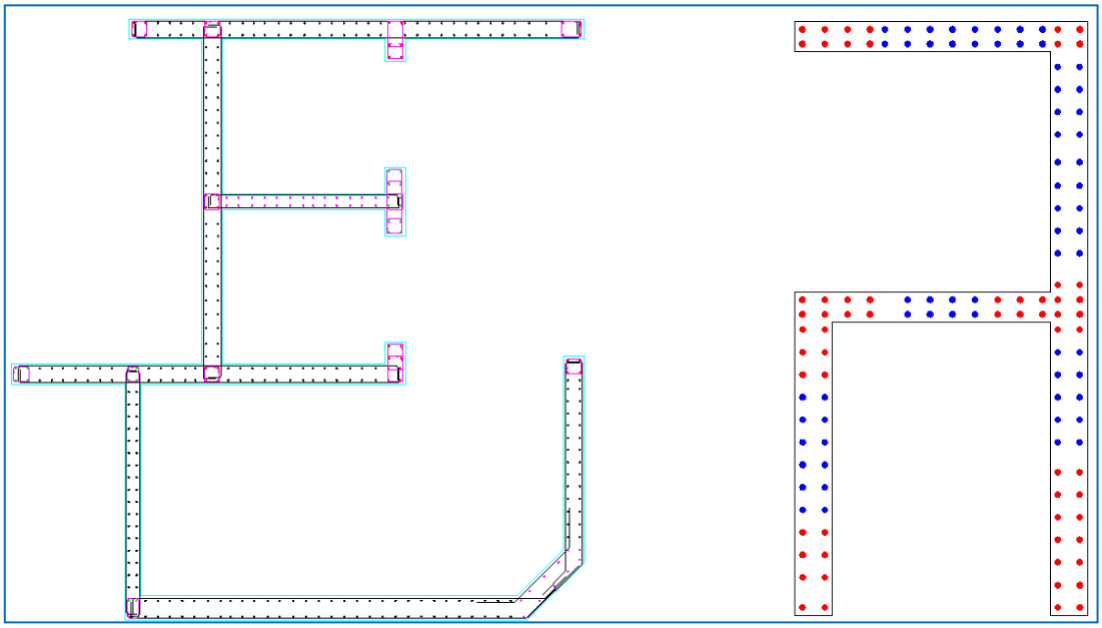
Introduction

WALLS-EXPERT is an innovative application for analysis and design of reinforced concrete core walls of any shape.



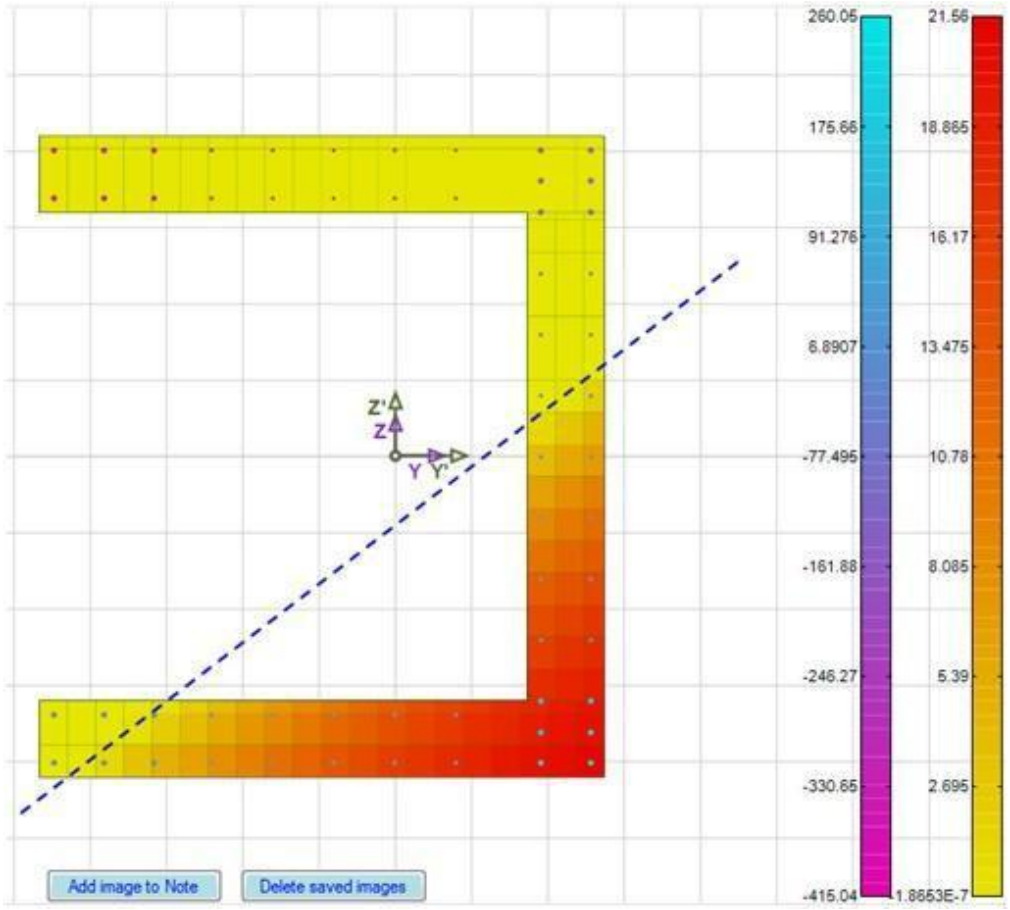
WALLS-EXPERT is conceived for Engineers who need an easy and powerful tool to design a simple wall, or a complex group of walls (core walls).

The main features of **WALLS-EXPERT** are:

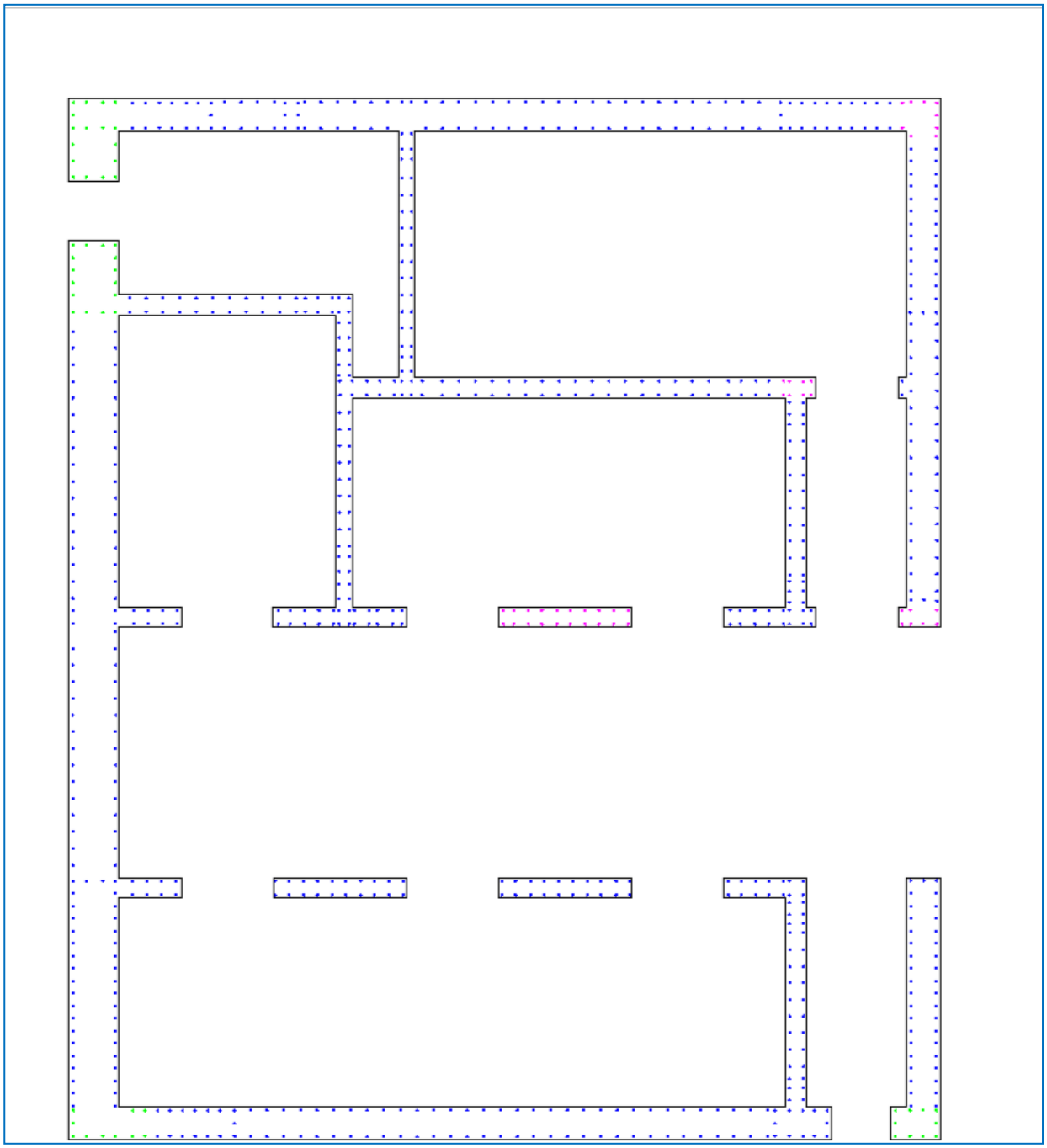
- ***Friendly graphical interface*** allowing for a quick introduction of loads and parameters.
- ***Direct import of the core geometry and reinforcement from AutoCAD***. Basically, the user draws the geometry of the walls by the mean of closed contours (polylines) and represent the reinforcement by donuts, just the same way when generating classical design drawings. By grouping the donuts with layers named alike the reinforcement diameter, **WALLS-EXPERT** will straightforwardly recognize the geometry of the core and the reinforcement. It is the most efficient and quick way to ***import elements of any complexity***.
- When walls composing the core are not connected, **WALLS-EXPERT** interprets voids as ***doors***, allowing thus the consideration of realistic situations where doors are inevitably present.

- ***Rectangular block and parabolic-linear models for concrete*** are available. ***Bilinear models (with hardening) for reinforcement*** can be used as well. The user can choose to either use the reinforcement in compression or not. Strength reduction factors can be used when designing walls according to ***ACI318-19***.
- ***Graphical display of colored maps for stresses in concrete and reinforcement.*** This unique feature allows the user to fully investigate the results and optimize the reinforcement in the walls by detecting zones where the reinforcement bars are highly stressed. Both non-linear (real) and elastic stresses can be displayed.
- ***Generation of (N,My,Mz) interaction diagrams*** to verify the capacity of the section.
- ***Shear design of all wall segments considering doors and windows, according to ACI318-19.*** The software calculates the area of horizontal and vertical reinforcement required to resist shear forces resulting from seismic and wind forces.
- Generation of high-quality design report where results are displayed by the mean of ***graphics and tables***.

WALLS-EXPERT ***combines simplicity with state-of-the-art technologies*** to help you solve your most complicated problems within minutes.

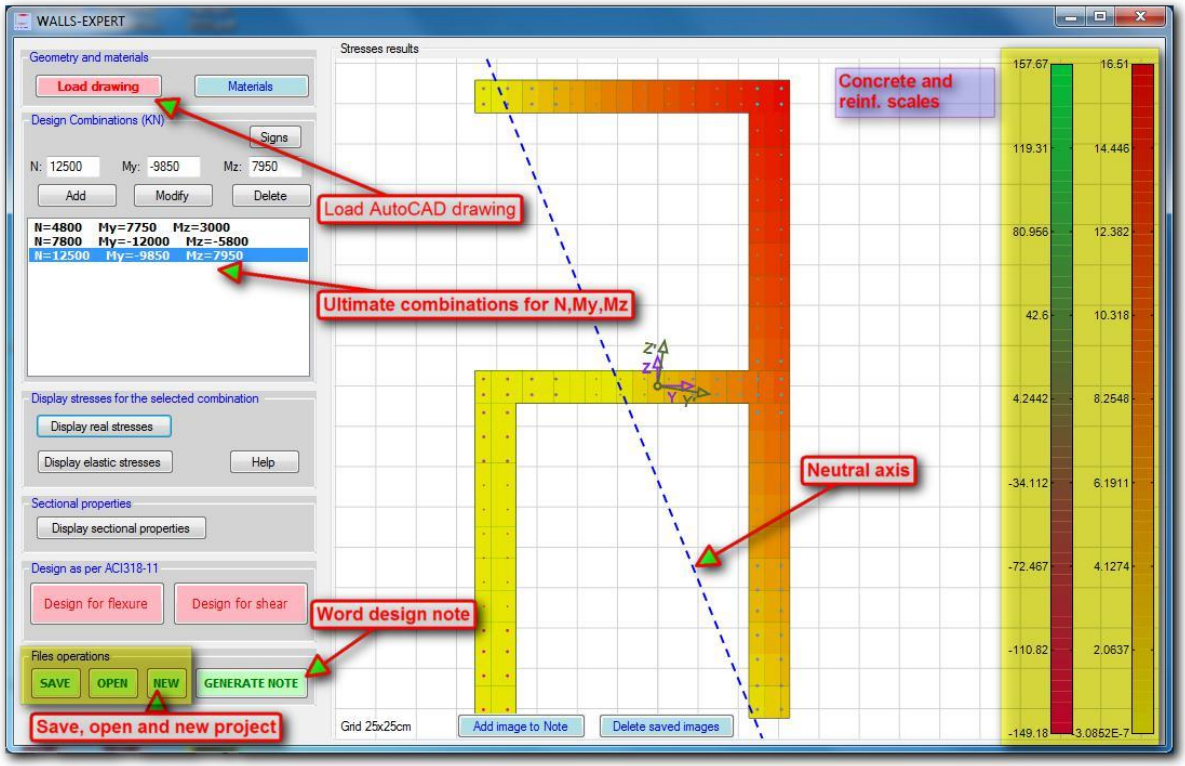


Real stresses (MPa) in concrete and reinforcement



Import from AutoCAD of a complex core that comprises door openings.

General overview



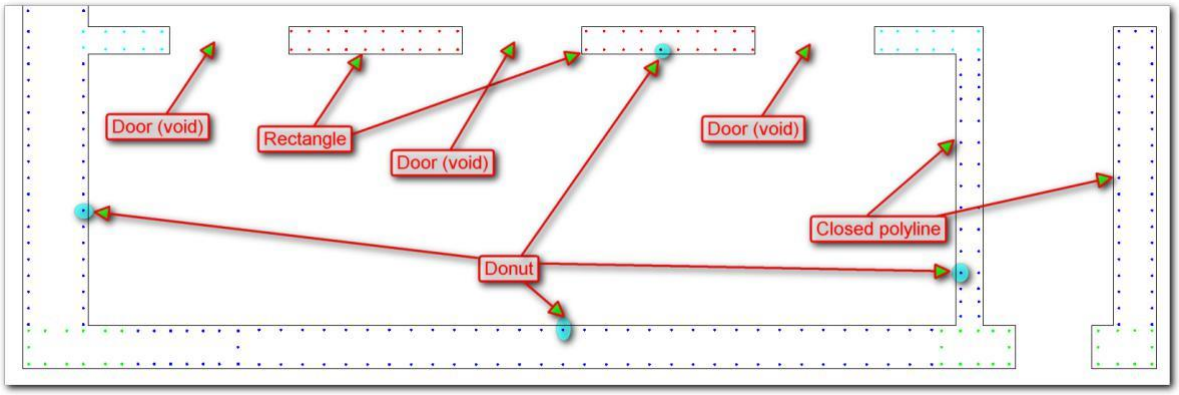
The Main Window contains seven zones:

- ***"Geometry and materials"***. Used to load an AutoCAD file of the drawing of the walls and to control the materials parameters.
- ***"Design Combinations (KN)"***. Ultimate combinations of loads (units are in KN) are introduced for the axial force, bending moment about axis y and bending moment about axis z for axial forces-flexure interaction analysis.
- ***"Display stresses for the selected combination"***. Real (non-linear) compression/tension stresses for concrete and reinforcement are displayed for the selected combination. Elastic stresses for the gross section can be shown as well.
- ***"Section properties"***. Displays the mechanical characteristics of the section (areas, inertias…)
- ***"Design as per ACI318-19"***. The core can be fully verified according to the American Standards (ACI318-19 metric). The generated interaction diagrams for axial forces-moments incorporate strength reduction factors for tension and compression-controlled sections. Horizontal and vertical reinforcement are calculated considering full requirements of ACI.
- ***"Files operations"***. Allows saving in a separate file all data introduced to the software (combinations for bending and shear analysis, material properties…), opening an existing project, creating a new one from scratch, and generating a Word design report.
- The graphical display where stresses are displayed with two scales, for concrete and steel.
- ***"Import from ETABS"*** introduces in the program the combinations that are created in ETABS®. It is possible hence to design the core in WALLS-EXPERT.

Loading a drawing from AutoCAD

WALLS-EXPERT offers a simple and efficient way to input the geometry and the reinforcement of core walls: Using one of the most traditional CAD software's available on the market, AutoCAD.

The screen capture below shows a complex group of walls pertaining to a core, generated on AutoCAD.



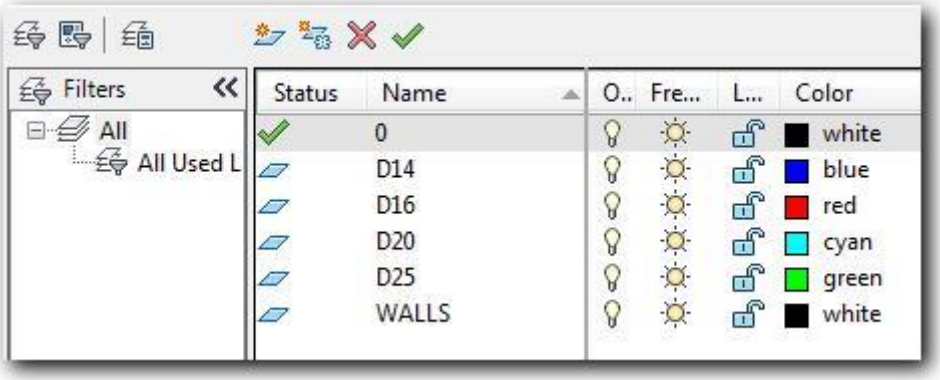
During the drawing phase, the following simple rules should be respected:

- The geometry of the wall(s) should be achieved by using ***closed polylines*** or ***rectangles*** belonging to layer ***"WALLS"***. When two walls do not have common edges, the void between them is interpreted as a door. Since the whole group of walls will behave in the program as a unique section, the user must ensure that the lintel above the opening is stiff enough to make the core behave as one unit under lateral loads. If two distinct walls are not linked withy a lintel, then they will resist independently lateral loads (not like one entity) and consequently they should not be introduced both in the core.
- Walls of complicated geometry can consist of one polyline having multiple segments or a series of adjacent rectangles. If you draw the wall by using the line command, do not forget at the end to convert the series of lines to a closed polyline.
- Arcs and curves cannot be represented by using the arc or circle command in AutoCAD. Always approximate arcs by a series of segments.
- All reinforcement should be represented as ***donuts*** (use the donuts command in AutoCAD). The diameter and the color of the donuts are not important. The program identifies the diameter of the rebar by reading its ***layer's name***. The screen capture below is extracted from AutoCAD layer's menu. All donuts should belong to layers named ***"D#"*** where # is an integer number representing the ***diameter in mm of the rebar***.

- It is advised to keep in the drawing to be loaded by the program only the layers shown in the image below. Thus, the drawing to be created should contain only the geometry of the wall. (polylines in layer "WALLS") and the reinforcement (donuts in layers "D#").
- ***The unit of the drawing is always meters and the scale 1***. Example: wall of length 3.75m and width 0.25m should be represented by rectangle 3.75x0.25.
- **The drawing of the wall must be saved in version AutoCAD 2010**.
- ***Many samples of walls drawings*** can be found in the folder "Drawings samples" located inside "WALLS-EXPERT" folder.

NB:

Only the longitudinal (vertical) reinforcement has to be represented in the drawing, by the mean of donut elements. ***The horizontal reinforcement is calculated by the program when designing for shear***.



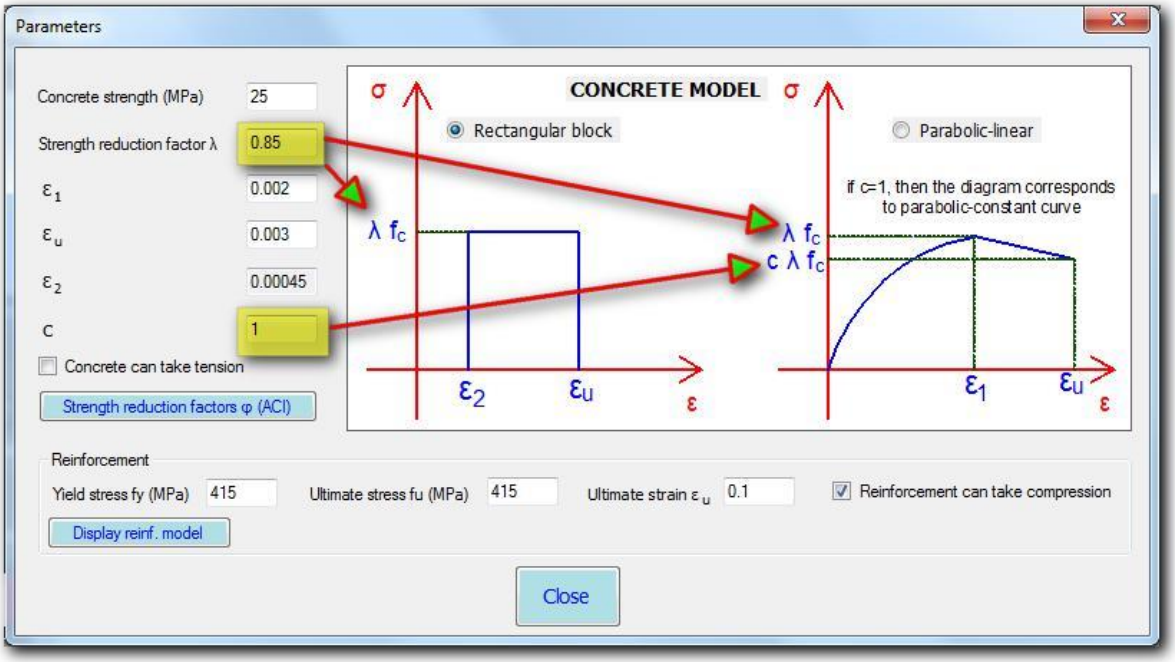
Sample of layers to be used in a wall drawing

Materials

Two concrete models are available in the software.

- The classical rectangular block model generally adopted by most of Codes.
- The more realistic parabolic-linear model

Any of both models can be used to generate the interaction diagrams, but the program always uses the parabolic-linear one (more stable numerically), even if not selected, to display real stresses in concrete and reinforcement.



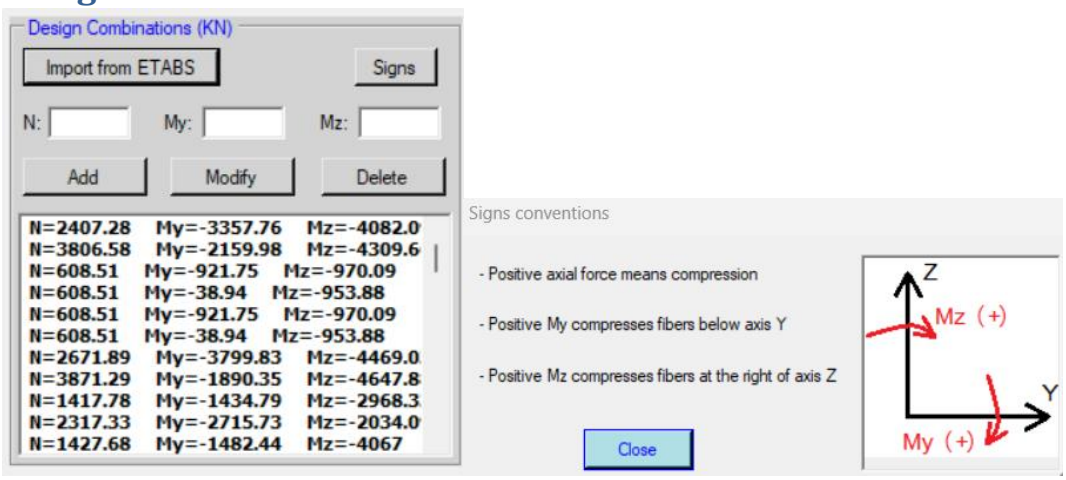
A bilinear curve is used to model steel material. By setting the ultimate stress equal to the yield one (no hardening), we retrieve the common stress-strain model that can be found in most of the Standards.

By using strength-reduction factors, it is possible to verify the flexural capacity of the section ***according to any Code*** *and particularly ACI318.*

The user can prevent compression reinforcement to participate in the strength of the section.

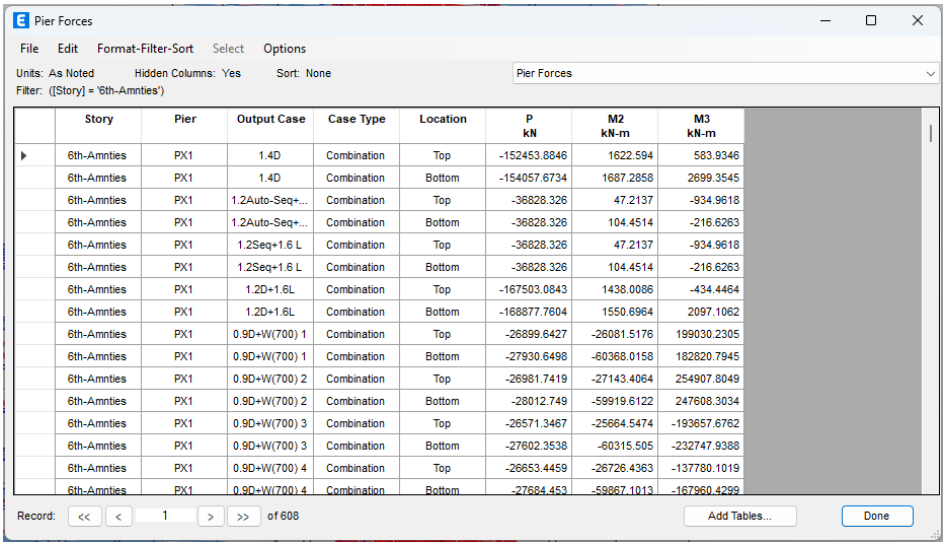
Ultimate strain (set to 10% in the figure above) corresponds to the rupture point of the reinforcement.

Design combinations



The user should introduce acting forces as ***ultimate combinations; they are applied by the program at the center of gravity of the core***. The introduced load cases are used by the software to verify the flexural capacity of the section (interaction diagrams) and to display the resulting normal stresses maps in steel and concrete. Reference (Y,Z) is displayed in the figure above. Units are in KN and cannot be changed.

Here are the steps to follow when importing loads from ETABS.

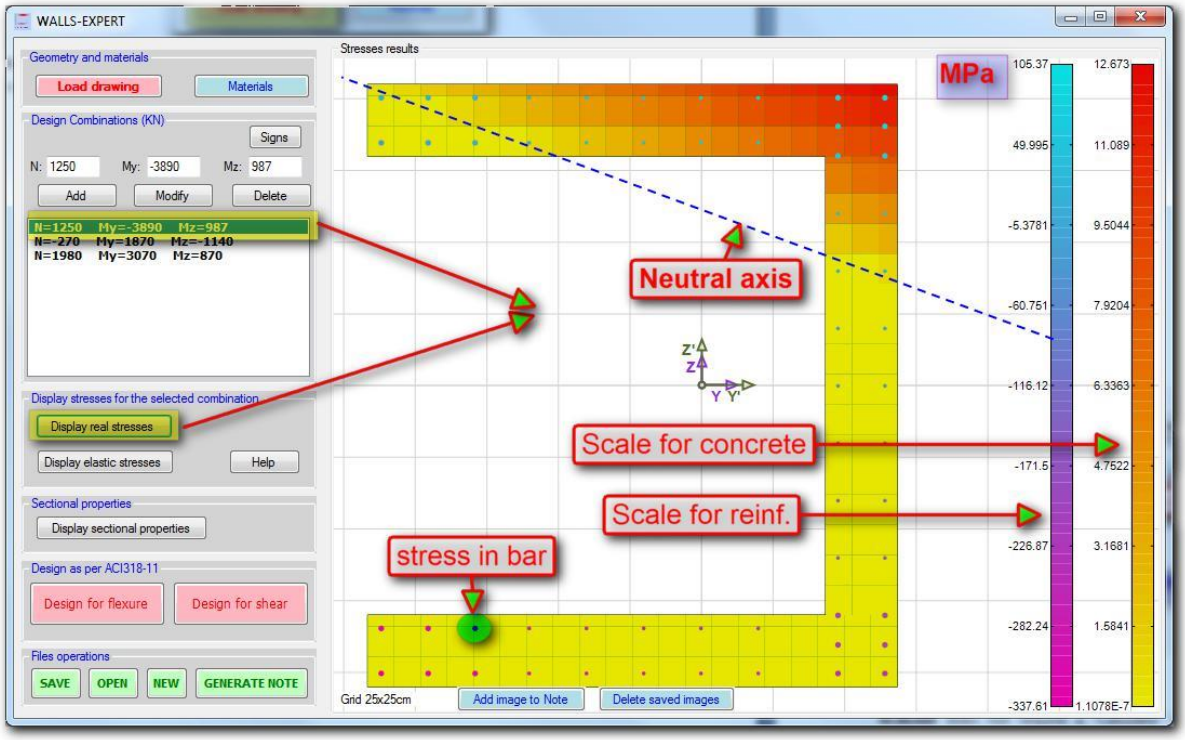


All the walls of the core should be assigned the same pier name. For instance, if the wall has a “U” shape, all three walls should have the same pier label. It is possible then to display the forces P,M2,M3 (refer to the figure above) by opening the table related to this pier. The table should be identical to this figure (eight columns). The user can incorporate in the table all the created combinations. M2 and M3 are the moments about the local axis 2 and 3 of the pier.

- The table should be exported to Excel format, and opened in Excel.
- Finally, it should be saved as “Text (Tab delimited) (*.txt)”, which is a text file.
- Once created, open this text file by clicking on the button “Import from ETABS”.

Attention! M2 will be saved as My and M3 will be saved as Mz. Moments M2,M3 and My,Mz should have the same orientation with regards to the inertias of the core. Otherwise the imported drawing of the core from AutoCAD should be rotated accordingly.

Real and elastic stresses



Real stresses

For the selected combination, real stresses can be displayed with colored maps for both concrete and reinforcement (individually for each rebar).

This original feature can be used for two purposes:

1. **Reinforcement optimization of the core**. The user can check the distribution of stresses in concrete and steel for each combination. Reinforcement area can be reduced in zones where tension stresses are low, as it can be increased when stresses reach the ultimate limit fu, in case the design fails. This is why it is interesting to use this feature in conjunction with the "Design for flexure" in order to cleverly increase the strength of the section that fails under some combinations.
2. For cores designed to resist seismic loads, ACI requires the use of **boundary elements** and to identify their lengths based on the location of the neutral axis. This latter is displayed together with a gird of step 25cm. The user can hence easily deduce the dimension of the boundary edge for each wall of the core.

Elastic stresses

Elastic stresses are displayed for the selected combination, considering a linear elastic behavior of concrete and disregarding the presence of steel. The main purpose of the elastic stresses is to assist

the user to identify whether a boundary element is required or not (which is true if compression stresses exceed 0.2*f'c, as per ACI318).

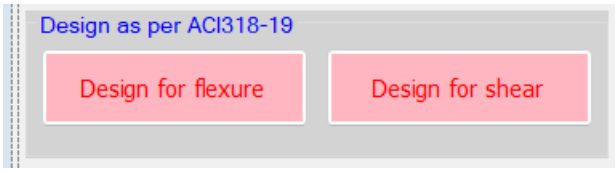
To modify the diameters of the reinforcement (if required), the user has to go back to the AutoCAD drawing. A good procedure consists of starting with a reinforcement configuration that satisfies minimal Code requirements, and to increase the reinforcement through iterations.

NB:

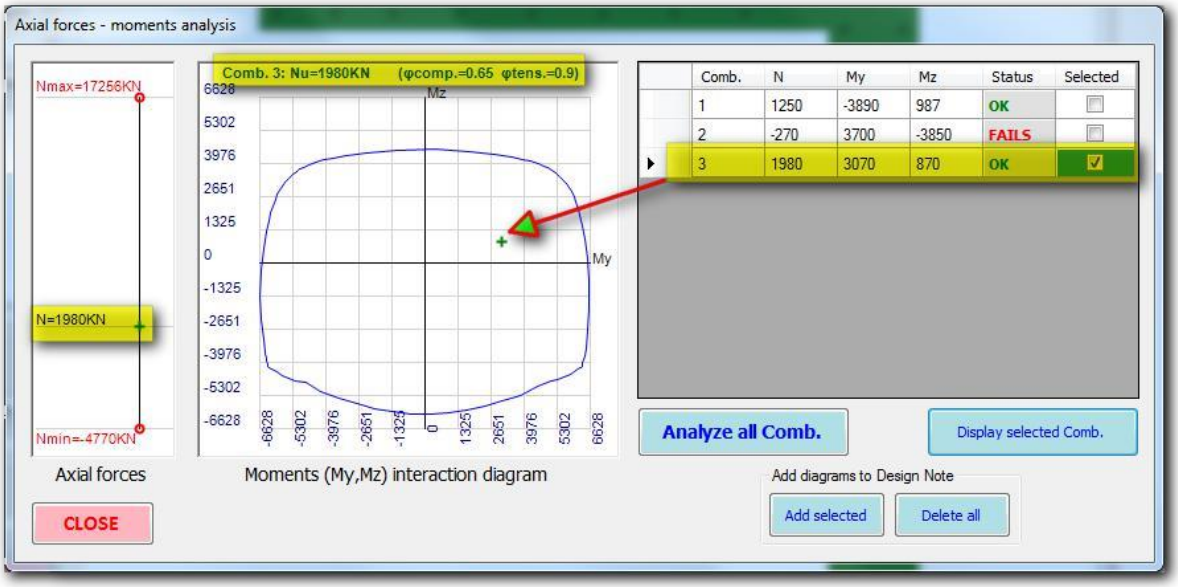
One should note that stresses are displayed even if the loading case falls outside the interaction diagram. The displayed map of stresses does not include the strength reduction factors. To perform a Code verification of the section regarding axial forces-bending, the ***"Design for flexure"*** procedure should be used.

Design according to ACI318-19 metric

The program can perform a full Code design of the core for flexure (with axial force) and shear.



Design for flexure



The program analyses all the introduced combinations and generates the related interaction diagrams.

For the selected combination (N,My,Mz), a capacity curve is generated in the My,Mz plane. This curve defines the flexural capacity of the section under the applied axial force N. The flexural capacity of the section is defined only by the geometry of the section and the strength of the materials. The loading case is represented on the display by a cross in green color if (My,Mz) falls within the domain, and in red color if (My,Mz) falls outside it. On the other hand, if the axial force exceeds the limits in compression or in tension of the section, the corresponding combination fails as well.

The interaction diagram considers the introduced values of the ***strength reduction factors for compression and tension-controlled sections***. These factors can be found in the ***"Parameters"*** window.

If the design fails under a combination, it is advised to check the corresponding real stresses map in order to identify the cause and hence to consider the proper measures.

Each diagram can be added to the Word design note by clicking the ***"Add selected"*** button.

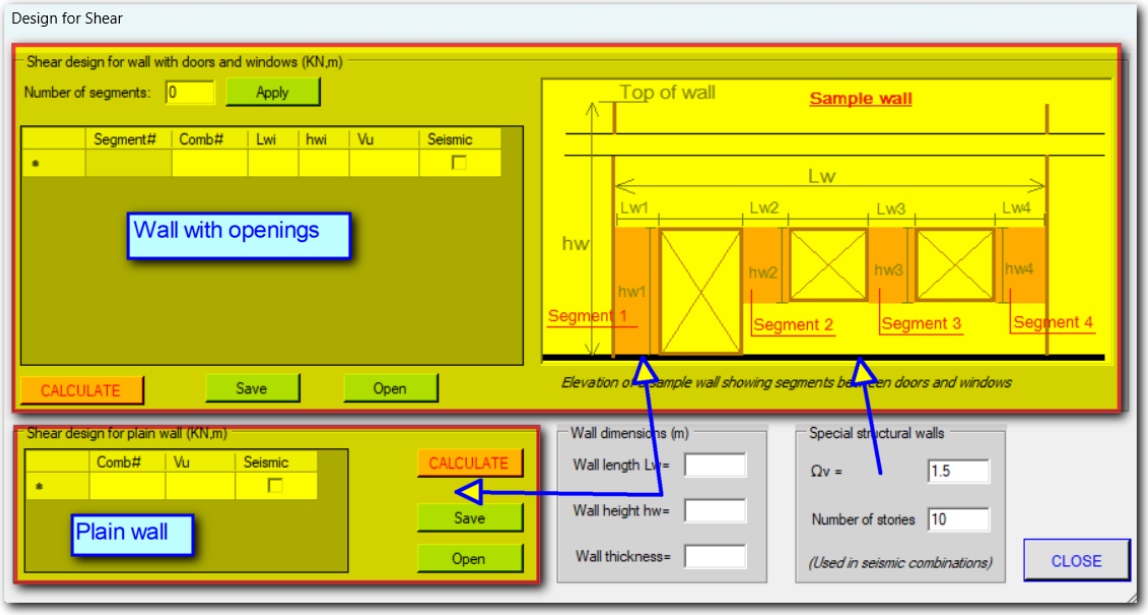
Design for shear

The design of walls regarding shear forces is a complicated procedure especially when both seismic and non-seismic combinations are present. The expressions to be used are not identical (the recommendations of both chapters 11 and 18 of ACI318-19 should be followed) and several conditions should be satisfied.

WALLS-EXPERT fully tackles the problem by verifying the resistance of the wall and by ***providing the areas of the required vertical and horizontal reinforcement***.

The following notation is used:

- A core consists of a group of walls.
- A wall could be plain or with openings (door and windows)
- If a wall contains openings, the element between two openings is called segment. If just one opening is present, then there should be two segments.



For each wall pertaining to the loaded core, the user should provide the information's requested in the tables above, whether the wall is plain or contains openings.

Two tables are displayed.

1. **<u>The upper one concerns walls with openings</u>**. The user should at first specify the number of segments present in the wall. Then he introduces the number of each segment (integer starting from 1 till the total number of segments) and that of the combination (integer starting from 1) applied to the segment in question. Beside the ultimate shear force on the segment, the user should indicate whether the combination is seismic or not and should provide the number of stories (above the critical section of the wall) and the coefficient Ω_v that is required to amplify the ultimate shear and to transform it into design shear, as per section 18.10.3.1 of ACI 318-19. These two entries are located in the groupbox at the bottom right of the form, used only for special shear walls when the seismic design category (SDC) is D,E or F, as per table R18.2 of the Code.

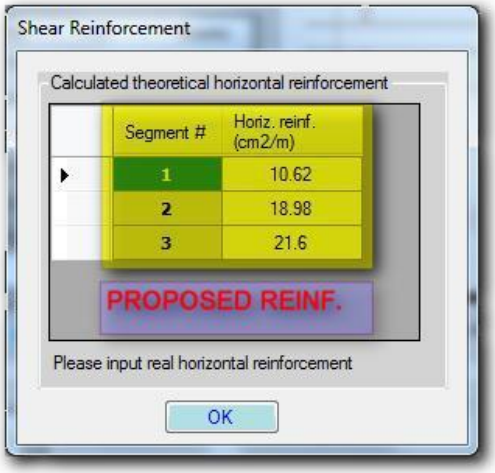
The global wall dimensions (total length, total height and thickness) are defined in the central groupbox. The height of the wall is defined from the section where we have the maximal moment (usually foundations or ground floor) till the top of roof in case the wall ends there. Once the wall is calculated, it is possible to save it and give it an individual name. Another wall can then be defined, calculated and saved until all the walls of the core are done. This operation is repeated as many times as there are walls with openings in the core.

2. **If the wall is plain**, then the lower table is to be filled. The user introduces the desired combinations (the ultimate shear forces that the wall should resist) and the wall dimensions in the central groupbox. Once calculated, the wall can be given a unique name and saved.

In both cases, whether the wall is plain or not, the dimensions of the wall should be always provided (length, total height and thickness) in the central groupbox.

- The user should calculate the wall before saving it by clicking on the ***"calculate"*** button.

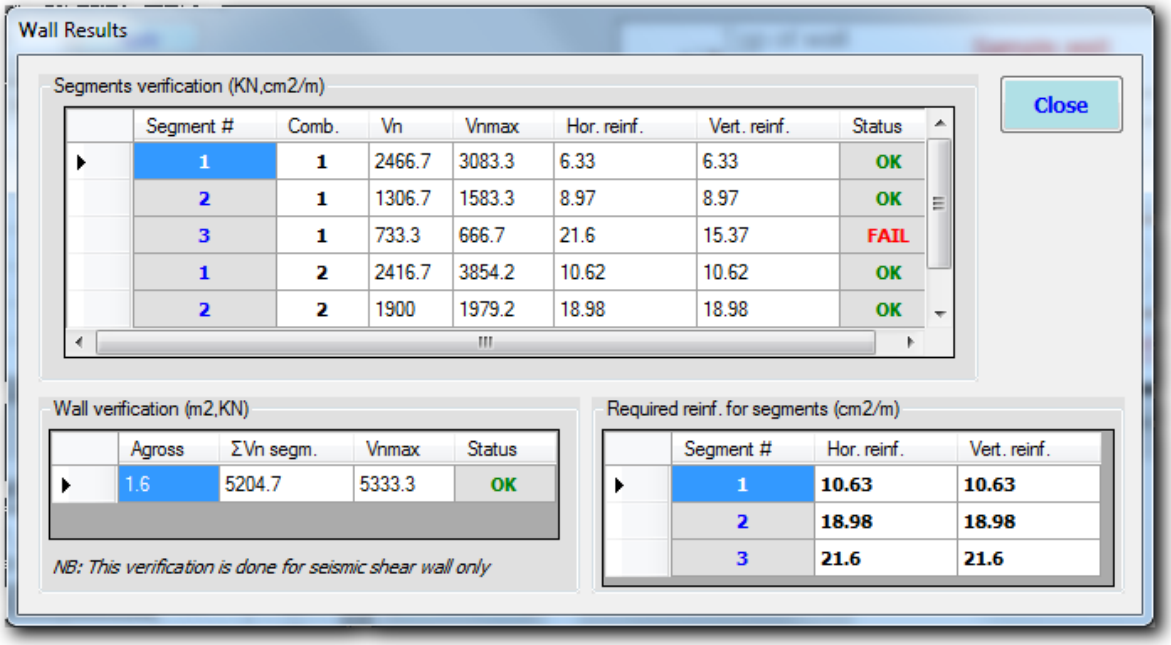
Once the ***"calculate"*** button is pressed, the program opens a window where the proposed **required** horizontal reinforcement is displayed. It can be modified by the user in order to match the value obtained when using the available diameters in the market (**provided** reinforcement).



Once the required reinforcement is displayed, another window is opened (refer to the screen capture hereafter).

The ***"Wall Results"*** window for walls with openings displays three tables.

- The "Segments verification" shows for each segment its nominal shear capacity, the required horizontal and vertical reinforcement and the status of the design.
- The "Wall verification" is displayed only if one or more seismic combinations are applied to the wall. Indeed, ACI318 requires verifying that the nominal shear capacity of the seismic wall exceeds the sum of nominal shear capacities of all segments belonging to the wall. If this condition is not met the design fails even if all segments are verified.



- The final required reinforcement for each segment is displayed as well, whether the design fails for some combinations or not. Table “Required reinf.” summarizes the required reinforcement to resist shear.

It is possible to design all walls present in the core, to save them, and to open them later on for eventual modifications.

The user should note that the introduced combinations for shear design differ from those introduced for flexural analysis. ***For shear, the forces to introduce are applied on each individual wall, whereas for flexure the introduced forces are applied on the whole core***.

Saving a project, generating a design note.

The ***"save"*** button allows to save all the introduced parameters and forces for flexure and shear design in a file to be saved on the hard disk.

The ***"new"*** button empties all the variables used by the software for the creation of a new project.

"Generate note" creates a detailed Word document where all results are displayed.