, it is just required to erase it from the model. Its label will consequently disappear.

If any members (such as columns or beams) are moved from their original layers to a user-created layer, they become inactive and they are not considered as structural by the software. This approach can be used when tracking an error in the model.

This sample of a floor (namely ST8 as per the layers names) shows a slab configuration with columns and walls below.

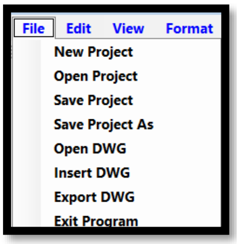
- Subslab is totally included inside the slab. It corresponds to a drop panel.
- The drop panel on the right is modeled as slab because it has a free edge.
- 2 beams segments are introduced from center-to-center of columns.
- Lines in gray color, at the edges of the slab, are intended to receive linear loads.
- Labels of all members belong to layers that end with the “label” text. These layers are automatically generated and filled with the labels of the members



After meshing a floor using the "Mesh Floor" menu, colored quadrangular elements appear in the layer named "STx-meshes". If the user selects one or more meshes and perform any drafting operation like delete, stretch, etc… this does not have any impact on the meshed plan.

Meshes are shown for information only and it is not possible to adjust them. Once the “Convert To Structure” command is triggered, all meshes are deleted.

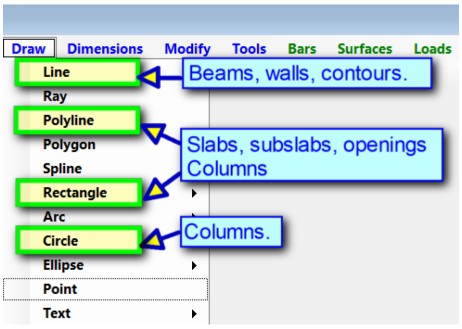
The File menu



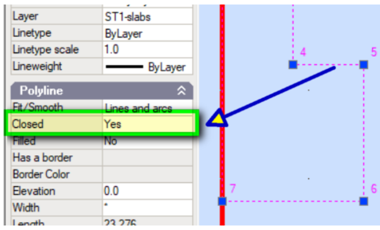
As previously mentioned, the easiest way to start a project consists of importing an already prepared AutoCAD drawing with the menu “Open DWG”. Once the project is saved with “Save Project”, a new folder is created where numerous proprietary files are generated. A BUILD-PRO project is a folder, not one single file. Command “Export DWG” converts back the model to an AutoCAD drawing.

It is possible to start a project from scratch with the command “New Project” which opens a new folder. Command “Open Project” requests the user to select the folder of an existing project.

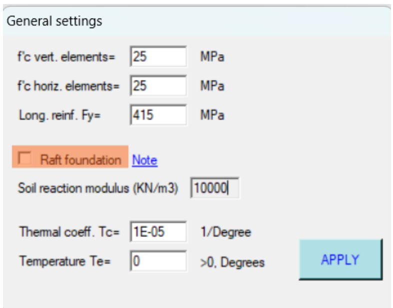
Four types of objects are available to model structural members. Lines are used to model beams, walls and contours. Slabs, subslabs and openings are always closed polylines. Columns are either rectangular or circular. If drawn with polylines, the quadrangle should have 90º corners.



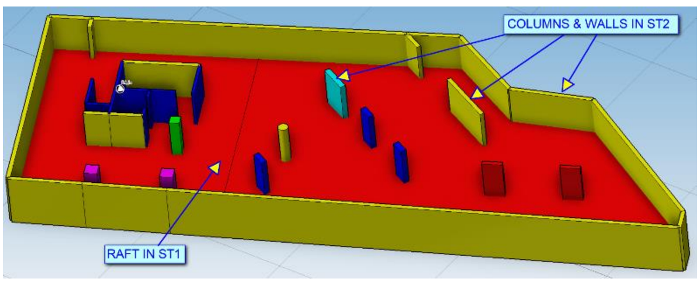
The remaining objects (polygon, arc, text, etc…) are not recognized as structural members. The user can use them in separate layers, to eventually annotate the drawing. The circle object is to be used for columns only. Polylines consist solely of segments; arcs are excluded. You can create a polyline by drawing the slab’s edges with lines and joining them using the “Join” command in the “Modify” menu. As shown in the figure at the right, the created polyline should be closed. Once the polyline is selected, the inspector window displays its closure state.

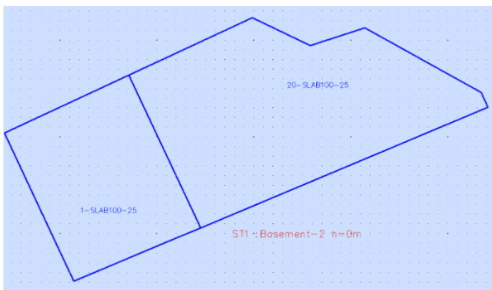


The Settings menu: General Settings



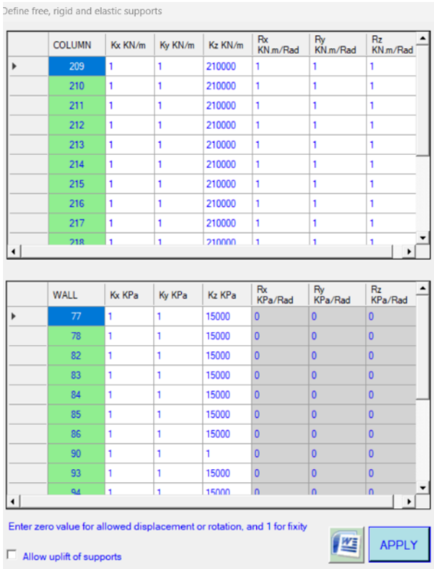
Once the project is saved, the user should open from the “SETTINGS” menu the General Settings and specify the foundations type. Before creating the first story -as depicted in STEP 5 of chapter 1.1 of the present document- it is important to indicate whether the foundations consist of mats or isolated footings. In the first case the height of ST1 should be set to zero and the slabs in this floor are rafts on elastic supports. It is not possible to draw columns and walls in this floor since these belong to ST2. In the second case, ST1 is a classical floor where slabs are suspended on the vertical elements that belong to the same story.





These figures illustrate the raft foundations case. Only the slabs are modeled in ST1. Columns and walls are in ST2.

The Settings menu: Define supports



Refer to STEP 10 (chapter 1.1) of the present document. Once the first and second floors are created, the software can automatically assign supports for columns and walls. If these are located over mat slabs, their labels won't appear in the table. In the case of a general raft, the supports table will be empty.

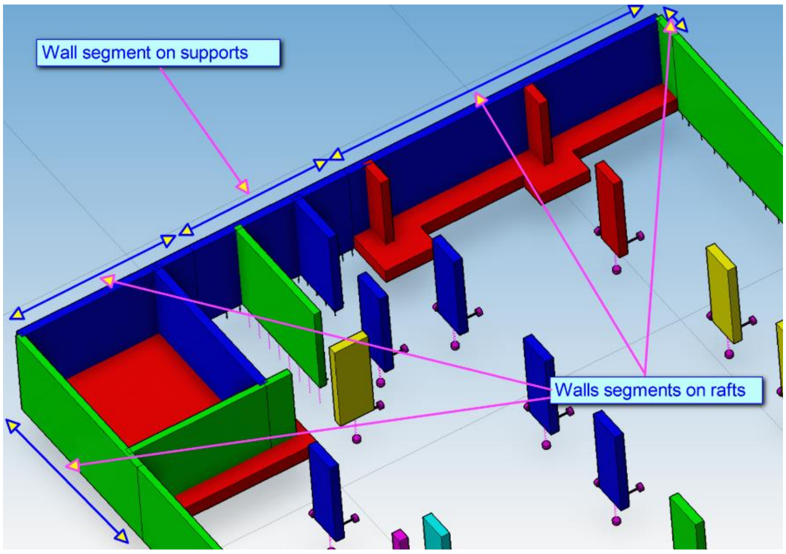
- Value “1” corresponds to fixed Degree Of Freedom.
- Value "0" releases the correspond DOF.
- Walls base is always free to rotate about its weak inertia
- Any value other than “0” or “1” is interpreted as stiffness

If uplift is allowed (checkbox at the bottom of the window), the program performs an iterative non-linear analysis where supports subject to tension are virtually eliminated. This works as well for mat foundations on soil and over piles.

⚠ Non-linear analysis is performed for combinations only. Load cases are not calculated when uplift is allowed, as this might lead to instabilities. For instance, wind load cases are inevitably unstable due to the absence of gravity loads. Only combinations results are available when uplift is activated.

⚠ When columns or walls are deleted or added to the model, the supports are not directly updated. The user should first “convert to structure” and then open the “define supports” window. Once the “Apply” button is clicked, the supports are re-assigned according to the new geometry.

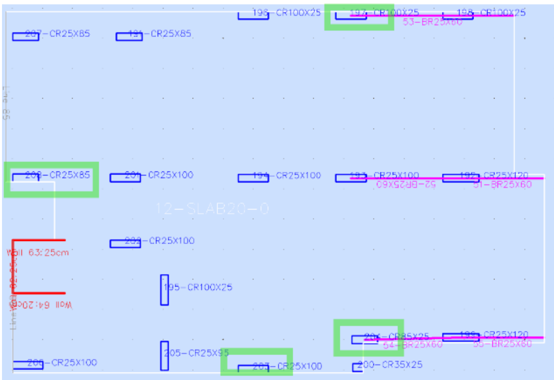
In the case of partial rafts, it is important for walls to be entirely located either over the mat slabs or outside them. This will help the software to correctly recognize the walls under which linear supports will be generated. In this figure, the peripheral wall in blue color is introduced by three segments; the intermediate one will receive linear supports.



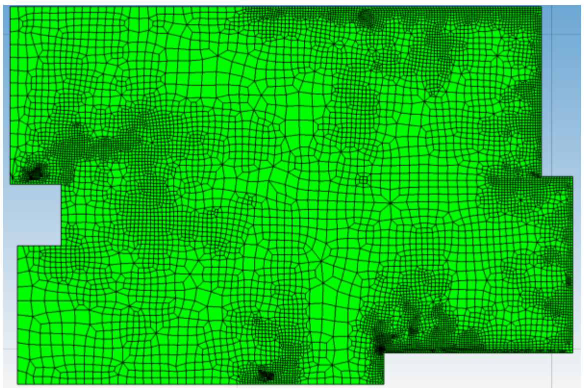
The Settings menu: Meshing parameters

Refer to STEP 10 (chapter 1.1) of the present document where meshing parameters are discussed.

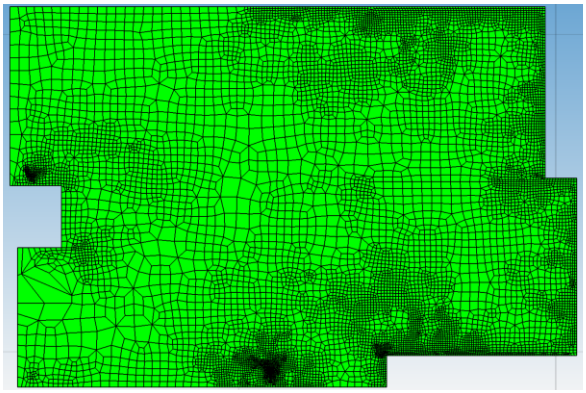
This window controls the size -and hence the distribution- of meshes in slabs and walls. Slab meshes are sized by specifying the average mesh size, whereas wall meshes are determined by their height and horizontal dimension. These parameters control the meshing when generated with the command “Mesh all slabs” from the “Building” menu, and when integrity check is performed, since this latter encompasses walls meshing.



Let’s consider the slab shown in this figure. Two meshing trials are considered, with mesh sizes 0.6m and 1m. The four areas that are highlighted in green correspond to the concentration of tiny meshes, consequent to the small distance between column’s center and slab edges. When the slab mesh size is increased to 1m, bigger meshes are generated in the middle of the slab, but more meshes are observed near the concentration zones. Regardless of the mesh size, mesh concentration at the designated areas cannot be avoided unless the slab's geometry is modified. Increasing mesh size for slabs does not necessarily lead to a smaller total number of generated meshes, in presence of mesh concentration. The software will have to generate more transition meshes between the 1m size and the smallest ones in order to respect coherence criteria’s.



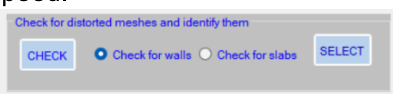
Average mesh size for slab: 0.6m



Average mesh size for slab: 1m

Walls in floor “ST3” (for instance) are located below slabs that belong to this floor. Since walls meshing is generated after that of slabs is executed, the distribution of walls meshes is controlled by the nodes at the bottom and top sides of the wall, which belong to floors “ST2” and “ST3”.
There is no need to modify walls meshes but in the flowing cases:

- ➢ Reduce the total number of meshes to increase calculation speed.
- ➢ Correct instability error encountered during analysis. In such case, check for distorted meshes from the “ANALYSIS” menu in the 3D Viewer. If such meshes are detected and belong to walls, then it is recommended to try first changing the height of the meshes in the floor concerned, before modifying their horizontal dimension.

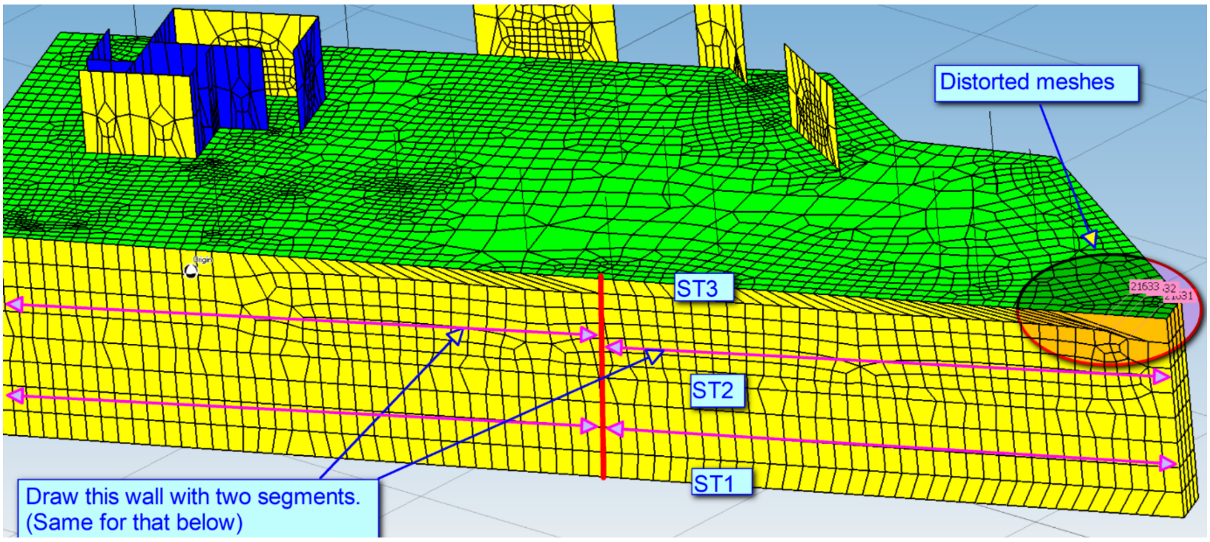


When analysis fails with instability error message, the user should check for distorted meshes. If such meshes are detected in walls, then modifying the height dimensions of wall's meshes might solve the issue. If the instability error persists with the presence of incoherent meshes, use bottom and/or top side coefficient "s".

The coefficient "s" (0.5<s<1.5) is per default equal to 1. It controls the meshing distribution at the bottom sides and top sides of all the walls that belong to the floor in question (columns "Walls-bot" and "Walls-top" in the table), as follows:

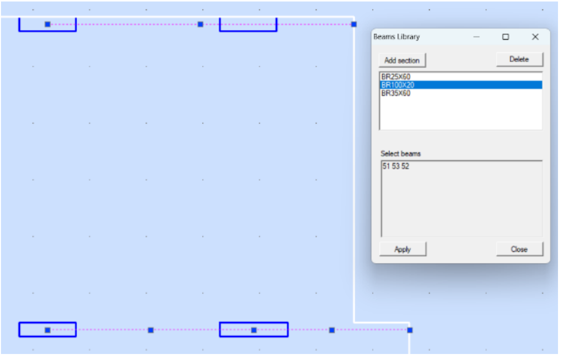
- If s<1 then the mesh length decreases and the number of meshes along the wall increases
- If s>1 then the mesh length increases and the number of meshes along the wall decreases

The figure displays distorted meshes in a wall within floor ST3. If an instability error occurs during analysis, these meshes are likely the cause. The use can introduce a smaller vertical size for the meshes, or a lower value for the coefficient "s" at the top side (ex.: s=0.9). Both methods might lead to regular meshing. However, a more expeditive way consists of dividing this wall into two segments, which will simplify the task of the meshing engine.



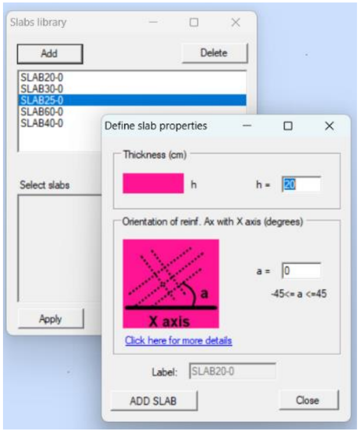
Instability message during analysis is not necessarily caused by incoherent meshes. If these are detected and fixed whereas run still fails, there might be isolated members not attached to the structure, like columns, walls, or beams. It is recommended then to consider an iterative approach based on the suppression of the suspected members or stories.

Assigning beams sections



The "BARS" menu option, "Create/Assign beams," launches a window enabling users to define beam sections and assign them to lines within the "STx-beams" layers. First the section should be highlighted from the upper part of the window. Second, it is possible to graphically select the beams lines with a window pointer. Once their numbers appear in the field, click the “Apply” button. The software filters only beams, even if other elements are selected.

Assigning slabs sections

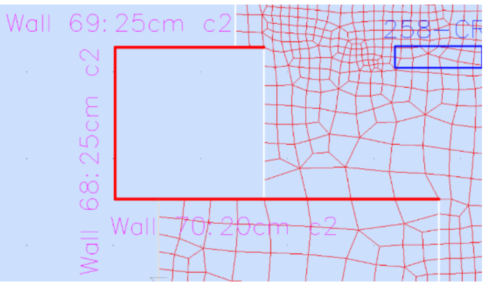


The process of creating and assigning slab sections follows a methodology comparable to that used for beams. When defining a new slab, two parameters must be specified: thickness and the angle between the reinforcement "Ax" and the horizontal "X" axis. The slab reinforcement comprises orthogonal meshes, "Ax" and "Ay," positioned at both the top and bottom faces. Although "Ax" typically corresponds to reinforcement parallel to the "X" direction, certain slab configurations may necessitate an inclined reinforcement mesh. Therefore, users are able to designate the angle "a" that "Ax" forms with the "X" axis. When viewing reinforcement maps, it is important to note that the "Ax" map displays reinforcement aligned according to the specified angle "a" (in degrees), while "Ay" consistently remains perpendicular to "Ax".

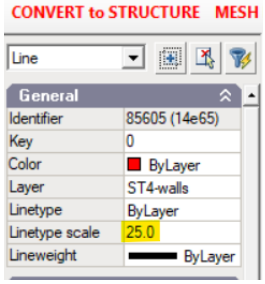
Assigning columns sections

Definition of columns is straight-forward. Rectangular and circular columns are automatically recognized by the software. Use the command “Rectangle” for rectangular columns or draw a closed polyline. The risk of this last option is that the sides of the closed polygon should be 100% perpendicular to each other’s, to make a perfect rectangle. The software identifies the rectangles in layer “STx-columns” and labels them with their dimensions. Once the “Convert To Structure” command is initiated, the labels of the columns should appear on the screen.

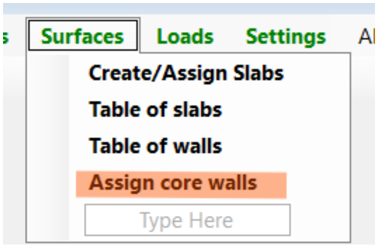
Assigning walls sections

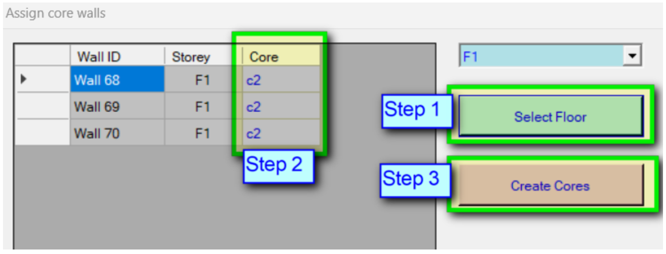


Walls are line objects in layers “STx-walls”. Once the drawing is converted to structure, the user can assign the width by selecting the wall(s) and introducing the value in the “Linetype scale” field (refer to the figure at the right).



It is possible to group walls into a core, by assigning to them the name of the core to which they belong. Menu “Surfaces”→”Assign to core walls” opens a table that displays all the walls that belong to the selected floor.





Once the name of the core is assigned to the concerned walls, the button “Create Cores” completes the operation. A story might comprise several cores. The name of one core should be unique for the whole building. For instance, if a story comprises two cores named C1 and C2, the same cores in the floor above (or below) should be named C1A and C2B or any other names but not C1 and C2. In the figure that shows an extract of the slab (here-above), one can notice that next to the name of each wall there is a mention of the core to which it belongs, ex: “Wall 69:25cm c2”.

Assigning cores does not have any impact on the analysis. The software will provide additional results which are the global axial forces and moments, applied in the centroid of the core, calculated by integrating the local forces in the walls. These forces are required for the reinforced concrete design of the core, performed in the 3D Viewer module.

Lines in “STx-contours” layers

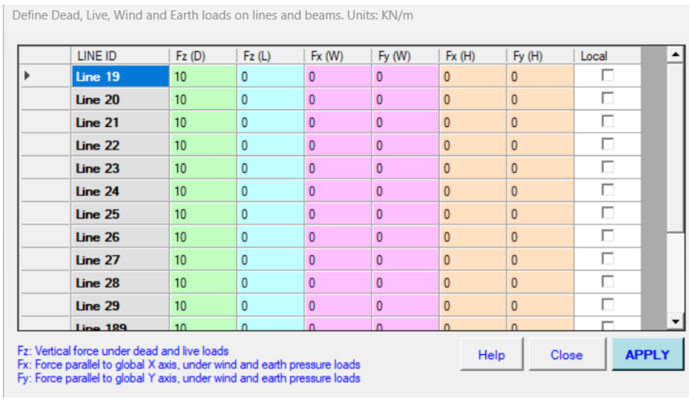
These lines are intended to receive linear loads within the slabs and along their edges, since linear loads cannot be applied directly to slab edges. Once the drawing is converted to structure, the lines are numbered.

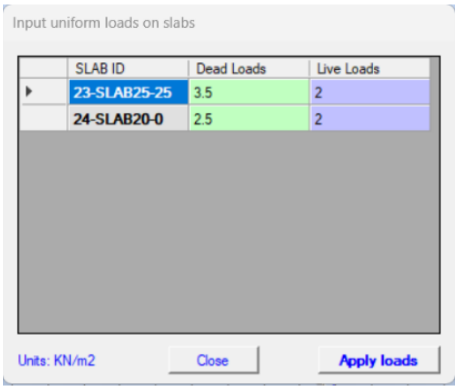
The Loads menu

Loads are fully addressed in STEP 10 of the present document.

Before introducing loads, the sections of the structural members should be defined.

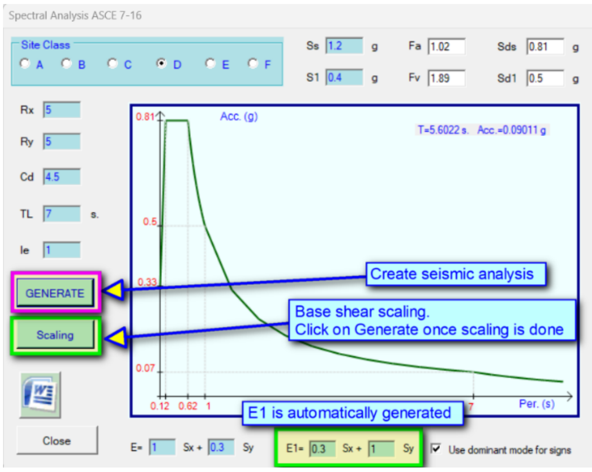
- It is possible to introduce uniform loads only, on lines (beams and lines that belong to "STx-contours" layers) and on slabs.
- Gravity and wind load cases are shown in these figures. It is not possible to add load cases. These are predefined in the software, with the labels D,L,W,H. W and H are lateral (horizontal) loads introduced in the global or local X and Y directions. D and L are gravity (vertical) loads introduced in the Z direction.
- Loads on slabs are always gravity loads (vertical direction), related to dead and live cases.





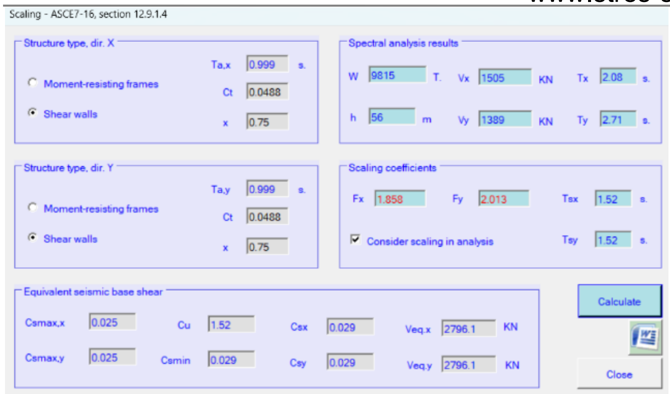
Typical load cases for lines (at the left) and slabs (right figure)

- Seismic load cases (named "E" and "E1") are generated from the menu "LOADS"→ "Seismic loads". These are linear combinations of the seismic forces Sx and Sy, in the global X and Y directions, as per ASCE 7-16. If quadratic combination is selected, only "E" is generated.
    - E=Sx+0.3*Sy or, if quadratic combination is selected: E=$\sqrt{S_x^2 + S_y^2}$
    - E1=0.3*Sx+Sy
- It is not possible to manually define seismic loads. These are created by the software, following the spectral analysis parameters.
- Modal analysis is defined in the 3D Viewer.



Spectral accelerations are oriented along X and Y axes. Once applied to the modes and quadratic combinations are generated, we obtain the seismic forces Sx and Sy which correspond to the global directions of the earthquake. These should be combined via linear (or quadratic) superposition, since the principal bending directions that correspond to the main modes are not necessarily oriented along X and Y. The user can introduce the factors of Sx and Sy for the "E" load case; "E1" is directly generated with switched factors.

Once scaling is performed, the "GENERATE" button must be clicked. Activate "Use dominant mode for signs" for a more realistic computation of in-plane axial forces and moments in shear walls. The program affects signs for the seismic forces -instead of being all positive- based on the sign they had in the corresponding dominant mode.

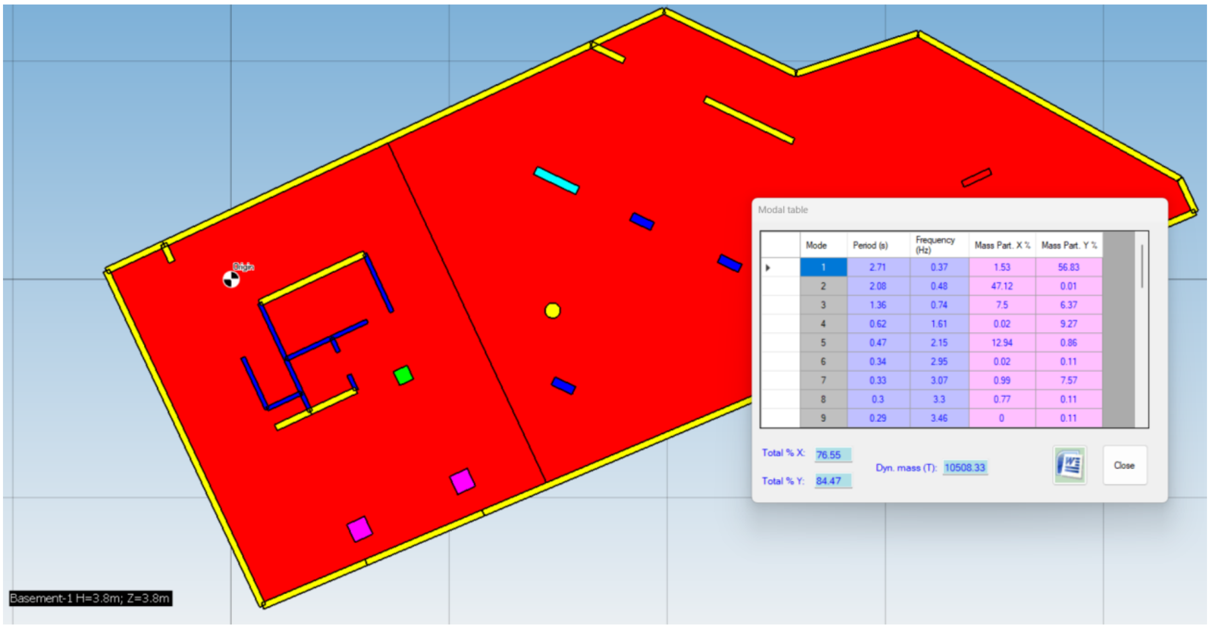


The purpose of scaling is to amplify the dynamic forces obtained from spectral analysis, based on equating dynamic and static base shears. Static base shear can be calculated by the software, once the fields in blue color are filled. Fx and Fy amplify Sx and Sy; the user can modify these factors.

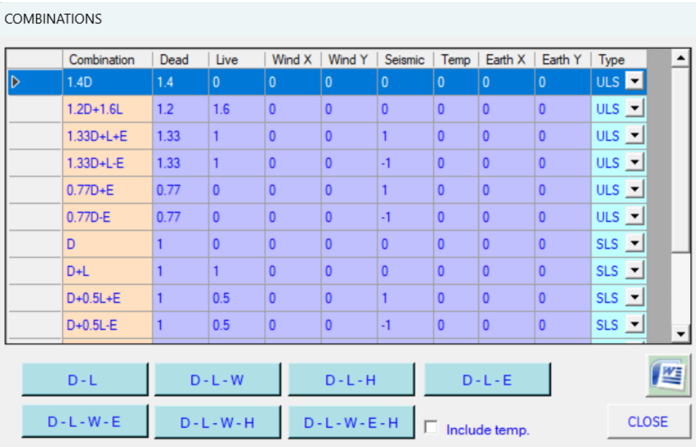
For scaling to be efficient, it is recommended to rotate the structure about Z axis, to align its principal modes with X and Y axes. Practically, it is equivalent to attain an orientation for which the first (or second) mode has the biggest mass participation in X and a minimal one in Y (or vice-versa).

Any modification to dead, live, wind and earth loads requires integrity check. The 3D Viewer does not refresh by itself. Modifications to seismic loads do not require integrity check.

This top view of a structure shows an inclined orientation (on purpose) with regards to the (X,Y) gird system. It might appear weird at first sight, but the first and second modes which have the biggest mass participation factors, are well oriented with regards to the introduced spectral accelerations. The resulting seismic forces in the members are hence more critical, and the scaling is more appropriate.



Combinations of load cases


COMBINATIONS

Combination	Dead	Live	Wind X	Wind Y	Seismic	Temp	Earth X	Earth Y	Type
1.4D	1.4	0	0	0	0	0	0	0	ULS
1.2D+1.6L	1.2	1.6	0	0	0	0	0	0	ULS
1.33D+L+E	1.33	1	0	0	1	0	0	0	ULS
1.33D+L-E	1.33	1	0	0	-1	0	0	0	ULS
0.77D+E	0.77	0	0	0	1	0	0	0	ULS
0.77D-E	0.77	0	0	0	-1	0	0	0	ULS
D	1	0	0	0	0	0	0	0	SLS
D+L	1	1	0	0	0	0	0	0	SLS
D+0.5L+E	1	0.5	0	0	1	0	0	0	SLS
D+0.5L-E	1	0.5	0	0	-1	0	0	0	SLS

Combinations are generated from the predefined load cases D,L,W,H,T and E. The last three ones are respectively earth, temperature and seismic. When introducing combination names, it is recommended to respect the labels of the load cases; this will help the software to identify combinations that include wind and seismic cases.
Earthquake combinations should include "E" in their names when referring to seismic case, as shown in the table. The software will automatically generate identical combinations that include "E1" case. It is possible to generate predefined combinations with the use of the buttons located under the table; these follow ASCE 7-16 guidelines.

⚠ If drift combinations are created (i.e. Cd*E) where Cd is the deflection amplification factor, it is recommended to assign them as SLS and not ULS. In reinforced concrete modules, the user can select the envelope of ULS combinations. If those intended for drift are classified as ULS, the software will consider them in the design, which is incorrect.

When the checkbox "Include temperature" is activated, the generated combinations incorporate the case "T" that denotes thermal loads. Thermal loads are introduced in the menu "SETTINGS"→ General settings window. A positive value should be entered, corresponding to temperature variation. The corresponding expansion strains will be applied to all the structural elements. Retraction strains (negative variation of the temperature) are considered in the combinations, since the load case "T" appears with factors (+1) and (-1).

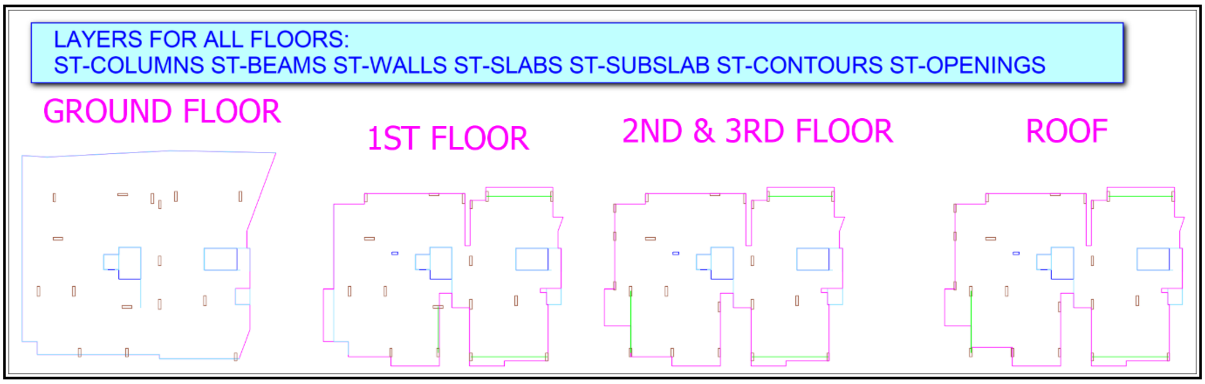
Self-weight of the structure (beams, columns, slabs and walls) is always generated by the software. It is embedded in the combinations within the load case "D", i.e. added to the introduced dead loads. For instance, combination "D" shown in the table above contains the self-weight of the structure and the introduced dead loads by the user.

The user can add and remove combinations. It is important to respect three rules related to the combination's name:

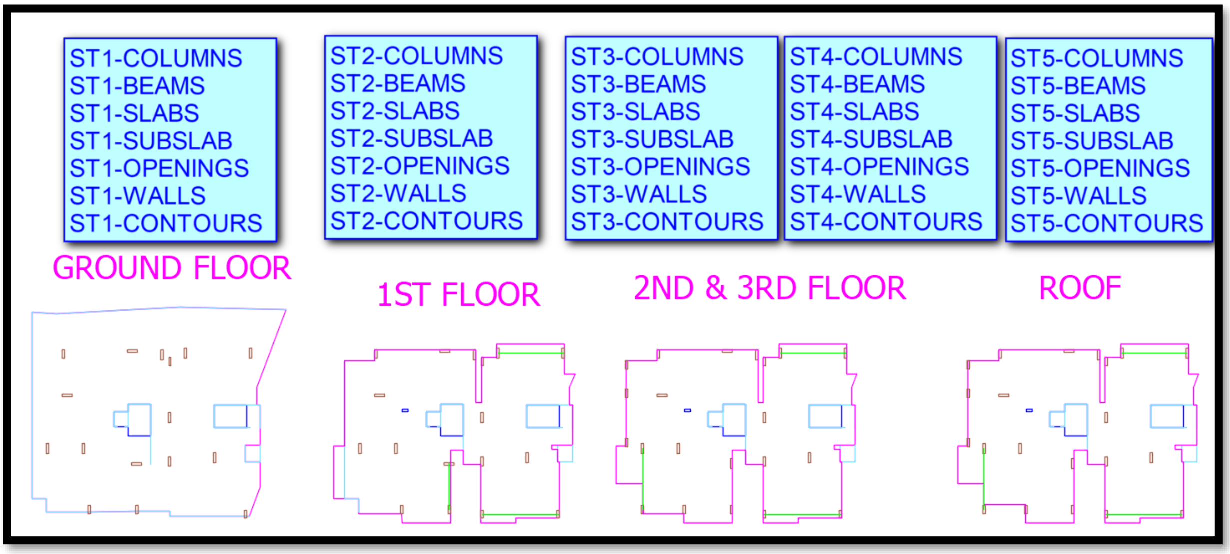
1. Combination's name should not contain spaces.
2. Combination's name should reflect the involved load cases, as shown in the table above.
3. The labels of the load cases are: "D,L,Wx,Wy,E,Hx,Hy". When load cases are present in the combination, their labels should also be present in the name of the combination. The program identifies wind and seismic loads from their labels in the combination's name.

Convert To Structure

As described in STEP 9 (chapter 1.1 of the present document), the “Convert To Structure” translates the lines and polylines to structural elements identified by their layers. Let’s consider the preparation drawing done on AutoCAD, imported into BUILD-PRO. For this five-floor building, all lines and polylines -for all floors- belong to these seven layers listed here-below. For BUILD-PRO, these lines are still drafting objects, not structural members. Make sure the titles of the floors (if they exist) are not included by mistake inside these layers. The drawing should comprise all floors of the building. If there are several identical floors, only one needs to be included.



Each time a new story is created with the “Stories Manager”, the related layers are generated. The user should change the object’s layers from their initial one to that corresponding to the type of the member. For instance, if we consider columns: “ST-COLUMNS” to “ST1-COLUMNS”. Once this procedure is applied to all the members of the created floor, the “Convert To Structure” command should be prompted. At this moment it becomes possible to affect the desired sections to the members, since the software recognizes the drafting objects as structural elements.

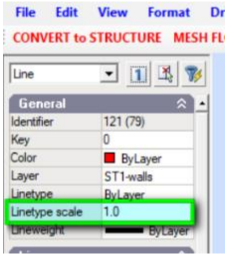


The “Convert to Structure” command transforms the lines and polylines into structural elements. In addition, it verifies the integrity of the floor and might trigger error messages if singularities in the model are detected. This verification and the related messages are done to all floors that are created with the stories manager window. In other words, it will apply to the entire building if all stories are created. That’s why it is strongly recommended to use it floor by floor to ease the detection of errors and confine them to a single floor. Do not create a new floor if the actual one is not converted to structure.

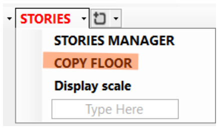
Once the lowest floor ST1 is converted to structure, and all potential errors resolved, it is possible to create a new floor, assign the objects from their original layers “ST-…” to their new corresponding layers “ST2-…”, and once again use the convert to structure command. This cycle is to be repeated to the last floor.

⚠ Once converted to structure and all sections have been assigned to columns, walls, beams and slabs, it is recommended to mesh the floor. This will guarantee the correctness and convenience of the concerned slab framing. If meshing fails, see the error management chapter in this document.

- Columns sections are automatically recognized, based on the dimensions of the introduced rectangular and circular shapes.
- To assign beams sections, select menu Bars → Create & assign beams.
- To assign slabs thicknesses, select menu Surfaces → Create & assign slabs.
- To assign walls widths, graphically select the walls and enter in the “linetype scale” field the width in centimeters. The default value 1.0 indicates that the width of the selected wall is not assigned yet. Refer to the figure at the right.



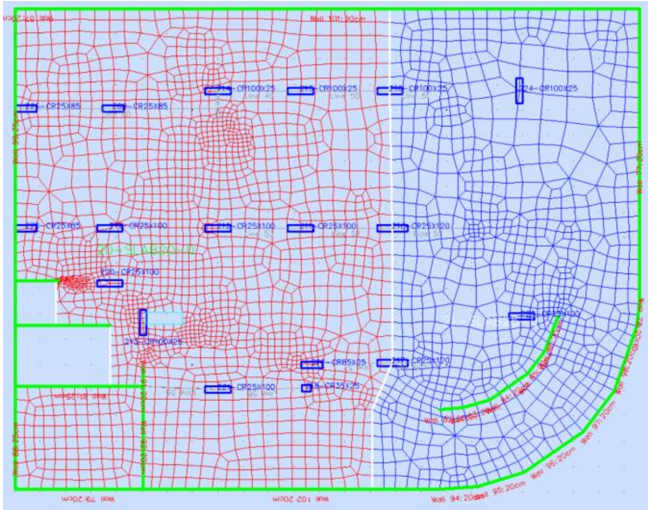
⚠ In the case of identical typical floors, it is possible to use the “Copy Floor” command from the “STORIES” menu. All members -including the defined loads- will be copied to the floor above.



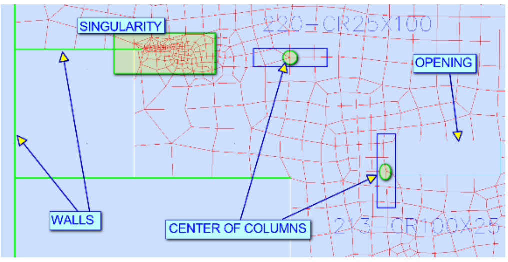
⚠ It is possible, during the modeling process, to open the 3D Viewer (menu “BUILDING”→”Open 3D Viewer” and then load the structure with the “LOAD” button. A three-dimensional view of the building appears. For more information, you can refer to “STEP 11” in chapter 1.1 of the present document. It is not possible to modify the geometry of the structure in the 3D Viewer. Additionally, changes made to the structure are not automatically updated in the viewer. The user should re-load the structure with the “LOAD” button to update the 3D view.

Meshing the slabs

Command “Mesh Floor” generates quadrangular meshes inside slabs and subslabs, within layers “STx-meshes”. During the modeling phase, once a floor is successfully converted to structure and all sections are assigned, it is advisable to mesh the slab with the intention to verify whether geometrical issues prevent this operation. The most common reasons for errors are the presence of nodes close to each other’s, narrow gaps between lines, polylines not closed… In general, it is recommended to simplify the geometry of the slab by following the recommendations of STEP 1 in chapter 1.1 of the present document.



Typical meshing of a slab

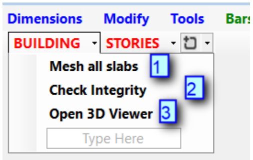


This figure is a close view of the slab at the location of the mesh concentration area. Mesh concentration is the consequence of high number of tiny meshes the software had to generate to solve a geometrical singularity. In the present case, the wall extends inside the slab just 10cm. Consequently, the meshing engine progressively reduces the general mesh size to avoid incoherent meshes. This leads to an unnecessary number of small meshes as shown in the figure. The solution consists of either trimming the wall to the edge of the slab or extending it further within the slab. If such cases occur at other locations in the slab, the total number of generated meshes will be excessive and will affect calculations time, the size of the project and the time required to display the results.

The “Mesh Floor” command meshes slabs and subslabs. Walls are meshed during the “Check Integrity” process which is called from the “BUILDING” menu. The “Mesh Floor” command generates meshes based on an average mesh size of 0.6m, not changeable by the user.

The meshing parameters from the “SETTINGS” menu controls mesh size for slabs, only when they are meshed from the menu “BUILDING” → “Mesh all slabs”. This command is recommended to be used right before checking integrity and launching calculations.

Menu “BUILDING”→ ”Check Integrity” performs numerous tasks, among them meshing walls. Furthermore, in the presence of implanted columns and walls, the affected slabs will be re-meshed. Check Integrity is the last step before opening the 3D Viewer and loading the model.



Mesh all slabs

It is possible to mesh all slabs at once from the menu “BUILDING”→ “Mesh all slabs”. It is recommended to do so once all floors are converted to structure. If each floor is individually meshed with the command “Mesh Floor”, there is no need to re-mesh all of them. The purpose is to make sure all floors are meshed before proceeding to the integrity check.

Menu “SETTINGS”→ “Meshing parameters” provides a full control on the mesh sizes for both slabs and walls. In opposition to the individual floor mesh command “Mesh Floor” where the mesh size is fixed to 0.6m, the “Mesh all slabs” considers the specified mesh size from the table. That’s why it is important to visit the meshing parameters table before using the “Mesh all slabs” command.

Meshing of slabs and walls

	Floor ID	Floor Name	Floor height (m)	Slabs (m)	Walls-Bot (0.5<s<1.5)	Walls-Top (0.5<s<1.5)	Walls-Vert (m)
	ST1	Foundations	0	0.6	1	1	0
	ST2	B1	2.6	0.6	1	1	0.43
	ST3	GF	3.5	0.6	1	1	0.58
	ST4	F1	3.1	0.6	1	1	0.52
	ST5	Attic	3.1	0.6	1	1	0.52
▸	ST6	Roof	3.5	0.6	1	1	0.58

Mesh all slabs: Important note:

> If the geometry of a slab is modified (edges, openings, subslabs…), or beams and columns are added, the slab must be re-meshed either through the ”Mesh Slab” command or by meshing all of them from the “BUILDING” menu. Once done, it is recommended to check integrity.

Check Integrity

That’s the last step before opening the 3D Viewer to run analysis. It is accessible from the menu “BUILDING”→ “Check Integrity” right after making sure all slabs are meshed.

It performs the following tasks:

- Verifies the soundness of the full structure geometry based on internal algorithms.
- Detects implanted columns and walls and re-meshes the concerned slabs to generate nodes at their location.
- Meshes all walls in the model. The software performs this operation with respect to the nodes of the slabs meshes, which are located at the top and bottom sides of the wall.
- Assigns supports under columns and to the generated nodes at the base of walls.
- Assigns loads to the generated quadrangular finite elements of the slabs.

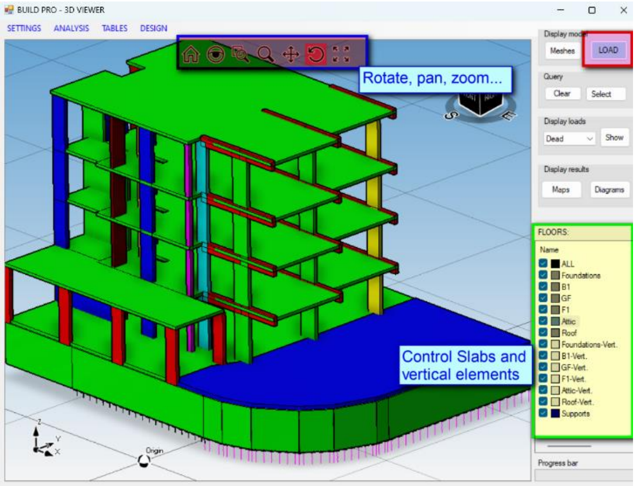
Check Integrity: Important note:

> Any modification to the geometry of the model (adding or deleting members) to the sections (modifying sections of columns, beams, walls or slabs) to the loads (adding, deleting or modifying loads) to the supports table (changing boundary conditions) to the meshing table (modifying mesh size) requires an integrity check. The changes will not be reflected in the 3D Viewer if the model is not re-loaded with the “LOAD” button.

2.2 3D VIewer Interface

The 3D Viewer is introduced in STEP 11, chapter 1.1 of the present document.

Accessed from the “BUILDING” menu, it offers a 3D view of the structure during modeling, performs analysis, and displays results.



The 3D Viewer does not allow changes to the structure, such as adding or removing members, making geometric alterations, or adjusting loads. These functions are available in the BUILD-PRO 2D interface.

The “LOAD” button imports the model from BUILD-PRO and refreshes the screen after seeing results or after displaying graphically the introduced loads.

The 3D Viewer provides the following features:

- Display the structural 3D model, including the applied loads
- Visualize the meshed model
- Detect and show distorted meshes
- Acquire information about the selected members (columns, beams, walls and slabs)
- Perform linear and non-linear static, and dynamic analysis
- Control the graphical display with the Settings menu
- Display tabulated analysis results
- Display graphical analysis results, like diagrams and maps for bars and panels
- Reinforced concrete design for all structural elements according to ACI 318-19 metric

➢ It is possible to hide or display slabs and vertical elements with the checkboxes located at the right of the display. The “vert.” text next to the name of the floors denotes the vertical members.

➢ The toolbox at the top of the screen contains classical graphical tools used to rotate, zoom, pan and execute other similar tasks.

It is possible, during the modeling process in BUILD-PRO, to see in parallel the model in the 3D Viewer. However, the structure to be loaded should not have members with undefined properties. Sections of columns, walls, slabs and beams should be defined prior to using the “LOAD” button in the 3D Viewer.

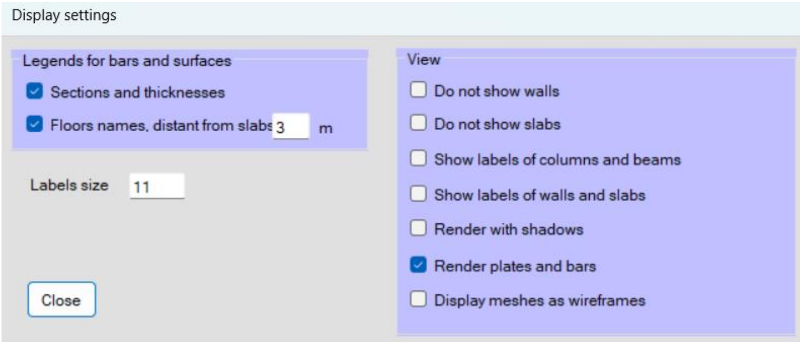
⚠ The “LOAD” button imports the model from BUILD-PRO only after successful conversion using “Convert to Structure.” Otherwise, it cannot be loaded in the 3D Viewer. Changes like editing members, floors, sections, loads, or supports will not update in BUILD-PRO unless you run “Convert to Structure” before loading the structure again.

⚠ It is not possible to run analysis or to graphically display the introduced supports under columns and walls if Integrity check is not performed. As previously explained, the “Check Integrity” command generates the numerical model to be ready for analysis.

2.2.1 The SETTINGS Menu

The “SETTINGS” menu opens a window that controls the graphical aspect of the model.

The selections take effect upon clicking on the “LOAD” button.

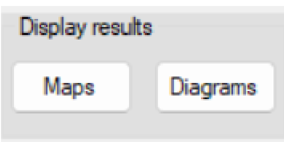


- "Sections and thicknesses" shows a color legend on the right, where each color matches a member section in the model. This lets users easily identify sections by color.
- “Floors names” displays the names and heights of the floors. It is possible to specify the distance of the text from the slabs, to improve visibility.
- “Labels size” controls the height of the text on the screen, including that for results.
- “Do not show walls” and “Do not show slabs” hide panels from the view. Useful when the user wants to display results on columns and beams only.
- “Show labels of columns and beams” displays the numbering of the members and their sections. Useful when reading tabulated results and building the analysis report.
- “Show labels of walls and slabs” displays the numbering of the panels and their thicknesses.
- “Render with shadow” improves the quality of the 3D view when the real shape of the members is displayed (refer to the following option).
- “Render plates and bars” provides a realistic view of bars and plates, by rendering their real size, instead of showing them with their centerlines.
- “Display meshes as wireframes” is effective when displaying the meshed model. The meshes are shown as contours, without interior filling.

Select members

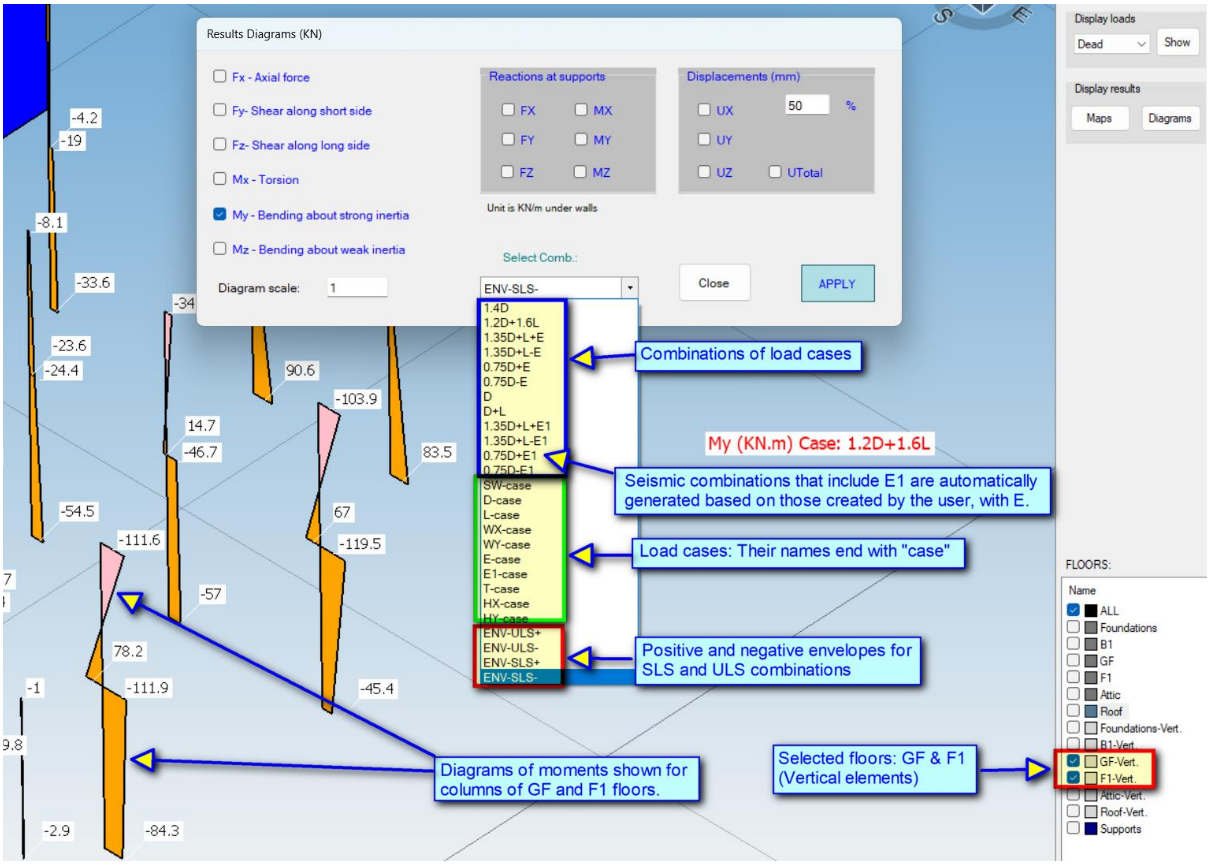
Once a member is selected, its properties are displayed. If the results are available and the selected members are beams or columns, then forces, moments and displacements are shown. It is not possible to modify the selected member.

Display results



Diagrams results are for beams and columns. Results for slabs and walls are shown as colored maps. The height of the results text is controlled with the “Labels size” field from the “SETTINGS” menu. The hide the displayed results and retrieve the original view, click on the “LOAD” button.

2.2.2 Results: Diagrams



To display results for columns, first the corresponding floors should be selected. Floors name's that end with "Vert." comprise vertical elements only. If the user wishes to display just columns, the "Do not show walls" option in the "SETTINGS" menu should be activated.

BUILD-PRO stands out for its simple results interpretation, with no node numbers or local systems for members to display:

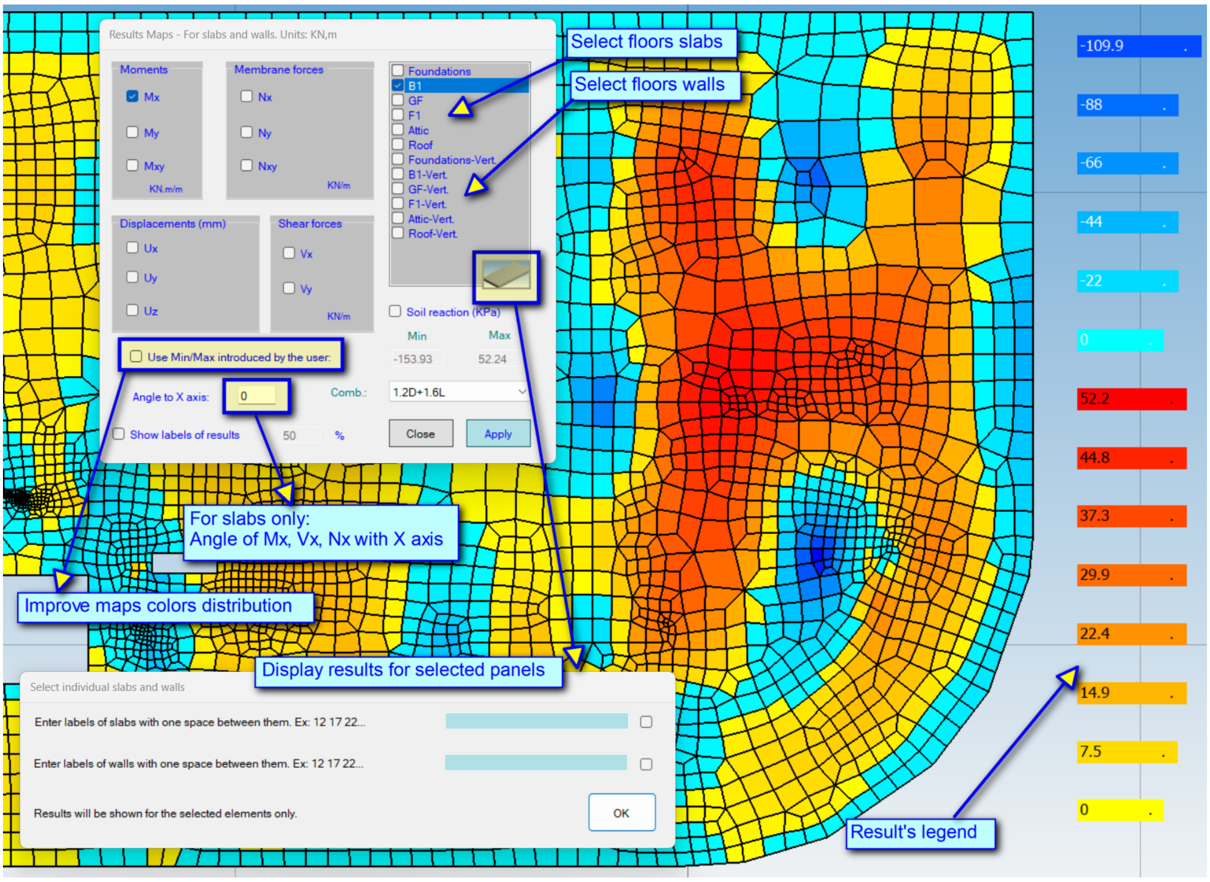
- For columns: Shear forces are along short and big sides; moments are about weak and strong inertias.
- For beams: Shear Fz and moment My are in the vertical plane, whereas Fy and Mz are in the horizontal plane of the slab.
- Positive axial force means compression.

It is possible to select a combination, a load case or an envelope of all ULS or SLS combinations. Load cases names end with "case". For instance, "D-case" corresponds to the dead loads of partitions, tiling and other components. "SW-case" comprises the self-weight of the structure (beams, columns, walls and slabs); it is automatically generated by the software.

> ⚠ All combinations that involve dead loads case "D" comprise implicitly the self-weight of the structure. To check the results for the individual cases of dead loads and self-weight, the user should select "SW-case" and "D-case".

Selecting any displacement checkbox (UX, UY, UZ, or UTotal) shows the same deformed structure. These options represent displacement in the X, Y, or Z directions and do not alter the displayed deflected shape. "UTotal" indicates the vector norm, showing the total deflection between two nodes. If a vibration mode is selected from the "Select Comb." drop menu, the mode shape is displayed without values, since a mode does not hold a seismic force.

2.2.3 Results: Maps



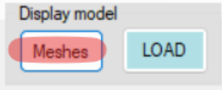
To display results for slabs and walls, first the corresponding floors should be selected from the right side of the window. Floors name's that end with "Vert." contain just walls. It is possible to select several floors. It is possible to refine selection by clicking on the button that has a panel image as background. A new window will open where the user can introduce the individual labels of the walls and slabs.

"Mx" represents bending moments parallel to the x-axis. The axis's orientation relative to global X is set in the "Angle to X axis" field; an angle of zero aligns Mx with X and My with Y. The angle, ranging from -45° to +45°, is specified in degrees. In some slab layouts, column's grids may not match the main (X,Y) system, so principal moments occur at specific angles, making this field useful.

- Mx is the out-of-plane bending moment. For slabs, it follows the orientation set in the "Angle to X axis" field. For walls, Mx is always directed vertically.
- My is the out-of-plane bending moment perpendicular to Mx. For walls, it is always oriented horizontally, in the plane of the wall.
- Mxy is the out-of-plane torsion moment.
- A positive value for the moment corresponds to tension at the bottom fiber of the slab, commonly called the "positive" moment at mid-span.
- Nx is the membrane force, it corresponds to the in-plane axial force in the panel, in the "x" direction. If "Angle to X axis" is zero, "x" is parallel to the global "X" axis.
- Ny is the in-plane axial force in the direction perpendicular to "x".
- Nxy is the in-plane shear force. For walls, it follows the wall's orientation.
- For walls, Nx is always oriented vertically, parallel to the "Z" axis. Ny is perpendicular to Nx, in the plane of the wall.
- Vx is the out-of-plane shear force, in the "Z" direction for walls. For slabs, the orientation of Vx is specified in the "Angle to X axis" field. If this angle is zero, Vx is parallel to "X" axis.

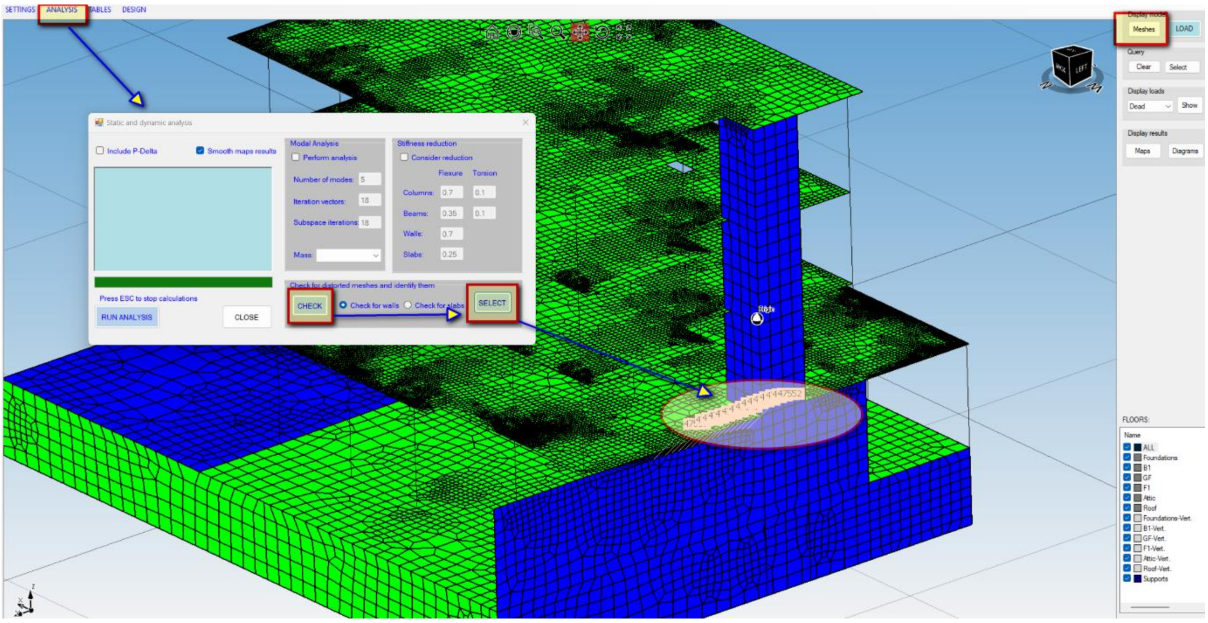
- Displacements are provided in the global system (X,Y,Z).
- It is possible to control the range of the results displayed by imposing minimum and maximum values. The scale of colors will fit within the defined interval, providing a better shades distribution.
- The scale of the results is displayed at the right of the screen, by the mean of two shades of colors. The red one is for positive values and the blue one for negative values.
- If "Show labels of results" is active, the program displays the results at the center of the finite elements, according to a density specified by the user.
- Once the "LOAD" button is clicked, the display retrieves its initial configuration.

2.2.4 Display meshes



This button visualizes the meshed structure. It helps the user to detect inconsistencies. Once meshes are displayed, the "ANALYSIS" menu provides a powerful tool that tracks incoherent meshes susceptible of instability errors during analysis. It is normal to have distorted meshes in a finite elements model. These do not necessarily lead to analysis breakdown, depending on their concentration and degree of distortion. If the run fails with instability message, it is recommended to check for the eventual wall's distorted meshes by clicking on the "CHECK" button. Then, the "SELECT" button will display the labels (in pink color) of the detected distorted meshes, as shown in the encircled area in the picture here-below.

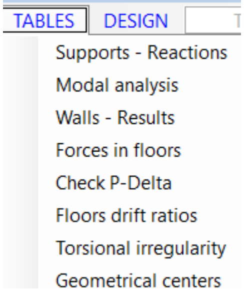
The "SETTINGS" menu in BUILD-PRO allows to solve meshing problems by controlling meshing parameters. Refer to chapter 2.1 of the present document for more information.



Instability message during analysis is not necessarily caused by incoherent meshes. If these are detected and fixed whereas run still fails, there might be isolated members not attached to the structure, like columns, walls, or beams. It is recommended then to consider an iterative approach based on the suppression of the suspected members or stories.

2.2.5 Tabulated results

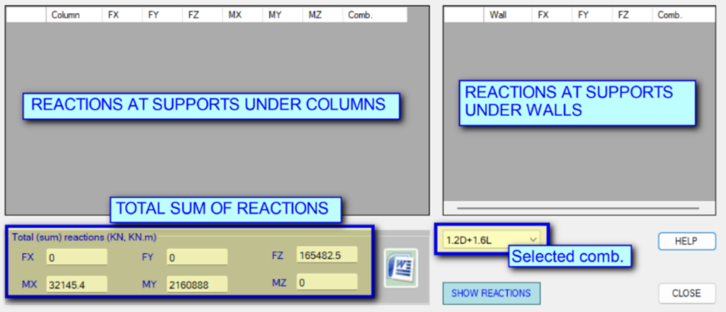
The program can show useful tabulated results:



- Reactions at the base of the structure
- Modal analysis results
- In-plane total forces in walls and cores
- Total forces at the base of each floor
- Check for P-Delta requirements
- Calculation of floors drifts ratios
- Analysis for torsional irregularity -normal or extreme.
- Determination of the center of gravity and center of rigidity for each floor.

2.2.5.1 Reactions table

Reactions under supports are provided in the following format:



- Table showing the reactions (FX,FY,FZ,MX,MY,MZ) under each column affected with a base support.
- Table showing the reactions (FX,FY,FZ) under each wall affected with a linear support.
- Total sum of reactions at the base of the building.

In the presence of general mat foundation, columns and walls are resting on slabs (no supports introduced). Tables for columns and walls are hence empty, as shown in the figure here-above. Only total reactions are provided, at the bottom of the window.

- Reactions are given according to the global system (X,Y,Z),
- Reactions under wall are the summation of all individual nodal reactions at the supports generated along the bottom side of the wall. For instance, if FZ=1000 KN, it corresponds to the total vertical force at the base of the wall.
- At the bottom of the window are shown the total sums of reactions for all forces (FX,FY,FZ) and moments (MX,MY,MZ) about the global axes.

The results shown in the window here-above are for the combination 1.2D+1.6L. Foundations consist of mat slabs (no supports introduced), justifying the absence of reactions under columns and walls. The total sum of reactions along X and Y are zero, which is expected since gravity loads do not lead to horizontal reactions. FZ is the total reaction, which corresponds in this case to the factored sum of all dead and live loads.

Total moments reactions MX,MY,MZ require careful interpretation. These are the static moments induced by FX, FY and FZ reactions:

- $MX=FZ*e_y+FY*e_z$
- $MY=FZ*e_x+FX*e_z$
- $MZ=FX*e_y+FY*e_x$

For gravity load combinations, e_x and e_z refer to the coordinates of the force FZ's application, which is the building's center of gravity. MX and MY do not have physical significance, as their values increase with the building's distance from the origin. FX, FY and MZ are always zero for such combinations.

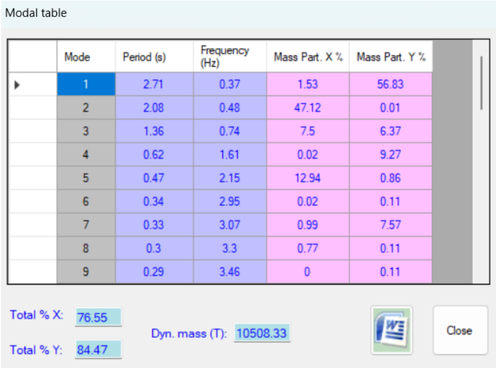
In load combinations involving both gravity and lateral forces, FY and FY denote the total reactions resulting from lateral loads, while FZ pertains to the vertical component associated with gravity. MX and MY do not exclusively represent moments induced by lateral loads, as they also include static moments generated by gravity effects.

If the user is interested in the total overturning moments MX and MY produced by wind or earthquake, only the simple load cases should be selected (Wx, Wy, E, E1…). Doing so, base shears correspond to FX and FY, and the overturning moments remain MX and MY.

Seismic base shears FX and FY are obtained by selecting seismic cases E and E1 (E1 is created when linear directional combination of Sx and Sy is selected). Combinations that involve cases E or E1 do not give correct values for base shears.

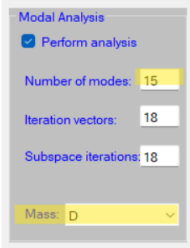
When the user chooses one of the envelope combinations (ULS+, ULS-, SLS+, SLS-), the program displays the highest (or lowest) reaction values for both columns and walls. The maximum or minimum FX (for example) value may come from a different combination than the one that gives the highest or lowest FZ value. This applies as well to the total sum of reactions shown at the bottom of the window.

2.2.5.2 <u>Modal analysis table:</u>



Mode	Period (s)	Frequency (Hz)	Mass Part. X %	Mass Part. Y %
1	2.71	0.37	1.53	56.83
2	2.08	0.48	47.12	0.01
3	1.36	0.74	7.5	6.37
4	0.62	1.61	0.02	9.27
5	0.47	2.15	12.94	0.86
6	0.34	2.95	0.02	0.11
7	0.33	3.07	0.99	7.57
8	0.3	3.3	0.77	0.11
9	0.29	3.46	0	0.11

The modal analysis table lists the vibration modes, with their number set in the analysis window. Mass participation percents are shown in X and Y directions of the global system.

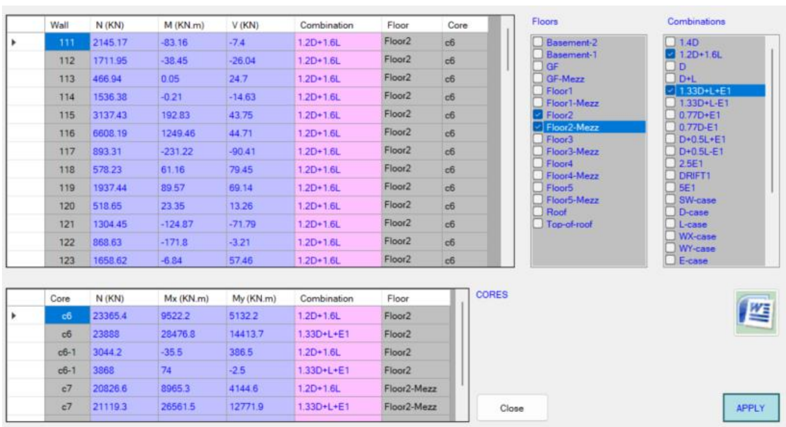


The dynamic mass shown at the bottom of the windows at the left corresponds to the combination that was selected to be converted to mass, while defining modal analysis. If the foundations system consists of isolated footings, the vibrating mass is equal to the FZ reaction under the converted combination, divided by the gravity acceleration g=9.81m/s^2. When there are mat foundations, most of their weight do not contribute to the total mass, since they are laterally restrained against translation. Therefore, the dynamic mass is less than the FZ reaction due to the converted load combination.

<u>Table of walls results</u>

First, the user selects the floors to which he wants to display the results. Second, the combinations(s) are selected. The upper table displays the in-plane axial, moment and shear forces in the walls that belong to the selected floor, for each combination. The lower table displays the axial forces and bending moments about global X and Y axes, for the defined cores.

- ➢ Numerical integration is used to derive the resultant forces in the walls, based on those inside the finite elements. The triplet (N,M,V) is of practical importance for reinforced concrete design. These in-plane forces are applied at the center of the wall,
- ➢ Same method is used to calculate (N,Mx,My) in cores. These are given at the centroid of the group of walls that constitute the core. The program can export these forces to WALLS EXPERT module for reinforced concrete design of single walls and cores. It is important to note that Mx and My are about the global axes X and Y.
- ➢ For both walls and cores, forces are computed at the base of the floor (bottom of walls).
- ➢ When envelopes are selected (ULS+,ULS-,SLS+,SLS-) the extremum values are displayed. These are not concomitant to combinations, since for the same wall, the maximal axial force and maximal moment might not come from the same combination.



Wall	N (KN)	M (KN.m)	V (KN)	Combination	Floor	Core
111	2145.17	-83.16	-7.4	1.2D+1.6L	Floor2	c6
112	1711.95	-38.45	-26.04	1.2D+1.6L	Floor2	c6
113	466.94	0.05	24.7	1.2D+1.6L	Floor2	c6
114	1536.38	-0.21	-14.63	1.2D+1.6L	Floor2	c6
115	3137.43	192.83	43.75	1.2D+1.6L	Floor2	c6
116	6608.19	1249.46	44.71	1.2D+1.6L	Floor2	c6
117	893.31	-231.22	-90.41	1.2D+1.6L	Floor2	c6
118	578.23	61.16	79.45	1.2D+1.6L	Floor2	c6
119	1937.44	89.57	69.14	1.2D+1.6L	Floor2	c6
120	518.65	23.35	13.26	1.2D+1.6L	Floor2	c6
121	1304.45	-124.87	-71.79	1.2D+1.6L	Floor2	c6
122	868.63	-171.8	-3.21	1.2D+1.6L	Floor2	c6
123	1658.62	-6.84	57.46	1.2D+1.6L	Floor2	c6

Core	N (KN)	Mx (KN.m)	My (KN.m)	Combination	Floor
c6	23365.4	9522.2	5132.2	1.2D+1.6L	Floor2
c6	23888	28476.8	14413.7	1.33D+L+E1	Floor2
c6-1	3044.2	-35.5	386.5	1.2D+1.6L	Floor2
c6-1	3868	74	-2.5	1.33D+L+E1	Floor2
c7	20826.6	8965.3	4144.6	1.2D+1.6L	Floor2-Mezz
c7	21119.3	26561.5	12771.9	1.33D+L+E1	Floor2-Mezz

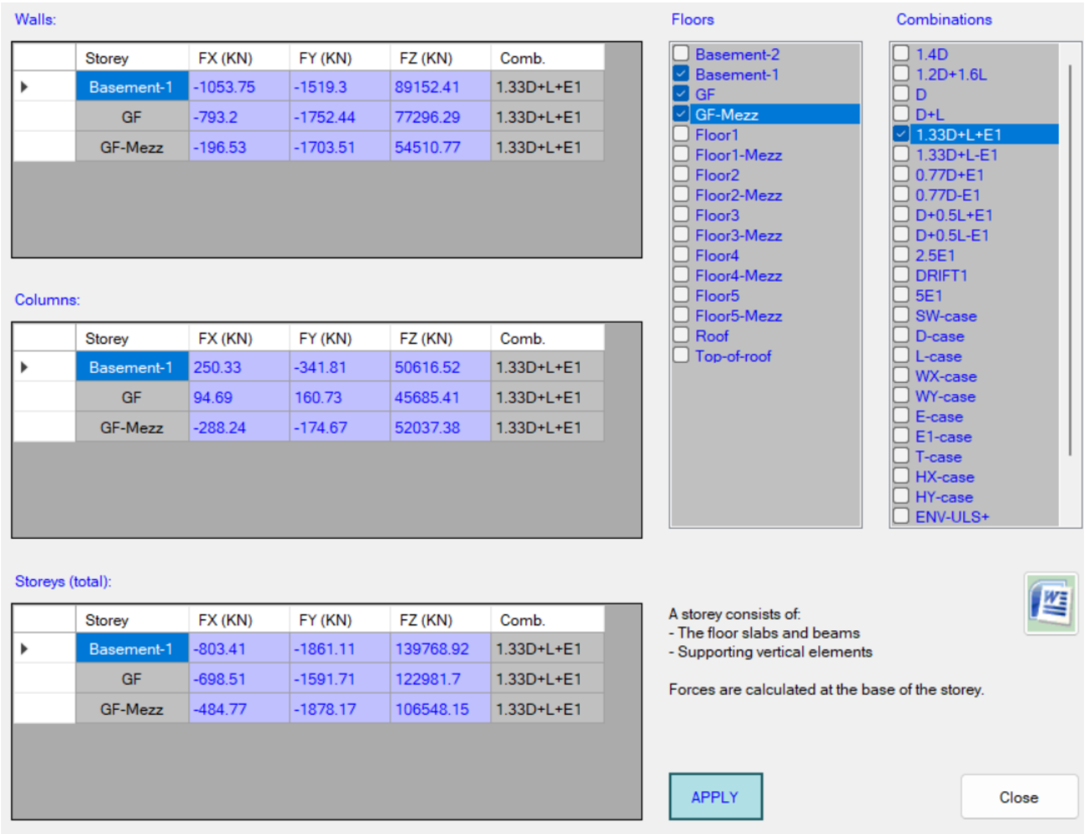
2.2.5.3 Table of forces in floors

This window presents three tables summarizing the total forces FX, FY, and FZ calculated for the entire floor. The process involves making a horizontal cut at the base of a story and summing up the forces in X, Y, and Z directions.

- The first table gives the forces that are resisted by the walls alone. For instance, FZ is the sum of all axial forces in walls that belong to the selected floors.
- The second table gives the forces that are resisted by columns alone. For instance, FX is the horizontal force in X direction computed as the sum of all FX forces in the floor's columns.
- In the third table, the forces are the sum of those for walls and columns. FX and FY can be interpreted as floor's base shears, whereas FZ is the total "weight" of the floor plus those above. For instance, these forces for the lowest floor correspond to the total reactions shown in the "Supports – reactions" table.
- These tables are particularly useful when seismic case "E" (or "E1") is selected. It helps the user to identify for each floor how the shear is shared between columns and walls. For instance, comparable share leads to the classification of the lateral-resisting system as mixed.
- Under gravity loads, the share of the total axial force between walls and columns helps in assessing whether the building is a bearing walls or frame system.

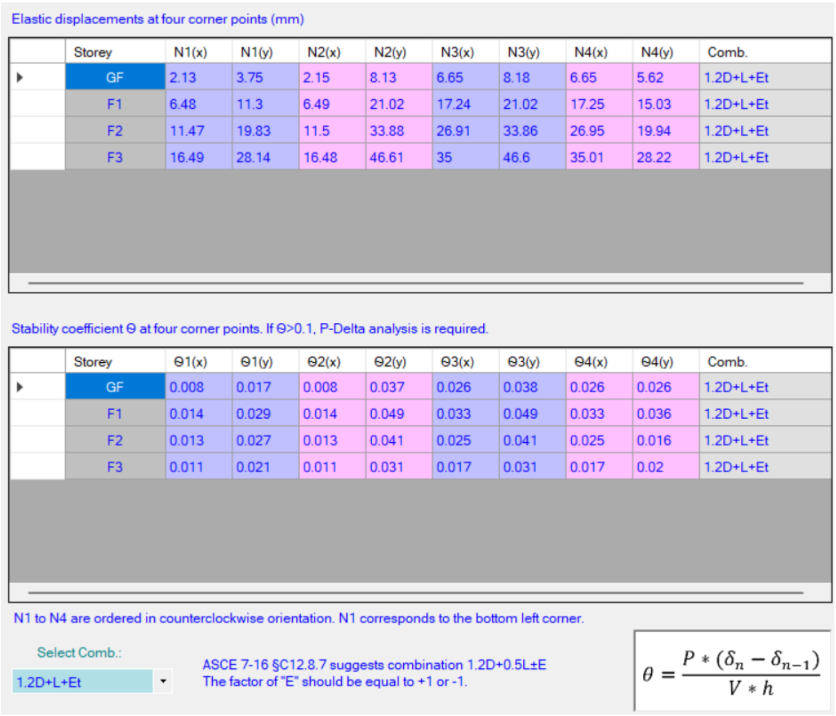
These results are intended to assist users in classifying the lateral-resisting system according to ASCE guidelines. They are computed at the base of the floor, i.e. at the bottom of columns and walls.

⚠ When raft foundations are present, FZ on the lowest floor excludes the weight of the raft as well as any dead and live loads applied to it.



2.2.5.4 Table of P-Delta check

ASCE 7-16 initiates the calculation of the stability coefficient Θ that is a simplified representation of the ratio {geometric stiffness / elastic stiffness}. When this ratio exceeds 0.1, second-order effects cannot be neglected, and a non-linear analysis based on the deformed configuration of the structure is required. The commonly used combination for this exercise is 1.2D+0.5L$\pm$E. The program derives the axial force in each storey, the seismic shear and its corresponding elastic drift, and calculates the stability coefficient at the four corners N1,N2,N3,N4 numbered counterclockwise, where N1 corresponds to the bottom left corner of the floor. If Θ>0.1 at any corner point, a message is displayed. The user can investigate the table of Θi (second table here-below) to decide whether a P-Delta analysis is required or not.


Elastic displacements at four corner points (mm)

Storey	N1(x)	N1(y)	N2(x)	N2(y)	N3(x)	N3(y)	N4(x)	N4(y)	Comb.
GF	2.13	3.75	2.15	8.13	6.65	8.18	6.65	5.62	1.2D+L+Et
F1	6.48	11.3	6.49	21.02	17.24	21.02	17.25	15.03	1.2D+L+Et
F2	11.47	19.83	11.5	33.88	26.91	33.86	26.95	19.94	1.2D+L+Et
F3	16.49	28.14	16.48	46.61	35	46.6	35.01	28.22	1.2D+L+Et

Stability coefficient Θ at four corner points. If Θ>0.1, P-Delta analysis is required.

Storey	Θ1(x)	Θ1(y)	Θ2(x)	Θ2(y)	Θ3(x)	Θ3(y)	Θ4(x)	Θ4(y)	Comb.
GF	0.008	0.017	0.008	0.037	0.026	0.038	0.026	0.026	1.2D+L+Et
F1	0.014	0.029	0.014	0.049	0.033	0.049	0.033	0.036	1.2D+L+Et
F2	0.013	0.027	0.013	0.041	0.025	0.041	0.025	0.016	1.2D+L+Et
F3	0.011	0.021	0.011	0.031	0.017	0.031	0.017	0.02	1.2D+L+Et

N1 to N4 are ordered in counterclockwise orientation. N1 corresponds to the bottom left corner.

Select Comb.: 1.2D+L+Et

ASCE 7-16 §C12.8.7 suggests combination 1.2D+0.5L±E
The factor of "E" should be equal to +1 or -1.

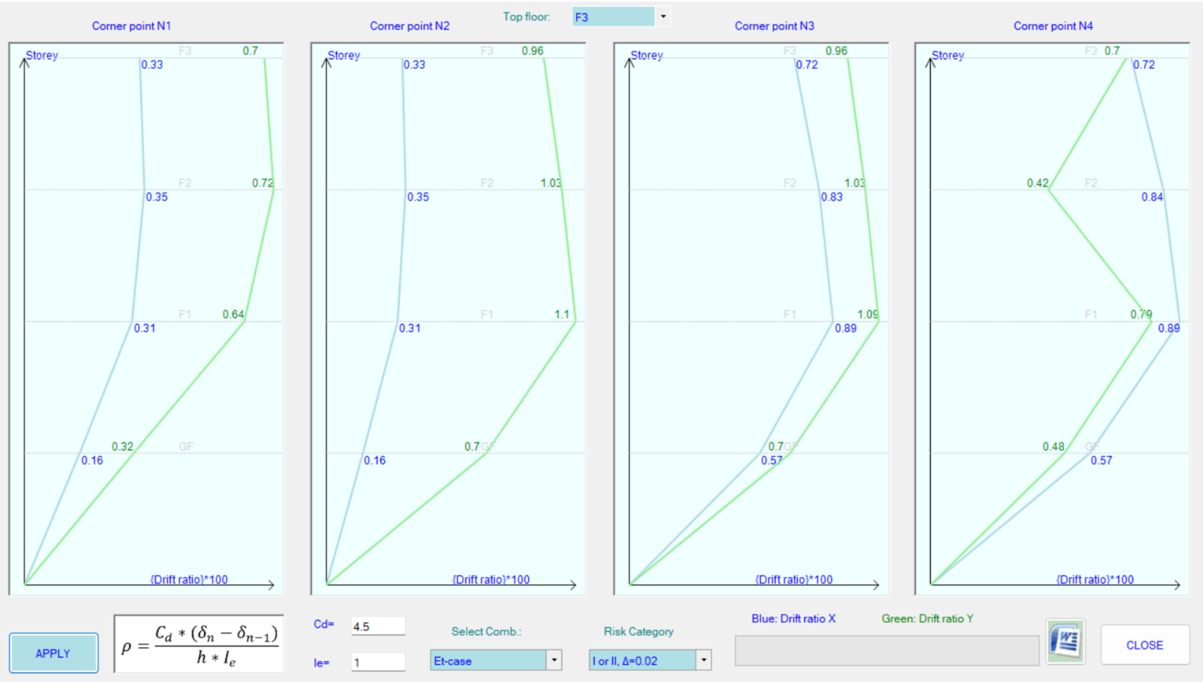
$$\theta = \frac{P * (\delta_n - \delta_{n-1})}{V * h}$$

It is important to note that if the geometry of two consecutive floors is noticeably different (recessed mezzanine floor for instance) the corner points will not be superposed. The stability coefficient at the affected location needs to be revised by the user. The program selects the location of the four corner points based on the geometry of each floor. Internal points (inside the slab) are not captured.

2.2.5.5 Storey drift ratios

Depending on the Risk Category of the building, ASCE 7-16 sets a limit for the floor drift ratios. For instance, storey drift ratio for a category I or II should not exceed 0.02, which corresponds in terms of displacements to the height of the storey divided by 50. The drift ratio is equal to the inelastic drift divided by the height of the storey and by the importance factor. The user needs to introduce the deflection amplification factor Cd, the importance factor Ie and the risk category.

The program draws at the location of each corner point a diagram that shows the variation of the drift ratio along the stories of the building. For clarify the plotted values are factored by 100.



In the case of a linear analysis, the drift ratios need to be checked under simple seismic cases.

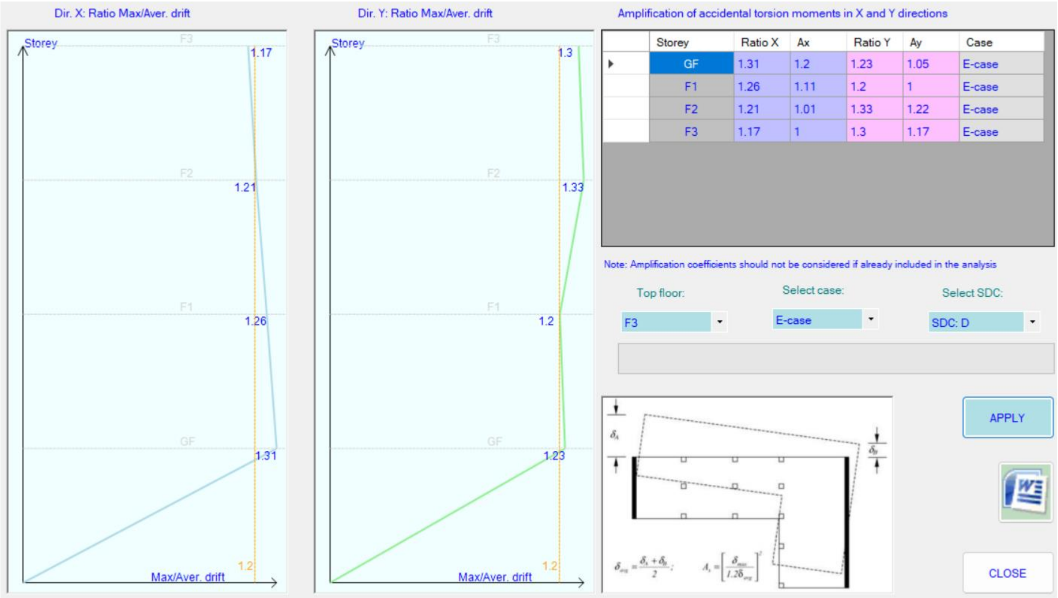
If P-Delta analysis is performed, seismic combinations should be considered since they account for drift amplification under gravity loads.

When the limit is exceeded, a red line is shown on the diagrams.

It is important to note that if the geometry of two consecutive floors is noticeably different (recessed mezzanine floor for instance) the corner points will not be superposed. Hence, the diagrams might display a singular jump that is not representative of the situation. In such cases, the user should manually check the drift ratios. The program selects the location of the four corner points based on the geometry of each floor. Internal points (inside the slab) are not captured.

2.2.5.6 **Torsional irregularity**

Torsional irregularity is quantified by comparing the maximal drift (usually corresponding to a slab's corner) to the average one, for the considered floor edge. Commonly, this can be done under simple seismic cases since the resulting drift is intrinsic to the seismic shear. The program computes drifts in X and Y directions and generates the corresponding diagrams where the ratio Max./Aver. is plotted for all floors. For the X direction, the right (or the left, depending on which is more critical) edge of the slab is considered. For the direction Y, either the upper or lower edge (depending on which is more critical) of the slab is considered.

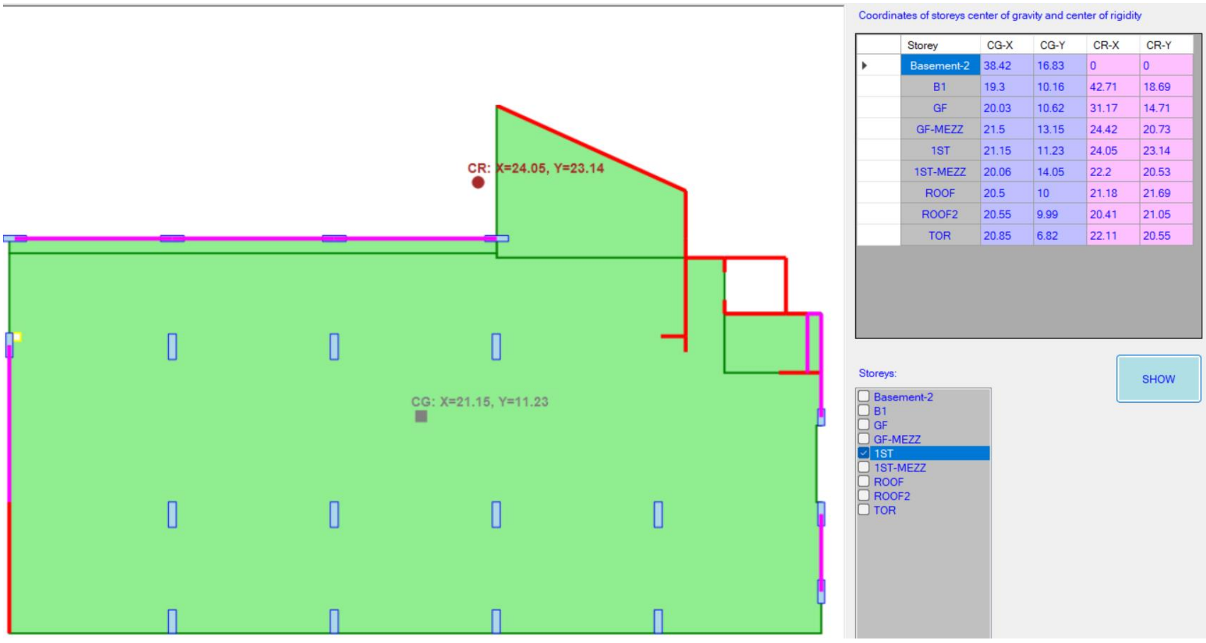


If the ratio Max/Aver. exceeds 1.2, the accidental torsional moments must be amplified with the coefficients Ax and Ay shown in the table. The user should open the eccentricities table (section 2.2.6 of the present document) and amplify the eccentricity ratios which are always 5% when torsional irregularity is analyzed. Once eccentricities are magnified, torsional irregularity should not be re-checked again.

If Seismic Design Category is E or F and the ratio>1.4, ASCE 7-16 suggests reviewing the configuration of the lateral-resisting system. If SDC=D and the ratio>1.4, then the redundancy factor ρ that factors the seismic case should be 1.3. The Code penalizes the structure in this case by increasing seismic forces by 30%.

2.2.5.7 Geometrical centers

Floors center of gravity (CG) and rigidity center (CR) are calculated from the “Analysis” menu (refer to section 2.2.6 in the present document). They are tabulated with their coordinates and shown graphically, for each storey.

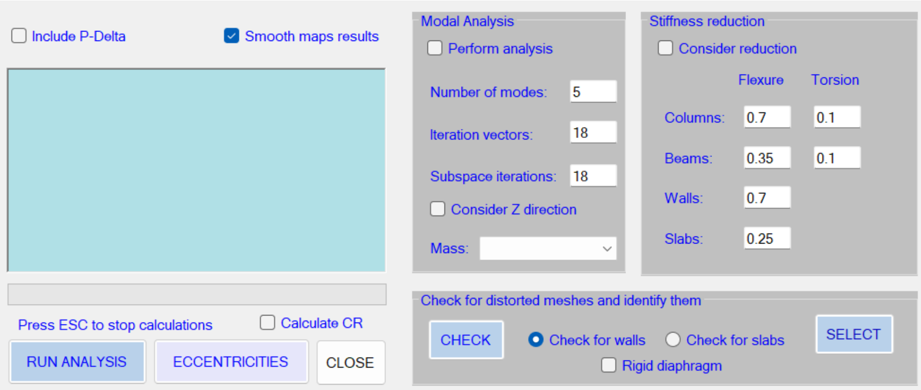


Storey	CG-X	CG-Y	CR-X	CR-Y
Basement-2	38.42	16.83	0	0
B1	19.3	10.16	42.71	18.69
GF	20.03	10.62	31.17	14.71
GF-MEZZ	21.5	13.15	24.42	20.73
1ST	21.15	11.23	24.05	23.14
1ST-MEZZ	20.06	14.05	22.2	20.53
ROOF	20.5	10	21.18	21.69
ROOF2	20.55	9.99	20.41	21.05
TOR	20.85	6.82	22.11	20.55

The program calculates them based on the stiffness of the whole building. The center of gravity is determined from all dead loads, including the self-weight of the columns and walls that pertain to the storey in question.

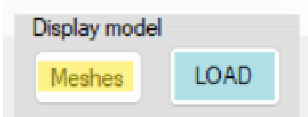
The program uses these centers for the determination of the eccentricities in X and Y directions. Once e_x and e_y are obtained, the program can identify in which direction the center of gravity needs to be shifted to obtain the critical accidental torsional moment.

2.2.6 The ANALYSIS Menu



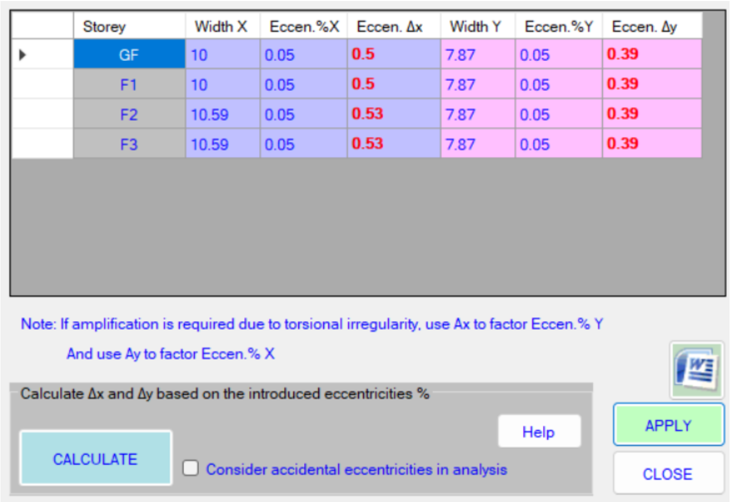
Calculations are performed from the 3D Viewer. For a classical static analysis, just click the “Run analysis” button.

- “Smooth maps results” applies stress averaging for adjacent elements.
- It is possible to reduce the elastic stiffness of the members, to simulate their cracked inertia. For walls and slabs, the reduction concerns the out-of-plane bending stiffness.
- “Include P-delta” option. This analysis refers to the second order effect associated with the lateral translation of the building; that’s called the P-Δ effect. The stiffness matrix is updated to reflect the deformed shape of the structure, leading to increased lateral displacements and bending moments. The geometrical stiffness of the columns and beams is included. The presence of compression forces reduces the global bending stiffness. Only combinations are calculated for P-Delta, because simple load cases (ex. Wind case alone) might lead to instabilities. If modal analysis is performed, P-Delta is applied to the combination that is converted to mass, leading to bigger periods.
- If in the supports table uplift is allowed, non-linear analysis is performed automatically. It concerns only supports and mat foundations. It should be confused with P-Delta.
- Modal analysis requires the selection of the combination that corresponds to the dynamic mass. For instance, ASCE 7 considers only dead loads to be included for residential buildings. In the figure above, combination “D” is selected; it comprises self-weight and the introduced dead loads. The remaining parameters need not to be modified unless requested by the program.
- The bottom of the window provides a tool to help the user solve unsuccessful analysis due to instability error message. The “CHECK” button detects distorted meshes in walls and visualize them. First the meshed model should be displayed. The “SELECT” button displays the detected distorted meshes. It is important to note that instability error is not necessarily related to the presence of such meshes. Refer to STEP 10 chapter 1.1 and to chapter 2.1 on how to fix distorted meshes.



- Chapter 3.4 discusses problems related to analysis.
- If “Calculate CR” is checked, the program calculates for each floor the location of the center of gravity and center of rigidity. This calculation needs to be performed only once, if both the geometry of the model and the loads are not modified. It is required to be done if the accidental torsional moments need to be incorporated into seismic cases. It is possible to graphically display the location of the centers with the menu “Tables”→”Geometrical centers”.
- The button “Eccentricities” enables the consideration of the accidental torsion moments which are equal to the base shear at the concerned story, times the accidental eccentricity (per default 5% of the floor dimension is X and Y directions).

Once the button “Apply” is clicked, the table displays the width of each floor in X and Y directions and calculates the eccentricities (in meter) starting from the introduced ratios. If torsional irregularities are applicable -which can be checked from the menu “Tables”→ “Torsional irregularities”-, the user can modify the defaults ratios of 5% and click on the “Calculate” button. The program will update the eccentricities by factoring each ratio by the floor width. Once the checkbox “Consider accidental eccentricities” is activated, the program will consider in the analysis the effect of the accidental torsional moments.



The program calculates the accidental torsional moment for each floor and adds it to the seismic cases “E” and “E1” with the appropriate signs (clockwise and counterclockwise) that lead to the most critical effect.

If the quadratic directional combination is selected (E=SRSS{Sx,Sy}), the program creates a new seismic case named “Et”, identical to the original one “E”, where the sole difference is the sign of the accidental torsion moment, to know:

- Seismic case “E” includes clockwise torsion moment.
- Seismic case “Et” includes counterclockwise torsion moment.

The seismic combinations introduced by the user with the case “E” are automatically duplicated, where “E” is replaced by “Et”.

The consideration of the accidental torsional moments affects the original seismic cases by adding to them a global torsion couple “T” about the vertical “Z” axis:

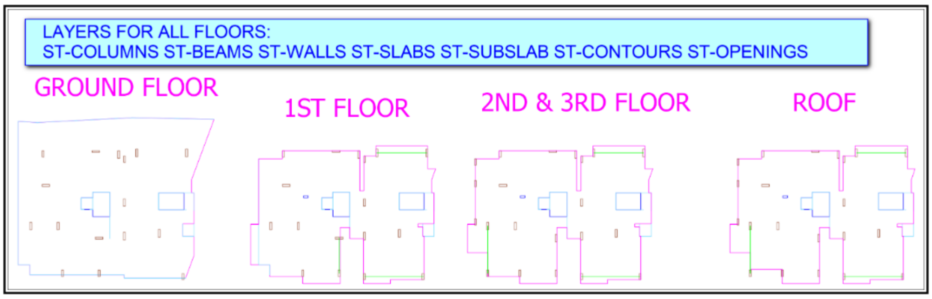
$E=E_{original}+T$ and $Et=E_{original}-T$. The seismic combinations are hence directly duplicated.

Chapter 3- Modeling insights

3.1 Prepare the drawing and import it

Preparation model in AutoCAD

The preparation drawing in AutoCAD should comprise all floors (beside those identical) with the structural objects (slabs, columns, openings…) affected to the layers shown in this picture. If the floors are superposed, centers of columns and lines of walls should be perfectly aligned. That's why it is recommended to copy the modeled floor based on a fixed reference point and then operate the required modifications. Once this procedure is repeated successively to all floors, this will guarantee the alignment of the vertical elements.

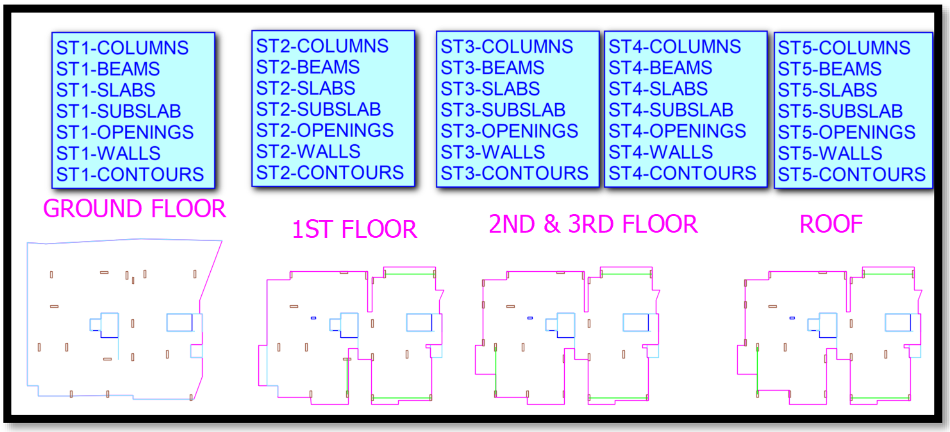


Import drawing to BUILD PRO

1. Import the drawing (Menu "FILE"→"Open Dwg")
2. Save the project (Menu "FILE"→"Save Project")
3. Open the Setting to specify foundations type (Menu "SETTINGS"→"General Settings")
4. Create the first floor (Menu "STORIES"→"Stories Manager")

Let's consider the preparation drawing done on AutoCAD, imported into BUILD-PRO. For this five-floor building, all lines and polylines -for all floors- belong to these seven layers listed here-above. For BUILD-PRO, these lines are still drafting objects, not structural members. Make sure the titles of the floors (if they exist) are not included by mistake inside these layers.

Once the first floor is created, the related layers are generated. The user should change the object's layers from their initial one to that corresponding to the type of the member. For instance, if we consider columns: "ST-COLUMNS" to "ST1-COLUMNS". Once this procedure is applied to all the members of the created floor, the "Convert To Structure" command should be prompted. At this moment it becomes possible to affect the desired sections to the members, since the software recognizes the drafting objects as structural elements. Once all structural members of ST1 are defined, the second floor can be created, members transferred to layers "ST2-…", convert to structure and affect the sections.

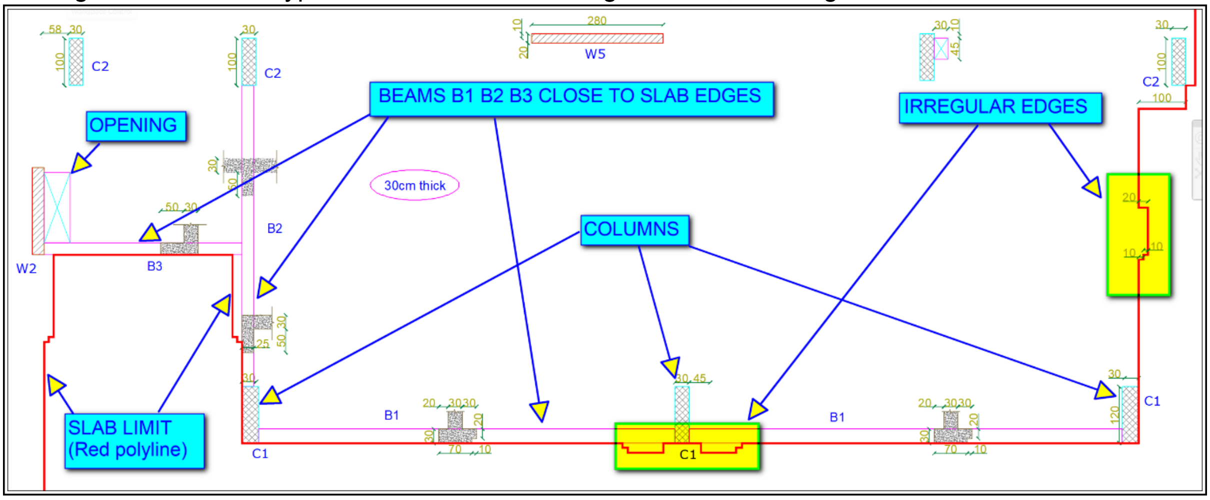


3.2 Basis of modeling

From structural drawings to the equivalent model

Seeming trivial at first impression, drawing the geometry of the slab is of utmost importance because of its direct consequences on the meshing and calculation steps. If not realized with great precision and care, the geometrical errors might prevent meshing from being generated or even lead to analysis problems. Some important principles should be followed at very early phases, during drafting.

This figure illustrates a typical real structural framing of a slab, showing beams, columns and walls.



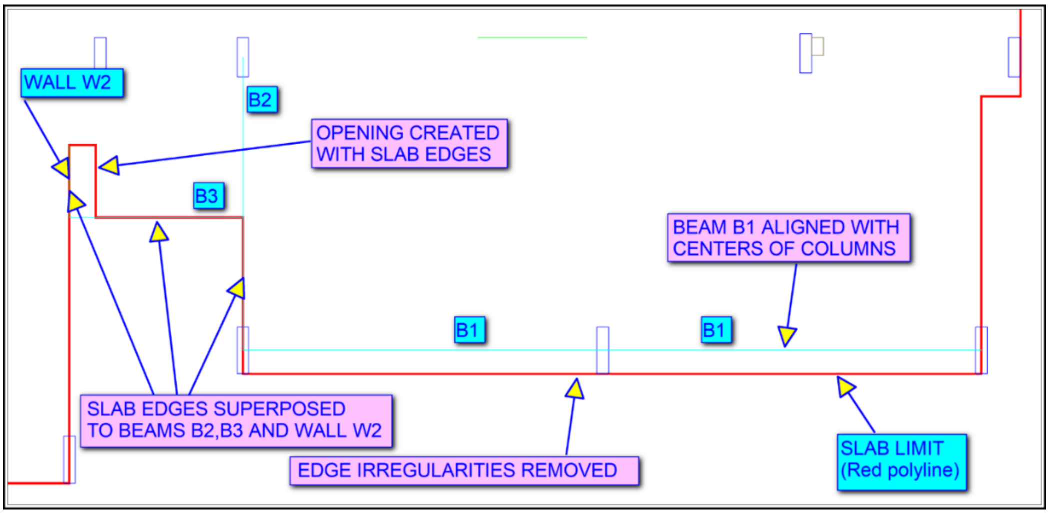
The following characteristics are noted:

- Edge beams, close to the slab limits
- Irregular slab edges related to architectural requirements
- Presence of opening at the face of wall W2

The equivalent preparation model is shown in the figure here-below.

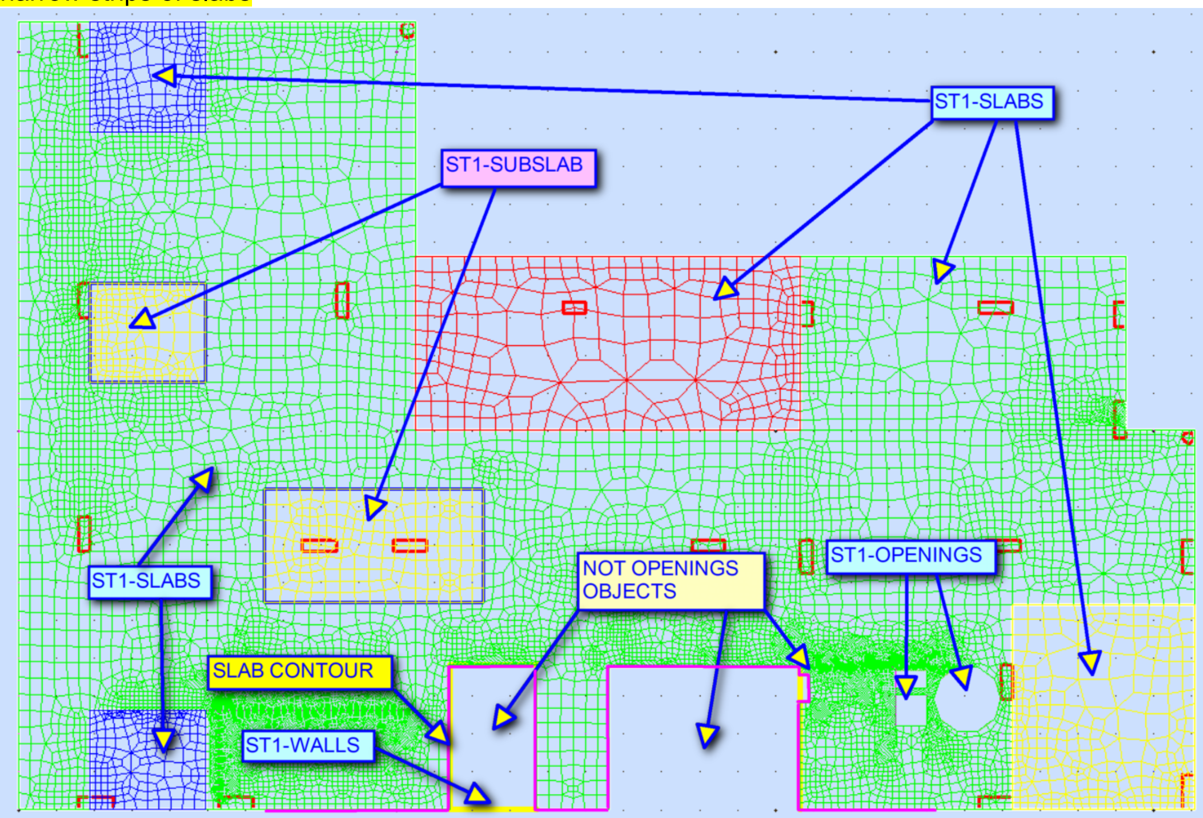
- Beams are line objects in layer “ST-BEAMS”. Similarly to the treatment of walls, these lines are typically created at mid-width of the webs. But in the present case, to avoid having narrow slabs strip with potential meshing problems, the edges of the slab are heavily modified to superpose over beams lines. In the other hand, beams should intersect with the center of the rectangles which represent the columns. That’s why beam B1 shifts upward to coincide with the centers of the three columns. The strip between B1 and the slab edge below is large enough to allow a successful meshing. It is recommended to draw beams as individual segments spanning between column’s center instead of one single segment (refer to the two-span beam B1).
- Slabs are modeled as closed polylines in layer “ST-SLABS”. Most of the time, it is not possible to follow exactly the architectural outline. It is highly advisable to remove decorative protrusions and recesses, as they complicate meshing and greatly increase the number of meshes. At the left of the figure, there is an opening at the corner of the slab, adjacent to wall W2. A particular treatment is required. It involves aligning W2 with slab edge, and modeling the opening with the slab edge itself, without the need to create an opening by drawing a rectangle in layer “ST-OPENINGS”. Beam B3 is as well aligned with the slab edge and coincides with the end node of the wall, though that is not mandatory. It is important to avoid two nodes distant less than 15cm (end nodes of: beam and wall, beam and column center, beam and slab vertex, etc…)
It is possible to represent the entire slab area with one single closed polyline, or several ones if different thicknesses and different loads are expected.
- To model sunken slabs or drop panels, closed polylines should be used, in layer “ST-SUBSLAB”. Such regions should be 100% located inside the slab, with not a single edge of a

subslab superposed to any edge of the slab (total island). If the sunken or the drop panel is located at the edge of the main slab, then it should be modeled as a slab and not as a subslab.



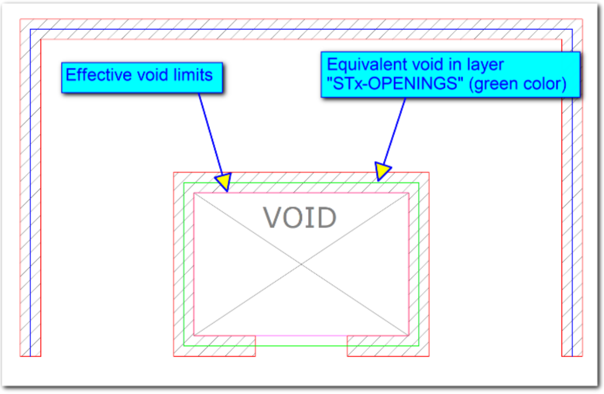
The equivalent model

The following figure is another illustration of the stated guidelines. Slabs not entirely enclosed within the parent slab should be in layer “STx-slabs”. Drop panels and sunken should be entirely surrounded by a parent slab, so that they are modeled in layer “STx-subslab”. At the bottom of the figure there is a lift case and a staircase. Due to their vicinity to the edge of the slab, the contour of this latter is studiously adjusted (refer to the lines in magenta color) to model the corresponding voids. A slab cut should be entirely located inside it, to be modeled with a closed contour in layer “STx-openings”. It is always recommended to model openings at edges of slabs with the slab contour itself, instead of keeping narrow strips of slabs



Openings in slabs often occur at the edges. In this case, it is not recommended to keep a small strip between the edge of the slab and that of the void. It is more efficient to simply adjust the limits of the slab to match those of the opening edge, as shown in the figure above.

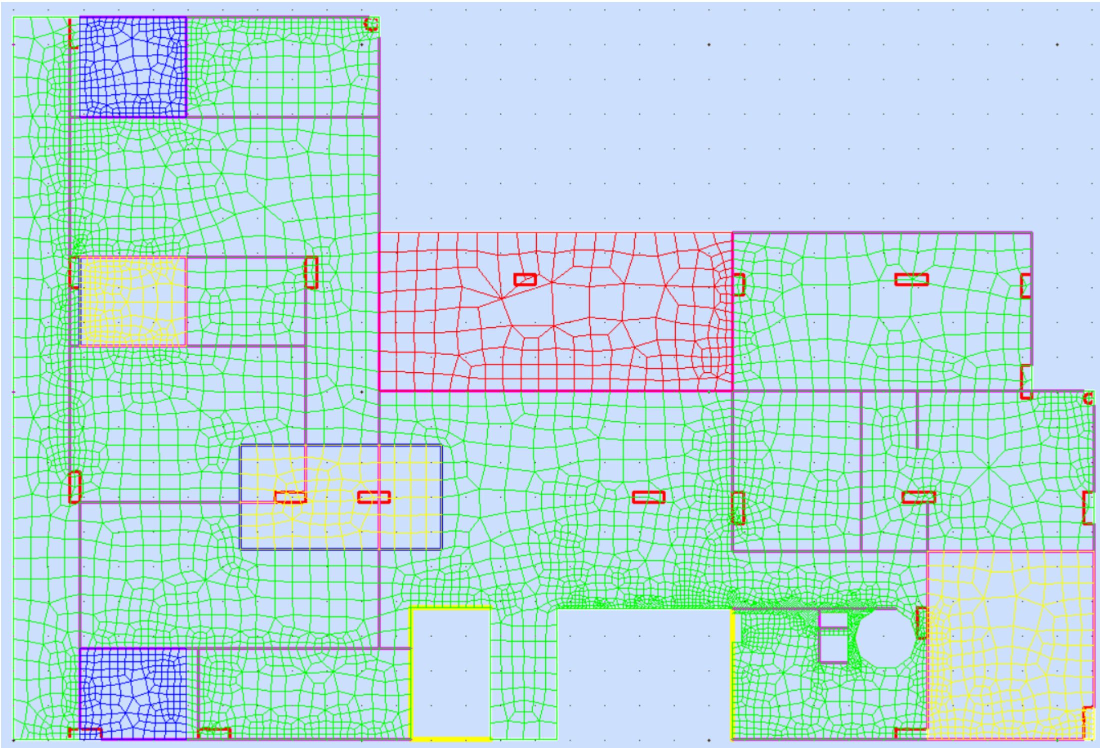
Note that openings, like slabs, are modeled with closed polylines. A circular opening should be discretized into a series of segments.



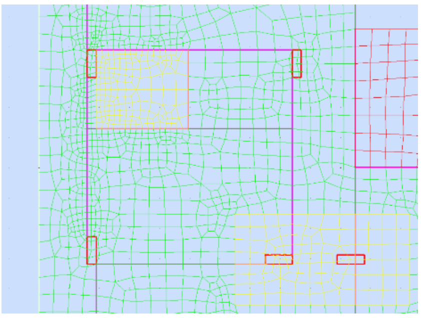
This another sample on how to deal with openings located entirely within a slab.
The effective lift shaft corresponds to the “X” symbol in gray color. The modeled opening in layer “STx-openings” (rectangle in green color) is larger. It is superposed exactly over the wall’s lines. Keeping its original size will lead to narrow strips between the opening and the walls lines, prone to errors in meshing.

Drawing lines: good practice

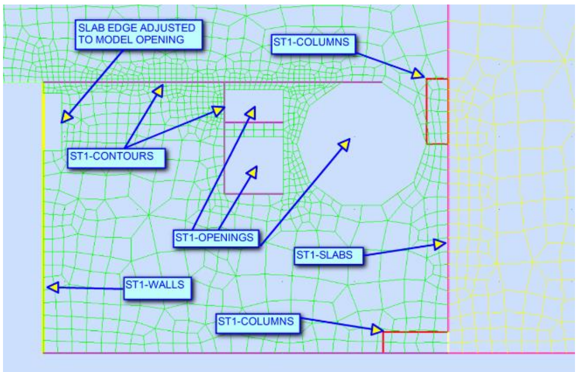
This figure displays the internal slab partitions for applying linear loads. The "STx-contours" layer enables linear load application on its lines (in magenta color), which are connected to the meshes and follow the same previously mentioned rules. Note lines intersections and alignments.



Internal partitions (lines in magenta), located in layer “STx-contours”

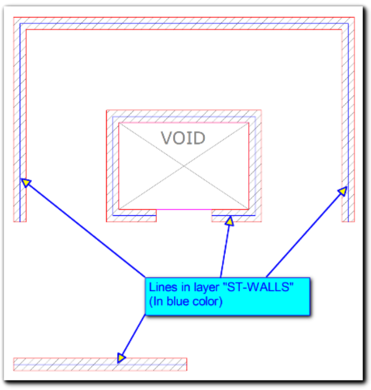


This figure -extracted from the previous one- shows a yellow-colored sunken slab (in layer “STx-subslab”) surrounded by lines in layer “STx-contours”. These lines correspond to the masonry blocks partitions, intended to receive linear loads. They are not modeled at mid-width of the walls but rather shifted to the edges of the sunken slab. If drawn at their architectural location, narrow strips will arise between the edge of the sunken and the line itself, leading to unnecessary complications.

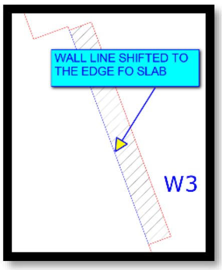


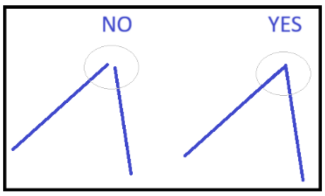
The structural wall at the left (in yellow color) is superposed to the edge of the slab. Furthermore, in the “Y” direction, its length is adjusted to match the upper corner of the slab, avoiding extending inside the slab, unless it has to, but in this case, it should be for more than 15cm depth. The partitions align precisely with the edges and lengths of the central openings, which is crucial. If partition lines extend slightly past the opening (such as in the X direction), nearby nodes can cause meshing congestion or process failure. Rather than a strict rule—such as requiring partition lengths to always match an opening edge—this is simply an indication meant to prevent nodes from being placed too closely together.

Lines in layer “STx-WALLS” represent walls, assumed to be under (supporting) the concerned slab.

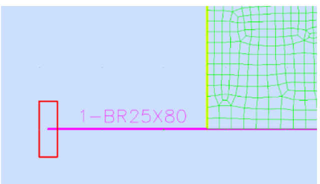


Typically, they are drawn at mid-width of the walls, although this is not mandatory, depending on the geometrical particularities. For instance, the line that corresponds to wall W3 (figure at the right) is shifted to the side of the wall, to match the slab limit. Keeping the line at mid-thickness will lead to a narrow slab strip that might prevent the slab from being meshed. In the figure at the left, lines are at the center of the walls, in the absence of constraints that might lead to their relocation.

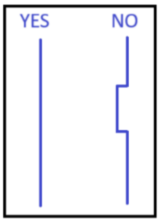




Draw correct intersections. When the ends of two non-parallel lines are intended to connect, they should. In the case where lines are not meant to meet, then do not keep a gap of less than 15cm. A small distance between two nodes will lead to dense meshing and a consequent large number of meshes. It is recommended, when possible, to avoid a meshed plan with high density of small meshes.



This figure shows the case where a column is located outside of the slab perimeter. The right end of the beam is connected to one mesh, which is fine. The left end of the beam must coincide with the center of the column, otherwise the beam is a cantilever. For the program, a circular or rectangular column is just a vertical line with upper and lower nodes. If nodes of column and beam do not perfectly meet, and the column is inside the slab, the problem becomes more critical. The program will be forced to generate very small meshes, which might halt meshing process and trigger error messages.

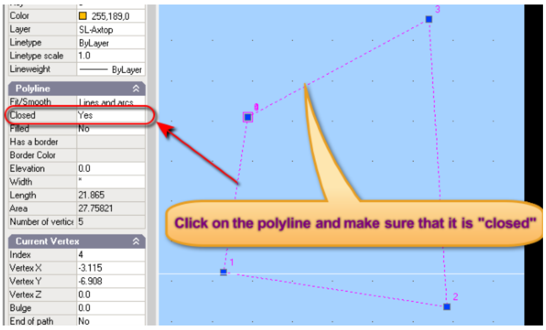


Simplify edges of slabs. Do not follow the architecture as it is. If the edge of a slab contains indentations or extrusions of amplitude less than 15cm, then do not consider them and adopt a straight edge instead. This practice avoid zones with high density of small meshes.

The emphasis on the accuracy and “cleanness” of the geometrical layout of the slab is intended to eliminate the biggest part of the meshing problems that user might encounter. The geometry of the slabs should be carefully prepared in AutoCAD.

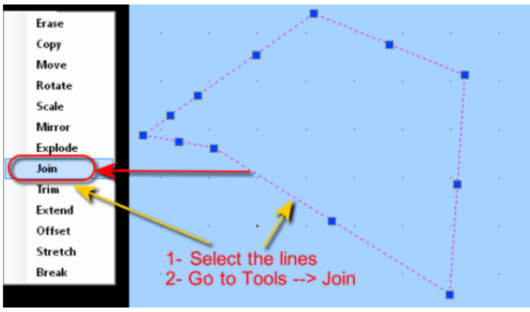
Drawing polylines: good practice

- Slabs, subslabs and openings should be closed polylines.

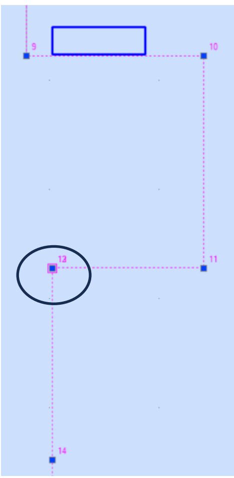


A polyline is not necessarily closed by default. Whether in AutoCAD or in BUILD-PRO, it is easy to verify the closure state.
Select it and choose “yes” in the field next to the “Closed” option.
The “Convert to structure” command detects unclosed polylines and displays the appropriate warning message.

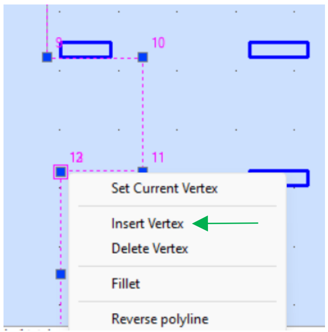
- To create a polyline from lines, follow these three steps:



a)Draw the lines in the appropriate layer
b)Select the lines with the mouse.
c)Go to “Tools” → “Join”



Polyline vertices are numbered in order, starting from 0 up to the last vertex. If two vertices overlap—often a result of repeated editing during drafting—the 3D Viewer will display an error upon loading the structure. Overlapping vertices need to be removed. You can spot them by visually inspecting their labels. Once the polyline is selected, vertices labels appear. The figure at the left shows overlapped vertices of labels 12 and 13. The shown sequence of numbering is 9-10-11-12-13-14, where labels 12 and 13 are superposed.
Right-click directly on the vertex and select "Delete" from the window that appears, to resolve the issue.

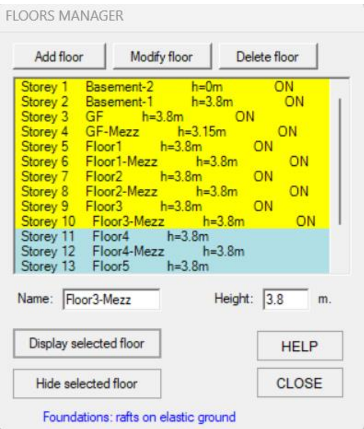


It is possible to add a new vertex to modify the geometry of a polyline, which is of big help during drafting. The new vertex will be inserted between two consecutive vertices. Just put the mouse over the vertex and right-click, a small menu will appear.

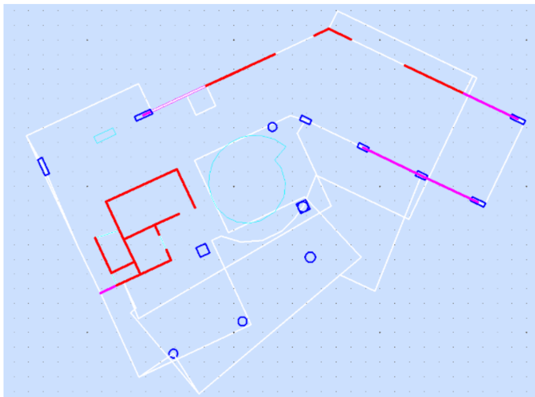
Superposing the floors

In BUILD-PRO, once all floors are created and their corresponding structural elements are defined, it is necessary to superpose them based on a fixed reference point that is common to all of them. The command “Move” from the “Modify” menu is used to align each floor over the other.
As mentioned, it is recommended to keep this step to the last, making sure that all floors have been successively converted to structure. As noted in STEP 9 (chapter 1.1), do not create the next floor unless the current one is successfully converted, since the “Convert to structure” command affects all the created floors and can make errors harder to spot.

This figure shows a defined reference base (an intersection of walls) that is not verifying along stories. The 1rst, 2nd, 3rd and roof floors will be moved over the ground floor, using this base point.



It is possible to display all floors by selecting them from the stories manager and clicking on the “Display selected floor” button. The displayed floors are highlighted in yellow color. Keep in mind that displaying a floor is simply equivalent to turning on its corresponding layers, which is possible to do manually (though it’s less efficient) via the layers window.



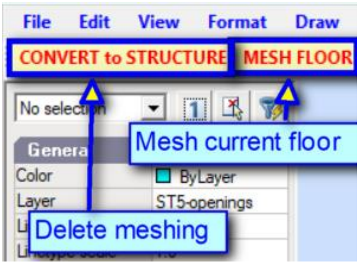
The figure on the right shows a sample of several superposed floors, all displayed. The main interest is to inspect the alignment of columns and walls, which is crucial for the integrity of the structure. One can zoom for inspection and turn off selected layers without having to hide the entire floor.

3.3 Why meshing is not generated

Introducing mesh generation

Once a floor is displayed from the stories manager, the “MESH FLOOR” command generates the finite elements meshes in the plan. It is recommended to mesh the floor once it is converted to structure and all member’s properties assigned. The program fixes the average mesh size to 60cm.

During modeling phase, it is advised to mesh the current floor before creating the next one. If the meshing operation fails, it means there are geometrical problems that must be resolved.



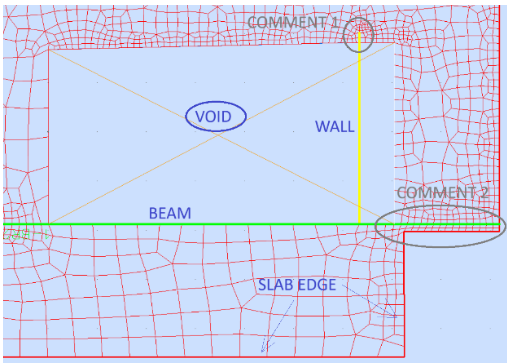
Once the meshes of the current floor are successfully generated, the user can decide whether to go for the next floor or improve meshing by simplifying the geometry of the slab. A good meshing should not comprise areas with high concentration of small meshes.
The “Convert to structure” command deletes generated meshes for the current floor, which reduces drawing size and improves command speed without compromising model integrity.

Meshes are stored in layers “STx-meshes”. It is possible to hide or display this layer from the layers manager.

Before checking integrity and running analysis from the 3D Viewer, the user can generate the meshes for all the slabs of the building, with one click. Meshes are not displayed in the drawing itself and hence the user can see them from the 3D Viewer only. Walls meshes are generated from the “Check integrity” command.

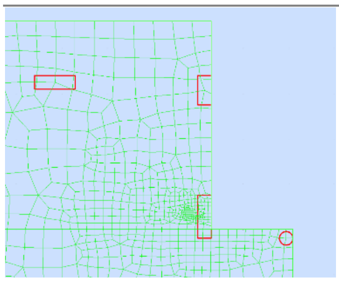
The “Mesh all slabs” command generates meshes which size is manageable by the user, from the “SETTINGS”→ “Meshing parameters” menu. For each floor a custom mesh size can be affected.

Reasons for the Production of High-Density Meshes



This slab comprises one big central opening, a wall in yellow color and one beam in green color. The first comment concerns the extension distance of the wall beyond the side of the void, as depicted in the figure. If the distance between the end of the wall and the edge of the void (or slab) is less than 15cm, a high concentration of meshes is expected. The second comment concerns the distance between the beam and the slab edge. In general, when two lines are parallel, the distance between them should not be less than 15cm. Otherwise, it is advised to superpose them.

If the above-mentioned gaps are very small (less than few centimeters), which means consequent to drafting inaccuracies, an error message appears, stating the failure of the meshing task.



Columns located at the edge of the slab are not in general a source of concern. However, in some cases where the center of the column is close to the corner of the slab, concentration of meshes might appear.

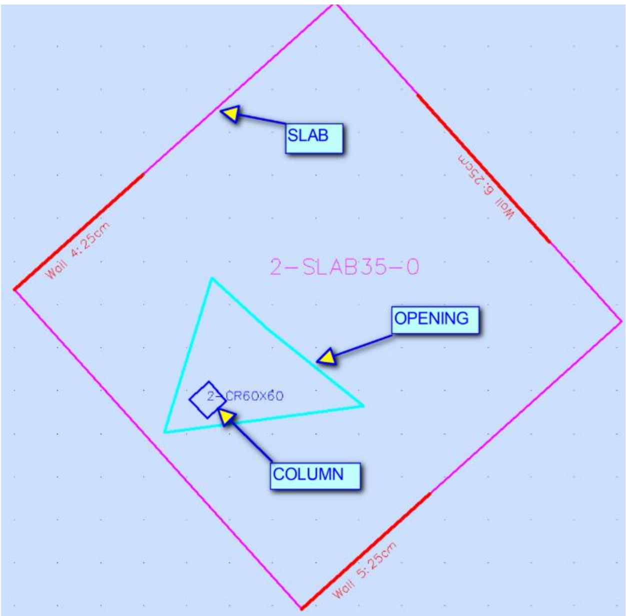
Note that the circular column, although close to the node of the slab, does not exhibit mesh concentration because the mesh size in this area is already close to the gap in question. Mesh concentration is consequent to the adaptive reduction of the mesh size from its original size to that imposed by two nearby nodes.

Error in generating meshes

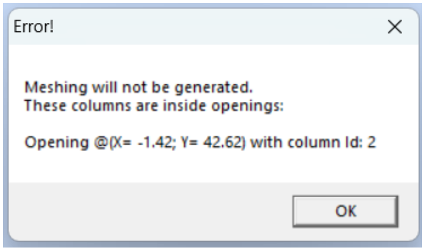
The program is equipped with a powerful algorithm that checks for common drafting errors like overlapped contours (slab-subslab, slab-opening, slab-slab, etc…), lines overlapping (beam-wall) and many other cases that the user can easily fix, thanks to the displayed self-explanatory messages.

⚠ Distinction should be made between error messages and warning messages. A warning does not halt calculations or meshing processes. It just alerts the user about a detected singularity.

Here-below is a sample of an error detected by the program, preventing meshing generation.

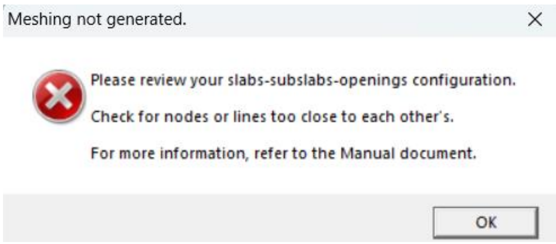


Let's consider the case of a column located inside a closed polyline that belongs to layer "STx-openings". BUILD-PRO will generate a meshing error message for this case.



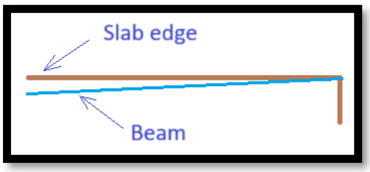
The user should not confuse between a column outside the slab -which is common to happen- and column inside the opening contour. If a column is outside the slab, hence not attached to meshes, the program displays a warning message for information only. If the structure imposes the presence of a column inside an opening, the user should remove the opening and generate it with the edges of the slab, by dividing the slab into two, for example.

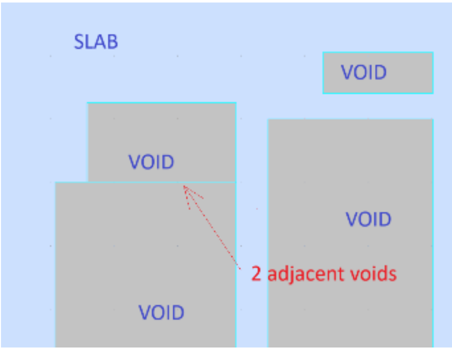
The explicit nature of the displayed message makes solving the issue an easy task. However, in some cases, the program cannot deliver an accurate description of the encountered error as shown hereafter.



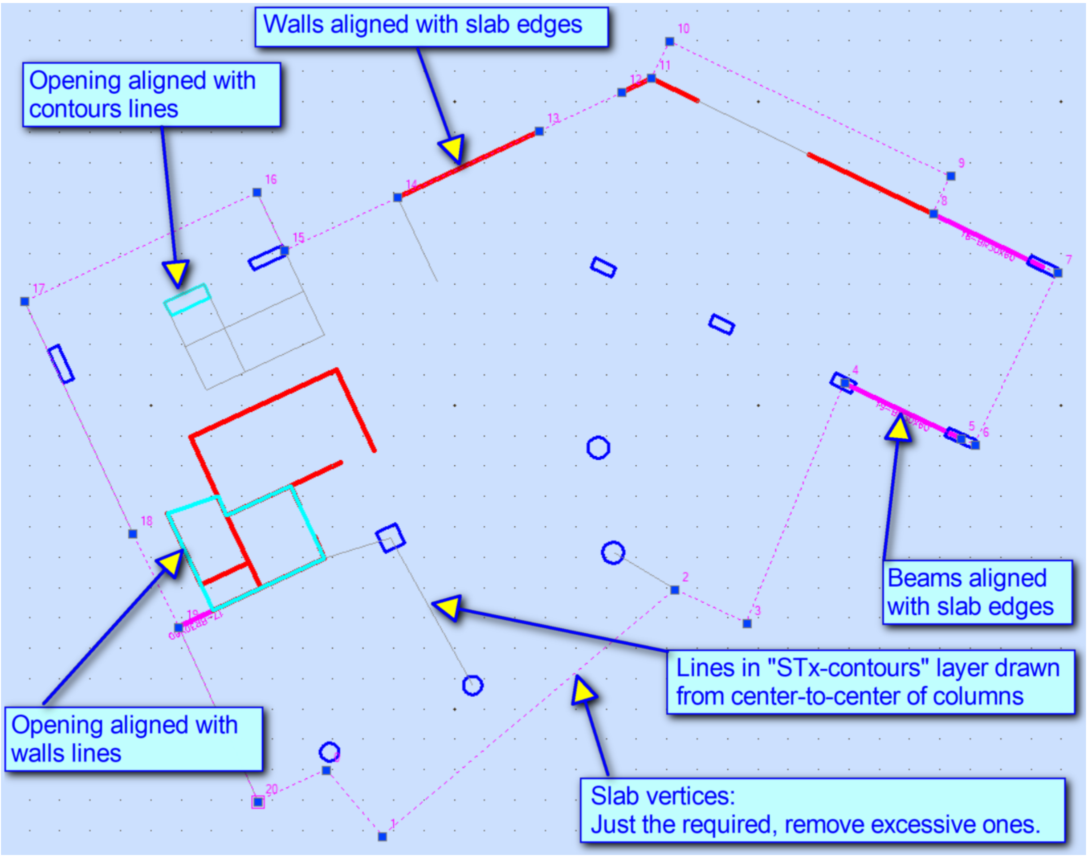
When the program fails to generate slab's meshing, and is unable to describe the problem, this error message is displayed. It is always possible to fix this issue by reviewing the geometry of the concerned floor, based on the guidelines provided in chapter 3 of the present document.

Here is a list of the most common issues:

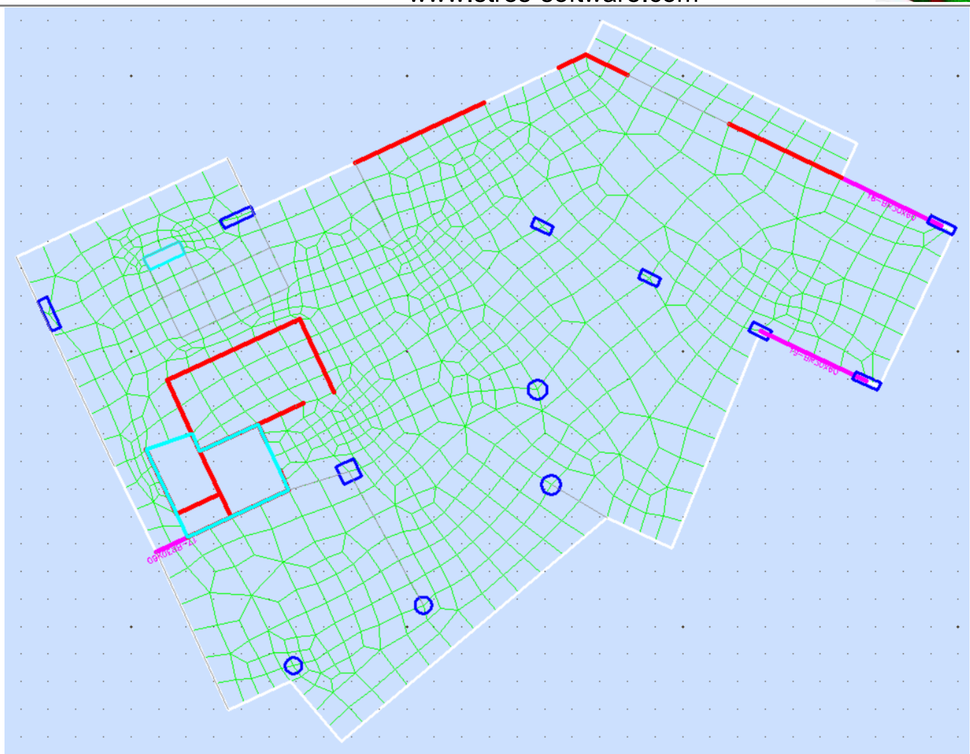


- Small angles between two lines that belong to slabs, openings, beams, walls…Such case imposes the generation of distorted meshes beyond the acceptance criteria's set by the engine.
- Two nodes very close to each other's. A node can be the end of a line, the vertex of a polyline or the center of a column. For instance, in the case of implanted columns or walls on the slab below, the user might not notice the issue unless the floors below and above are both displayed on the screen. Hence visual inspection is required.
- 

  When two openings have a common side, meshing cannot be generated. 2 adjacent voids are a wrong configuration. An opening should be surrounded by slab meshes from all sides. The two adjacent openings shown in the picture should be merged into one unique contour.
  This comment applies as well for subslab sharing a side with a slab.
- Two lines (parallel or not) with a narrow strip between them might lead to meshing errors. These lines could be a beam (or a wall) and the edge of the slab, which is common in structural drawings. It is recommended in such cases to shift the beam (or the wall) to the edge of the slab.



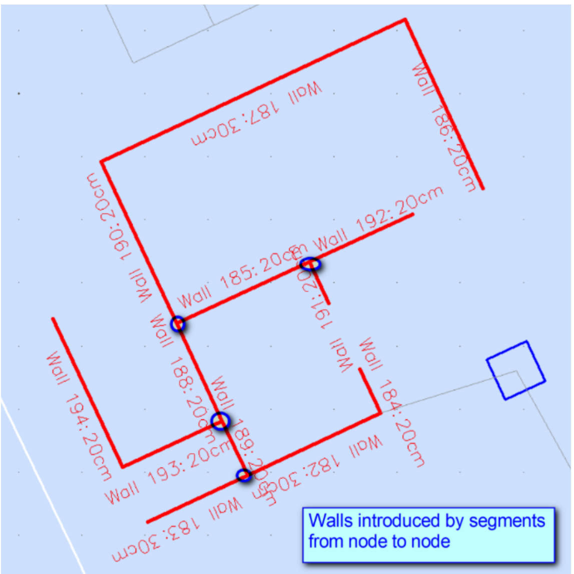
Model of a floor with guidelines comments



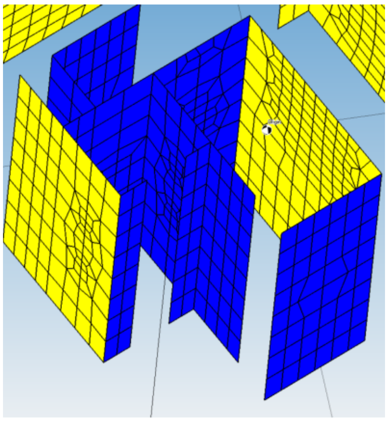
Meshed floor

When it is not possible to identify the problem, narrow the error investigation by filtering the geometry: Select the doubtful element and put it in layer zero. This way the element is not deleted from the drawing, but it will not be considered in the structural model. Each time an element is stored in layer zero (or any layer that does not begin with “STx-“) the “MESH FLOOR” command should be launched. Once the meshing is successfully generated, it will be easy for the user to detect and solve the problem.

Good practice for core walls

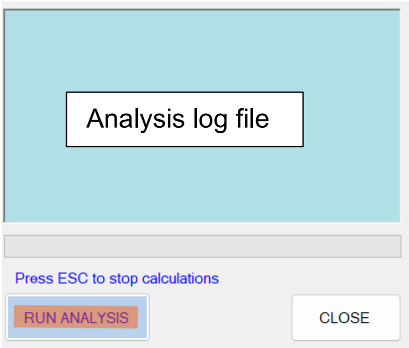


Walls are meshed during the “Check integrity” process. When a group of walls intersect, it is strongly recommended to model the walls by segments between intersection points.



3.4 Analysis errors

Why analysis fails



Click “RUN ANALYSIS” to start analysis. For your first attempt, choose static linear analysis; if this fails, more advanced options will likely fail as well. That’s why it is recommended to disregard non-linear and seismic analysis and just keep the static linear for the first run.

If eventually the static linear analysis fails, it will be easier for the user to detect and fix the error.

The program displays different types of error messages for the attention of the user, depending on their category.

Here-below is a summary of the finite element method solution procedure, in the case of the static linear analysis.

1. Preprocessing. It consists of gathering and verification of geometry, loads, materials and supports.
2. Element formulation. The program builds the local stiffness matrices of the elements based on their shape functions and calculates the equivalent nodal loads.
3. Assembly. The global system formation is done to solve the equation F=K*U. The global stiffness “K” matrix is built, and the force vector “F” constructed.
4. Application of the boundary conditions (supports).
5. Solution of the linear system. The displacements vector “U” is calculated based on the inversion of the stiffness matrix “K”.
6. Post-processing. Starting with the obtained displacements, the program derives strains then stresses and finally forces and moments.
7. Validation of the analysis by performing equilibrium checks.

An analysis is successful when every step is completed. The Integrity check that is performed prior to analysis guarantees steps 1 and 4.

Errors in linear static analysis

- Error message: “Unstable structure”. If such message is encountered in linear static analysis, it is most probably related to the presence of isolated meshes -mainly in walls-, not connected to the surrounding. It is encountered during step 4 listed here-above. Such problems arise at the intersections wall-slabs, when the software cannot generate at the top or bottom sides of the wall the required nodes that permit the generation of coherent meshes. This subject was already tackled at the following locations in the present document:
    - Chapter 1.1: STEP 10, Meshing parameters
    - Chapter 2.1: Meshing parameters
    - Chapter: 2.2: Display meshes
    - Chapter 3.3: Good practice for core walls

  In the error message displayed, the location of the problematic meshes is specified by coordinates. The user can hence take the appropriate measures to fix the issue.

- Error message: “Error during static analysis”. The program cannot identify the culprit but proposes a list of successive trials that are candidates for such error. The most common problem might be related to distorted meshes (highly irregular quadrangle) that forces the analysis to abort at step 2, during element formulation phase. The user should run the tool that identify distorted meshes -mainly in walls- and take the appropriate actions.

Errors in non-linear static analysis

Non-linear static analysis is either:

- Automatically performed when the user allows uplift of supports
- When P-delta analysis is selected.

In all cases, a linear analysis should be first attempted, to verify the soundness of the structure.

- If the non-linear analysis fails and the header of the error message is “P-Delta analysis did not converge” or “Non-linear analysis did not converge”, then it simply means that a combination of load cases leads to the structure’s instability. For instance, the overturning moment might be bigger than the stabilizing one. The combination in question is easily identified from the analysis log. The user can try removing this combination from the model.

A non-linear analysis failure is almost always related to the above-mentioned problem.

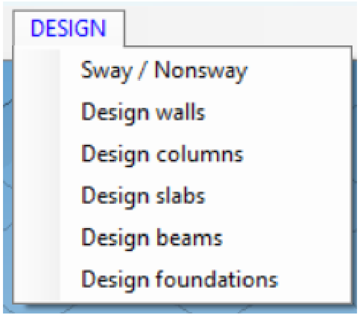
Errors in seismic analysis

The program displays two types of error messages both with the same header: “Error during dynamic analysis”. First, make sure the structure is stable under static analysis.

- In the first type, the message clearly points to insufficient RAM on the computer. The program requires at least 32 GB of RAM (64 GB are recommended) for modal analysis. The solver engine needs to store the stiffness matrix in the physical memory. The user can try to reduce the requested storage by increasing mesh size in walls and slabs. If still the error message persists, the only solution is to add more RAM to the PC.
- The second type of error message proposes a trial-and-error procedure, based on removing one or more stories from the building.

Chapter 4- Design of reinforced concrete members

Design of reinforced concrete members is performed according to ACI 318-19 metric.



The program supports the design of the main structural elements in a building, including columns, walls, slabs, beams, and foundations. ACI 318 requirements are closely followed, and concise reports can be generated in Word format.

In the upcoming versions, the program will be able to create detailed drawings for the designed members, which is of considerable help for the user.

4.1 Sway / Nonsway

When designing a slender column, it is important to identify the sidesway type of the floor to which it belongs. Buckling length calculation and the moment magnification procedure follow distinct sections in ACI 318, depending on the classification of the frame and in particular the concerned story, as sway or nonsway.

The program follows the requirements of section 6.2.5 (ACI 318-19) where columns are considered braced against sidesway if the flexural stiffness of the shear walls in the considered direction is at least twelve times that of the columns, for the considered story. If this condition is not satisfied, the Code asks to run P-Delta analysis (section 6.6.4.3 of ACI 318-19) and to verify that the increase in column end moments due to second order effects does not exceed 5 percent of the first-order end moments.

The criteria $\frac{Stiffness\ shear\ walls}{Stiffness\ columns} < 12$ does not necessarily mean that the columns of the concerned story should be designed as sway, since an additional condition consequent to the non-linear analysis applies. However, the program classifies any story with a stiffness ratio below twelve as sway, leaving the user to decide whether to perform P-Delta analysis and hence adopt the final decision.

In this table, the ratio walls to columns stiffness exceeds twelve for all floors, in both X and Y directions. If some walls constitute a core, it is highly recommended to assign them as part of a core, to prevent the program from considering their individual stiffness instead of that of the core (refer to chapter 2, “assigning walls sections”).

Floor	Sum of col. inertias (X)	Sum of walls inertias (X)	Sum of col. inertias (Y)	Sum of walls inertias (Y)	Ratio X dir.	Ratio Y dir.	Swayness X dir.	Swayness Y dir.
Basement-1	0.062	115.319	0.052	51.445	1853.28	981.89	Non sway	Non sway
GF	0.066	3.497	0.086	2.953	53.26	34.38	Non sway	Non sway
F1	0.066	3.497	0.086	2.953	53.26	34.38	Non sway	Non sway
F2	0.066	3.497	0.086	2.953	53.26	34.38	Non sway	Non sway
ROOF	0.066	3.497	0.086	2.953	53.26	34.38	Non sway	Non sway

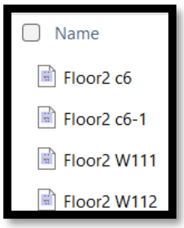
Only flexural stiffness of walls and columns is considered in the computation of the ratios.

4.2 Design of walls

Walls in the selected stories are designed for the chosen combinations. It is possible to select all ultimate combinations when “ENV-ULS+” or ”ENV-ULS-“ is checked. In this case, as shown in the table, once the button “Export to Walls Expert” is clicked, all the created ultimate combinations are listed. If seismic design category is “D”, “E” or “F” then the “Special” option must be selected, so the design is performed according to section 18.10 of ACI 318-25 (Special concrete walls).

The upper table shows the results for all the walls of the selected floors, whereas the lower one is dedicated to the created cores, in case these are created by the user. If a wall belongs to a core, that’s indicated in the last column of the upper table.

Design of walls is performed in Walls Expert software. BUILD-PRO creates a folder where all the walls that pertain to the selected floors are stored with a comprehensive name, as shown in the figure at the right. The first part of the name is that of the story, the second part is that of the wall or of the core in case the user created cores.

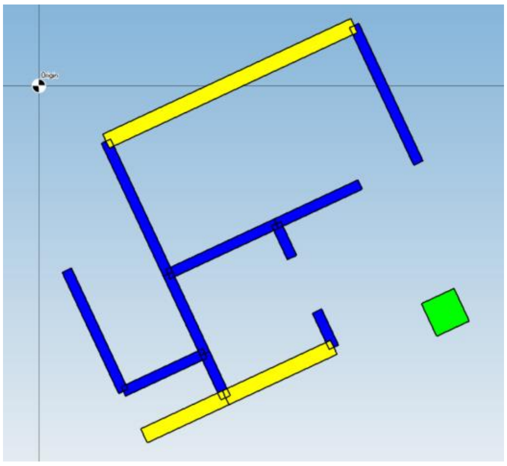


For instance, cores c6 and c6-1 are exported and likely for walls W111, W112… The user can directly open these files with Walls Expert and find all the combinations. The only remaining step is to load the drawing of the wall or that of the core. For more details about Walls Expert, please refer to its manual.

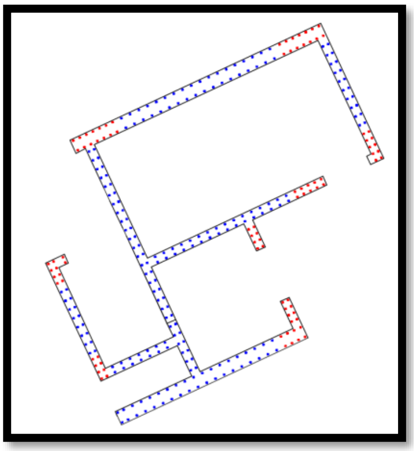
- Design of individual walls considers the axial force, the in-plane moment and the in-plane shear. These forces are concentrated at mid-length of the wall. The in-plane moment is always about the strong inertia of the wall, whatever its orientation is in the plan (X,Y). The in-plane shear is always oriented along the length of the wall, independently of tis orientation in the plan (X,Y).
- Design of cores considers the axial force and the two moments about global X and Y axes, generated at the centroid of the core. These moments are not oriented according to the principal flexural system of the core but calculated about the global X and Y axes. For shear design, Walls Expert considers the in-plane shear force for each wall considered individually, as exported by BUILD-PRO. The values of the shear forces for the individual walls that form the core are not shown in the lower table for clarity reasons.

Once the user opens the file of a wall or a core, the geometry and loads are automatically applied. The remaining task consists of loading the AutoCAD drawing of the wall/core, where the proposed longitudinal reinforcement is drawn. It is important to consider the following:

- For core walls: The core orientation should be the same in BUILD-PRO and in the AutoCAD drawing. The reason is that the triplet (N,Mx,My) is calculated at the centroid of the core, according to the global system (X,Y,Z).

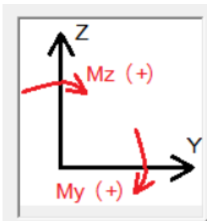


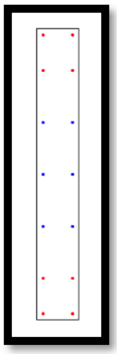
Top view showing the core in BUILD-PRO



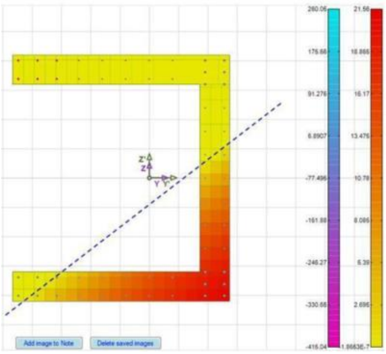
AutoCAD drawing of the core

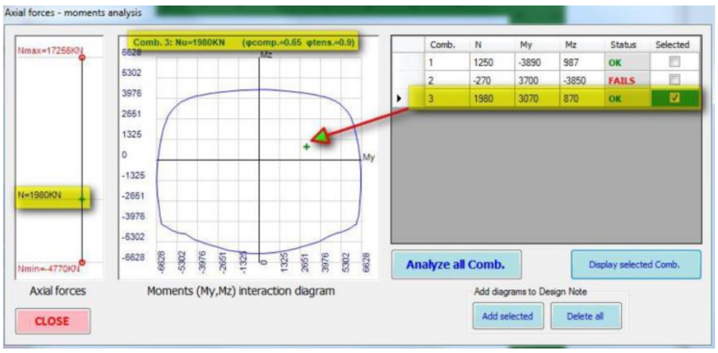
- For individual walls: Whatever is the orientation of the wall in BUILD-PRO, it should always be drawn in AutoCAD as shown in this picture. This way, the in-plane moment is oriented about the “Y” axis of Walls Expert. Walls Expert always assumes that the in-plane moment is My. The figure at the right displays the axes system in Walls Expert.



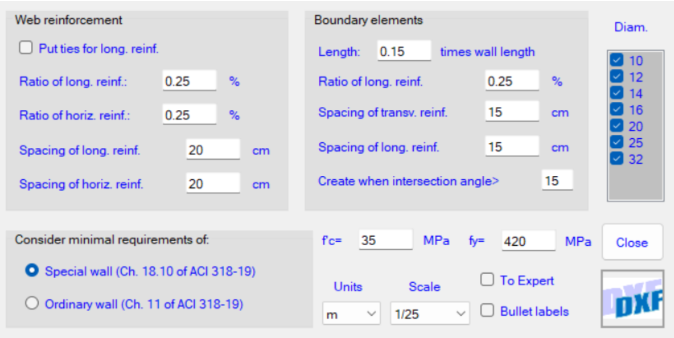


Walls Expert performs a complete flexural and shear design, generates interaction diagrams, and produces a comprehensive report. The core shape can be of any complexity.



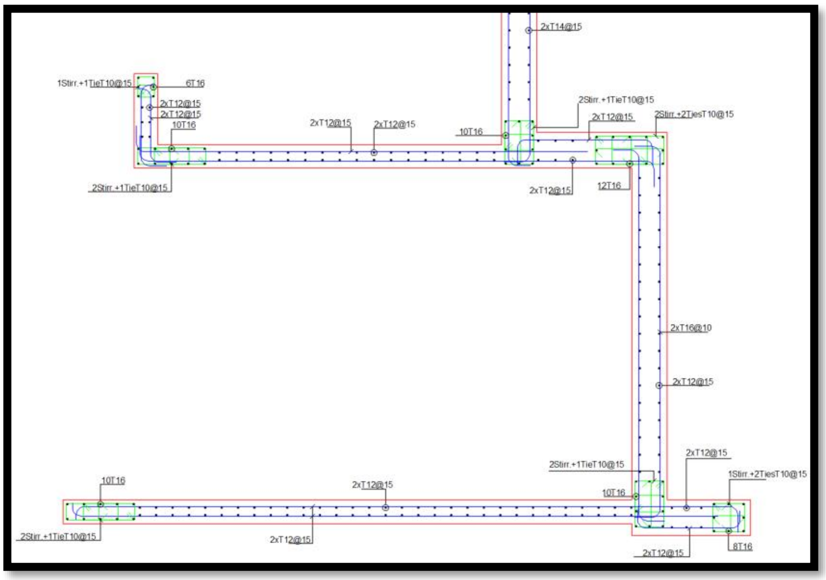


4.2.1 Drawings generation



The program can generate high-quality shop drawings (in DXF format) through a powerful interface that comprises all the required parameters, for any complex wall or core geometry. The user needs to select the storey for which the drawings of all the related walls and cores will be created. The reinforcement is controlled by the user since for the particular case of walls, these are designed in WALLS EXPERT application. The choice of ordinary/special allows the program to apply the minimal Code requirements for longitudinal, horizontal and transverse reinforcement with regards to steel ratios and spacing. The drawings generated comply with the Code's provisions.

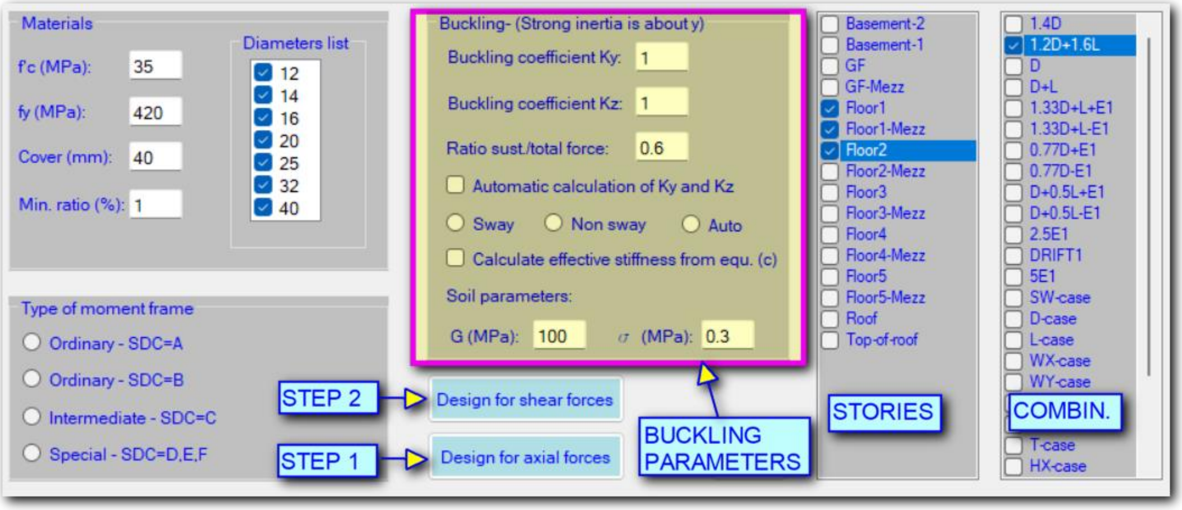
- For web reinforcement, the user can control the spacing of the longitudinal and horizonal reinforcement, and their ratios.
- For the boundary region, the user can specify its length as a fraction of the wall's length, the desired ratio & spacing of the longitudinal reinforcement and finally the spacing of the confinement hoops and ties. The program creates at the end of each wall seismic -or ordinary- boundary regions. If two walls intersect with an angle less than that specified, the program considers these walls as aligned and does not create boundaries.
- Concrete and steel strengths are used to calculate the minimal reinforcement ratio to be placed in boundary regions.
- The use can control the bars diameters to be used.
- If the checkbox "Bullets labels" is checked, the program creates a bar bending schedule table that details the lengths and shapes for all bars, stirrups and ties with total weights.
- If the checkbox "To Expert" is checked, the program creates an additional drawing that can be opened with WALLS EXPERT, ready for design.



Sample of core wall drawing generated by the program

4.3 Design of columns

Columns in the selected stories are designed for the chosen combinations. It is possible to select all ultimate combinations when “ENV-ULS+” or ”ENV-ULS-“ is checked.



Materials

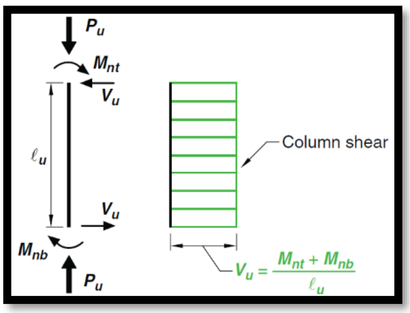
The listed diameters apply to longitudinal reinforcement. Transverse reinforcement diameters start at 10 mm. The program provides real reinforcement based on this list, as shown hereafter. The user can specify the minimal longitudinal reinforcement ratio (1% per default). Cover value is measured to transversal reinforcement.

Type of moment frame

Depending on the seismic design category of the structure (SDC), design and detailing of columns follow separate formulations in ACI 318-19.

- SDC=A → Section 18.3
- SDC=B → Section 18.3.3
- SDC=C → Section 18.4
- SDC=D,E,F → Section 18.7

Seismic design category has a direct impact on the design for shear. For SDC=B and above, the ultimate shear strength ΦV_n is deduced from the free-body diagram with end moments assumed equal to the nominal flexural strength of the column, under the axial force that is present in the investigated combination.



Buckling parameters

The program provides extensive options for calculating column buckling effects. First the unsupported length l_u (clear length of the column) is automatically determined based on the thicknesses of the top and bottom slabs and the eventual presence of drop beams.

The effective length factor k (buckling coefficient) that factors l_u can be introduced by the user or automatically calculated by the program, based on the alignment charts of Jackson and Moreland (figure R6.2.5.1 of ACI 318). "K_y" is the buckling coefficient about the strong inertia I_y.

The effective length factor, when requested to be computed by the program, depends on two settings:

- Structure swayness that is either set by the user for the whole building or defined per floor (the "Auto" mode), based on the results of the assessment performed by the program in section 4.1 of the present document. Swayness estimation should hence be done prior to column's design.
- The stiffness reduction factors for columns, beams and slabs should be assigned prior to analysis, as requested in section 6.6.4.4.3 of ACI 318-19. The alignment charts use the stiffnesses of the frame members to compute the effective length factor k.

The key parameter in buckling analysis is the determination of the critical buckling load P_c.

$$P_c = \frac{\pi^2 (EI)_{eff}}{(kl_u)^2}$$

ACI proposes three expressions in section 6.6.4.4.4 to estimate the effective stiffness $(EI)_{eff}$.

$$\text{(a)}\ (EI)_{eff} = \frac{0.4E_c I_g}{1+\beta_{dns}}$$

$$\text{(b)}\ (EI)_{eff} = \frac{(0.2E_c I_g + E_s I_{se})}{1+\beta_{dns}}$$

$$\text{(c)}\ (EI)_{eff} = \frac{E_c I}{1+\beta_{dns}}$$

Expression (a) is the least accurate since it does not involve the actual reinforcement in the column. The program applies by default expression (b). When the column is of important slenderness (ratio of l_u to width more than 10) the default formulation of the effective stiffness leads to moderate values for the critical buckling load P_c. Consequently, the moments magnification factor δ may exceed 1.4 and condemn the concerned columns. The user has the choice of selecting the use of expression (c) where the effective inertia of the column (here-below) is calculated based on section 6.6.3.1.1(b) where reinforcement, ultimate moments and ultimate axial force are incorporated, for better accuracy of the inertia.

$$\left(0.80 + 25\frac{A_{st}}{A_g}\right)\left(1 - \frac{M_u}{P_u h} - 0.5\frac{P_u}{P_o}\right) I_g$$

The ratio "sust. / total force" corresponds to β_{dns} in the Code; obtained by dividing the axial force of the sustained loads by the total one P_u, in the same considered combination. The program does not calculate this ratio; it considers for all columns the simplified value of 0.6 that the user can modify.

⚠ Soil parameters (shear modulus and allowable bearing capacity) are used to compute the rotational stiffnesses of the isolated foundations under columns, in the absence of rafts. Based on the bearing capacity, the program deduces an estimated size for each footing and calculates rotational stiffnesses from Pais & Kausel expressions (1988). If "Automatic calculation of K_x and K_y" is not selected, the program assigns for all columns the introduced values for the buckling coefficients and hence does not calculate the foundations stiffness.

Design for axial forces

Once the corresponding button is clicked the following table is displayed.

Column	Height-y (m)	Height-z (m)	Floor	Combination	N (KN)	My,b (KN.m)	Mz,b (KN.m)	My,t (KN.m)	Mz,t (KN.m)
37-CR70x70	3.52	3.52	Floor1	1.4D	3262.9	27.8	-47.9	-30.2	43.9
37-CR70x70	3.52	3.52	Floor1	1.2D+1.6L	3234.9	31.3	-52.7	-33.8	49.7
37-CR70x70	3.52	3.52	Floor1	1.33D+L+E1	3511.1	63.5	-9.3	-53.7	8.5
37-CR70x70	3.52	3.52	Floor1	1.33D+L-E1	3236	-1.2	-96.3	-13.4	90.1
37-CR70x70	3.52	3.52	Floor1	0.77D+E1	1932.2	47.7	17.2	-36.7	-16.6
37-CR70x70	3.52	3.52	Floor1	0.77D-E1	1657.1	-17.1	-69.9	3.6	64.9
37-CR70x70	3.52	3.52	Floor1	2.5E1	343.9	80.9	108.8	-50.4	-102
38-CR40x105	3.52	3.52	Floor1	1.4D	4858.4	47.3	-43	-158.5	36.5
38-CR40x105	3.52	3.52	Floor1	1.2D+1.6L	4959.8	47.4	-46.7	-164	40.5

For each combination, the table shows the columns in the selected floors along with their corresponding information, to know:

- Clear lengths about the strong and weak inertias, denoted previously as "l_u". The local system "y" is always oriented about the strong inertia, to prevent confusion. In the presence of drop beam in one direction, the height of the column is not the same for strong and weak inertias. The displayed unsupported lengths $l_{u,y}$ and $l_{u,z}$ will be factored by the corresponding buckling coefficients k_y and k_z to obtain the buckling length. The program deduces the clears lengths based on the thickness of the slabs below and above, with consideration to the depth of the eventual beams. The user is invited to verify these values and modify them if required.
  - Double-height columns are also considered. This case occurs when a column connects not to the slab directly above, but to the next higher one. The program computes the clear height and affects the corresponding moments at top and bottom nodes.
- For each selected combination, the axial force and moments at top and bottom nodes are shown. For instance, "$M_{z,b}$" is the moment about the weak inertia, at the bottom of the column. These figures are accessible for the user to change them before performing the design.

Once the design is performed, the following summary table is displayed.

Column	Combination	Total reinf.	#short	#long	N (KN)	Mc,y (KN.m)	Mc,z (KN.m)	Ky	Kz	λ	N-crit. (KN)	δ	Ratio
37-CR70x70	1.33D+L+E1	20T20-1.28%	6	6	3511.1	63.5	9.3	1	1	17.42	93094	1	0.397
38-CR40x105	1.33D+L-E1	20T20-1.5%	4	8	5342.9	176.2	86.9	1	0.92	28.05	31602	1	0.688
39-CR40x105	2.5E1	20T25-2.34%	4	8	-1843	142.2	135.1	1	0.9	27.44	40820	1	0.498
40-CR40x105	1.33D+L-E1	20T20-1.5%	4	8	4389.7	75.9	65.5	1	0.92	28.05	31602	1	0.565
41-CC70x70	1.33D+L+E1	15T20-1.22%	0	0	4735.9	49.5	57.6	1	1	20.11	53420	1	0.686
42-CR40x120	1.33D+L-E1	24T16-1.01%	4	10	4395.4	35.5	113.2	1	0.93	28.35	30795	1	0.523
43-CR40x120	1.33D+L+E1	24T16-1.01%	4	10	5219.2	179.4	7.8	1	0.93	28.35	30795	1	0.621
44-CR90x90	1.33D+L-E1	24T25-1.45%	7	7	4456.8	99.6	63.4	1	1	13.55	276644	1	0.299
45-CR90x90	1.33D+L-E1	24T25-1.45%	7	7	5128.8	108.5	93	1	1	13.55	276644	1	0.344
148-CR40x215	1.33D+L-E1	38T20-1.39%	4	17	6713.6	698.9	48	1	1	30.48	55297	1	0.427
46-CR70x70	1.33D+L+E1	20T20-1.28%	6	6	3207.1	69.4	30.2	1	1	17.32	94161	1	0.362
47-CR40x105	2.5E1	20T20-1.5%	4	8	-488.4	50.4	185.3	1	0.87	26.37	35744	1	0.598

For each column, the governing combination (that with the highest working ratio) is selected. The reinforcement ratio and the arrangement of the reinforcement are shown. For instance, column #38 is reinforced with 8T20 along its long side and 4T20 along its short one, making a total of 20 bars. The axial force corresponding to the critical combination is shown, together with the magnified moments $M_{c,y}$ and $M_{c,z}$. The buckling coefficients k_y and k_z (about strong and weak inertias) intervene in the calculation of the slenderness $\lambda = \frac{k*l_u}{r}$; the program displays in the table the bigger slenderness and bigger magnification factor δ although these are calculated for both directions. "N-crit." Is the critical buckling force, calculated according to expression (b) or (c). The working ratio is the maximum of axial and moment demand/capacity; when it exceeds one, it means the section fails.

Design for shear forces

Shear design depends on the longitudinal reinforcement, so the user must complete the axial load and moment design first. When the seismic design category is B and above, the shear capacity of the column is governed by the nominal moments which are related to the longitudinal reinforcement.

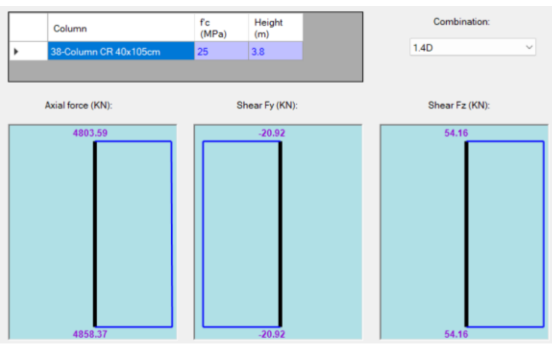
For each column, the critical combination is shown; it corresponds to the maximal density of transversal reinforcement obtained among all combinations. As it may frequently happen, transversal reinforcement can be governed by minimal code requirements like seismic dispositions. In such case, several combinations will require identical amount of reinforcement and the combination shown in the table is not necessarily the "critical" one.

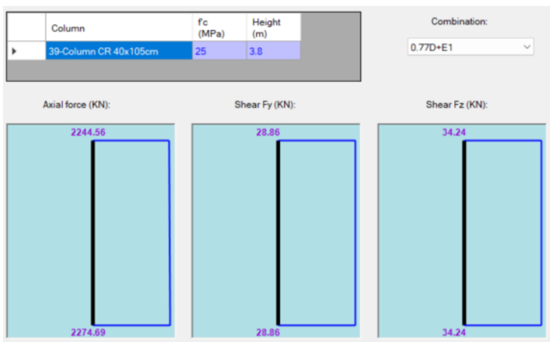
- V_y: Shear parallel to the short side. V_z: Shear parallel to the long side.

Column	Combination	N (KN)	Vy (KN)	Vz (KN)	Mny (KN.m)	Mnz (KN.m)	Vcy (KN)	Vcz (KN)	Vny (KN)	Vnz (KN)	Avy - L0	Avy - rem.	Avz - L0	Avz - rem.	L0 (m)
37-CR70x70	1.4D	3262.9	24.2	15.3	1522.3	1522.3	947.8	947.8	947.8	947.8	6T10@16cm	6T10@16cm	6T10@16cm	6T10@16cm	0.7
38-CR40x105	1.4D	4858.4	20.9	54.2	2273.5	911.3	983.8	1091.3	983.8	1091.3	8T10@16cm	8T10@16cm	4T10@16cm	4T10@16cm	1.05
39-CR40x105	0.77D+E1	2274.7	521.5	1478.9	2381.1	917.8	676.3	753.8	1566.9	2056.8	8T10@10cm	8T10@10cm	4T10@10cm	4T10@10cm	1.05
40-CR40x105	1.4D	4087.1	1.7	13.7	2209.1	878.7	938.1	1040.5	938.1	1040.5	8T10@16cm	8T10@16cm	4T10@16cm	4T10@16cm	1.05
41-CC70x70	1.4D	4383.9	2.2	14.6	1120	1121.1	1080.2	1080.2	1080.2	1080.2	2T10@16cm	2T10@16cm	2T10@16cm	2T10@16cm	0.7
42-CR40x120	1.4D	4264.5	50.5	4.8	2480.6	866.6	1020.4	1135.8	1020.4	1135.8	10T10@12cm	10T10@12cm	4T10@12cm	4T10@12cm	1.2
43-CR40x120	1.4D	4632.9	7.5	71.9	2540.1	889.7	1072.9	1194.2	1072.9	1194.2	10T10@12cm	10T10@12cm	4T10@12cm	4T10@12cm	1.2
44-CR90x90	1.33D+L+E1	3624.2	1742	1742	3066	3066	1320.2	1320.2	3176.7	3176.7	7T12@14cm	7T12@14cm	7T12@14cm	7T12@14cm	0.9
45-CR90x90	1.33D+L+E1	4931.9	1904.6	1904.6	3352.1	3352.1	1523	1523	3512.1	3512.1	7T12@13cm	7T12@13cm	7T12@13cm	7T12@13cm	0.9
148-CR40x215	2.5E1	-384.9	436.8	2489.3	4381.2	768.7	680.7	778.4	680.7	3536.1	17T10@10cm	17T10@10cm	4T10@10cm	4T10@10cm	2.15
46-CR70x70	1.4D	2990.1	38.1	15.2	1478.9	1478.9	906.2	906.2	906.2	906.2	6T10@16cm	6T10@16cm	6T10@16cm	6T10@16cm	0.7
47-CR40x105	1.4D	4172.4	8.9	40.7	2218.1	882.9	950.1	1053.9	950.1	1053.9	8T10@16cm	8T10@16cm	4T10@16cm	4T10@16cm	1.05

Let's consider columns #38 and #39 which are respectively reinforced with 20T20 and 20T25, as shown in the design for axial loads table. Since SDC=C, for each seismic combination the design shear force is that corresponding to the flexural nominal capacity: $V_u = \frac{M_{nt}+M_{nb}}{l_u}$ which is in general much bigger than that given by the combination itself. For instance, the shear forces in column #39, present in combination 0.77D+E1 (where E1 is the seismic case) are shown in the below figure at the right. These are of moderate magnitude in comparison with the design shear forces shown in the table here-above, which are calculated based on the flexural nominal capacity diagram.

V_y and V_z are not necessarily the shear forces present in the displayed critical combination. SDC requirements for seismic combinations lead to the computation of shear from the free-body diagram with end moments assumed equal to the nominal flexural strength of the column

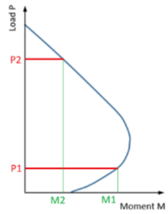




Column #38: forces from comb. "1.4D" *Column #39: forces from comb. "0.77D+E1"*

One can note, however, that the critical combination for column #38 is the gravity one where design shear forces V_y and V_Z correspond to those given by the combination itself. The reason for that is the low longitudinal reinforcement ratio (20T20) for which the corresponding nominal moment leads to transversal reinforcement less than the minimal Code requirements. Hence, the program selected the gravity case as the critical one. The higher reinforcement of column #39 (20T25) leads to a bigger shear force since the nominal moment is more important; design shear force parallel to the long side becomes V_z= 1478.9 KN, exceeding the shear capacity of concrete. The program reduced the spacing of the hoops to 10cm in order to obtain a nominal shear strength V_{nz}=2056.8 KN, ensuring that $\Phi V_{nz} > V_z$.

M_{ny} and M_{nz} are the nominal moments about strong and weak inertias. These are displayed for the critical combination, even if it is not a seismic one. For instance, the nominal moments under combination 1.4D for column #38 as shown in the table are not in fact used to compute design shears since it's a gravity combination.



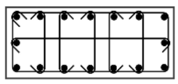
It is fundamental to understand that nominal moments are not the same for all combinations since they are calculated for the corresponding axial force that is present in the combination. As shown in the figure, a combination with a low compression P1 may -or may not- lead to a nominal moment M1 much bigger than that resulting from a combination with a higher compression force P2.

For column #39 where the critical combination is seismic, the axial force N=2274.7 KN is used to compute the corresponding nominal moments M_{ny} and M_{nz} respectively about strong and weak inertias, leading to the obtention of the design shear forces V_z and V_y respectively along long and short sides of the column.

V_{cy} and V_{cz} correspond respectively to the shear capacities of concrete alone, along short and long sides of the column. By comparing, for instance, V_{cy} of columns #38 and #39, it may appear singular to not see the same value, since both columns have same dimensions. The reason is that the program uses equation "a" of table 22.5.5.1 (ACI 318-19) that accounts for the contribution of the axial force. This explains the lower values of V_c for column #39 for the displayed combinations.

The user can sense the intricate nature of design for shear in the particular case of columns.

V_{ny} and V_{nz} are the nominal shear strengths of the section respectively along short and long sides. When transversal reinforcement is not required, the nominal strength equals that of concrete V_c.

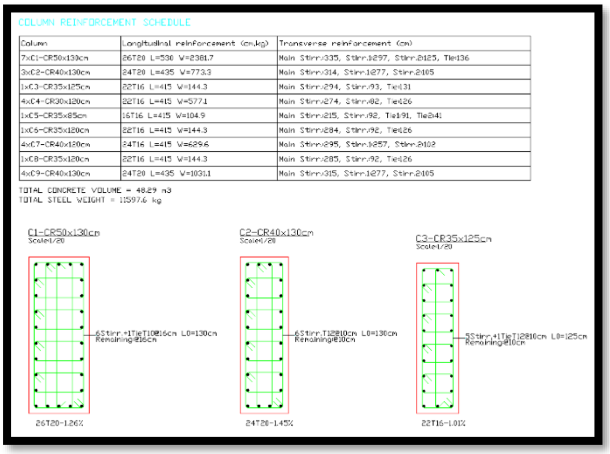


A_{vy}-L_0 and A_{vz}-L_0 give the number of legs along short and long sides, within the critical length L_0. The program assumes a fixed pattern where each longitudinal bar is tied to the opposite one, and this all over the height of the column. Hence, the number of legs along short and long sides corresponds to that of longitudinal bars at these faces. The given spacing is required all over the critical length L_0 that is calculated in the last column of the table.

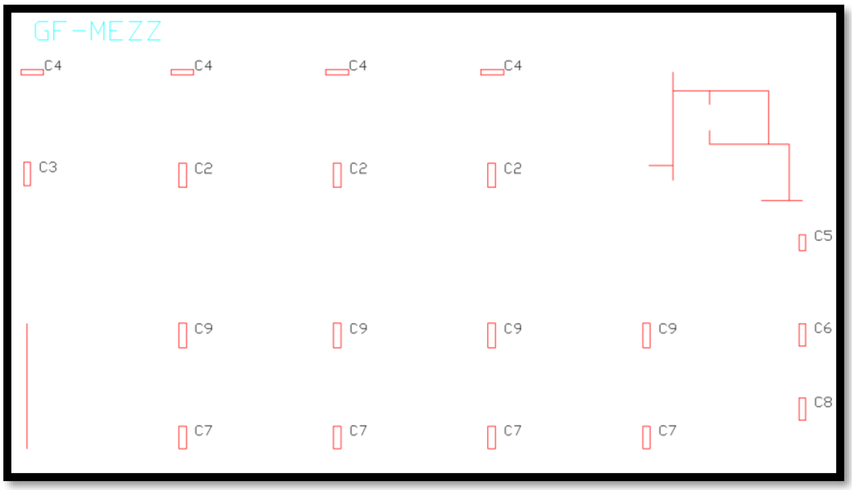
A_{vy}-rem and A_{vz}-rem provide the transversal reinforcement in the remaining length of the column, i.e. outside the critical lengths L_0. When seismic combinations are present and design shear is computed from nominal moments, the spacing of the transversal reinforcement may remain the same all over the length. Shear diagram in the free-body model is constant, imposing equal reinforcement area in both the critical zone and outside it.

For circular columns (refer to that with label 41 in the table), the program assumes that one circular hoop is placed, which leads to the consideration of two legs in the design. When columns are part of special moment-resisting frame (SDC=D,E or F), the program assumes that additional 2 ties are added to the hoop, in both directions.

4.3.1 Drawings generation



The program can generate high-quality shop drawings (in DXF format). Once, for the selected floors, the design for longitudinal and transversal reinforcement is performed, the program groups all the columns according to their common geometry, reinforcement and height. Identical columns receive the same label and hence have the same detail. A key plan is generated for each floor, showing the generated labels. It is possible hence, with one click, to generate the details of all the columns in the building. Reinforcement quantities consider splicing length of longitudinal bars at floors and the exact geometry of stirrups and ties. The spacing of transversal reinforcement respects the provisions of the Code depending on the selected seismic design category.



Key plan showing the automatic labeling of the columns

4.4 Design of slabs

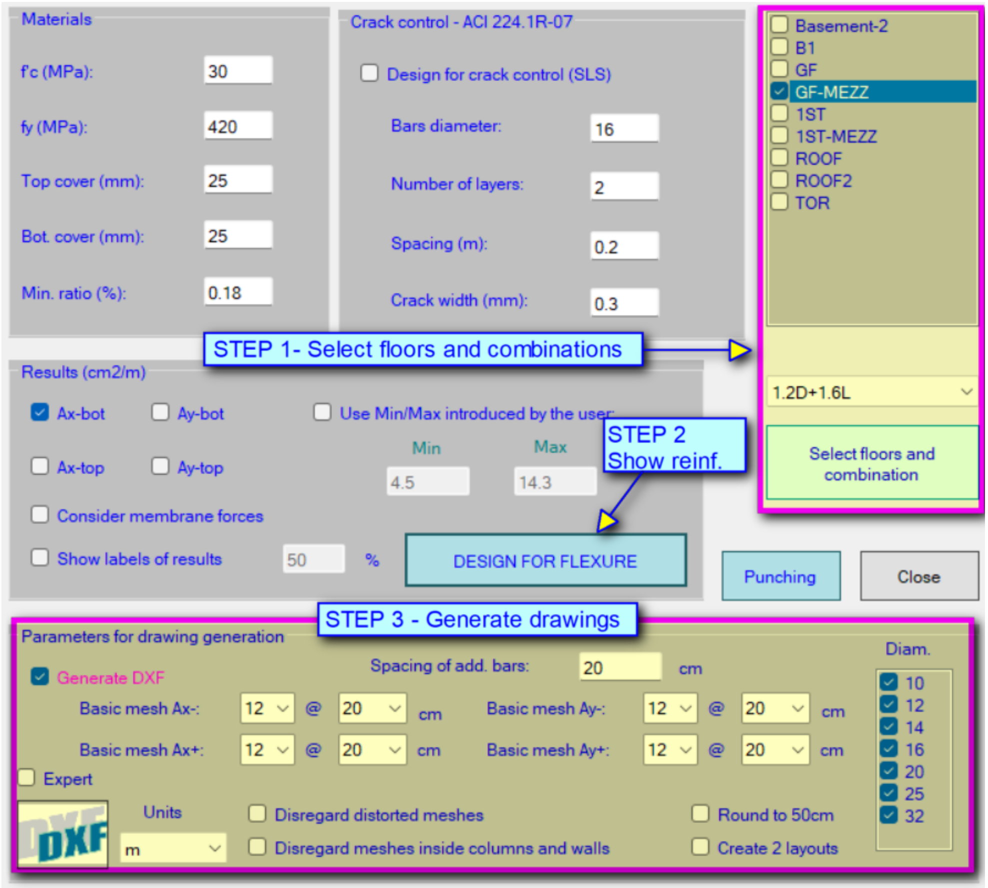
4.4.1 Design for flexure and membrane forces

The design process consists of three steps.

- Step 1: Select the floor and the combination. All slabs that belong to the floor will be designed. When “ENV-ULS+” or ”ENV-ULS-“ is selected, all ultimate combinations are considered.
- Step 2: Once material properties are defined, the user should select the reinforcement values to be calculated (top or bottom, x or y directions) and click on the “Design” button.
- Step 3: In case the user wishes to generate drawings.

Whenever the user changes the floor or combination, the “Select floors and combinations” button must be clicked.

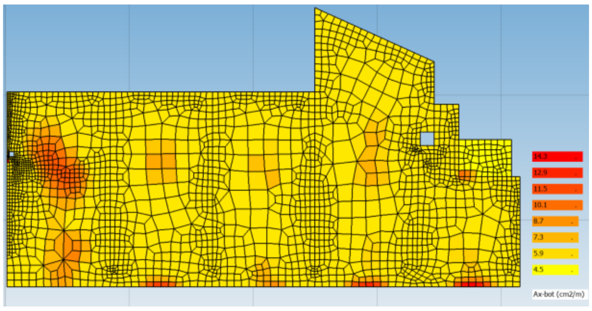
Once floor and combination are selected, the “Design” button is clicked to display the chosen reinforcement values.

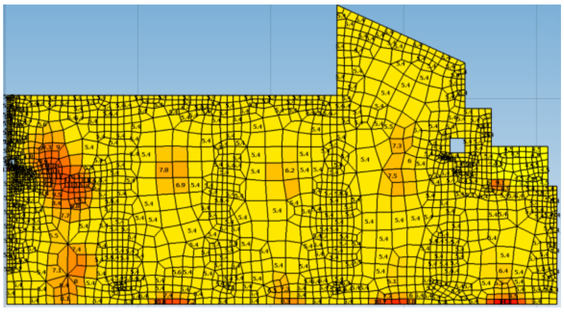


The orientation of the “Ax” reinforcement is defined during the creation of the slab. Refer to “Assigning slabs sections” in chapter 2.1 of the present document.

When “Consider membrane forces” is selected, the program incorporates tension axial forces in the design of reinforcement. Tension forces are substantial in case the combinations involve temperature or seismic forces.

The reinforcement is shown by the mean of colored maps, with a legend. The user can display labels (reinforcement values) inside the centers of the finite elements.



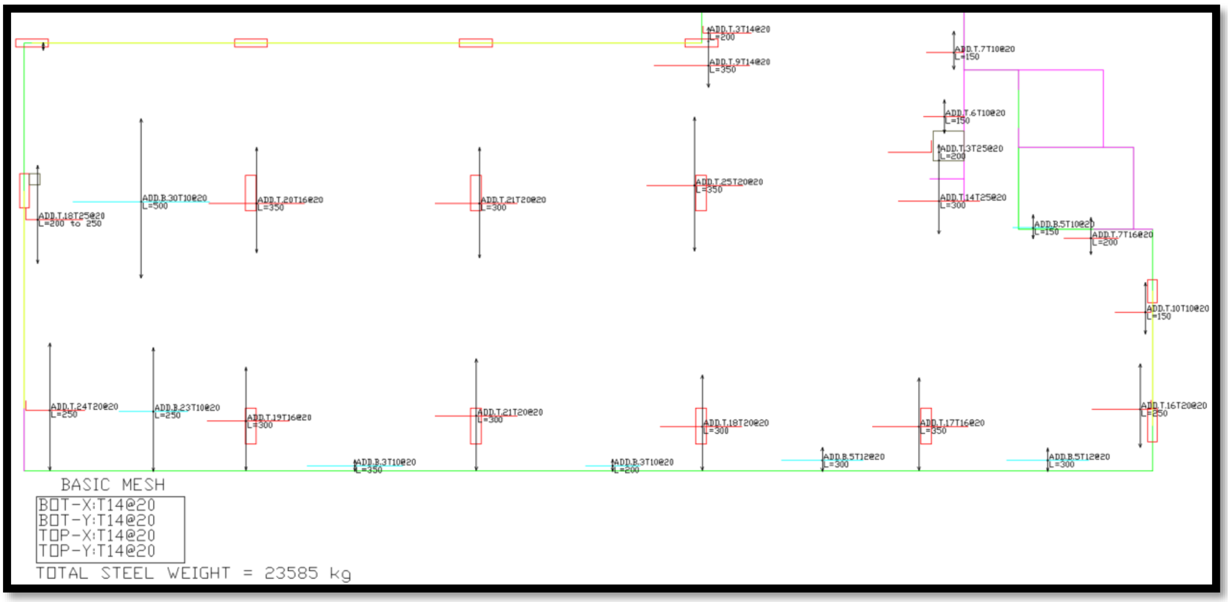


It is possible to control the colors range of the displayed map by defining limits to the maximal and minimal reinforcement values. Meshes where the reinforcement is beyond the defined interval are shown with gray color.

If the slab is exposed to corrosive environment, the user can select a service combination (commonly "D+L") and specify the crack width for which the reinforcement will be determined. The program performs the design according to the provisions of ACI 224.1 R-07, by calculating the required reinforcement that satisfies the imposed crack width limit.

4.4.1.1 Drawings generation

Before generating the drawings, the user should first calculate the required reinforcement Ax and Ay for both top and bottom layers (refer to step 2 above).



The program generates the provided reinforcement and its real total weight. The built algorithm -unique in the market- reads the required reinforcement from the maps, and uses several criteria's that involve the arithmetic mean, count weighting measures, area weighting measure, RMS indicator, percentiles interpretations, and maximum cap bounds to create human-like reinforcement with the correct length and lateral extent. Bars terminating at slab and opening edges are hooked.

- The user needs first to specify the desired basic mesh (bar diameter and spacing) and the spacing of the additional reinforcement that the program will provide for top and bottom layers.
- The program selects from the diameters list the lowest diameters that satisfy the demand balance. The demand balance is equal to the required total reinforcement area, minus that provided by the basic mesh. The corresponding additional reinforcement bars will be created.

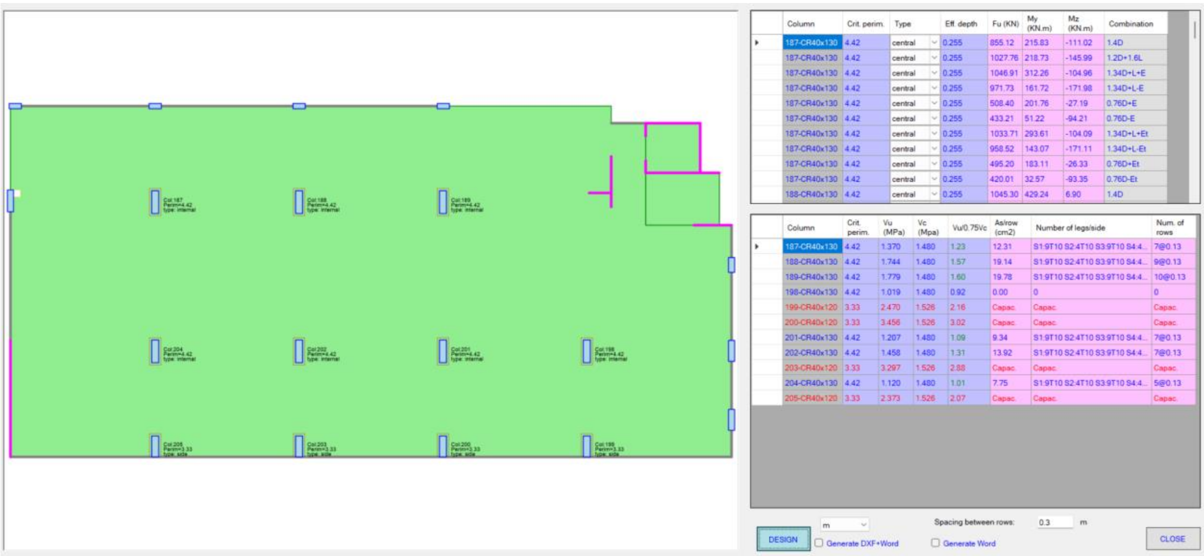
The program provides two important features that help in producing realistic reinforcement.

1. The checkbox “Disregard distorted meshes”, once checked, prevents the program from reading reinforcement area inside meshes which shape is noticeably different from the reference square element. For instance, meshes where angle at a corner is less than 20 degrees are disregarded. Furthermore, when sides ratio exceeds three, the quality of the result deteriorates. Distorted meshes are responsible of the “jumps” that the user encounters while interpreting results.
2. Meshes inside the perimeter of columns and walls need not to be considered. If the related checkbox is active, the program excludes meshes that are located within these areas.

- ❖ The “Round to 50cm” feature rounds-up bars length to 50cm.
- ❖ Per default, the program generates four separate layouts of reinforcement to cover Ax+, Ax-, Ay+ and Ay-. If the checkbox “Create 2 layouts” is active, only two layouts are generated: One for {Ax+,Ax-} and one for {Ay+,Ay-}.
- ❖ If the checkbox “Expert” is active, the program creates a second DXF drawing that is intended to be opened with SLABS EXPERT. This drawing reflects the additional reinforcement provided and hence saves a huge amount of time to introduce them from scratch in SLABS EXPERT. The user needs only to set the materials characteristics, the basic mesh, and the loads before calculations. SLABS EXPERT calculates the cracked long-term deflection based on the provided reinforcement. It helps the user to complete the design of the slab.

4.4.2 Design for punching shear

To perform design for punching shear, the user should first select one floor and the combination. If “ENV-ULS+” or “ENV-ULS-“ is selected, all ultimate combinations are considered.

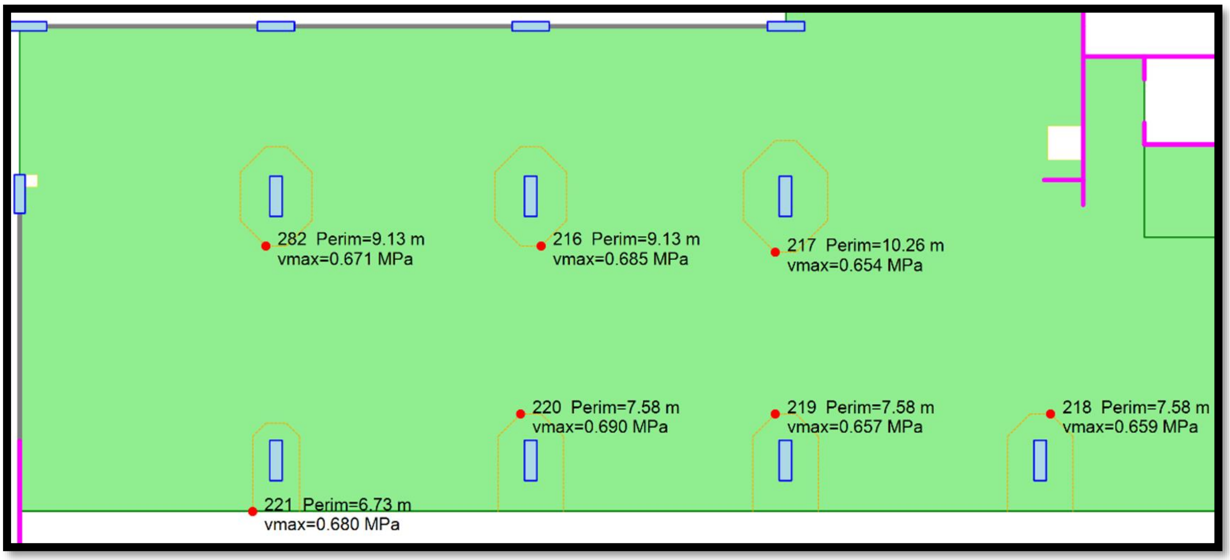


Punching shear design follows the requirements of ACI 318-25 and ACI 421.1R-20. Central, side and corner columns are automatically identified; the inner critical perimeter (located at d/2 from that of the column) is generated with regards to slab and opening edges. The upper table shows the input data that is used for the design. The slab layout shows the critical perimeters (figure here-above). Columns attached to beams or walls are not designed for punching shear.

Based on the shear force and the unbalanced moments, the program calculates the maximal shear stress V_u at the corners of the critical perimeter. If it exceeds the ultimate strength of the section ($V_u > \Phi V_n = \Phi * 0.5 * (f'c)^{0.5}$), the slab thickness shall be increased. In this case, the column’s results are highlighted in red, with the word “Capac.”, signaling the failure case. Eventually, the user can modify the forces shown in the upper table and investigate further the design.

If $V_u > \Phi V_c$, where V_c is the shear stress capacity of concrete, then reinforcement is required, provided as peripheral rows of ties; each row holding the same number of ties. The ratio $V_u/\Phi V_c$ is in this case shown in the table with green color. The program displays for each side of the critical perimeter (numbered S1 to S4 in case four sides are involved) the required number of ties to be placed.

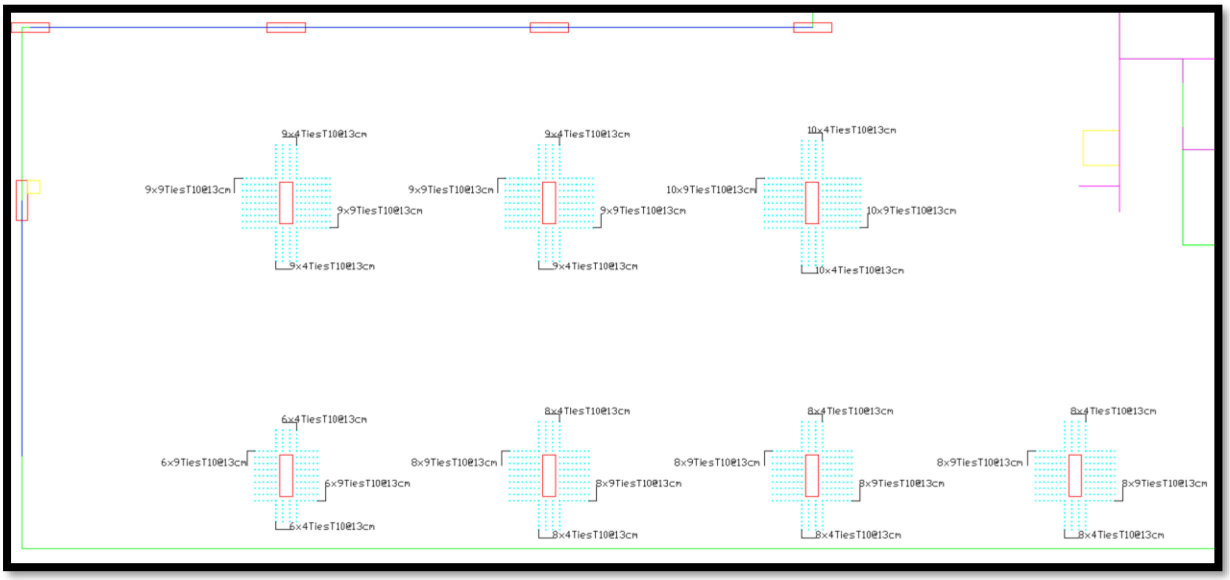
The number of rows is deduced from the outer critical perimeter (octagonal shape). The program iterates on the perimeters to reach that where the shear stress is less than the limit specified by the Code. The required number of rows is shown in the last column of the lower table.



Extract from the design report: Outer perimeters with their length and maximal shear stress

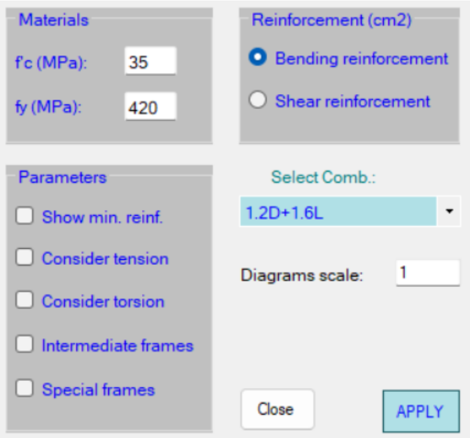
4.4.2.1 <u>Drawings generation</u>

It is possible to generate a DXF drawing that shows the layout of the ties.



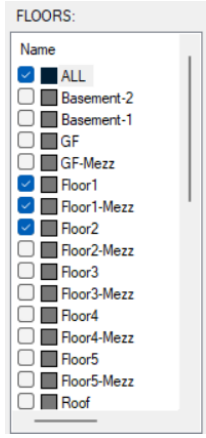
4.5 Design of beams

4.5.1 Design for bending, shear and torsion



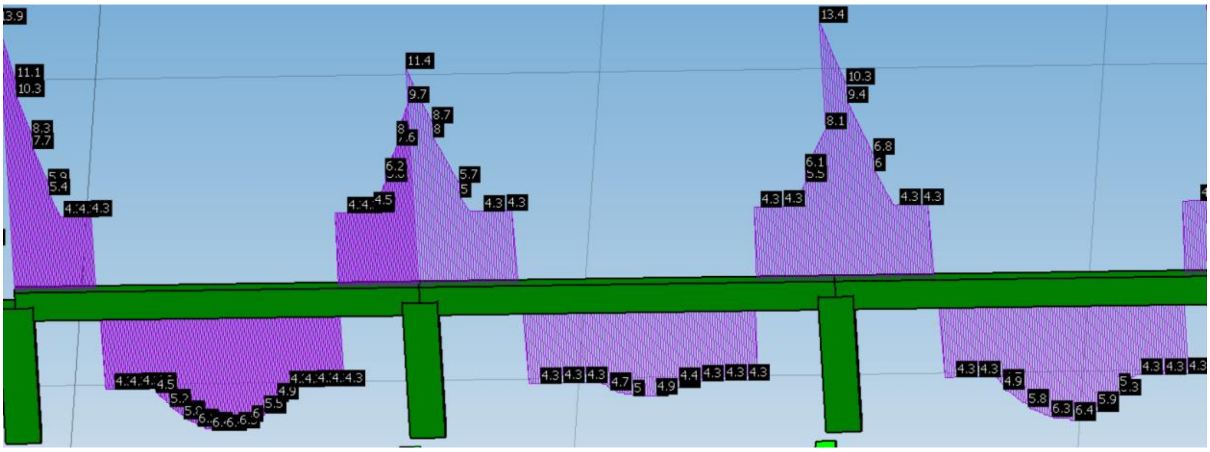
Design of beams is performed at ULS in the actual version of the program.

Once the concerned floors are selected, the user can perform design for flexure and shear, under the selected ultimate combination. If “ENV-ULS+” or ”ENV-ULS-“ is chosen, the calculated reinforcement corresponds to the envelope of all ultimate combinations.



Bending reinforcement

Top and bottom area (cm^2) of longitudinal reinforcement are displayed as diagrams, where that above the beam denotes top reinforcement, and that below, bottom reinforcement. In addition, the user can include in the design the effect of tension axial forces. If “Show min. reinf.” is selected, the program will respect the Code requirement for minimum reinforcement. When the maximal reinforcement ratio is exceeded, the appropriate message is displayed.



Shear and torsion reinforcement

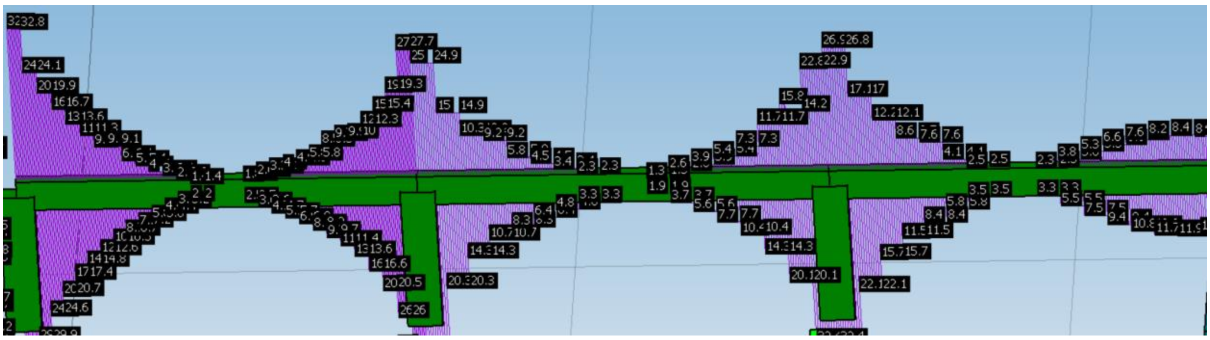


Diagram for shear reinforcement is displayed above the beam. It corresponds to the density per linear meter of transversal reinforcement (cm^2/m). The program automatically considers the Code-minimal reinforcement requirements, depending on the value of the shear force. When torsion moments are included, a new diagram is shown below the beam; it corresponds to the required longitudinal reinforcement. In this figure, the bottom diagram displays the area of the required longitudinal reinforcement (cm^2) due to torsion.

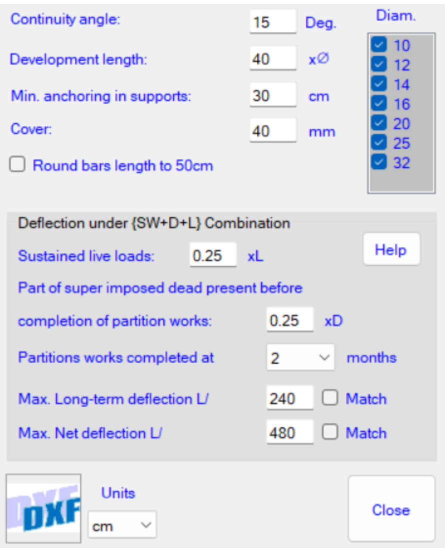
- It is important to note that torsional moments have a direct impact on transversal reinforcement. Torsional strength is provided by additional transversal and longitudinal reinforcement.

When the intermediate or special moment-resisting frames option is selected, the program adds to the shear that is present in seismic combinations the part computed from the free-body diagram, assuming nominal moments at the ends of the beam (intermediate frame) or probabilistic moments (special frame). The related Code sections are 18.4.2.3 and 18.6.5. Seismic dispositions affect only shear reinforcement.

When the beam fails for shear, the appropriate message is displayed.

4.5.2 <u>Drawings generation and deflection calculation</u>

4.5.2.1 <u>Drawings generation</u>



The program offers a powerful tool that generates high-quality shop drawings once the design task is completed. Beams that belong to the concerned floor are first translated into the formwork configuration, with identification of their supports (columns, walls and supporting beams).

If a chain of beams with a continuity angle less than that specified (for instance 15º as per the input in the form at the left) the chain is translated into a continuous beam of several spans. If the intersecting angle exceeds 15º, beams are dissociated.

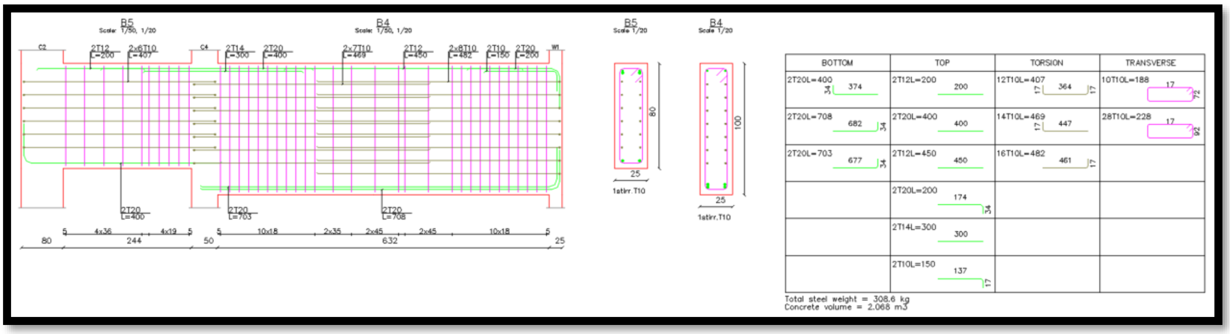
The program “reads” top and bottom longitudinal reinforcement independently from this criterion. A beam can have a single span with top reinforcement at supports. Each span is divided into stations that correspond to the slab’s nodes which are superposed to the beam segment. At each station, the program identifies the longitudinal and transversal reinforcement areas: That’s the demand, the required reinforcement. The provided reinforcement is based on the demand of the stations that are located outside the perimeter of columns and walls, as follows:

- Bending longitudinal reinforcement considers the demand located within the clear span
- Torsional longitudinal reinforcement considers the demand located within the clear span
- Shear transversal reinforcement considers the demand that starts from a distance equivalent to the span effective depth from face of supports.

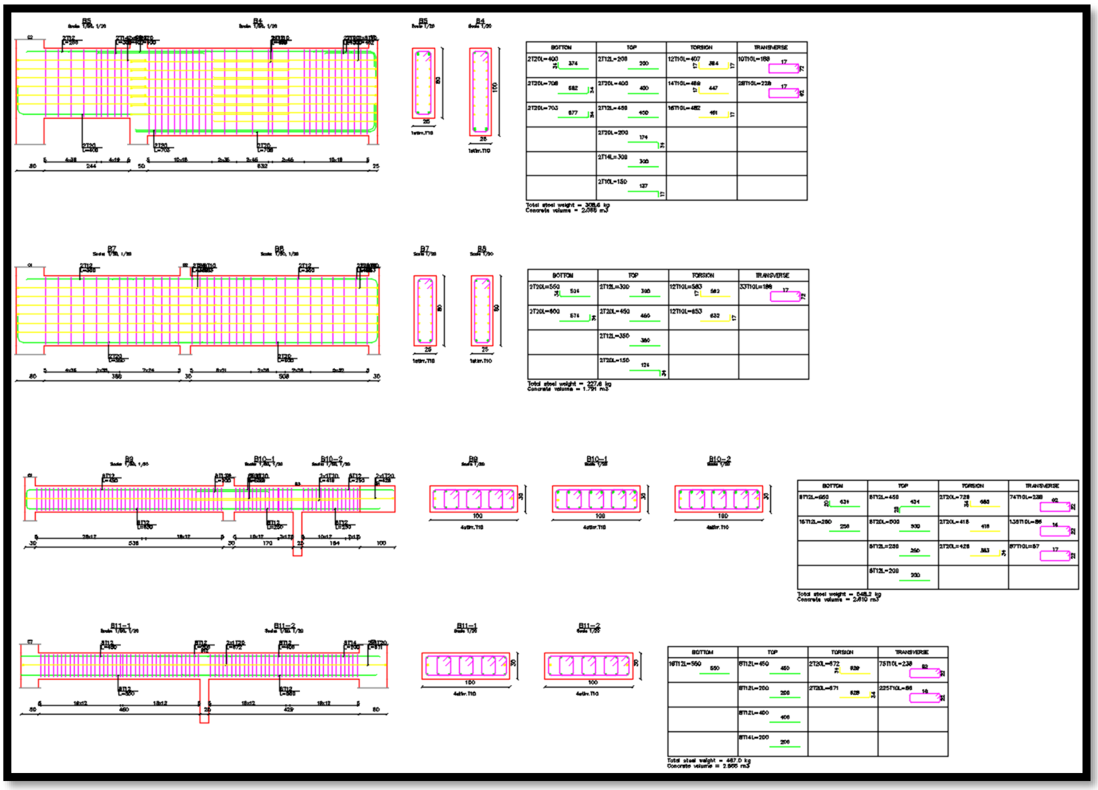
Furthermore, the program implements several Code-related features:

- At end supports, if the straight development length does not fit, standard 90º-hook is used. If the height of the beam cannot accommodate it, it is replaced with a 180º-hook.
- At intermediate supports, the program calculates the required development length for bottom reinforcement, with a minimum of 30cm. If seismic dispositions are set, the bars are fully developed.

- Longitudinal torsion reinforcement is always developed as per Code requirements. If the straight development length does not fit within the support, a horizontal 90º-hook is used.
- Bottom and top reinforcement are curtailed as per the Code provisions. Lengths of the second and higher-order layers are hence calculated and do not follow approximate rules.
- Skin reinforcement is provided when the depth of the beam exceeds 90cm.
- Transversal reinforcement follows the demand diagram. The program starts by selecting the smallest diameter (Φ10mm) and calculates the required spacing between rows. If this spacing is too small, a bigger diameter is selected. When seismic dispositions are required, the program follows the minimal spacing requirements of the Code.
- For each beam, an elevation with section on each span is generated. A detailed bar bending schedule displays, for each type of reinforcement, total and partial lengths. Steel and concrete quantities are shown at the bottom of the table.

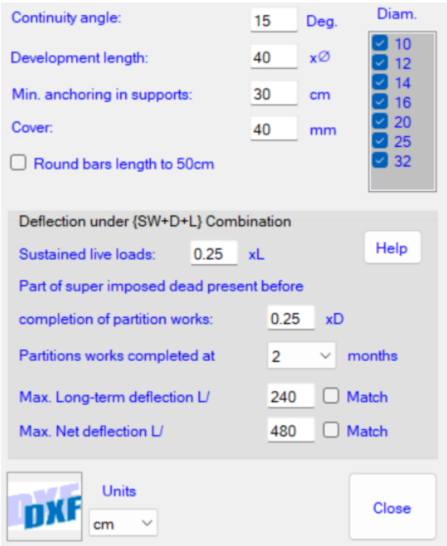


Sample of the shop drawing of a beam – as generated by the program



Series of beam as generated by the program

4.5.2.2 Deflection calculations



The “DXF” button generates shop drawings for all beams that belong to the selected story. Simultaneously, calculations of cracked time-dependent deflections are performed and detailed in a Word report that shows diagrams and tabulated results.
Considering the elastic displacement of the supports, it is challenging to compute deflections from a 3D model. The implemented algorithm starts by identifying the vertical settlements of the supports under the service combination (SW+D+L) and establishes a reference line from which the deflections are reported. The purpose is to provide the relative displacement and not the absolute one, in order to compare it to the Code limits.
The provided reinforcement (top & bottom) and the service moment are used to compute the effective inertia at each station. The double integration of the curvature over the whole beam gives the deflection diagram, with a new redistribution of the service moments. This iterative process will converge to the final instantaneous “cracked” deflection.

Several load cases are considered:

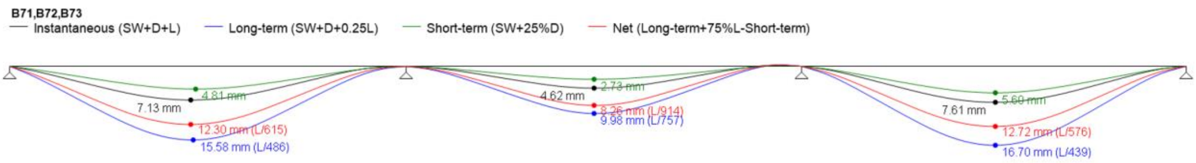
- Comb1: “SW+D+L”
- Comb2: “SW+D+αL”
- Comb3: “SW+βD”

- The coefficient α represents the fraction of live loads that is sustained.
- The coefficient β represents the fraction of imposed dead loads that is executed at the time where partition works are completed.
- The user needs to specify the time “t_1” at which partition works are completed.
  - At 1 month, creep coefficient is 0.5
  - At 2 months, creep coefficient is 0.75
  - At 3 months, creep coefficient is 1
- The program assumes that after five years, the creep coefficient reaches a maximum of 2.

- Case LT: Deflection at 5 years (long-term) is calculated with Comb2 at t=5 years.
- Case ST: Short-term deflection corresponds to Comb3 at t_1.
- Net deflection: Case LT - Case ST + {Comb1-Comb2}.

Long-term deflection is usually compared to the Span/240 limit. In the presence of brittle partitions, the net deflection should be compared to Span/480.

The program displays the obtained deflection diagrams with the maximum values for each span.



The ratio of compression reinforcement ρ’ is considered when computing the time-dependent amplification coefficient $\lambda = \frac{\zeta}{1+50\rho'}$.

Control of deflections

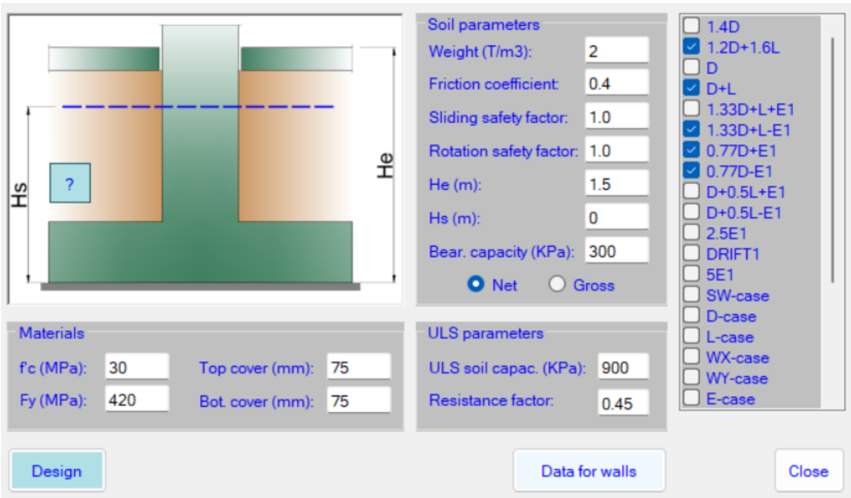
If the “Match” checkbox is active and the obtained maximal deflection is less than the specified limit, the ULS provided reinforcement (tension and compression) is increased to improve the flexural stiffness of the concerned span. This process is automated by the program. The drawings generated reflect the final state of the reinforcement provided.

The adjustment process consists of increasing the diameters of the original reinforcement that is obtained from ULS design and adding new layers. This does not guarantee the obtention of deflections satisfying the imposed criteria.

4.6 Design of foundations

Design of shallow isolated footings under columns and walls is fully performed by the program, based on an innovative analytical-finite element concept.

- Footings under columns: Axial forces and moments about strong and weak inertias of the column are considered, in addition to the contribution of the shear forces along both sides.
- Footings under walls: Only forces acting about strong inertia are considered, to know: Axial force, in-plane moment, in-plane shear. The program assumes that out-of-plane effects are negligible.



The hybrid analytical-numerical formulation that is incorporated in the program is based on the finite element modeling of the footing and the neck above, with shell elements. Elastic springs -which stiffness can be introduced by the user- are affected to the footing. Forces are assigned at the top of the neck. The analysis gives the distribution of the soil reaction, moments and shears in the footing. That's where the numerical analysis ends. The obtained forces are then computed according to the analytical equations of ACI 318-19, providing a design in full compliance with the Code, with regards to flexure, shear and punching shear.

This method is particularly efficient in presence of bi-axial bending, when eccentricities lead to uplift.

- "He" is the excavation depth, calculated from the top of the ground slab to the bottom face of the footing. The weight of this volume is calculated based on the introduced density of the soil, disregarding that of the concrete that pertain to the footing. The excavation depth intervenes in the following cases:
  - If gross bearing capacity is introduced, weight of backfill is considered in the calculation of the soil reaction as an additional load.
  - Stability analysis for sliding and rotation incorporates the weight of backfill.
  - In the case of uplift, weight of backfill is used to calculate the top reinforcement in the footing. This latter is acting like a cantilever loaded with backfill.
  - It is recommended to introduce a realistic value for "He", avoiding depths less than the thickness of the footing.
- "Hs" is the assumed level of the supports (boundary conditions) as introduced in the 3D model. The origin is the bottom face of the footing. If Hs=0, the program assumes that the level of the supports in the model corresponds to that of the foundations to be executed. If Hs>0, the program interprets this case as if the real foundations are deeper, and hence an extended neck is required. Additional moments are added to all combinations, equal to the shear forces times Hs. In presence of tie beams linking columns together, shear forces at the base of the columns will be transferred into the beams as axial forces, without leading to additional moments. In this case, the user should set Hs equal to zero.

- Soil-concrete friction coefficient is used for sliding verification.
- Safety factors for sliding and rotation need not to be greater than one when ASCE 7-16 combinations are introduced. Service combinations that include 0.6D and ultimate combinations that include 0.9D are for stability use; the common "1.5" safety factor is already embedded in these combinations. The program verifies stability for ULS and SLS combinations.
- The soil bearing capacity is considered for all service combinations. When specified as "Net", backfill weight is not considered when calculating the soil reaction. The "Net" and "Gross" descriptions have same effect on the ultimate bearing capacity that is mentioned hereafter.
- The ULS soil capacity is used in all ultimate combinations. It is factored by a reduction factor indicated in table 12.13-1 of ASCE 7-16.

⚠ Both ULS and SLS soil capacities should be satisfied. The footing is sized accordingly. It is mandatory that the user selects at least one service combination; the most common one being D+L. Reinforcement is calculated at ULS only, considering the large cover that is usually used in isolated footings (75mm).

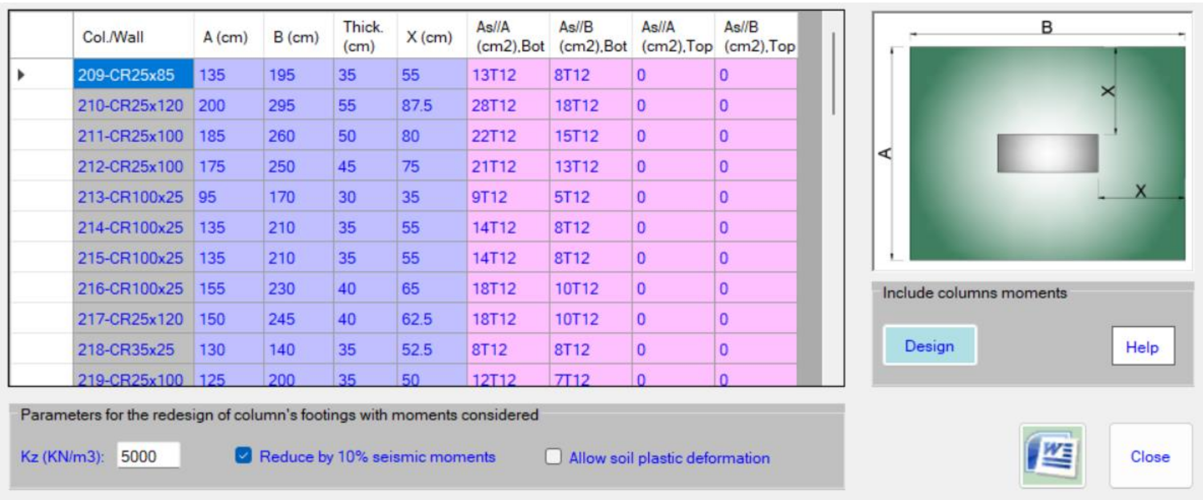
Once clicked, the "Desing" button performs the following tasks:

- ✓ Footings under columns are sized and their reinforcement calculated under axial forces only.
- ✓ Footings under walls are sized and their reinforcement calculated under full loads.

The reason for disregarding moments at this step is to provide a first estimation of column's footings dimensions, allowing the user to assess the effect of the moments on the size of the footing.

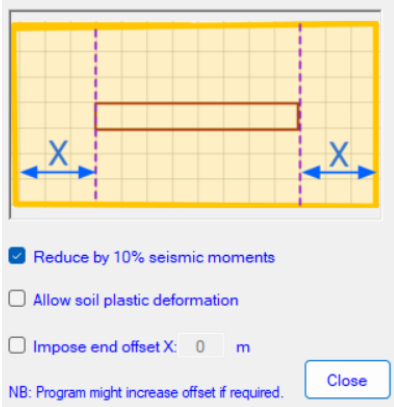
The following table is displayed, showing footings dimensions, the offset "X" that is of help during drafting, and the total top and bottom reinforcement.

- The program always assumes for columns and walls that the edges of the footing are equally distant from those of the column or wall. It is hence possible to deduce the outline of the footing by operating an offset of "X" to that of the column or wall.
- "A" is always the dimension parallel to the small side of the column/wall. "B" is that parallel to the big side.
- As//A is the reinforcement parallel to the side "A" of the footing. It corresponds to the total area of reinforcement to be placed across the side "B".
- Top reinforcement is zero when the footing is not subject to an uplift, even when column's moments are considered. The top face of the footing is subject to tension when partial uplift occurs. In this case, the weight of the backfill of depth "Hs" induces a moment that requires top reinforcement that in most of the cases corresponds to the minimum of the Code.

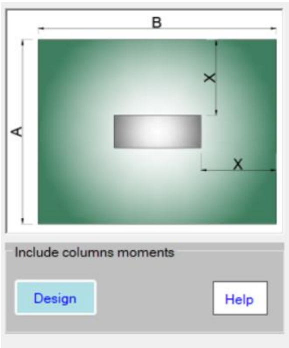


Col./Wall	A (cm)	B (cm)	Thick. (cm)	X (cm)	As//A (cm2),Bot	As//B (cm2),Bot	As//A (cm2),Top	As//B (cm2),Top
209-CR25x85	135	195	35	55	13T12	8T12	0	0
210-CR25x120	200	295	55	87.5	28T12	18T12	0	0
211-CR25x100	185	260	50	80	22T12	15T12	0	0
212-CR25x100	175	250	45	75	21T12	13T12	0	0
213-CR100x25	95	170	30	35	9T12	5T12	0	0
214-CR100x25	135	210	35	55	14T12	8T12	0	0
215-CR100x25	135	210	35	55	14T12	8T12	0	0
216-CR100x25	155	230	40	65	18T12	10T12	0	0
217-CR25x120	150	245	40	62.5	18T12	10T12	0	0
218-CR35x25	130	140	35	52.5	8T12	8T12	0	0
219-CR25x100	125	200	35	50	12T12	7T12	0	0

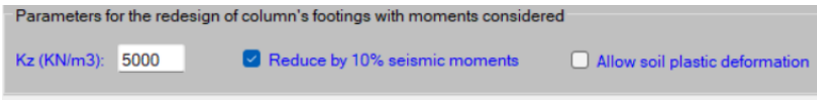
The equal offsets ruling that is adopted in the program can be bypassed for walls. When clicking the “Data for walls” button, the user can specify the following parameters:



- Reduce by 10% seismic moments, as permitted by ASCE 7-16 in section 13.13.4.
- Allow plastic soil deformations. The presence of moments leads to linearly varying (triangular) distribution for soil reaction that might exceed at the tip the ULS allowable capacity. It is possible to use an equivalent rectangular distribution of same resultant, that satisfies equilibrium (Meyerhof approach). Practically, this permits to exceed the soil capacity by 4/3 (33%) and hence save in the size of the footing. Only seismic combinations and ULS with wind can benefit from the plastic deformation allowance.
- Finally, the user can impose the value of the offset “X” in the longitudinal direction of the footing. This is helpful since some walls might be located at the edge of the plot, or a pit.



In order to design column’s footings with inclusion of the moments, the user should click on the “Design” button, after considering the three fields at the bottom of the window. These are like those of walls, beside the Winkler coefficient Kz (soil vertical stiffness). Low values are recommended (ex.: 5000 KN/m^3) since they lead to bigger flexural moments. The program starts an iterative procedure, increasing at each time the size of the footing by 5cm, till soil capacity and stability conditions are both satisfied.



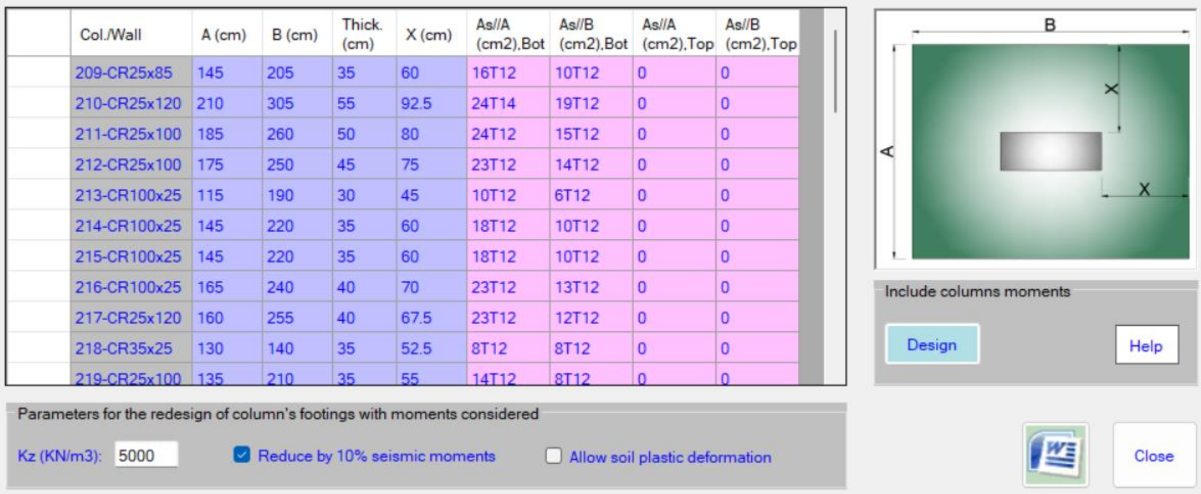
Once calculations are performed, the results of stability analysis are displayed, and the initial table for sizing and reinforcement is updated.

Col./Wall	Sliding	Rot. about A	Rot. about B	Status
209-CR25x85	6.69	65.76	63.48	OK
210-CR25x120	9.14	71.04	362.13	OK
211-CR25x100	11.72	50	275.49	OK
212-CR25x100	16.25	30.96	215.65	OK
213-CR100x25	4.88	8.39	14.96	OK
214-CR100x25	4.88	76.3	38.11	OK
215-CR100x25	4.14	52.26	36.33	OK
216-CR100x25	6.38	82.22	40.45	OK
217-CR25x120	5.75	39.37	65.53	OK
218-CR35x25	47.23	98.11	95.3	OK
219-CR25x100	8.59	18.39	39.42	OK
220-CR25x100	8.8	6.44	14.95	OK
221-CR25x100	16.77	11.21	49.56	OK
222-CR85x25	8.88	47.46	51.95	OK
223-CR25x100	4.64	9.36	25.1	OK

Rotation FOS: 1.0 Sliding FOS: 1.0 Close

Safety factors against sliding, rotation about strong inertia of the column/wall (titled “about A” in the table) and about its weak inertia (titled “about B” in the table) are shown. These are the minimal safety factors that are obtained among all investigated combinations. When sliding criteria fails, the user displays in the “Status” column the note “Use tie beams”. Also to note that for walls, the safety factor corresponding to the rotation about B side (weak inertia of the wall) is not calculated. The program considers only in-plane forces for walls.

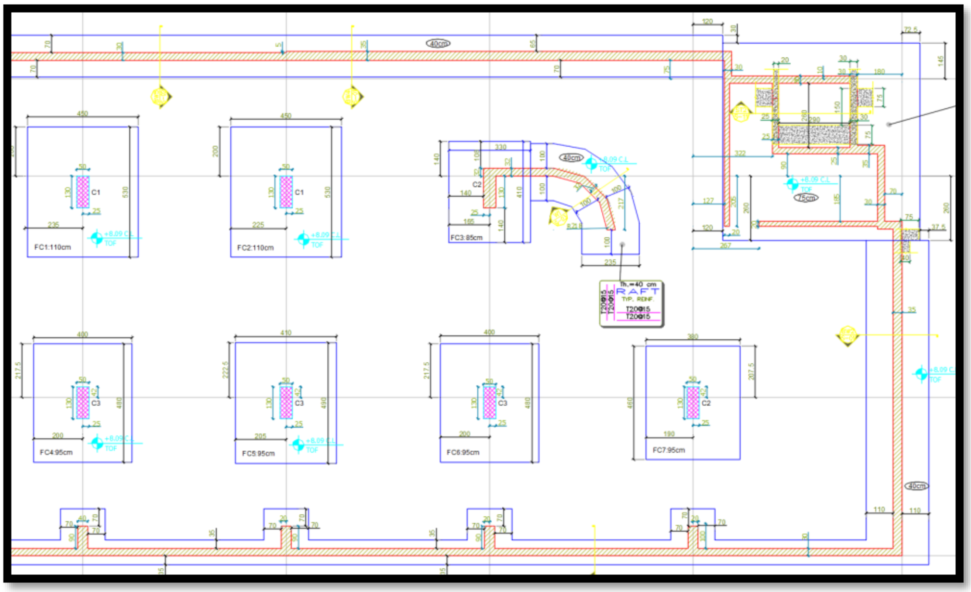
The updated column's results are shown in this table. By simple comparison, one can note the slight increase in size and in reinforcement related to the consideration of the bending moments. Results for walls are unchanged since the first design already included all in-plane forces.



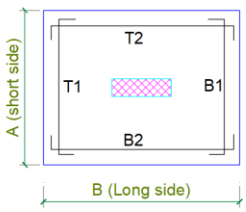
Col./Wall	A (cm)	B (cm)	Thick. (cm)	X (cm)	As//A (cm2),Bot	As//B (cm2),Bot	As//A (cm2),Top	As//B (cm2),Top
209-CR25x85	145	205	35	60	16T12	10T12	0	0
210-CR25x120	210	305	55	92.5	24T14	19T12	0	0
211-CR25x100	185	260	50	80	24T12	15T12	0	0
212-CR25x100	175	250	45	75	23T12	14T12	0	0
213-CR100x25	115	190	30	45	10T12	6T12	0	0
214-CR100x25	145	220	35	60	18T12	10T12	0	0
215-CR100x25	145	220	35	60	18T12	10T12	0	0
216-CR100x25	165	240	40	70	23T12	13T12	0	0
217-CR25x120	160	255	40	67.5	23T12	12T12	0	0
218-CR35x25	130	140	35	52.5	8T12	8T12	0	0
219-CR25x100	135	210	35	55	14T12	8T12	0	0

4.6.1 Drawings generation

The program can generate the foundations layout and tabulated reinforcement for isolated footings under columns and walls. Footings are labeled and have dimensions.



Foot. ID	A(cm)	B(cm)	Thick. (cm)	Abot//B	Abot//A	Atop//B	Atop//A
FC1	450	530	110	40T25	48T25	40T12	48T12
FC2	450	530	110	40T25	48T25	40T12	48T12
FC3	330	410	85	30T20	36T16	30T12	36T12
FC4	400	480	95	36T20	44T20	36T12	44T12
FC5	410	490	95	36T20	44T20	36T12	44T12
FC6	400	480	95	36T20	44T20	36T12	44T12
FC7	380	460	95	34T20	42T20	34T12	42T12



H: Thickness of footing
B1: Bottom reinf. parallel to short side A
B2: Bottom reinf. parallel to long side B
T1: Top reinf. parallel to short side A
T2: Top reinf. parallel to long side B

Chapter 5- Sample models.

The user can download the demo version of the program from STRES website:

BUILD-PRO - STRES Software

In the same page, there is a separate link that contains sample models.

Inside the directory “SAMPLE MODELS,” you’ll find folders, each corresponding to a model (*Every BUILD-PRO project is a folder, not a single file*) that can be readily opened with BUILD-PRO. The user can explore them and speed up the learning process.

Additionally, the AutoCAD preparation drawing of each model is provided.

END OF DOCUMENT