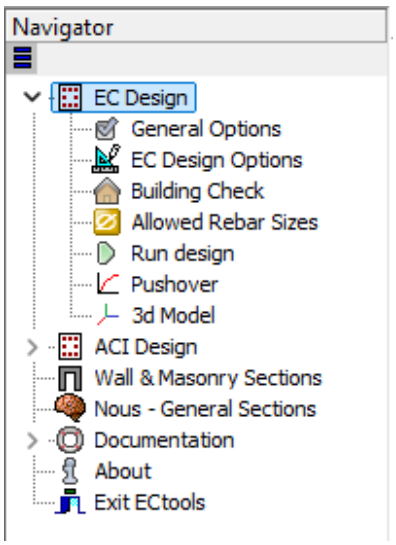
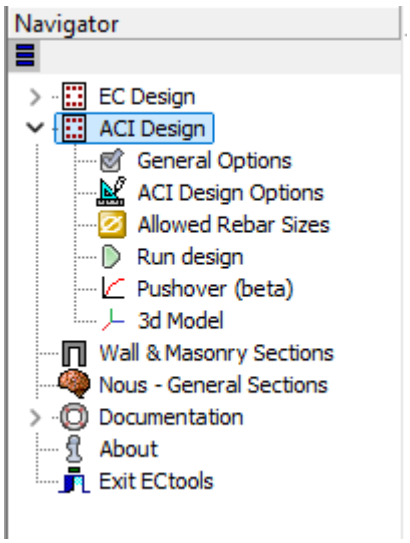
1. General description

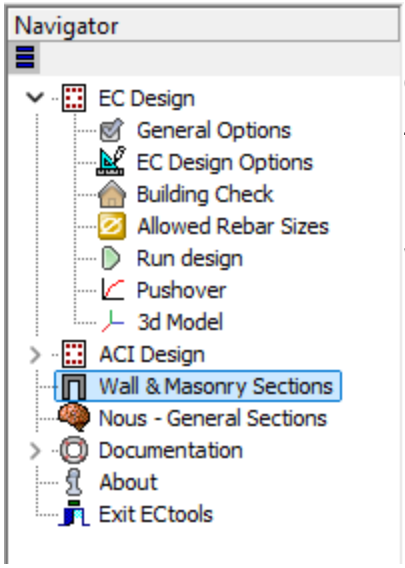
Having installed the program and the user license, the program can be run from *Start Menu ⇨ ECtools ⇨ ECtools* or the corresponding desktop icon (if it has been created during installation). From version 4.0 the program runs in a uniform environment with a navigator in the left side of the GUI providing all the functionality of the previous versions:



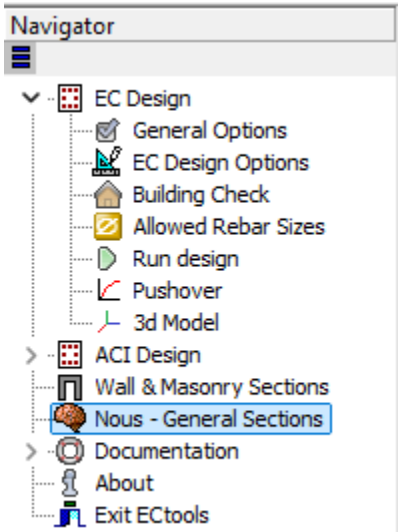
EC Design : Comprises of the main design and check application for RC and/or masonry structures (Standards EC2, EC8, EC6). It collaborates directly with the RC/masonry walls editor and the autonomous section check and design program Nous, which is described below.



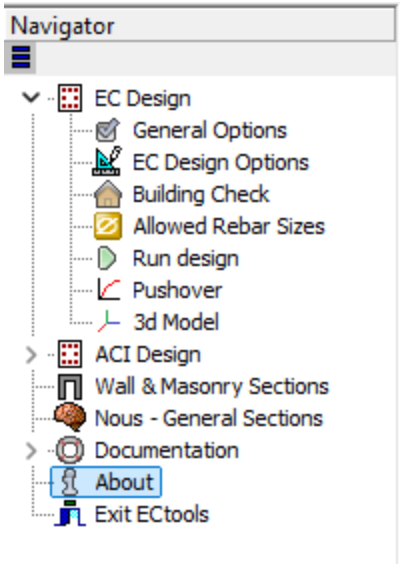
ACI Design : Comprises of the main design and check application for RC structures (Standards ACI318-11 & ASCE/SEI7-10). It collaborates directly with the RC/masonry walls editor and the autonomous section analysis and design program Nous, which is described below.



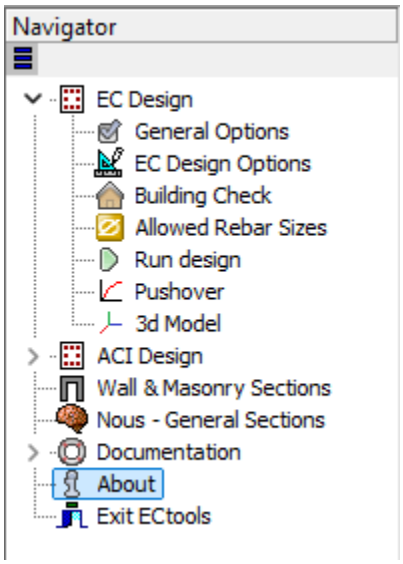
RC/Masonry walls : Graphics editor for creating the geometry of RC walls, masonry blocks and bed-joints that are modelled with linear or area elements in ETABS. It creates text files of the form *.wal* for the RC walls and *.mas* for the masonry elements that are necessary for the design and check runs.



Nous : Autonomous program for the check and design of arbitrary RC and masonry sections. It can accept jacketed sections for strengthening designs of existing structures, creates files of the form *.sd* that are required for checking/designing, includes the computing algorithm for checking and design of arbitrary sections in biaxial bending with axial force and collaborates interactively with the design application forming a reliability auditing tool for the program results.

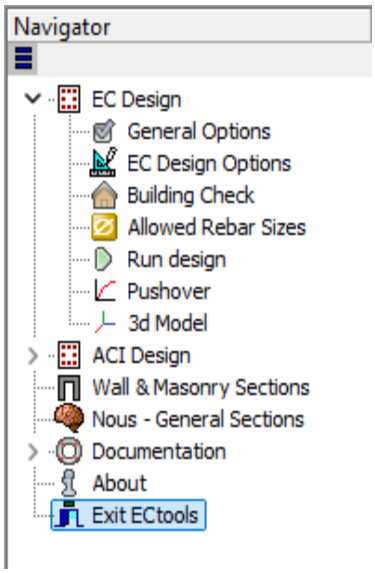


About : Displays the basic program information as described in section 1.2 (current version, user license information).



Documentation:

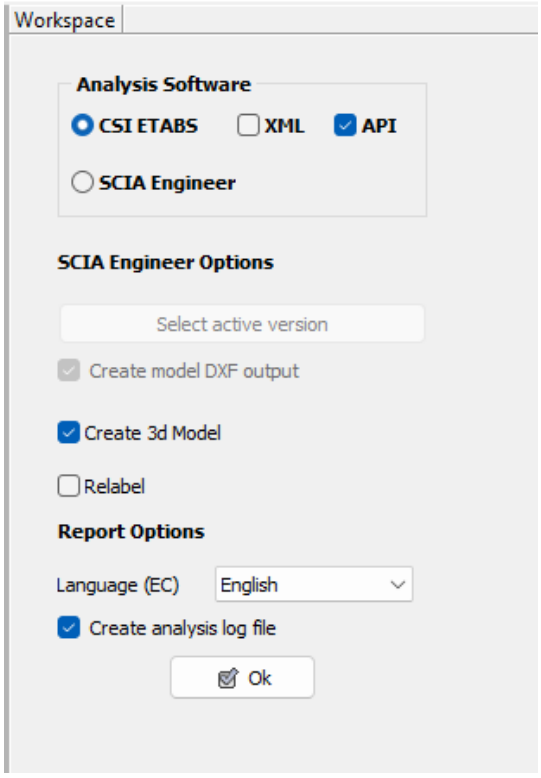
Displays the current user manual(Adobe® Reader required).



Exit : Terminates the program.

2. Overview of the design procedure

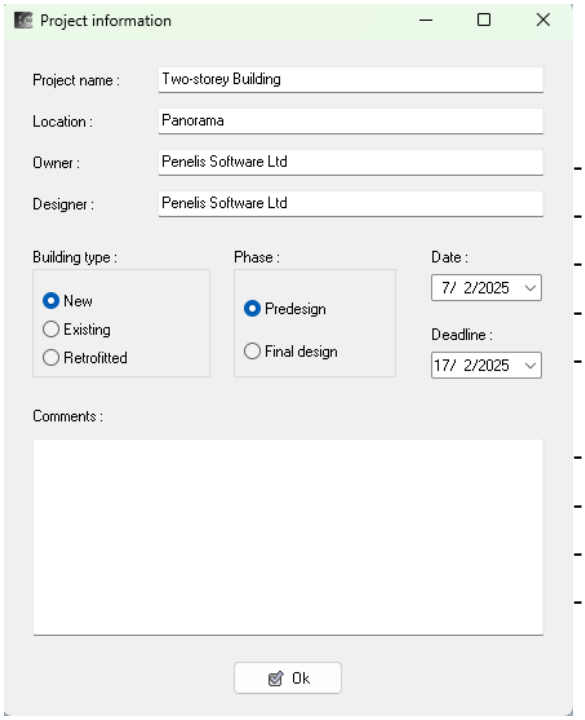
2.1 Exporting data from ETABS 2019-current version



The procedure of exporting data from ETABS 2019-current is handled automatically by ECtools – ETABS API communication. The user simply provides ECtools with the appropriate file path. The file should be solved and opened in ETABS during the design process. For ETABS versions prior to v2019 information can be found in our manuals

2.2 Project file creation

The program has the capability to store all the data of a project in a separate file of the form .etp, which includes all the design options (described in latter sections), the locations of the Input and Output files on the hard drive and additional information that can be entered by the user from *File* ⇨ *Project information*.

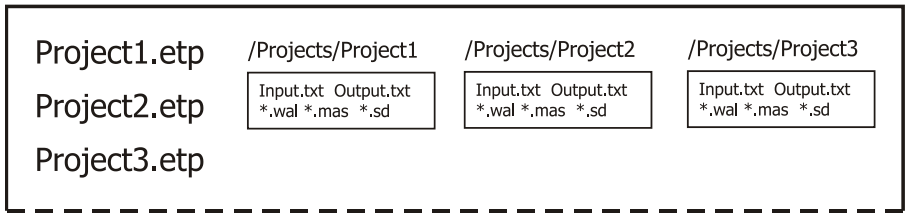


- Project name
- Location
- Client
- Designer
- Project type (new construction, existing, strengthening)
- Project phase (preliminary, final design)
- Project date
- Deadline
- Additional comments

File ⇨ *Open project file* : Opens a stored project file (.etp extension) from the hard drive. Also, an .etp file can open in ECtools by double-clicking.

File ⇨ *Save project file* : Saves the current project (settings, additional information, etc.) in a file of the form .etp. It is noted that the project file does not need to be saved in the same folder with the Input, Output and section files. The following folder structure is recommended for the project folders :

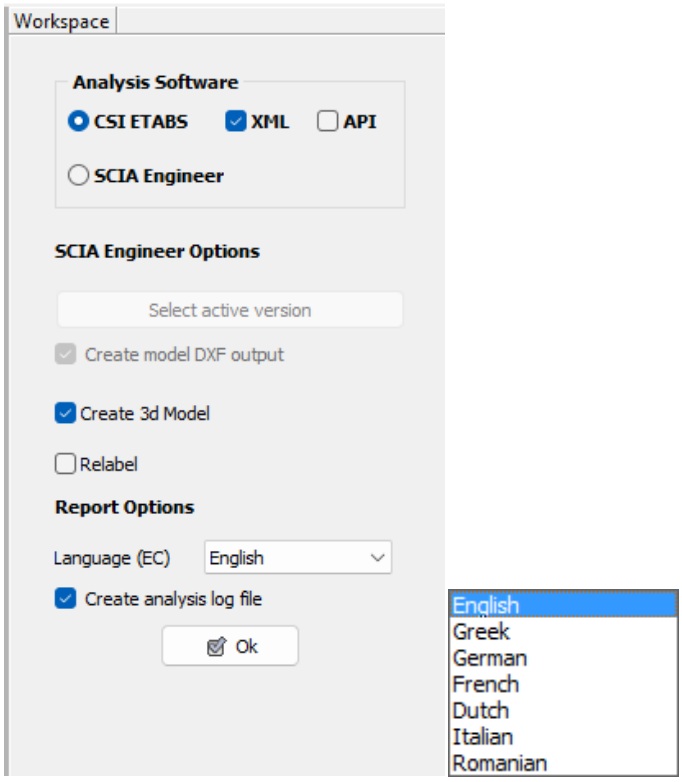
/Projects



File ⇨ *Create compressed project file (zip)* : Creates a compressed .zip file that includes all the files in the project folder. The user is initially prompted for the .etp file of the project and then a .zip file is created having the same name in the project folder. This function is particularly useful for easy exchange of all the project files between different users.

2.3 General Options

The program options window is opened from the navigator *General options* :

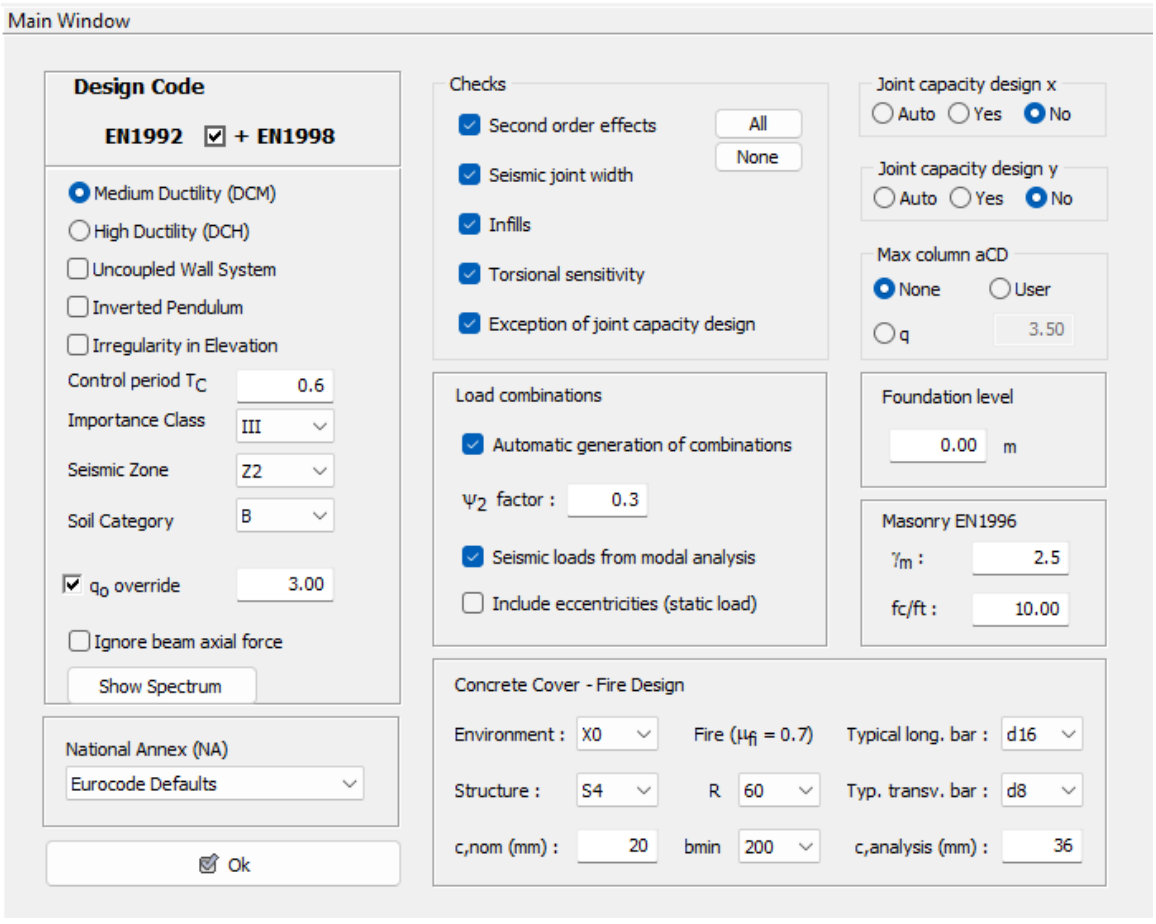


- *Create log file* : The user can select if a full results file is to be created after the design is complete, and can define the file name. The full results file includes separate results for all the load combinations and not only for the most onerous ones that appear in the design calculations issue.
- *Language:* The user has the option to select from a list of available languages for the creation of the design report:

- Relabel: Enables relabelling of the model frames this will be explained extensively later in this manual.
- Create 3d Model : Enables the creation of a 3d model in the main window consisting of the elements that are to be designed . This mode is only available via the API for ETABS or via the xml for S.EN . The reason is that ECtools draws all the necessary geometry through the clients model created in the parent software that the client uses.

2.4 Design options

The design options window opens from the navigator ⇨ ⚙ *Design options* :



- Design Code: The design can be performed according to Eurocode 2 (Eurocode 2 option) provisions only, or the Combination of Eurocode 8 and Eurocode 2 (Eurocode 8 option). Additional information is provided in the Eurocode Theory Manual.

- Ductility Class: For Eurocode 8 design option, the desired ductility class of the building should be selected, either Ductility Class Medium or Ductility Class High.
- Structural System: For Eurocode 8 design option, the structural system has to be defined in order to calculate the behaviour factor q.

$$q=q_o k_w$$

The structural system is automatically defined according to the features of the ETABS/S.EN. model as following, unless it is an uncoupled wall system, an inverted pendulum or there is irregularity in elevation and hence the user has to declare it.

The basic structural systems are shown in the following table along the selection criteria for each system.

Structural System Type	Wall contribution in base shear (%)
Wall System	n>65
Dual System- wall equivalent	65<n<50
Dual System-frame equivalent	50<n<35
Frame System	n<35

- *q_o override:* The user can override the automatically calculated q_o factor (and the resulting q factor) with a value different from the one suggested by EC8, since the program automatically selects the maximum allowed value. The override for smaller values of q_o than the code prescribed ones is allowed, **while the override for higher values of q_o requires additional documentation and justification such as pushover analysis.**
- *Control Period T_C*: The end corner of the spectra plateau has to be defined in order to calculate the μ_φ factor (local ductility of elements).

- *Ignore beam compression:* This option allows the design of the RC beams with zero (0) compression force (the tension force is not altered). This option is valuable for models without diaphragm constraint (as in S.EN.) where a compressive axial force for beams may render their design unconservative, or even unsafe.

- *Fire design-Environmental conditions* : The environmental conditions to which the building is exposed to, are selected according to EC2, which reflects to specific reinforcement cover. This information is used to check if the structural elements conform with EN1992 fire checks for the prescribed fire duration R.

- *Checks* : The checks (design criteria) that are to be performed according to EC8 are defined. These include 2nd order effects, joint widths, fill materials, torsionalsusceptibility[1] and omission of joint capacity design checks[2]. See section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.** for additional information.
- *Load combinations* : The user can choose whether the necessary design load combinations are to be created automatically by ECtools, based on the load cases that have been defined in the ETABS or S.EN. model. Also, the ψ_2 coefficient can be defined and whether the eccentricity load cases are to be included in the above-mentioned combinations. Additional information is provided in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**.

- *Joint capacity design check along the x and y axes* : The user can choose whether the joint capacity checks are performed during the structural design. Available options are a) to be dependant on the outcome of the corresponding joint capacity design omission check based on EC8, b) to be always performed irrespectively of the result of the joint capacity design omission check, c) to be omitted irrespectively of the result of the joint capacity design omission check.

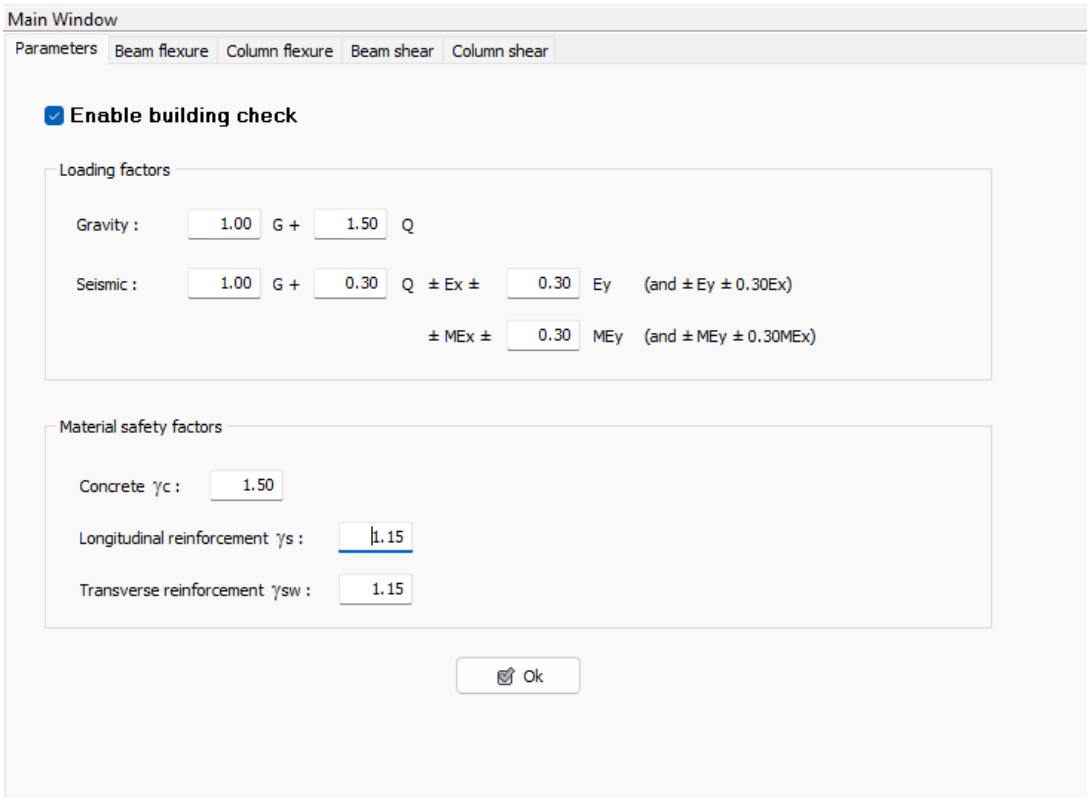
- *Masonry* : The masonry material factor (γ_m) and the ratio between the compressive and tensile capacity (f_c / f_t) of masonry is entered.

- *Foundation level [m]* : By definition, each RC wall base is assumed as the founding level of the corresponding Pier that has been defined in the ETABS/S.EN. model (the node with the lesser vertical co-ordinate z). Should the user wish to define a common founding level for all the RC elements in the model, the level (elevation) that is to be defined as the common founding level is entered in the field. The common RC wall founding level can be altered latter during the design run (described in more detail in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**).

- *National Annex:* As it is known the Eurocodes program has allowed some diversification in the application of the Eurocodes at the several member states. These diversifications are achieved through the Nationally Defined parameters which are included at the National Annexes.

2.5 Building Check Options

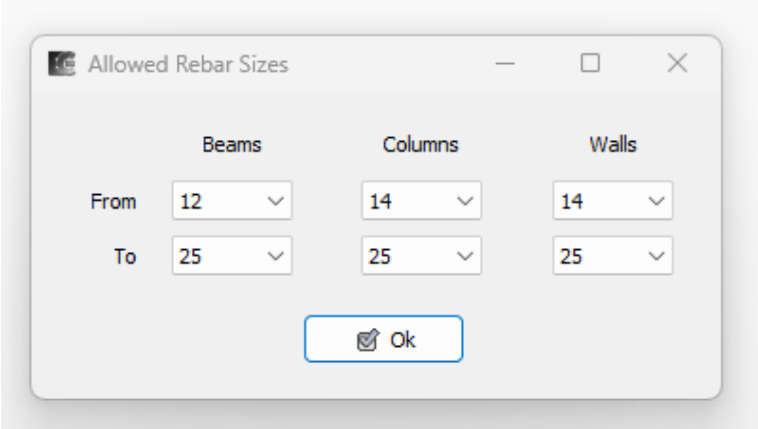
The Building Check options window is opened from the menu *Options* ⇨ 🔧 Building Check *options*:



For checking an existing structure according to EC2 or EC2 and EC8, the option *Enable building Check* must be selected and the load and material safety factors to be input in the relevant fields. These factors will replace the default EC2 factors. During the design run the user is prompted for the existing element reinforcements that are entered in table format, so as to perform the individual structural element capacity checks.

2.6 Reinforcing bar diameter sizes

The reinforcing bar diameter sizes window is accessed from the menu *Options* ⇨ *Rebar sizes* :



The user can set limits to the bar sizes to be used as bending reinforcement in beams, columns and RC walls and which will be proposed in the design calculations issue.

2.6 Running the Design

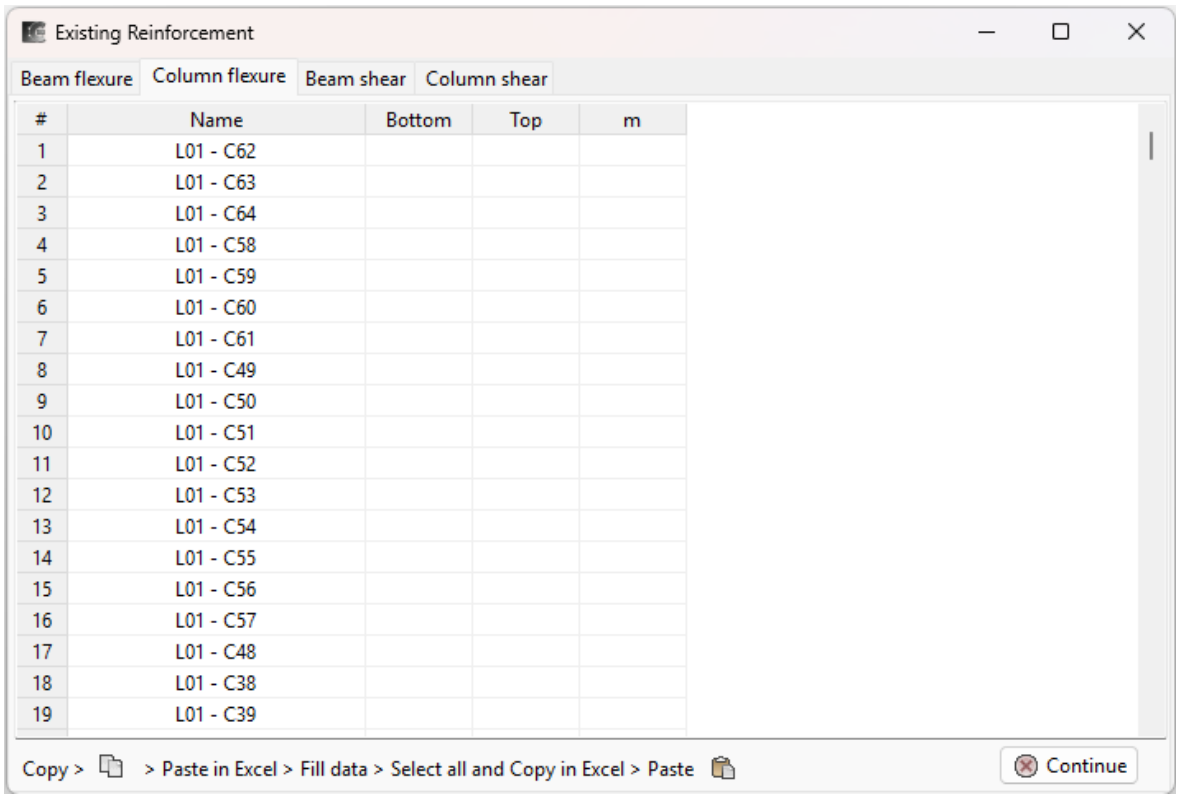
Having entered the Input and Output files in the relevant fields and once the design parameters are set according to the previous sections of this manual (it is recommended at this point to save the project file), the design can be run be clicking on the *Run Design* button.

The design begins by reading the input (via .txt, xml, API), whilst simultaneously information regarding the number of materials, sections, structural elements, etc. contained in that file, is displayed in the adjacent window.

*Note : Should an error message appear through reading of the Input file or if the displayed data does not correspond to the ETABS model, it is possible that not all the available data categories (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**) were selected during the creation of the Input file, or that there is data in the model with names exceeding 9 characters or containing blanks.*

Once the input is read and only in the case the building check option has been activated (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**), the existing beam and column reinforcement data (for bending and shear) input window is displayed in table format[3].

[3] If the *Building Check* license has not been activated, then only one indicative beam and column appear in the table.



The user needs to enter the following data :

- Beam bending tab. Existing bending reinforcement :

Top left	Top centre	Top right
Bottom left	Bottom centre	Bottom right

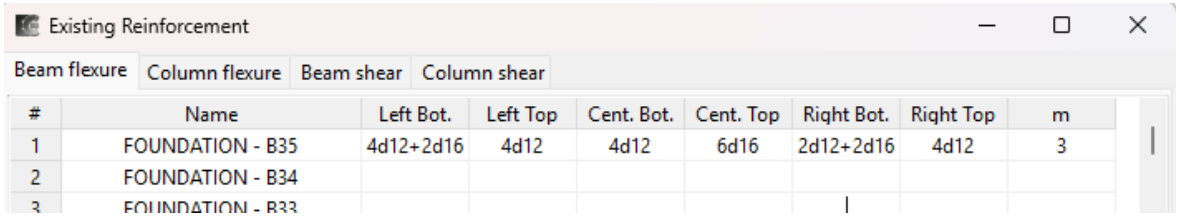
And, optionally, the local behaviour factor (m) that replaces the global behaviour factor (q) for each beam.

- Column bending tab. Existing bending reinforcement :

Top
Bottom

and the local behaviour factor (m) that replaces the global behaviour factor (q) for each column.

Entering the existing bending reinforcement data can be in the form of a single bar size, e.g. 4Φ12[4] or multiple bar sizes e.g. 4Φ12+2Φ16+... . Moreover, the total area of bending reinforcement in cm^2 can be entered, instead of individual bars.



[4] The latin character 'F' or 'D' can be used instead of 'Φ'.

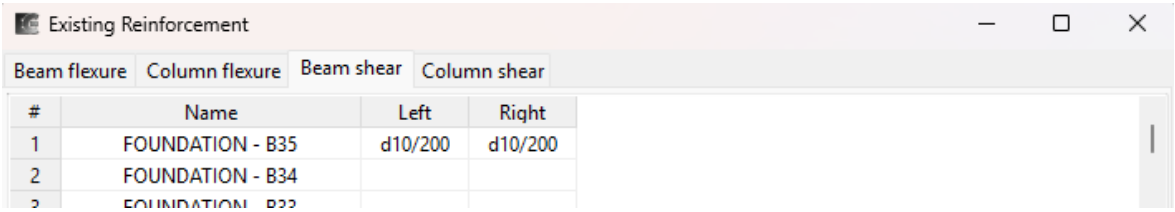
- Beam shear tab. Existing shear reinforcement :

Top left	Top centre	Top right
Bottom left	Bottom centre	Bottom right

- Column shear tab. Existing shear reinforcement :

Top
Bottom

Entering the existing shear reinforcement data can be in the form of a single bar size, e.g. Φ10/200 or multiple bar sizes e.g. Φ10/200+2Φ16+... . Additionally, the link legs can be entered in brackets e.g. Φ10/200(4) (if omitted, it is assumed that the link has two legs). Moreover, the ratio of A_s/s can be entered instead of individual bars.



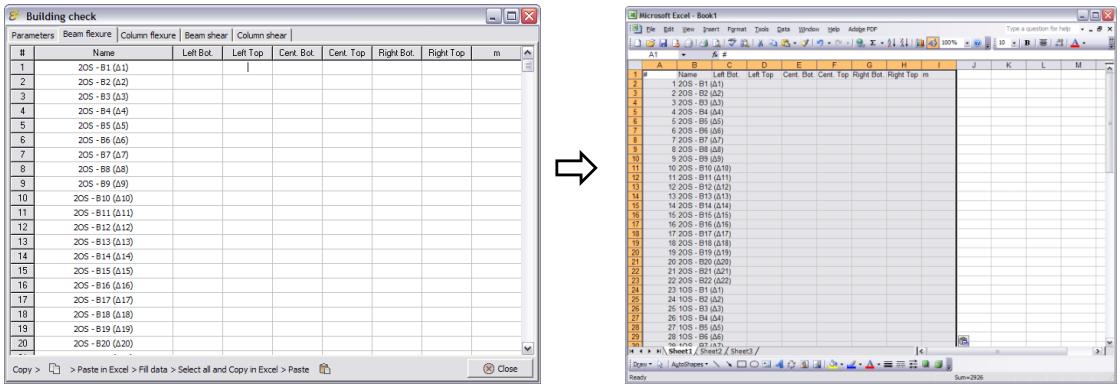
Note 1 : Should any cells be left blank after the process is completed (by clicking on the Close *button), these will be assumed to be zero and also the local behaviour coefficients m will be assigned the value of the global behaviour coefficient q (as entered in the design options, see section* ***Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.****).*

Note 2 : Once the process is completed, the tables containing the existing reinforcement data are saved in the folder where the Input and Output files are

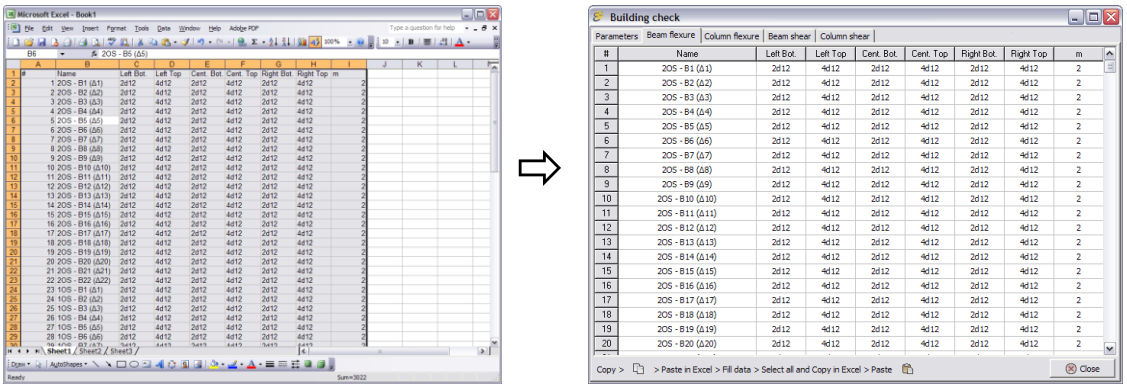
stored under the name <Input file name>_Checkx.txt (x = 1,2,3,4 referring to the 4 respective tables). This ensures that the table data does not have to be re-entered, should the analysis be rerun, and enables editing of the entered data.

To assist the manual input of the existing reinforcement data described above, it is possible to follow the procedure previously described in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.** for multiple creation of masonry elements. Specifically :

1. In each of the four tabs, clicking the Copy button copies the table data to the Clipboard. The data can then be pasted to a spreadsheet (e.g. Microsoft Excel®).

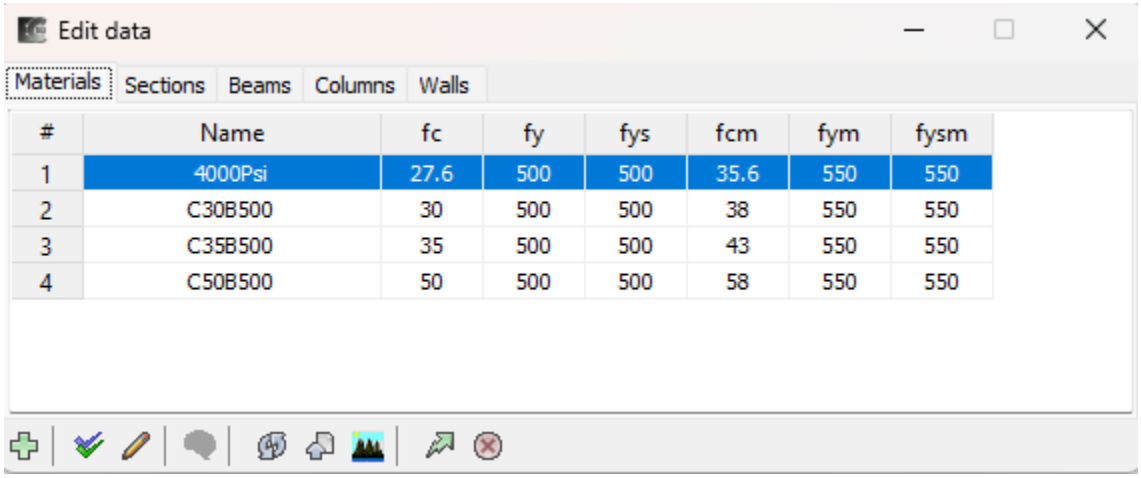


2. The existing reinforcement data for each tab is entered in the spreadsheet as described above. Next, select the whole table (Ctrl-A) and copy (Ctrl-C) and paste it in the relevant tab by clicking the button.



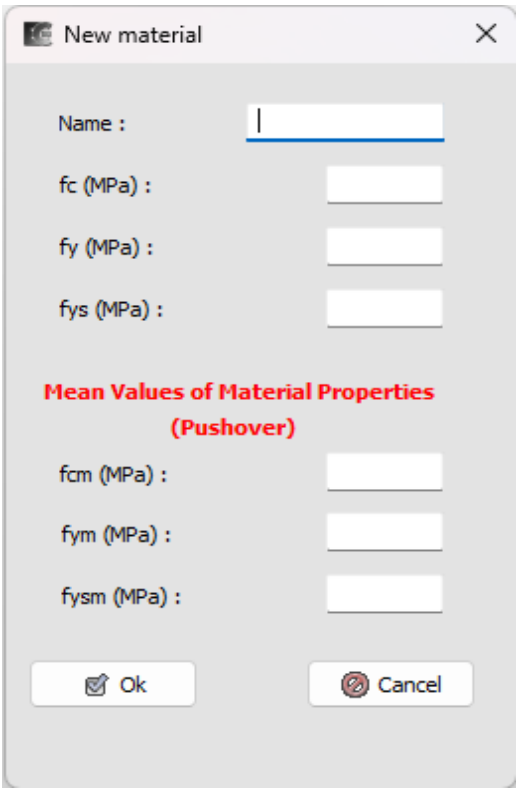
Next, the data editing window is displayed, where the user can alter the model data before the Output file is read and the design is run. The data editing window includes the following tabs :

- *Materials* : Displays the material names and properties table included in the model. The following editing options are available :

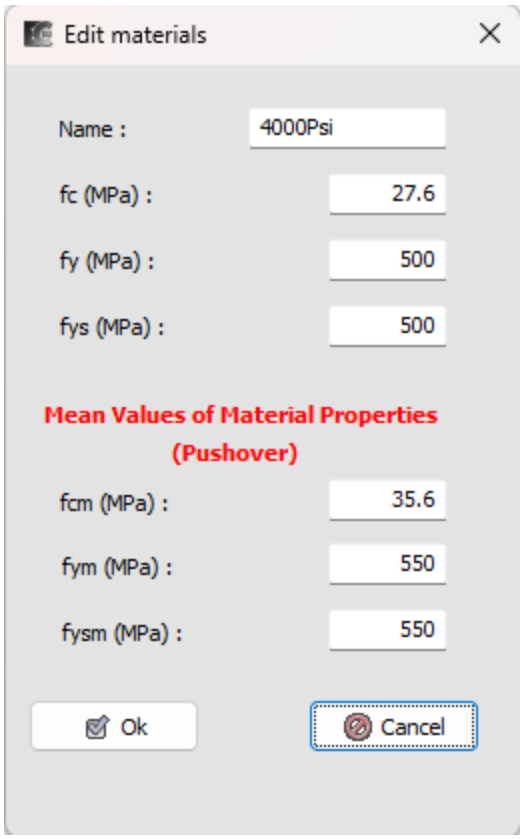


#	Name	fc	fy	fys	fcm	fym	fysm
1	4000Psi	27.6	500	500	35.6	550	550
2	C30B500	30	500	500	38	550	550
3	C35B500	35	500	500	43	550	550
4	C50B500	50	500	500	58	550	550

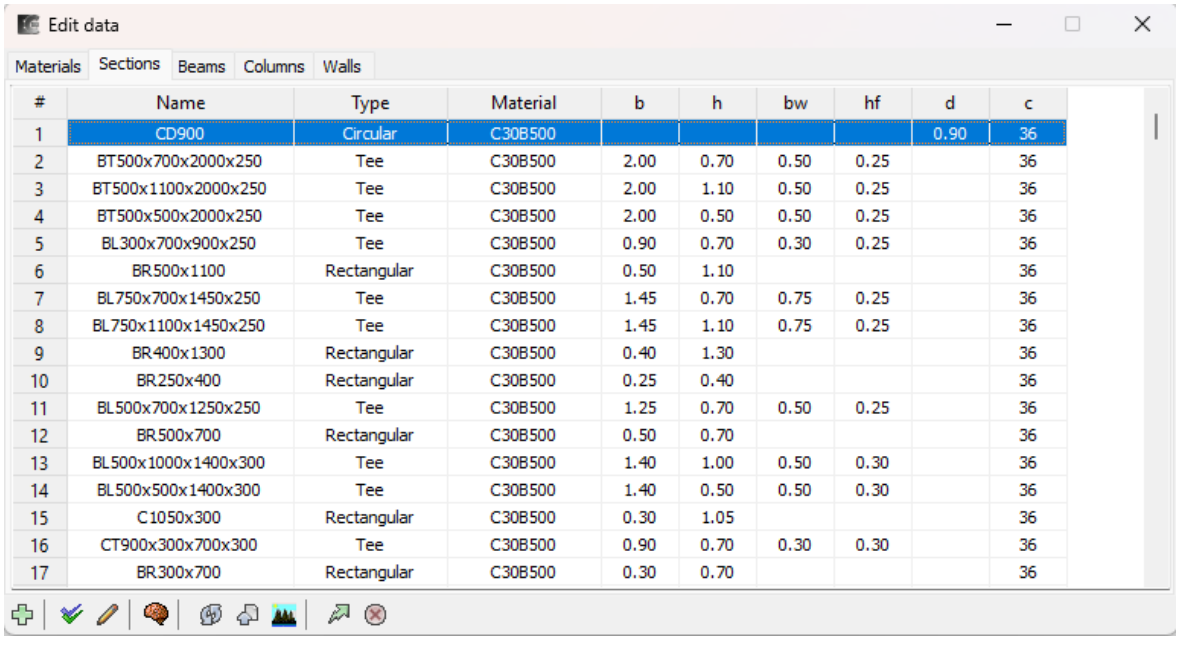
button : Adds a new material in the table. Material properties are entered in the figure shown on the right. For masonry materials, the f_{vko} and f_{lim} properties are entered initially at the f_y and f_{ys} fields respectively, enabling the program to automatically recognise these as masonry materials.



button or double click : Edits the selected material properties. It is also possible to edit multiple materials from the table either by clicking the particular button or by right-clicking on the multiple selection. For multiple material editing, all the selected materials should be of the same type, i.e. all concrete or all masonry.

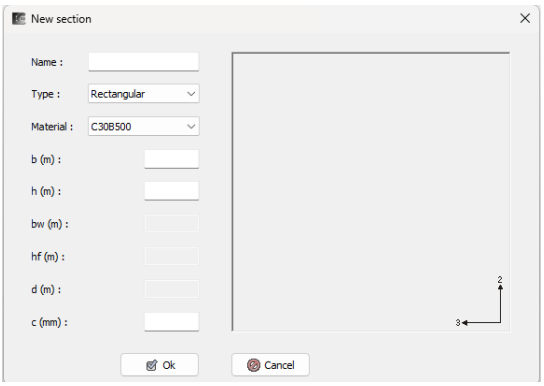


- *Sections* : Displays the section names and characteristics table contained in the model. The following editing options are available :

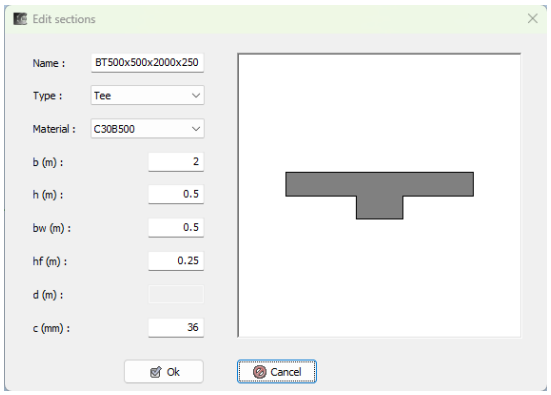


#	Name	Type	Material	b	h	bw	hf	d	c
1	CD900	Circular	C30B500					0.90	36
2	BT500x700x2000x250	Tee	C30B500	2.00	0.70	0.50	0.25		36
3	BT500x1100x2000x250	Tee	C30B500	2.00	1.10	0.50	0.25		36
4	BT500x500x2000x250	Tee	C30B500	2.00	0.50	0.50	0.25		36
5	BL300x700x900x250	Tee	C30B500	0.90	0.70	0.30	0.25		36
6	BR500x1100	Rectangular	C30B500	0.50	1.10				36
7	BL750x700x1450x250	Tee	C30B500	1.45	0.70	0.75	0.25		36
8	BL750x1100x1450x250	Tee	C30B500	1.45	1.10	0.75	0.25		36
9	BR400x1300	Rectangular	C30B500	0.40	1.30				36
10	BR250x400	Rectangular	C30B500	0.25	0.40				36
11	BL500x700x1250x250	Tee	C30B500	1.25	0.70	0.50	0.25		36
12	BR500x700	Rectangular	C30B500	0.50	0.70				36
13	BL500x1000x1400x300	Tee	C30B500	1.40	1.00	0.50	0.30		36
14	BL500x500x1400x300	Tee	C30B500	1.40	0.50	0.50	0.30		36
15	C1050x300	Rectangular	C30B500	0.30	1.05				36
16	CT900x300x700x300	Tee	C30B500	0.90	0.70	0.30	0.30		36
17	BR300x700	Rectangular	C30B500	0.30	0.70				36

button : Adds a new section in the table. Section properties are entered in the window shown on the right. Section type can be rectangular, T-, L-, U- or circular section. Depending on the section type, the relevant data input fields are activated and the section outline is drawn for checking purposes.

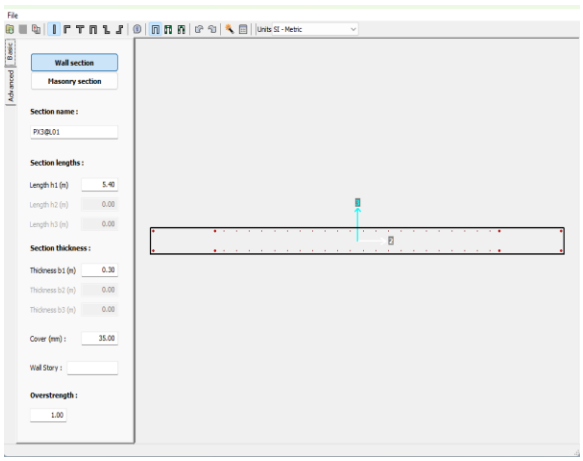


button or double click : Edits the selected section geometry. It is also possible to edit multiple sections from the table either by clicking the particular button or by right-clicking on the multiple selection. For multiple sections editing, all the selected sections should be of the same type.

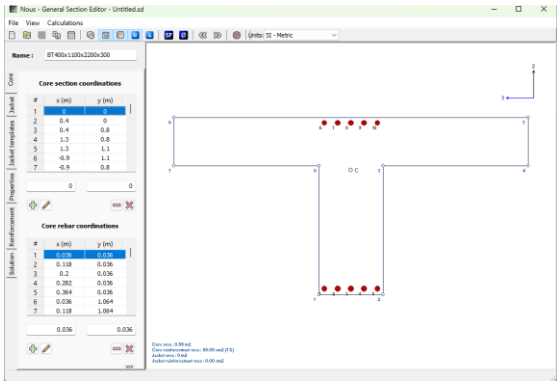


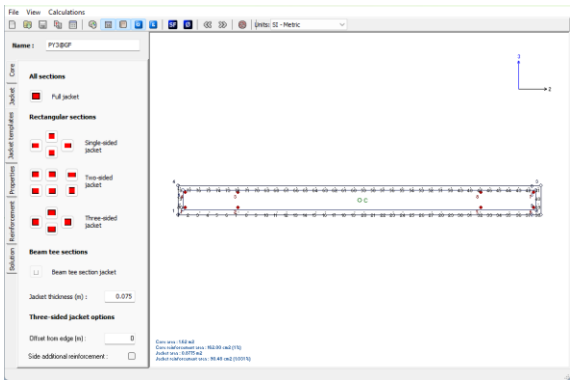
If the selected section is an RC wall or masonry wall section, then it is opened in the corresponding editor, as described in section **Σφάλμα! Το αρχείο**

προέλευσης της αναφοράς δεν βρέθηκε..

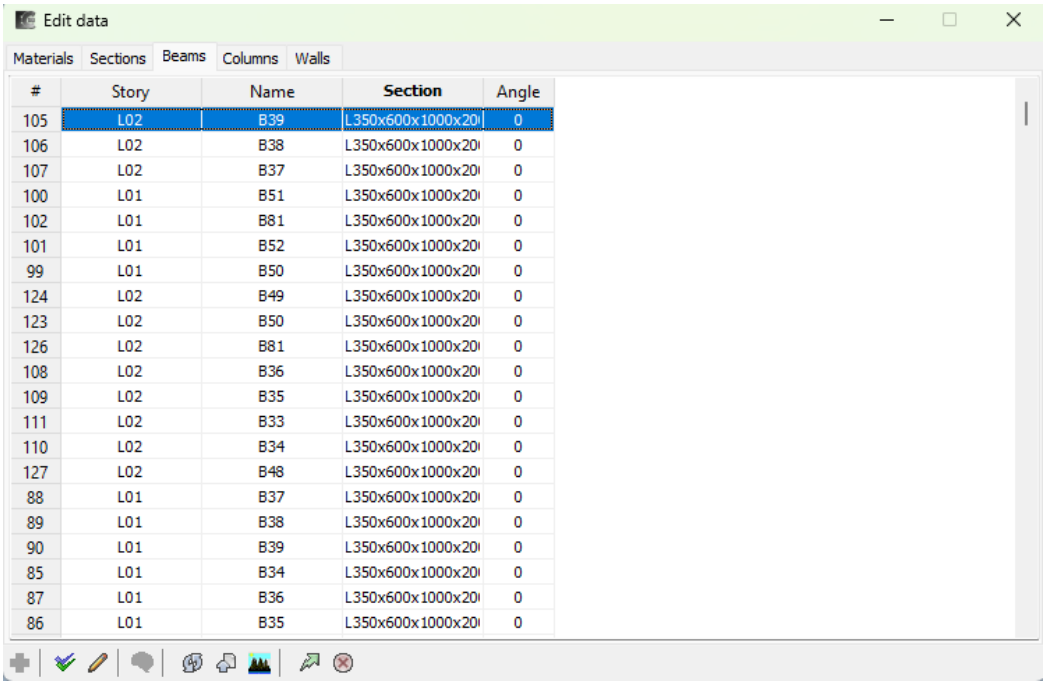


🧠 button : Opens the selected section for editing in Nous (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**). If the section is modified (e.g. by adding a jacket), then it is saved on the hard drive as a .sd file and is entered in the table as a jacketed section by clicking the update button (described latter on). Also, jacketed sections can be edited in Nous by double-clicking.

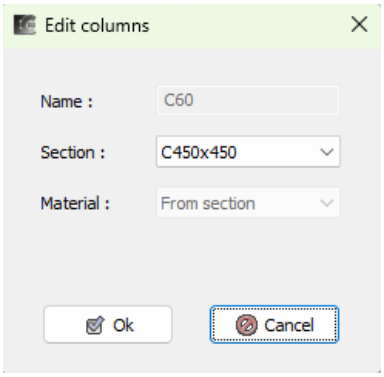




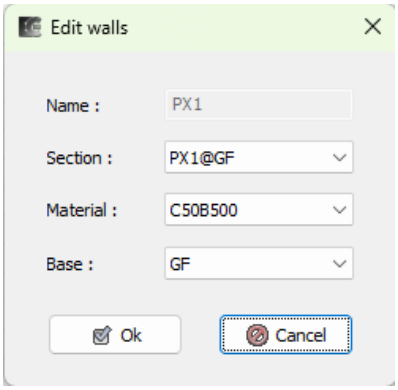
- *Beams / Columns / RC Walls / Masonry Walls* : Displays the structural element names and characteristics contained in the model. Editing of structural elements is performed by clicking on the button or by double-clicking for a single selection or by right-clicking for multiple selection.



For beams and columns, only the section names can be modified. The section material will be that of the newly applied section.



For RC walls the section, material (choice of concrete materials only) and base (founding level) name that is to be used in design can be modified. Initially, the founding level of every RC wall is assumed to be the respective Pier base that has been defined in the ETABS model or the common founding level defined in the design options (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**).



For masonry elements, the section and material name (choice of masonry materials only) can be edited.

Note : The sections table displays those jacketed sections that have been created in ETABS Section Designer as polygonal lines with point reinforcements.

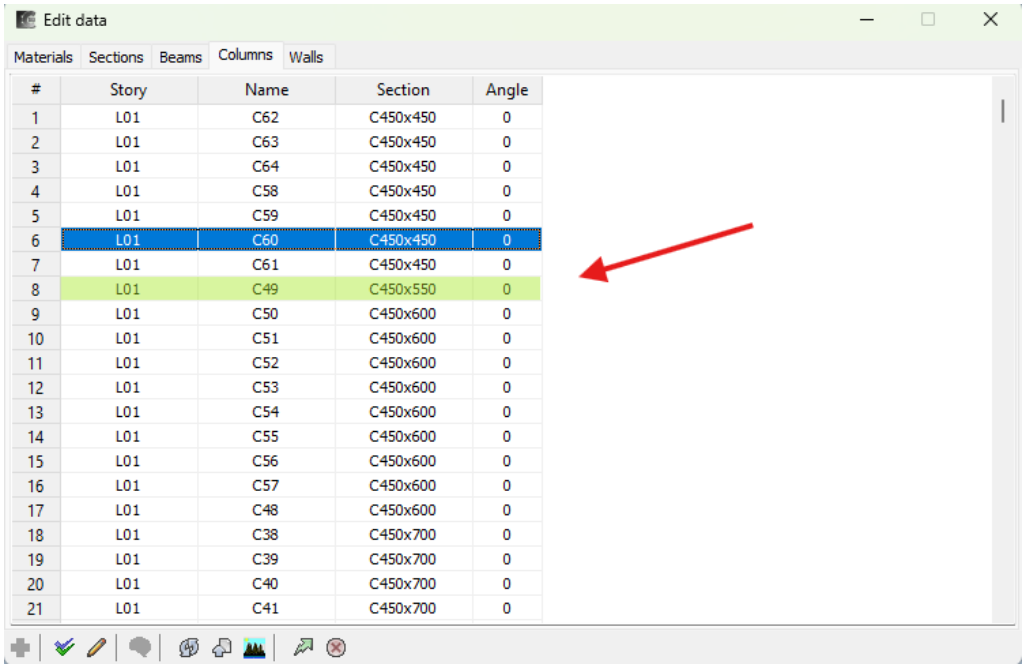
- The general options of the data editing window, irrespective of the selected tab, are as follows :

button or Ctrl-A : Selects all the table data. Clicking again selects only the top row of the table.

button : Updates the jacketed sections in the table from the files contained in the folder where the Input and Output files are stored. Updating is necessary before the design is run, if any additional jacketed sections are created in Nous.

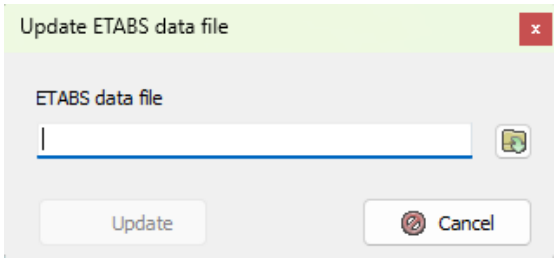
button : Replaces existing sections with jacketed sections (strengthened sections) located in the same folder where the Input and Output[5] files are stored. These strengthened sections are highlighted in green (see figure below) in the column, RC and masonry wall tables. The strengthened section creation process is described in detail in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**.

[5] Requires the *Strengthening* license to be activated.



#	Story	Name	Section	Angle
1	L01	C62	C450x450	0
2	L01	C63	C450x450	0
3	L01	C64	C450x450	0
4	L01	C58	C450x450	0
5	L01	C59	C450x450	0
6	L01	C60	C450x450	0
7	L01	C61	C450x450	0
8	L01	C49	C450x550	0
9	L01	C50	C450x600	0
10	L01	C51	C450x600	0
11	L01	C52	C450x600	0
12	L01	C53	C450x600	0
13	L01	C54	C450x600	0
14	L01	C55	C450x600	0
15	L01	C56	C450x600	0
16	L01	C57	C450x600	0
17	L01	C48	C450x600	0
18	L01	C38	C450x700	0
19	L01	C39	C450x700	0
20	L01	C40	C450x700	0
21	L01	C41	C450x700	0

button : Updates the ETABS data file (file with a .E2K or .$ET extension) to all the modifications that have been performed in the data editing tables window. (not available in S.EN. mode)



The ETABS data file name that contains the model is entered in the next window and by clicking the *Update* button, the user is prompted for the new file name to be created. This new file can be loaded in the ETABS environment from the menu *File ⇨ Import ⇨ ETABS .E2K Text File*. It is noted that all the jacketed sections created in Nous, are input as ETABS Section Designer sections. Also, all the changes to the correlations between sections and **frame** elements that have been performed in the data editing window are incorporated in the new data file.

button : Cancels the design procedure. The information and data editing window is closed and the design process reverts to the starting stage (start of current section).

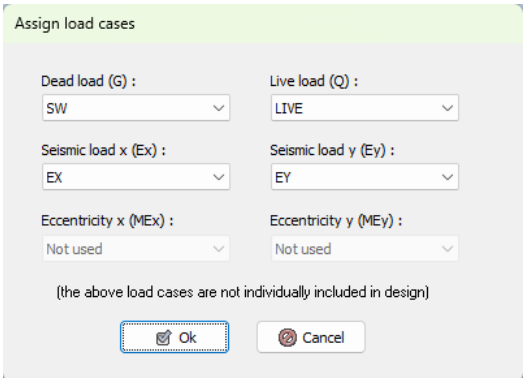
button : Starts the design process. Once all the necessary changes have been performed in the data editing window, the process moves on to read the results file (Output) and perform the design.

Note : All the columns displayed in the window can be alphabetically sorted by left-clicking on the title of each column. By clicking again, the column is sorted in reverse alphabetical order. The title of the sorted column is displayed in ***bold****.*

Running the design – Reading the results (Output)

Depending on the selected options, the following windows for additional data input are displayed when the design is run :

- Load case correlation : If the automated creation of loading combinations has been selected in the design options (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**) the user is asked to correlate the load cases created in ETABS with the G, Q, Ex, Ey, MEx and MEy load cases (the last two are asked for only if eccentricity load cases have been selected).



Note 1 : ETABS load cases that are correlated in the above window are not included in the design as stand alone load cases.

Note 2 : For designs according to EC2 + EC8, even if the automated creation of load combinations has not been selected in the design options, the user is

prompted to correlate the load cases G, Q, Ex and Ey for performing the various checks against the clauses of the Eurocodes.

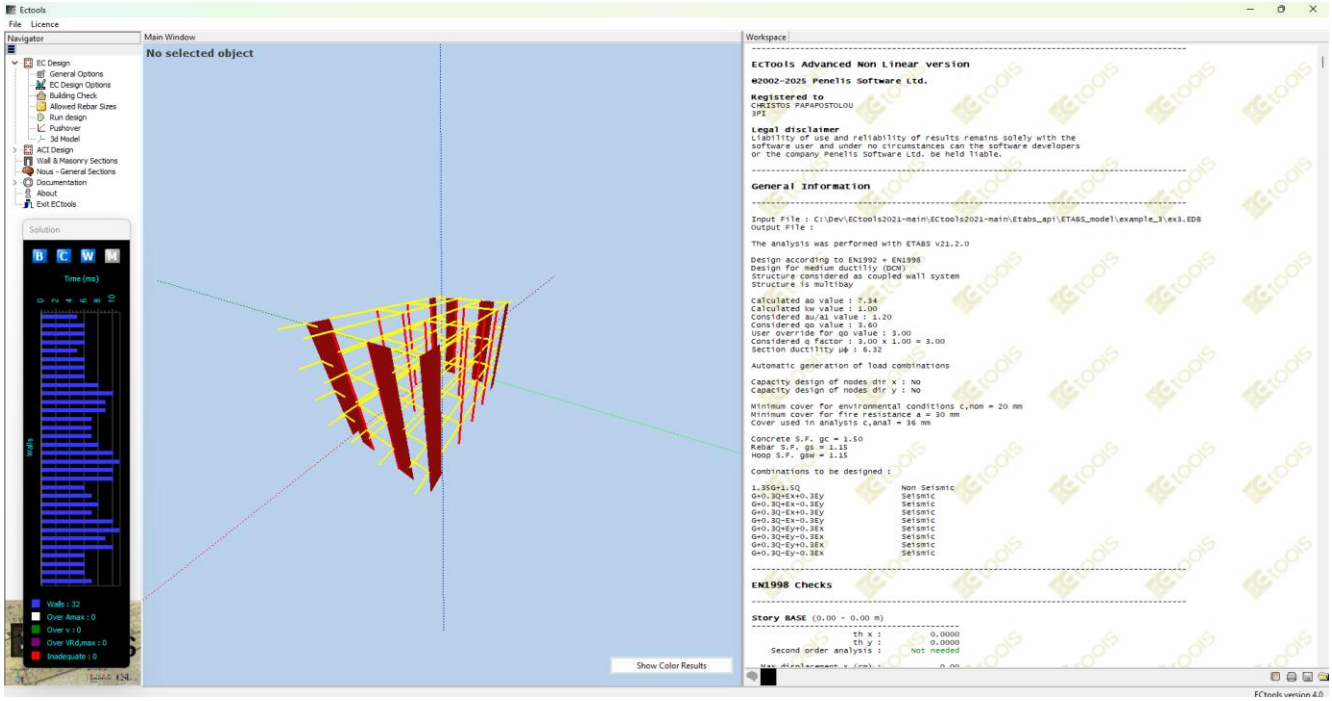
- Define additional earthquake load combinations: For designs according to EC2 + EC8 and for the load cases (and combinations) created in ETABS (except those already correlated in the previous window) the user is prompted for whether these are to be treated as earthquake combinations when designing against EC8. Clicking on the square box (☐) by the load case or combination name selects that case or combination as an earthquake one (✓).

Design calculations issue

Depending on the number of the structural elements, the load combinations and the speed of the computer used, the design process can be completed in a few seconds to a few minutes. Aggregate data of the design process and a bar-chart indicating the solution time for each element (in milliseconds – ms) are displayed in the information window during the design run. The design calculations issue window is displayed once the design process is complete.
The solution time bar-chart includes the following categories :

button : Displays the beam data. The various colours represent the following :

- Beams that have been designed or that are adequate in the cross-section capacity design checks (safety factor larger than unity).
- Beams where the maximum reinforcement percentage (A_{max}) as defined in the Eurocodes has been exceeded.
- Beams where the concrete core shear capacity (V_{Rd2}) has been exceeded in design.
- Beams that are inadequate in the cross-section capacity design checks (safety factor smaller than unity).



button : Displays the column data. The various colours represent:

- Columns that have been designed or that are adequate in the cross-section capacity design checks (safety factor larger than unity).
- Columns where the maximum reinforcement percentage (A_{max}) as defined in the Eurocodes has been exceeded.
- Columns where the maximum ratio (v) of applied axial force to axial capacity as defined in the Eurocodes has been exceeded.
- Columns where the concrete core shear capacity (V_{Rd2}) has been exceeded in design.
- Columns that are inadequate in the cross-section capacity design checks (safety factor smaller than unity).

button : Displays the RC walls data. The various colours represent:

- RC walls that have been designed or that are adequate in the cross-section capacity design checks (safety factor larger than unity).
- RC walls where the maximum reinforcement percentage (A_{max}) as defined in the Eurocodes has been exceeded.
- RC walls where the maximum ratio (v) of applied axial force to axial capacity as defined in the Eurocodes has been exceeded.
- RC walls where the concrete core shear capacity (V_{Rd2}) has been exceeded in design.
- RC walls with jacketed section (.sd) that are inadequate in the cross-section capacity design checks (safety factor smaller than unity).

button : Displays the masonry walls data. The various colours represent the following :

- Masonry walls that are adequate in the cross-section capacity design checks (safety factor larger than unity).
- Masonry walls that are inadequate in the cross-section capacity design checks (safety factor smaller than unity).

Note : In the cases mentioned above, if the section is a jacketed one (.sd extension) or a masonry wall, then the design is not performed but merely a cross-section adequacy check carried out. Consequently, the only possible states in the bar-chart are (adequate) or (inadequate).

On the solution time bar-chart the following actions can be performed :

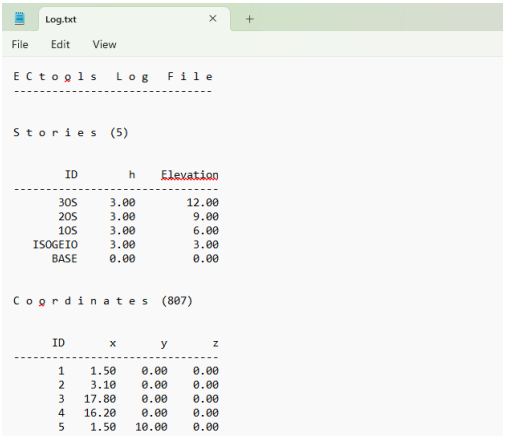
- Left click ⇨ Displays the selected element in the design calculations issue.

- Double left click ⇨ Opens the selected element in Nous. This action is useful for creating strengthened sections, as described in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**.
- Scroll wheel up/down ⇨ Scrolls through elements up or down respectively.
- Left button drag ⇨ Zoom in/out on the bar-chart.
- Right button drag ⇨ Pans up/down on the bar-chart.

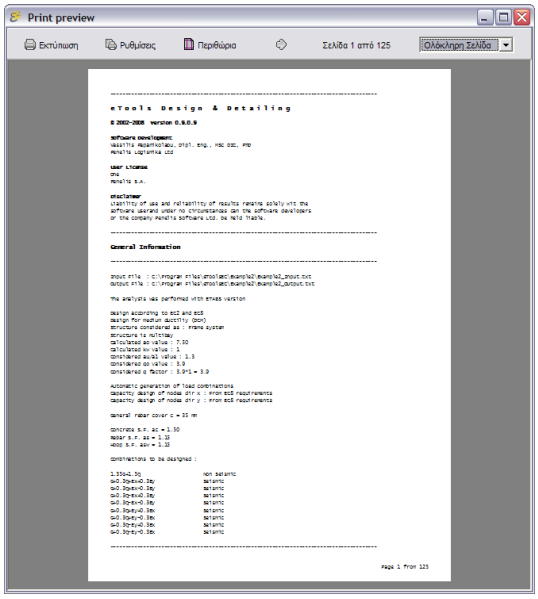
When a structural element is selected on the bar-chart, the design/check results are displayed in the design calculations issue window. The name of the selected element is displayed at the lower part of the window in the form <floor> - <element>. Additional functions are as follows :

🧠 button : Opens the selected element in Nous. Performs the same action as double left-clicking on the solution time bar-chart.

📄 button : Displays the full results file, if this has been enabled in general program options (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**).



button : Print preview of the design calculations issue. The following options are available in the print preview window :



Print

Printer setup

Margins (show/hide)

Previous page

Next page

Zoom in/out on page

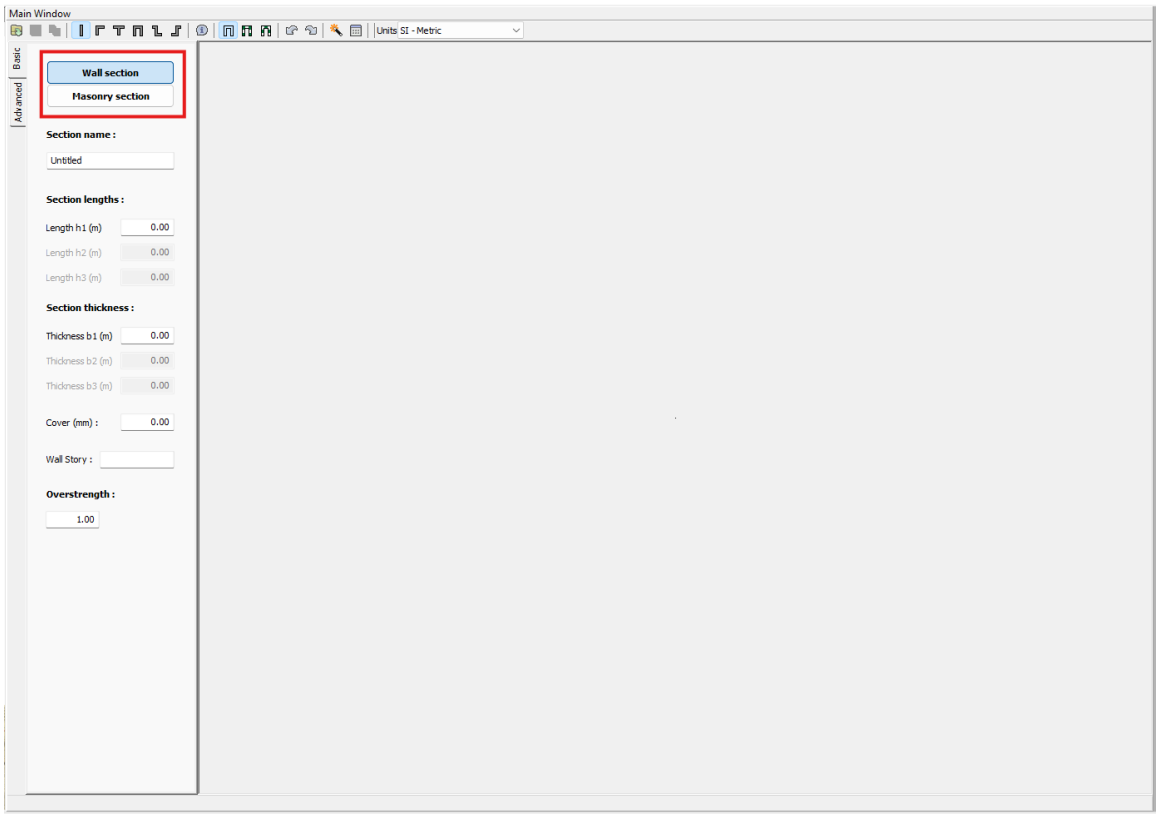
Close preview

button : Print design calculations issue.

button : Save design calculations issue in rich (.rtf) or plain (.txt) text format.

3. Masonry wall creation

The RC/masonry wall editor is run from the corresponding option of the main menu. Initially, the user must choose between RC or masonry section from the respective options :

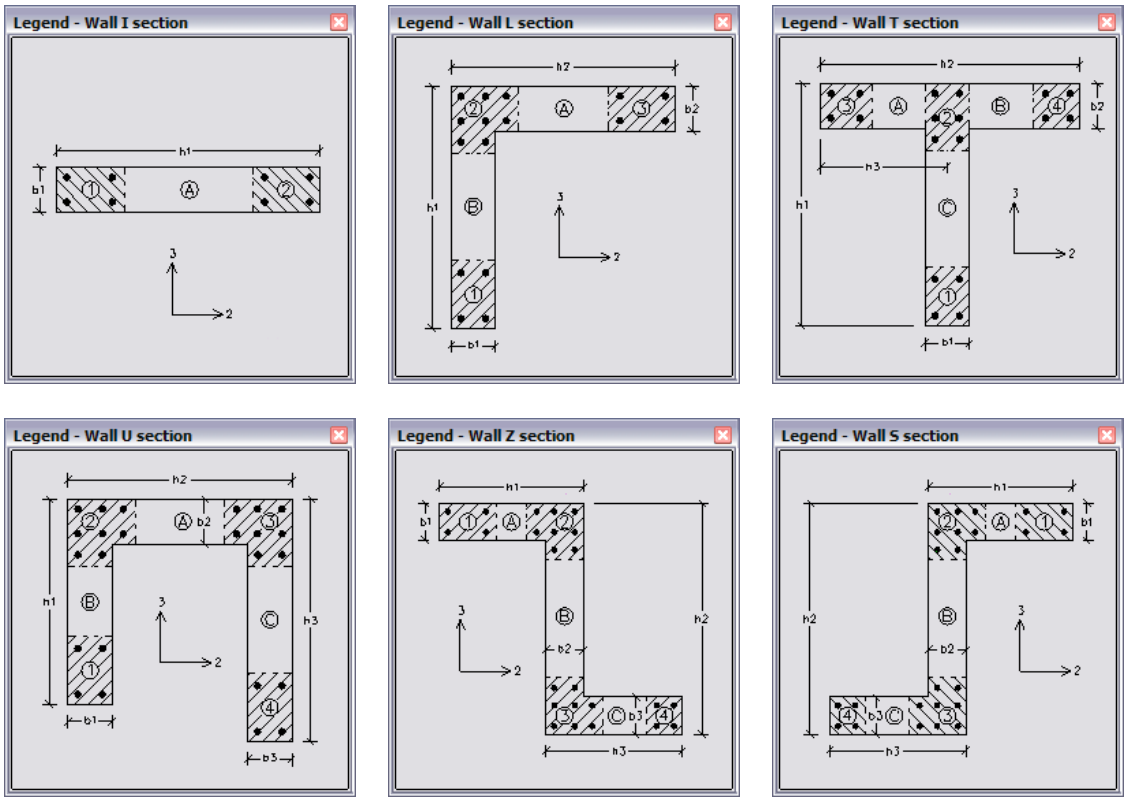


3.1 RC wall creation

First the section type is selected from the available options from the respective toolbar buttons :

- Rectangular section. Geometric data : b1, h1.
- L-section. Geometric data: b1, b2, h1, h2.
- T- section. Geometric data: b1, b2, h1, h2, h3.
- U- section. Geometric data : b1, b2, b3, h1, h2, h3.
- Z- section. Geometric data: b1, b2, b3, h1, h2, h3.
- S- section. Geometric data: b1, b2, b3, h1, h2, h3.

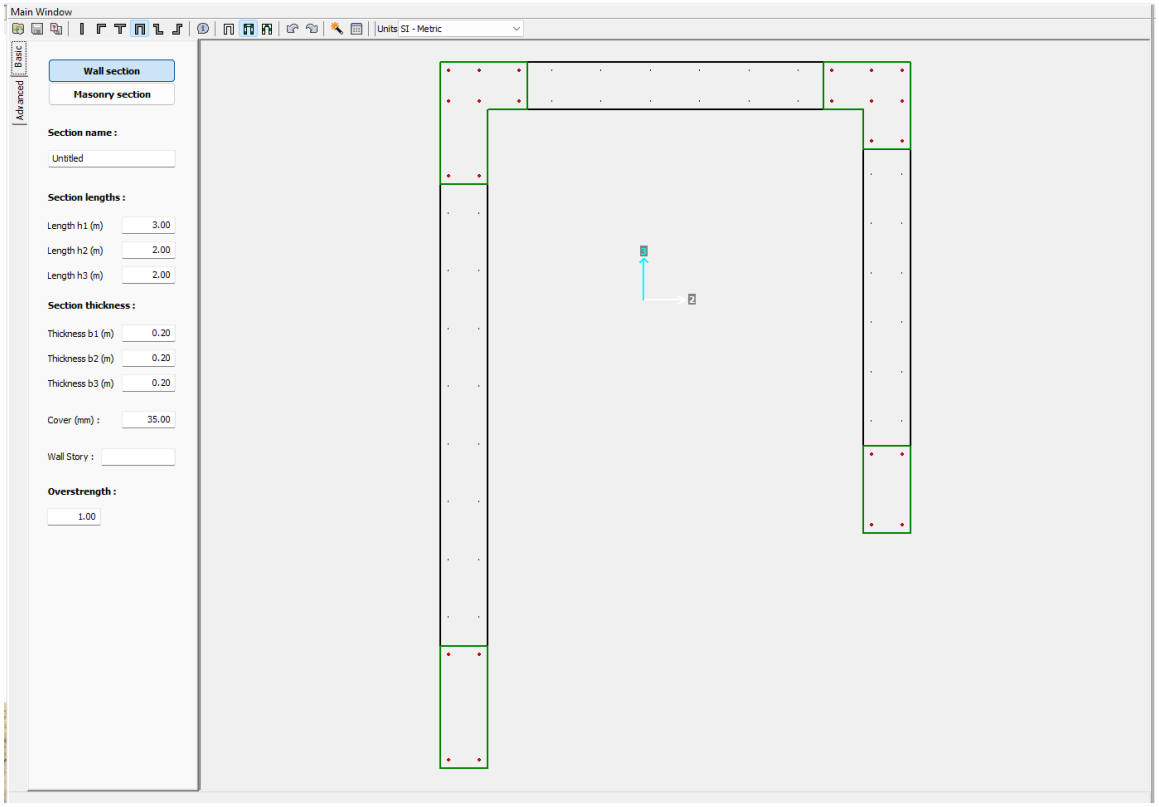
The geometric data for each section type is displayed in the following figure (these are displayed on screen by clicking the button) :



Having input the geometric data of the selected section type in the corresponding fields and the reinforcement cover, the section shape is created by clicking the button. Particular attention must be drawn at this point, since the local axes 2 and 3 displayed in the editor **must match** the respective axes in the ETABS model (on plan). Should this not be the case, the section must be rotated using the appropriate buttons and by +90º and -90º respectively (the axes remain constant).

When the section is ready, its name should be input in the relevant field, which **must match** the respective Pier name assigned in ETABS. Next, the section file must be saved in the same folder where the *Input* and *Output* design files were

stored, as described in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**. To save the file, either click the button. The file name will be the same as the section name[6], with a .wal extension, e.g. WALL1.wal in the following example :



All the aforementioned activities are not required when in S.EN. mode, since these sections are automatically created.

Additional editor options are as follows :

- button: Opens a prepared RC wall section (file with a .wal extension) from the hard drive.

- button: Saves on the hard drive a previously stored section under a different name.

[6] It is reminded that there is a limit of 9 characters (no spaces allowed) for the section name.

button : Displays the RC wall outline (initial figure).

button : Displays the outline of the RC wall's end columns.

button : Displays the outline of the RC wall's core(s).

button : Displays the outline of the RC wall's core(s).

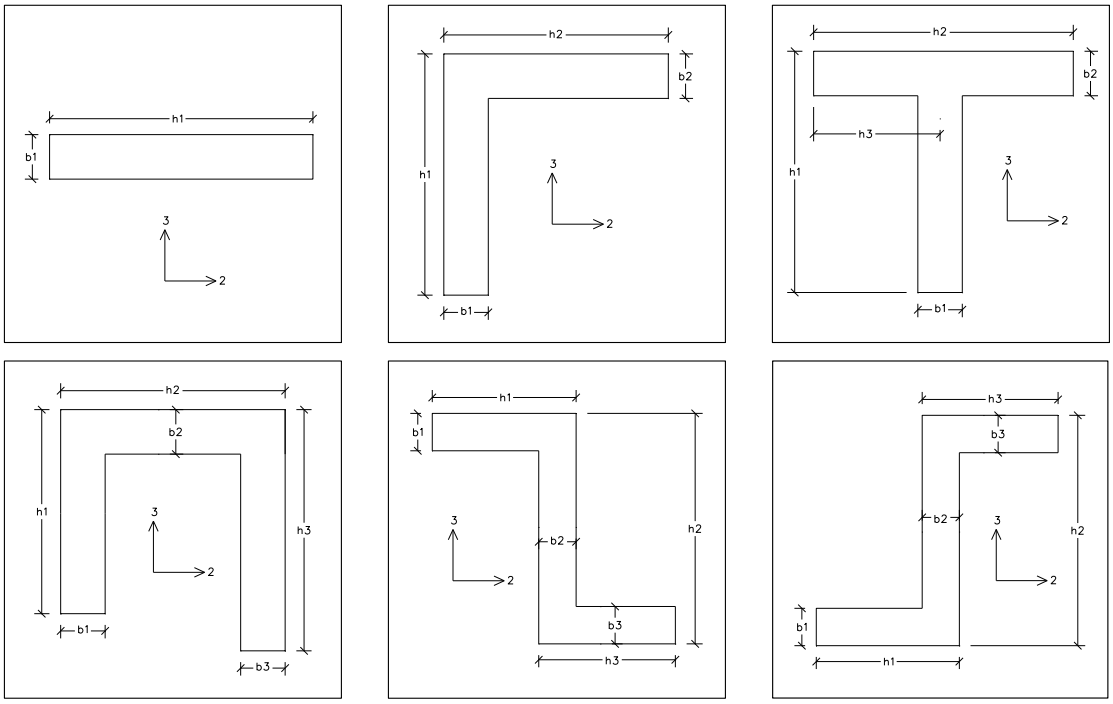
- *Optional parameter Wall Story* : Should the RC wall dimensions vary from floor to floor in the ETABS model, then separate sections (.wal files) must be created having the same name (which must always match the corresponding Pier name in the ETABS model) and by inputting the corresponding floor name in the relevant field. The created file has the form <section name>@<floor name> (e.g. WALL1@STORY1) and is automatically identified during the design run. (this ia done automatically when in S.EN. mode)

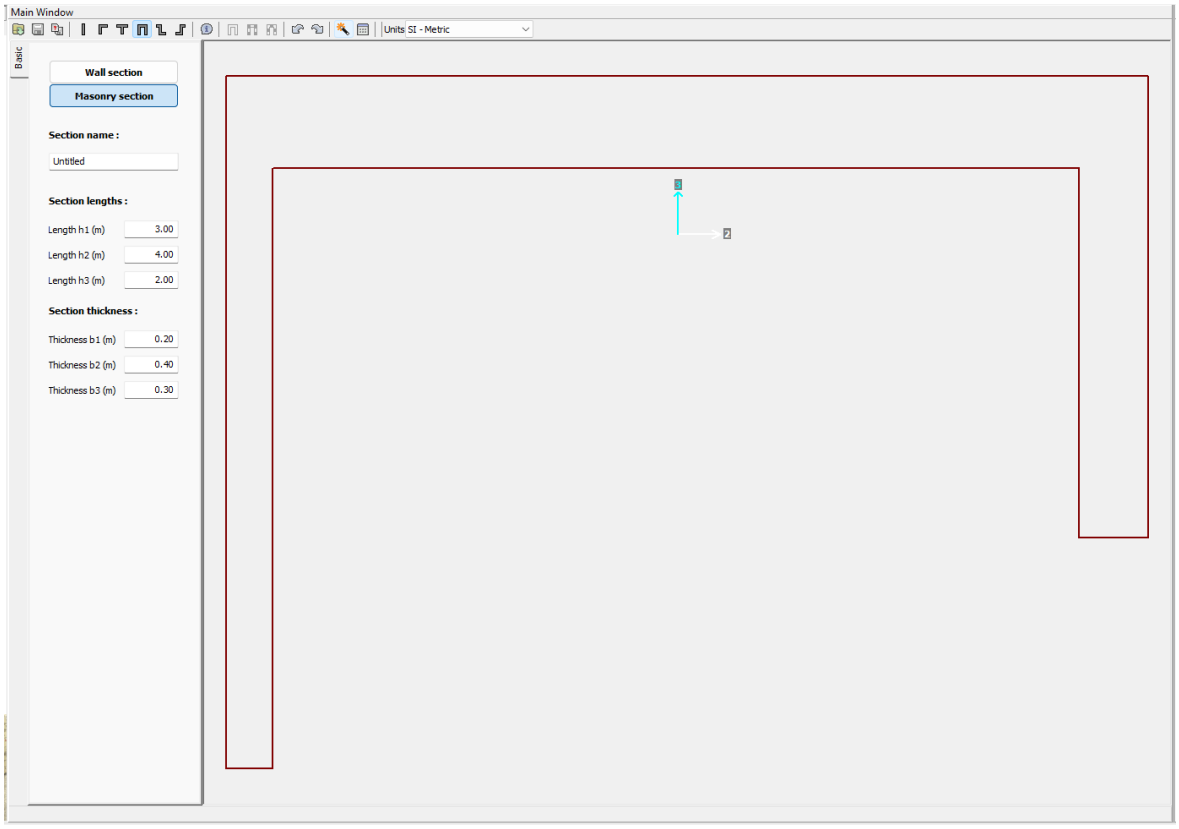
3.2 Masonry wall creation

The same procedure as for RC wall section creation applies for the masonry wall section creation. The only differences are that the created file names have the .mas extension (e.g. MASONRY1.MAS) and their creation is not subject to regulatory clauses governing minimum dimensions. It is again highlighted that the local axes 2 and 3 displayed in the editor **must match** those in the ETABS model and the editor section name **must match** the corresponding Pier/Spandrel name assigned in the ETABS model.

This of course is not the case for the software in S.EN. mode, since then the *.mas sections are created automatically, when they correspond to one of the predefined shapes.

The geometric data for each section type are shown in the following figure (these can be displayed on the screen by clicking the button) :

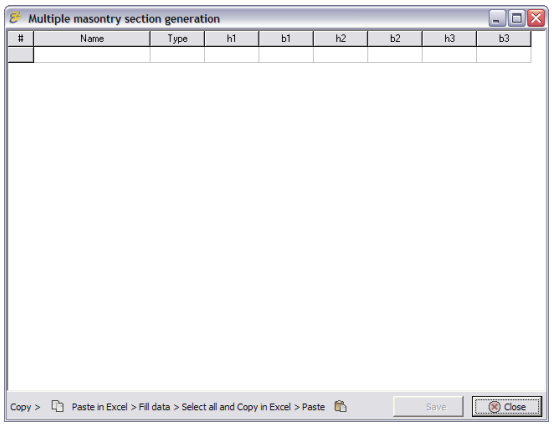


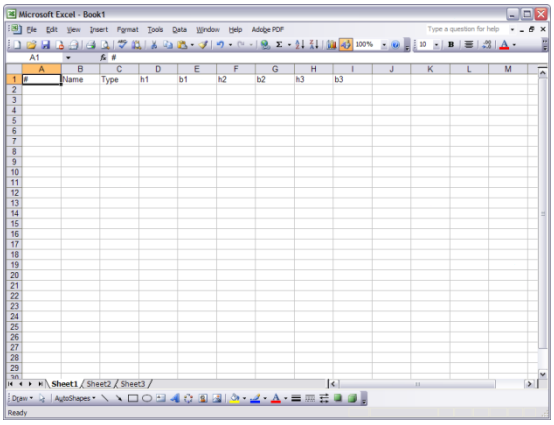


Usually in masonry wall models the number of masonry areas and intermediate RC beams (lateral supports) can be very large and the creation of the necessary

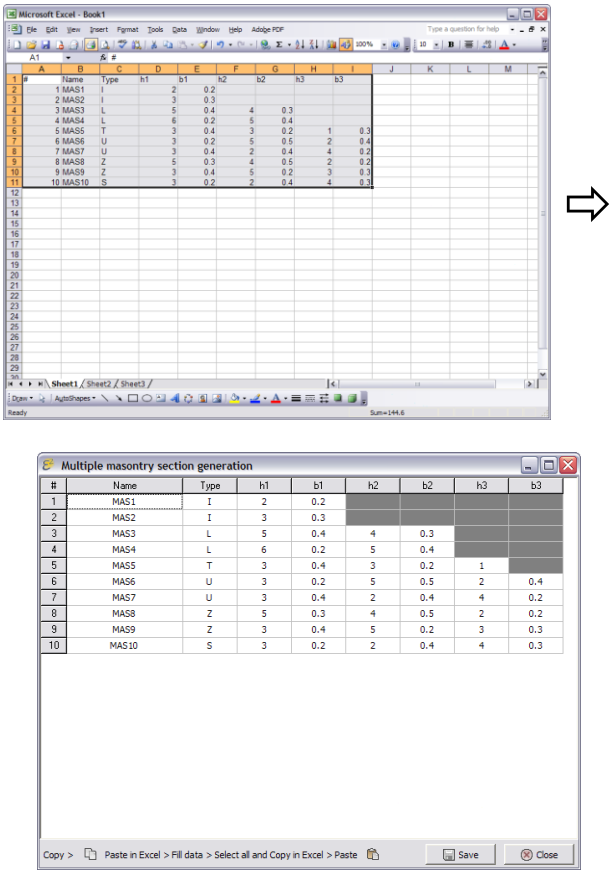
elements following the above procedure can be cumbersome. To facilitate the creation of the masonry sections, when in Etabs mode, there is the option to create multiple masonry elements, which is displayed by clicking on the button at the toolbar. The procedure described below is to be followed in the popped up window :

1. By clicking the button (Copy) the form is copied on the Clipboard. Next, it is pasted in a spreadsheet (e.g. Microsoft Excel®).





2. The geometric data is filled in the spreadsheet for each section (one per line). In the section type it is filled I, G, T, P, Z, S for rectangular, L-, T-, U-, Z- or S-section respectively. Next, select the whole table (Ctrl-A) and copy (Ctrl-C) and paste it in the multiple element creation window by clicking the button.



3. By clicking the *Save* button, the folder[7] where all the masonry wall section files (.mas) are saved is selected. Should any of the sections require further editing (e.g. rotation), they can be opened in the editor, modified accordingly and saved.

All the above are only used for reviewing the sections when in S.EN. mode.

4. Arbitrary section designer - Nous

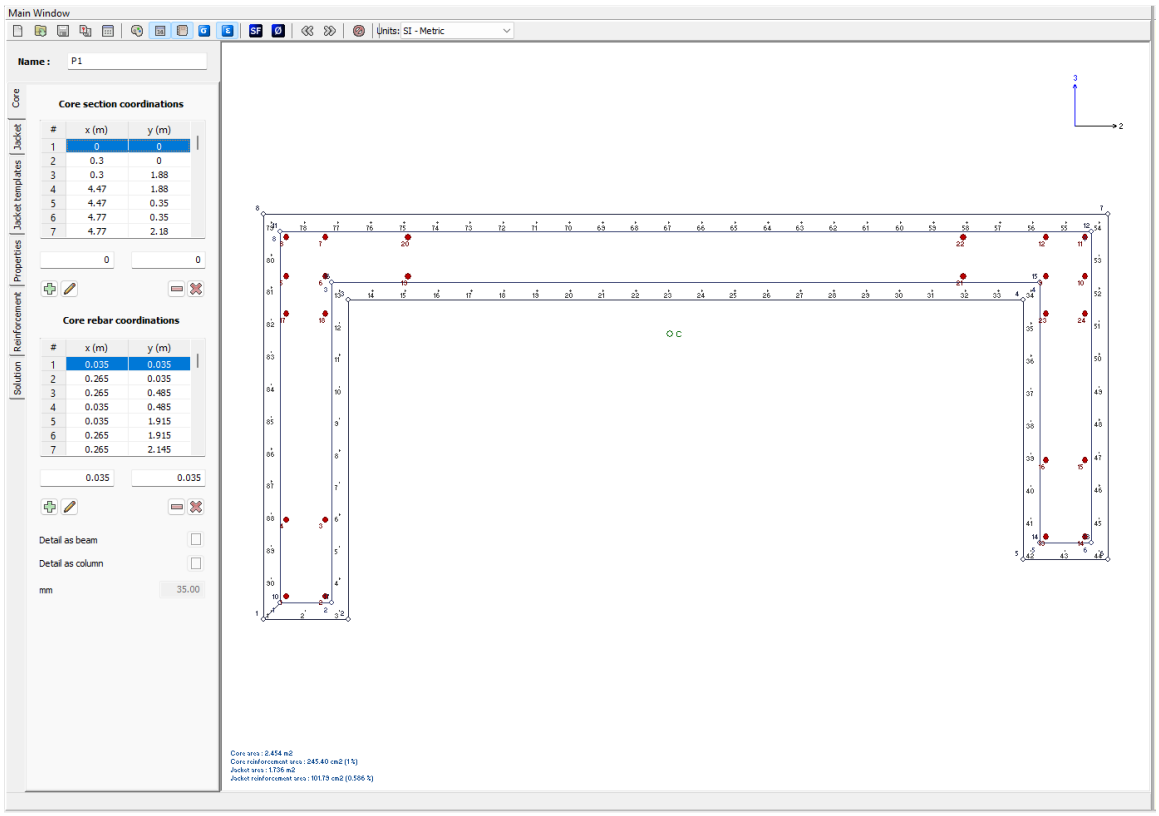
Nous is an autonomous program for check and design of arbitrary sections. It provides the option of geometrically defining a new arbitrary section with the option of adding a jacket[8] and can edit all the simple sections originating from the

[7] The same folder where the Input and Output files from the design are stored.
[8] Requires the *Strengthening* license to be activated.

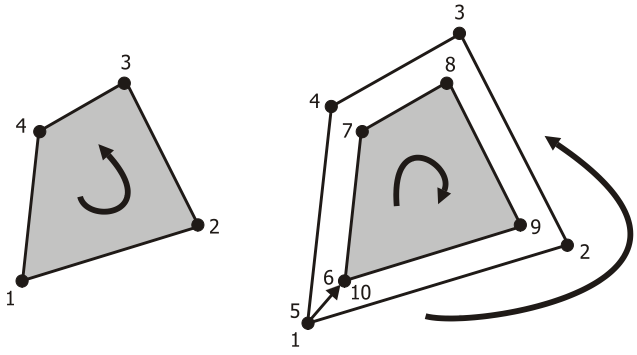
section design (rectangular, T-, L-, U- and circular sections) as well as the RC and masonry wall sections (.wal and .mas) that were created in the relevant editor (see section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**). Moreover, it includes the computational algorithm for check and design of arbitrary sections in biaxial bending with axial force. It collaborates interactively with the design application (described in section **Σφάλμα! Το αρχείο προέλευσης της αναφοράς δεν βρέθηκε.**) and formulates a reliability auditing tool for the program results.

The program includes 6 tabs (to the left of the window). The top two are identical and concern the geometric description of the core and the jacket8 of the section. The corner point co-ordinates for the core and the jacket and the centreline co-ordinates for the reinforcing bars are entered at the left side of the window. Data manipulation is accomplished by using the following buttons :

- Adds a new point (corner point for the section or centre point for the reinforcing bar) using the co-ordinates that are input in the corresponding fields below.
- Edits the selected point (corner point for the section or centre point for the reinforcing bar) in the table by pasting the co-ordinates input in the corresponding fields below.
- Deletes the selected point (corner point for the section or centre point for the reinforcing bar) from the table.
- Delete all points (corner point for the section or centre point for the reinforcing bar) from the table. If the core or jacket points are deleted, then the respective reinforcement bars are also deleted.



Note : The geometric description of the section core must be in an anti-clockwise manner, whilst that of the jacket in an anti-clockwise manner for the external face and a clockwise manner for the internal face (in contact with the core).



The following additional options are activated when Nous is run via the design procedure during editing of rectangular and T-sections :

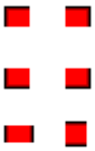
- Reinforce core as beam (top-bottom) with Ø12 bars.
- Reinforce core as column (perimeter reinforcement) with Ø12 bars.

In the above options, the value of the field *Reinforcement Cover (mm)* is taken into consideration.

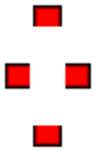
The third tab of the program includes the automatic jacket[9] creation for an arbitrary, rectangular or T- core section. The jacket reinforcing bars are distributed around the perimeter and to the centre of the jacket thickness with an initial bar diameter Ø12, which can be altered later. Data manipulation is accomplished by using the following buttons :

Perimetric jacket. Can be applied to all cases.

One-sided jacket. Can only be applied to rectangular sections when Nous is run via the design process.



Two-sided jacket. Can only be applied to rectangular sections when Nous is run via the design process.



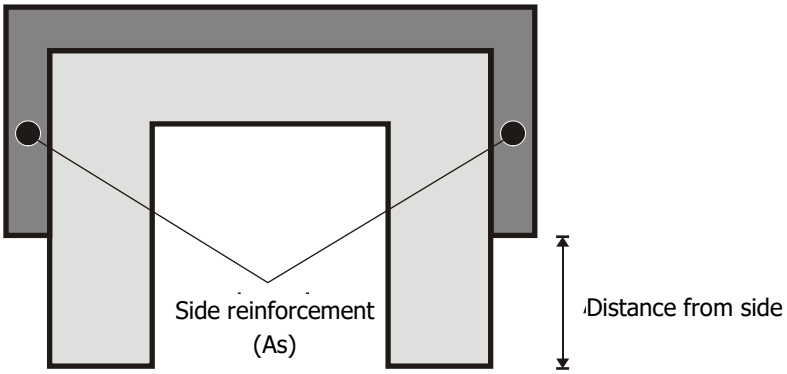
Three-sided jacket. Can only be applied to rectangular sections when Nous is run via the design process.

T-section jacket. Can only be applied to T-sections when Nous is run through the design process.

Jacket thickness : Defined in m in the corresponding field.

[9] Requires the *Strengthening* license to be activated.

Additional options for three-sided jackets (only when Nous is run through the design procedure for rectangular sections) :



In the fourth tab, the fields for the material properties for the core and the jacket are displayed in the following form :

$f_c = \frac{\alpha \cdot f_{ck}}{\gamma_c}$ e.g. for C20/25 concrete : $f_c = \frac{0.85 \cdot 20}{1.5}$ (MPa)

$f_y = \frac{f_{yk}}{\gamma_s}$ e.g. for B500C steel : $f_y = \frac{500}{1.15}$ (MPa)

The fifth tab includes the areas and the corresponding diameters for the core and jacket reinforcement, as these have been defined in the first two tabs. Data manipulation is accomplished by using the following buttons :

- Edits the area (in cm^2) or the diameter (in mm) of the selected reinforcement in the table that has been defined in the corresponding field. When the area is altered in this field, then the corresponding diameter is calculated automatically and vice-versa.
- Redistributes the areas/diameters of all the reinforcement according to the percentage (% times the concrete area of the section) that is input in the

adjacent field, whilst simultaneously it equates all the areas/diameters for all the reinforcement.

- Redistributes the areas/diameters of all the reinforcement according to the percentage (% times the concrete area of the section) that is input in the adjacent field, whilst the current ratio of areas/diameters is retained among the reinforcement.

The total reinforcement areas for the core and the jacket is always displayed under the respective reinforcement tables.

In the sixth tab the applied forces to the section are input, more specifically, the axial force N (in KN, negative sign for compression) and the moments M2 and M3 (in KNm) that correspond to the local axes 2 and 3 (displayed at the top left corner of the section shape). The correlation between the moments and axes is vectorial and follows the rule of thumb.

The various program functions are displayed on the toolbar. These are the following :

New section button : Begins input of a new section, deleting any previous section data. A warning message appears if the previous section has not been saved.

Open... button : Opens a prepared section (file with .sd or .wal or .mas extension) from the hard drive.

Save button : Saves on the hard drive the current section (file with .sd extension) under the name displayed in the relevant field (above the tabs).

Save as... button : Saves on the hard drive a previously saved section under a different name.

Clone sections button : The .sd files for cloning are initially chosen and then the user is prompted for the floor names for which the selected sections are to be cloned to. This function is particularly useful if it is required to provide identical jackets to identical sections at different floors of the model.

Background colour switch button : Switches the background colour in the graphics window from black to white and vice-versa, as well as the graphic colours of the section according to the user preferences.

Labels button : Displays the numbering of the section corner points and of the reinforcement positions.

Information button : Displays various information in the graphics window of the section :

- Number of iterations
- Number of reinforcement changes during design
- External forces (N, Mx, My)
- Internal forces (N_i, Mx_i, My_i)
- Neutral axis transitions for every iteration (dx, dy, dφ)
- Difference between external and internal forces (dN, dMx, dMy)
- Neutral axis location (x_p, y_p, φ_p) from ultimate limit state design
- Safety and capacity ratios from ultimate limit state design
- Safety and capacity ratios from linear elastic design (if there is no solution in ultimate limit state design, e.g. in cases of very high axial forces – small bending moments – unusual forms of cross-section)
- Centre of gravity for the section (x_o, y_o)
- Elastic neutral axis location (x_n, y_n)
- Core and jacket concrete area

- Core and jacket reinforcement areas and percentages

Stresses button : Displays the internal stresses of the section and the corresponding values at the corner points and reinforcement locations. Moreover, the outline of the concrete compression zones for parabolic and rectangular stress distribution are displayed.

Deformations button : Displays the internal section deformations and the corresponding values at the corner points and reinforcement locations.

Safety factor button : Calculates the section safety factor (SF) under ultimate limit state design (failure for SF < 1, adequacy for SF ≥ 1). The safety factor is displayed in the graphics window of the section (if the display information button has been selected) as well as in the sixth tab, under applied forces. Additionally, the section capacity ratio (λ) is displayed, which is the invert of the safety factor (failure for λ > 1, adequacy for λ ≤ 1).

Required reinforcement button : Designs the section, calculating by iterative process the required reinforcement so that the safety factor is equal to 1. Should a jacket be present, the design is with respect to the jacket's reinforcement.

Next iteration button : Displays the next iteration of the safety factor calculating process under ultimate limit state design.

Previous iteration button : Displays the previous iteration of the safety factor calculating process under ultimate limit state design.

Restart button : Cancels the safety factor calculation process and returns to the initial elastic state. If the design has been performed, the program returns to the initial reinforcement solution.

5. Pushover

5.1 Scia Engineer

ECtools uses the analysis model of Scia Engineer and the proposrtioning (reinforcement) results from ECtools to prepare a nonlinear analysis model for the execution of pushover analysis based on ATC 41-13 and FEMA 356.

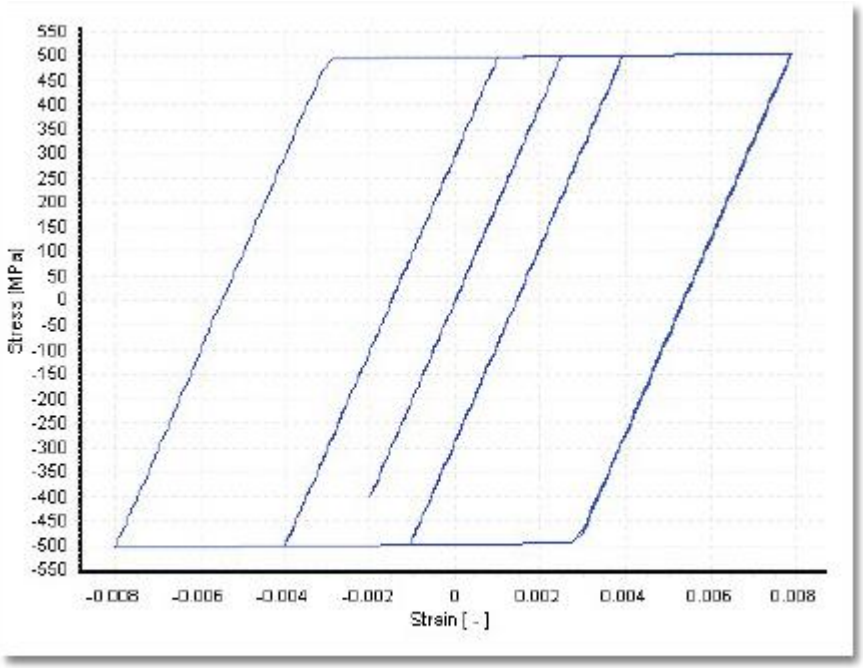
The nonlinear analysis is performed using a dedicated Seismostruct engine which is included in the nonlinear version of ECtools.

More information on seismostruct is available here while the verification is here

The nonlinear model is prepared using 1d nonlinear finite elements and rigid links to create the cores.

The material properties are:

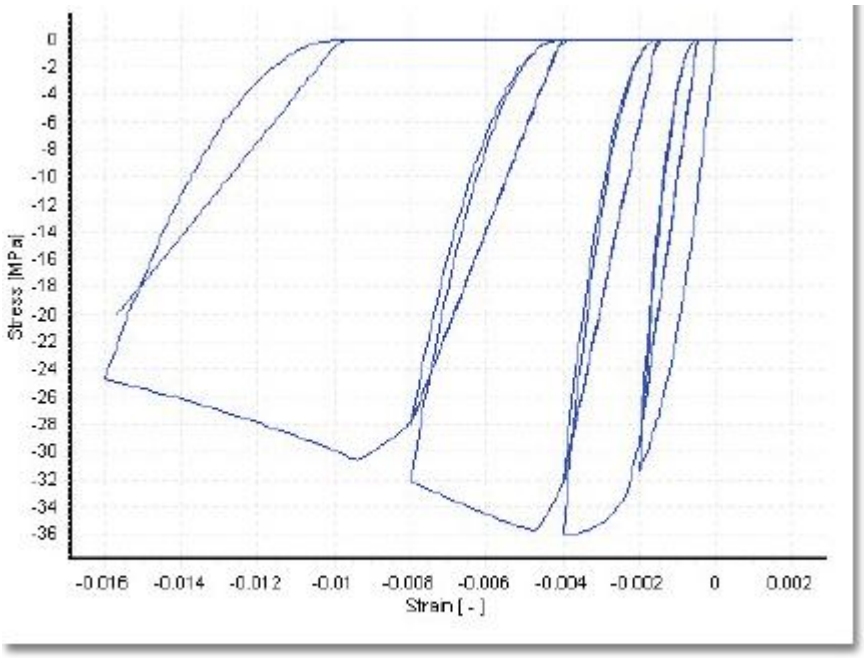
- Steel with uniaxial bilinear stress--strain model with kinematic strain hardening, whereby the elastic range remains constant throughout the various loading stages, and the kinematic hardening rule for the yield surface is assumed as a linear function of the increment of plastic strain.



Bilinear steel model

- Concrete Uniaxial nonlinear constant confinement model, initially programmed by Madas [1993], that follows the constitutive relationship

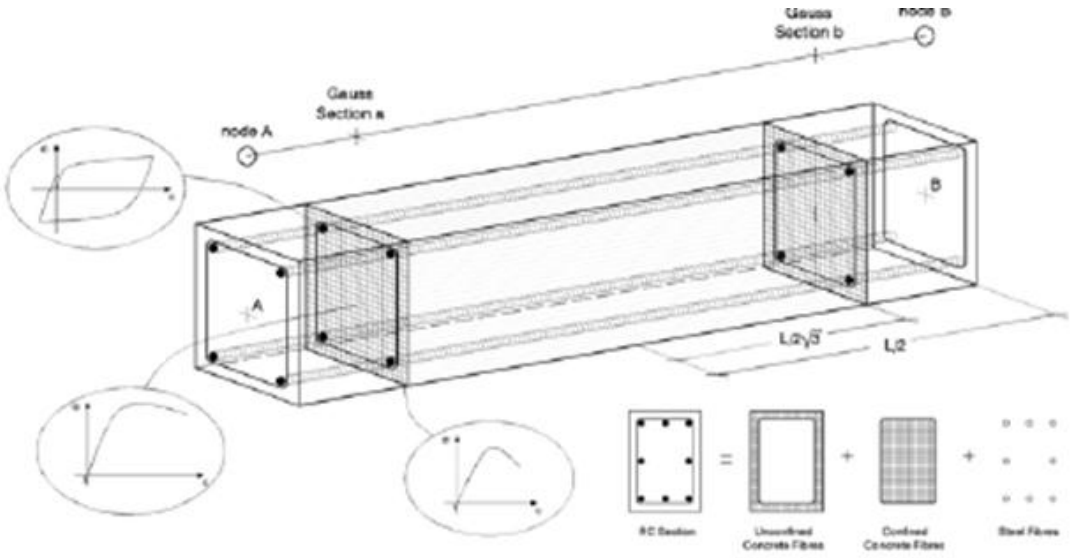
proposed by Mander et al. [1988] and the cylic rules proposed by Martinez--Rueda and Elnashai [1997].



Mander et al. nonlinear concrete model

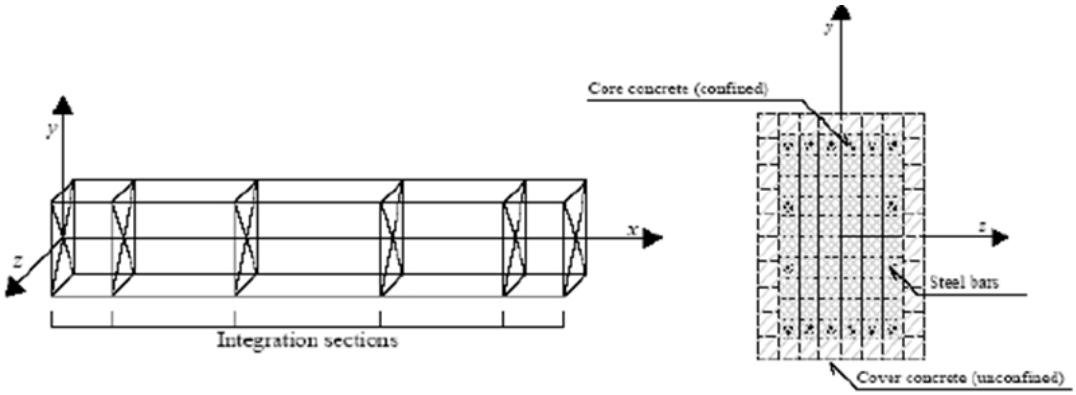
The analysis is based on a fibber element apporach whereas:

- each element is divided into sections



Discretisation of a typical reinforced concrete cross-section

- each section is divided into fibbers (subsections)



Gauss-Lobatto integration sections

- each fibre is associated with a uniaxial stress-strain relationship

The sectional stress-strain state of beam-column elements is then obtained through the integration of the nonlinear uniaxial stress-strain response of the individual fibres