ETABS MATE® User Manual

Document Edition: EN610

FARASA Engineering Group

FARASA Engineering Group

Document Number: EN2608216102

ETABS MATE
Concrete Structure Detailing Software

ETABS MATE

Concrete Structure Detailer Software

Software User Manual

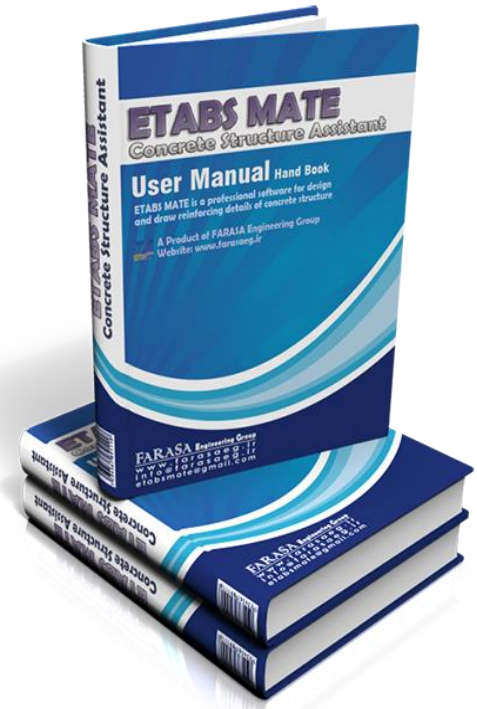
Document Edition: EN610

Document Language: English

Revision Date: June 2026

Publisher: FARASA Engineering Group

Document Number: EN2608216102



Copyright & Terms of Use

ETABS MATE
Concrete Structure Detailing Software

Table of Contents

ETABS MATE
Concrete Structure Detailing Software

About The Software

Structural design engineers consistently encounter numerous challenges throughout the detailing, rebar detailing, and drafting of construction drawings for concrete structures. The breadth of analytical outputs, the large number of elements, the diversity of design conditions, and the necessity for high precision render this process exceedingly time consuming and susceptible to various human errors. Furthermore, in many projects, the detailed design of elements must be carried out concurrently based on the outputs of multiple structural models, further compounding the difficulty and time required to complete this process.

Consequently, the concept of designing a detailing software capable of automatically calculating and then drafting rebar details for concrete buildings with the highest degree of accuracy, while maintaining the highest level of interaction with the design engineer, gradually took shape. The combination of this concept with many years of experience in the fields of structural design, the development of specialized structural software, and structural finite element programs led to the creation of a completely independent, professional, intelligent, and powerful software. This software features a highly flexible, practical, and user friendly interface for the rebar detailing, detailing, and drafting of construction drawings for concrete structures, the result of which is now before you.

ETABS MATE, in a thoroughly professional and independent manner, and without any restrictions on the number of elements, the number of stories, or the project area, performs the process of detail calculation and construction drawing drafting in a fraction of a second with very high accuracy and speed. In addition to displaying the calculated details within its graphical and editable environment, it delivers high quality construction drawings in the standard drawing format of **AutoCAD** software.

The professional and unique capabilities of **ETABS MATE** on one hand, and its attractive, user friendly graphical environment with extensive user interactivity on the other, have led to the software's widespread adoption by both veteran engineers in major consulting firms and young design engineers across numerous countries worldwide. To date, this software has been successfully employed in the design and detailing process of thousands of small to medium scale projects and hundreds of large to very large scale and complex projects exceeding one hundred thousand square meters in area.

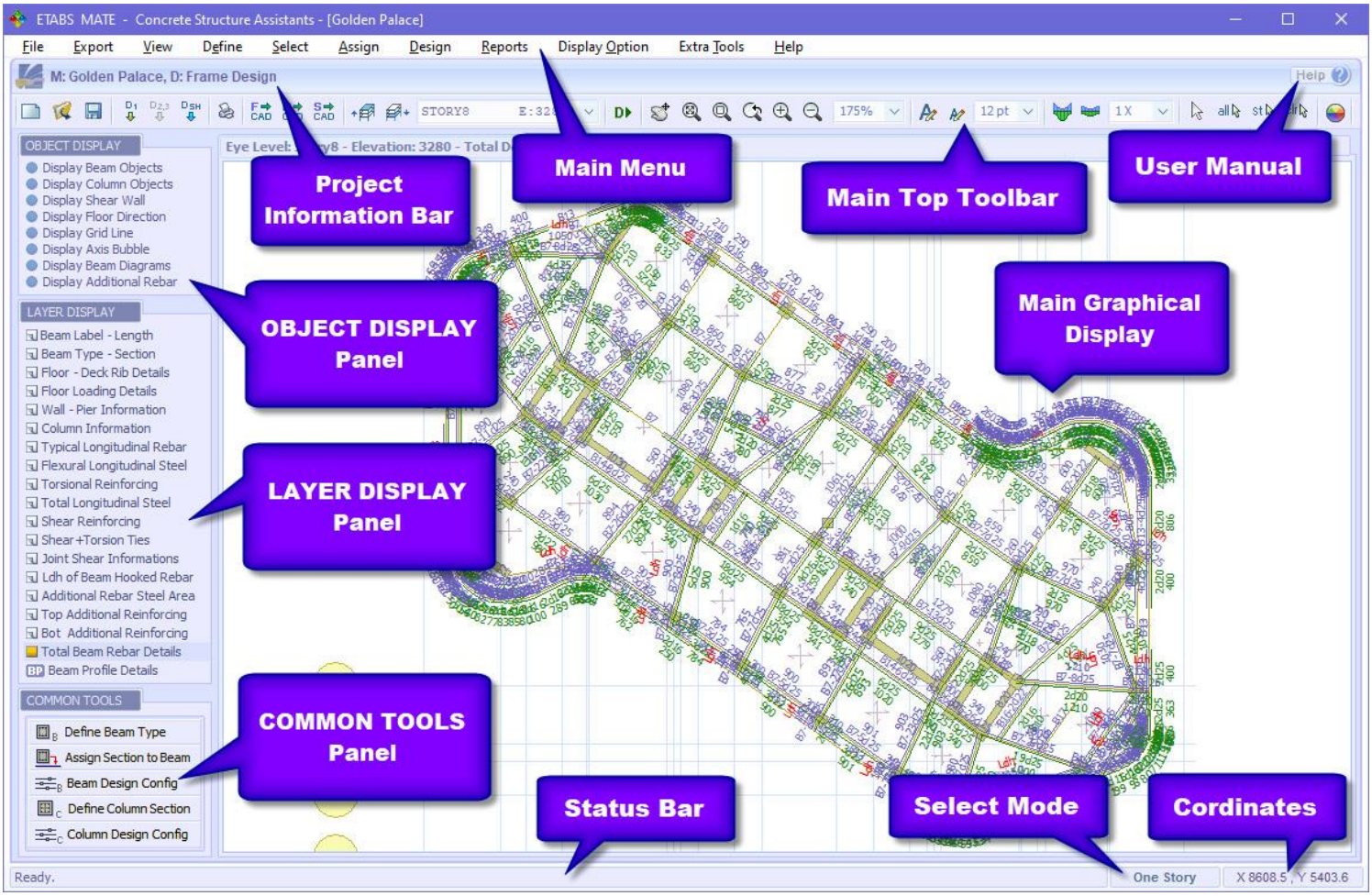
The primary goals and approach of the designers and developers of **ETABS MATE** are to ensure simplicity, convenience, extreme accuracy and speed, coupled with the highest possible level of user interactivity. In this regard, the software's user interface was designed so that designers familiar with common engineering software such as **ETABS** and **AutoCAD** can easily relate to the tools, menus, and graphical user interface. Therefore, learning the software is very straightforward, and users will achieve the necessary proficiency within a short period, enabling them to complete the detailed design and detailing of a large concrete structure in just a few minutes, thereby saving many hours and days for more important tasks aligned with their professional activities.

ETABS MATE
Concrete Structure Detailing Software

Software Interface Overview

The user interface of **ETABS MATE** has been designed in a highly simple, intuitive, and user friendly manner, enabling all engineers familiar with conventional engineering software such as ETABS and AutoCAD to easily interact with the tools and graphical environment of the software. The main components of the software are identified in the image below.

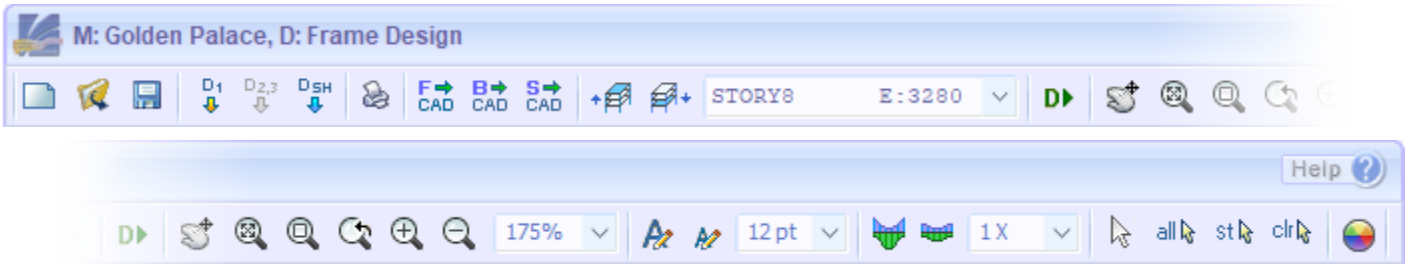
The software's user interface is designed to provide users with easy access to all parts of the software. To this end, in addition to the main menu and the toolbar at the top of the software, a set of highly important and practical panels has been embedded on the left side of the main graphical environment. Using the tools available in each of these palettes, you can control the display of various structural elements, or readily view element information, various design details such as bending moment, shear, and torsion values and diagrams, along with many other design parameters, as well as the detailing particulars of structural elements required throughout your workflow, directly within the graphical environment. Furthermore, should the need arise, you can edit the detailing specifications of various structural elements, which will be elaborated upon in subsequent sections.



ETABS MATE
Concrete Structure Detailing Software

Main Toolbar of the Software

Certain commands that are used continuously and with high frequency throughout the use of the software, along with several essential tools, have been placed in the main toolbar at the top of the software. This allows users to manage their project and execute the detailing process with greater ease and speed.



The tools available in this toolbar are explained in detail below:

Icon	Tool Description	Shortcut
	Create a new project	Ctrl + N
	Open a project saved in **ETABS MATE** format with the EMC file extension	Ctrl + O
	Save the project in the proprietary **ETABS MATE** file format (*.EMC)	Ctrl + S
	Import the ETABS model text file in e2k format into the software	Ctrl + T
	Import the first frame design data file (allowable formats: RTF, XML, MDB, ACCDB)	Ctrl + D
	Import the second, third, ... frame design data file into the software	
	Import the first shear wall design data file (allowable formats: TXT, XML, MDB, ACCDB)	Ctrl + H
	Send the structural details drawn in the graphical view to the printer	Ctrl + P
	Export frame drawings, including columns and beam details in plan, to AutoCAD	F3
	Export beam profiles structural drawings to AutoCAD	Ctrl + F3
	Export shear wall detail structural drawings to AutoCAD	F4

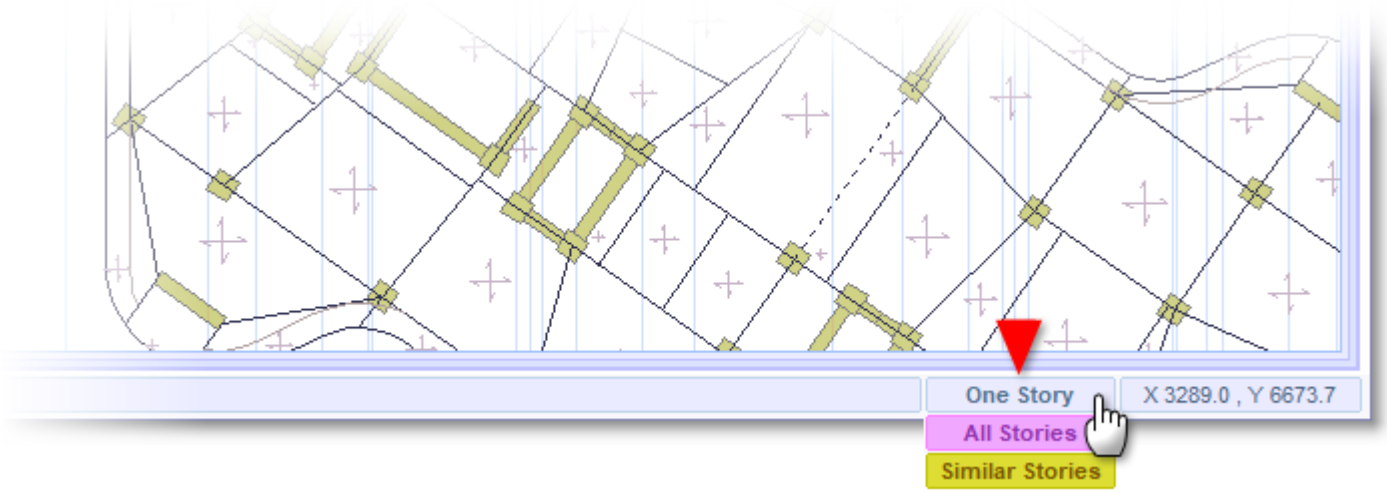
ETABS MATE
Concrete Structure Detailing Software

Icon	Tool Description	Shortcut
	Move to the next story or one level higher in the structural elevation levels	Ctrl + U
	Move to the previous story or one level lower in the structural elevation levels	Ctrl + J
STORY1 E:385	Move to a desired elevation level or story among elevation levels	
	Design all structural details	F2
	Pan the structure using the mouse within the graphical view	SPACE
	Zoom to fit the entire structure within the graphical view	Ctrl + E
	Zoom to a specific region by drawing a rectangle over the desired area	Ctrl + W
	Restore the previous zoom state or previous view position of the structure	Ctrl + B
	Zoom in incrementally on the structure within the graphical view	Ctrl + Q
	Zoom out incrementally on the structure within the graphical view	Ctrl + A
100 %	Manually specify the zoom magnification of the structure	
	Increase the font size of text annotations within the graphical view	Ctrl + I
	Decrease the font size of text annotations within the graphical view	Ctrl + K
12 pt	Manually specify the font size for element detail text annotations	
	Increase the magnification factor for beam diagrams within the graphical view	
	Decrease the magnification factor for beam diagrams within the graphical view	
1 X	Manually specify the magnification factor for beam diagrams	
	Configure colors in various sections of the software or select the software themes	
	Single element selection tool for selecting structural elements using the mouse	
all	Select all beam elements of the structure across all stories	
st	Select all beam elements of the structure on the active story	
clr	Deselect all currently selected elements	

To facilitate easier element selection, a set of selection methods has also been implemented in the software, which will be described in the following sections.

Structural Element Selection Methods

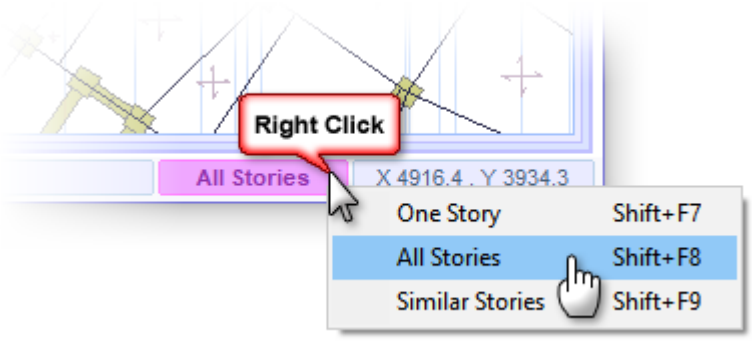
To facilitate a more convenient and efficient structural element selection process, provisions have been made allowing the user to employ one of the selection methods, including One Story, All Stories, and Similar Stories, based on their specific requirements. As illustrated in the image below, the most straightforward way to switch between selection methods is by using the multi state button One Story located in the lower right corner of the main software window.



With each click on this multi-state button, the selection method will toggle sequentially and cyclically through the available selection modes. Each of these modes is described below:

One Story : In this mode, selecting a beam selects only that beam on the current story.
All Stories : In this mode, selecting a beam selects all similar beams across all stories.
Similar Stories : In this mode, selecting any beam will result in the selection of all similar beams across all stories belonging to a story group. It is worth noting that the definition of similar story groups is explained in the Project Story Editing section.

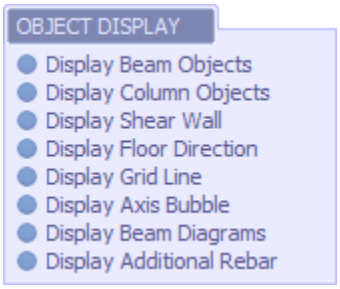
Furthermore, as shown in the adjacent image, the selection mode can also be changed as desired by right clicking on the multi state button, using the shortcut keys **Shift+F7**, **Shift+F8**, and **Shift+F9**, or via the Select menu and the Select Mode option. Once the desired mode is set, the intended elements can then be selected.



ETABS MATE
Concrete Structure Detailing Software

OBJECT DISPLAY Panel

The **OBJECT DISPLAY** panel is designed to control the display of structural elements and detailing information within the graphical view. The items available in this panel are described in the Below:



Display Beam Objects

When enabled, beam elements of the structure are visible in the graphical view.

Display Column Objects

When enabled, column elements of the structure are visible in the graphical view.

Display Shear Wall

When enabled, shear wall elements of the structure are visible in the graphical view.

Display Floor Direction

When enabled, the direction indicator of the floor joists is displayed on the plan.

Display Grid Line

When enabled, the grid lines defined in the ETABS software are displayed.

Display Axis Bubble

When enabled, the label of each grid line is displayed as a bubble.

Display Beam Diagram

When enabled, in layers that display flexural, torsional, or shear reinforcement quantities, the corresponding diagram is also displayed.

Display Additional Rebar

When enabled, the calculated additional reinforcement on beams is displayed.

ETABS MATE
Concrete Structure Detailing Software

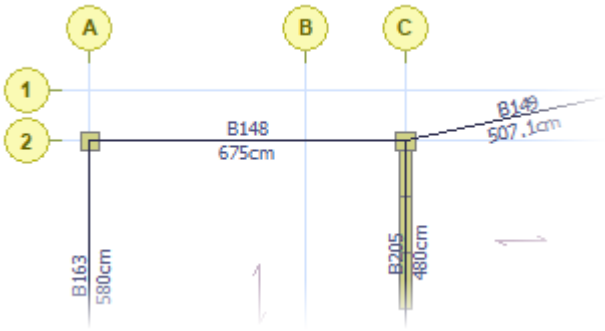
LAYER DISPLAY Panel

The **LAYER DISPLAY** panel is of particular importance in the software's usability and is designed to manage the display of element information, structural design details such as moment, shear, and torsion values and diagrams, and many other parameters, as well as to edit the detailing details of structural elements within the software's graphical environment.

As mentioned, this panel is one of the most important and key parts of the software, through which you will have access to much of the structural information of your project. In addition to viewing these details, in many cases you will also be able to edit the details designed by the software, allowing you to check or make necessary edits to the details before producing the structural drawings. The following sections describe some of the items available in this panel.

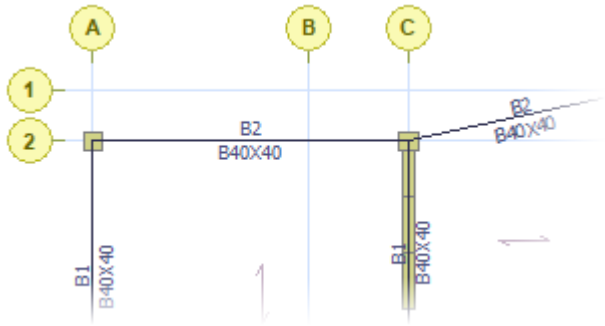
Beam Label – Length

In this layer, the geometric information of the structural beams is displayed on them. This information includes the name assigned to the beam elements in ETABS at above the beam, as well as the length of each beam at below the beam, which is inserted on each beam in the software's graphical environment.



Beam Type – Section

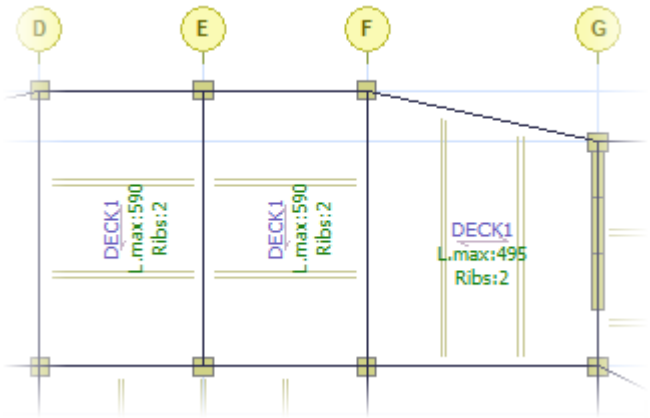
In this layer, the modeling information of the beams is displayed on them. This information includes the beam type assigned to each beam at the top and the beam section name at the bottom of each beam, which appears in the software's graphical environment. Additionally, right-clicking on any beam brings up the user interface for viewing information and editing details.



ETABS MATE
Concrete Structure Detailing Software

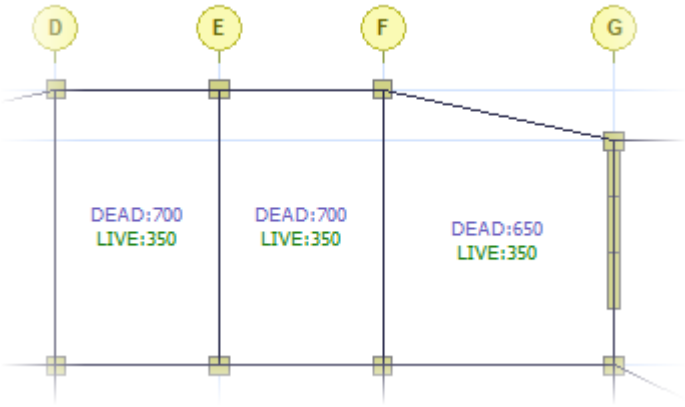
Floor – Deck Rib Details

In this layer, the details of ribs in joist-block slabs are drawn. In addition, the slab section name, the longest joist length, and the number of slab ribs are displayed on the structural plan within the software's graphical environment. These settings can be edited and configured via the **Define** menu and the **Deck Ribs Drawings Parameters** option.



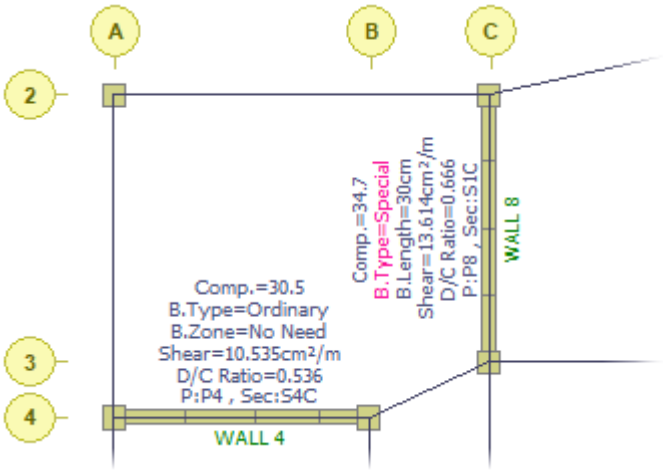
Floor Loading Details

In this layer, all loads applied to the floors are displayed in user friend software's graphical environment. Checking the applied loads using this layer, as it shows all applied loads together, can be very effective in preventing load application errors. It is worth noting that floor loads are only read and displayed in the software if the loads have been entered individually in the ETABS software and a Load Pattern has not been used.



Wall - Pier Information

In this layer, useful and practical information about shear walls is displayed in the software's graphical environment. This information includes: the pier name, the section name, the demand-to-capacity ratio, the required shear reinforcement area, the boundary element length, the boundary element type, and the maximum compressive stress along the wall section. Additionally, in this layer, clicking on any wall brings up the user interface for viewing and editing the details of the selected shear wall for all stories.

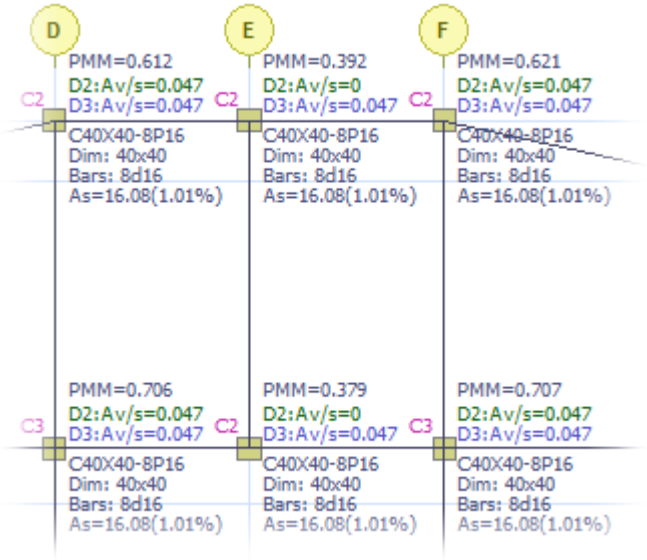


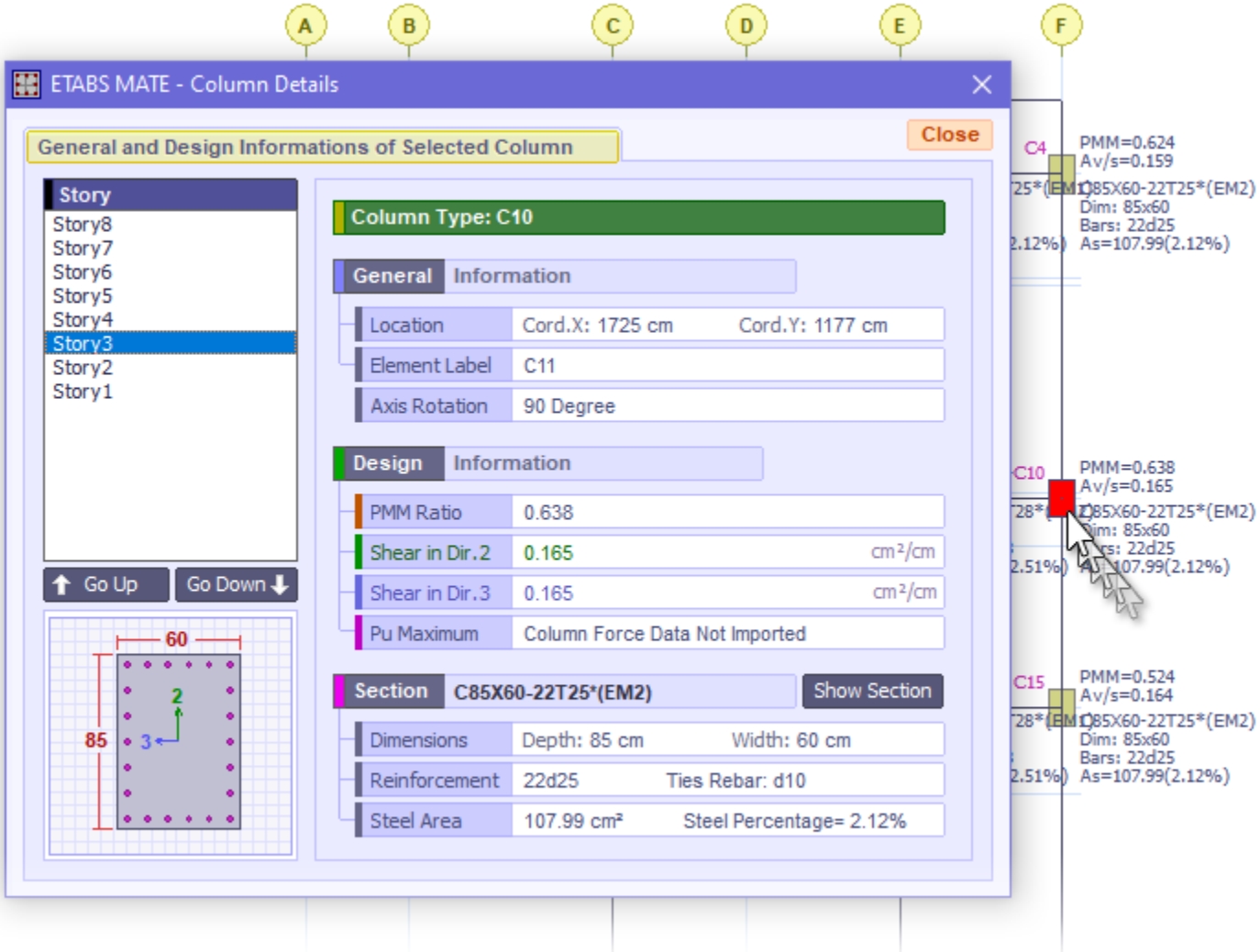
ETABS MATE
Concrete Structure Detailing Software

Column Information

In this layer, useful and practical information about columns is displayed in the software's graphical environment. This information includes: the column type, the demand-to-capacity ratio (PMM), the required shear reinforcement area in the 2 and 3 directions of the section, the section name assigned to the column, the dimensions, the rebars defined in the section, the existing steel area in the section, and the steel percentage of the section.

By clicking on any column brings up the column information user interface, as shown in the image below.

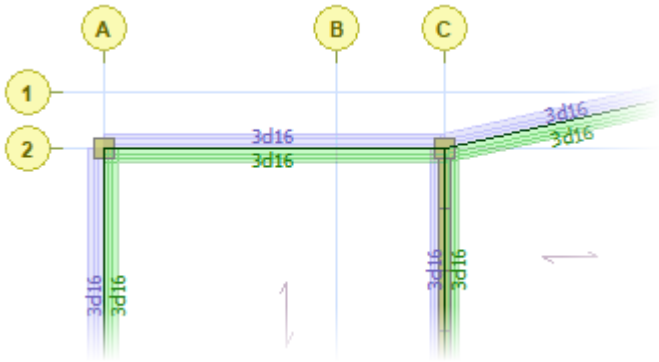




As shown in the image above, **the modeling details, design information including the PMM Ratio, the required transverse reinforcement of the section in the 2 and 3 directions, the maximum compressive force applied to the column, as well as the section properties assigned to the column**, are displayed. Furthermore, using the story list or the navigation keys, you can also view the details of columns located directly above or below the selected column, thereby easily observing and verifying the vertical details of each Column Type. Additionally, by using the button of **Show Section**, you can view the complete section information, including the longitudinal and transverse reinforcement details.

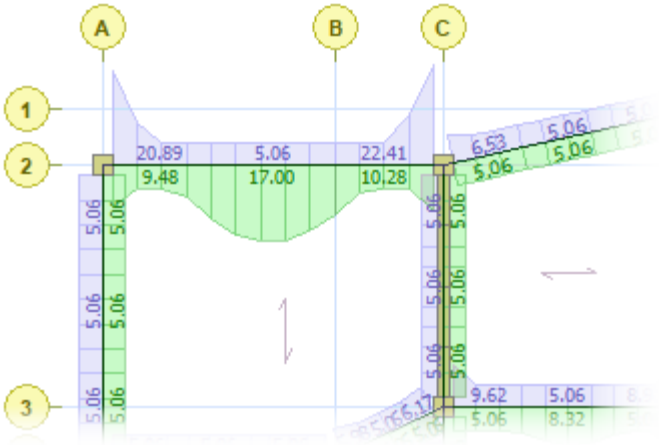
Typical Longitudinal Rebar

Selecting this item displays the details of the continuous rebars assigned to the beams, either by the software or by the user, on all beams in the plan view within the graphical view. As shown in the image, these details include the number and diameter of the continuous rebars at the top and bottom of the beams.
It is worth noting that the details of the continuous rebars assigned to the structural beams can be viewed and edited by the user via the **Define** menu and the **Define Beam Type – Section** option.



Flexural Longitudinal Steel

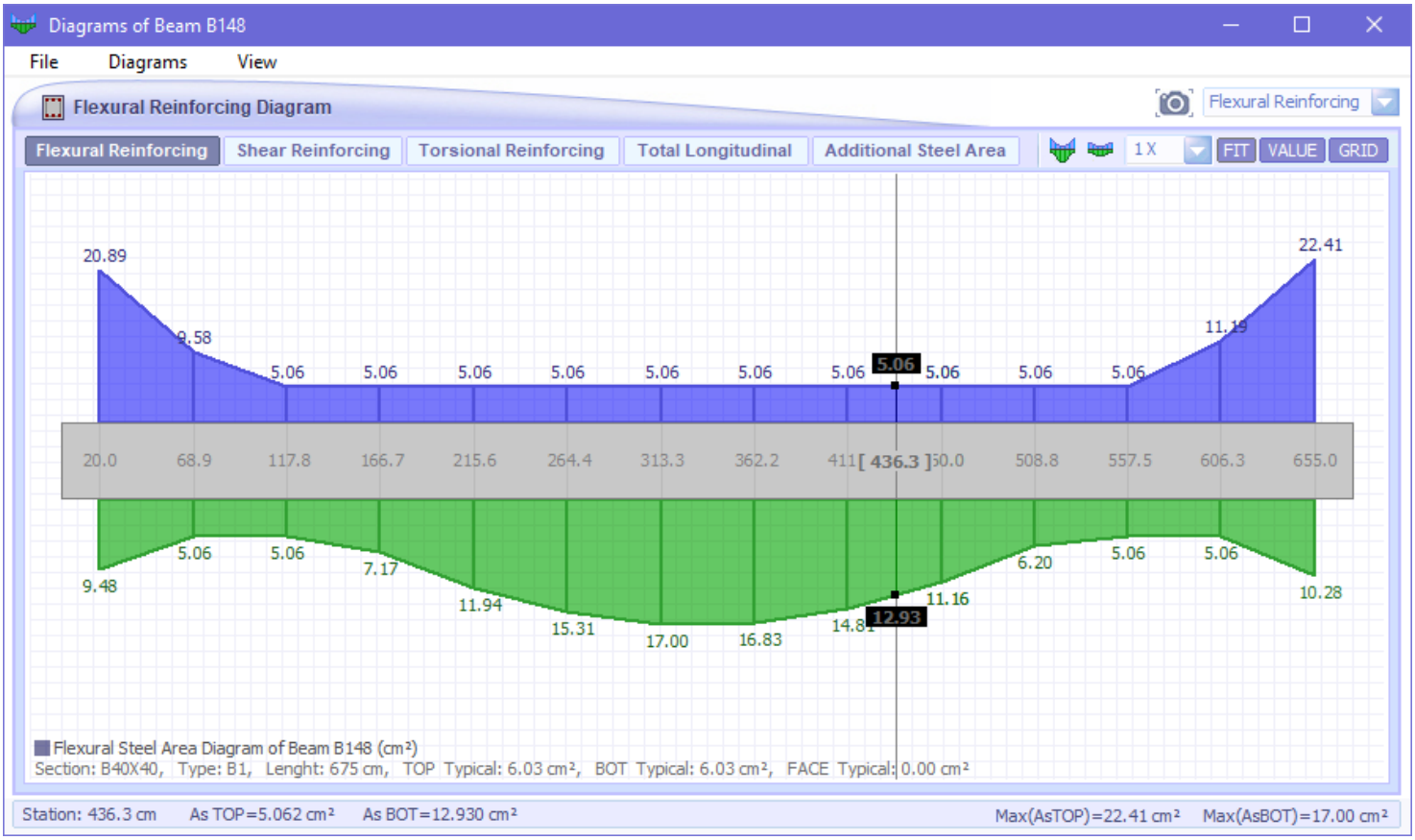
By this item, the flexural steel area designed by ETABS for the beams is displayed in the software's graphical environment. These details include the flexural steel area enveloped from all imported design files for the start, middle, and end of the beam at the top and bottom positions, as well as the flexural steel area diagram required for the top and bottom of the beam.



Note that in this layer, right-clicking on any beam opens a user interface for the detailed view of the beam's flexural reinforcement diagram, which will be described in the following sections.

User Interface for Displaying Beam Steel Diagrams

In the layers displaying design details, right-clicking on any beam will graphically display the user interface for the detailed view of the steel diagrams for that beam. As shown in the image below, by moving the mouse over the beam, all information related to the desired position is easily accessible graphically.



This user interface incorporates useful tools for controlling the reinforcement diagrams, which are described below:

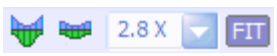


Diagram scale control tools

Option to show or hide station values on the diagram

Option to show or hide grid lines beneath the diagram

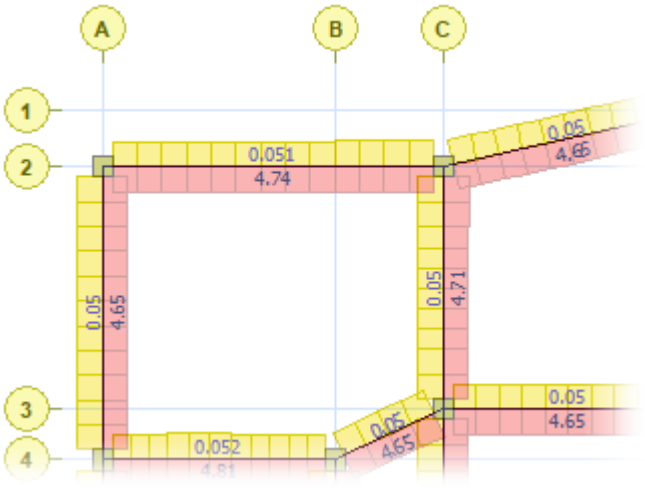

Tool to save the drawn diagram as an image file

Furthermore, numerous other options and tools for controlling the display of beam diagrams have been designed, which are accessible from the top menu of this interface.

ETABS MATE
Concrete Structure Detailing Software

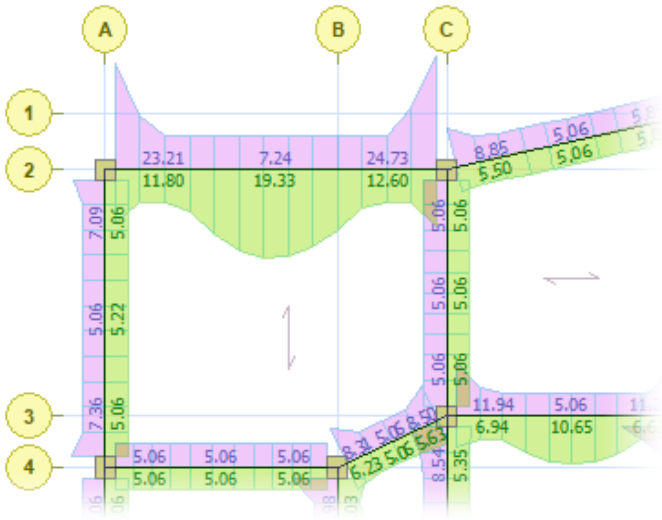
Torsional Reinforcing

In this layer, the details of the transverse and the longitudinal steel Area due to torsion calculated by ETABS, as well as the corresponding diagrams, are displayed on the structural beams in the software's graphical display. These details are, respectively: the amount of transverse torsional steel (At/s) at the top of the beam, and the required longitudinal torsional steel area, which is inserted at the bottom of the beams in plan of structure.



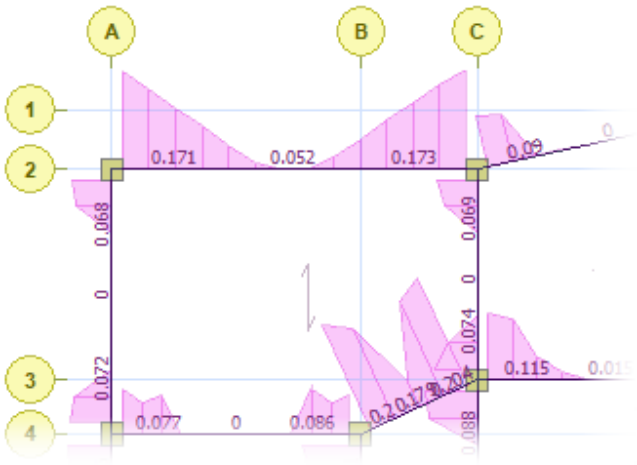
Total Longitudinal Steel

In this layer, the details of the total longitudinal steel area due to flexure and torsion calculated by ETABS, as well as the corresponding diagrams, are displayed on the structural beams in the software's graphical environment. It is worth noting that in this layer, right-clicking on any beam brings up the user interface for the detailed view of the total longitudinal steel diagram of the beam.



Shear Reinforcing

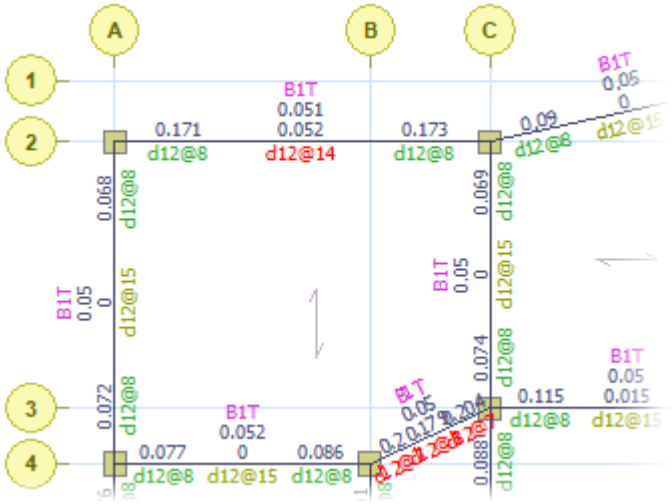
In this layer, the details of the shear steel calculated by ETABS, as well as the corresponding diagrams, are displayed on the structural beams in the software's graphical environment. These details include the amount of transverse shear steel (Av/s) for the start, middle, and end positions of the beam, which is inserted at the top of the beam.



ETABS MATE
Concrete Structure Detailing Software

Shear + Torsion Ties

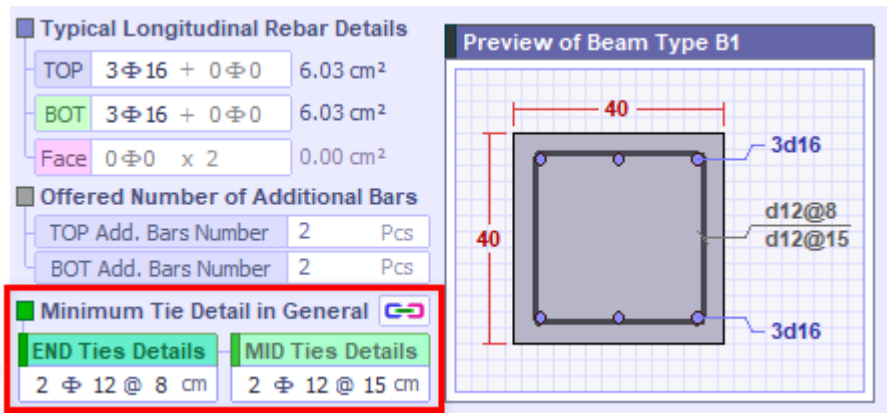
This item displays the details of transverse reinforcement, i.e., stirrups, on beams within the graphical view. These details, listed sequentially from the top row, are as follows: the beam type, the transverse reinforcement due to torsion (At/s), the transverse reinforcement due to shear (Av/s) at the top of the beam, and finally, the stirrup details assigned to the beams at the start, middle, and end locations, shown at the bottom of the beams.



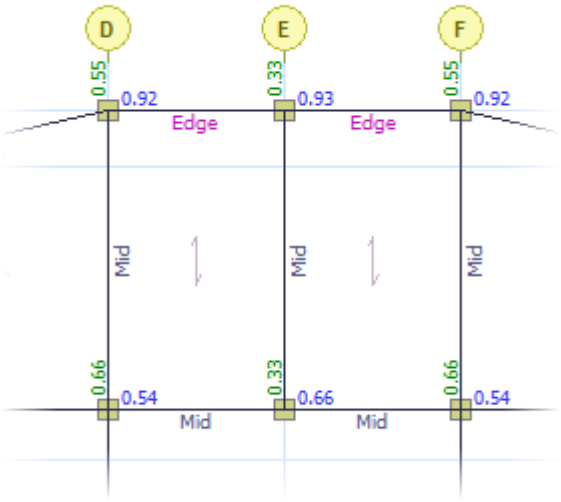
As observed in the image, some transverse reinforcement details are displayed by **red**. This indicates that the minimum transverse reinforcement quantity considered by the program, based on the relevant code or as specified by the user for the beam section, is less than the quantity required by the ETABS analysis. Consequently, **ETABS MATE** has automatically calculated the required transverse reinforcement that satisfies the transverse reinforcement quantity computed by ETABS software.

Therefore, the **red** color does not signify that the transverse reinforcement is insufficient. Rather, it simply indicates that more transverse reinforcement was required than the minimum code-specified reinforcement initially considered for the beam section. The software has therefore been forced to calculate the required transverse reinforcement based on the transverse steel required at that station, and display it in **red**. Conversely, the **green** color indicates that the minimum transverse reinforcement considered for the beam section is sufficient.

The minimum transverse reinforcement quantity considered for each beam section can be viewed and edited via the **Define** menu and the **Define Beam Type – Section Details** option.



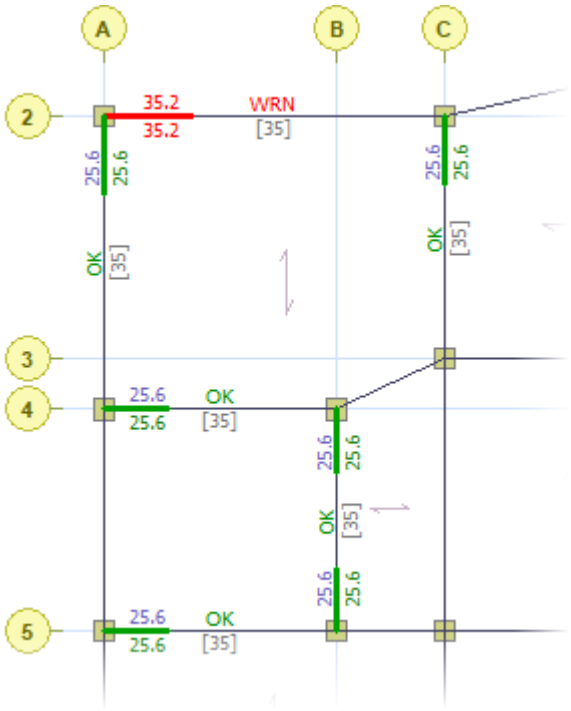
Joint Shear Information



Selecting this item displays the joint shear ratio calculated by the software in the graphical view for both column directions on all columns, as shown in the image. The accurate calculation of the joint shear ratio requires precise knowledge of the actual reinforcement details, the positioning of beams relative to the column section, whether the beams connected to the column are cantilevered, and many other parameters.

Given that some of these parameters are not available in the ETABS software, the joint shear ratio calculated by ETABS is, in many cases, inaccurate. Since the beam reinforcement calculated in **ETABS MATE** represents the actual as-built reinforcement, and considering that many calculation parameters are configurable, the capability to calculate the joint shear ratio based on ACI 318 has been incorporated into the software. The software automatically computes these critical parameters without any limitations regarding the angles of beams connected to the joint.

Ldh of Beam Hooked Rebar

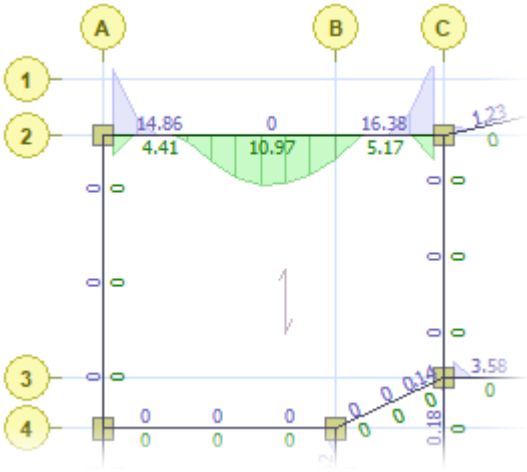


Selecting this item displays the calculated values of the required development length for hooked bars. As shown in the image, the required development length for the top hooked end bars is displayed at the top of the beam, and the required development length for the bottom hooked end bars is displayed at the bottom of the beam. Furthermore, the development length provided by the column dimension in the beam direction is displayed in the middle of the beam, enclosed within [ ] brackets in gray.

If the calculated hooked development length is less than the length provided by the column dimension in the beam direction, the calculated required development length is displayed in blue for the top bars and in green for the bottom bars of the beam, indicating that the development length is adequately provided. Otherwise, the calculated required development length is displayed in red, signifying that the required development length is not satisfied, and a WRN warning is also displayed in the middle portion of the beam. It is worth noting that by right-clicking on any beam, you can view the detailed calculations of the development length for the hooked end bars.

Additional Rebar Steel Area

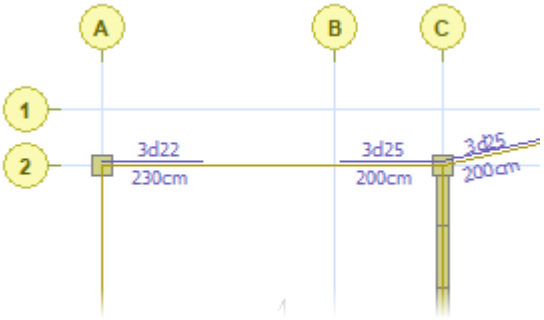
Selecting this item displays the required additional reinforcement values for each station, along with their corresponding diagrams, on the structural beams within the graphical view. The area of additional reinforcement is obtained by subtracting the continuous reinforcement area from the flexural reinforcement diagram and then distributing the longitudinal torsional reinforcement, after deducting the area of the face bars defined in the section, between the top and bottom reinforcement using one of the torsional reinforcement distribution methods selected by the user.



TOP Additional Reinforcing

Selecting this item displays the details of the top additional reinforcement of the beams, including the number, diameter, and length of the additional rebars, on the beams within the graphical view.

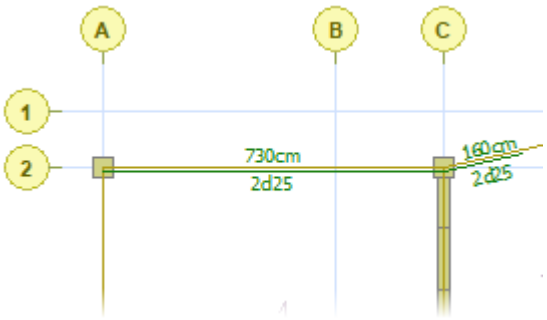
It is worth noting that in this layer, right-clicking on any beam opens a user interface for viewing information and editing the rebar detailing of the selected beam.



BOT Additional Reinforcing

Selecting this item displays the details of the bottom additional reinforcement of the beams, including the number, diameter, and length of the additional rebars, on the beams within the graphical view.

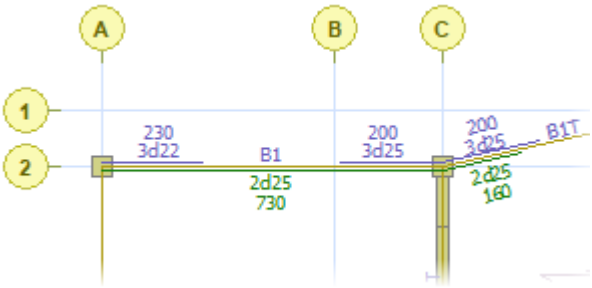
It is worth noting that in this layer, right-clicking on any beam opens a user interface for viewing information and editing the rebar detailing of the selected beam.



ETABS MATE
Concrete Structure Detailing Software

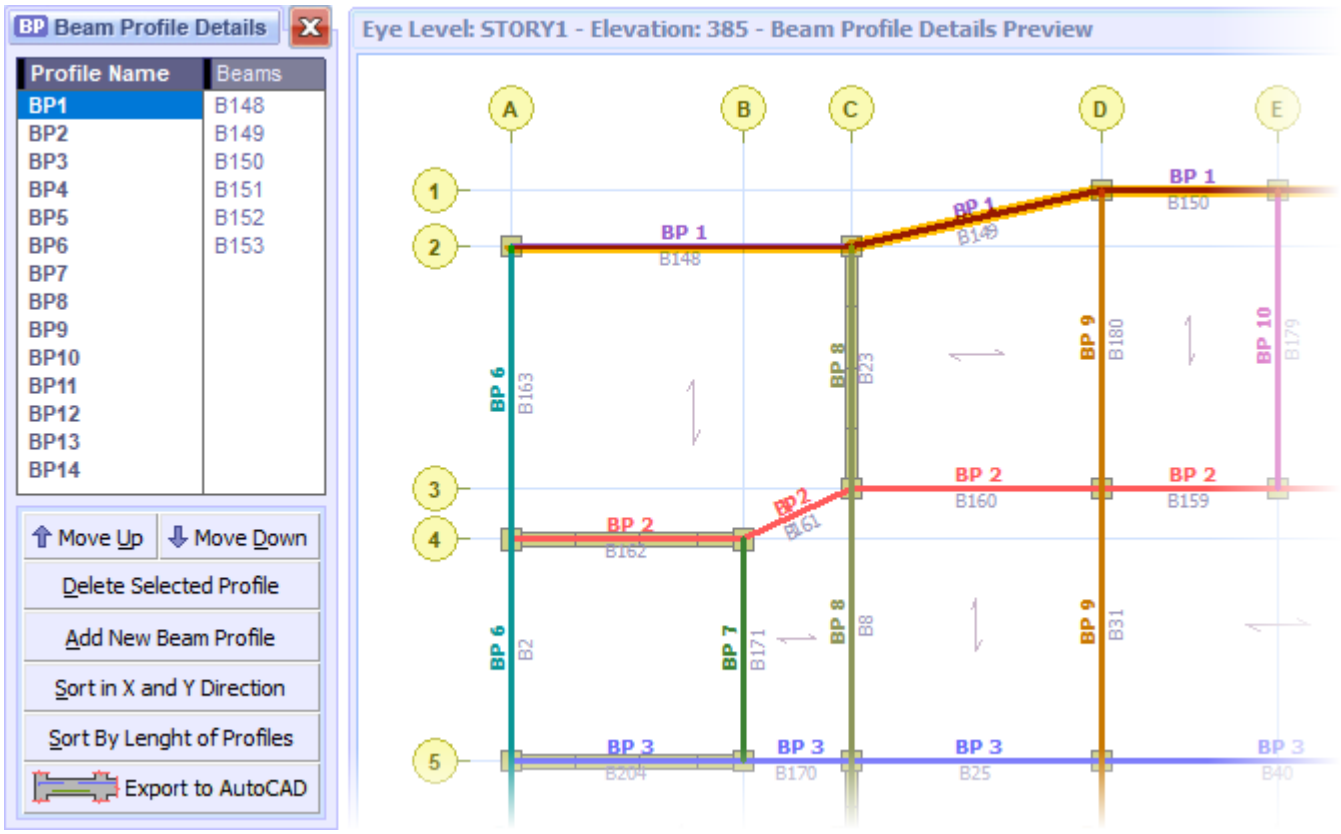
Total Beam Rebar Details

In this layer, the details of the top and bottom reinforcement bars of the beams, including the number, diameter, and length of the reinforcement bars, are displayed on the beams in the software's graphical environment. In this layer, right-clicking on any beam brings up the user interface for viewing information and editing the reinforcement details of the selected beam.



Beam Profile Details

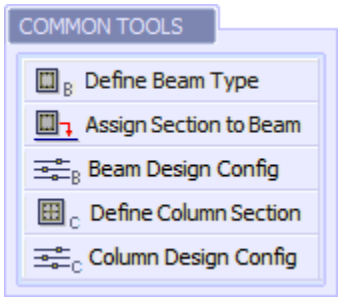
In this layer, the beam longitudinal profiles that automatically detected by the software are displayed in the graphical environment. At the bove the beams, the name of the Beam Profile containing the beam is inserted, and below the beam, the label of the beams within each profile are inserted. Using this layer, you can fully manage the beam profiles.



Clicking on any profile in the list or within the graphical view highlights the corresponding profile in the graphical view. The **Beam Profile Details** section includes additional tools such as the **Move Up** and **Move Down** buttons for editing the order of profiles, **Delete Selected Profile** for removing a profile, **Add New Beam Profile** for adding a new longitudinal profile, **Sort in X and Y Direction** and **Sort By Length of Profiles** for sorting the profiles, and the **Export to AutoCAD** button for generating the longitudinal profile structural drawings of the beams.

COMMON TOOLS Panel

This panel incorporates several frequently used tools, which are continuously accessed by users, into the side palette of the software for greater convenience. These tools are described below:



Define Beam Type

This tool opens a user interface for viewing and editing the details of beam sections. Through this interface, you can edit details such as the number and diameter of continuous rebars, the minimum transverse reinforcement details of the section including the diameter, spacing, and number of crossties, the section name, and many other beam section details, or define new sections.

Assign Section to Beam

This tool allows you to assign a newly defined section to the selected beam or beams.

Beam Design Config

This tool opens a user interface for configuring the detailed design of structural beams. Through this interface, you can view, edit, and customize numerous parameters governing the detailed design of beams.

Define Column Section

This tool opens a user interface for viewing and editing the details of column sections. Through this interface, you can view or edit the minimum transverse reinforcement details of the section, including the diameter, spacing, and number of crossties.

Column Design Config

This tool opens a user interface for configuring the detailed design of structural columns. Through this interface, you can view, edit, and customize numerous parameters governing the detailed design of columns.

ETABS MATE
Concrete Structure Detailing Software

Quick Start in Four Simple Steps

ETABS MATE is a highly flexible and user-friendly software solution, designed to provide a smooth learning experience and efficient workflow for structural engineers. To use the software for reinforcement detailing and construction drawing generation, the structure must first be modeled, analyzed, and designed in ETABS. Fortunately, importing the structural model into **ETABS MATE** does not require any special element naming conventions in ETABS, allowing engineers to model their projects entirely according to their standard workflow.

Once the structural modeling and design process in ETABS is completed and all design results are satisfactory, the reinforcement detailing and drawing production process can be easily continued within **ETABS MATE**. The entire detailing and drawing generation workflow is completed in only four simple steps:

1 Exporting the Structure from ETABS

At this stage, the structural geometry and design data are exported from ETABS.

2 Importing the Structure into ETABS MATE

In this step, the model and structural design result files are imported into **ETABS MATE**.

3 Reinforcement Detailing of Structural Elements

By executing the detailing command, reinforcement details for structural elements are generated automatically and with exceptional speed.

4 Generation of Structural drawings

After completing the detailing process and performing the required reviews and adjustments, the construction drawing generation command can be executed.

The following provides a brief overview of these four simple steps. However, it is strongly recommended to carefully study all the material presented in this guide to ensure the correct use of the software and to become fully acquainted with all its advanced capabilities and features, enabling you to utilize them professionally. Furthermore, a wealth of supplementary material is available in the Articles section of the website, which is highly beneficial for more advanced and professional use of the software.

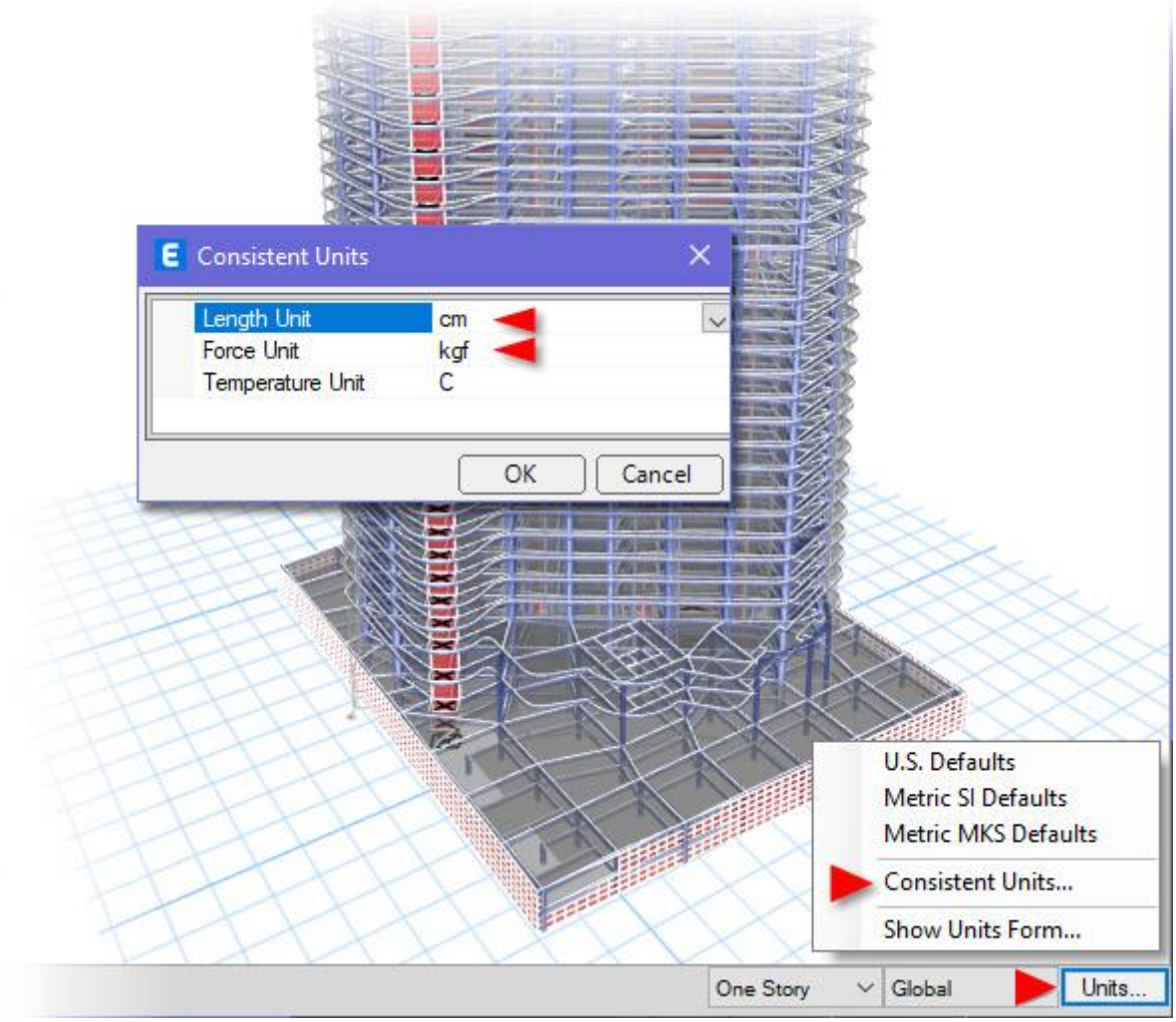
ETABS MATE
Concrete Structure Detailing Software

1 Exporting the Structure from ETABS

To export the text file and the design results files from the ETABS software, the program's unit system must first be set to **Kgf** and **cm** to ensure that the calculated values from ETABS are transferred correctly and with sufficient accuracy.

Setting the Unit System in ETABS Software for File Export

To set the unit system in ETABS software, first, as shown in the image below, click on the **Units** button located at the lower right corner of the ETABS program. Then, from the menu that appears, select the **Consistent Units** option to open the unit selection window. In this window, select **cm** for the Length Unit and **kgf** for the Force Unit. Next, from the **File** menu, select the **Export** option to save the desired files.

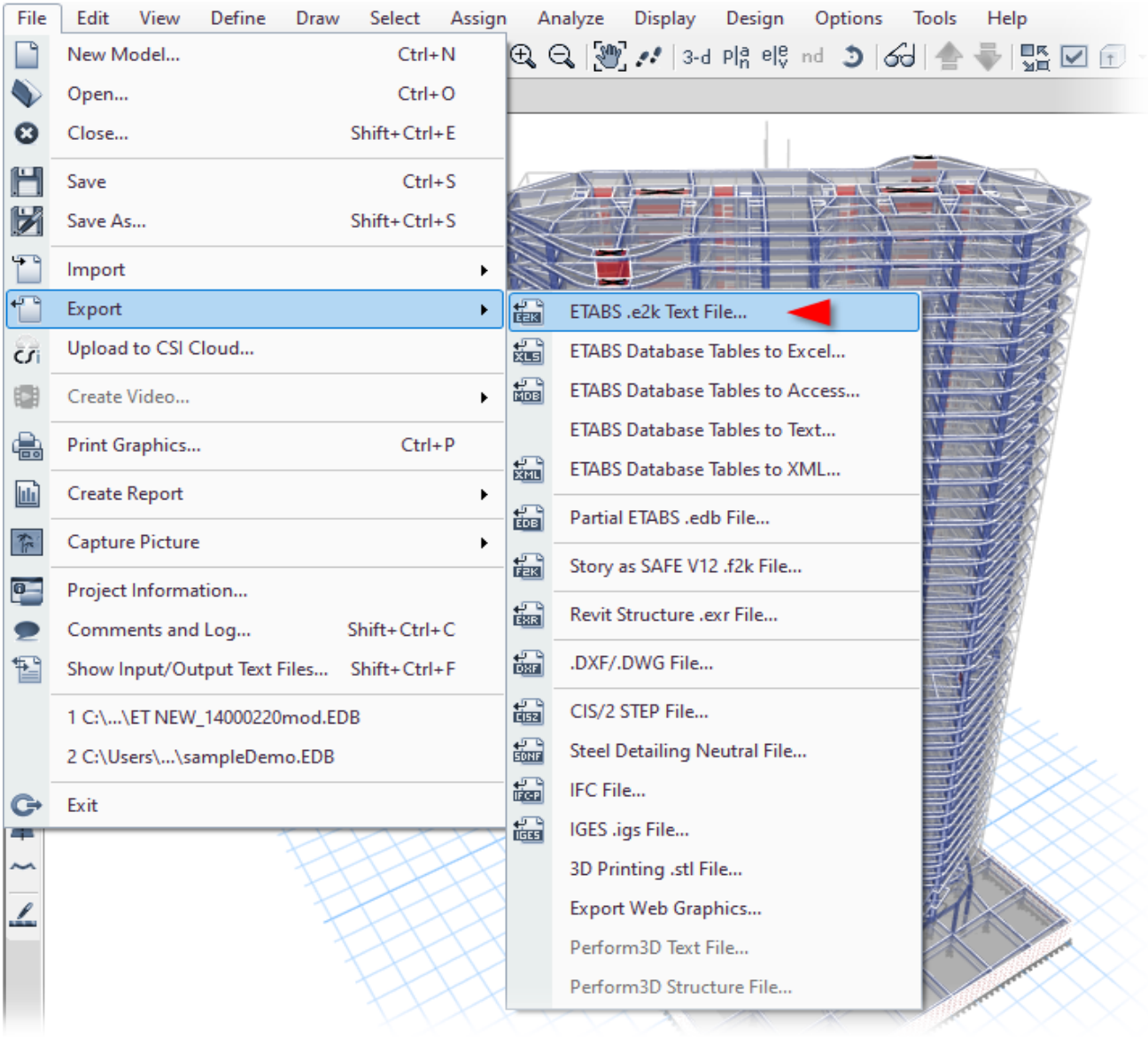


ETABS MATE
Concrete Structure Detailing Software

Exporting the Structural Geometry File

First, set the unit system of the ETABS software to **Kgf, cm**. Then, as shown in the image below, follow the steps corresponding to your ETABS version to save the model text file:

File Menu > Export > ETABS .e2k Text File...



Note 1: Before generating the structural model text file, ensure that the unit system of the ETABS software is set to **Kgf, cm**.

Note 2: If you encounter any issues or error messages while exporting the e2k file in the ETABS software, unlock the model and then export the file. Subsequently, re-analyze and redesign the model before proceeding to the next steps.

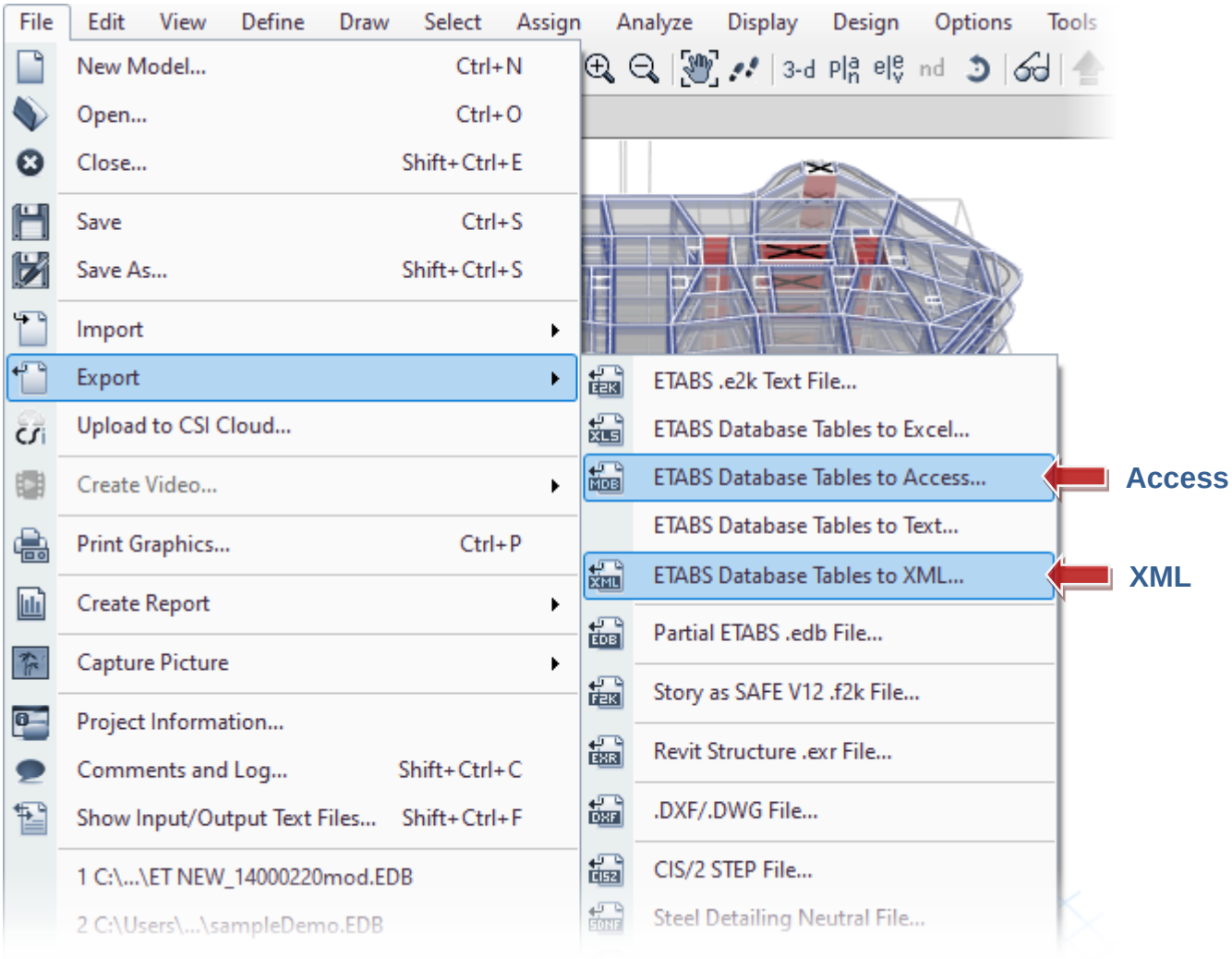
Exporting the Structural Design Results File

First, set the ETABS unit system to **Kgf, cm**, and then use the **Access** or **XML** outputs as follows:

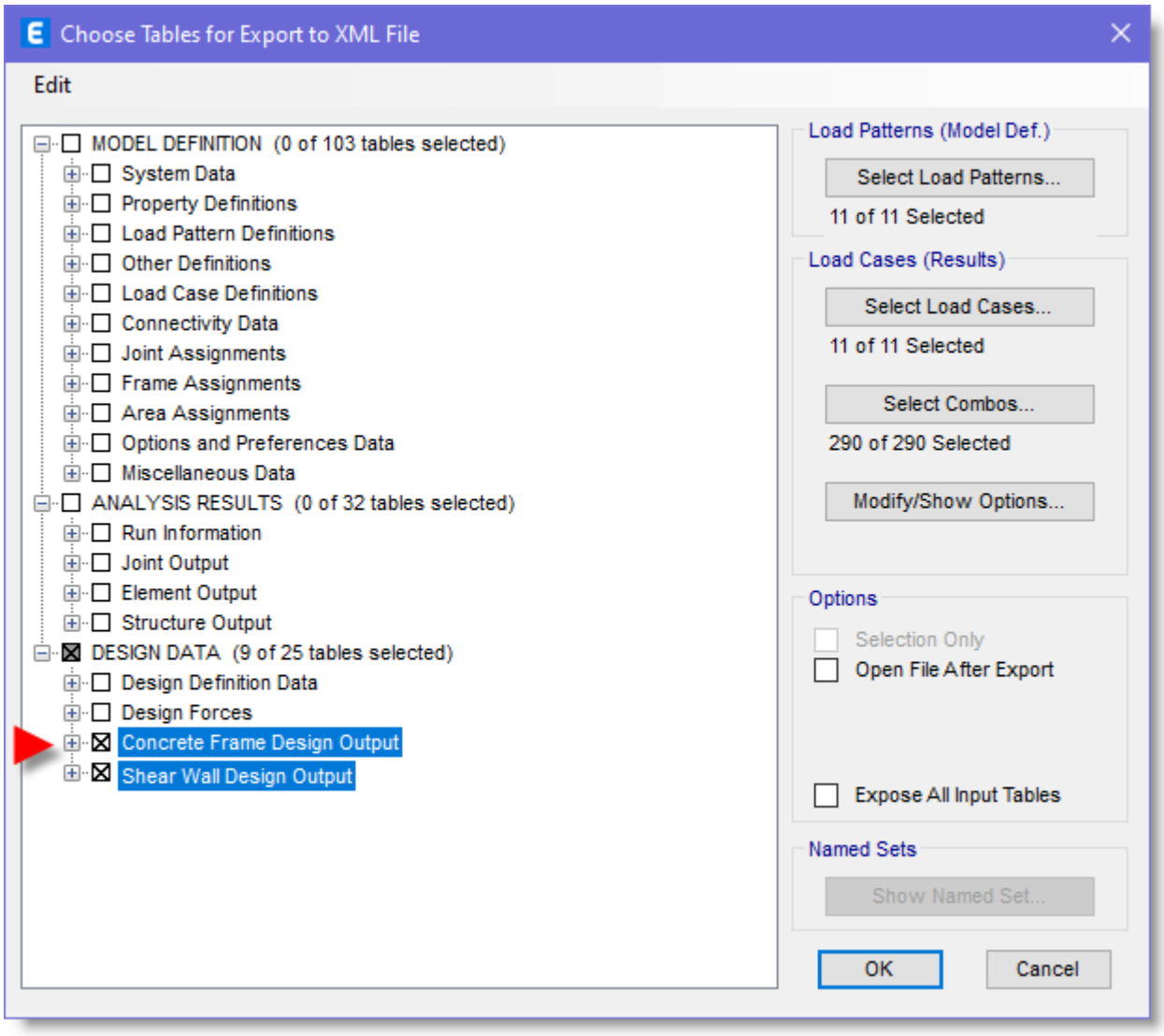
File > Export > ETABS Database Table to Access …

Or

File > Export > ETABS Database Table to XML …



After selecting the output format for the structural design data, the **Choose Tables** user interface will appear, as shown in the image below. In this interface, select the **DESIGN DATA** item under **Concrete Frame Design Output** to save the design results for the frame, i.e., the beams and columns of the structure, and likewise, select the **Shear Wall Design Output** sub-item to save the design results for the shear walls. Deselect all other options. Then, by specifying a name and save location for the output file, the file will be generated by the software.



Of course, you can also save only the frame design results or only the shear wall design results as a single Access or XML file. To do this, repeat the steps above; however, to save only the frame design results, select solely the **Concrete Frame Design Output** option in the Choose Tables interface. Similarly, to save only the shear wall design results, select solely the **Shear Wall Design Output** option, and deselect all other options. Alternatively, you may save both the frame and shear wall design results in a single file.

Note 1: Prior to performing the above step and saving the design results file, the structure must have been analyzed and designed; otherwise, the design tables shown in the image above will not be available.

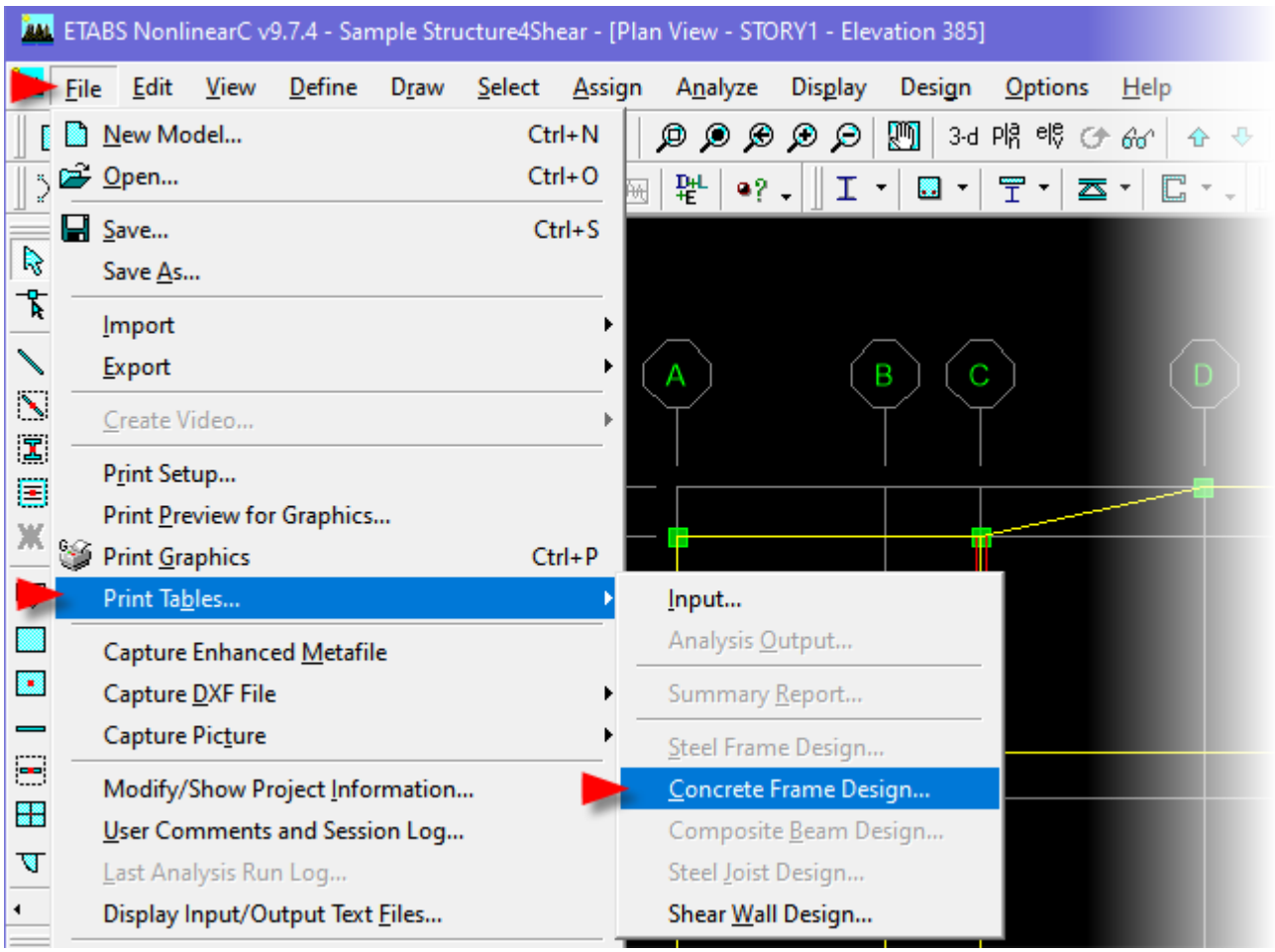
Note 2: Before saving the design results file from the File menu, ensure that the unit system of the ETABS software is set to **Kgf, cm**. Furthermore, when ETABS prompts you for the output unit system during the table export process, set the units to **Kgf, cm** once again.

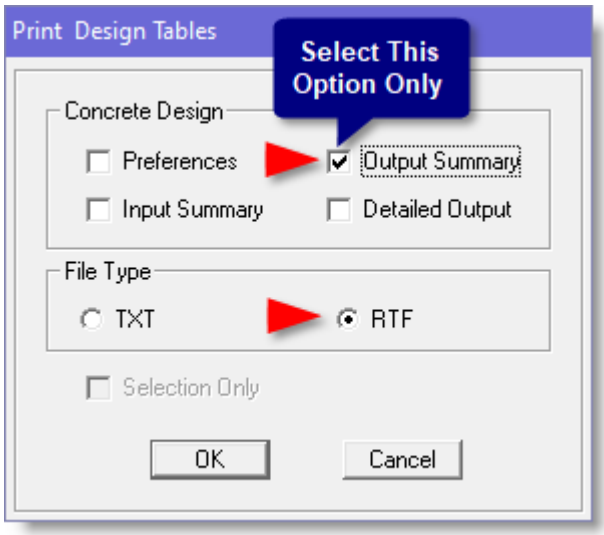
ETABS MATE
Concrete Structure Detailing Software

Exporting the Structural Design Results File from ETABS 9

First, set the unit system of ETABS to **Kgf, cm**. Then, to save the frame design results file, you may use the **RTF**, **TXT**, and **Access** formats as follows:

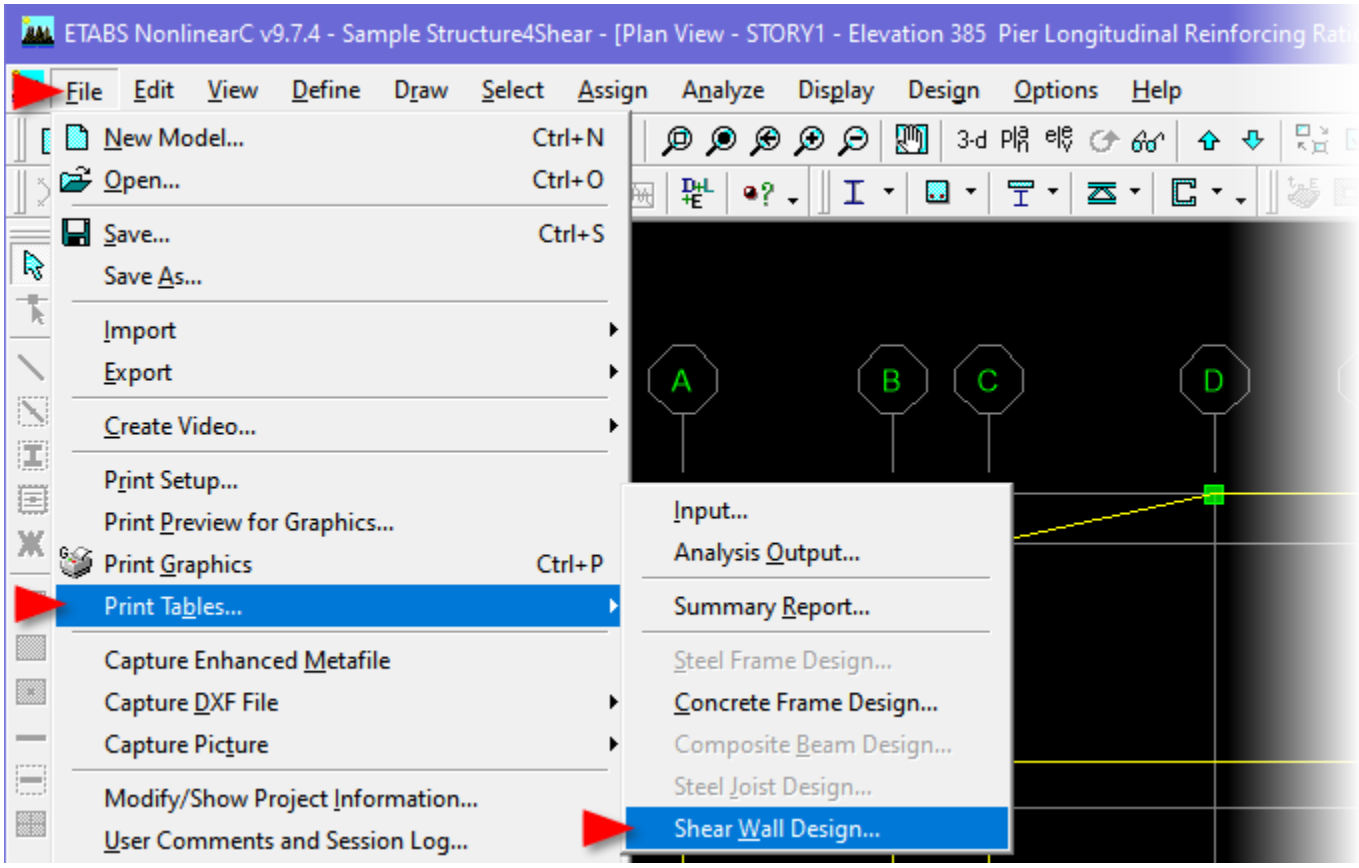
File Menu > Print Tables... > Concrete Frame Design... > ☑ Output Summery

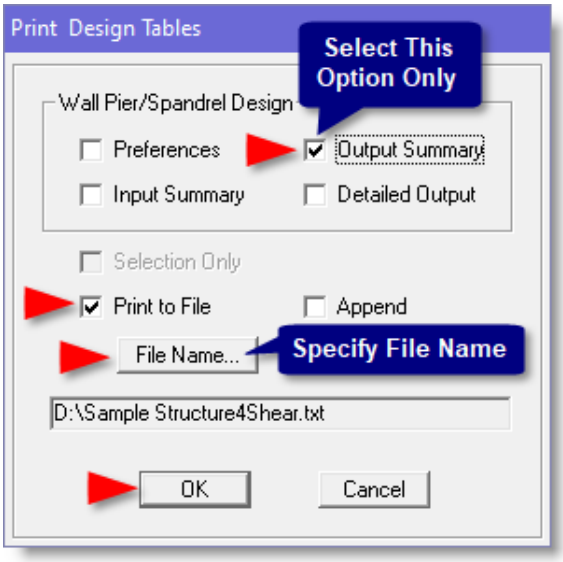




Similarly, in **ETABS 9**, to save the shear wall design data file in **TXT** format, proceed as follows:

File Menu > Print Tables... > Shear Wall Design...





Note: Using the **Access** output in **ETABS 9** to generate the structural design files is possible as:

File Menu > Export > Save Input/Output as Access Database File ...

ETABS MATE
Concrete Structure Detailing Software

2 Importing the Structure into ETABS MATE

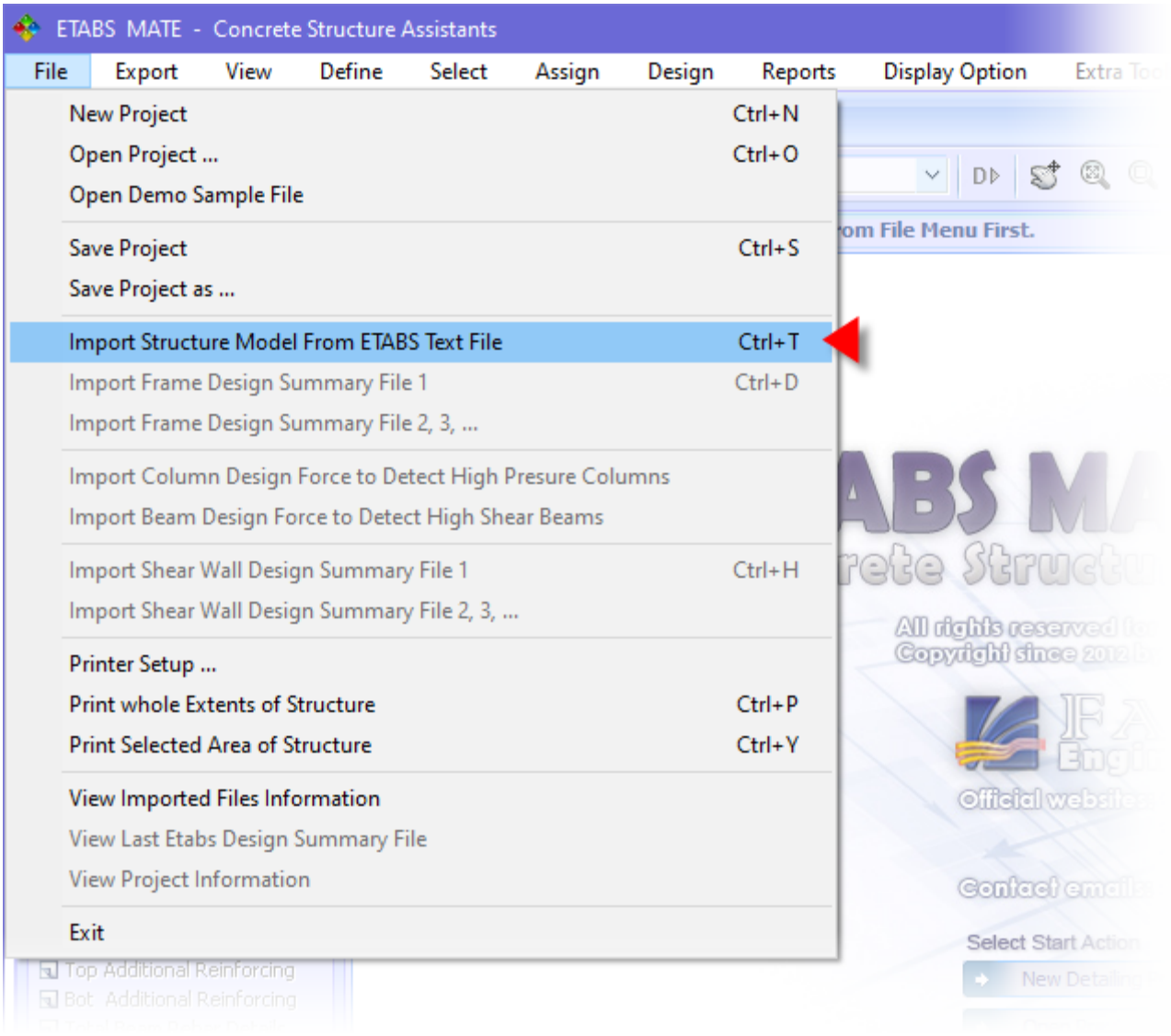
In this step, the text file and the design files generated in the previous step must be imported into the software as follows.

Importing the Structural Model Text File into the Software

First, import the structural model text file, i.e., the **e2k** file, into the software as follows:

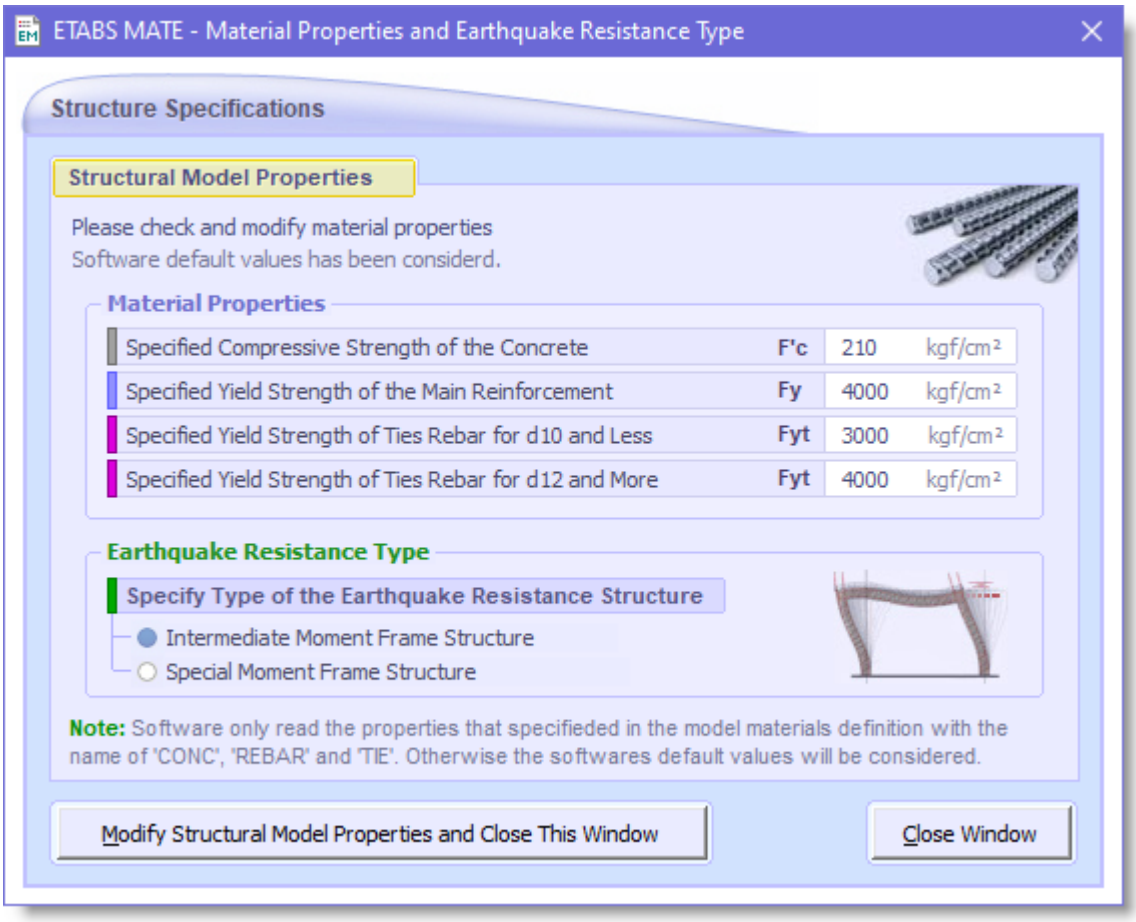
File Menu > Import Structure Model From ETABS Text File (Ctrl + T)

You may also use the icon in the main toolbar at the top of the software to import the model text **e2k** file. Upon importing the model text file, the structure will be visible within the main graphical display of the software.



Configuring Material Properties and Earthquake Resistance Type

After importing the model text file, the user interface for configuring the project material properties and specifying the earthquake resistance type will appear, as shown in the image below. Through this window, you must specify the **Earthquake Resistance Type** and verify the project material properties, which the software automatically extracts from the **CONC**, **REBAR**, and **TIE** materials defined in the structural model. If necessary, you may modify these properties. In this window, **F'c** represents the compressive strength of concrete, **Fy** represents the yield strength of the main rebars, and **Fyt** represents the yield strength of the transverse reinforcement. As can be seen, a different yield strength can be assigned to transverse reinforcement with a diameter of 12 mm and above compared to transverse reinforcement up to 10 mm in diameter.



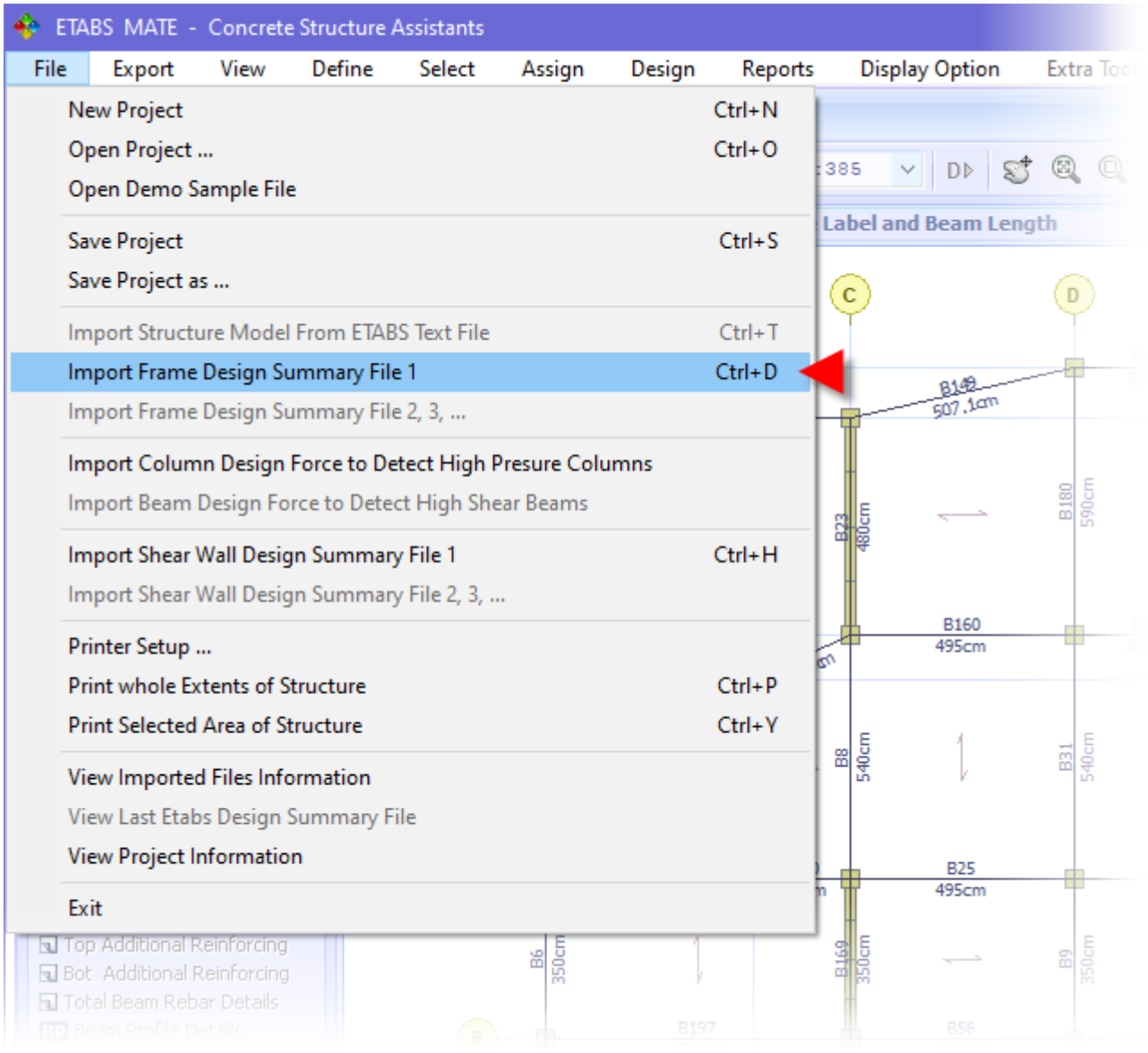
NOTE: If materials named **CONC, REBAR**, and **TIE** have been defined in the ETABS model for the concrete specifications, longitudinal rebars, and transverse reinforcement, respectively, **ETABS MATE** will automatically read the properties of these materials and display them in the window above. Otherwise, the software will display its default properties. In this case, you must adjust these parameters in accordance with those specified in the ETABS software to ensure full consistency between both programs.

Importing the Structural Design Results File into the Software

After importing the text file, the design data file must also be imported into the software as follows:

File Menu > Import Frame Design Summery File 1 (Ctrl + D)

You may also use the icon in the main toolbar at the top of the software to import the structural design results file.

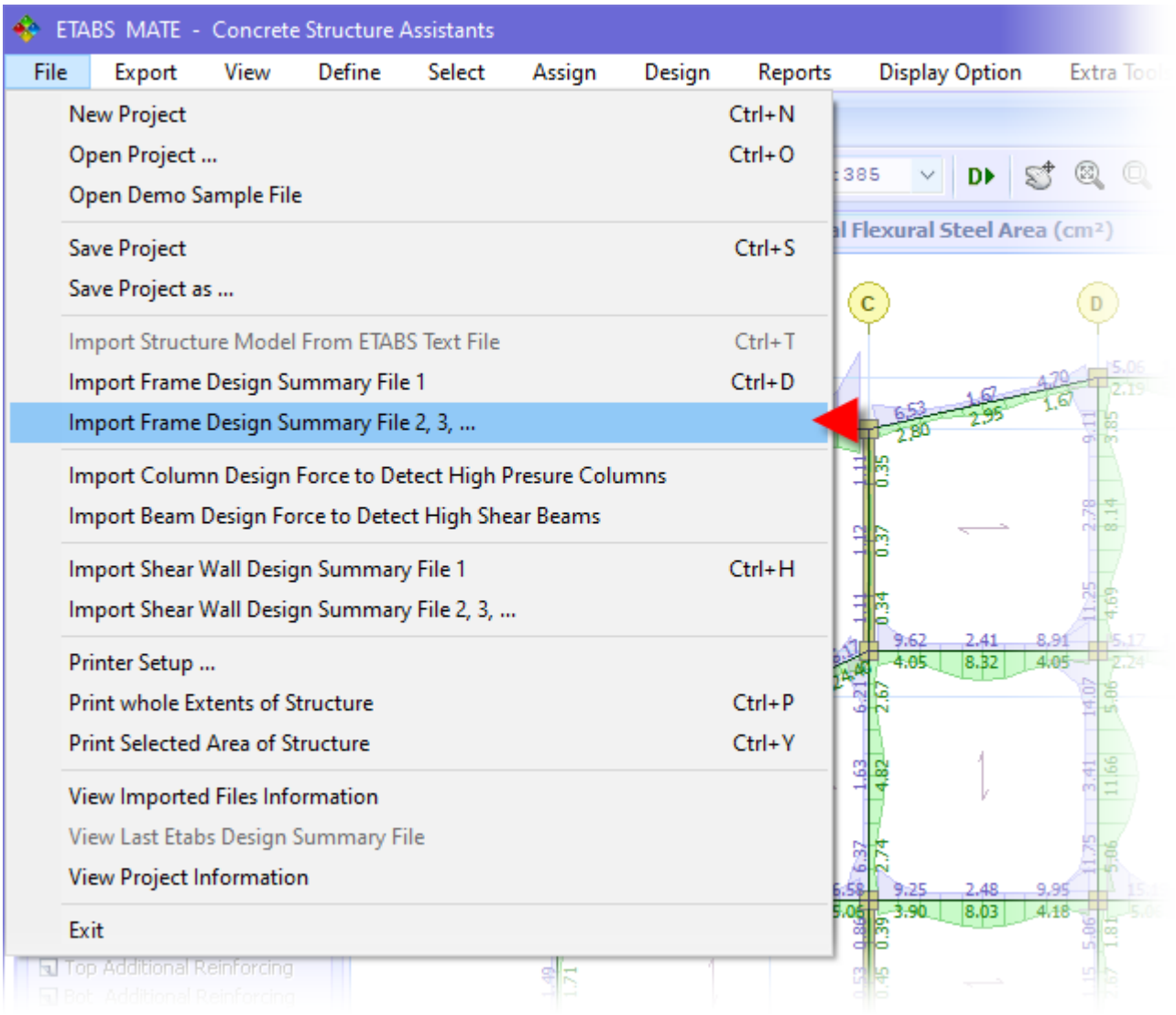


Here, import the design results file, which was exported from ETABS in **Access** or **XML** format, into the software to extract the design results of the structural elements. Upon completion of the design results import, all required design information will be displayed on the structural elements within the graphical view.

Importing Multiple Design Data Files and Generating Their Envelope

If you wish to import more than one design data file for a structure, so that the software uses the envelope of all of them for the structural detailing, use the icon , or roceed as follows:

File Menu > Import Frame Design Summery File 2, 3, ...



In this case, for each point of every element, the software obtains the maximum relevant data from all design files and then considers this maximum value for that point. These maximum values will also be displayed on the elements within the graphical view. Accordingly, the rebar detailing will be performed based on these maximum values to ensure that the resulting details and the drafted structural drawings encompass all design files imported into the software.

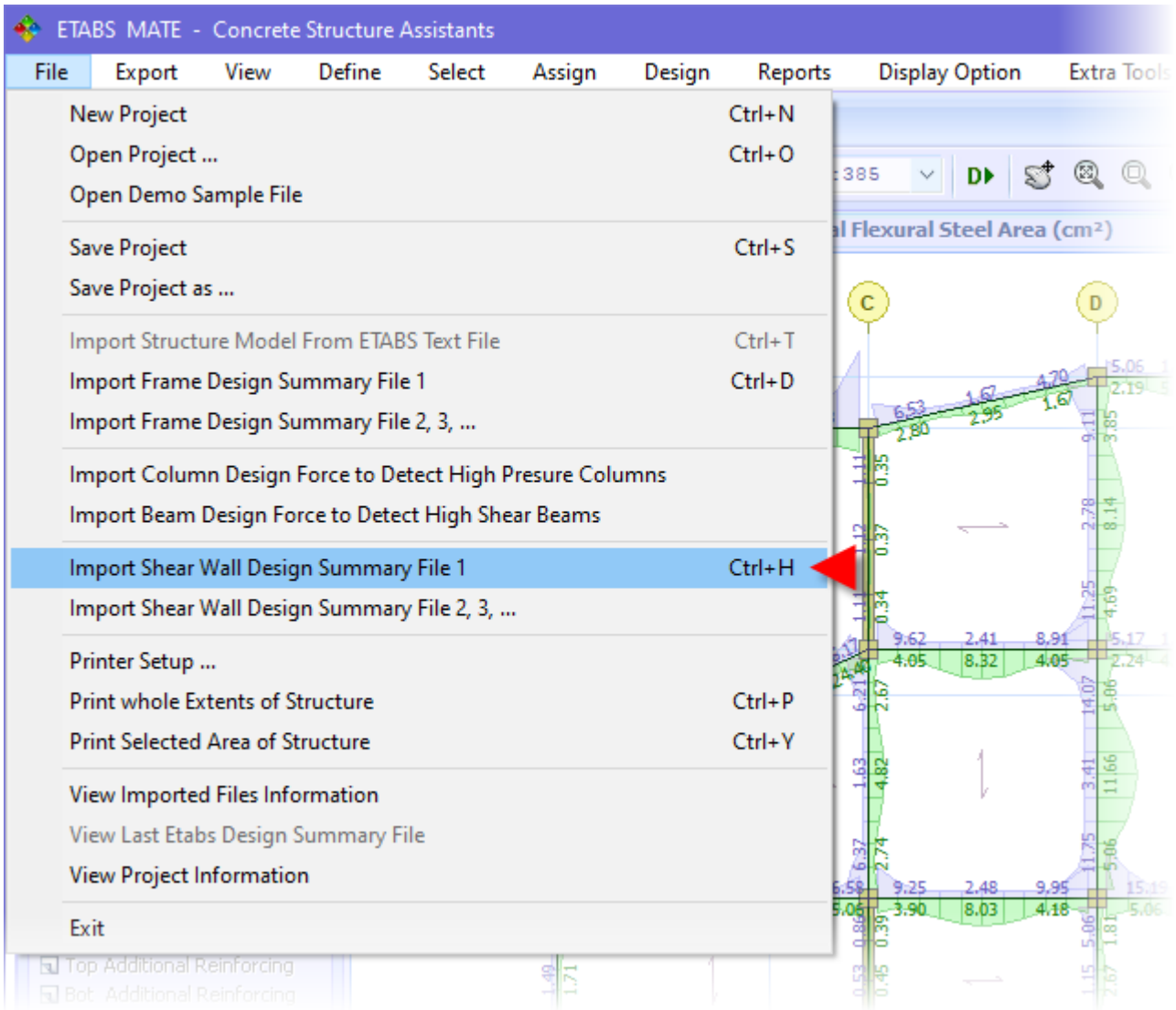
NOTE: It is strongly recommended that the element names in all design files imported into the program through this command be entirely consistent, as the software extracts design information from the design file based on the element name.

Importing the Shear Wall Design Data File into the Software

If your structure also includes shear walls, import the wall design data file, which was saved in the first step, into the software as follows:

File Menu > Import Shear Wall Design Summery File 1 (Ctrl + H)

You may also use the icon in the main toolbar at the top of the software to import the shear wall design data file.

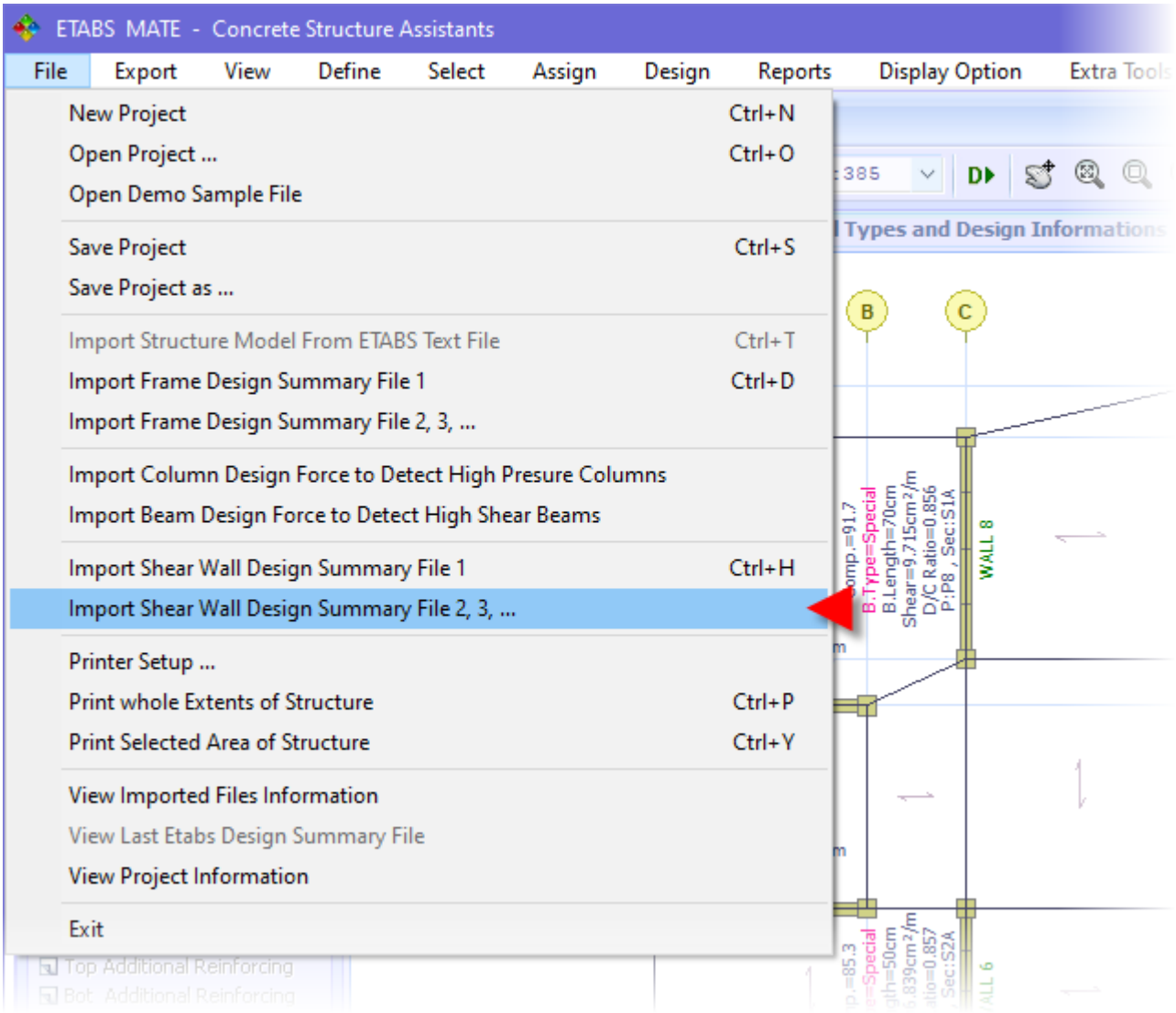


After importing the shear wall design data file, the **Wall – Pier Information** layer from the **Layer Display** panel is automatically activated. This displays all wall information on the structural shear walls within the graphical view, including the pier name, the section name assigned to the wall, the demand-to-capacity ratio of the wall section, the required shear reinforcement quantity of the wall, the calculated boundary zone length for the wall, and the type name assigned to the wall.

Importing Multiple Shear Wall Design Files and Enveloping Them

If you wish to import more than one shear wall design data file for a structure, so that the software uses the envelope of all of them for detailing the walls, proceed as follows:

File Menu > Import Shear Wall Design Summery File 2, 3, …



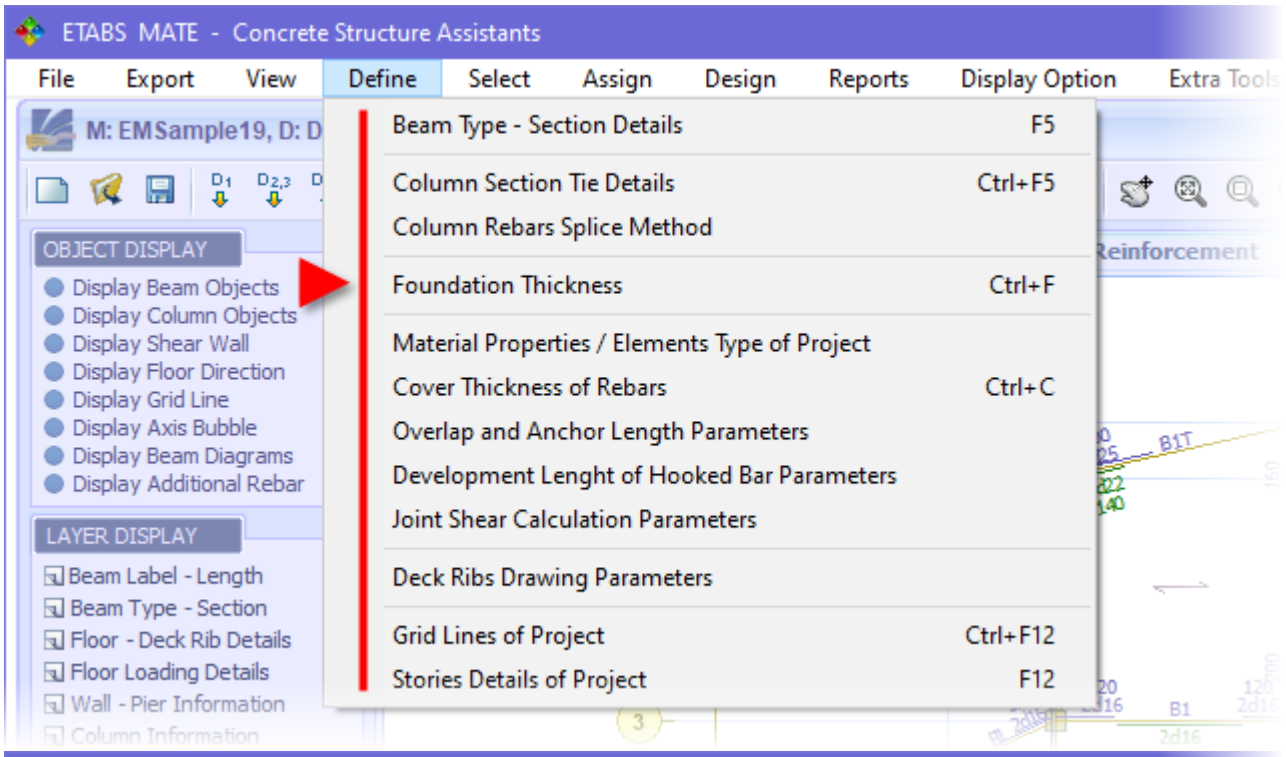
As shown in the image above, highly useful information about the walls, as well as their boundary element type, is displayed on the walls. Should any issues exist with certain wall design parameters, the software highlights these parameters in red on the element for easy identification.

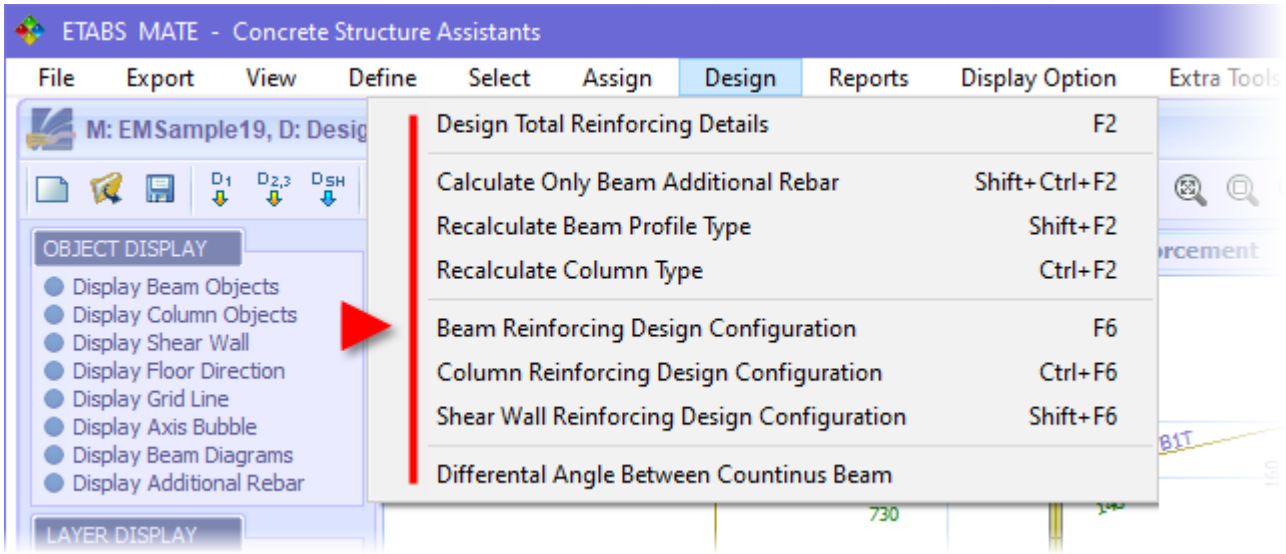
Furthermore, right-clicking on any wall opens the shear wall properties user interface, through which you can view all the aforementioned information and the sections assigned to the wall across all stories, or edit certain wall design details at various story levels.

ETABS MATE
Concrete Structure Detailing Software

3 Detailing of Structural Elements

After importing the structure, the software automatically applies default settings for the design parameters of the structural elements. Prior to performing the rebar detailing, you may, if necessary, review or modify these default settings, such as the strength properties of concrete and steel, the concrete cover for structural rebars, the settings governing the calculation of development length and splice length of rebars, the continuous and transverse reinforcement of sections, as well as the software's design configuration settings. These adjustments can be made using the options available in the **Design** menu and the **Define** menu, all of which will be elaborated upon later in this guide.





Design Reinforcement Details of Structural Elements

After importing the structural model and design data into the software and performing the necessary configurations, you can execute the design reinforcement details command and classify the project elements.

To do so, you may use the icon in the main toolbar at the top of the software, or alternatively, use the **Design** menu or the keyboard shortcut as follows:

Design Menu > Design Total Reinforcing Details (F2)

Executing this command opens the **Structure Reinforcing Design** window, which displays the design process of the various structural elements to the user. As shown in the image below, this process begins with the classification of the structural columns and the design of their transverse reinforcement. Subsequently, the rebar detailing of the shear walls is calculated, and finally, the design process for the beam additional reinforcement is performed, along with the necessary classifications. The entire detailing process is executed very rapidly by the software; for a typical building of approximately ten thousand square meters in area, this process takes less than one second to complete.

Upon completion of the reinforcement design of the structure, the designed details are displayed by the software within the graphical view. By changing the display layers through the **Layer Display** panel, you can access all these software-designed details. After the design is complete, you will be able to generate the structural drawings, which will be explained in detail in the following sections.

4 Generating Structural Drawings

Upon completing the structural detailing and performing the necessary checks and corrections, you can issue the command to generate the structural drawings by the software. Thanks to the powerful internal drafting engine coded within the program, the process of generating structural drawings is extremely fast; for typical buildings, this process is completed in merely a fraction of a second, and the drafted structural drawings are saved in the standard AutoCAD format. It is worth noting that the generated drawings can be easily viewed and edited by all versions of the AutoCAD software. This capability allows you to utilize all the features of the widely used AutoCAD software to perform final revisions.

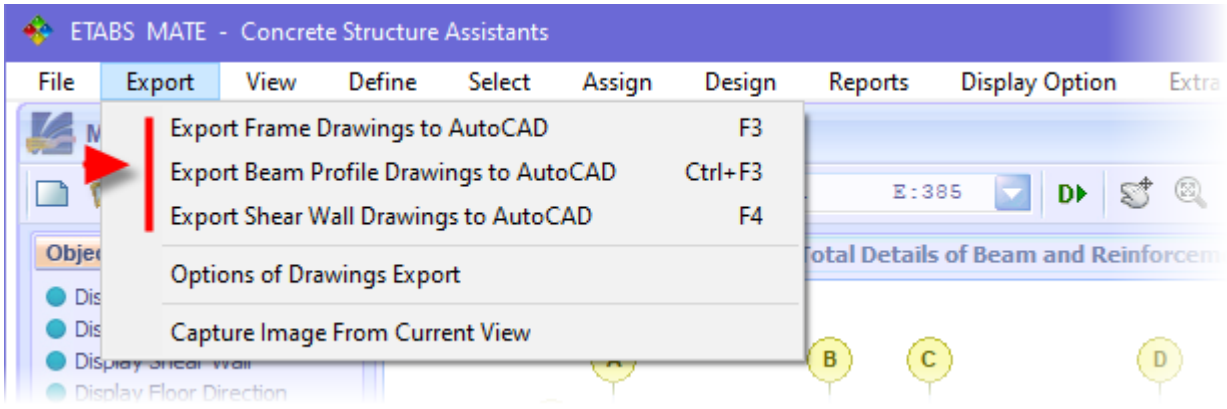
For greater program flexibility, the structural drawings are generated using three separate commands. To produce the frame structural drawings, the longitudinal beam profiles, and the shear wall drawings of the project, along with their corresponding rebar schedules, you may use the tools available in the main toolbar of the software, as identified in the image below. These are described in the following section:

F CAD Generate frame structural drawings, including columns and beam details in plan (F3)

B CAD Generate beam detail structural drawings as longitudinal profiles (Ctrl + F3)

S CAD Generate shear wall detail structural drawings for the project (F4)

As shown in the image below, in addition to the tools available in the main toolbar at the top of the software, users can also utilize the options in the **Export** menu to generate and export structural drawings. Furthermore, through the **Option of Drawings Export** setting, you can configure the drawing unit for the structural drawings, the font style of the drawings, and the generation of the rebar schedule Excel file accompanying the drawings.



ETABS MATE
Concrete Structure Detailing Software

Several Critical Notes

To ensure the correct use of the software, please observe the following points when modeling your project in ETABS.

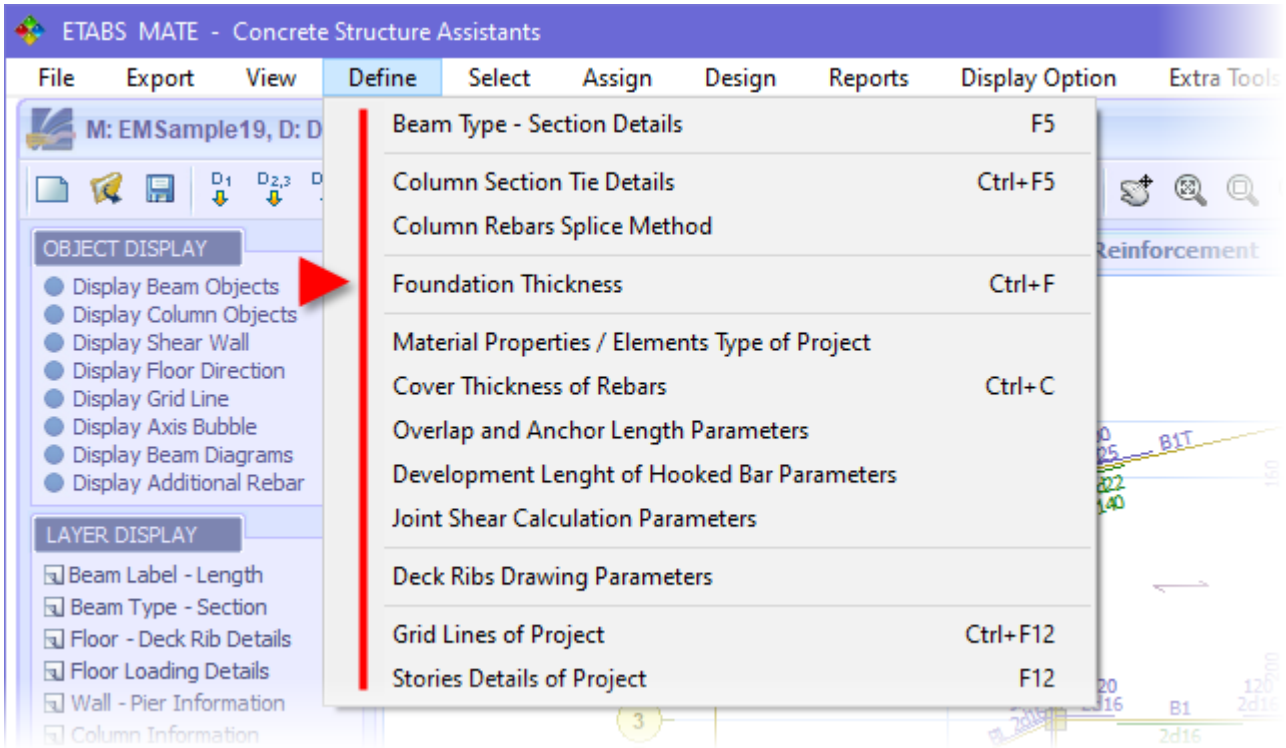
- When assigning names to sections, stories, piers, etc., in the structural model, never use illegal characters, particularly the comma character. The use of the comma character can cause serious issues when reading text files.
- Upon completing the modeling, and especially when the sections assigned to model elements have been changed, always execute the **Auto Relabel All** command from the **Edit** menu in the ETABS software. Then, re-analyze and redesign the model, regenerate the e2k file and the design output files, and import them into **ETABS MATE**. It is worth noting that using the **Auto Relabel All** command not only organizes element naming but also eliminates many model errors. However, a very important point must be carefully considered: after executing this command, any files derived from the original file, such as 25%, 50%, or any other variation, must also be regenerated from the original file to ensure that element names remain consistent across all files, as design results are extracted from the tables based on element names.
- Before exporting the structural design results using the **Export** option in the **File** menu, ensure that the unit system of the ETABS software is set to Kgf, cm. Furthermore, when ETABS prompts you for the output unit system, select Kgf, cm once again. Otherwise, the values will not be exported from ETABS with the correct units. This recommendation is due to a bug present in certain versions of the ETABS software.
- The rebars used in the ETABS software must be in the format **16d, 18d, 20d, … or 16, 18, 20, ….** In other words, the rebar names must be either a single number representing the rebar diameter in millimeters, such as 20, or the rebar diameter in millimeters followed by a suffix of Latin characters, such as 20d. If the rebars in the project are defined in a different format, they must be redefined accordingly. To do this, you can edit and modify the rebar definitions using the **Reinforcing Bar Size** user interface in ETABS. To access this interface in ETABS, proceed as follows: **ETABS > Define menu > Section Properties > Reinforcing Bar Sizes**
- Note that the rebar cover parameters for various elements in the ETABS and **ETABS MATE** software must be in complete agreement. The cover parameter for different elements in **ETABS MATE** corresponds precisely to the code-specified cover, i.e., the distance from the concrete surface to the outer face of the transverse reinforcement. Please be aware that in some versions of ETABS, the cover you specify refers to the distance to the rebar axis and not the code-specified cover. Therefore, setting the cover parameter in ETABS must be performed with greater care, taking this issue into account, to ensure full consistency between both software programs.

ETABS MATE
Concrete Structure Detailing Software

Defining Structural Properties of Project Elements

As shown in the image below, through the **Define** menu in the software, you can view or define numerous structural parameters, which include the following options. These are described in detail in the subsequent sections.

- Define parameters and properties of beam sections
- Define parameters and properties of column sections
- Specify the column rebar splice type
- Specify the foundation thickness
- Define the strength properties of concrete and rebars, and the structural ductility type
- Specify the concrete cover for rebars of various structural elements
- Configure the calculation parameters for the development and splice length of rebars
- Configure the calculation parameters for the development length of hooked bars
- Configure the calculation parameters for the joint shear ratio
- Configure the drawing and calculation parameters for ribs in joist and block floors
- Define the properties and specifications of the project grid lines
- Define the specifications of the project stories

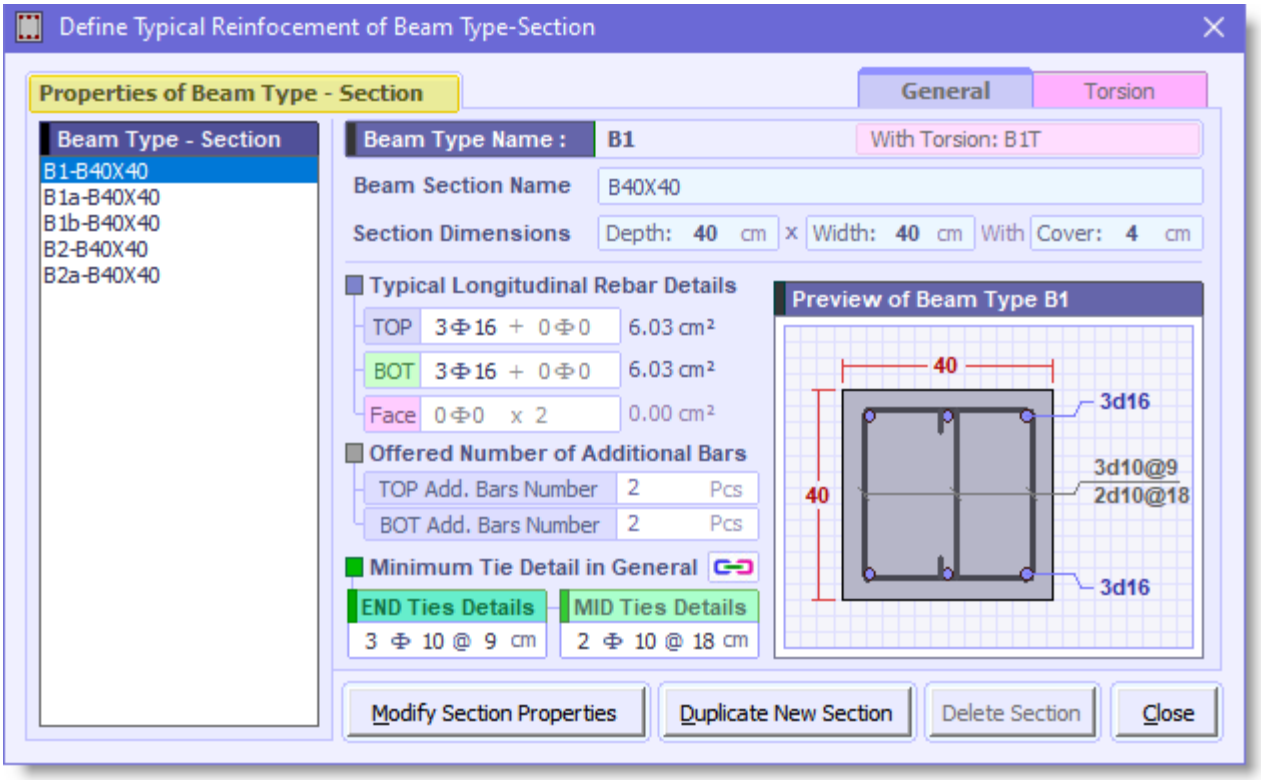


ETABS MATE
Concrete Structure Detailing Software

Defining Beam Section Properties

Define Menu > Beam Type - Section Details (F5)

After importing the structural model, the software automatically extracts all beam sections defined in the project that are utilized in the structure. Subsequently, based on code provisions, it automatically assigns continuous rebars and minimum transverse reinforcement, i.e., stirrups, to each one. Through this user interface, you can manage all beam sections of the project, and view or edit the properties assigned to each of these sections.



By selecting any **Beam Type - Section**, you can view or edit its properties, which are as follows:

TOP: Number and diameter of continuous top rebars (combination of two diameters also permitted)

BOT: Number and diameter of continuous bottom rebars (combination of two diameters also permitted)

FACE: The number and diameter of the continuous face rebars located on the side faces of the section.

END Ties Details: The minimum transverse reinforcement details at the special end regions of the beam, comprising the number of legs, diameter, and spacing of the ties. For example, as illustrated in the image, the detail **3 Ø 10 @ 9 cm** signifies the use of transverse reinforcement with a diameter of 10 mm, with three legs (i.e., one closed stirrup plus one crosstie), spaced at 9 cm.

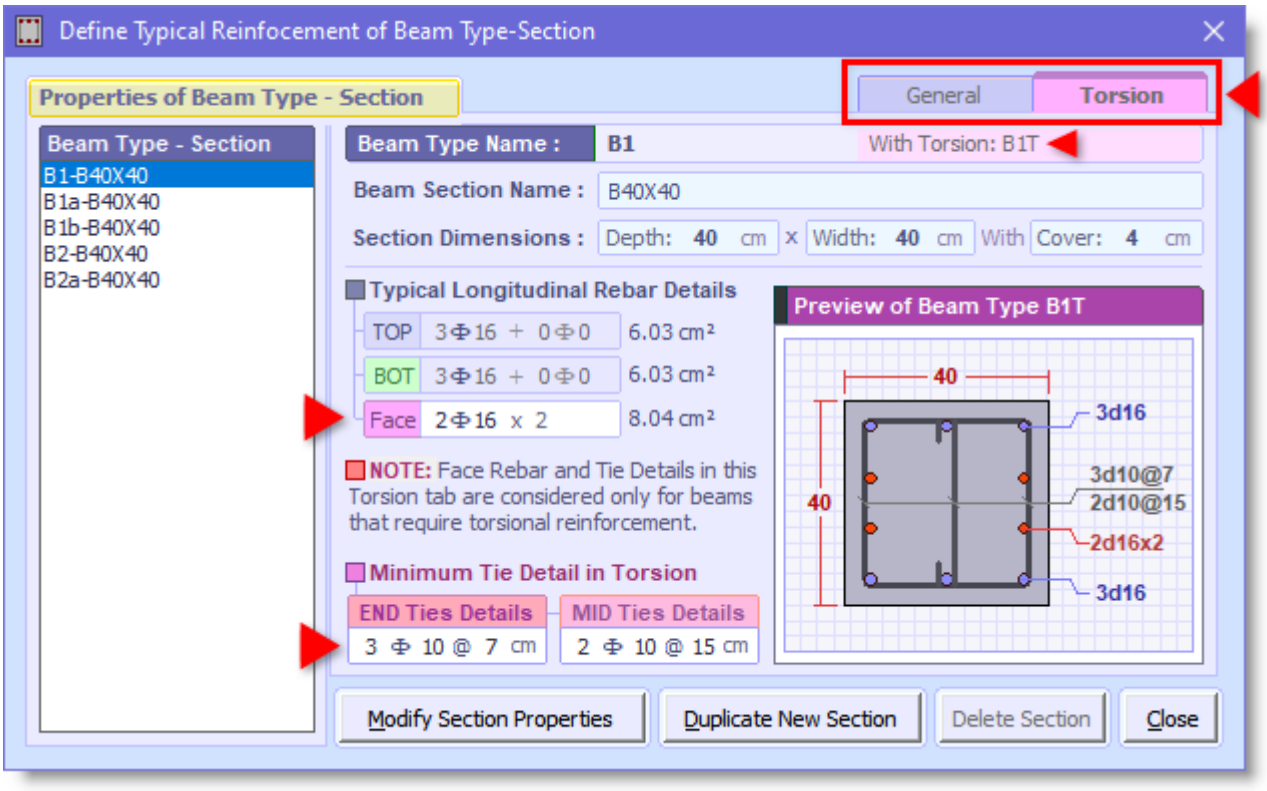
MID Ties Details: The minimum transverse reinforcement details at the mid-span region of the beam, comprising the number of legs, diameter, and spacing of the ties. For example, the detail **2 Ø 10 @ 18 cm** signifies the use of transverse reinforcement with a diameter of 10 mm, with two legs as a closed stirrup, spaced at 18 cm.

TOP Add. Bars Number: User's preferred suggestion for the number of top additional rebars.
BOT Add. Bars Number: User's preferred suggestion for the number of bottom additional rebars.

It is worth noting that these user-suggested values for the number of additional rebars represent only the user's preference and do not compel the software to calculate the additional rebars strictly with these quantities. In other words, the software attempts to accommodate the user's preference within its computational procedures. However, if a smaller number of additional rebars suffices to meet the required reinforcement quantity, the software will adopt that smaller quantity in its detailing. Conversely, if a greater number of additional rebars than the user's preferred quantity is required to satisfy the needed reinforcement, that larger quantity will be adopted.

General and Torsion Tabs

As shown in the image, this user interface contains two tabs, **General** and **Torsion**. This allows you to define a **General** and a **Torsion** version with different sets of details for each section.



If different details are defined for the **General** and **Torsion** versions of a section, the program operates as follows: if the beam does not require torsional reinforcement, the software uses the properties defined in the **General** version of the section for the beam detailing. If the beam requires torsional reinforcement, the software uses the properties defined in the **Torsion** version of the section for the beam detailing. For example, in the **General** version, you may choose not to define side face rebars; however, in the **Torsion** version of the section, considering it is used for a beam subjected to torsion, you may define side face rebars and also specify a greater amount of transverse reinforcement compared to the general section.

It is worth noting that the software automatically defines the side face rebars for the **Torsion** version of beam sections. The side face rebar details automatically defined by the software are calculated as a percentage of the minimum torsional reinforcement of the section. Users can, however, configure how this parameter is calculated. To do so, select **Beam Reinforcing Design Configuration** from the **Design** menu to open the beam design configuration user interface. Then, using the below parameter:

Percentage of Minimum Torsional Steel for Face Rebar in Torsion Section

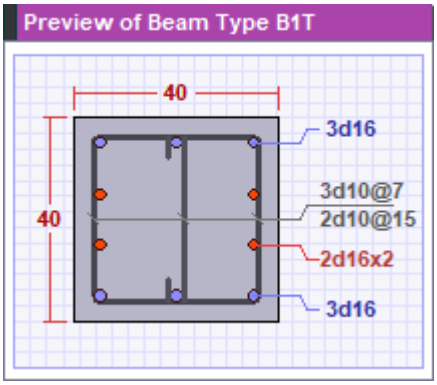
You can specify what percentage of the minimum torsional reinforcement of the section should be considered as side face rebars. It is worth noting that increasing the value of this parameter causes the software to consider a greater amount of side face rebars by default for beam sections. This results in a larger portion of the torsional reinforcement being accommodated in the beam's side faces, thereby requiring a smaller quantity for the additional top and bottom rebars. This can lead to better detailing and a reduction in the joint shear ratio. On the other hand, since it is preferable for torsional reinforcement to be distributed uniformly across all four faces of the section, this parameter cannot be set to an excessively large value. Therefore, considering the project characteristics, the user should determine the optimal value for this parameter to achieve an efficient design.

Furthermore, you can enable or disable the use of the Torsion version of beam sections for beams that require torsional reinforcement. To do so, in the same beam design configuration user interface, you can use the below parameter:

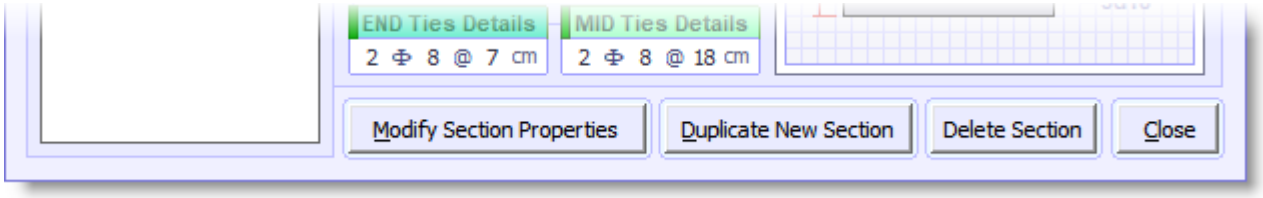
Use Torsion Properties for Beam that Require Torsion Reinforcement

to activate or deactivate the use of the Torsion version of beam sections. By deactivating this parameter, the software will no longer use the Torsion version of the sections and will employ the details defined in the General version for the detailing of all project beams, regardless of whether they require torsional reinforcement.

Upon making any changes to the beam details and properties, these changes will be visible in the **Preview Section** area, allowing the user to have a completely accurate and realistic perception of the details defined for the beam section.



Additionally, several buttons have been designed at the bottom of the beam section definition user interface, which are described below:



Modify Section Properties

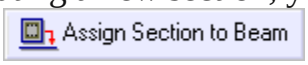
After making changes to any section, pressing this button applies the modifications.

Delete Section

This button allows you to delete unused sections that are not assigned to any beam.

Duplicate New Section

This button allows you to define a new section with different continuous rebar, side face rebar, or transverse reinforcement details based on an existing section. To do so, first select one of the beam sections, for example B4, then change its name, for instance to B4a, modify the desired properties, and finally press the **Duplicate New Section** button to create the new section with the specified name and properties, which will then be added to the project beam section list.

After duplicating and creating a new section, you can assign this section to your desired beams. To do this, use the button 

in the side palette of the software, or navigate through the **Assign** menu and select **Assign Section to Beam (F7)** to assign new beam sections to the desired beams. First, select the beam or beams of interest, then press this button to display the beam section list. Select the desired section, and finally click the **Assign to Selected Beam** button. Note that the dimensions of the selected section must be consistent with the current dimensions of the beam section; otherwise, assigning the new section will not be possible. This is because any change in element dimensions requires re-analysis and redesign in ETABS.

The ability to duplicate new sections allows you, for certain structural axes where a large amount of additional reinforcement has been calculated, to duplicate and create a new section with a greater quantity of continuous rebars and assign it to the desired axis, so that after redesign, less additional reinforcement is calculated for them. Similarly, for beams with nonstandard stirrup configurations, you can duplicate new sections with a different number of legs, diameter, or spacing and assign them to the desired beams, ensuring that the stirrup details for those beams are applied exactly as you intended.

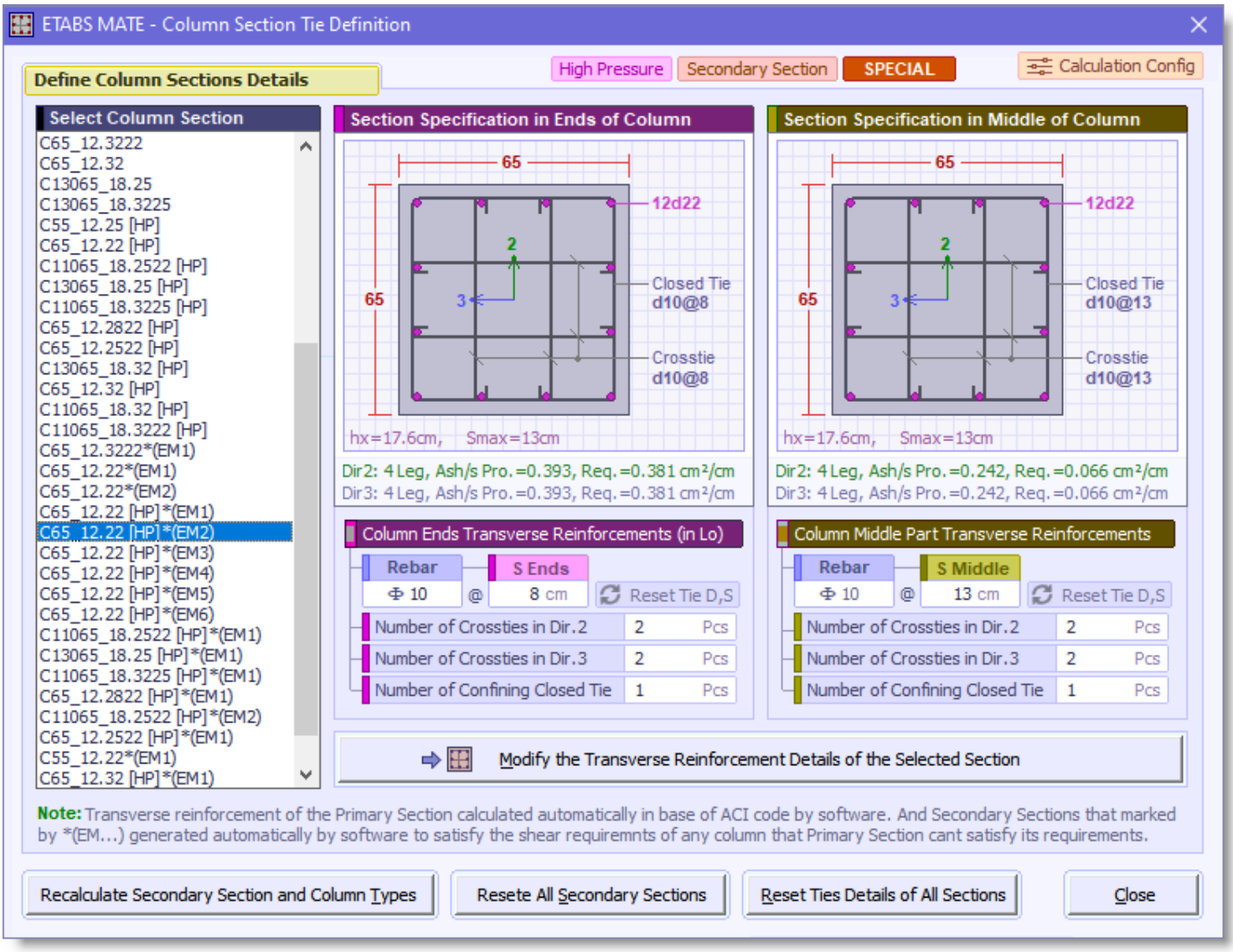
It is worth noting that after making any changes to the section properties, all previously designed beam rebar detailing will be discarded, and you must re-execute the design reinforcement details command to redesign the structural reinforcement detailing.

ETABS MATE
Concrete Structure Detailing Software

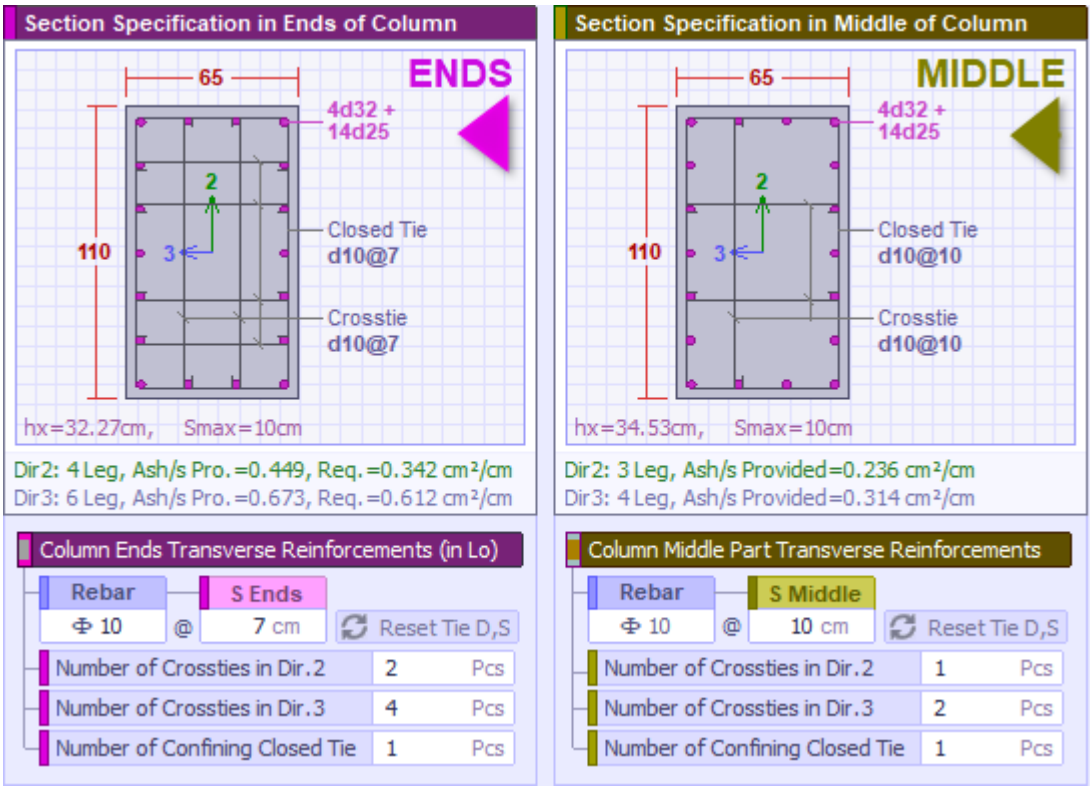
Defining Column Section Properties

Define Menu > Column Section Tie Details (Ctrl + F5)

After importing the structural model, the software automatically extracts all column sections defined in the project that are utilized in the structure. Subsequently, based on code provisions, it automatically assigns minimum transverse reinforcement to each one. Through the column section tie definition user interface, you can view or edit the closed tie and crosstie details of the project column sections. As shown in the image, all column information, including the ductility type, primary or secondary section designation, low-compression or High Pressure status, longitudinal rebar arrangement, crosstie configuration in each direction, tie diameter and spacing, the required transverse reinforcement quantity of the section based on the code, and much other information, can be viewed and edited for each section via this interface.



Given that code requirements for special end regions of length l_0 at the top and bottom of a column are considerably more stringent than those for the middle region, the software provides the capability to define the transverse reinforcement details of columns in the end regions **(Ends)** separately from the middle region **(Middle)**. In other words, as shown in the image below, in addition to the tie spacing, other parameters such as crosstie configuration and the number of closed perimeter ties of the section can be defined differently for the top and bottom end regions and the middle region of the column. This feature enables a significantly more economical and optimized design for columns, particularly in structures with special ductility.



The parameters shown in the image above are described below:

Rebar: The diameter of the transverse reinforcement in the **Ends** and **Middle** regions of the column.

S Ends: The spacing of transverse reinforcement in the **End** regions of the column.

S Middle: The spacing of transverse reinforcement in the **Middle** region of the column.

Number of Crossties in Dir. 2: The number of crossties parallel to direction 2 of the column section.

Number of Crossties in Dir. 3: The number of crossties parallel to direction 3 of the column section.

Number of Confining Closed Tie: The number of perimeter confining closed ties of the section (single or double perimeter tie).

By selecting any section, indicators appear at the top of this interface, which are described below:

Primary Section

Indicates that the selected section is of the Primary type. The transverse reinforcement of primary sections is automatically designed by the software to satisfy the code requirements of the section, including limits on tie and crosstie spacing and diameter, as well as the A_{sh}/sb_c relationships.

Secondary Section

Indicates that the selected section is of the Secondary type. The transverse reinforcement of secondary sections is designed by the software to satisfy, in addition to code requirements, the shear reinforcement quantity A_V/s calculated by ETABS for the column or the requirements for High Pressure columns. In other words, if the transverse reinforcement quantity of the Primary section does not meet the column shear reinforcement demand, the software automatically generates the Secondary section and designs its transverse reinforcement quantity to satisfy the column shear reinforcement demand. In addition to defining secondary sections, the software also automatically assigns these sections to the corresponding column. Secondary sections are named by the software using a combination of the primary section name and the suffix ***(EM…)**.

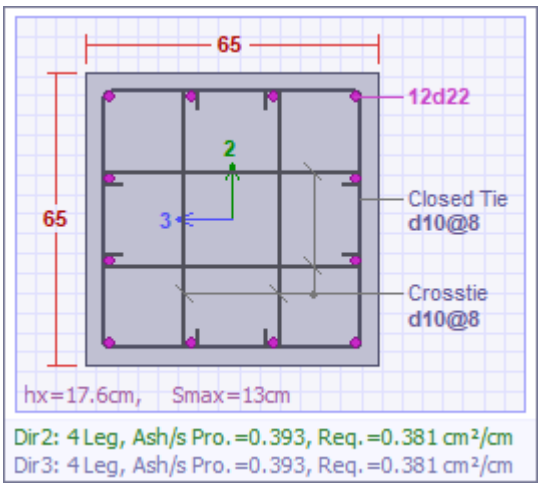
High Pressure

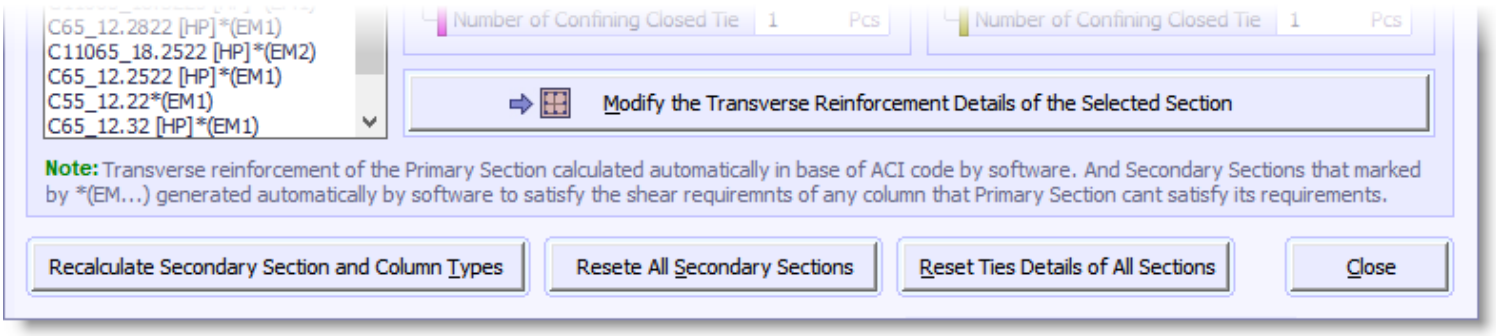
Indicates that the selected section is used for a column identified by the software as being under high compression. These sections are automatically named by the software using a combination of the primary section name and the suffix **[HP]**.

SPECIAL

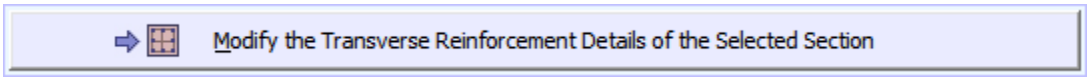
Indicates that the transverse reinforcement of the column sections has been designed under the special ductility condition.

As shown in the image, other highly useful and practical information is also visible in this user interface, including the Section Dimensions, the details and configuration of the Longitudinal Rebars, the details of the Closed Ties and Crossties, the Number of Transverse Reinforcement Legs in each direction of the section, the parameter S_{max}, i.e., the maximum permissible tie spacing based on the ACI Code, the value of the parameter hx, which is used in calculating the tie spacing, the parameter Ash/s Req., i.e., the required transverse reinforcement quantity of the section, and the parameter Ash/s Pro., i.e., the provided transverse reinforcement quantity supplied by the ties and crossties in each direction of the section.

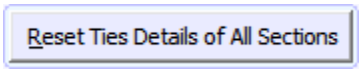




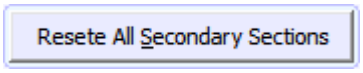
The buttons located at the bottom of this user interface are described below:



This button must be used to save the changes made, after editing the properties for any section.

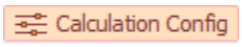


This tool recalculates the transverse reinforcement details of all primary sections based on the ductility type, code requirements, and the parameters defined in the column design configuration user interface. It then regenerates the secondary sections, designs their transverse reinforcement based on the shear demands of the corresponding element, and subsequently re-executes the column classification process. In other words, using this tool discards all user edits made to the column sections, and the details of all sections, including primary and secondary sections, are reset to the software's computational defaults.



This tool deletes all secondary sections, then regenerates them based on the primary section configuration, redesigns their transverse reinforcement details based on the shear demands of the corresponding element, and subsequently re-executes the column classification process. This tool is useful when the user has modified the transverse reinforcement details of the sections and wishes to have the transverse reinforcement details of the secondary sections redesigned based on the edited primary sections.

This tool re-checks the transverse reinforcement details of the secondary sections based on code requirements, the shear reinforcement quantity A_V/s calculated by ETABS, and the requirements for High Pressure columns. If, based on user edits, the generation of new secondary sections is necessary, new secondary sections are generated, and the software then re-executes the column classification process.



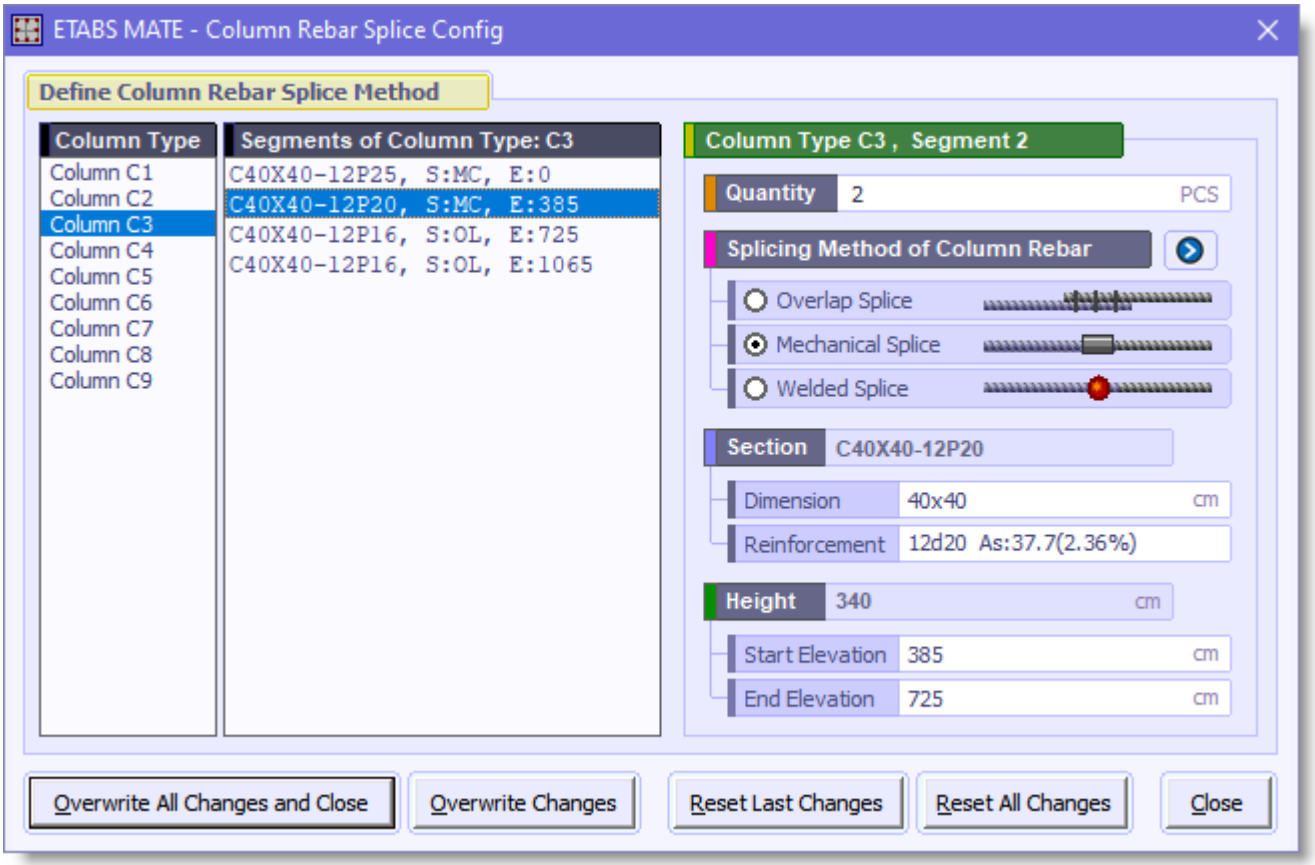
This tool provides access to the user interface for configuring the design and detailing of columns.

Specify Splice Type of Column Rebars

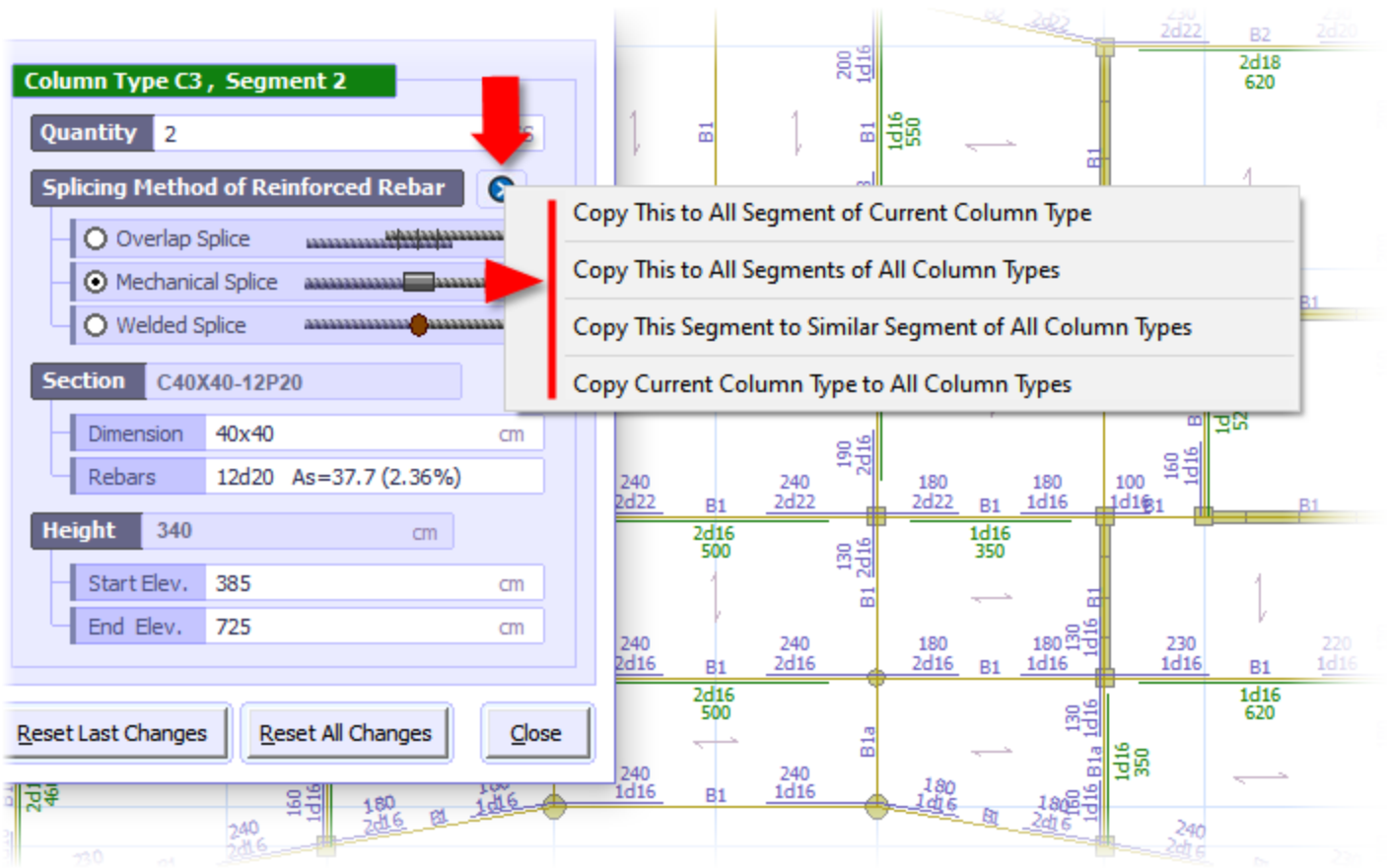
Define Menu > Column Rebars Splice Method

This option allows you to select the splice type of the longitudinal rebars for each column type and at the desired stories from three different methods: **Overlap Splice** (**OL**), **Mechanical Splice** (**MC**), **and Welded Splice** (**WL**).

To do this, first select the desired column type from the **Column Type** section. Upon selecting a column type, the list of column segments across various stories and their splice types is displayed in the **Segment of Column Type** section. In this list, select the desired segment, then choose the intended splice type for the rebars of that column segment from the **Splicing Method of Reinforced Rebar** section, and press the **Overwrite Changes** button. For example, as shown in the image below, for column type C3, a mechanical splice (**S:MC**) is used for the first and second stories, and an overlap splice (**S:OL**) is used for the third and fourth stories.



Upon selecting each segment, highly useful information is displayed in the right-hand section of the window, including the number of columns in each column type, the splice type, the assigned section name, section dimensions, section rebars, rebar area and reinforcement ratio, and the start and end elevations of the selected segment, greatly facilitating your decision regarding the splice type. By using the icon you can apply the specified splice type to other segments as described below.



To copy the splice type of one segment to other segments, first click on the icon to open the replication menu, as shown in the image above. Each item of this menu is described below.

- **Copy This to All Segment of Current Column Type**

This item applies the splice type of the current segment to all segments within the selected column type. In other words, all column segments from bottom to top become identical.

- **Copy This to All Segment of All Column Types**

This item applies the splice type of the current segment to all segments across all column types. In other words, all column segments from bottom to top, across all column types, become identical. This option is particularly useful when, for example, you wish to set the splice type of all columns to mechanical splice.

- **Copy This to Segment to Similar Segment of All Column Types**

This item applies the splice type of the current segment to all elevation-wise similar segments across all column types. In other words, for instance, the first story will have the same splice type across all columns.

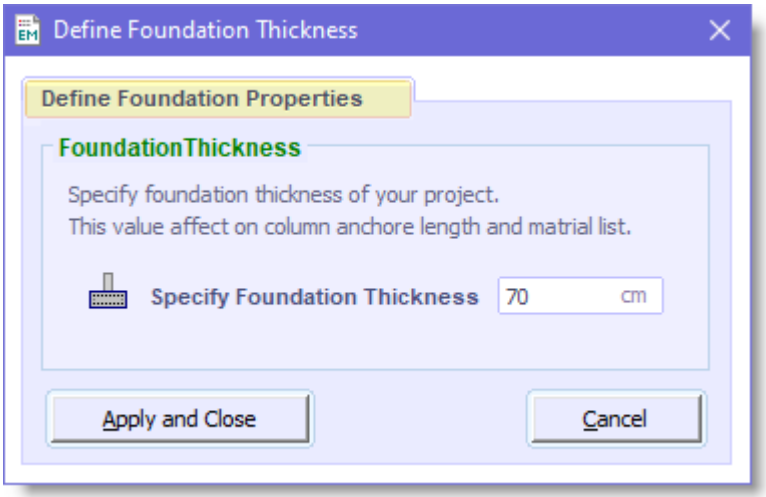
- **Copy Current Column Type to All Column Types**

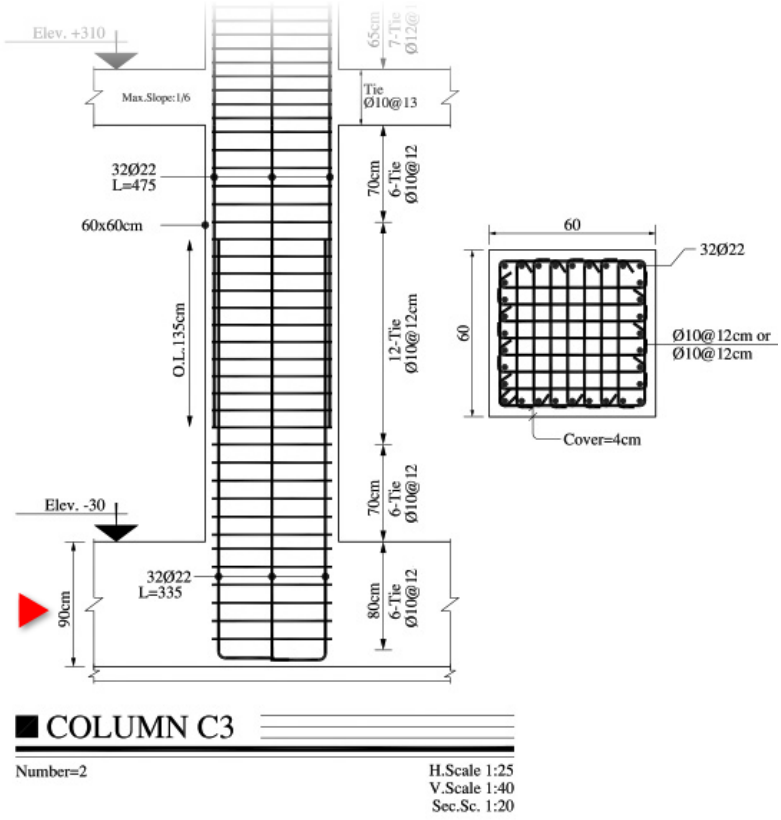
This item applies the splice types of the various segments of one column type to all elevation-wise similar segments across all column types. In other words, if you wish to have a mechanical splice for the first and second stories of the column types and an overlap splice for the remaining stories, first edit one of the column types and then use this option to apply the settings configured for that column type to all other column types.

Specifying Foundation Thickness

Define Menu > Foundation Thickness

As shown in the image below, this option allows you to specify the foundation thickness of the structure. The value of this parameter affects the embedment length of columns and shear walls into the foundation and, as illustrated in the image below, is also annotated on the foundation in the column longitudinal profile structural drawings.

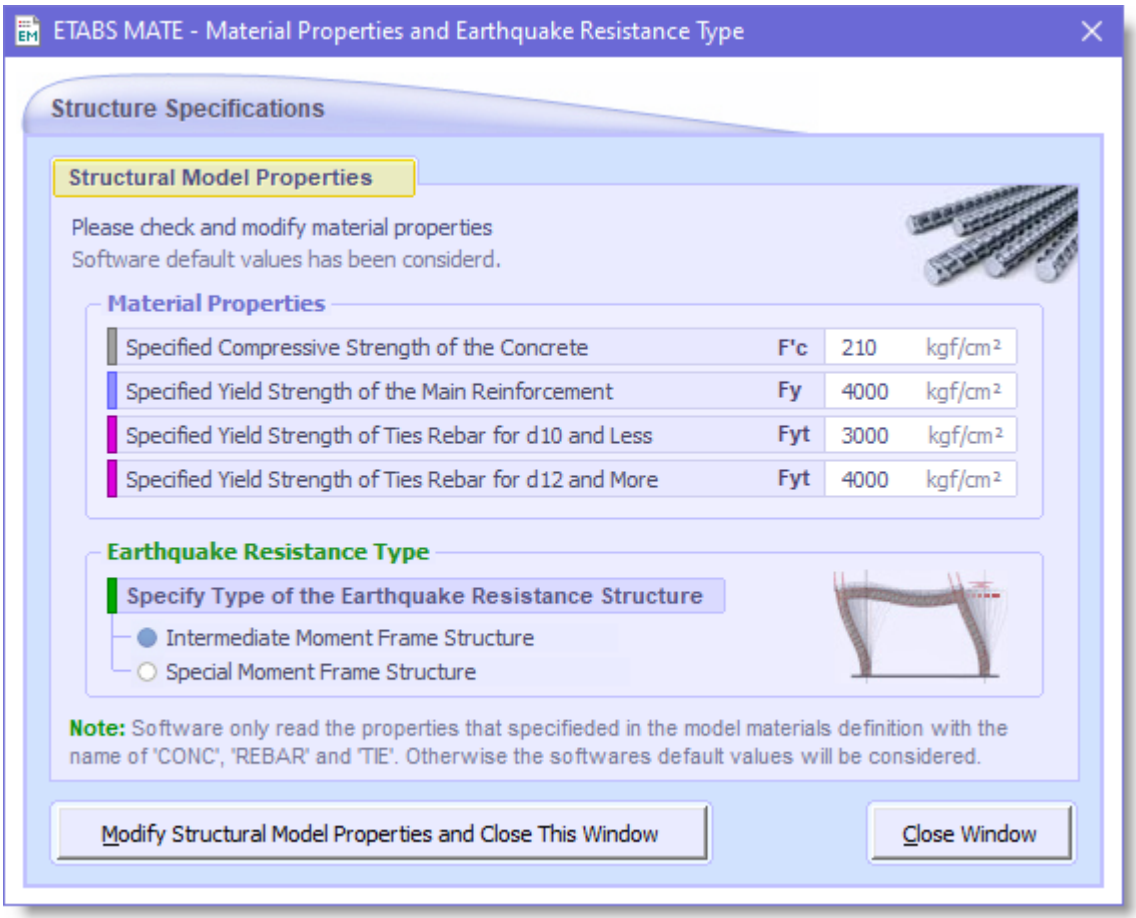




Specifying Material Properties and Earthquake Resistance Type

Define Menu > Material properties of projects

As shown in the image below, this window allows you to define the material properties of the project and the earthquake resistance type of the structure. In the **Material Properties** section, the project material specifications can be configured. These specifications are as follows: **F'c**, the compressive strength of concrete; **Fy**, the yield strength of the main rebars; and **Fyt**, the yield strength of the transverse reinforcement. As can be seen, a different yield strength can be assigned to transverse reinforcement with a diameter of 12 mm and above compared to transverse reinforcement up to 10 mm in diameter. Furthermore, in the **Earthquake Resistant Type** section, you can specify the earthquake resistance type of the structure. It is worth noting that certain parameters in this window are editable only prior to importing the design data files. Therefore, it is recommended to perform the necessary settings after importing the text file; for this purpose, upon importing the text file, this user interface automatically appears to allow these settings to be configured.

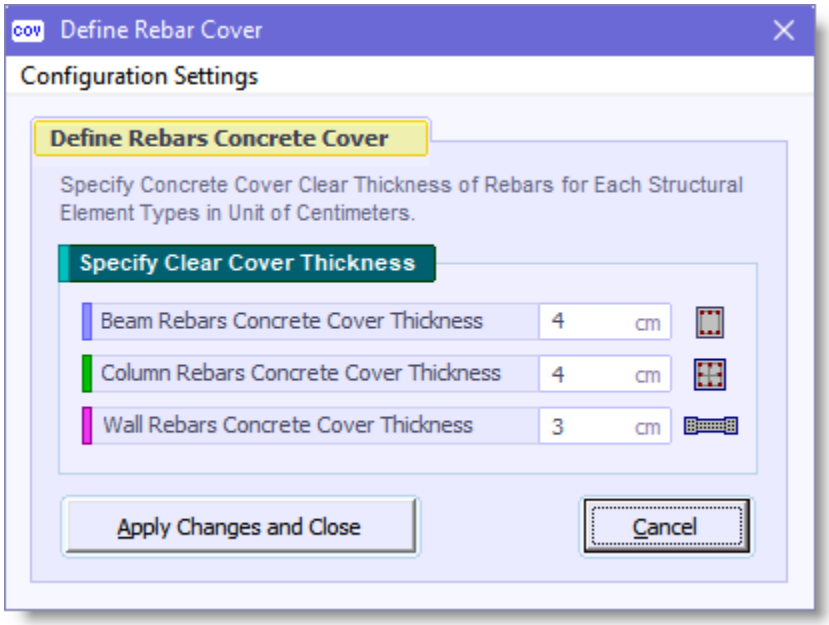


The software automatically extracts the properties defined for the **CONC**, **REBAR**, and **TIE** materials in the ETABS structural file, if they exist, then applies them to these parameters. If it does not find materials with the aforementioned names, it will apply its own default properties.

Specifying Concrete Cover Thickness of Rebars

Define Menu > Cover Thickness of Rebars

As shown in the image below, this option allows you to configure the concrete cover thickness over the rebars for various structural elements. It is worth noting that these parameters represent code-specified values; for instance, in beams and columns, this parameter is the distance from the concrete surface of the element to the outer face of the transverse reinforcement.



These values influence several design processes, including the spacing of transverse reinforcement in beams and columns, the calculation of the number of rebars in the section of shear walls, and many other software processes. They are also annotated on the drawn sections of the elements in the structural drawings. Furthermore, note that the cover defined for the elements in the ETABS software must be entirely consistent with the cover entered in **ETABS MATE**. Also, be aware that in some cases, the cover entered in ETABS refers to the distance to the rebar axis rather than the code-specified cover; therefore, specifying the cover in ETABS must be performed with greater care, taking this issue into consideration.

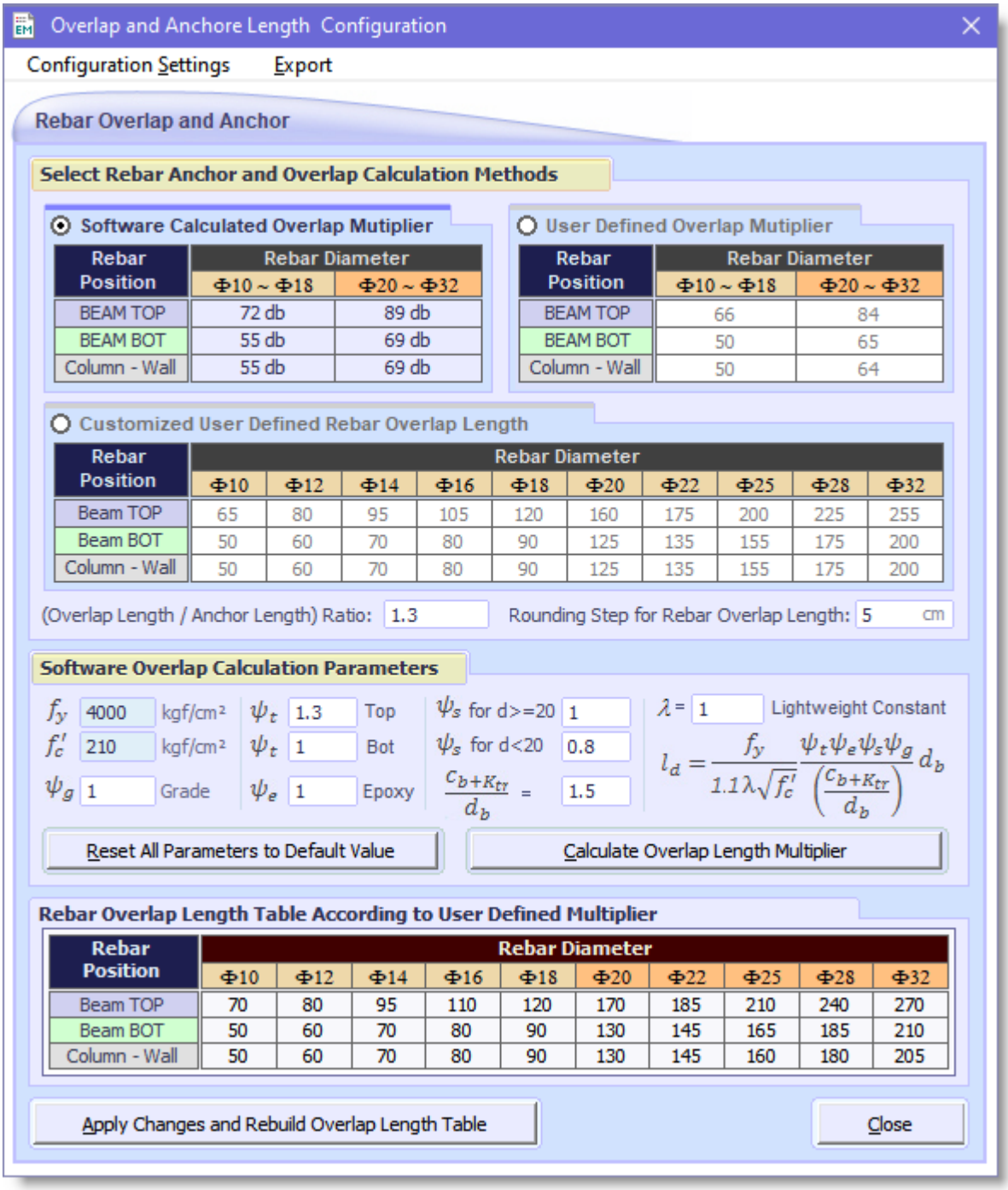
After configuring the concrete cover values for the elements, you can save the specified values via the **Configuration Setting** menu and the **Save User Default Configuration** option, so that the software uses these values as the default concrete cover for rebars in other projects. This menu also contains additional items for deleting saved settings or restoring the software's default values for the parameters in this user interface.

ETABS MATE
Concrete Structure Detailing Software

Configuring Splice and Development Length of Rebars

Define Menu > Overlap and Anchor length parameters

This option allows you to configure the parameters governing the calculation of the development length and splice length of structural rebars, which are used in various reinforcement design processes of the software. As shown in the image below, the software provides users with three different methods for calculating the development length and splice length of rebars. Each of these three methods is described below.



Rebar Position	Rebar Diameter Φ10 ~ Φ18	Rebar Diameter Φ20 ~ Φ32
BEAM TOP	72 db	89 db
BEAM BOT	55 db	69 db
Column - Wall	55 db	69 db

Rebar Position	Rebar Diameter Φ10 ~ Φ18	Rebar Diameter Φ20 ~ Φ32
BEAM TOP	66	84
BEAM BOT	50	65
Column - Wall	50	64

Rebar Position	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
Beam TOP	65	80	95	105	120	160	175	200	225	255
Beam BOT	50	60	70	80	90	125	135	155	175	200
Column - Wall	50	60	70	80	90	125	135	155	175	200

f_y 4000 kgf/cm² ψ_t 1.3 Top ψ_s for d>=20 1

f'_c 210 kgf/cm² ψ_t 1 Bot ψ_s for d<20 0.8

ψ_g 1 Grade ψ_e 1 Epoxy $\frac{c_b+K_{tr}}{d_b}$ = 1.5

λ = 1

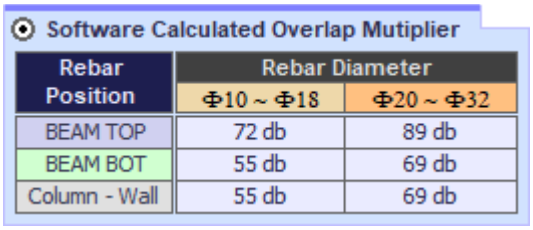
$$l_d = \frac{f_y}{1.1\lambda\sqrt{f'_c}} \frac{\psi_t\psi_e\psi_s\psi_g}{\left(\frac{c_b+K_{tr}}{d_b}\right)} d_b$$

Rebar Position	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
Beam TOP	70	80	95	110	120	170	185	210	240	270
Beam BOT	50	60	70	80	90	130	145	165	185	210
Column - Wall	50	60	70	80	90	130	145	160	180	205

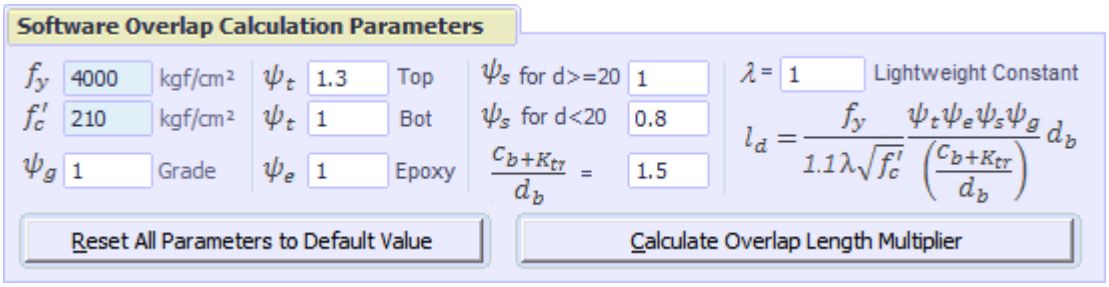
In this user interface, you are presented with three different options for the method of calculating the development and splice lengths:

1 Software Calculated Overlap Multiplier

In this mode, the software automatically calculates the development length and splice length factors for various rebar sizes and positions in accordance with ACI 318 provisions, based on the compressive strength of the concrete and the yield strength of the project rebars. To use this mode, activate the **Software Calculated Overlap Multiplier** option, as shown in the image below.

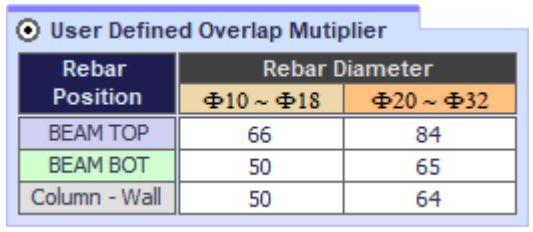


In this mode, the software calculates the development length and splice length factors for various rebar sizes and positions based on the formula shown in the image below, using the user-configurable parameters in the **Software Overlap Calculation Parameters** section, as well as the compressive strength of the concrete and the yield strength of the project rebars. These factors are then employed in the calculations and detailing of the project.



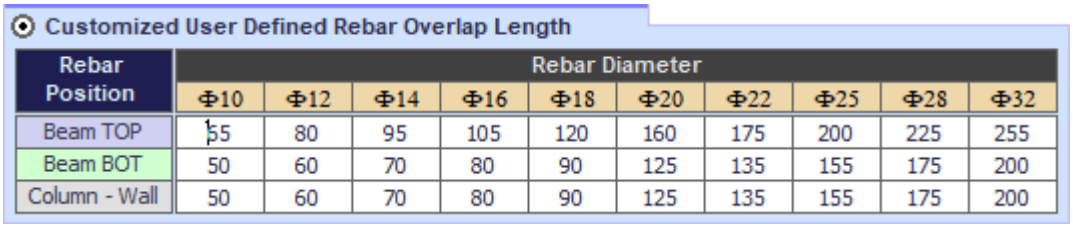
2 User Defined Overlap Multiplier

In this mode, the software calculates the development length and splice length based on factors specified by the user for determining the splice length. To use this mode, first activate the **User Defined Overlap Multiplier** option, as shown in the image below, and then specify the splice length calculation factors using six parameters for different diameters and positions.



③ Customized User Defined Overlap Length

In this mode, users can directly specify their desired splice length individually for each rebar size and for various structural positions of the rebar in the corresponding table. To use this mode, first activate the **Customized User Defined Overlap Length** option, as shown in the image below, and then edit the splice length of the rebars for each individual rebar diameter at various possible positions in the corresponding table, as illustrated in the image below.



After selecting one of the three methods described above for calculating the development length, and after performing the relevant settings, press the **Apply Changes and Rebuild Overlap Length Table** button to apply the changes. In addition, the splice length table for all rebar sizes and various rebar positions will be updated according to the selected splice method, which will then be visible at the bottom of the aforementioned window, as shown in the image above.

Two additional parameters are present in this window, which are described below:

Overlap Length / Anchor Length Ratio:
This parameter allows the user to specify the ratio of splice length to development length. By default, and in accordance with the code, this parameter is set to 1.3. In other words, the software also obtains the development length of the rebars by dividing the splice length by this number, i.e., 1.3.

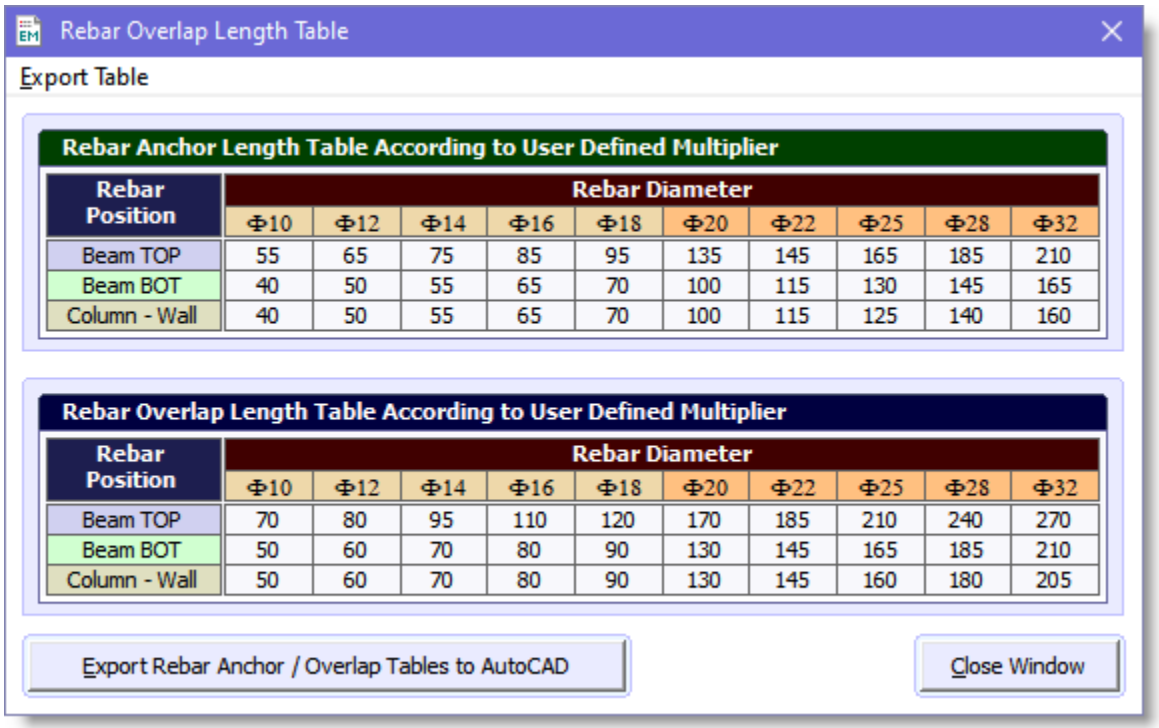
Rounding Step for Rebar Overlap Length:
This parameter allows the user to determine the rounding increment for the splice and development lengths of the rebars. By default, this parameter is set to 5 centimeters, meaning that the splice and development lengths will always be multiples of five.

It is worth noting that the development length and splice length tables for the rebars of structural elements can always be viewed via the **Report** menu and the **Rebar Overlap and Anchor Length** option, and can also be exported to AutoCAD.

Viewing the Splice and Development Length Tables of Rebars

The development length and splice length tables for the rebars of various structural elements can always be viewed via the **Report** menu and then the **Rebar Overlap and Anchor Length** option, as shown below.

Report Menu > Rebar Overlap and Anchor length

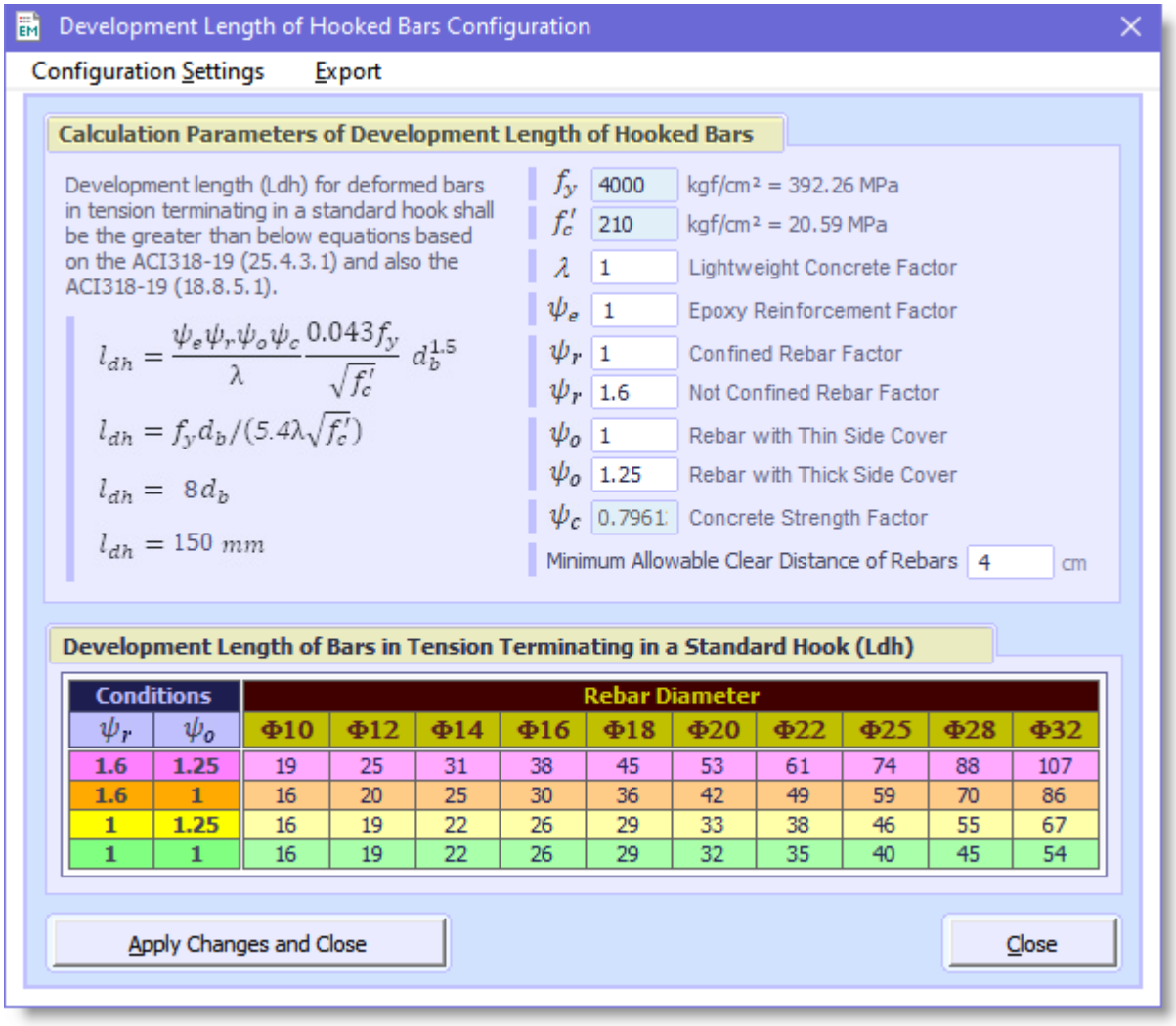


Additionally, if necessary, you can export these tables as an AutoCAD file using the **Export Rebar Anchor / Overlap Table to AutoCAD** button or the **Export Table** menu.

Hooked Bar Development Length Configuration

Define Menu > Development Length of Hooked Bars Parameters

This user interface allows you to configure all parameters governing the calculation of the development length of hooked bars in end beams. As shown in the image, all of these parameters are configured by default based on ACI 318-19, and normally no changes are required. However, users may modify these parameters according to their own requirements if necessary, and the software will calculate the development length of the hooked bars based on the relationships and parameters defined in this user interface. It is worth noting that these settings can be saved and restored via the **Configuration Setting** menu. Furthermore, the software provides an exportable table containing the development lengths of hooked bars based on rebar size and various conditions, as shown in the image below. Using the **Export** menu at the top of this user interface, you can also export this table as an AutoCAD file.

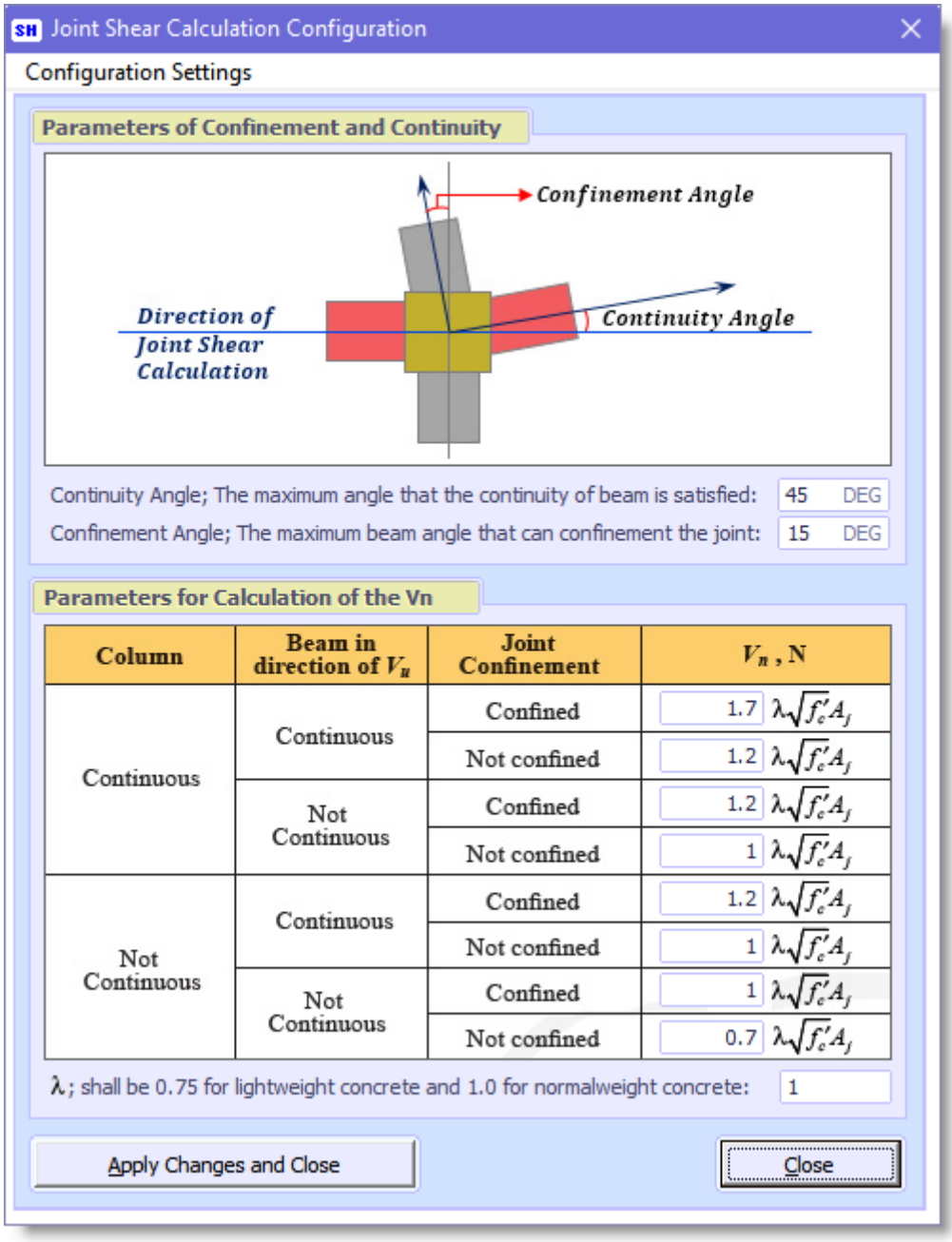


Conditions		Rebar Diameter									
ψ_r	ψ_o	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
1.6	1.25	19	25	31	38	45	53	61	74	88	107
1.6	1	16	20	25	30	36	42	49	59	70	86
1	1.25	16	19	22	26	29	33	38	46	55	67
1	1	16	19	22	26	29	32	35	40	45	54

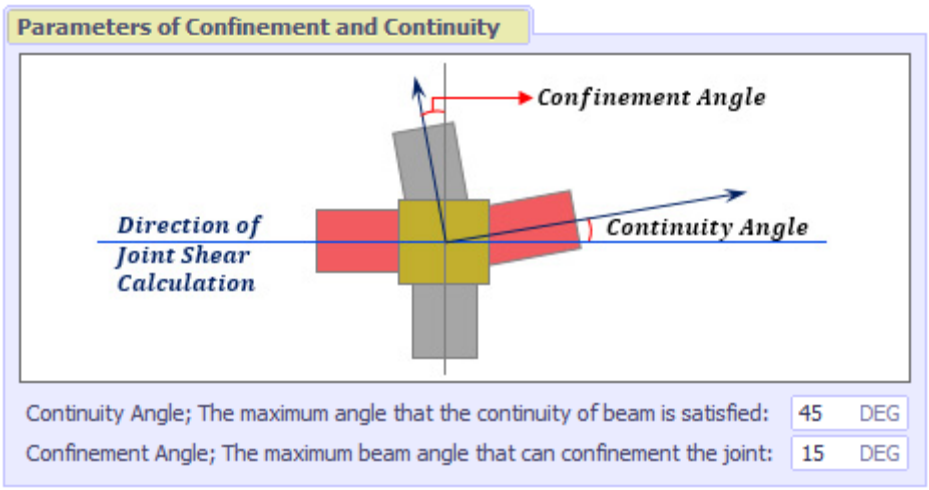
Joint Shear Calculation Parameters

Define Menu > Joint Shear Calculation Parameters

By using the user interface of **Joint Shear Calculation Configuration**, as shown in the image below, you can configure the parameters governing the calculation of the joint shear ratio. These parameters include the maximum continuity angle of the beam in the direction for which the joint shear is being calculated, the maximum confinement angle provided by beams perpendicular to the direction of joint shear calculation, and the coefficients for calculating V_n under various joint conditions, which are further described below.



As shown in the image below, the upper section of the form contains the continuity and confinement parameters, which are described below:



Continuity Angle:

This parameter defines the maximum angle of beams relative to the direction of joint shear calculation for a beam entering the joint to be considered continuous. In other words, if the angle of any beam entering the joint is less than this angle, the software will assume that beam to satisfy the continuity condition. This parameter is set to 45 degrees in the software, and users can easily adjust it to their desired value.

Confinement Angle:

This parameter defines the maximum angle of beams relative to the direction perpendicular to the direction of joint shear calculation for the possibility of providing confinement to the joint by the beams entering the joint. In other words, if the angle of any beam entering the joint is less than this parameter, the software will assume that beam to satisfy the confinement condition for the joint. This parameter is set to 15 degrees by default in the software, and users can easily adjust it to their desired value.

Note that the values of these parameters are not specified in any reference, and users must set them to their desired values based on their own engineering judgment.

As shown in the image below, the lower section of the **Joint Shear Calculation Configuration** interface contains the parameters for calculating V_n under various joint conditions.

Since the coefficients for calculating V_n have undergone changes in code revisions, these coefficients have been made user-editable, allowing the user to easily configure the method of joint shear calculation. By default, these parameters are set based on ACI 318-19, as illustrated in the image below.

Parameters for Calculation of the Vn

Column	Beam in direction of V_u	Joint Confinement	V_n, N
Continuous	Continuous	Confined	1.7 $\lambda\sqrt{f'_c}A_j$
		Not confined	1.2 $\lambda\sqrt{f'_c}A_j$
	Not Continuous	Confined	1.2 $\lambda\sqrt{f'_c}A_j$
		Not confined	1 $\lambda\sqrt{f'_c}A_j$
Not Continuous	Continuous	Confined	1.2 $\lambda\sqrt{f'_c}A_j$
		Not confined	1 $\lambda\sqrt{f'_c}A_j$
	Not Continuous	Confined	1 $\lambda\sqrt{f'_c}A_j$
		Not confined	0.7 $\lambda\sqrt{f'_c}A_j$

λ; shall be 0.75 for lightweight concrete and 1.0 for normalweight concrete: 1

The λ parameter, which pertains to lightweight concrete, is also editable at the end of this section. As shown in the image below, the λ parameter is set to 1 by default, which corresponds to normal-weight concrete.

Using the **Configuration Setting** menu at the top of the **Joint Shear Calculation Configuration** interface, the user can save all parameters in this interface after applying the desired changes. This allows the software to automatically use the user's customized parameters in subsequent sessions, eliminating the need to reconfigure these parameters for future projects.

A detailed explanation of the calculation methodology, as well as the specific settings and control of the joint shear ratio by **ETABS MATE**, is provided in the joint shear control section.

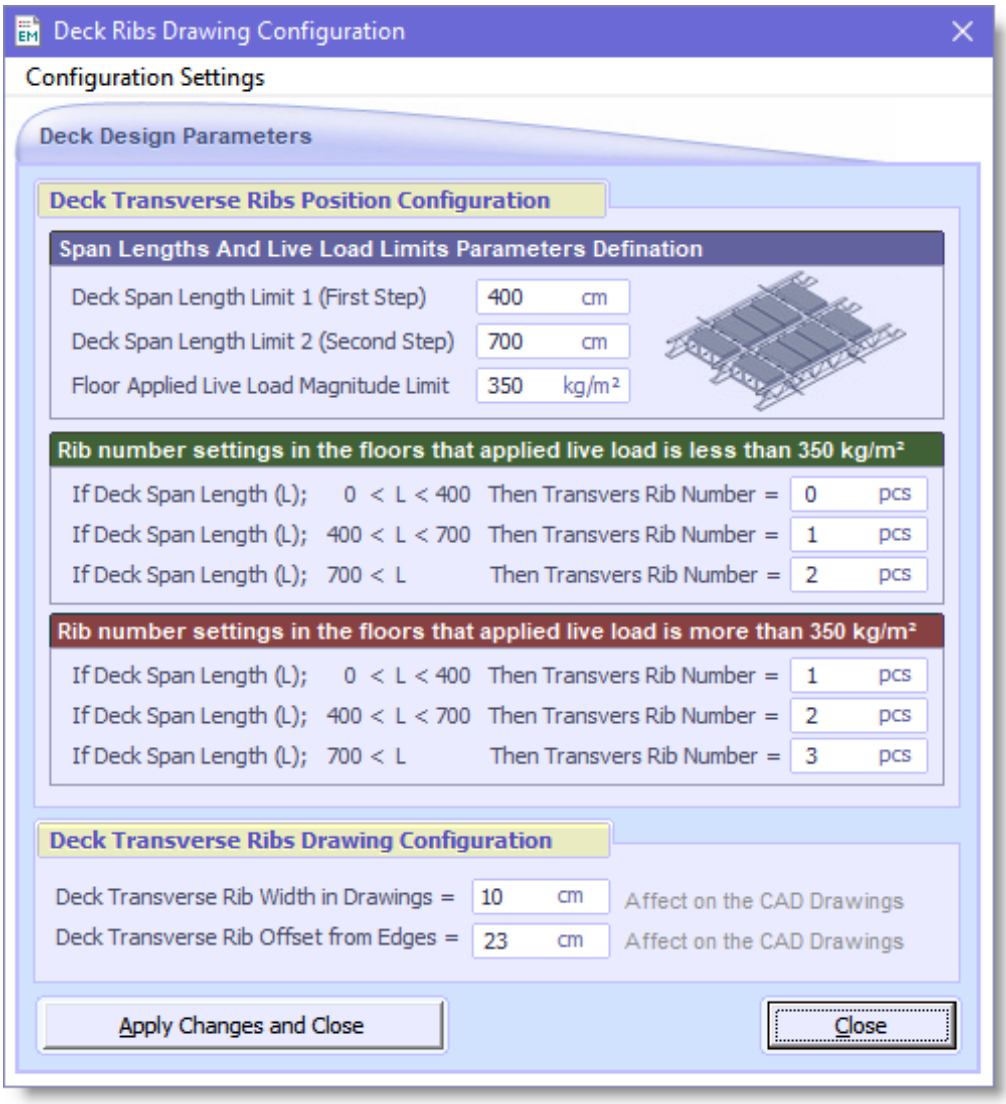
Deck Ribs Drawings Configuration

Define Menu > Deck Ribs Drawings Parameters

This option allows you to define the parameters governing the calculation of the number of ribs in joist and block floors. The settings of this user interface are performed based on three different span length ranges:

- Floor span less than First Step (default: 400 cm)
- Floor span between First Step and Second Step (default: 400 to 700 cm)
- Floor span greater than Second Step (default: 700 cm)

The number of ribs for each floor is also calculated based on the live load magnitude, specified as the **Floor Applied Load Magnitude Limit**, which is set here to 350 kg/m² and is adjustable in the first section of this user interface. Note that floor loads are only read by the software if the dead and live loads have been entered individually in the ETABS software and a Load Pattern has not been used. In the second and third sections of this user interface, you can specify the number of floor ribs for each of the conditions mentioned above.



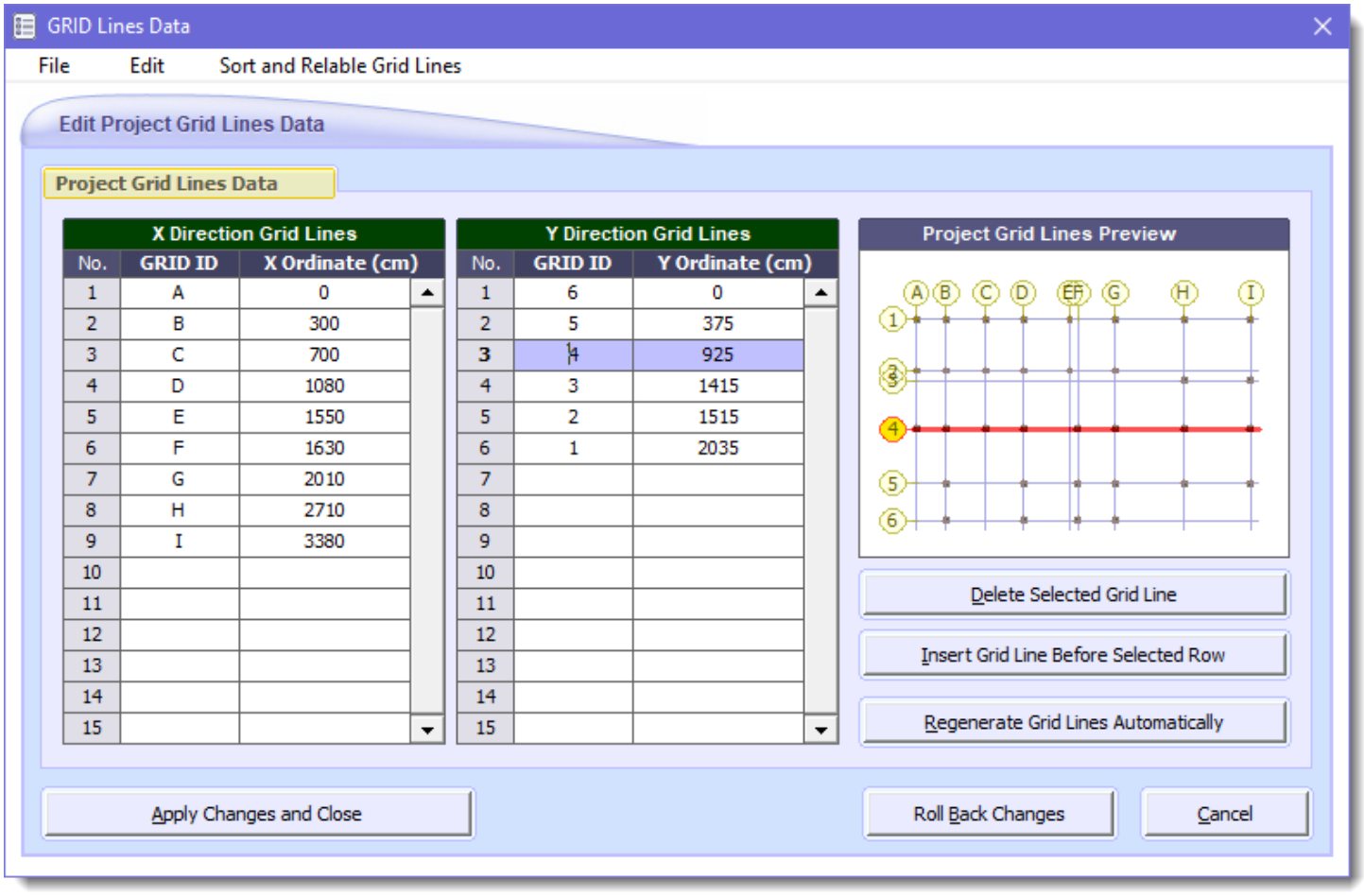
ETABS MATE
Concrete Structure Detailing Software

Grid Lines Configuration

Define Menu > Grid Lines of Project (Ctrl + F12 **)**

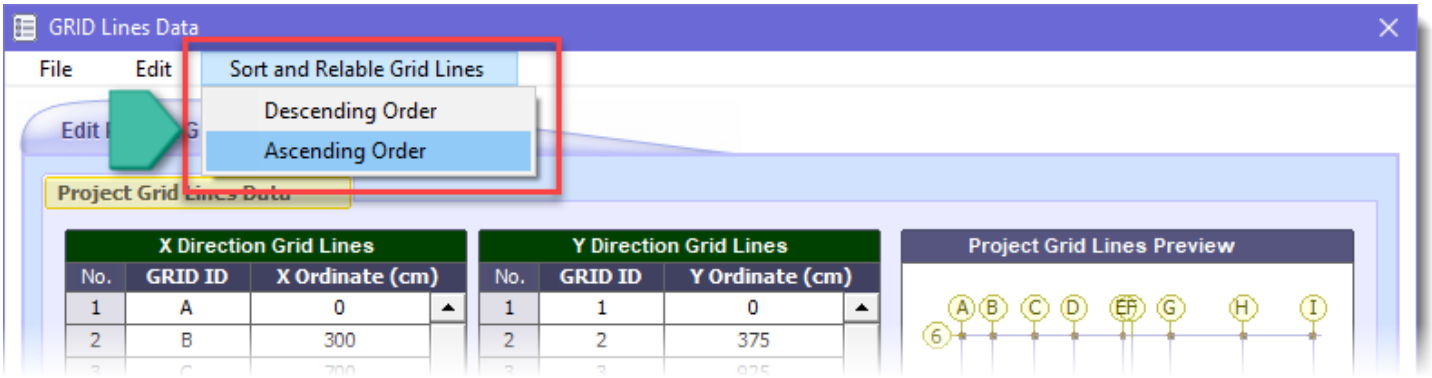
The project grid lines information interface allows you to view the information of the grid lines defined in the structural model and, if necessary, edit this information, including the names or coordinates of the project grid lines.

In this window, in addition to editing the names or coordinates of the grid lines, the ability to delete or add new rows is also provided. To do this, use the **Delete Selected Grid Line** button to remove the selected row, or the **Insert Grid Line Before Selected Row** button to add a row before the selected row. Another very useful tool has also been designed in this interface, allowing you to quickly and automatically define all project grid lines in an organized manner, ensuring no grid line is omitted. For this purpose, use the **Regenerate Grid Lines Automatically** button to delete all previous project grid lines and then generate and name new grid lines based on the coordinates of the columns of the lowest story of the project in both the horizontal and vertical directions.



This interface also provides the capability for the user to automatically organize the project grid lines in terms of coordinates and label names. To do this, from the **Sort and Relabel Grid Lines** menu, use the **Descending Order** option to sort the grid labels in descending order, or the **Ascending Order** option to sort the grid labels in ascending order.

Additional practical tools for convenient and comprehensive editing of the project grid line data are also available to users through the **Edit** menu in this interface.



Furthermore, this interface allows the user to save the modified grid line information as a text file, enabling them to update the ETABS e2k text file to reflect the changes made in **ETABS MATE**. To do this, after applying the necessary changes, use the **Save Text File for ETABS 9** option from the **File** menu to save the grid line information in a format compatible with the ETABS 9 text file. You may also use the **Save Text File for ETABS 16, 20, …** option from the **File** menu to save the grid line information in a format compatible with ETABS 16 and later text files, as shown in the below:

```
$ GRIDS
  GRIDSYSTEM "G1"  TYPE "CARTESIAN"  BUBBLESIZE 125
  GRID "G1"  LABEL "A"  DIR "X"  COORD 0 VISIBLE "Yes"  BUBBLELOC "End"
  GRID "G1"  LABEL "B"  DIR "X"  COORD 300 VISIBLE "Yes"  BUBBLELOC "End"
  GRID "G1"  LABEL "C"  DIR "X"  COORD 700 VISIBLE "Yes"  BUBBLELOC "End"
  GRID "G1"  LABEL "D"  DIR "X"  COORD 1080 VISIBLE "Yes"  BUBBLELOC "End"
  GRID "G1"  LABEL "E"  DIR "X"  COORD 1550 VISIBLE "Yes"  BUBBLELOC "End
   ....
```

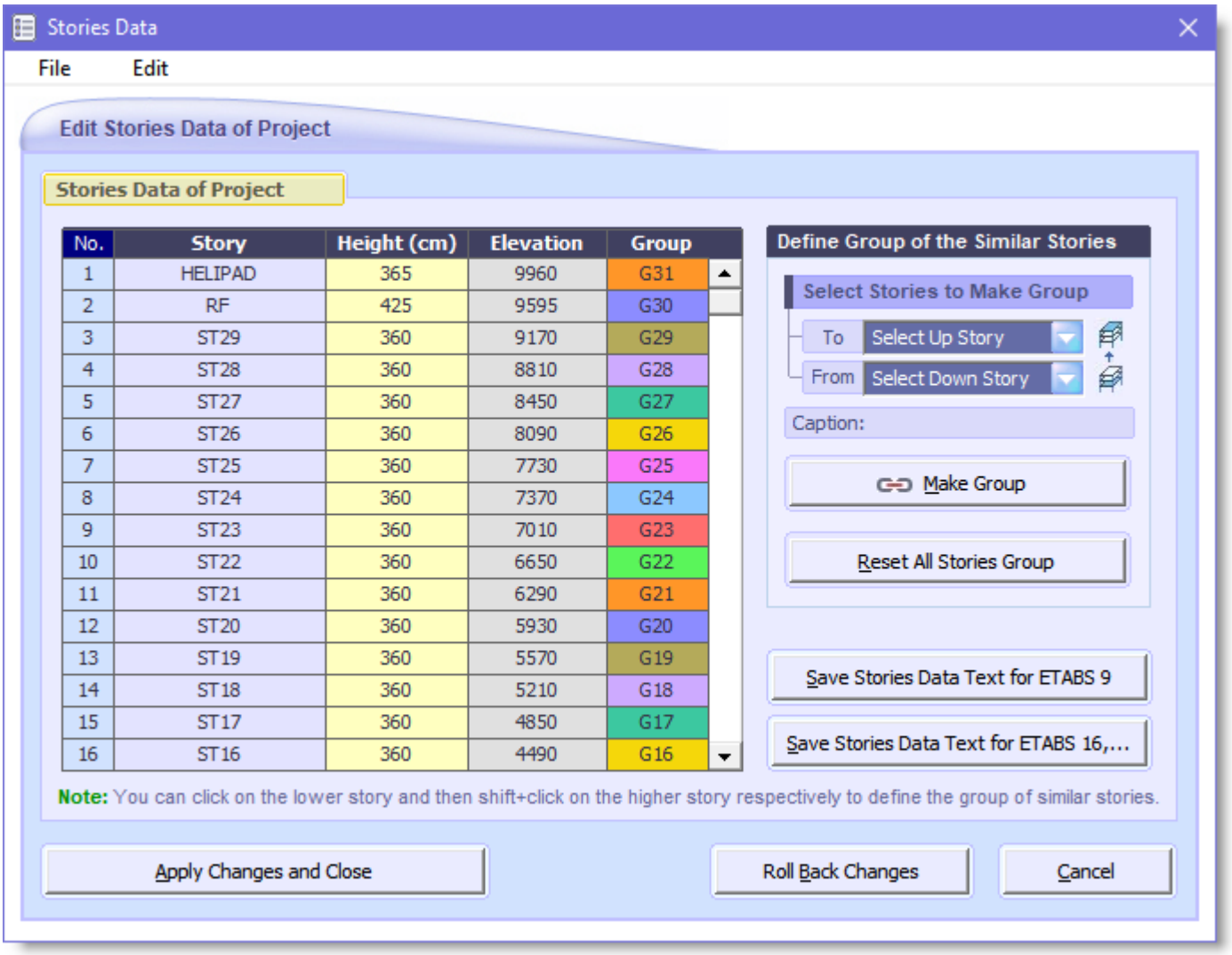
At the bottom of this interface, pressing the **Apply Changes and Close** button saves and applies the modifications made. The **Roll Back Changes** button allows you to discard the changes and revert to the initial settings.

ETABS MATE
Concrete Structure Detailing Software

Story Details Configuration

Define Menu > Stories Details of Project (F12)

The story information interface allows you to view the details of the project stories defined in the structural model and, if necessary, edit them. As shown in the image below, the editable information includes the story height and the base elevation of the structure.



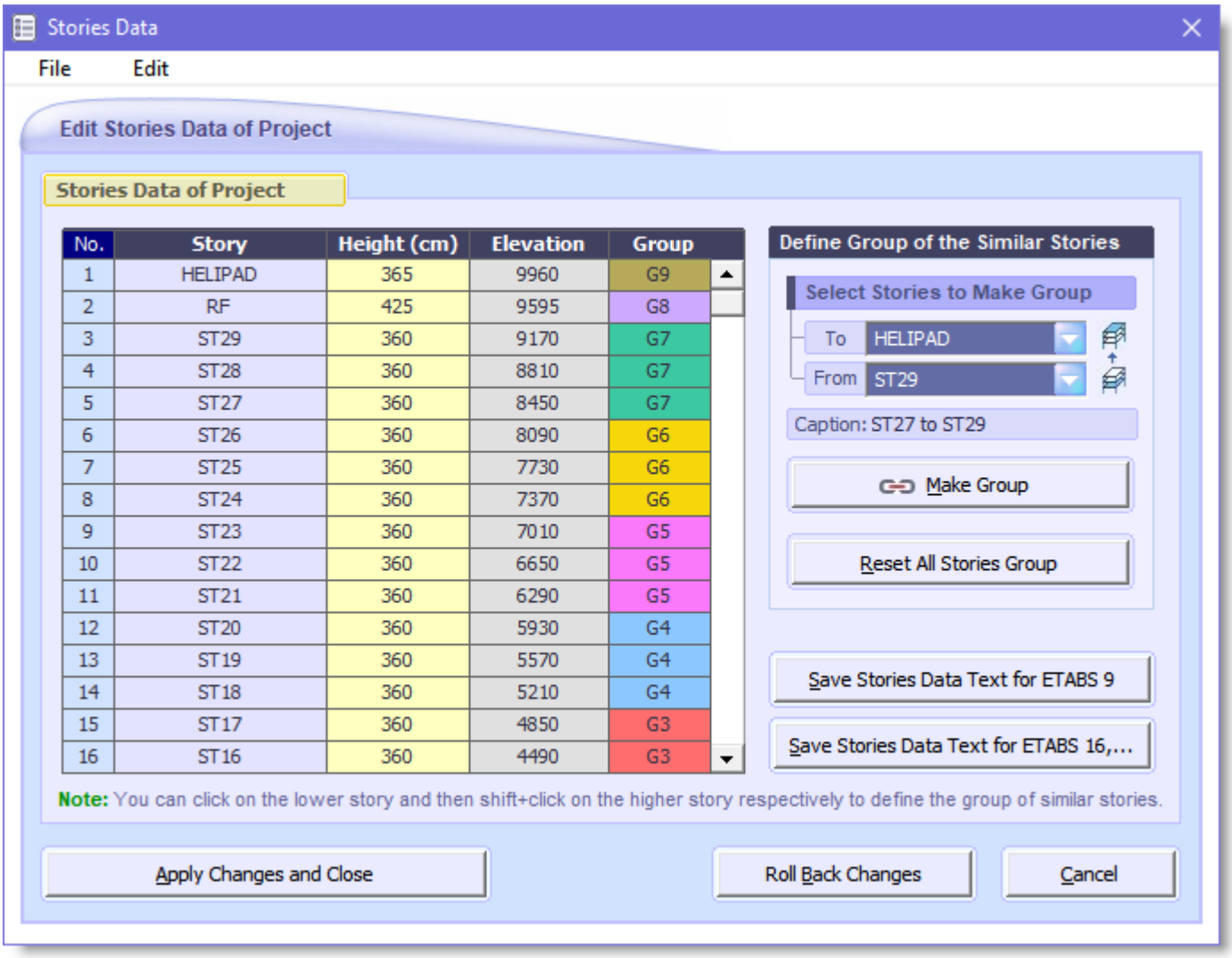
Please note that editing the story information will also be reflected in the structural drawings. Therefore, if modifications are made, the original model in the ETABS software must be corrected accordingly. As shown in the image above, this interface also includes tools for saving a text file compatible with the ETABS software and for grouping similar stories, which will be described in the following section.

ETABS MATE
Concrete Structure Detailing Software

Similar Story Groups Definition

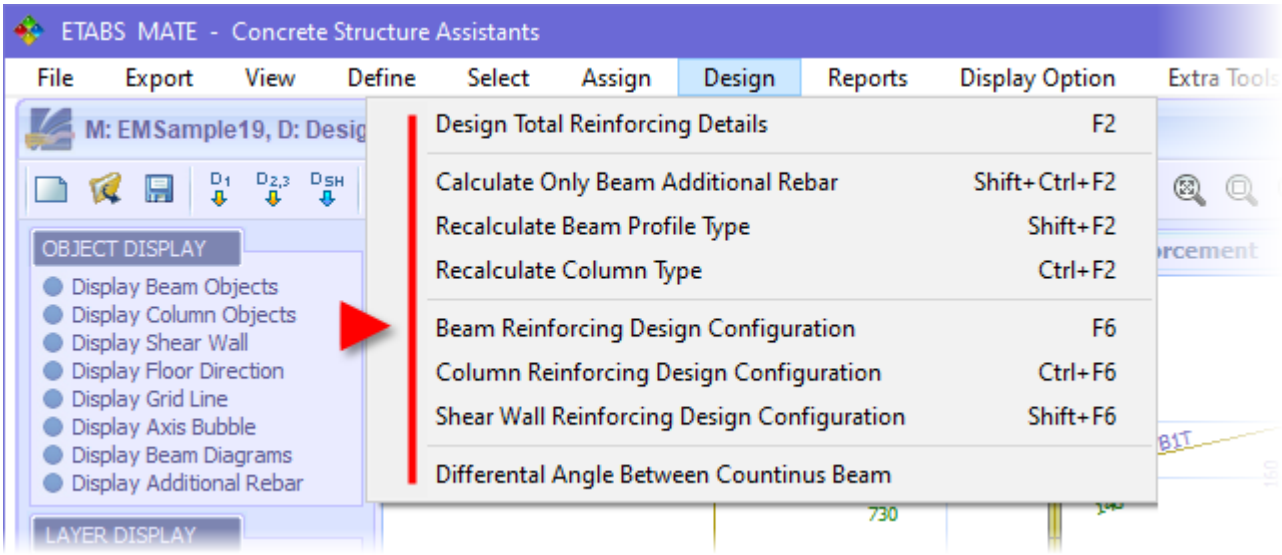
Defining similar story groups allows you to perform the element selection process across similar stories much more conveniently and quickly. To do this, in the project information interface, within the **Select Group of the Similar Stories** section, first select the lowest story of the group from the **From** dropdown list, or click on the name of the lowest story in the story list. Then, select the highest story of the group from the **To** dropdown list, or Shift+click on the name of the highest story in the story list to define the range of similar stories. Next, press the **Make Group** button to define the similar story group, as shown in the image below.

For ease of identification, the software names each group as **G1**, **G2**, **G3**, **...** and also displays each group with a distinct color.



Design Parameters Configuration

The parameters governing the reinforcement detailing of structural elements are set by default to the most optimal and conventional configurations. However, users can easily configure these parameters according to their preferences to fully customize the calculation methodology. If necessary, to configure the reinforcement design of various structural elements, you should use the options available in the **Design** menu.



As shown in the image below, the lower section of this menu pertains to the settings and configuration of parameters for the reinforcement detailing of various structural elements. These are briefly introduced here.

Beam Reinforcing Design Configuration (F6)

Configuring the design parameters governing the reinforcement detailing of beam elements.

Column Reinforcing Design Configuration (Ctrl + F6)

Configuring the design parameters governing the reinforcement detailing of column elements.

Shear Wall Reinforcing Design Configuration (Shift + F6)

Configuring the design parameters governing the reinforcement detailing of shear wall elements.

Differential Angle Between Continues Beam

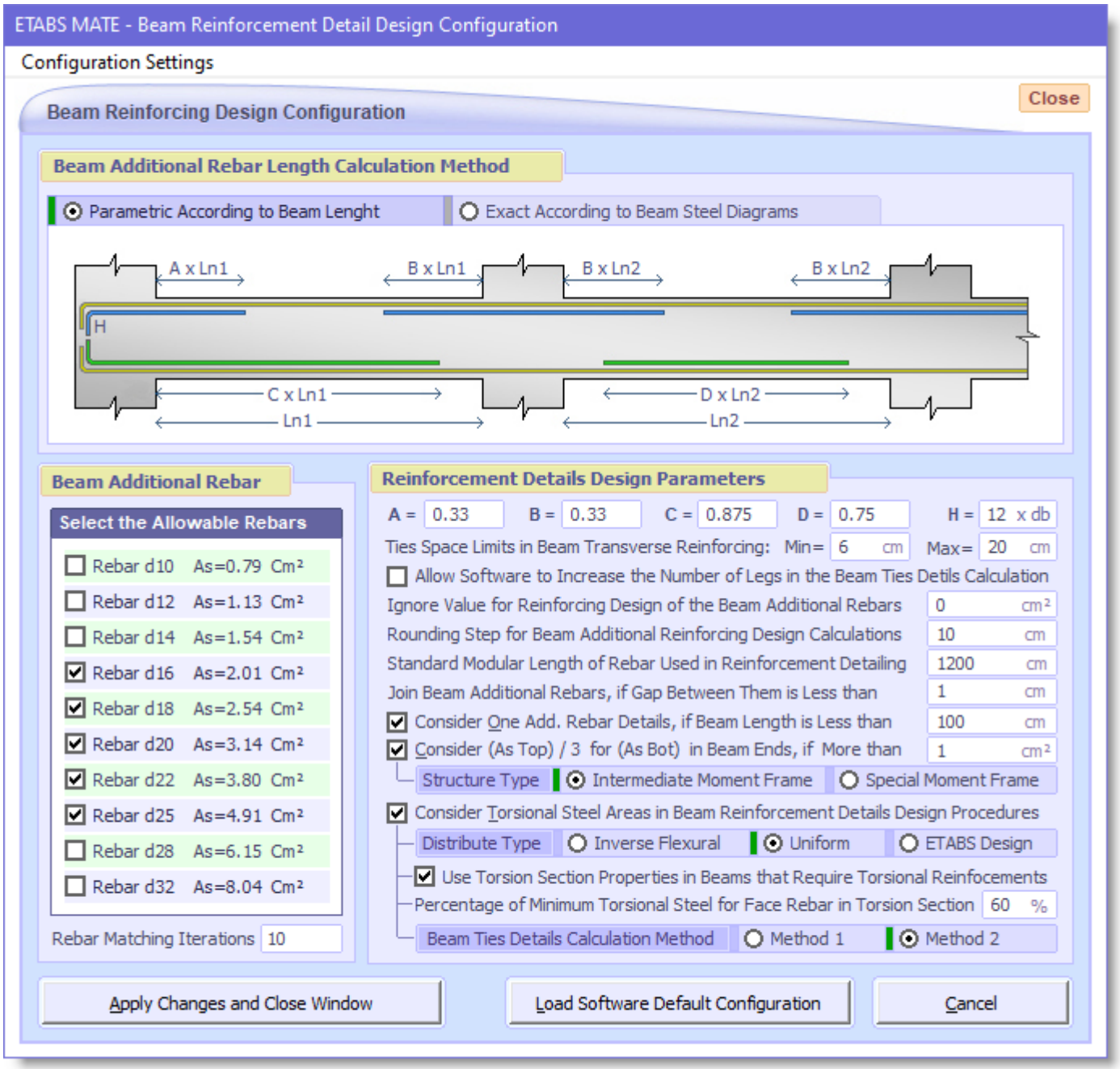
Settings for the conditions under which beams aligned consecutively are considered continuous.

All of these sections will be described in full detail in the following.

Beam Detailing Configuration

Here, the configuration of the software's design parameters for the reinforcement detailing of beams is explained. All of these parameters are set by default to the most optimal configuration. However, users can easily adjust these parameters according to their preferences to fully customize the software's design methodology. To access this user interface, proceed as follows using the **Design** menu. These parameters are described in the following section.

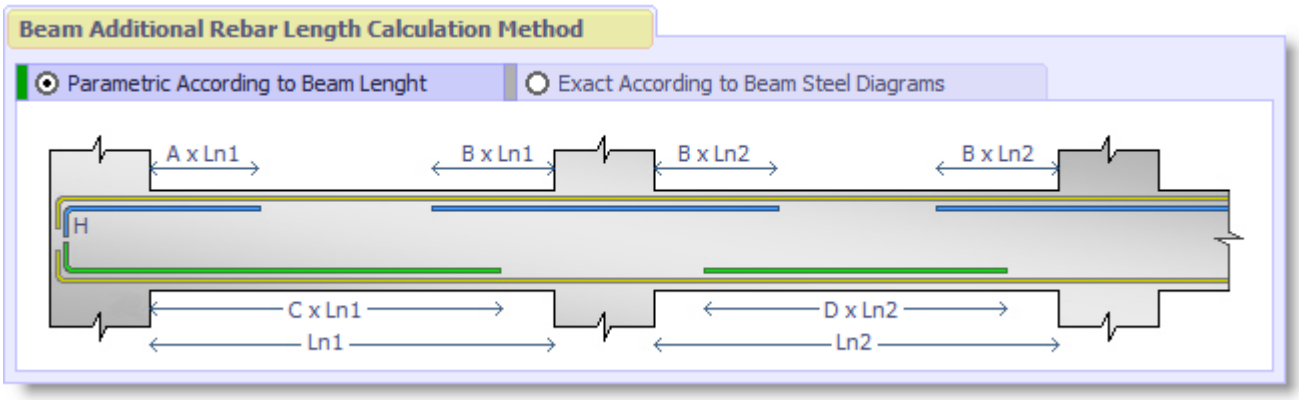
Design Menu > Beam Reinforcing Design Configuration (F6)



Beam Additional Rebar Length Calculation Methods

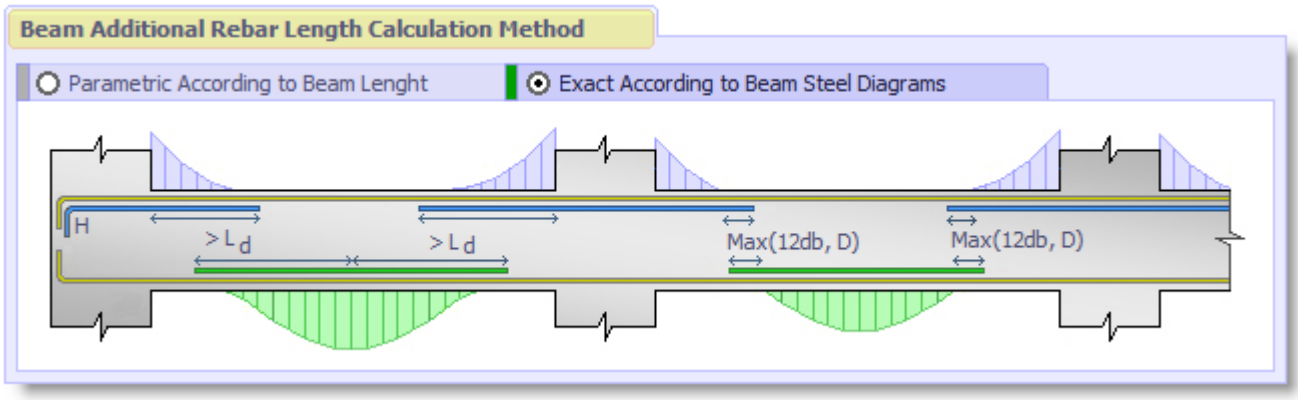
This option allows you to specify the method for calculating the length of additional rebars in beams. Two different methods have been coded for this purpose, which are described below:

1. Parametric According to Beam Length



In this method, as shown in the image above, the calculation of additional rebar lengths is performed based on fixed parameters, expressed as coefficients of the beam length, along with the relevant code provisions. This method is based on an approximate approach using a set of constant coefficients, resulting in more conservative lengths. The configuration of the coefficients used in this method is explained in detail in the following sections.

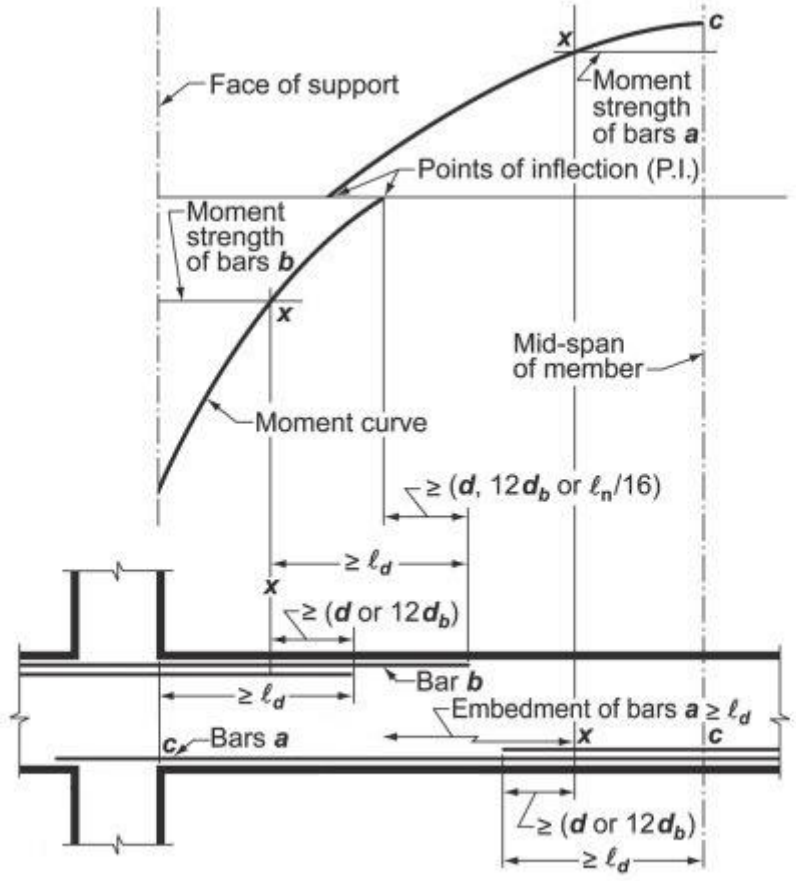
2. Exact According to Beam Steel Diagrams



In this method, the lengths of additional rebars are calculated precisely and automatically by the software, based on determining the theoretical and practical cutoff points of the rebars using the steel diagrams, in conjunction with the relevant code provisions. This method yields more economical additional rebar lengths. The calculation procedure and the related provisions are described in detail in the following sections.

In the **Exact According to Beam Steel Diagrams** method, the lengths of additional rebars are calculated precisely based on the actual requirements of the beam by determining the theoretical cutoff point and subsequently the practical cutoff point using the steel diagrams, in accordance with the provisions illustrated in the image below.

In this method, the flexural additional reinforcement area diagram is first obtained by subtracting the continuous reinforcement area from the flexural reinforcement area diagram. The longitudinal torsional reinforcement, after deducting the area of the side face rebars defined in the section, is then distributed over the flexural additional reinforcement diagram values to obtain the total additional reinforcement diagram. Using this diagram, the theoretical cutoff point of the rebar is calculated by the software through interpolation between the diagram data. Subsequently, by extending the length from the theoretical cutoff point by the maximum of (beam depth ***d*** *and* ***12d_b***), the practical cutoff point, and consequently the additional rebar length, is automatically calculated by the software. In the next step, the software verifies that the calculated additional rebar length exceeds the required straight development length from the column face for the end additional rebars of the beam, or twice the development length for the mid-span additional rebars, thereby determining the precise length of the additional rebars.

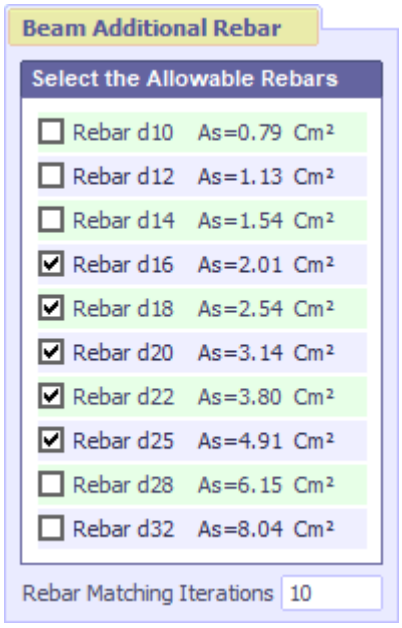


Development of flexural reinforcement in a typical continuous beam.

ETABS MATE
Concrete Structure Detailing Software

Permissible Rebar Sizes for Additional Reinforcement

Beam Additional Rebar

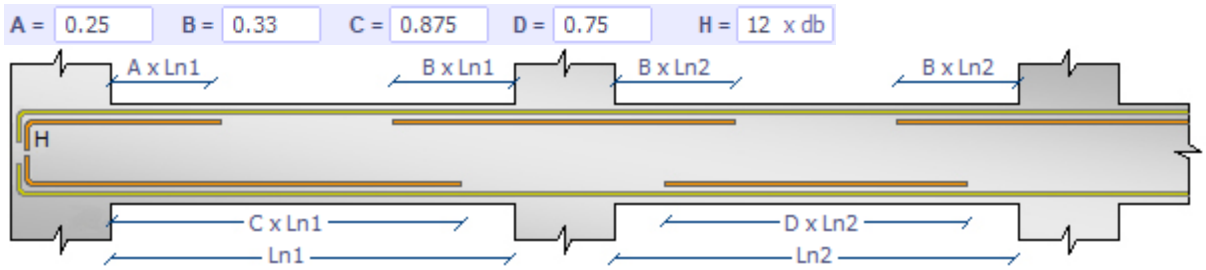


In this section, you can specify which rebar sizes the software is permitted to use for calculating the additional reinforcement. In many projects, to avoid an excessive variety of additional rebars, engineers prefer that only specific sizes be used for the additional reinforcement. To this end, you can select, from this list, the rebar sizes that the software is allowed to use in the beam additional reinforcement design process.

A **Rebar Matching Iteration** parameter is also available here, which is used in the intelligent process of matching additional rebars on either side of a column or additional rebars that intersect one another. By default, this parameter is set to 10, which is suitable for structures with a maximum of 30 spans. However, for very large structures with more than 30 spans, this parameter should be increased to achieve the desired result.

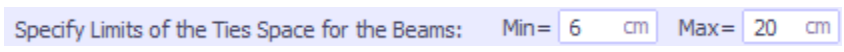
General Settings for Beam Additional Reinforcement and Ties

A, B, C, D and H Parameters



If you have selected the **Parametric According to Beam Length** method for calculating the additional rebar lengths, you can easily configure the parameters related to the coefficients for calculating additional rebar lengths, namely **A, B, C, and D**, as identified in the image, in this section. All of these parameters are set by default based on ACI recommendations. The **H** parameter represents the hook length coefficient for rebars anchored with a 90˚bend.

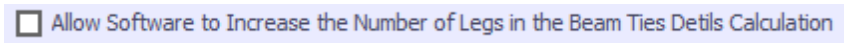
Specify Limits of the Ties Space for the Beams



In this section, you can specify the permissible limits for the minimum and maximum tie spacing in beams. For instance, the minimum permissible tie spacing is set to 6 cm, meaning this is the extent to which the software can reduce the tie spacing. If, even after reducing the spacing to this limit, the required transverse reinforcement quantity cannot be provided, the software will be compelled to increase the tie diameter to satisfy the required transverse reinforcement quantity.

Allow Software to Increase the Number of Legs in the Beam Ties Details Calculation

By activating this option, you allow the software to increase the number of crossties in the beam section as much as necessary, to satisfy the required transverse reinforcement of the beam.



When this option is enabled, the software's procedure for providing the transverse reinforcement calculated by ETABS is as follows: first, it reduces the tie spacing, then increases the number of crossties, and finally, if the required reinforcement is still not satisfied, increases the tie diameter.

Ignore Value for Reinforcing Design of the Beam Additional Rebars

In some cases, the area of the additional rebars is slightly less than the reinforcement quantity calculated by ETABS (e.g., 0.1 cm^2). In such situations, some engineers prefer to ignore these small deficiencies to avoid increasing the number or diameter of the additional rebars. This parameter allows you to specify this tolerance.

The software default for this parameter is zero, meaning the additional reinforcement area must not be less than the calculated reinforcement area. For example, a value of 0.1 means a maximum tolerance of 0.1 cm^2.

Rounding Step for Beam Additional Reinforcing Design Calculations

This parameter allows you to specify the rounding increment for rebar lengths. By default, this parameter is set to 10 cm, meaning the lengths reported by the software will always be multiples of 10.

Note that, for conservatism, length rounding is always performed upward. For example, lengths of 102 cm and 107 cm will both be rounded to 110 cm. However, the software incorporates intelligent algorithms that prevent the rounding process in situations where rounding the rebar length would cause the rebar to protrude from the corresponding element, or where overlapping of rebars is not feasible, ensuring the actual rebar length is reported.

Standard Modular Length of Rebar Used in Reinforcement Detailing

This parameter allows you to specify the standard modular length of rebar bars. By default, this parameter is set to 1200 cm.

Given that the standard rebar length may vary across different manufacturers or countries, the software provides the capability for the user to configure the modular length of the continuous beam rebars.

Join Beam additional Rebars, if Gap Between Them is Less than …

In this section, you can specify the conditions for joining additional rebars when their length is relatively large compared to the beam length. The default value for this parameter is 1 cm.

For example, if the gap between the start and end additional rebars in a beam is less than 1 cm, the software, for ease of reinforcement placement, joins them together and considers the additional rebar as continuous.

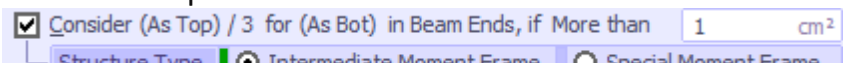
Consider One Add. Rebar Details, If Beam Length is Less than …

In this section, you can specify the conditions for joining additional rebars when the beam length is short. The default value for this parameter is 100 cm. This means that if a beam length is less than 100 cm, for instance in the case of cantilever beams, the software considers only one set of reinforcement details for the entire beam.

If this option is disabled, the software will perform the reinforcement detailing based on the calculation results without considering reinforcement congestion.

Consider (As TOP) / 3 for (As Bot) in Beam Ends, if More than …

In this section, you can require the software, in accordance with code provisions to prevent the formation of a plastic hinge, to place 1/3 (or 1/2 for special ductility) of the required top reinforcement at the supports as bottom reinforcement. This option is enabled by default, and the software automatically compares the bottom and top reinforcement quantities for each beam. If the bottom reinforcement quantity is less than one-third (or one-half) of the top reinforcement, the software will consider one-third (or one-half) of the top reinforcement as the bottom reinforcement at the supports. If you do not wish to apply this requirement, you may disable this option.



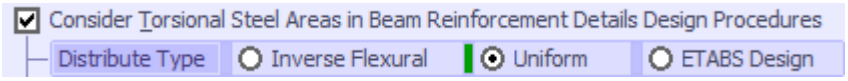
Furthermore, since the earthquake resistance type of the structure is important in how this parameter is applied, as shown in the images, you can also set the **Structure Type** to Intermediate or Special in this section.

Additionally, in this section, you can define the threshold for applying this quantity to the bottom of the beam. By default, this value is set to 1 cm², meaning that if the quantity of reinforcement transferred from the top to the bottom of the beam exceeds 1 cm², this quantity will be used in the beam reinforcement detailing; otherwise, it will not be incorporated into the detailing. If you wish to disregard this threshold and strictly follow the code, set this value to zero cm².

Torsional Reinforcement Settings

Consider Torsional Steel Area in Beam Reinforcing Design Procedures

In this section, you can specify whether the longitudinal and transverse reinforcement due to torsion, calculated by ETABS, should be applied in the additional rebar calculation and beam tie design processes.



When this option is enabled, you can use three different methods in the **Distribute Type** section for distributing the longitudinal torsional reinforcement between the top and bottom flexural reinforcement of the beam, which are described below:

1) Inverse Flexural

In this method, the excess longitudinal torsional reinforcement (total longitudinal torsional reinforcement minus the reinforcement provided in the beam side faces) is distributed between the top and bottom additional reinforcement in inverse proportion to the flexural reinforcement accumulation at the top and bottom of the beam. For example, if the flexural reinforcement at the top of the beam is greater than at the bottom, a larger share of the torsional reinforcement will be allocated to the bottom of the beam. This distribution method results in a more uniform distribution of the total required flexural and torsional reinforcement area across the entire section.

2) Uniform

In this method, the excess longitudinal torsional reinforcement is added equally to the additional flexural reinforcement at the top and bottom of the beam section. Note that the excess longitudinal torsional reinforcement is obtained by subtracting the area of the side face rebars (if defined in the beam section) from the longitudinal torsional reinforcement. Therefore, by defining side face rebars in the section for beams subjected to torsion, a uniform distribution of torsional reinforcement across all faces of the beam can be achieved. This option is enabled by default.

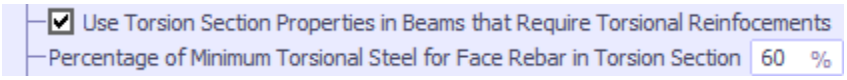
3) ETABS Design

In ETABS version 20.3 and later, a new report titled **Total Longitudinal Reinforcing** has been added to the Design Output section. In this report, ETABS provides the maximum combined longitudinal reinforcement due to flexure and torsion for the beams across all design load combinations. In some cases, these values are less than the sum of the maximum longitudinal flexural reinforcement and the maximum longitudinal torsional reinforcement.

If ETABS version 20.3 or later is used for modeling, this option becomes available and can be used for calculating the additional reinforcement. In this mode, the additional reinforcement is calculated directly based on the aforementioned report, which ETABS provides for the total longitudinal reinforcement, representing the combined longitudinal reinforcement due to flexure and torsion in the beams.

Use Torsion Section Properties for Beam that Require Torsion Reinforcement

This option allows you to enable or disable the use of the torsional version of the section for detailing.



As explained in the beam section definition section, the software automatically defines **General** and **Torsion** versions for all beam sections. When this option is enabled, the program's procedure for beam detailing is as follows: if the beam does not require torsional reinforcement, the software uses the properties defined in the **General** version of the section for beam detailing. If the beam requires torsional reinforcement, it uses the properties defined in the **Torsion** version of the section.

By disabling this parameter, however, the software will no longer use the **Torsion** version of the sections and will employ the details defined in the **General** version for the detailing of all project beams, regardless of whether they require torsional reinforcement

Percentage of Minimum Torsional Steel for Face Rebar in Torsion Section

The side face rebars automatically considered by the software for the torsional version of beam sections are calculated as a percentage of the minimum torsional reinforcement of the section. When the above option for using the torsional version of sections is enabled, this parameter also becomes active, allowing you to specify what percentage of the minimum torsional reinforcement of the section should be considered as side face rebars in the beam section.

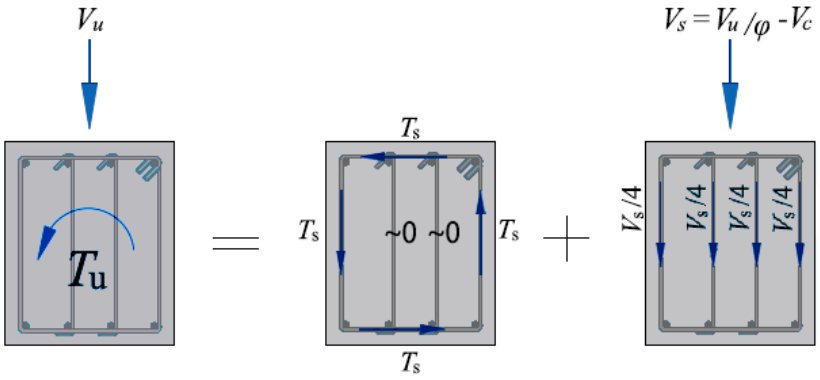
Note that increasing the value of this parameter causes the software to consider a greater amount of side face rebars by default for beam sections. This results in a larger portion of the torsional reinforcement being accommodated in the beam's side faces, thereby requiring a smaller quantity for the additional top and bottom rebars. This can lead to better detailing and a reduction in the joint shear ratio. On the other hand, since it is preferable for torsional reinforcement to be distributed uniformly across all four faces of the section, this parameter cannot be set to an excessively large value. Therefore, considering the project characteristics, the user should determine the optimal value for this parameter to achieve an efficient design.

Beam Ties Details Calculation Method

Two different methods, designated as **Method 1** and **Method 2**, have been coded in the software for calculating beam transverse reinforcement details. These are shown in the below.

This option allows you to select one of these methods for calculating the transverse reinforcement details of beams based on your engineering judgment, which are described below.

In a section subjected to the combined effects of shear force and torsional moment, as shown in the figure below, only the peripheral legs of the closed tie are capable of resisting the torsional moment, and the contribution of the internal legs can be neglected in the torsional transverse reinforcement calculations. However, for shear force resistance, all legs are capable of carrying the shear force.



Accordingly, two different methods have been provided for calculating the ties, which are described below:

1) Method 1:

In the first method, the transverse reinforcement calculation for beams subjected to the combined effects of torsional moment and shear force is performed based on the maximum value obtained from the following relationships:

Total legs of transverse reinforcement, including crossties $> \frac{A_v}{s} + 2\frac{A_t}{s}$

Total legs of closed transverse reinforcement, excluding crossties $> 2\frac{A_t}{s}$

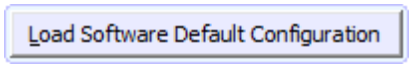
2) Method 2:

In the second method, assuming the diameter of the closed tie and the crossties to be identical, the following formula is used to calculate the transverse reinforcement details of sections subjected to the combined effects of torsional moment and shear force:

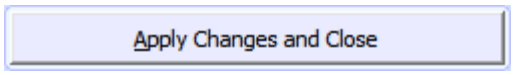
$$\frac{A_b}{s} = \frac{1}{n}\frac{A_v}{s} + \frac{A_t}{s}$$

Where, in these relationships: Av is the transverse reinforcement due to shear, At is the transverse reinforcement due to torsion, Ab is the area of one tie leg, s is the spacing between ties, n is the total number of legs, including the legs of the closed tie and the crossties.

The following describes some of the buttons and menus available in this user interface:



This button updates all parameters in this interface to the program's default values.



This button saves the changes made in this interface. Afterward, the reinforcement design and structural detailing must be re-executed for the changes to take effect in the structural detailing.

Also through the **Configuration Setting** menu at the top of this interface, you can save or restore the settings configured here. The items available in this menu are described below:

Configuration Setting > Save as User Default Configuration

If you wish to save the settings configured in this interface as the program's default settings, use the **Configuration Setting** menu and the **Save as User Default Configuration** option. Note that once these settings are saved as the default configuration, the software will automatically load them in subsequent sessions and use these user-saved settings instead of its original defaults.

Configuration Setting > Delete User Default Configuration

If you wish to delete the user default settings so that the software launches with its original default settings in subsequent sessions, use the **Configuration Setting** menu and the **Delete User Default Configuration** option to remove the customized settings, allowing the software to start with its factory default configuration.

Configuration Setting > Load Software Default Configuration

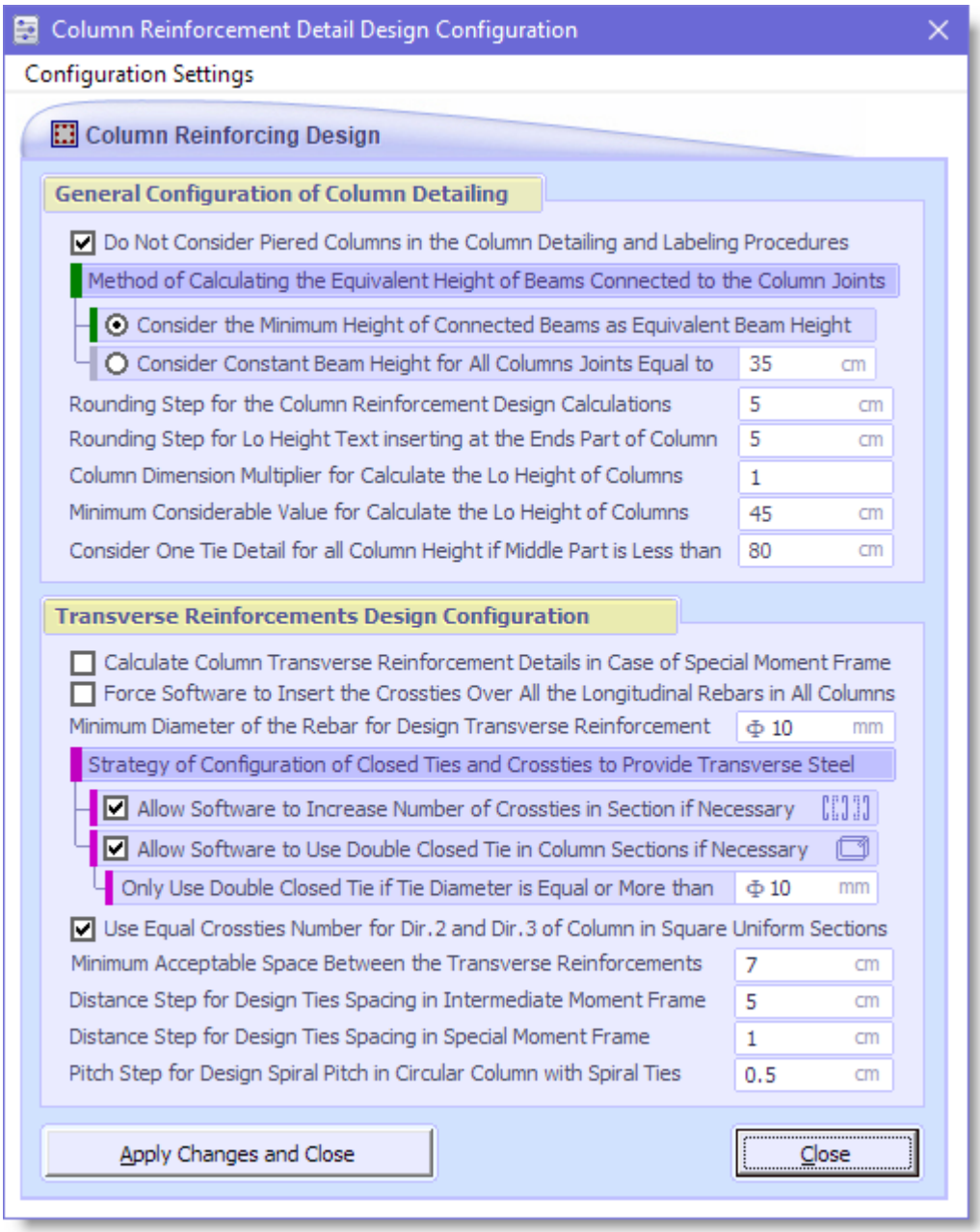
If you wish to reload the software's default settings, use the **Configuration Setting** menu and the **Load Software Default Configuration** option to restore the program's default settings.

ETABS MATE
Concrete Structure Detailing Software

Column Detailing Configuration

The following describes the configuration of the software's design parameters for the reinforcement detailing of columns. All of these parameters are set by default to the most optimal configuration. However, users can easily adjust these parameters according to their preferences to fully customize the software's design methodology. To access this user interface, proceed as follows using the **Design** menu. These parameters are described in the following section.

Design Menu > Column Reinforcing Design Configuration (Ctrl + F6)

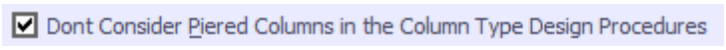


ETABS MATE
Concrete Structure Detailing Software

General Column Detailing Settings

Don't Consider Piered Column in the Column Type Design Procedures

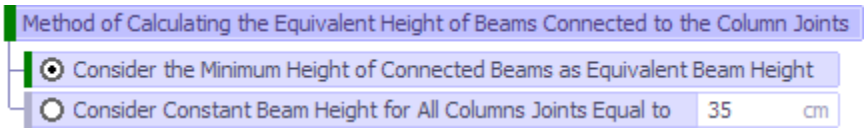
Columns located at the ends of shear walls are considered part of the shear walls, and it is preferable that they not be included in the structural column classification process. The software provides the capability for the user to specify, via this parameter, whether or not end columns of shear walls should be included in the column classification.



When this option is selected, the software will disregard **Piered Columns** in the column classification, and consequently, no label will be assigned to them in the column layout plan of the structural drawings. However, when this option is not selected, the software will classify the columns regardless of whether they are piered, and these columns will accordingly be labeled in the structural column layout plan. Note that if this parameter is not selected and the piered columns are classified, the software will account for the weight of these columns twice in the material quantity takeoff and rebar schedules: once in the column material calculation and once in the shear wall material calculation.

Method of Caculating the Equivalent Height of Beams Connected to the Column Joints

Since the parameter l_0, i.e., the length of the critical region of the column, must be calculated based on the clear height of the column, the equivalent height of the beams framing into the columns must first be calculated for each column type to determine the column clear height. For this purpose, as shown in the image below, two different methods are provided, either of which may be used at your discretion. These are described below.



1) Consider the Minimum Height of Connected Beams as Equivalent Beam Height

In this mode, the software first gathers the data of all beams connected to the ends of the columns belonging to a given column type at each story, and then considers the minimum height of these beams as the equivalent beam height at that story. Since this method yields a larger clear height, it can calculate the length of the critical region more conservatively and more closely aligned with the actual project conditions.

2) Consider Constant Beam Height for All Columns Joints Equal to …

This option allows you to specify a constant value for the beam height. In this mode, the software applies this constant value as the equivalent beam height across all stories and for all column types.

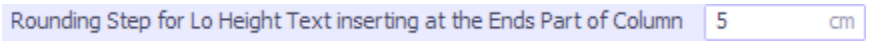
Rounding Step for the Column Reinforcement Design Calculations

This parameter allows you to specify the rounding increment for rebar lengths. By default, this parameter is set to 5 cm, meaning the lengths reported by the software will always be multiples of 5.

Note that, for conservatism, length rounding is always performed upward. However, the software incorporates intelligent algorithms that prevent the rounding process in situations where rounding the rebar length would cause the rebar to protrude from the corresponding element, or where overlapping of rebars is not feasible, ensuring the actual rebar length is reported.

Rounding Step for Lo Height Text inserting at the Ends Part of Column

This parameter allows you to specify the rounding increment for l_0, which is the length of the critical region of the column.



By default, this parameter is set to 5 cm, meaning the lengths reported by the software will always be multiples of 5. Note that, for conservatism, length rounding is always performed upward.

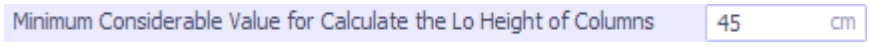
Column Dimension Multiplier for Calculate the Lo Height of Columns

This option allows you to apply a multiplier for calculating the l_0 length of columns. Since l_0 must not be less than the largest column dimension, you can also define the minimum l_0 based on a multiplier of the column dimension. By default, and in accordance with the code, this parameter is set to 1.

In other words, this parameter will be multiplied by the l_0 length that the software calculates based on the code.

Minimum Considerable Value for Calculate the Lo Height of Columns

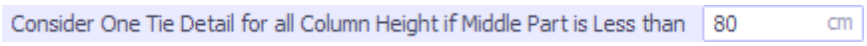
This option allows you to define a minimum value for the l_0 parameter, which is the height of the critical region of the column. By default, and in accordance with the code, this parameter is set to 45 cm.



Using this parameter, you can ensure that the length of the critical region of the column is never considered less than the value defined here.

Consider One Tie Detail for all Column Height if Middle Part is Less than ...

Typically, column ties are detailed differently for the end critical regions and the middle region. However, if the column height is short or the column dimensions are large, the end critical regions may become large, resulting in a very short middle region. In this case, only a few ties are placed in the middle region, which can cause the detail text annotations to overlap. Under these conditions, many engineers prefer that the entire column be tied uniformly. For this purpose, a parameter has been included in the design settings, allowing the user to specify that if the height of the middle region of the column falls below this parameter, the entire column height will be tied uniformly based on the details calculated for the critical region of the column.



As shown in the image below, this parameter can be defined through the column detailing configuration interface. By default, this parameter is set to 80 cm.

Column Transverse Reinforcement Calculation Settings

Calculate Column Transverse Reinforcement Details in Case of Special Moment Frame

When this option is enabled, the column tie details are calculated for special ductility. Otherwise, they are calculated for intermediate ductility. After changing this option, pressing the **Apply Changes** button recalculates the tie details for all column sections based on the selected ductility condition.

Note that changing this parameter causes the tie details of all column sections to be recalculated, and consequently, all user-defined modifications to the column section tie details will be reset to the program's default settings.

Force Software to Insert the Crossties Over All the Longitudinal Rebars in All Columns

By default, the software restrains alternate longitudinal column rebars with crossties. However, when this option is selected, the software will provide crossties for all longitudinal column rebars.

This option is disabled by default, allowing the software to calculate the most optimal configuration for the section transverse reinforcement based on the configured settings.

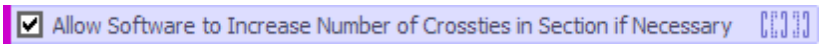
Minimum Diameter of the Rebar for Design Transverse Reinforcement

This option allows you to specify the minimum diameter for calculating the column transverse reinforcement. By default, and in accordance with the code, this parameter is set to 10 mm.

If you wish to change the minimum diameter of all ties, this parameter allows you to easily modify the minimum tie and crosstie diameter for all sections, without the need to edit each section individually. Upon changing this parameter, the software automatically recalculates the transverse reinforcement details of the columns for all sections based on the newly defined minimum diameter.

Allow Software to Increase Number of Crossties in Section if Necessary

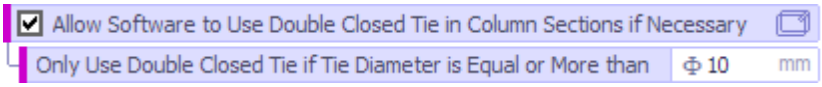
By enabling this option, you allow the software, during the column transverse reinforcement design process, to increase the number of crossties in each direction as much as possible, if required, to satisfy the transverse reinforcement demand.



This option is enabled by default, allowing the software, where possible, to satisfy the required transverse reinforcement quantity by increasing the number of legs before increasing the tie diameter.

Allow Software to Increase Number of Crossties in Section if Necessary

By enabling this option, you allow the software to use double closed ties in the column transverse reinforcement design, if necessary.

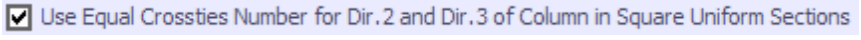


This option is enabled by default, allowing the software to use double closed ties where necessary to avoid increasing the tie diameter.

You can also specify the minimum diameter from which the software is permitted to use double closed ties. By default, the software is permitted to use double closed ties from a diameter of 10 mm and above.

Use Equal Crossties Number for Dir.2 and Dir.3 of Column in Square Uniform Sections

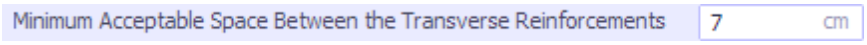
By enabling this option, the software applies the same number of crossties in both directions 2 and 3 for square column sections. This option is enabled by default.



Enabling this option can be useful for preventing construction errors.

Minimum Acceptable Space Between the Transverse Reinforcements

This option allows you to specify the minimum permissible tie spacing. By default, this parameter is set to 7 cm.



In other words, by configuring this parameter, you can ensure that in the column detailing, the tie spacing never falls below your specified value. However, it should be noted that code-specified minimum values may also govern in this regard.

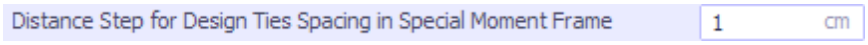
Distance Step for Design Ties Spacing in Intermediate Moment Frame

This option allows you to specify the rounding increment for tie spacing in the case of intermediate ductility. By default, this parameter is set to 5 cm.

Note that, for conservatism, spacing rounding is always performed downward, and code-specified minimum values also govern in this regard.

Distance Step for Design Ties Spacing in Intermediate Moment Frame

This option allows you to specify the rounding increment for tie spacing in the case of special ductility. By default, this parameter is set to 1 cm.



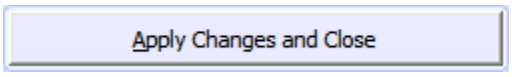
Note that, for conservatism, spacing rounding is always performed downward, and code-specified minimum values also govern in this regard.

Pitch Step for Design Spiral Pitch in Circular Column with Spiral Ties

This option allows you to specify the rounding increment for the spacing in spiral ties. By default, this parameter is set to 0.5 cm.

Note that, for conservatism, spacing rounding is always performed downward, and code-specified minimum values also govern in this regard.

The following describes some of the buttons and menus available in this user interface:



This button saves the changes made in this interface. Afterward, the reinforcement design and structural detailing must be re-executed for the changes to take effect in the structural detailing.

Saving the Column Detailing Configuration Settings

At the top of the column design configuration interface, there is a **Configuration Settings** menu through which you can save or restore your customized settings, delete the saved settings, or restore the program's default settings. The items available in this menu are described below:

Configuration Setting > Save as User Default Configuration

This option allows you to save the settings configured in this interface as the program's default settings. Note that once these settings are saved as the default configuration, the software will automatically load them in subsequent sessions and use these user-saved settings instead of its original defaults.

Configuration Setting > Load User Default Configuration

If you wish to reload all user-saved settings and configurations, you can use this option.

Configuration Setting > Delete User Default Configuration

This option allows you to delete the user default settings so that the software launches with its original default settings in subsequent sessions.

Configuration Setting > Load Software Default Configuration

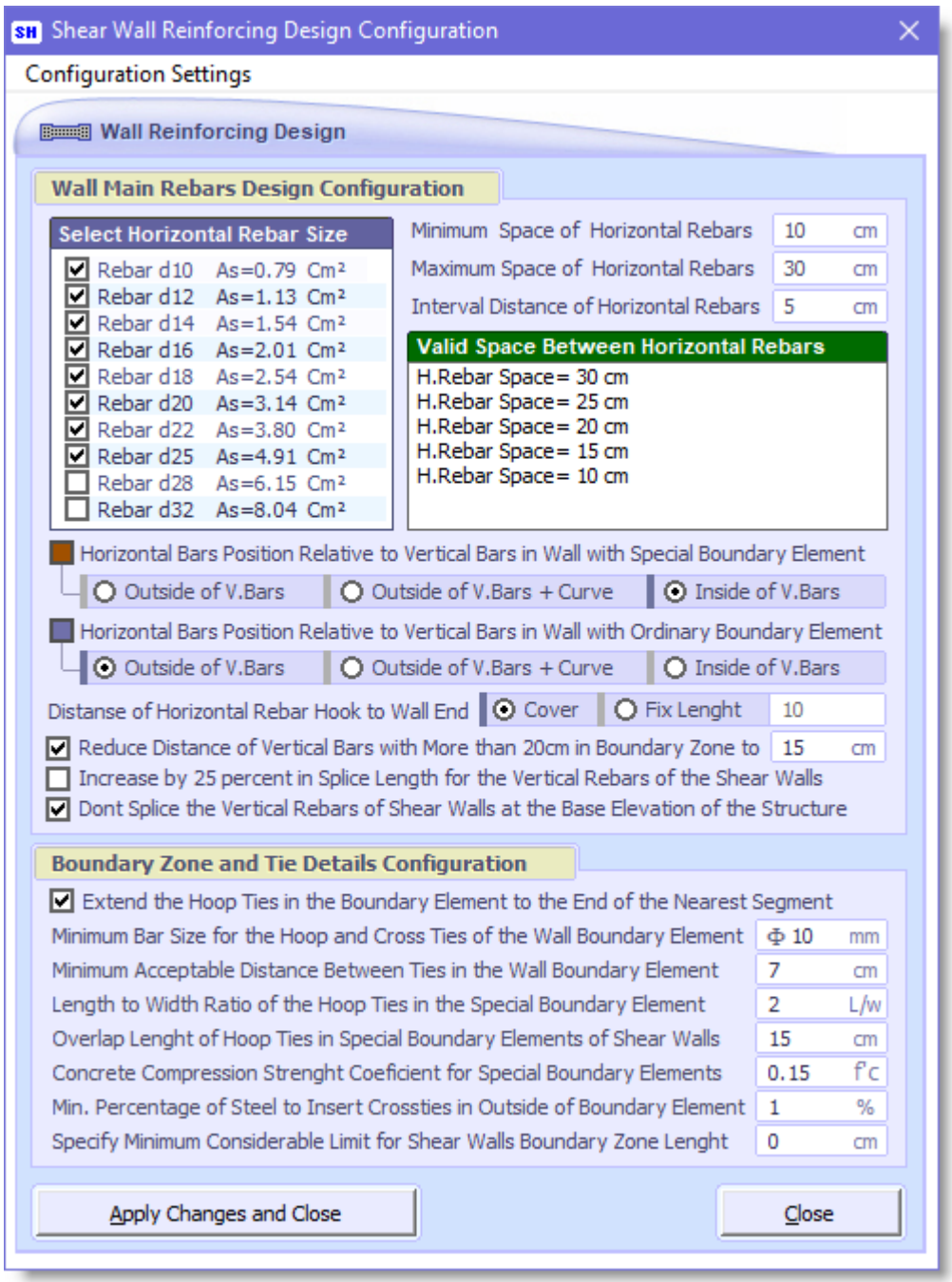
This option allows you to restore the software's default settings.

ETABS MATE
Concrete Structure Detailing Software

Shear Wall Detailing Configuration

All design parameters for shear wall detailing are set by default to the most optimal configuration. However, this user interface allows you to edit and customize these parameters according to your preferences. To access this user interface, proceed as follows using the **Design** menu. These parameters are described in the following section.

Design Menu > Shear Wall Reinforcing Design Configuration (Shift + F6)

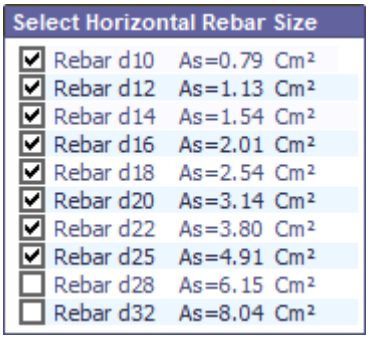


ETABS MATE
Concrete Structure Detailing Software

Shear Wall Main Rebar Settings

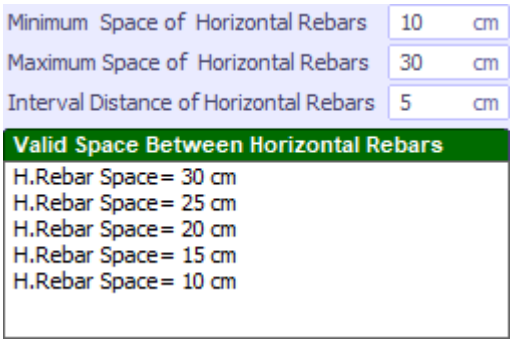
Select Horizontal Rebar Size

In this section, you can specify which rebar sizes the software is permitted to use for calculating the horizontal wall reinforcement. In many projects, to avoid an excessive variety of rebars or to account for certain construction considerations, engineers prefer that only specific sizes be used for the horizontal wall reinforcement. To this end, you can select, from the sizes available in this list, the rebar sizes that the software is allowed to use in the horizontal wall reinforcement design process.



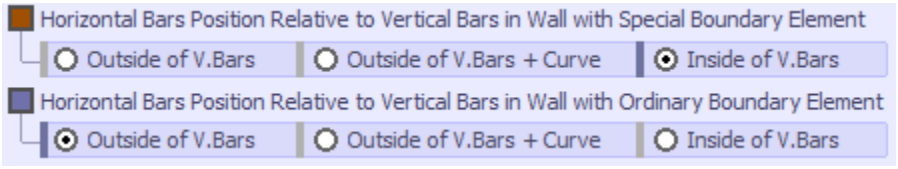
Valid Space Between Horizontal Rebars

In this section, you can specify the permissible spacings that the software may use for calculating the horizontal wall reinforcement, using three parameters as follows. First, specify the minimum and maximum permissible spacing between the horizontal rebars using the **Minimum Spacing** and **Maximum Spacing** parameters, as shown in the adjacent image. Then, define the permissible spacing increment using the **Interval Distance** parameter. Any change to any of these three parameters will update the list of permissible spacings between the horizontal rebars, and the software will ultimately use only the values present in this list for calculating the horizontal reinforcement details of the shear walls.



Horizontal Bars Position Relative to Vertical Bars in Wall

In this section, you can select the positioning of the horizontal rebars relative to the vertical wall rebars from three different methods, according to your preference. As shown in the image below, the capability has also been provided to configure this parameter separately for walls with special boundary elements and walls with ordinary boundary elements.

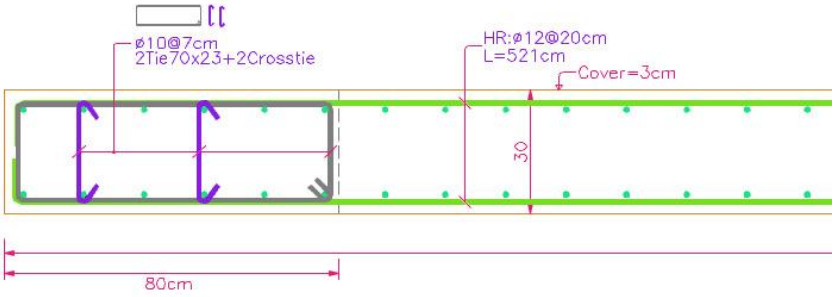


As shown in the image, for walls with special boundary elements, the **Inside of V. Bars** parameter is enabled by default, so that the horizontal rebars are placed on the inside of the vertical rebars, allowing them to be anchored within the core of the boundary element without bending and with greater ease of construction. For walls with ordinary boundary elements, the **Outside of V. Bars** parameter is enabled by default, so that the horizontal rebars are placed on the outside of the vertical rebars. Further explanation of these parameters, along with the corresponding images, is provided in the following sections.

The three different configurations and their corresponding detailing are described below:

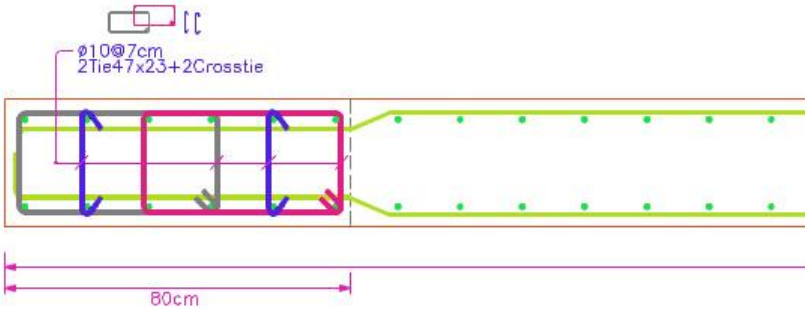
- **Outside of V.Bar:**

  In this configuration, as shown in the image below, the horizontal rebars are placed on the outside of the vertical rebars in the wall section.



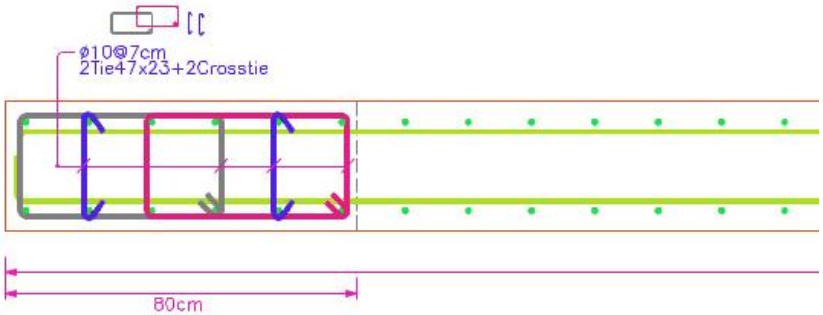
- **Outside of V.Bars + Curve:**

  In this configuration, as shown in the image, the horizontal rebars in the mid-portion of the wall web are placed on the outside of the vertical rebars in the section. However, before reaching the boundary element, they are bent inward, past the vertical rebars, to be anchored within the core of the boundary element.



- **Inside of V.Bars:**

  In this configuration, as shown in the image below, the horizontal rebars are placed on the inside of the vertical rebars in the wall section.



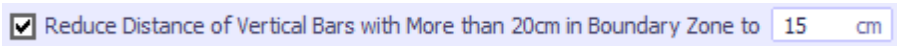
Distance of Horizontal Rebar Hook to Wall End

In this section, you can select and specify the method for calculating the distance from the end hook of the horizontal wall rebars to the end of the wall, choosing between two different methods according to your preference.

As shown in the image below, two different methods are provided for this purpose. In the first method, the software automatically calculates the distance from the hook to the end of the wall based on the specified covers. In the second method, the user can specify a constant length for this purpose. As you are aware, based on ACI 318, this length should be taken as not less than 15 cm. In the software, this parameter is set to 10 cm by default.

Reduce Distance of Vertical Bars With More than 20cm in Boundary Zone to …

By enabling this parameter, if the spacing between the vertical wall rebars exceeds 20 cm, the software automatically reduces the spacing between the vertical rebars in the boundary zone of the wall to your specified value.



As you are aware, the spacing between compressive rebars should not exceed 20 cm. However, in the upper stories of the structure, due to reduced forces, spacings greater than 20 cm are typically considered for the vertical rebars. Using this parameter, you can easily and automatically control this without making changes in the Section Designer.

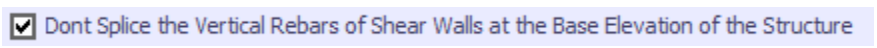
Increase by 25 percent in Splice Length for the Vertical Rebars of the Shear Walls

By enabling this parameter, the splice length of the vertical shear wall rebars is increased by 25 percent.

If this parameter is not enabled, the splice length of the vertical wall rebars is calculated normally.

Dont Splice the Vertical Rebars of Shear Walls at the Base Elevation of the Structure

This parameter allows you to specify whether the splicing of the vertical wall rebars occurs at the foundation level or at the first floor level.



As you are aware, since the foundation level is a critical section and splicing of the vertical wall rebars should not occur at this location, this parameter is enabled by default, ensuring that the first splice of the vertical rebars takes place at the first floor level of the structure.

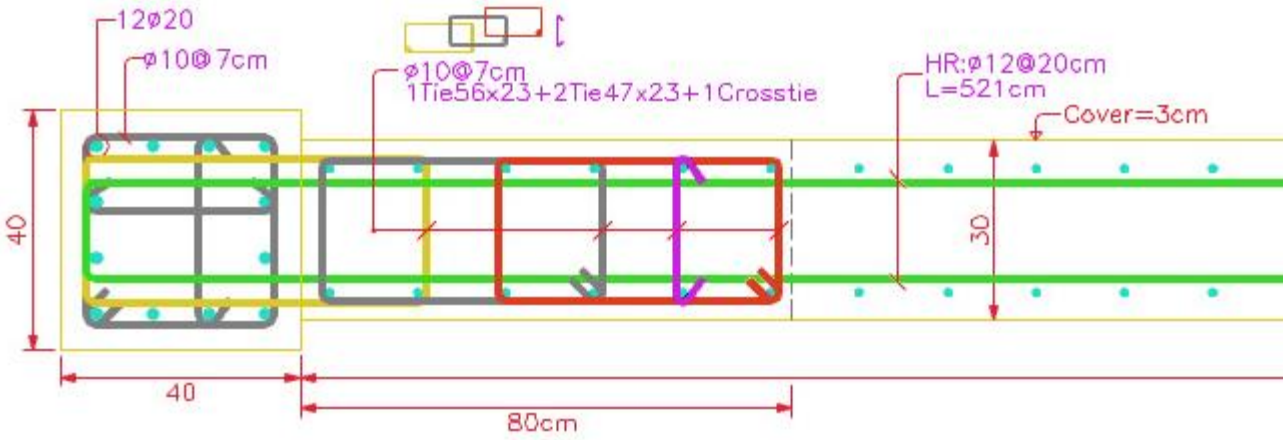
Transverse Reinforcement and Boundary Element Settings

The second section of this interface pertains to the tie and boundary element settings, which are described below:

Extend the Hoop Ties in the Boundary Element to the End of the Nearest Segment

When this parameter is enabled, the software connects the boundary element to the adjacent column by means of a closed tie.

With this parameter enabled, if a column exists at the end of the wall, the software uses a closed tie to connect the wall web boundary element to the end column, such that it encloses at least two of the wall boundary element rebars and the full dimension of the adjacent column. As shown in the image below, the boundary element of the wall web and the boundary element of the column are connected to each other by the yellow closed tie.



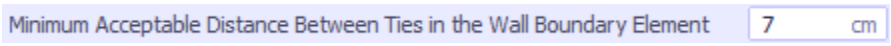
Minimum Bar Size for the Hoop and Cross Ties of the Boundary Element

This parameter allows you to specify the minimum bar size for the closed ties or crossties of the wall.

This parameter is set to 10 mm by default. Note that this parameter defines the minimum diameter, and in the calculations performed, the diameter of the hoops or crossties may exceed this value.

Minimum Acceptable Distance Between Ties in the Wall Boundary Element

This parameter allows the user to specify the minimum acceptable vertical spacing between ties in the shear walls.



This parameter is set to 7 cm by default, meaning the software is not permitted to reduce the tie spacing below this value. If necessary, the software will satisfy the required shear reinforcement quantity by increasing the tie diameter.

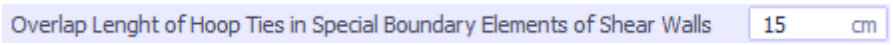
Minimum Acceptable Distance Between Ties in the Special Wall Boundary Element

This parameter allows you to specify the maximum length-to-width ratio for the closed hoops in the special boundary element of the wall.

If the wall boundary element is of the special type, the hoops of the boundary element must be detailed as overlapping chains with a maximum length-to-width ratio of two, in accordance with ACI 318 provisions. For this reason, this parameter is set to 2 by default. However, you may edit this parameter based on the specific conditions of your project.

Minimum Acceptable Distance Between Ties in the Special Wall Boundary Element

This parameter allows you to specify the minimum overlap length of the chained closed hoops in the special boundary element of the wall.



If the wall boundary element is of the special type, the hoops of the boundary element must be detailed as chains with a minimum overlap of 15 cm between the hoops, in accordance with ACI318 provisions. For this reason, this parameter is set to 15 cm by default. You may edit this parameter based on the specific conditions of your project.

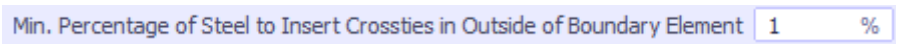
Concrete Compression Strength Coefficient for Special Wall Boundary Element

This parameter allows you to specify the concrete compressive strength coefficient that serves as the basis for the software's determination of whether the boundary element of the shear walls is special or ordinary.

Based on ACI 318, the software compares the maximum compressive stress of the wall at each story with the value 0.15 f'c. If the maximum compressive stress of the wall exceeds 0.15 f'c, the software automatically designates the boundary element type for that story as special; otherwise, it designates it as ordinary. For this reason, this parameter is set to 0.15 by default. However, you may edit this parameter based on the specific conditions of your project.

Min. Percentage of Steel to Insert Crossties in the Outside of Boundary Element

This parameter allows you to specify the minimum reinforcement percentage for providing crossties in the mid-portion of the wall web.



If the quantity of vertical reinforcement in the shear wall exceeds 1%, crossties are required for the lateral restraint of these rebars. Accordingly, the software automatically calculates the reinforcement percentage of the vertical web rebars, and if this quantity exceeds 1%, it draws the crossties of the wall web.

It should be noted that the software automatically calculates the vertical spacing of the wall web crossties based on the minimum of: the wall thickness, ***16d_b***, and ***48ds***. It is worth noting that restraining all web rebars is not necessary. If the clear distance between a restrained rebar and the adjacent unrestrained rebar is less than 15 cm, the crossties may be placed alternately, which the software also controls automatically.

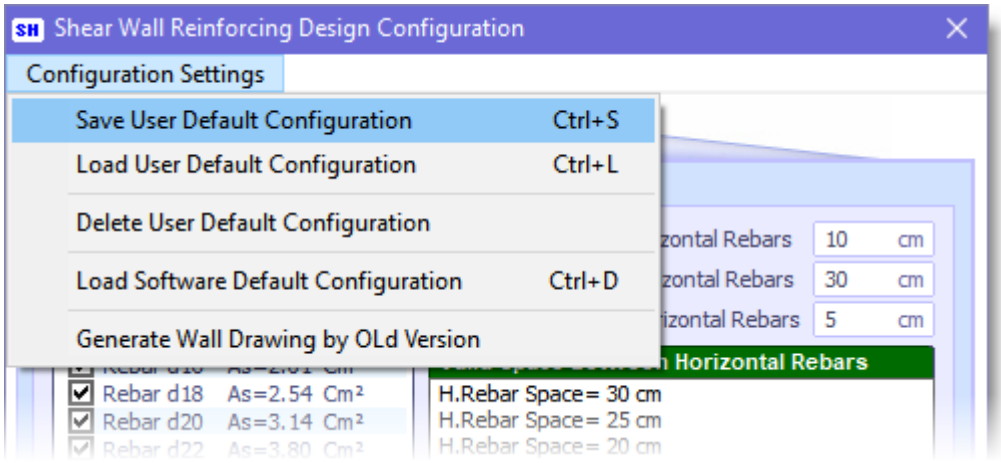
Min. Percentage of Steel to Insert Crossties in the Outside of Boundary Element

This parameter allows you to specify a minimum boundary element length for all walls.

If the value of this parameter is set to zero, the software uses the length calculated by ETABS as the basis for its calculations. However, if a value greater than zero is specified for this parameter, the software compares the calculated boundary element length with this parameter. If this parameter is greater than the calculated length, the software uses this parameter value as the boundary element length; otherwise, the calculated length is used as the basis.

Saving the Shear Wall Detailing Configuration Settings

At the top of the shear wall design configuration interface, there is a **Configuration Settings** menu through which you can save or restore your customized settings, delete the saved settings, or restore the program's default settings.



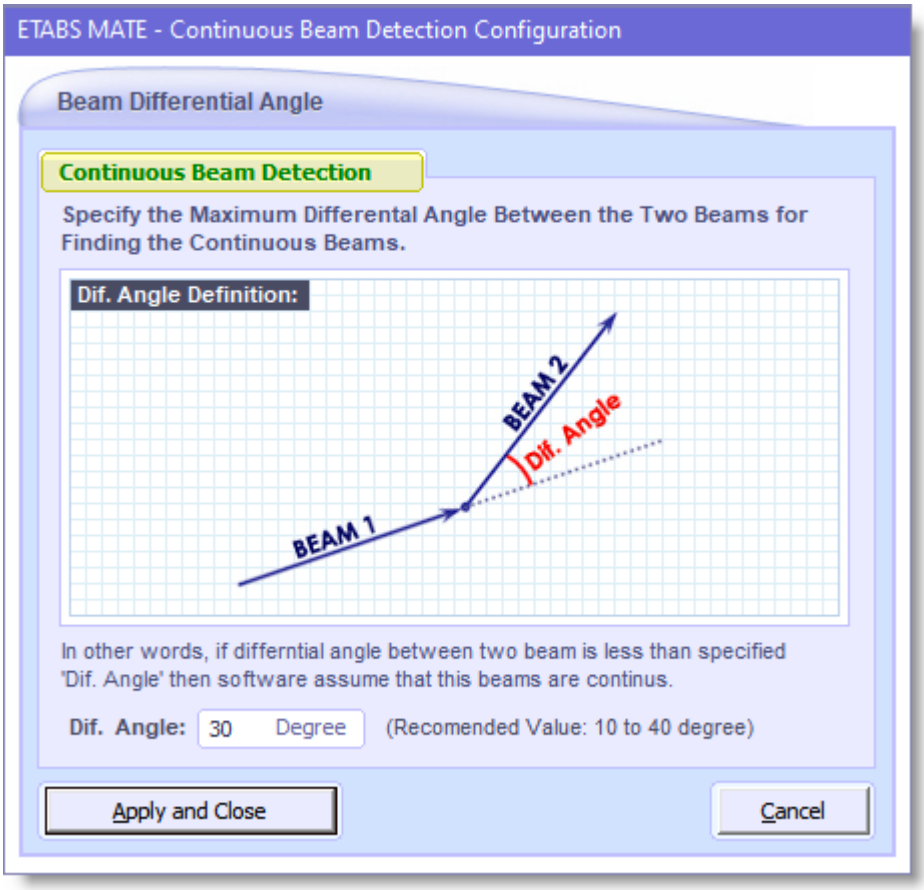
Furthermore, by enabling the **Generate Wall Drawings By Old Version** option in this menu, the software performs the calculations and generates the shear wall drawings based on the earlier ACI 318-2008 code.

Differential Angle Between Continuous Beams

This parameter allows you to specify which beams the software considers as continuous and which beams it considers as end beams of the frame. This parameter is crucial in the beam reinforcement detailing process and in the automatic identification of longitudinal profiles of continuous beams. To configure this parameter, proceed as follows from the **Design** menu.

Design Menu > Differential Angle Between Continues Beam

Executing this command opens the **Beam Differential Angle** window, as shown below, in which the definition of the **Dif. Angle** parameter is graphically illustrated in red.



This parameter is set to a default value of 30 degrees, meaning that if the angle indicated in the image above is less than 30 degrees, the software considers **Beam 2** as continuous with **Beam 1**, and these two beams are considered **Continuous** from the intersection point. However, if the angle between the two beams indicated in the image above is greater than 30 degrees, the software considers these beams as **Not Continuous** from the intersection point.

ETABS MATE
Concrete Structure Detailing Software

Design and Detailing of Structural Elements

After importing the structure into the software and performing the necessary configurations, you can execute the structural detailing command. To do so, you may use the **Design Icon** in the main toolbar at the top of the software or press the **F2** key on the keyboard.

You may also use the **Design** menu as follows:

Design Menu > Design Total Reinforcing Details (F2)

Executing this command opens the **Structure Reinforcing Design** window, which displays the progress of the design process for the various structural elements to the user. As shown in the image below, this process begins with the classification of the structural columns and the design of their transverse reinforcement. Subsequently, the reinforcement detailing of the shear walls is calculated, and finally, the design process for the beam additional reinforcement is performed, along with the necessary classifications.

The detailing process is executed by the software with high accuracy and speed, such that for a typical building of five thousand square meters in area, the entire process is completed in less than half a second.

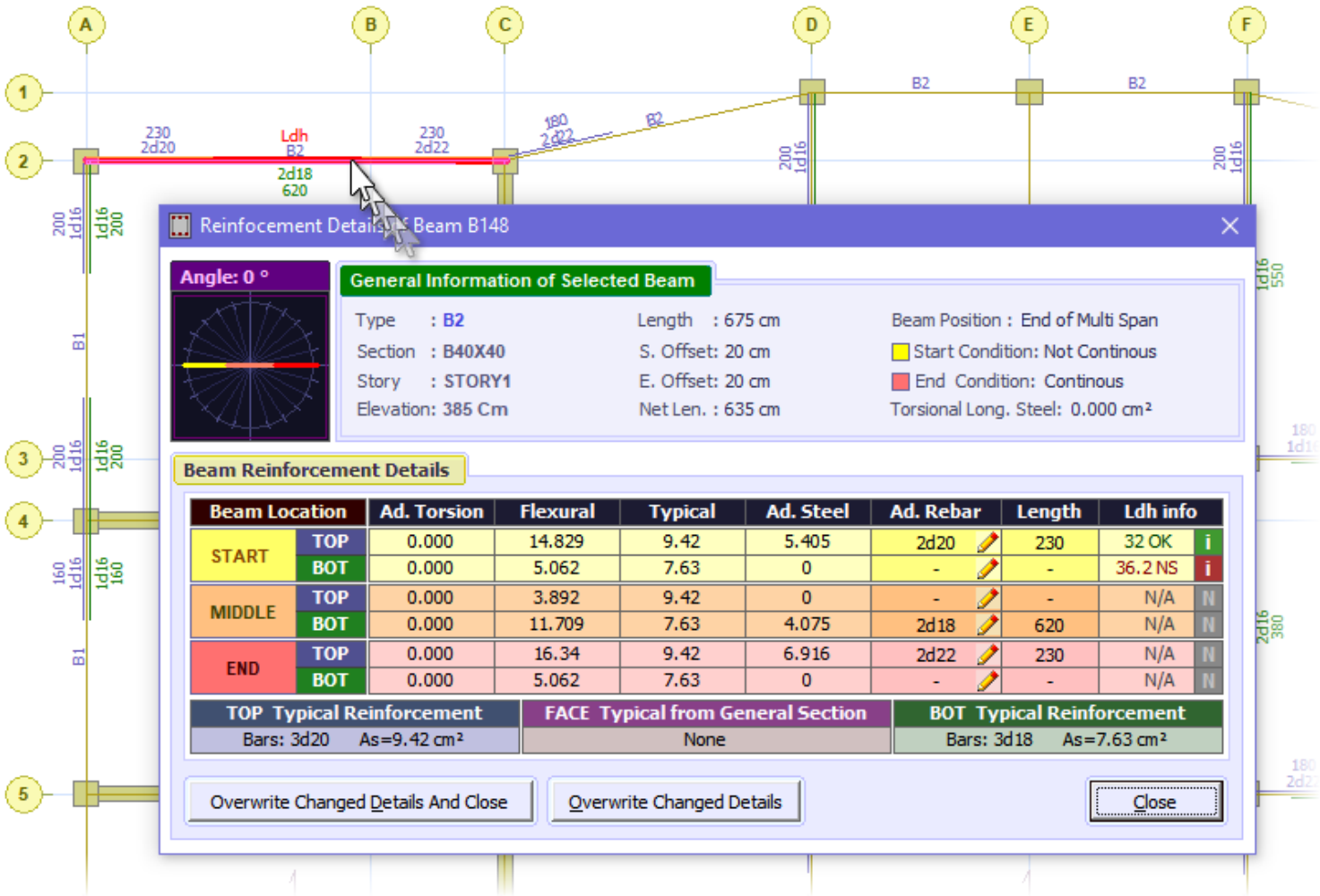
Upon completion of the reinforcement design of the structure, all designed details are displayed by the software within the graphical view.

By changing the display layers through the **Layer Display** panel, the user can access all of the designed details and, if necessary, edit the details designed by the software.

ETABS MATE
Concrete Structure Detailing Software

Editing Beam Additional Reinforcement Details

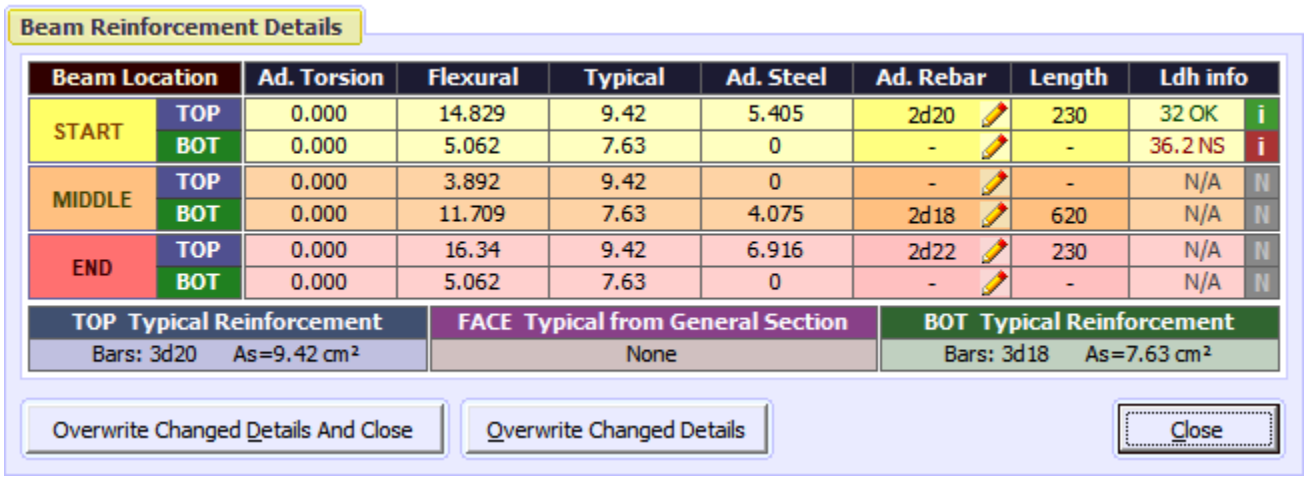
After the design and detailing proccess is complete, the beam additional reinforcement details are displayed on the structural plan in the **Total Beam Rebar Details** layer. In addition to viewing the details within the graphical view, these details can also be edited. To do this, right-click on any beam in the **Total Beam Rebar Details** layer, and then the design details of selected beam will appear, as shown in the image below.



This user interface displays all the geometric and reinforcement detailing information of the selected beam. The interface consists of two sections, which are described below:

In the **General Information of Beam** area, the general beam information is displayed, including the Beam Angle in plan, the Start and End positions of the beam (indicated in yellow and red), the Section Type, Section Name, Story Name, Elevation, Total Beam Length, the Start and End Offsets of the beam (calculated by the software based on the column dimension), the Net Length of beam (calculated by subtracting the start and end offsets from the total beam length), the Beam's Position in the frame, the Continuity Status (Continuous or Not Continuous) at the start and end of the beam in the frame, and the Torsional Reinforcement quantity of the beam.

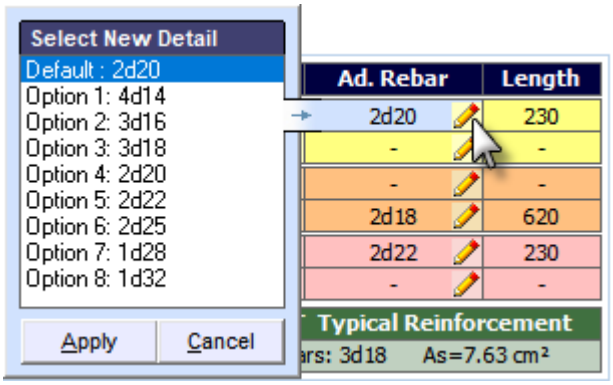
In the **Beam Reinforcement Details** part, the beam reinforcement detailing is displayed and can be edited:



- **Beam Location:** The position of the rebar in the beam.
- **Ad. Torsion:** The additional longitudinal torsional steel distributed at the top and bottom locations.

  Note that the additional longitudinal torsional reinforcement is obtained by subtracting the side face rebar area from the total longitudinal torsional reinforcement.
- **Flexural:** The total flexural reinforcement area at the given location
- **Typical:** The total typical reinforcement area of the beam at the given location
- **Ad. Steel:** The required additional reinforcement area at the given location
- **Ad. Rebar:** The additional reinforcement details calculated by the software at the given location

To edit the additional reinforcement details, you can directly type your desired details in this field using the format "***Number* d *Diameter***" such as **2d18**, or click on the pencil icon next to the additional rebar details to view a list of possible options, as shown in the image. Then, select your desired option from this list and press the **Apply** button to confirm.

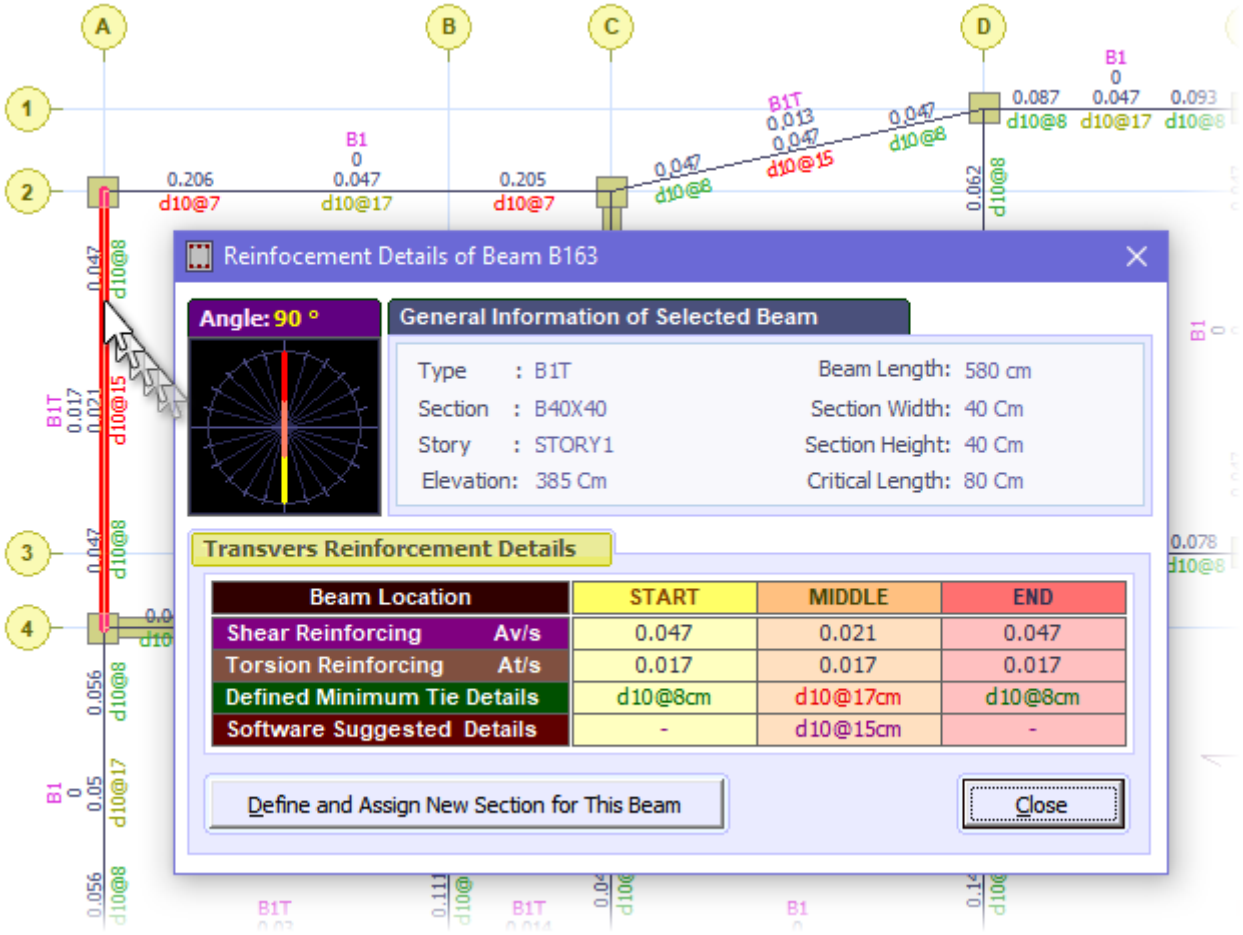


Note that all options in this list satisfy the required reinforcement quantity; however, the software has selected the most optimal configuration based on its internal procedures.

- **Length:** The additional rebar length calculated by the software, which is editable
- **Ldh info:** The development length details of hooked bars in the end beams of the frame

Editing Beam Tie Details

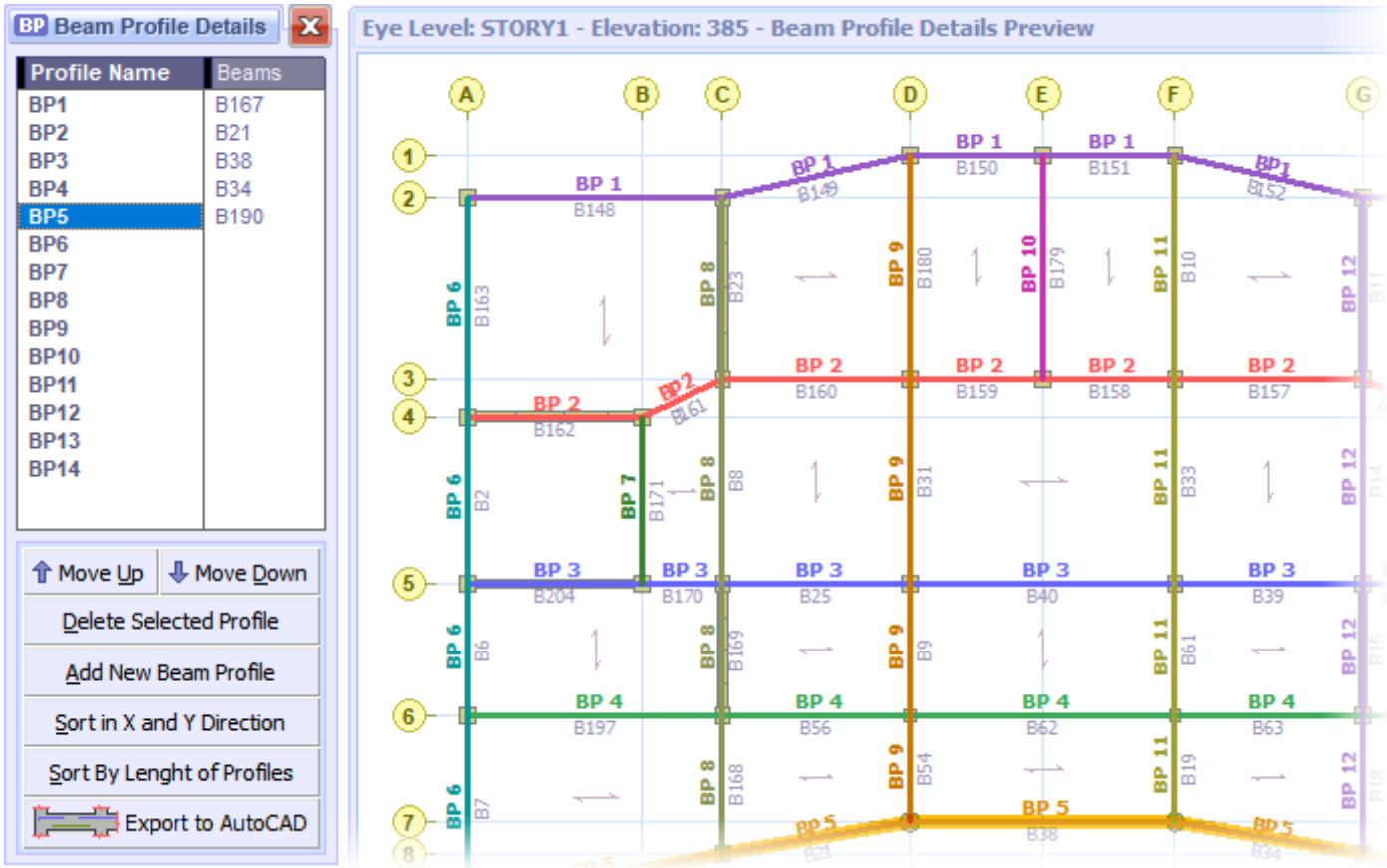
After the design and detailing proccess is complete, if you activate the **Shear+Torsion Ties** layer, the tie details are displayed on the beams in plan. These details, listed sequentially from top to bottom, are: the beam type, the transverse reinforcement due to torsion (A_t/s), the transverse reinforcement due to shear (A_V/s) at the top of the beam, and the tie details of the beams at the start, middle, and end locations, shown at the bottom of the beams.



As shown in the image, some tie details are displayed in red. The red color indicates that the minimum tie quantity defined by the program or by the user in the beam section, at the given location, is less than the transverse reinforcement quantity due to shear and torsion calculated by ETABS. Consequently, the software has been required to increase the tie quantity to satisfy the required transverse reinforcement of the beam at that location. Therefore, the red color does not signify inadequate ties; it merely indicates that a greater amount of transverse reinforcement was required compared to the minimum quantity defined for the section, prompting the software to calculate the necessary tie and display the new details in red on the beam to distinguish them. Conversely, the green color signifies that the minimum tie quantity defined for the beam section is sufficient at the given location. Note that by right-clicking on any beam, the tie details will appear as shown in the figure above, and all calculation details are viewable in this window.

Managing and Editing Beam Profiles

After the design and detailing proccess is complete, if the **Beam Profile Details** layer is selected from the **LAYER DISPLAY** panel, the beam longitudinal profiles automatically identified by the software are displayed in the graphical view, as shown in the image below. Clicking on any profile in the graphical view or in the beam profile list highlights the selected profile in the graphical view, and the names of the beam elements comprising the profile are also displayed in the beam list.



The **Beam Profile Details** section contains tools for editing and managing the beam longitudinal profiles, which are described below.

■ Move Up:

This button allows you to move the selected longitudinal profile upward in the list, and consequently to a higher position in the structural drawing.

■ Move Down:

This button allows you to move the selected longitudinal profile downward in the list, and consequently to a lower position in the structural drawing.

■ Delete Selected Profile

This button allows you to delete the selected longitudinal profile. To do so, first select the desired profile and then press this button to delete it.

■ Add New Beam Profile

This button allows you to define a new longitudinal profile and add it to the existing profile list. To do so, first click the **Add New Beam Profile** button to open the beam selection panel. Then, to define a new profile, click on the desired beam to select the first beam of the profile; it will be added to the profile beam list. Continue sequentially clicking on the subsequent beams until the last beam of the profile is selected, and finally click the **Add Beam Profile** button to define the new beam longitudinal profile and add it to the story's longitudinal profile list. Note that the beams must be selected in sequence, and the beams of a profile must be aligned.

■ Sort in X and Y Direction

This option allows you to sort the naming order of the beam profiles in plan based on their positions in the horizontal and vertical directions of the plan, facilitating easier identification of each profile in the drawings. In this mode, horizontal beam profiles are named first, from top to bottom, followed by vertical beam profiles, named from left to right.

■ Sort in By Length of Profiles

This option allows you to reorder the naming of the beam profiles based on the number of spans, from the highest number of spans to the lowest. This sorting method can be useful in certain projects for a more convenient arrangement of beam profiles in the structural drawings.

■ Export to AutoCAD

This option provides access to the user interface for configuring and generating the beam longitudinal profile structural drawings, and allows you to execute the command for generating the beam longitudinal profile structural drawings.

Upon completing the necessary edits, click the icon  at the top of the **Beam Profile Details** panel to close this panel and restore the main software panels.

If necessary, you can issue the command for re-identification and reclassification of the beam longitudinal profiles from the **Design** menu:

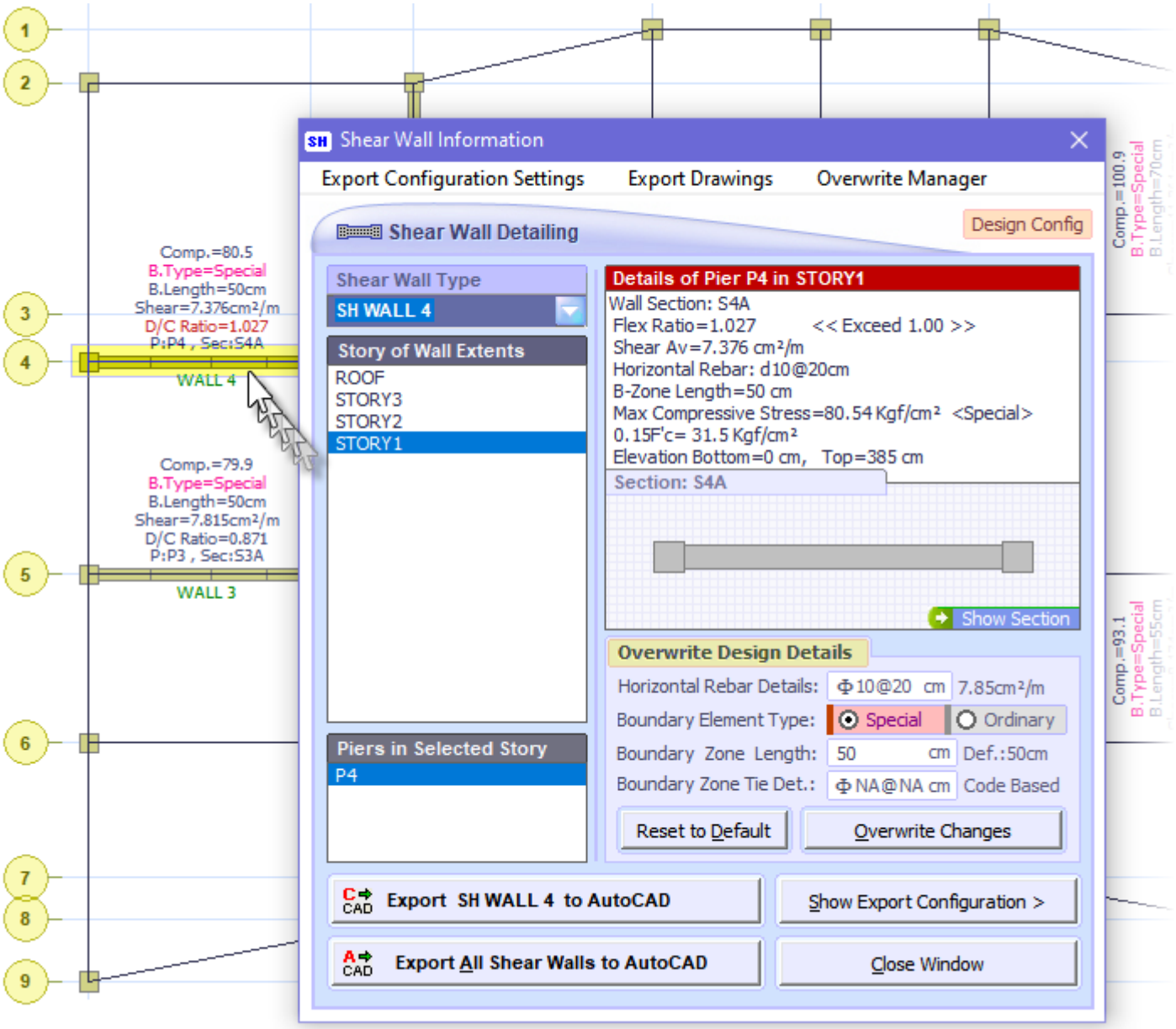
Design Menu > Recalculate Beam Profile Type Shift + F2) (

Executing this command resets the beam profile classification, and the software re-executes the beam longitudinal profile identification and classification procedures, re-identifying and reclassifying the beam profiles.

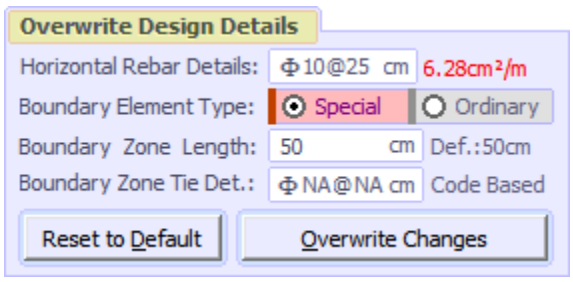
ETABS MATE
Concrete Structure Detailing Software

Editing Shear Wall Details

By selecting the **Wall – Pier Information** layer, a significant amount of important information regarding the shear walls is displayed on the structural plan. In addition to viewing the details within the graphical view, these details can also be edited. To do so, right-click on any wall, and the shear wall details interface will appear, as shown in the image below.



Through this interface, you can view and edit the design information and details of each shear wall across all stories. To do so, right-click on any wall, or select a wall from the wall dropdown list and then select a story; all details of the wall at the selected story will be displayed, and the wall section will also be shown graphically in the section area. Clicking on the wall section image allows you to view further details of the wall section. In the **Overwrite Design Details** pane, you can also edit the wall details, as described in the following.



This pane of the **Shear Wall Information** interface allows you to view and, if necessary, edit the details designed by the software for the shear walls.
All parameters in this pane are described in detail below.

■ Horizontal Rebar Details

This option displays the horizontal rebar details calculated by the software based on the wall design parameters. These details, including the horizontal rebar diameter and spacing, can also be edited. If the horizontal rebar details entered by the user are less than the required calculated quantity, the software displays the horizontal reinforcement area per unit length in red, as shown in the image, to alert the user that the selected details are insufficient and that more appropriate details must be entered by increasing the rebar diameter or reducing the spacing between the horizontal rebars.

■ Boundary Element Type: (Special / Ordinary)

This parameter specifies the boundary element type of the wall. The software automatically determines the boundary element type for each story based on the maximum compressive stress values of the wall. However, the user can also use this option to specify the **Special** or **Ordinary** type for the boundary element at each story of the wall. Note that automatic determination of the wall boundary element type by the software is only possible if ETABS version 16 or later has been used for the structural design.

■ Boundary Zone Length:

The software displays the boundary element length calculated by ETABS in this field. However, the user can also edit this parameter for each story of the wall.

■ Boundary Zone Tie Details:

This part allows you to modify the transverse reinforcement details of the wall boundary element. The software automatically calculates the transverse reinforcement details of the wall based on the code. When the phrase **ØNA@NAcm | Code Based** appears in this parameter, it means that the user has not entered specific details for the wall ties, and the software calculates these details based on the code. However, if the user enters specific details in this field, the software applies the user-entered details for the wall ties at that story and will no longer calculate these details.

After making changes to any of these parameters, press the **Overwrite Changes** button to save the modifications, or press the **Reset to Default** button to reset all changes for the wall at the given story to the software's default settings.
Additionally, through the **Overwrite Manager** menu, you can delete all changes for all walls and revert to the software's default values.

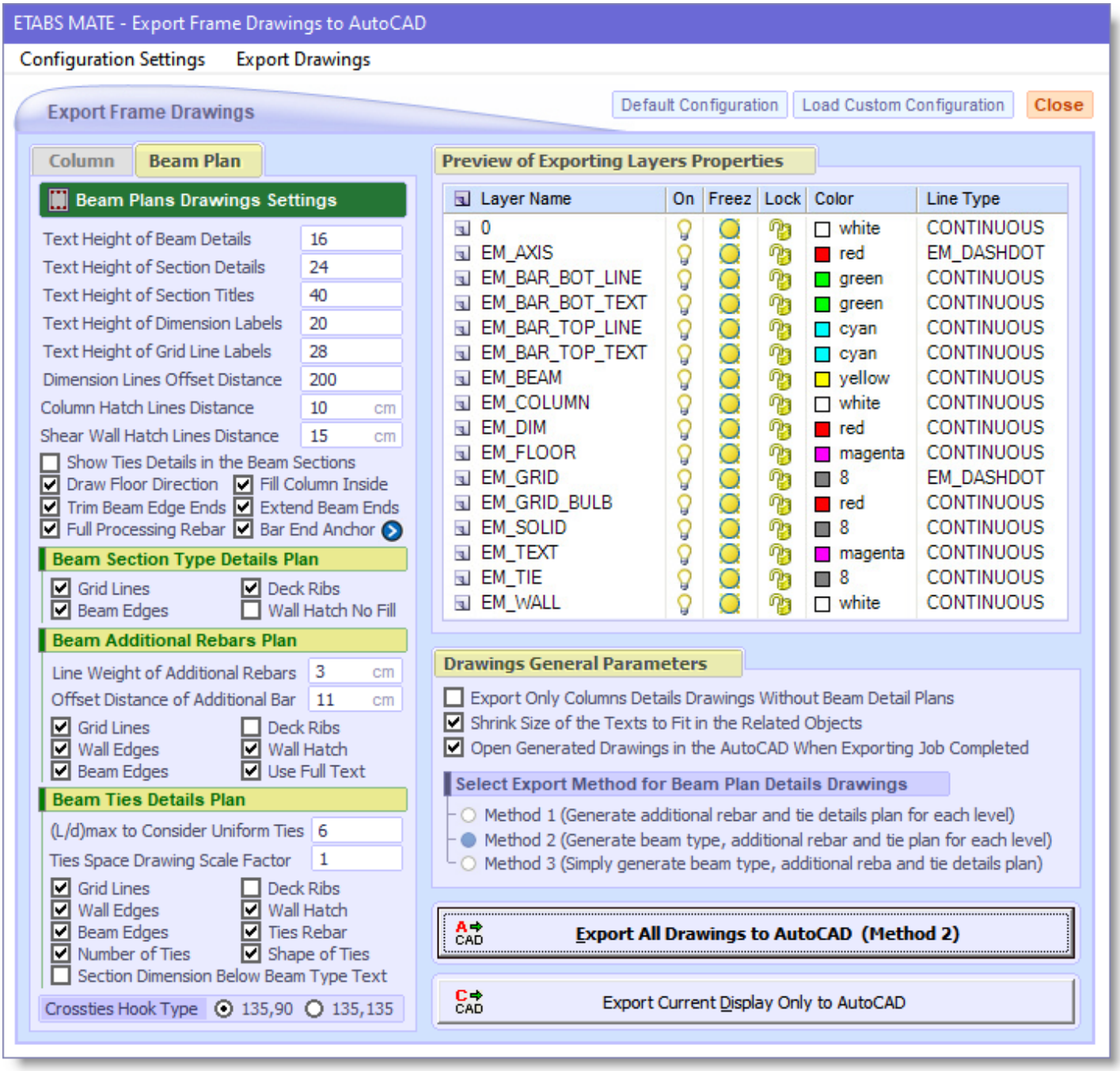
ETABS MATE
Concrete Structure Detailing Software

Frame Structural Drawings Configuration

The frame drawings include the Column and Wall Location Plan, the Beam Structural Details in plan (containing beam information, additional rebar details, and tie details along with beam section details), and the Column Structural Drawing as elevation profiles and column sections. To generate these drawings, use icon F→CAD in the main toolbar at the top of the software, or proceed as follows:

Export Menu > Export Frame Drawings to AutoCAD (F3)

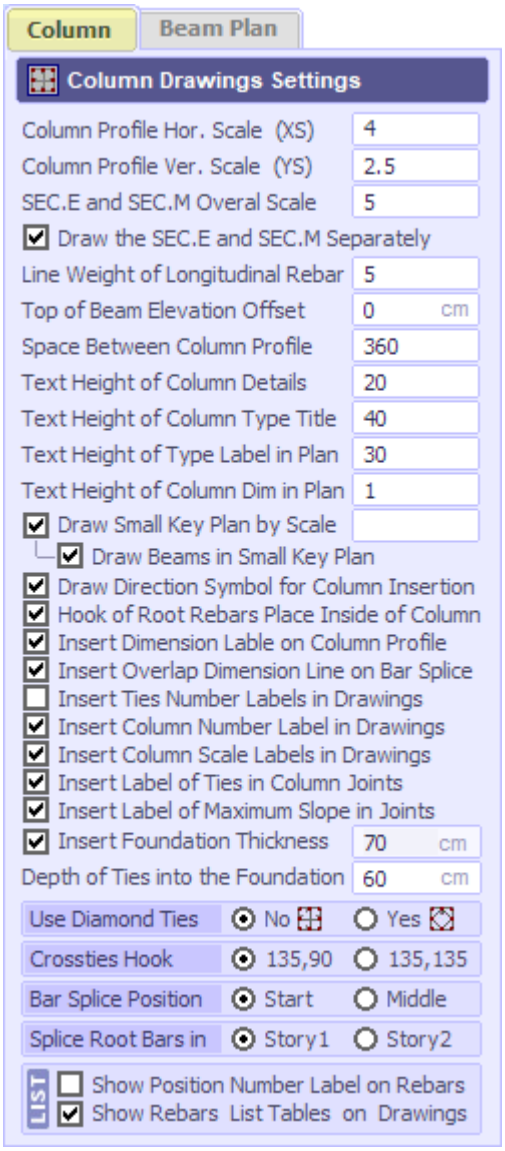
Using this tool opens the drawing configuration interface, as shown in the image below.



Column Drawing Parameters Configuration

Export Menu > Export Frame Drawings to AutoCAD (F3)

Through this interface and the tabs Column | Beam Plan at the top of this window, you can access the parameters for generating the beam detail drawings in plan and the column drawings, and configure them if necessary. To configure the column drawing parameters, first select the **Column** tab, as shown in the image below, to display the column drawing parameters.
These parameters are described below:



■ Column Profile Hor. Scale (XS)

The horizontal scale for drawing the column elevation profile.

■ Column Profile Ver. Scale (YS)

The vertical scale for drawing the column elevation profile.

■ SEC.E and SEC.M Overal Scale

The scale for drawing the end and middle column sections alongside the column elevation profile.

☑ Draw the SEC.E and SEC.M Separately

This option combines the Ends and Middle sections into a single section if possible. Since the transverse reinforcement configuration of columns may differ between the Ends and Middle regions, the software draws two sections, **SEC.M** and **SEC.E**, alongside the column by default, to clearly indicate the transverse reinforcement configuration over the full column height. To reduce the drawing sheet size when necessary, this option has been provided in the software to combine these two sections into one, where feasible.
This option is enabled by default to display the details in two separate sections for better clarity. However, for structures with intermediate ductility that do not have complex transverse reinforcement details, this option can be disabled to reduce the drawing sheet size.

■ Line Weight of Longitudinal Rebar:

The line weight of the longitudinal rebars in the column longitudinal profile.

■ Top of Beam Elevation Offset:

The reduction or increase of the top-of-beam elevation annotation placed on the columns.

■ Space Between Column Profile:

The drawn spacing between column longitudinal profiles.

■ Text Height of Column Details:

The text height, or font size, of the column detail annotations.

■ Text Height of Column Type Title:

The font size of the column type title, which is placed below each column type.

■ Text Height of Type Label in Plan:

The font size of the column type labels in the column location plan.

■ Text Height of Column Dim in Plan:

The font size of the column dimensions in the column location plan.

☑ Draw Small Key Plan, By Scale … :

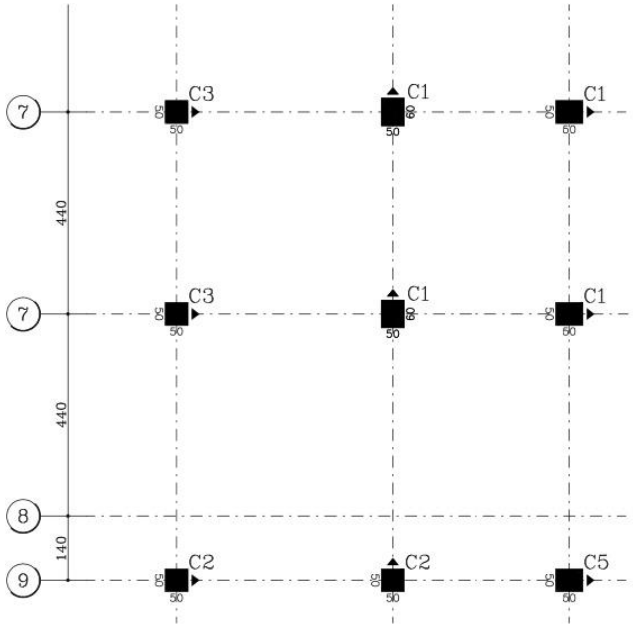
Placing a small-scale column location plan below each column type. In this section, you can also specify the scale for inserting the small key plan below the column type.

☑ Draw Beams in the Small Key Plan:

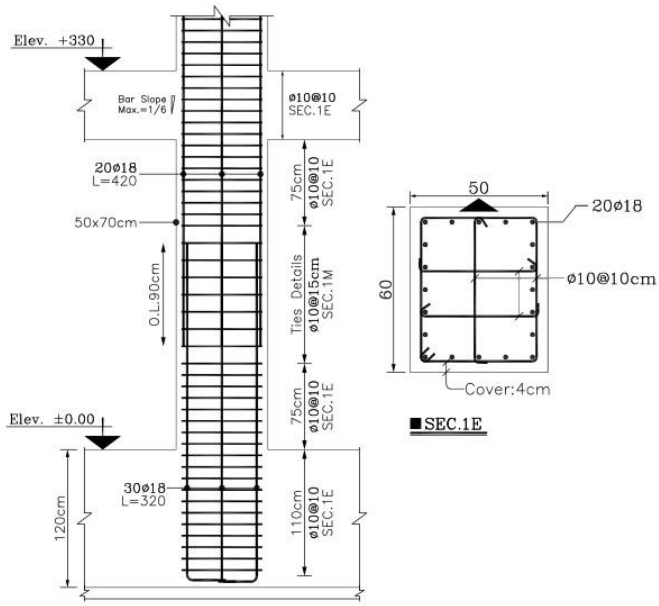
Drawing the beams in the column location plan below each column type.

☑ Draw Section Symbol for Column Insertion:

Placing a directional indicator for the column orientation in the column plan.
Since the column orientation in plan is critically important for rectangular columns or square columns with different numbers of rebars on the section faces, the software provides the capability to indicate the column orientation, as shown in the image, using a directional symbol on the column section and in the column location plan, to prevent construction errors.



■ COLUMN LOCATION PLAN



■ COLUMN C5

Placing a directional indicator for the column orientation can play a significant role in preventing construction issues and errors, particularly in projects using rectangular columns or square columns with different reinforcement on the faces. However, since only the lowest column section is displayed in the location plan, care must be taken during structural modeling in ETABS to ensure that the upper column sections are also drawn with exactly the same orientation as the lowest column section. In other words, within each column type, all column sections from the lowest story to the highest story must be inserted and modeled with an identical rotation angle.

☑ Hook of Root Rebars Place Inside of Column:

This option controls the direction of the hook bend of the column root rebars in the foundation. When enabled, the hook bends of the column root rebars are drawn toward the center of the column. When disabled, they are drawn toward the outside of the column. This option is enabled by default, in accordance with the code.

☑ Insert Dimension Label on Column Profile:

This option controls the annotation of column dimensions on the column elevation profile. When enabled, the column section dimensions are annotated on the column elevation profile.

☑ Insert Overlap Dimension Line on Bar Splice:

This option controls the insertion of dimension lines for the rebar splice length. When enabled, a dimension line indicating the splice length of the column rebars is drawn on the column elevation.

☑ Insert Ties Number Label in Drawings:

This option controls the annotation of the number of ties on the tie details. When enabled, the number of ties is annotated on the tie details in the column elevation profile.

☑ Insert Column Number Label in Drawings:

This option controls the annotation of the count of each column type below the column title. When enabled, the quantity of each column type used in the project is annotated below the column profile title.

☑ Insert Column Scale Label in Drawings:

This option controls the annotation of the column drawing scale below the column title. When enabled, the drawing scale of the columns is annotated below the column type title.

☑ Insert Label of Ties in Column Joints:

This option controls the annotation of the tie details within the column joint. When enabled, the tie details of the column critical region within the beam-column joint are annotated.

☑ Insert Label of Maximum Slope in Joints:

This option controls the annotation of the maximum permissible slope of rebar bends. When enabled, the text indicating the maximum permissible slope of the column rebar bends within the beam-column joint is annotated.

☑ Insert Foundation Thickness ...:

This option controls the insertion of a dimension line for the foundation thickness in the column longitudinal profile. When enabled, a dimension line indicating the foundation thickness is drawn in the column longitudinal profile. In this section, you can also specify the foundation thickness.

■ Depth of Ties into the Foundation ...:

This option allows you to specify the depth to which the ties of the column special region are drawn along the column root bars within the foundation.

■ Use Diamond Ties  :

This option controls the use of diamond ties in columns. When enabled, for square columns with an equal number of rebars on each face and requiring three legs of transverse reinforcement in each direction, the software uses a diamond tie instead of drawing crossties.

■ Tiepine Hook Type (⊙ 135-90, ○ 135-135):

This option allows you to specify whether the crosstie hooks in columns are drawn with 135° bends at both ends, or with a 135° bend at one end and a 90° bend at the other.

■ Bar Splice Position (⊙ Start, ○ Middle):

This option allows you to specify whether the software splices the longitudinal rebars at the bottom of the story or at the mid-height of the story.

For structures with intermediate ductility, this option is set to **Start** by default. For structures with special ductility, this option is set to **Middle** by default.

■ Splice Root Bars in (⊙ Story 1, ○ Story 2):

This option allows you to specify whether the column root bars are spliced at the first story or at the second story.

In other words, you can determine whether the column splices occur at the foundation level, or whether the root bars extend to the first floor level and are then spliced at the second story. This option can be useful in many cases where the reinforcement ratio of the columns at the first story exceeds the allowable limit, or where you wish to reduce the column rebar weight by eliminating the splice length of the root bars, resulting in a more economical structural design. However, it should be noted that in this case, due to the greater height of the root bars, the reinforcement placement of the first-story columns will be more difficult compared to the conventional case.

In all of the software's structural drawing generation interfaces, you will find the following settings under the **LIST** section, which pertain to the numbering of element rebars based on their position in the structure, also referred to as the **Position Number Label**, as well as the generation of rebar schedule tables alongside the structural drawings. These are described below.

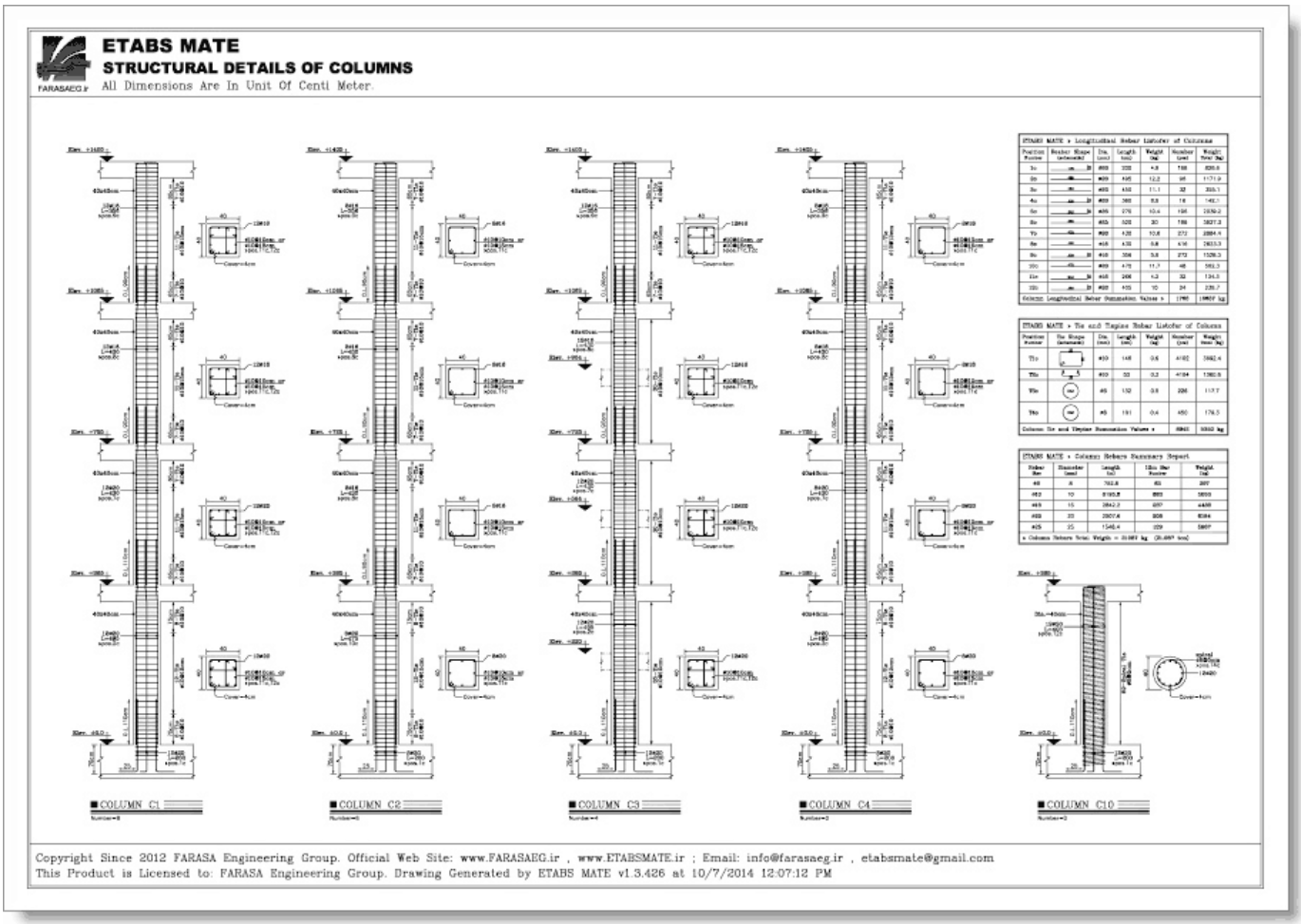
☐ Show Position Number Label on Rebars:

A position number is automatically assigned to each rebar drawn in the structural drawings. When this option is enabled, this position number is also annotated on the corresponding rebar alongside the other details, allowing you to easily locate the desired rebar in the rebar schedule tables.

☑ Show Rebar List Tables on Drawings:

When this option is enabled, the rebar schedule tables are automatically generated alongside the structural drawings. These tables are presented separately by story, element, and rebar type.

A sample of the **Column Structural Drawings** generated by the software for a structure is shown in the image below.

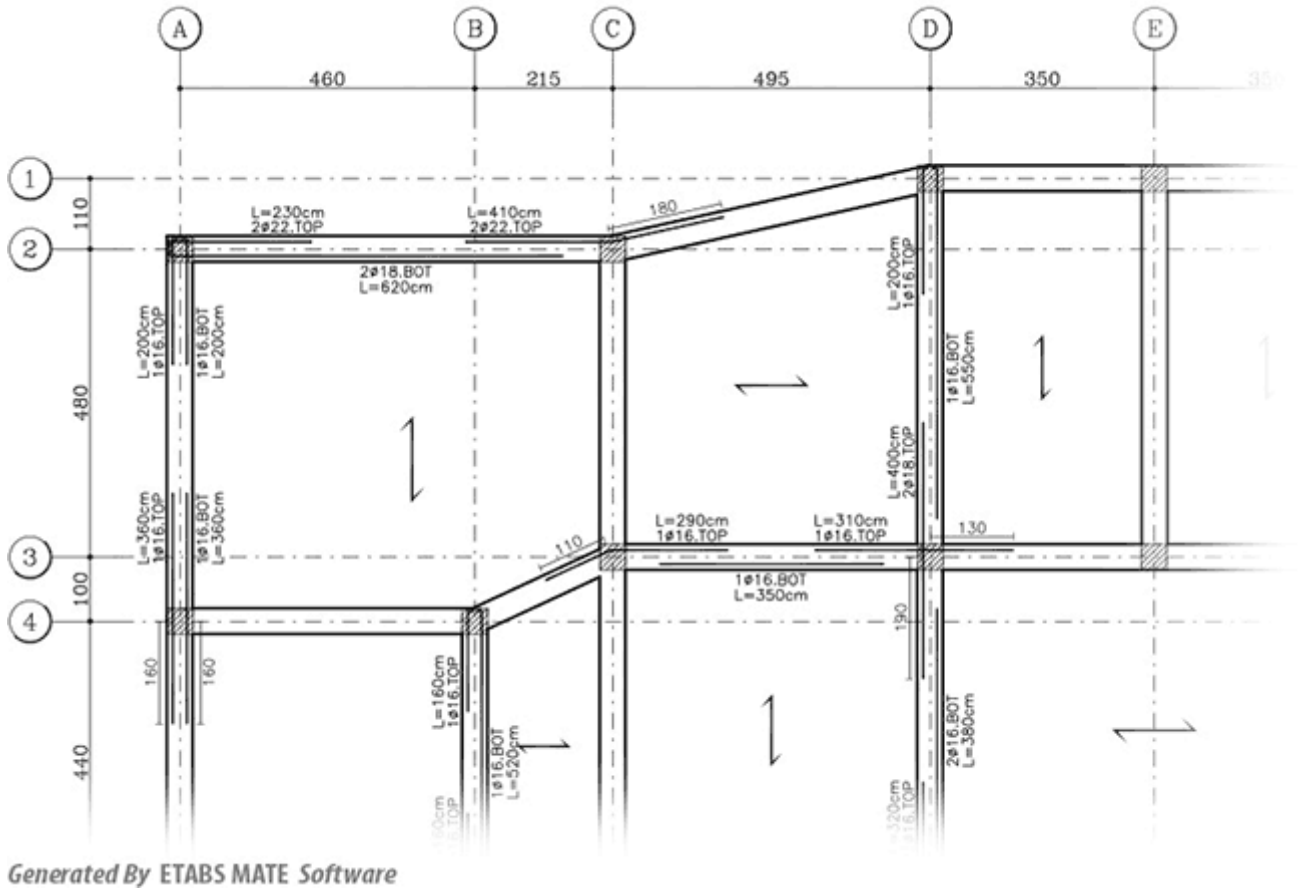


ETABS MATE
Concrete Structure Detailing Software

Beam Details in Plan Drawing Configuration

ETABS MATE provides the beam structural drawings using two different methods: drawing the beam details in plan, and drawing the beam details as longitudinal profiles. In the first method, the details of the beams of each story, including the beam section dimensions and information, additional reinforcement details, and beam tie details, are drawn on the plan of each story using a highly simple, innovative approach, which is further described below. In the second method, the beam details are drawn and presented as longitudinal profiles, which will be explained in subsequent sections.

A sample of the beam detail drawing method in plan is shown in the image below. As can be seen, this method offers numerous advantages, including simplicity, a significant reduction in the number of drawing sheets, and very easy and rapid interpretation of the drawings by engineers and rebar placers. With only the beam detail sheet in hand, rebar placers can complete the reinforcement placement of all beams of an entire floor easily and with minimal error. Furthermore, verification of the results by the design engineer and the project supervising engineer is also greatly facilitated. In many projects where the beam details are not complex, this method can be effectively used for greater simplicity. The drawing parameters for this method are described in the following.

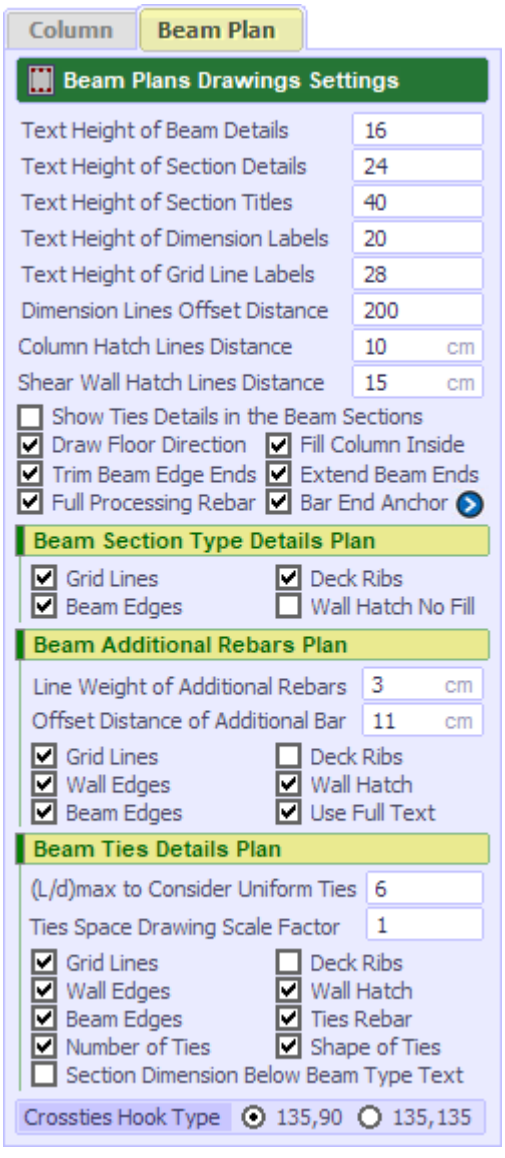


Generated By ETABS MATE *Software*

The configuration of beam detail drawing on the story plan is performed through the frame drawing interface. To access this, use the icon F→CAD in the main toolbar at the top of the software, or proceed as follows via the **Export** menu:

Export Menu > Export Frame Drawings to AutoCAD (F3)

By using the **Beam Plan** tab at the top of the frame drawing interface Column | Beam Plan, you can access the parameters for generating beam detail drawings on the plan of each story, and configure them if necessary, as shown in the image below. These parameters are set to the most optimal values to minimize the need for modification. Each of these parameters is described below.



■ Text Height of Beam Details

The font size of the beam detail annotations, such as the diameter and length of additional rebars.

■ Text Height of Section Details

The font size of the beam section detail annotations.

■ Text Height of Section Titles

The font size of the beam section titles.

■ Text Height of Dimension Labels

The font size of the dimension line annotations.

■ Text Height of Grid Line Labels

The font size of the structural grid line labels.

■ Dimension Lines Offset Distance

The offset distance of the structural grid dimension lines from the edge of the structure.

■ Column Hatch Lines Distance

The spacing between the column hatch lines in the beam plan.

■ Shear Wall Hatch Lines Distance

The spacing between the shear wall hatch lines in the plan.

☐ Show Ties Details in The Beam Section

This option controls the annotation of the transverse reinforcement details on the beam section. When enabled, the minimum tie details defined in the beam section are annotated on the section ties. When disabled, the phrase **See Tie Plan** is placed instead of the tie details. Since the minimum ties defined in the section are insufficient to satisfy the calculated reinforcement in many beams, this option is disabled by default so that the beam tie details are read from the tie plan.

☑ Draw Floor Direction

This option controls the display of the floor direction indicator on the plan.

When enabled, the direction of one-way floors is indicated by a linear arrow, and the direction of two-way floors, such as slab, waffle, U-Boot, etc., is indicated by a cross arrow on the floors.

Note that if the floor section name in the ETABS structural model begins with the letters S, W, or U, the software considers it as a two-way floor; otherwise, it is considered as one-way. (These letters are the initial characters of Slab, Waffle, and U-Boot, which are two-way floor types.)

☑ Trim Beam Edge Ends

This option controls the automatic trimming of beam line extensions at beam intersections on the beam details plan.

When enabled, the excess beam line extensions at beam intersections are automatically trimmed by the software to produce a cleaner drawing.

☑ Extend Beam Ends

This option controls the automatic extension of beam edges on the plan.

When enabled, at beam intersections where beam lines do not meet, these lines are automatically extended by the software to connect, producing a cleaner drawing.

☑ Full Processing Rebar

This option controls the full processing of additional rebars.

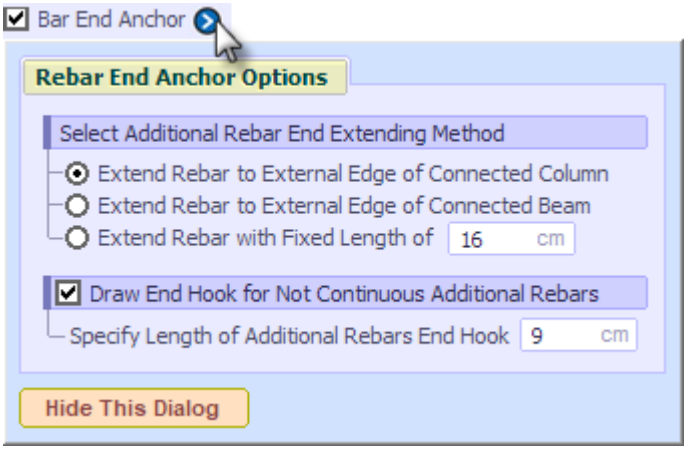
When enabled, all intelligent routines for unifying adjacent additional rebars, intelligent dimensioning of additional rebars, and drawing the end hooks of rebars are applied to all additional rebars, delivering significantly cleaner details for the additional reinforcement drawing on the plan.

☑ Bar End Anchor

This option controls the drawing of the end hook of rebars on the plan.

When enabled, the development hook of the end rebars is drawn at the discontinuous end of the beam.

Furthermore, if you click on the icon , as shown in the adjacent image, a set of additional parameters related to the method of extending the additional rebars will appear. These include: extending the rebar to the edge of the end column, extending the rebar to the edge of the perpendicular beam, extending the rebar by a constant length, as well as the option for drawing the hook and the drawn length of the 90-degree end hook of the end rebars. You can configure these parameters as well.



The software provides three separate plans for drawing all beam information and the details of the additional reinforcement and beam ties on the floor plan. These plans are: the **Beam Section Type Details Plan**, which includes the dimensional details and the section name of the beams; the **Beam Additional Rebars Plan**, which contains the additional reinforcement details, including the number, diameter, and length of the additional rebars; and the **Beam Tie Details Plan**, which contains the tie details, including the diameter, number of legs, and spacing of the beam ties. The settings for these three plans are configured as follows:

In the **Beam Type Details Plan** section, as shown in the image, the parameters related to the beam dimensions and sections plan are configurable.

☑ Grid Lines:

This option controls the display of the project grid lines. You can use this option to show or hide the structural grid lines in this plan.

☑ Deck Ribs:

This option controls the drawing of the floor ribs. You can use this option to show or hide the floor ribs in this plan.

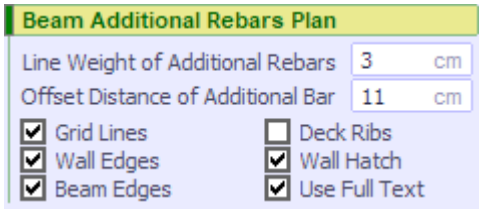
☑ Beam Edges:

This option controls the drawing of the floor ribs. You can use this option to show or hide the floor ribs in this plan.

☑ Wall Hatch No Fill:

This option allows you to specify whether a line hatch, instead of a solid fill hatch, is used for the interior of shear walls.

In the **Beam Additional Rebars Plan** section, as shown in the image, the parameters related to the beam additional reinforcement plan are configurable.



■ Line Weight of Additional Rebars

This parameter controls the drawn line weight of the additional rebars. You can use this option to specify the drawn line weight of the additional rebars in the additional reinforcement plan.

■ Offset Distance of Additional Bar

This parameter controls the offset distance of the additional rebars. You can use this option to specify the offset distance at which the additional rebars are drawn from the beam axis in the additional reinforcement plan.

☑ Grid Lines

This option controls the drawing of the grid lines. You can use this option to show or hide the structural grid lines on the story plan.

☑ Deck Ribs

This option controls the drawing of the floor ribs. You can use this option to show or hide the floor ribs in the additional reinforcement plan.

☑ Wall Edges

This option controls the drawing of the shear walls. You can use this option to specify whether the wall edge lines are drawn in the additional reinforcement plan, or omitted to reduce drawing congestion.

☑ Wall Hatch

This option controls the drawing of the hatch inside the shear walls. You can use this option to specify whether the wall hatch is drawn in the additional reinforcement plan, or omitted to reduce drawing congestion.

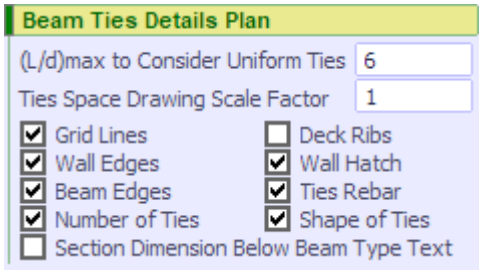
☑ Beam Edges

This option controls the drawing of beams with two edge lines. You can use this option to specify whether the beam is drawn with its edge lines, or as a single line along the beam axis.

☑ Use Full Text

This option controls the annotation of full beam details. You can use this option to specify whether the beam details are annotated in full or in an abbreviated format.

In the **Beam Ties Details Plan** section, as shown in the image, the parameters related to the beam tie details plan are configurable.



■ (L/d)max to Consider Uniform Ties

This parameter specifies the maximum length-to-depth ratio of the beam for applying uniform tie details. You can use this option to define the maximum ratio of beam length to beam depth for which uniform tie details are considered over the entire beam length. By default, this parameter is set to 6. For example, if the beam depth is 40 cm, for beams with a length less than 240 cm, the software automatically applies the maximum tie details over the entire beam length. Otherwise, the details are annotated separately for the start, middle, and end regions of the beam.

■ Ties Space Drawing Scale Factor

This parameter controls the scale factor for the drawn spacing between ties. You can use this option to specify a multiplier for the drawn spacing between ties in the tie details plan.

☑ Grid Lines

This option controls the drawing of the grid lines. You can use this option to show or hide the structural grid lines in the beam tie details plan.

☑ Deck Ribs

This option controls the drawing of the floor ribs. You can use this option to show or hide the floor ribs in the beam tie details plan.

☑ Wall Edges

This option controls the drawing of the shear walls. You can use this option to specify whether the wall edge lines are drawn in the beam tie details plan, or omitted to reduce drawing congestion.

☑ Wall Hatch

This option controls the drawing of the shear walls. You can use this option to specify whether the wall edge lines are drawn in the beam tie details plan, or omitted to reduce drawing congestion.

☑ Beam Edges

This option controls the drawing of beams with two edge lines. You can use this option to specify whether the beam is drawn with its edge lines, or as a single line along the beam axis.

☑ Ties Rebar

This option allows you to specify whether the beam details are annotated in full or in an abbreviated format.

☑ Number of Ties

This option controls the annotation of the number of ties on the beam. You can use this option to specify whether the number of ties is annotated on the beams in the beam tie details plan.

☑ Shape of Ties

This option controls the drawing of the tie shape on the beam. You can use this option to specify whether the tie shape, along with the crossties, is drawn as a small diagram on the beams.

☐ Section Dimension Below Beam Type Text:

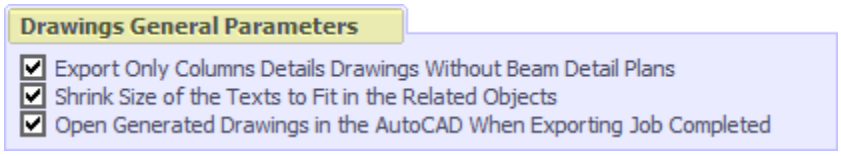
This option allows you to specify whether the beam dimensions are annotated below the beam section name in the beam tie details plan.

■ Crossties Hook Type

This option allows you to specify whether the beam crosstie hooks are drawn with 135-degree bends at both ends, or with a 135-degree bend at one end and a 90-degree bend at the other.

Drawing General Parameters

This interface also includes several general options, which are described below:



☐ Export Only Columns Details Drawings Without Beam Details Plans

When this option is enabled, only the column structural drawings are generated, and the beam details in plan are not drawn. If you are drawing the structural beams as longitudinal profiles, you can enable this option since the beam detail plans will no longer be needed, allowing the software to generate only the column structural drawings with greater speed and a smaller file size.

☑ Shrink Size of the Texts to Fit in the Related Objects

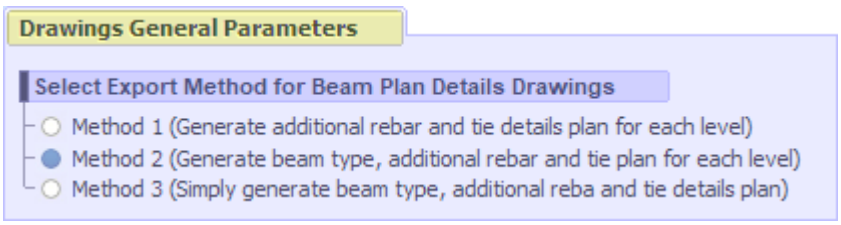
When this option is enabled, if the length of the detail text annotation for an element exceeds the length of that element, the software automatically reduces the font size to proportionally fit the detail text length to the corresponding element length.

☑ Open Generated Drawings in the AutoCAD When Exporting Job Completed

When this option is enabled, upon completion of the drawing generation, the software automatically opens the drawing in AutoCAD. For this purpose, the default Windows application for opening DXF files must be set to AutoCAD.

Methods of Beam Detail Drawing in Plan

Three different methods are provided for drawing the beam details in plan, which are described below:



- **Method 1 (Generate additional rebar and tie Details plan for each level)**

In this method, to reduce the number of sheets, the beam details are drawn in two plans in a more compact format, rather than three. In other words, the first plan, which contains the beam section dimensions and name, is omitted, and its details are added to the second and third plans.

- **Method 2 (Generate beam Type, additional rebar and tie plan for each level)**

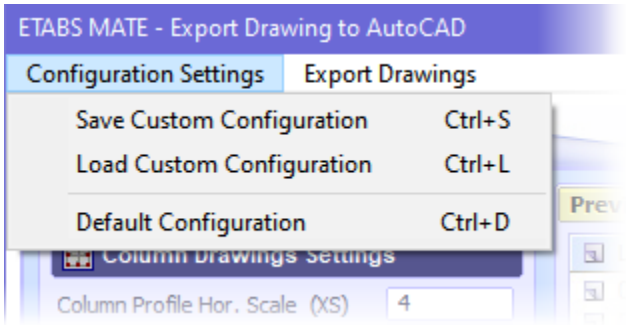
In this method, three separate plans are provided for the floor beam details: the beam section dimensions and section name plan, the beam additional reinforcement details plan, and the tie details plan. This is the software default.

- **Method 3 (Simply generate beam type, additional rebar and tie detail Plan)**

In this method, three separate plans are provided for the floor beam details: the beam section dimensions and section name plan, the beam additional reinforcement details plan, and the tie details plan. This is the software default.

Saving and Restoring Customized Frame Drawing Parameters

As shown in the image, through the **Configuration Setting** menu at the top of this interface, you can save or restore your customized settings, or reset all parameters in this interface to the software's default values.

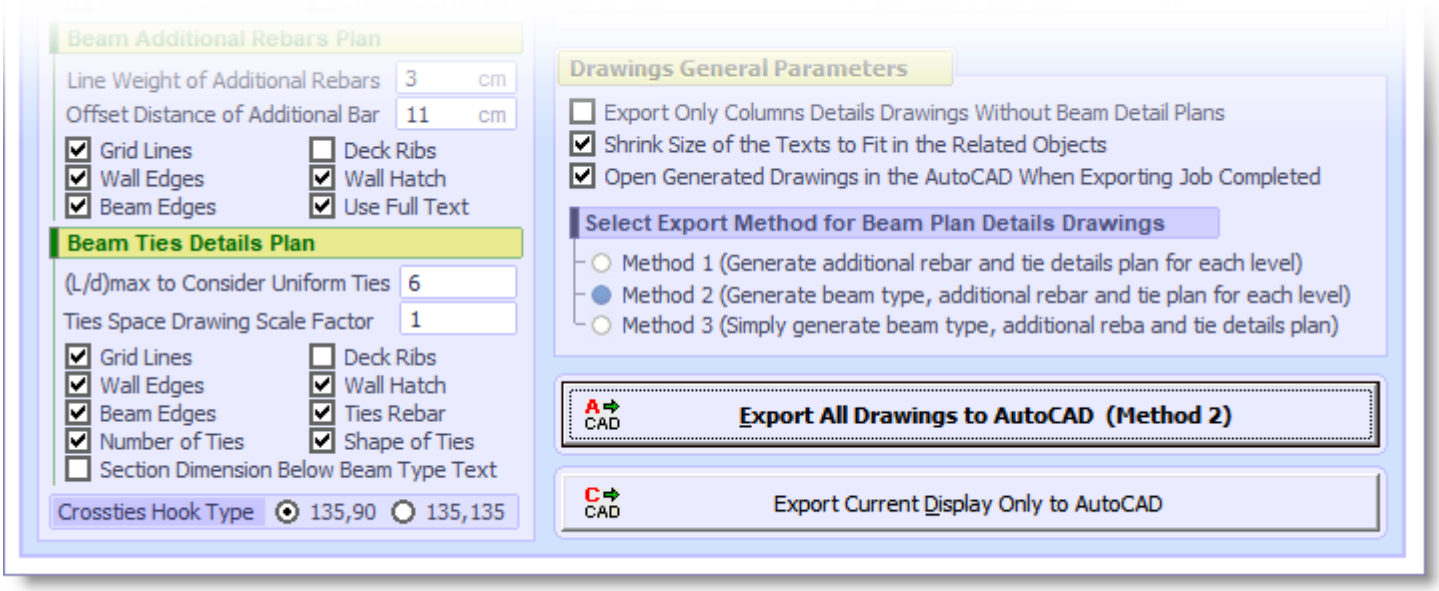


Exporting Frame Structural Drawings

The frame structural drawings include the column and wall location plan, the project beam structural drawings in plan, and the column structural drawings, along with the project column rebar schedules. To generate these drawings, use the icon in the main toolbar at the top of the software, or proceed as follows via the **Export** menu:

Export Menu > Export Frame Drawings to AutoCAD (F3)

Executing this command opens the frame drawing configuration interface. As shown in the image, two buttons are located at the bottom of this interface for exporting the drawings, which are described below.



This button generates all frame structural drawings, including the column and wall location plan, the project beam structural drawings in plan (containing beam information, additional rebar details, and tie details, along with beam section details), and the column structural drawings as elevation profiles and column sections, together with the project column rebar schedules. The drawings are saved as a single file in the AutoCAD drawing format at a location you specify. Note that the beam rebar schedules are generated when exporting the beam structural drawings as longitudinal profiles, which will be described in the following sections.

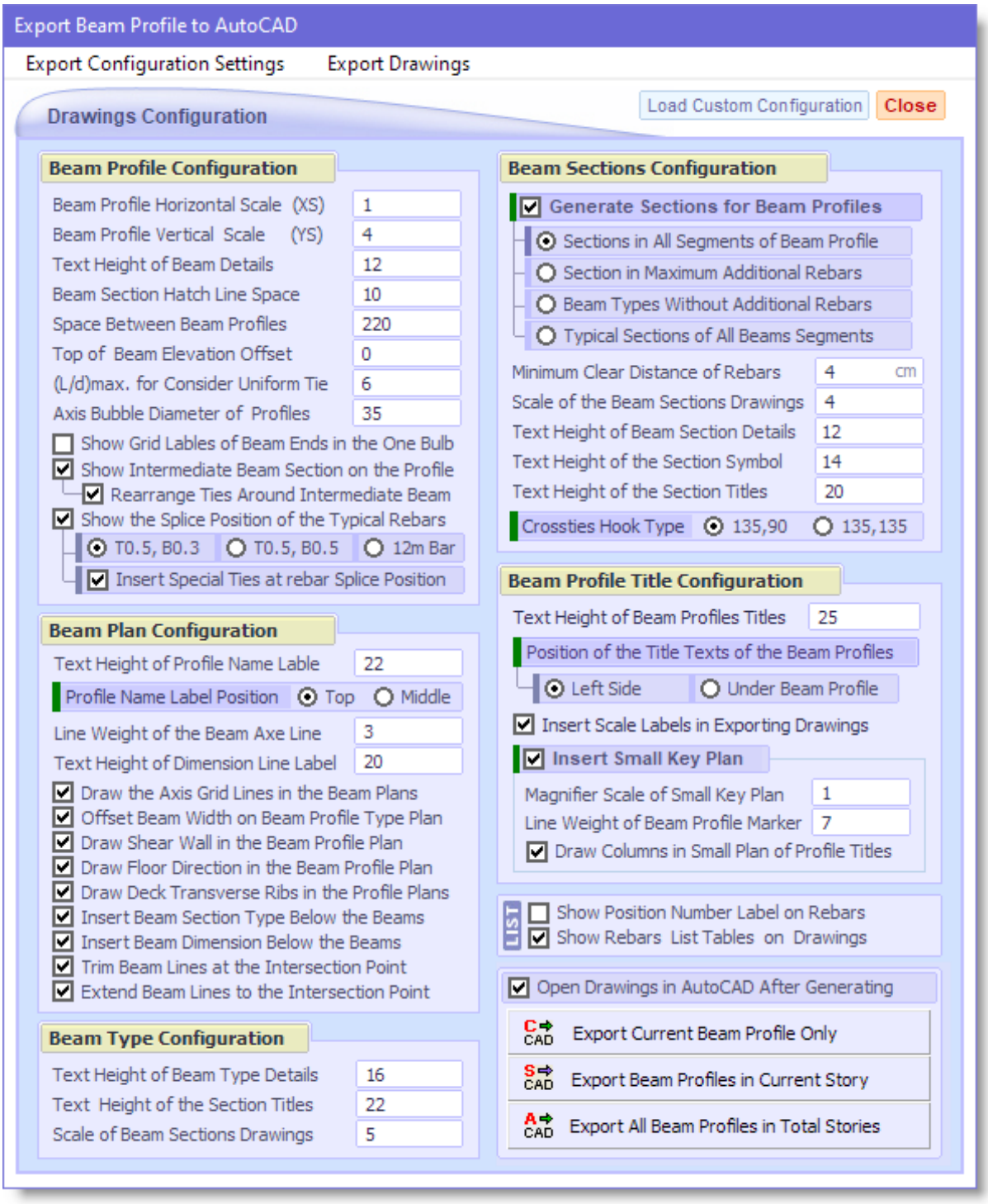
This button allows you to save the details currently displayed in the graphical view, based on the active layer in the **LAYER DISPLAY** palette, as an AutoCAD file.

Beam Profile Drawing Configuration

To configure the settings for drawing the beam profiles, proceed as follows:

Export Menu > Export Beam Profile Drawings to AutoCAD (Ctrl + F3)

Alternatively, clicking the icon B→CAD in the main toolbar at the top of the software opens the beam longitudinal profile drawing configuration interface, as shown in the image below.

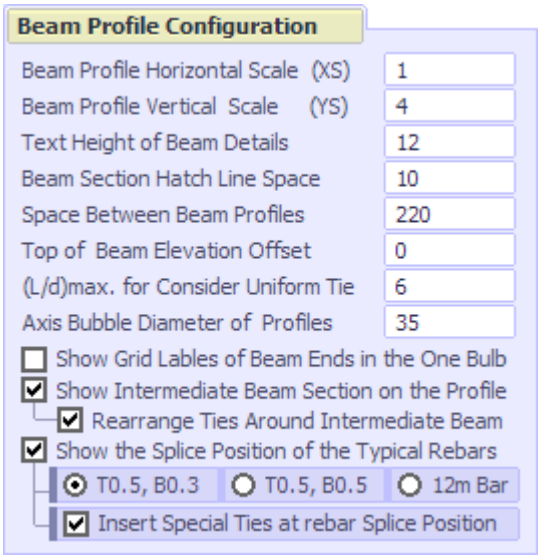


The beam longitudinal profile drawing configuration interface includes several different panes for configuring the various parts of the beam detail structural drawings as longitudinal profiles.

A brief description of each is provided below:

Beam Profile Configuration Palette

This pane contains the settings for drawing the longitudinal profiles. The parameters in this section are fully described below.



■ Beam Profile Horizontal Scale (XS)

This parameter defines the horizontal scale for drawing the beam longitudinal profile.

■ Beam Profile Vertical Scale (YS)

This parameter defines the vertical scale for drawing the beam longitudinal profile.

■ Text Height of Beam Details

This parameter defines the text height, or font size, of the beam detail annotations.

■ Beam Section Hatch Line Space

The spacing of the hatch lines for the cross-section of a secondary beam framing into the beam profile.

■ Space Between Beam Profile

This parameter defines the drawn spacing between beam longitudinal profiles. You can use this parameter to adjust the vertical spacing between profiles.

■ Top of Beam Elevation Offset

This parameter defines the reduction or increase of the top-of-beam elevation annotation placed on the beam longitudinal profile.

■ (L/d)max. for Consider Uniform Tie

This parameter allows you to specify the maximum length-to-depth ratio for applying uniform tie details over the entire beam length. By default, this parameter is set to 6. For example, if the beam depth is 40 cm, for beams with a length less than 240 cm, the software automatically applies the maximum tie details over the entire beam length. Otherwise, the details are annotated separately for the start, middle, and end regions of the beam.

■ Axis Bubble Diameter of Profile

This parameter defines the diameter of the grid axis bubbles drawn on the columns in the beam longitudinal profile.

☐ Show Grid Labels of Beam Ends in the one Bulb

When this option is enabled, the horizontal and vertical axis labels are placed in a single bubble. When disabled, each is drawn separately in two axis bubbles.

☑ Show Intermediate Beam Section on the Profile

When this option is enabled, the cross-sections of secondary beams framing into the beam longitudinal profile are drawn hatched on the beam longitudinal profile at their exact modeled positions.

☑ Rearrange Tie Space Around Intermediate Beam

When this option is enabled, the ties on either side of secondary beams in the beam longitudinal profile are given special consideration. In accordance with the code, when a secondary beam frames into a main beam, special ties must be provided on both sides of the secondary beam within the main beam. For this reason, this option is enabled by default.

☑ Show the Splice Position of the Typical Rebars

When this option is enabled, the splice of the typical rebars is displayed on the beam longitudinal profile.

When the above option is enabled, this setting allows you to select the splice position of the typical rebars using one of the following methods:

- **T0.5, B0.3**

  Top splice at the mid-span of the beam, and bottom splice at one-third of the beam span length.

- **T0.5, B0.5**

  Top splice at the mid-span of the beam, and bottom splice at the mid-span of the beam.

- **12m Bar**

  Rebar splice at the end of a 12-meter bar length, disregarding the splice position.

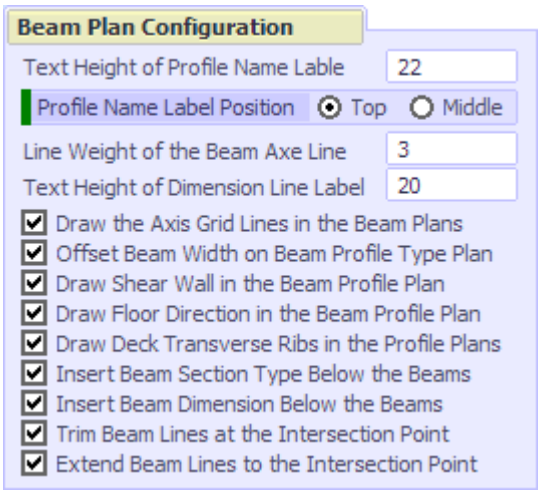
☑ Insert Special Ties at Rebar Splice Position

When this option is enabled, the ties at the splice location of the typical rebars in the beam longitudinal profiles are given special consideration.

In accordance with the code, for structures with special ductility, the tie spacing at the splice location of the typical rebars must not exceed the smaller of 10 cm or $d/4$. For this purpose, this option is enabled by default for the special ductility condition, so that the software automatically reduces the tie spacing at the typical rebar splice location.

Beam Plan Configuration Palette

This palette contains the parameters for drawing the location plans in the beam structural drawings as longitudinal profiles. The parameters available in this palette are fully described below.



■ Text Height of Beam Profile Details

The text height, or font size, of the beam detail annotations in the location plan.

■ Profile Name Label Position (⊙ Top, ○ Middle)

This option controls the placement of the profile name on the beams in the location plan. You can use this option to specify whether the beam longitudinal profile name is placed above the beam or within the beam depth.

■ Line Weight of Beam Axe Line

This parameter defines the line weight of the beam axis line in the location plan.

■ Text Height of Dimension Line Label

This parameter defines the text height, or font size, of the dimension line annotations.

☑ Draw Axis Grid Line in the Beam Plans

This option allows you to show or hide the structural grid lines in the location plan.

☑ Offset Beam Width on Beam Profile Type Plan

This option allows you to specify whether the beam is drawn with its edge lines, or as a single line along the beam axis.

☑ Draw Shear Wall in the Beam Profile Plan

This option allows you to show or hide the shear walls in the location plan.

☑ Draw Floor Direction in the Beam Profile Plan

This option allows you to show or hide the floor direction icon in the location plan.

☑ Draw Deck Transverse Ribs in the Beam Profile Plan

This option allows you to show or hide the floor ribs in the location Plan.

☑ Insert Beam Dimension Below the Beams

This option allows you to show or hide the beam dimension text in the location plan.

☑ Trim Beam Lines at the Intersection Point

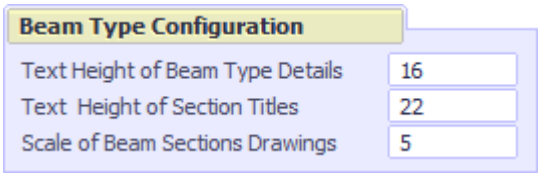
This option allows you to enable or disable the automatic trimming of beam edge lines at intersections in the location plan.

☑ Extend Beam Lines at the Intersection Point

This option allows you to enable or disable the automatic extension of beam edge lines at intersections in the location plan.

Beam Type Configuration Pallet

This palette contains the settings for drawing the beam section types. The parameters in this section are described below.



■ Text Height of Beam Type Details

The text height, or font size, of the beam type detail annotations on the sections.

■ Text Height of Section Titles

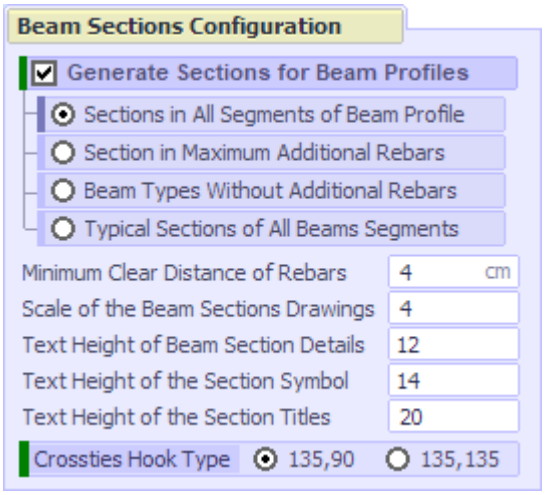
The text height, or font size, of the beam section type titles.

■ Scale of Beam Section Drawings:

The scale for drawing the beam section types.

Beam Section Configuration Pallet

This palette contains the settings for drawing the sections on the beam profiles. The parameters in this section are fully described below.



☑ Generate Sections for the Beam Profiles

When this option is enabled, cross-sections of the beam longitudinal profile are automatically generated and drawn alongside the beam longitudinal profiles. Four different practical section generation methods are defined in the software, as follows:

- **Sections in All Segments of Beam Profile**
- **Sections in Maximum Additional Rebars**
- **Beam Type Additional Rebars**
- **Typical Sections of All Beams Segment**

Based on the characteristics of your project, you can select one of these methods for the automatic generation of beam profile sections. These four methods are described in the following.

Section in All Segments of Beam Profile

In this method, the software automatically generates a cross-section for each segment of the beam profile, including the typical and additional rebar arrangement, and places them alongside the beam longitudinal profile.

Section in Maximum Additional Rebars

In this method, the software automatically generates a cross-section at the location of the maximum rebar concentration, including the typical and additional rebar arrangement, and places it alongside the beam longitudinal profile.

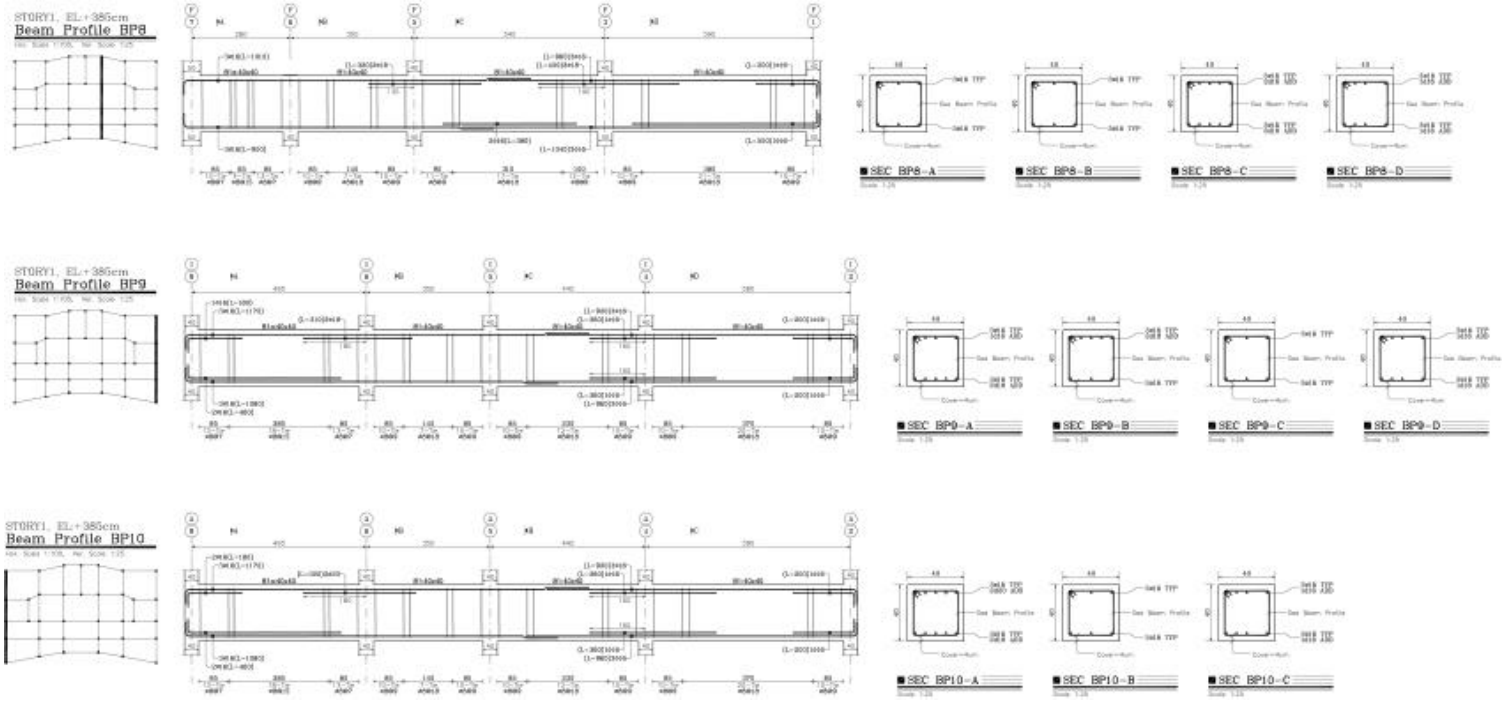
Beam Types Without Additional Rebars

In this method, the software automatically draws the beam types used in the profiles alongside the beam longitudinal profile. Note that in this method, only the typical rebars are drawn, without the additional rebars.

Typical Sections of All Beam Segments

In this method, the software automatically generates cross-sections for each segment of all beam profiles, including the typical and additional rebar arrangement. After classifying the sections, it places all of them below the beam longitudinal profiles. This method is similar to the first method, with the difference that in this method, the beam sections of the entire story are classified, and instead of displaying them adjacent to each beam profile, they are displayed at the end of the story's beam profiles.

A sample of the beam longitudinal profile cross-sections generated using the first method is shown in the image below.



■ Minimum Clear Distance of Rebars

This parameter defines the minimum permissible clear distance between the rebars present in the beam section. Note that this applies to both the typical rebars and the additional rebars.

This parameter has a significant effect on the rebar arrangement in the cross-sections and should be set based on the maximum aggregate size of the concrete, code considerations, and user preference. If the distance between the rebars in the section falls below the value of this parameter, the software automatically moves the rebars to the layer below to satisfy the minimum spacing defined by this parameter. Based on this parameter, the software automatically determines the most appropriate and symmetrical arrangement of the typical and additional rebars in each beam section, while maintaining the fixed position of the typical rebars along the entire length of the beam profile, and performs the corresponding drawings accordingly. To distinguish the typical rebars from the additional rebars in the beam sections, the software displays the typical rebars as solid circles and the additional rebars as hollow circles.

Note that this parameter also affects the calculation of the development length of hooked bars, as described in the relevant section.

■ Scale of Beam Section Drawings

This parameter defines the scale for drawing the beam longitudinal profile cross-sections alongside the beam profiles.

■ Text Height of Beam Section Details

This parameter defines the text height, or font size, of the cross-section detail annotations alongside the beam profiles.

■ Text Height of Section Symbol

This parameter defines the text height, or font size, of the cross-section detail annotations alongside the beam profiles.

■ Text Height of the Section Titles

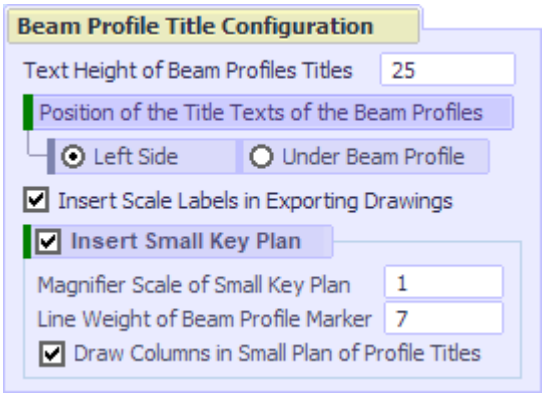
This parameter defines the text height, or font size, of the cross-section titles alongside the beam profiles.

■ Crossties Hook Type

This option allows you to specify whether the beam crosstie hooks are drawn with 135-degree bends at both ends, or with a 135-degree bend at one end and a 90-degree bend at the other.

Beam Profile Title Configuration Pallet

This palette contains the settings for the longitudinal profile title and the small key plan of the beam longitudinal profiles. The parameters in this section are fully described below.



■ Text Height of Beam Profile Titles

The text height, or font size, of the beam longitudinal profile titles.

■ Position of the Beam Profile Title

◉ Left Side

Places the title and the small key plan to the left of the profile.

○ Under Beam Profile

Places the title and the small key plan below the longitudinal profile.

☑ Insert Scale Labels in Exporting Drawings

This option allows you to show or hide the scale annotation in the beam longitudinal profile drawings.

☑ Draw Small Key Plan

This option allows you to show or hide the small key plan in the beam longitudinal profile drawings. When this option is enabled, the following parameters and options related to the settings of the beam longitudinal profile key plan are also activated, and are described below:

■ Magnifier Scale of Small Key Plan

This option allows you to define the magnification factor of the small key plan in the beam longitudinal profile drawings.

■ Line Weight of Beam Profile Marker

This option allows you to define the line weight of the highlight indicating the position of the beam longitudinal profile in the profile location key plan.

☑ Draw Columns in Small Plan of Profile Title

This option allows you to show or hide the columns in the small key plan.

In all of the software's structural drawing generation interfaces, you will find the following settings under the **LIST** section, which pertain to the numbering of element rebars based on their position in the structure, also referred to as the **Position Number Label**, as well as the generation of rebar schedule tables alongside the structural drawings. These are described below.

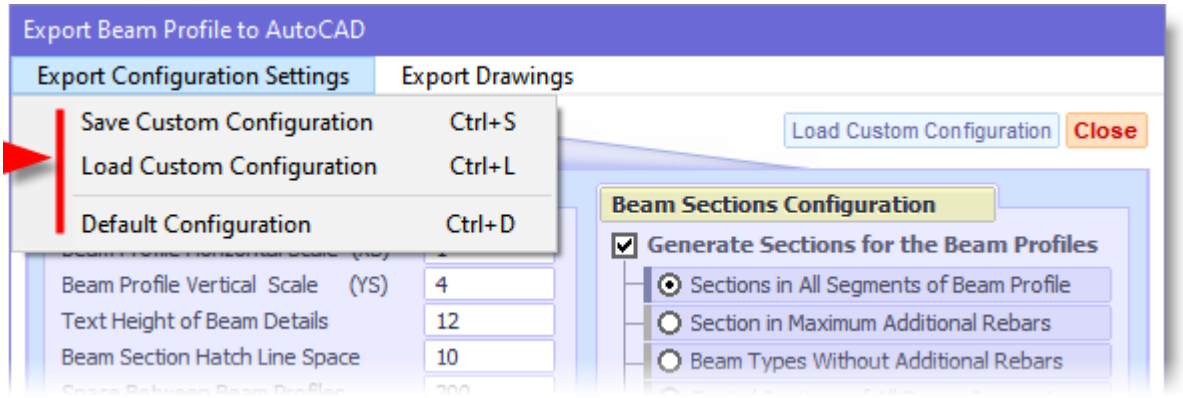
☐ Show Position Number Label on Rebars:

A position number is automatically assigned to each rebar drawn in the structural drawings. When this option is enabled, this position number is also annotated on the corresponding rebar alongside the other details, allowing you to easily locate the desired rebar in the rebar schedule tables.

☑ Show Rebar List Tables on Drawings:

When this option is enabled, the rebar schedule tables are automatically generated alongside the structural drawings. These tables are presented separately by story, element, and rebar type.

Through the **Configuration Setting** menu at the top of this interface, you can save or restore your customized settings, or reset all parameters in this interface to the software's default values.



Exporting Beam Profiles Drawings

To generate the beam longitudinal profiles, the location plan along with the sections, and the corresponding rebar schedules, proceed as follows:

Export Menu > Export Beam Profile Drawings to AutoCAD (Ctrl + F3)

Alternatively, clicking the icon B→CAD in the main toolbar at the top of the software opens the beam longitudinal profile drawing configuration interface, through which you can generate the structural beam details as longitudinal profiles.

At the bottom of the **Export Beam Profile to AutoCad** interface, three buttons are provided for generating the beam profile structural drawings, which are described below:

C CAD Export Current Beam Profile Only

Generate structural drawings for only the selected beam profile in the graphical view.

S CAD Export Beam Profiles in Current Story

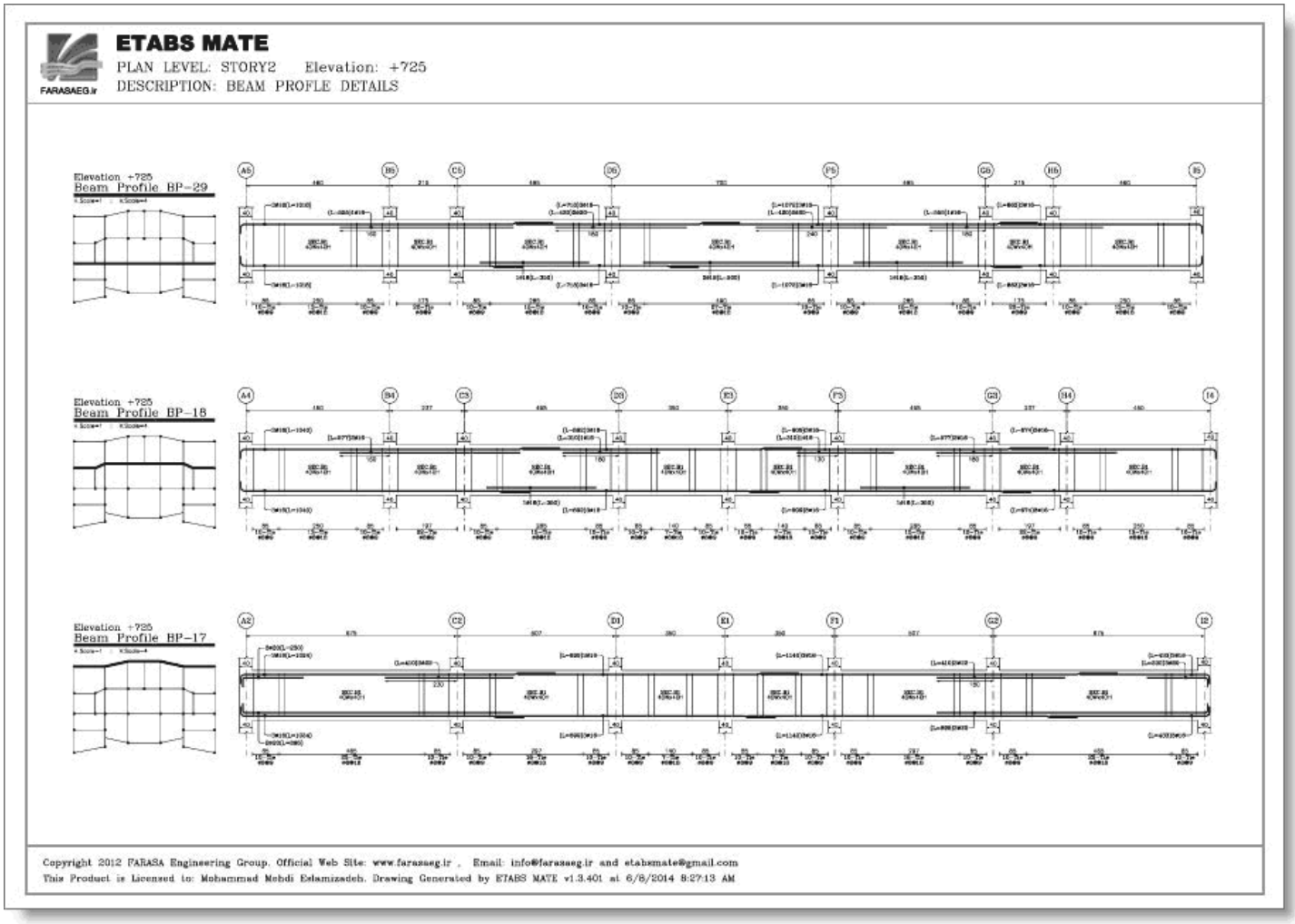
Generate structural drawings for all beam profiles on the active story in the graphical view.

A CAD Export All Beam Profiles in Total Stories

Generate structural drawings for all longitudinal profiles across all stories, i.e., generate all beam profiles of the structure.

☑ Open Drawings in AutoCAD After Generating

When this option is enabled, upon completion of the drawing generation, the software automatically opens the drawing in AutoCAD. For this purpose, the default Windows application for opening DXF files must be set to AutoCAD.



ETABS MATE
Concrete Structure Detailing Software

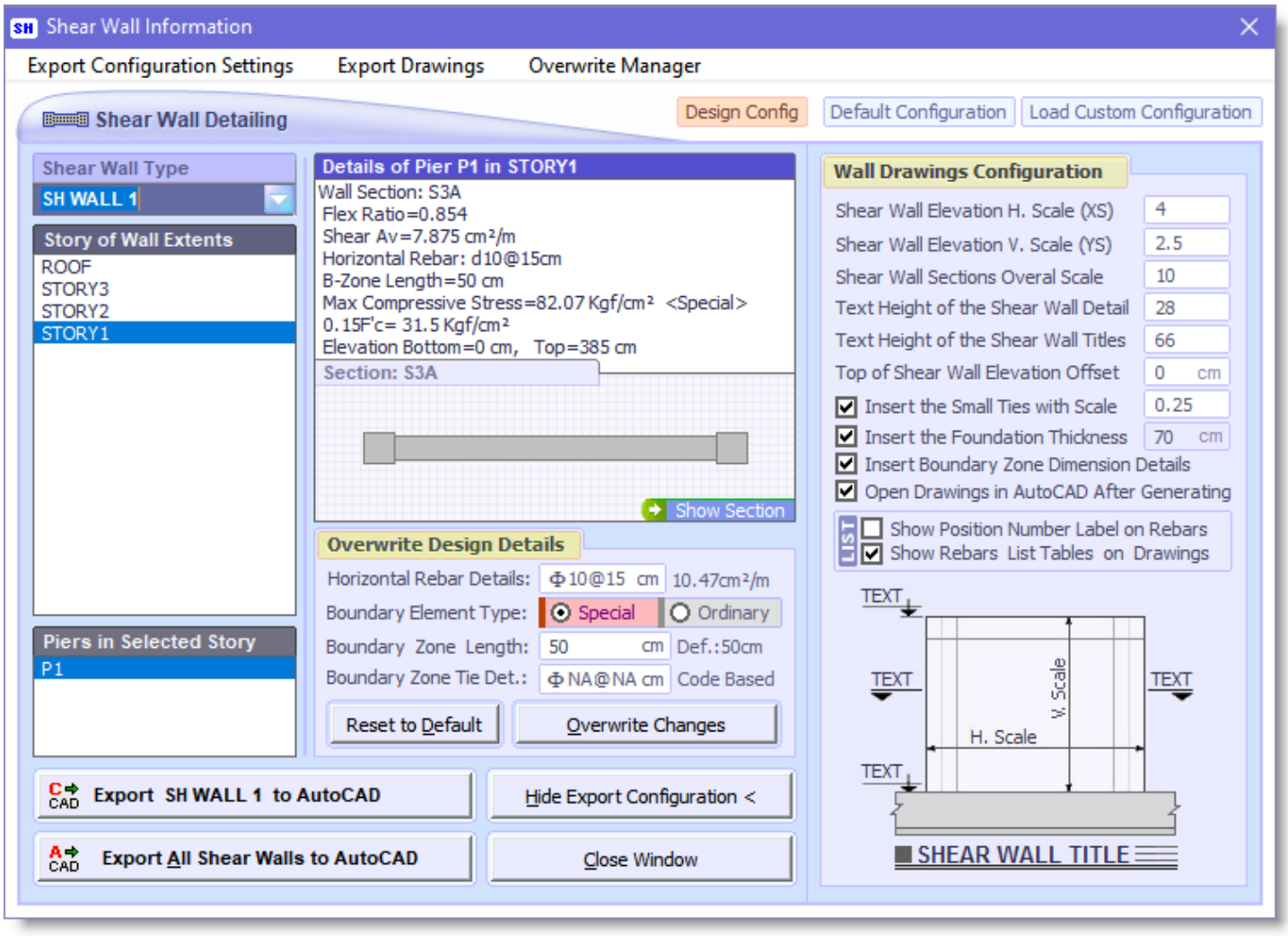
Shear Wall Drawing Configuration

To generate the shear wall structural drawings, including the wall elevation profile and its sections with complete rebar detailing and geometric properties, use the icon S→CAD in the main toolbar at the top of the software, or proceed as follows via the **Export** menu:

Export Menu > Export Shear Wall Drawings to AutoCAD (F4)

Upon executing the drawing command, the shear wall drawing interface will appear, as shown in the image below.

In this interface, you can edit the details designed by the software for various parts of each shear wall. You can also configure the shear wall drawing parameters if necessary, and then generate the wall structural drawings.



The drawing parameters of this interface are described in the following.

ETABS MATE
Concrete Structure Detailing Software

■ Shear Wall Elevation H. Scale (XS)

The horizontal scale for drawing the shear wall elevation view.

■ Shear Wall Elevation V. Scale (YS)

The vertical scale for drawing the shear wall elevation view.

■ Shear Wall Section Overal Scale

The overall scale for drawing the shear wall sections.

■ Text Height of the Shear Wall Details

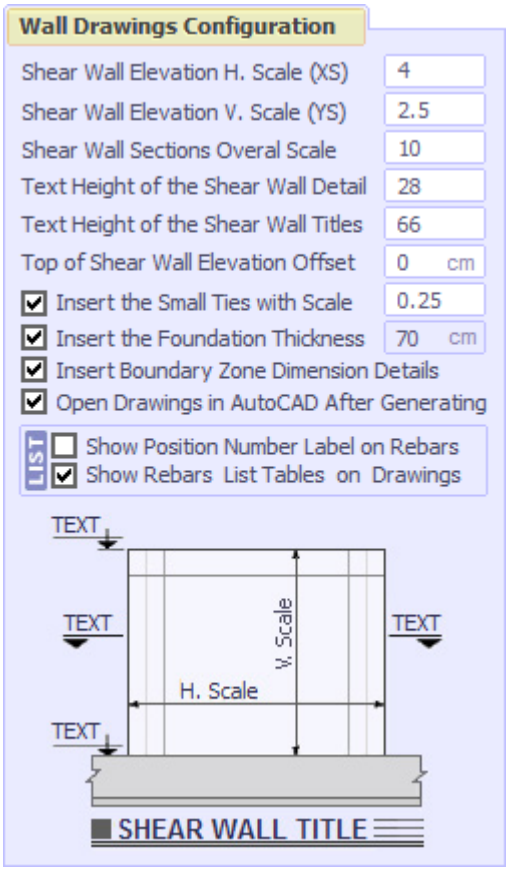
The text height, or font size, of the shear wall detail .

■ Text Height of the Shear Wall Titles

The text height, or font size, of the shear wall titles.

■ Top of Shear Wall Elevation Offset

The reduction or increase of the top-of-wall elevation annotation placed on the profile.



☑ Insert the Small Ties with Scale ...

Draws small ties on the tie details with the magnification factor specified in this section, for a better understanding of the closed tie and crosstie arrangement of the wall.

☑ Insert the Foundation Thickness ...

Specifies whether the foundation thickness dimension line is drawn below the wall, and provides the option to define the foundation thickness of the structure.

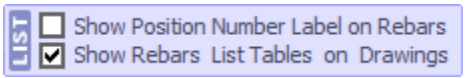
☑ Insert Boundary Zone Dimension Details

Specifies whether the boundary element length dimension lines are drawn on the shear wall sections.

☑ Open Drawing in AutoCAD After Generating

Opens the structural drawing in AutoCAD upon completion of the drawing generation by the software.

In all of the software's structural drawing generation interfaces, you will find the following settings under the **LIST** section, which pertain to the numbering of element rebars based on their position in the structure, also referred to as the **Position Number Label**, as well as the generation of rebar schedule tables alongside the structural drawings. These are described below:



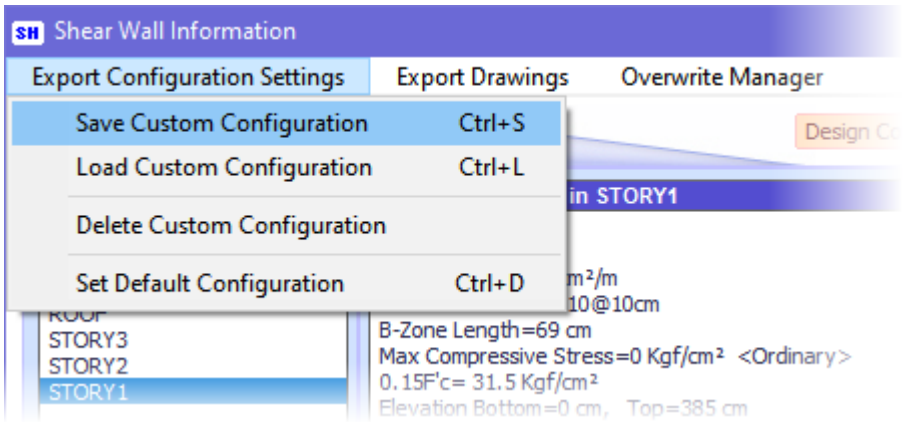
☐ Show Position Number Label on Rebars:

A position number is automatically assigned to each rebar drawn in the structural drawings. When this option is enabled, this position number is also annotated on the corresponding rebar alongside the other details, allowing you to easily locate the desired rebar in the rebar schedule tables.

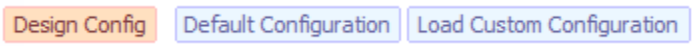
☑ Show Rebar List Tables on Drawings:

When this option is enabled, the rebar schedule tables are automatically generated alongside the structural drawings. These tables are presented separately by story, element, and rebar type.

Furthermore, through the **Configuration Setting** menu at the top of this interface, you can save or restore your customized settings, or reset all parameters in this interface to the software's default values.



Note that at the top of this interface, as shown in the image below, buttons are provided for quick access to certain commands. These are described below:



Load Custom Configuration: Allows you to load your customized settings

Default Configuration: Allows you to restore the software's default settings,

Design Config: Provides convenient access to the shear wall design configuration interface.

Two buttons are provided in this interface for generating the wall structural drawings, which are described in the following.

ETABS MATE
Concrete Structure Detailing Software

Exporting Shear Wall Drawings

To generate the shear wall structural drawings, including the wall elevation profile and its sections with complete rebar detailing and geometric properties, use the icon S→CAD in the main toolbar at the top of the software, or proceed as follows via the **Export** menu:

Export Menu > Export Shear Wall Drawings to AutoCAD (F4)

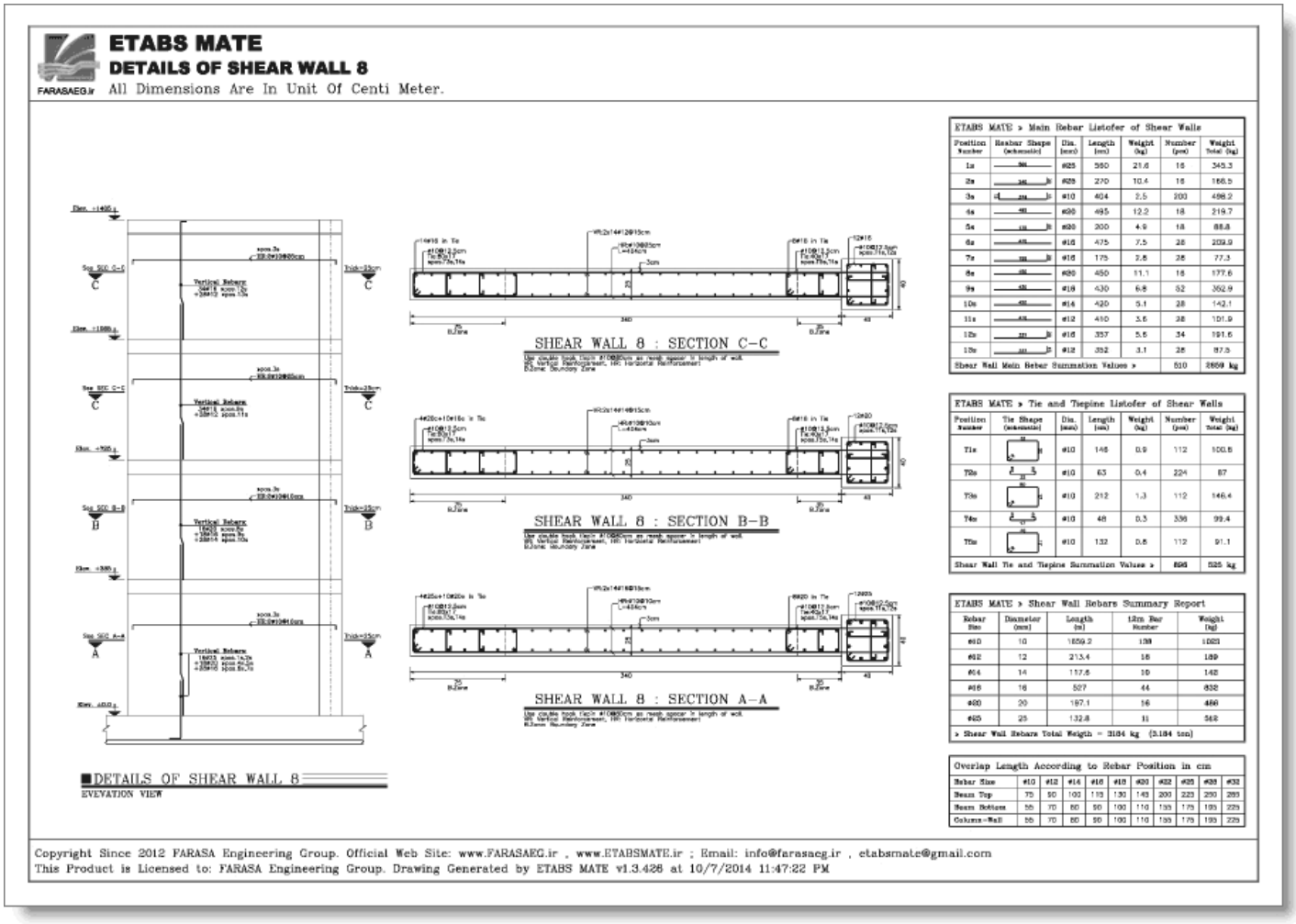
Two buttons are provided in this interface for generating the wall structural drawings, which are described in the following.

C→CAD Export SH WALL 1 to AutoCAD

This button allows you to generate the structural drawings for only the selected shear wall.

A→CAD Export All Shear Walls to AutoCAD

This button allows you to automatically generate the structural drawings for all shear walls of the structural project, as shown in the image below, and save them in a file in the standard AutoCAD drawing format.

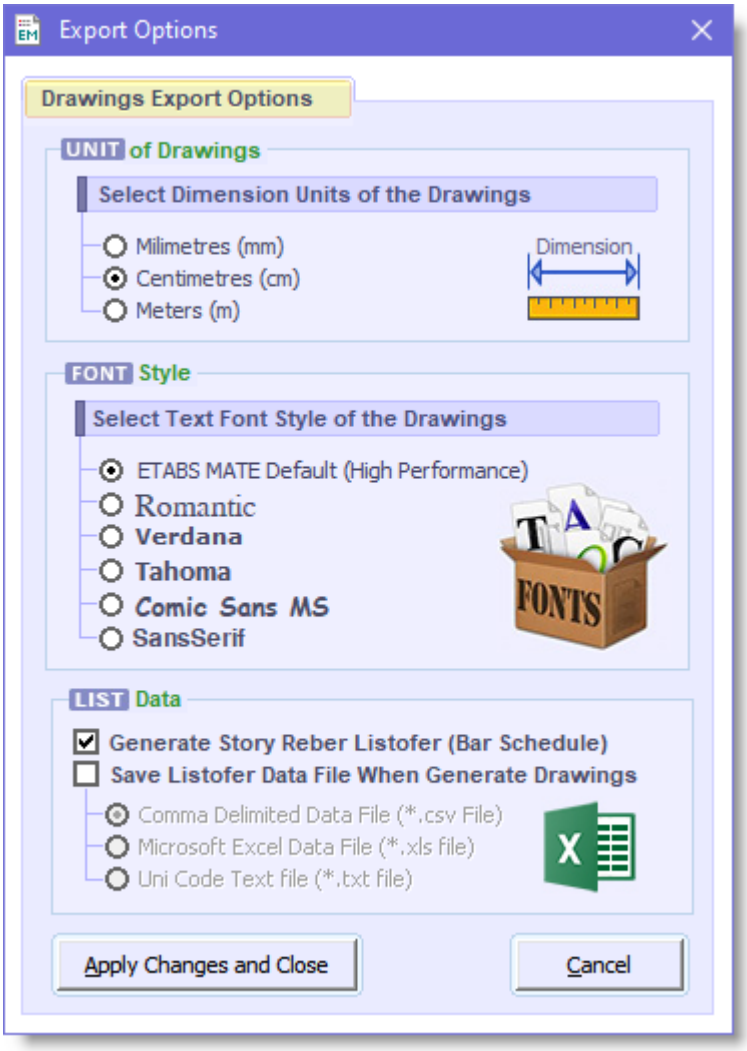


Export Options

You can easily select the drawing unit for the structural drawings and configure certain outputs, such as the Excel output, within the software. To do so, proceed as follows:

Export Menu > Option of Drawings Export

Executing this command opens the export options interface, as shown in the image below. The parameters of this interface are described below.



Units of Drawing

The first part of this interface pertains to setting the drawing unit for the drawings.

From the **UNIT of Drawings** section, As shown in the image, you can select millimeters, centimeters, or meters as the unit for generating your structural drawings.

Font Style of Drawings

The second part of this interface pertains to the font settings for the structural drawing text annotations.

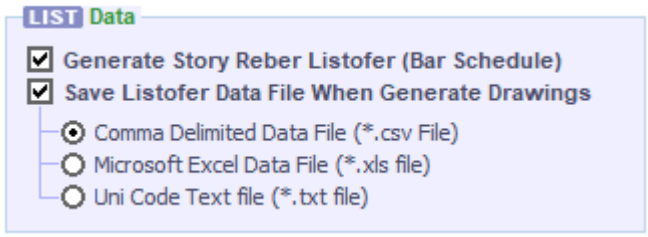
As shown in the image, from the **FONT Style** section, you can select your desired font for the structural drawing text annotations from the list of fonts suggested by the software.

Noting that the **ETABS MATE Default** font, which is selected by default, provides the highest drawing generation speed and is recommended for use in large projects.

Note that the font of the text annotations generated by the software is defined as a **Text Style** in AutoCAD. Given this feature, you can also apply your desired font in AutoCAD by changing the font in the **Text Style** settings.

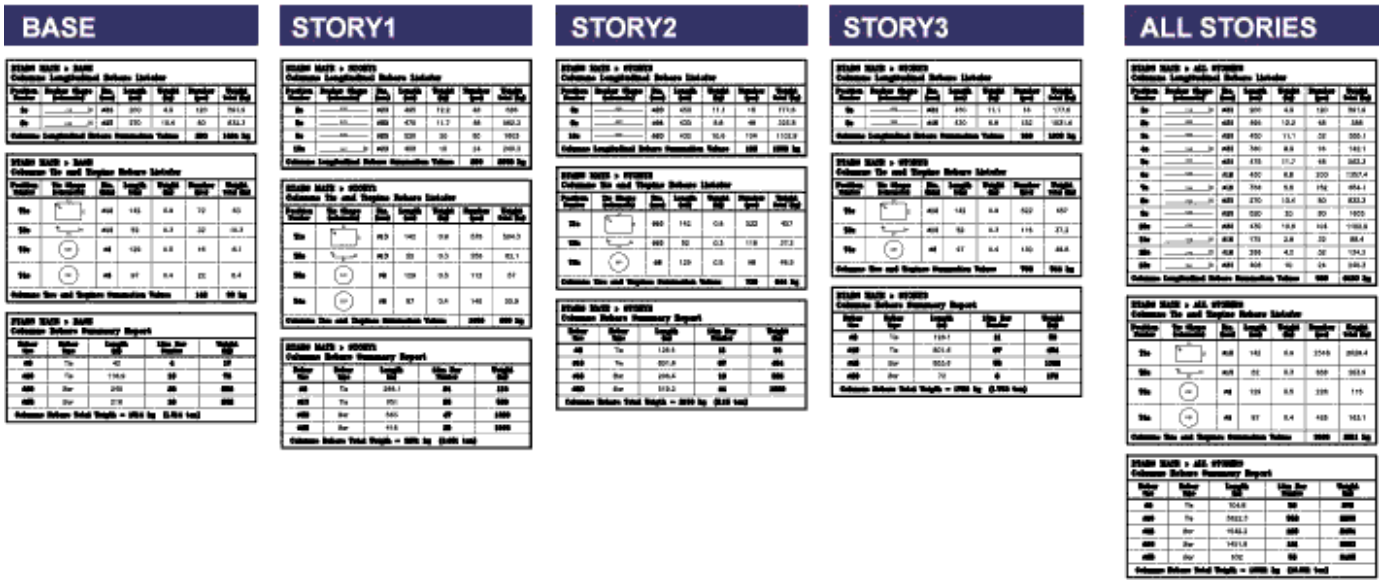
Rebar Schedule Parameters

The third part of this interface pertains to the rebar schedule settings and the Excel output of the structural material data. The parameters in this section are described below.



☑ Generate Story Rebar Listofer (Bar Schedule)

In this section, you can specify whether, in addition to the complete structural rebar schedule, story-level rebar schedules are also calculated and placed alongside the structural drawings. When this option is enabled, as shown in the image below, the story-level rebar schedules are also calculated and inserted alongside the drawings.



☐ Save Listofer Data File When Generate Drawings

By enabling this option, you can specify that the rebar schedule data, in addition to being placed as a table alongside the structural drawings, is also saved as a separate data file, for example, as an Excel file.

When this option is enabled, each time a structural drawing command is executed, a data file containing the rebar schedule tables is also saved with the same name as the structural drawing and in the same location as the AutoCAD drawing file. Furthermore, as shown in the image, you have several options for the file format in which the rebar schedule tables are saved, including saving as an Excel file or as a text file. You may use either based on your requirements.

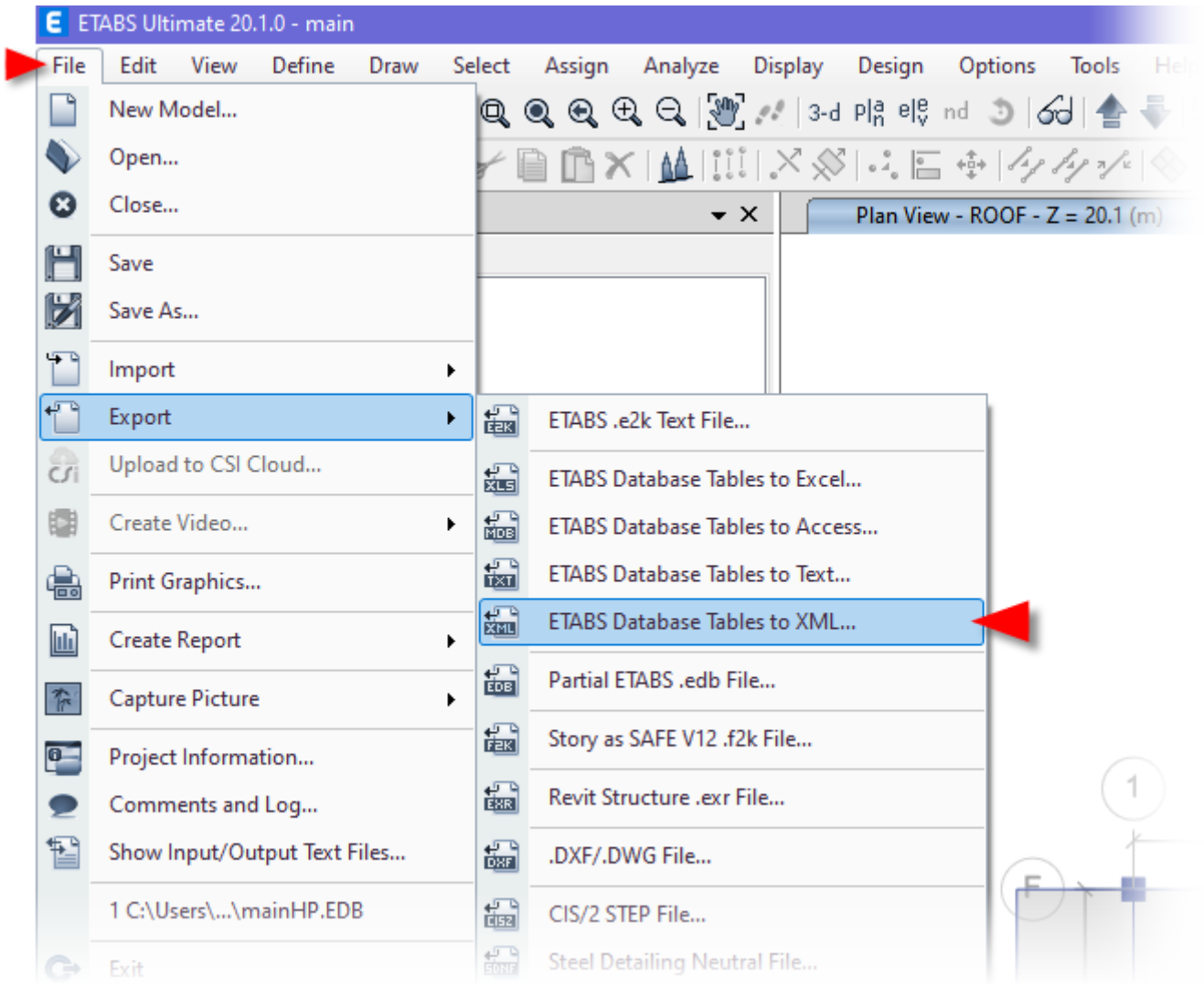
High Pressure Column Detailing Provisions

If the structural frame is designed for special ductility, specific provisions must be applied to the transverse reinforcement design of High Pressure columns.

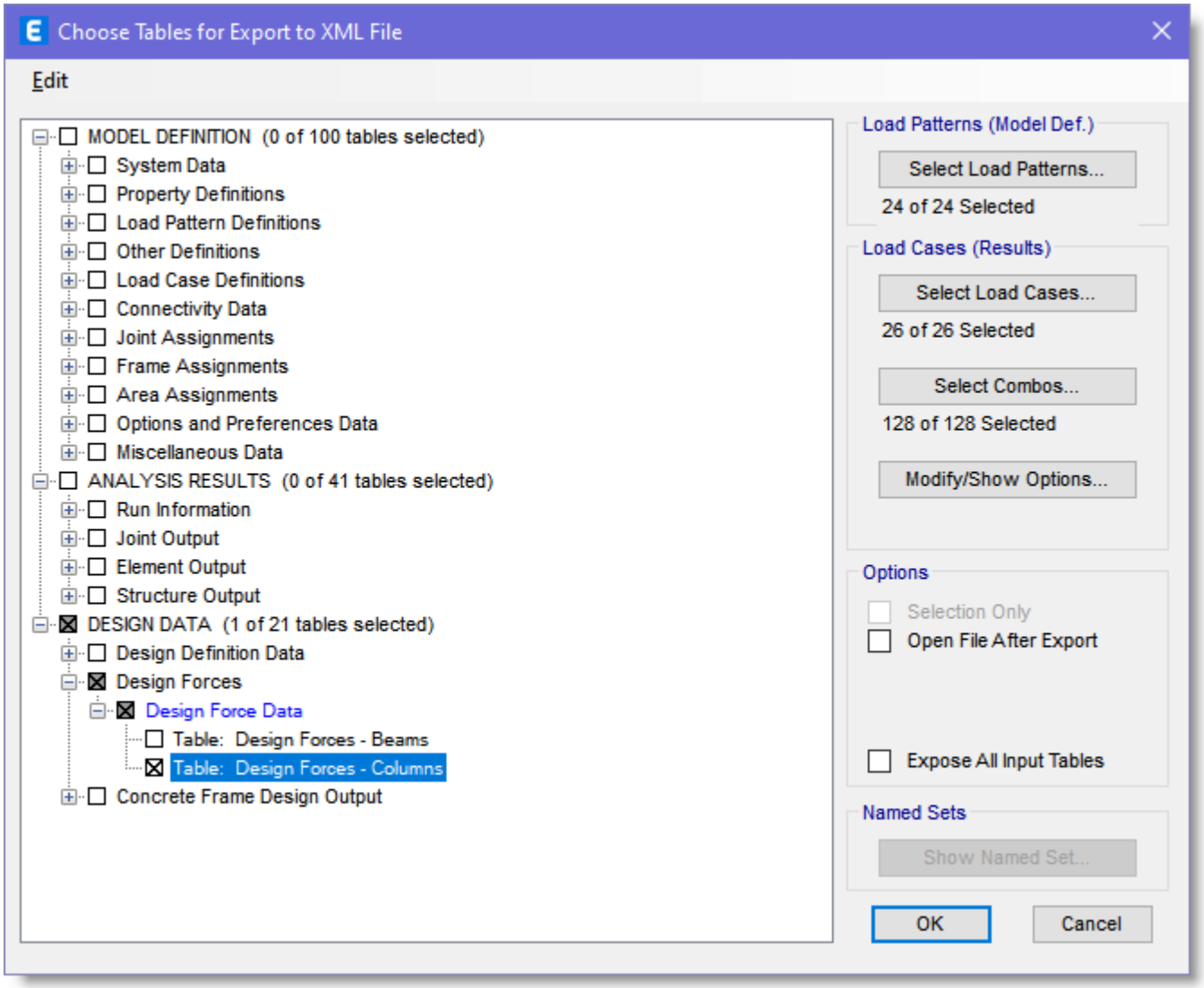
In accordance with the ACI code, High Pressure columns are defined as columns in which the compressive force $P_u > 0.3\ Ag\ F'c$, or in which $F'c > 70\ MPa$. These columns must first be identified, and then their specific provisions must be applied. Detailed explanations in this regard are provided below.

Automatic Identification of High Pressure Columns

First, the High Pressure columns must be identified by the program. To perform this process, the **Column Design Forces** table must first be exported from the ETABS software as an **XML** file. To do this, from the **File** menu in the ETABS software, first select **Export**, and then select **ETABS Database to XML ... ,** as shown in the image below.



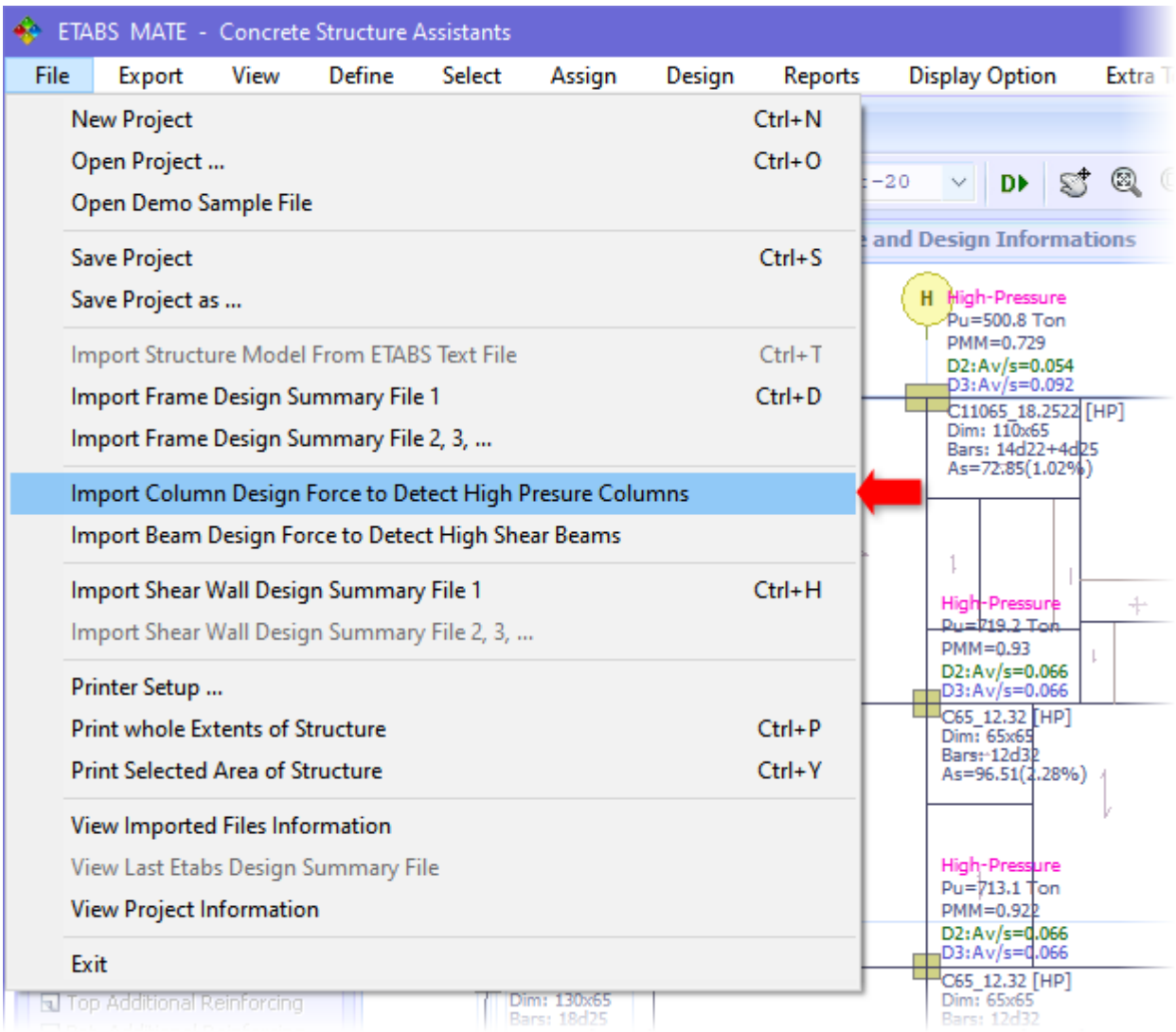
Then, as shown in the image below, in the **Choose Table for Export to XML File** interface, select only the **Design Forces – Columns** table (or the **Column Design Forces** table in earlier versions of ETABS software) and save the file containing the column design forces with **Kgf, cm** units at your desired location.



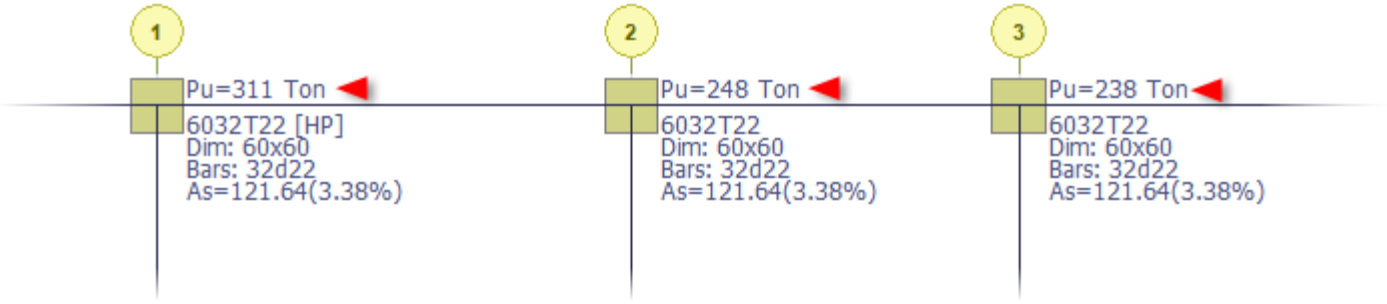
Now, this file containing the column design forces must be imported into **ETABS MATE**. To do this, from the **File** menu, select **Import Column Design Force to Detect High Pressure Columns**, and then select the **XML** file containing the column design forces that was saved in the previous step.

Note that this option in the **File** menu is only active when the earthquake resistance type of the structure was set to **Special Moment Frame** at the beginning of the structural geometry import into the software.

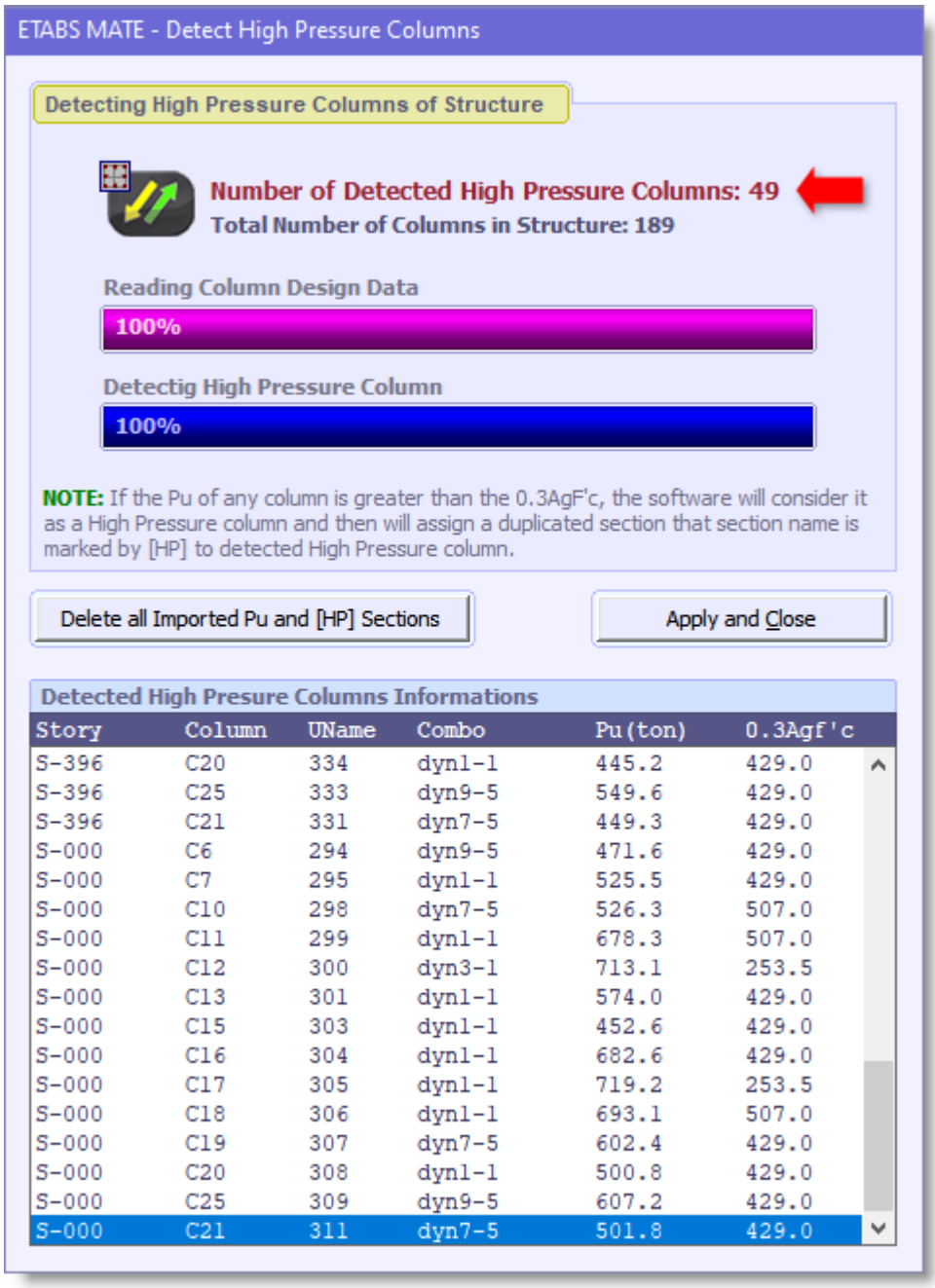
File Menu > Import Column Design Force to Detect High Pressure Columns



After importing the data file containing the Column Design Forces using the option shown in the image above, the software automatically determines the maximum compressive force P_u for each column at each story of the structure from among all the load combinations in the data file exported from ETABS, and displays this force on the structural columns, as shown in the image below.



After importing the file containing the column design forces, the software calculates the 0.3AgF'c values for each column and compares them with the maximum compressive force of that column. If $P_u > 0.3AgF'c$ or $F'c > 70MPa$, the software identifies that column as a **High Pressure** column and displays the details of the calculations performed, as shown below. As can be seen in the image, the number of columns identified as **High Pressure** by the software, as well as the total number of columns in the project, are displayed alongside other details in this interface for the user's information.



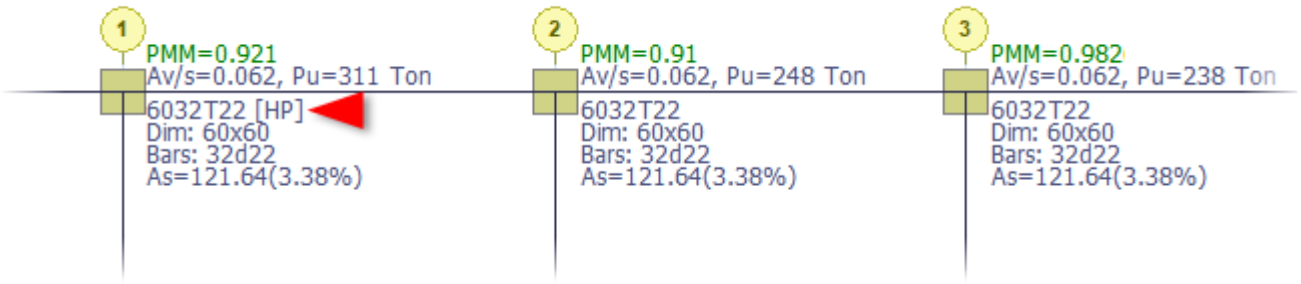
Story	Column	UName	Combo	Pu(ton)	0.3Agf'c
S-396	C20	334	dyn1-1	445.2	429.0
S-396	C25	333	dyn9-5	549.6	429.0
S-396	C21	331	dyn7-5	449.3	429.0
S-000	C6	294	dyn9-5	471.6	429.0
S-000	C7	295	dyn1-1	525.5	429.0
S-000	C10	298	dyn7-5	526.3	507.0
S-000	C11	299	dyn1-1	678.3	507.0
S-000	C12	300	dyn3-1	713.1	253.5
S-000	C13	301	dyn1-1	574.0	429.0
S-000	C15	303	dyn1-1	452.6	429.0
S-000	C16	304	dyn1-1	682.6	429.0
S-000	C17	305	dyn1-1	719.2	253.5
S-000	C18	306	dyn1-1	693.1	507.0
S-000	C19	307	dyn7-5	602.4	429.0
S-000	C20	308	dyn1-1	500.8	429.0
S-000	C25	309	dyn9-5	607.2	429.0
S-000	C21	311	dyn7-5	501.8	429.0

Applying Detailing Provisions for High Pressure Columns

After the software automatically identifies the High Pressure columns, as described in the previous section, it automatically assigns a new section with the suffix **[HP]**, which stands for High Pressure, to these columns.

As shown in the image below, the High Pressure columns, whose section names are marked with the **[HP]** suffix, are easily visible in the structural plan.

This new section has exactly the same properties as the original column section, with the difference that the transverse reinforcement provisions for high pressure columns have also been applied. These provisions are described in the following.



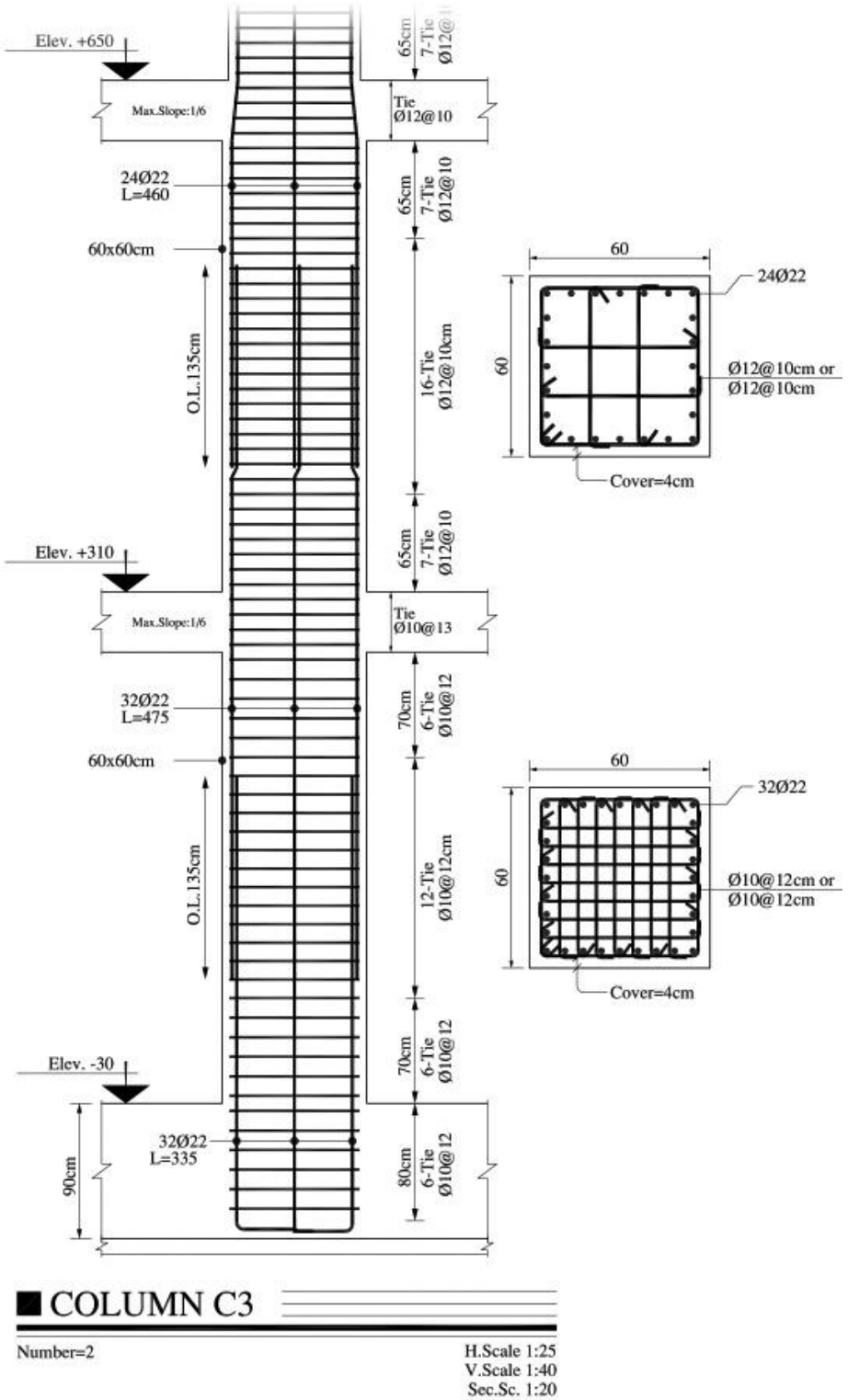
In accordance with ACI provisions, specific requirements must be observed in the design of **High Pressure Columns**. Among them, all rebars in the section of high pressure Columns must be restrained by tie corners or seismic hooks. In addition to satisfying relationships **a** and **b**, which are mandatory for all columns in structures with special ductility, relationship **c** must also be applied in the transverse reinforcement design of these columns. These relationships are provided below. When the software identifies a column as being in a High Pressure state, it automatically calculates all these provisions and applies them in the transverse reinforcement detailing.

a) $$A_{sh}/sb_c > 0.3\left(\frac{A_g}{A_{ch}} - 1\right)\frac{f'_c}{f_{yt}}$$

b) $$A_{sh}/sb_c > 0.09\frac{f'_c}{f_{yt}}$$

c) $$A_{sh}/sb_c > 0.2k_f k_n \frac{P_u}{f_{yt}A_{ch}}$$

A sample drawing generated by the software is shown in the image below. This column has been identified by the software as a **High Pressure column** at the first story and as a normal column at the second story, and the transverse reinforcement details of the column have been calculated accordingly.



ETABS MATE
Concrete Structure Detailing Software

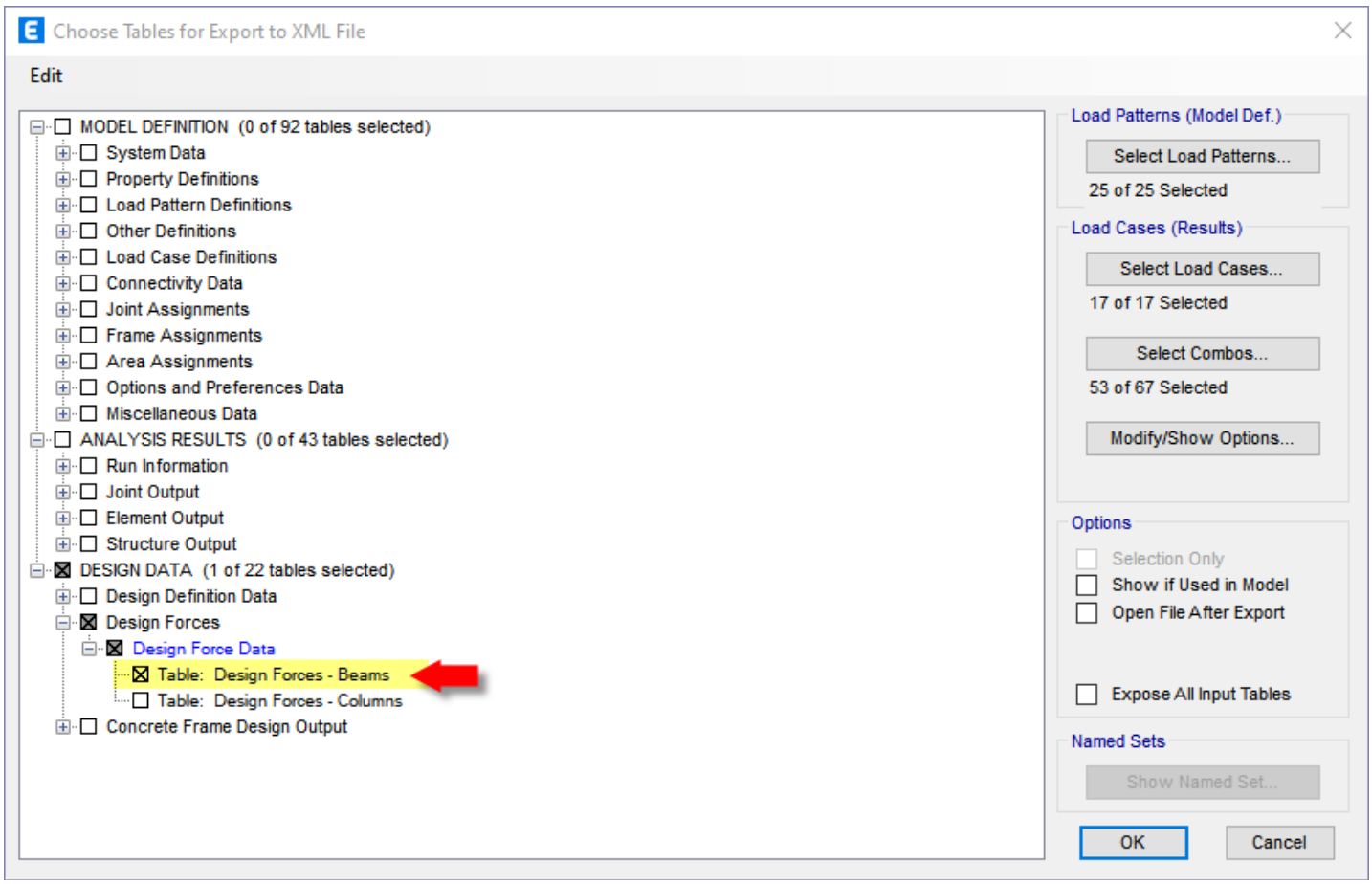
High Shear Beam Detailing Provisions

In accordance with ACI provisions, specific requirements must be observed in the transverse reinforcement design and detailing of **High Shear Beams**. In accordance with the code, **High Shear Beams** are defined as beams in which $V_s > 0.33\sqrt{f'_c}b_w d$. In these beams, the maximum spacing of shear reinforcement along the member length must not exceed the lesser of $d/4$ and **300 mm**, and the maximum spacing of the legs across the section width must not exceed the lesser of $d/2$ and **300 mm**. These beams must first be identified, and then their specific detailing provisions must be applied. Detailed explanations in this regard are provided below.

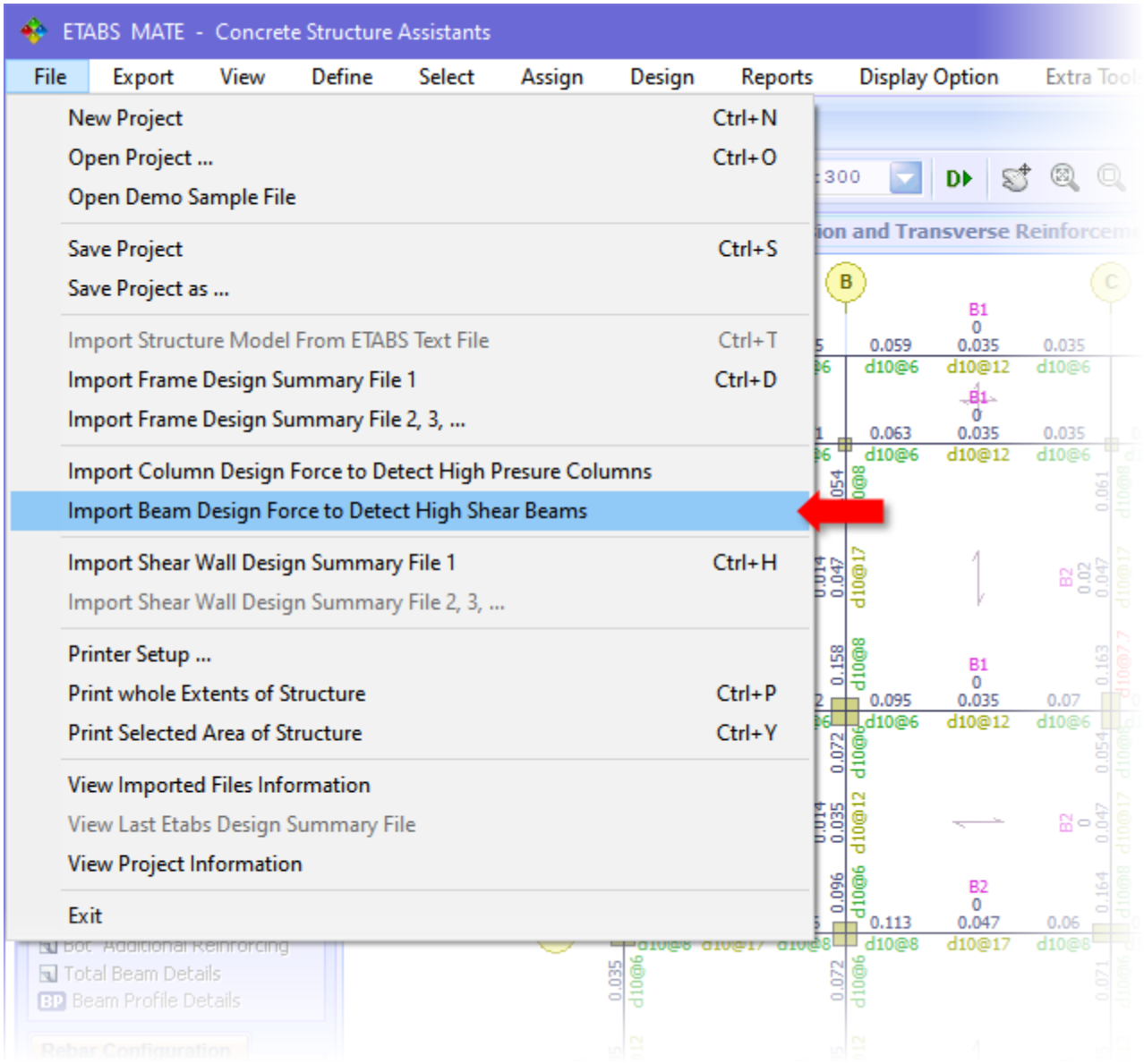
Automatic Identification of High Shear Beams

To identify **High Shear Beams** in structure, the software first automatically calculates the value of $0.33\sqrt{f'_c}b_w d$ for all structural beams and compares it with the V_s value imported from the ETABS software to identify the high shear beams.

To do this, first select **Export Database Tables to XML** from the **File** menu in the ETABS software, and then select the **Design Forces – Beams** table (or the **Beam Design Forces** table in earlier versions of ETABS), as shown in the image below.

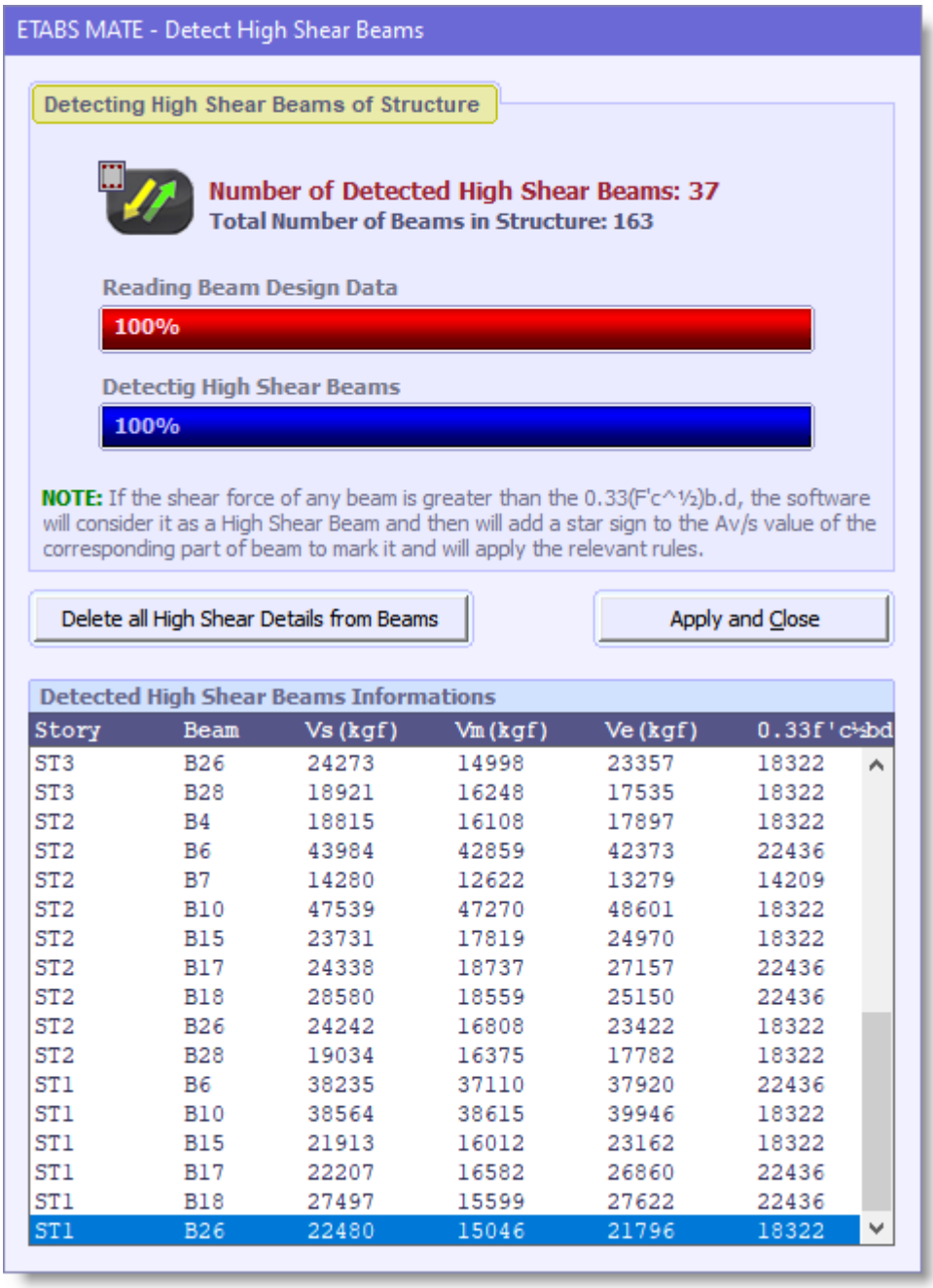


Then, in **ETABS MATE**, after importing the e2k file and the design data file, to automatically identify high shear beams, use **Import Beam Design Force to Detect High Shear Beams** from the **File** menu, as shown in the image below, and select the XML file containing the beam design forces that was saved in the previous step.



After importing the file containing the beam design forces, **ETABS MATE** extracts the beam design data from the corresponding table and then determines the maximum shear force V_s from all the imported load combinations for each of the start, middle, and end stations of the structural beams.

Afterward, the **Detect High Shear Beam** interface will appear, as shown in the image below, providing a report of the beams identified as high shear beams. Pressing the **Apply and Close** button will apply the high shear beam provisions. Note that if you do not wish to apply the high shear beam provisions in the project, pressing the **Delete all High Shear Details from Beam** button will restore all high shear beams to their original state.



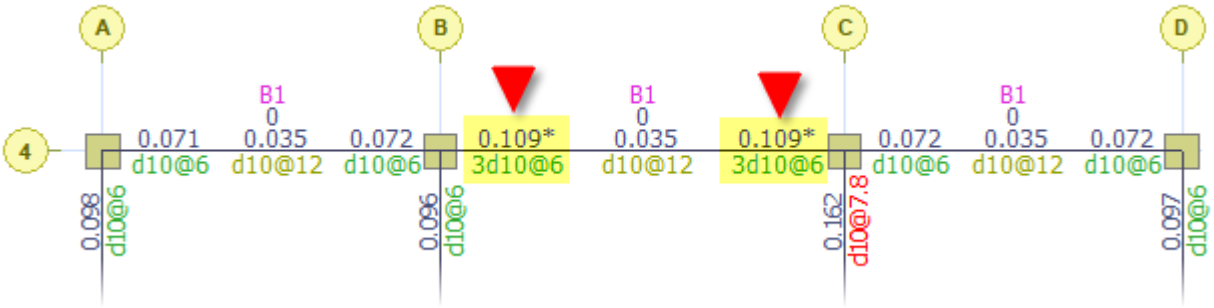
Story	Beam	Vs (kgf)	Vm (kgf)	Ve (kgf)	0.33f'c*bd
ST3	B26	24273	14998	23357	18322
ST3	B28	18921	16248	17535	18322
ST2	B4	18815	16108	17897	18322
ST2	B6	43984	42859	42373	22436
ST2	B7	14280	12622	13279	14209
ST2	B10	47539	47270	48601	18322
ST2	B15	23731	17819	24970	18322
ST2	B17	24338	18737	27157	22436
ST2	B18	28580	18559	25150	22436
ST2	B26	24242	16808	23422	18322
ST2	B28	19034	16375	17782	18322
ST1	B6	38235	37110	37920	22436
ST1	B10	38564	38615	39946	18322
ST1	B15	21913	16012	23162	18322
ST1	B17	22207	16582	26860	22436
ST1	B18	27497	15599	27622	22436
ST1	B26	22480	15046	21796	18322

Applying Detailing Provisions for High Shear Beam

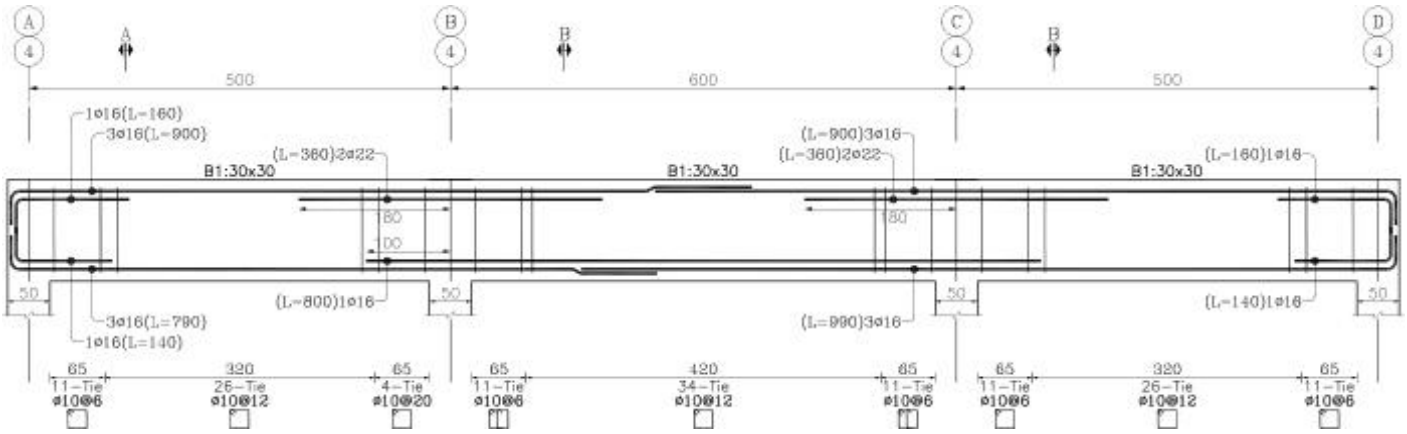
After the software identifies the high shear beams, activating the **Shear – Torsion Tie** layer from the **LAYER DISPLAY** palette appends an asterisk (*) to the end of the A_v/S values on the beams at the locations identified as high shear, as shown in the image below, allowing the user to easily distinguish the high shear beams from the other beams.

In these beams, the maximum spacing of shear reinforcement along the member length must not exceed the lesser of **d/4** and **300 mm**, and the maximum spacing of the legs across the section width must not exceed the lesser of **d/2** and **300 mm**. The software automatically applies these provisions in the beam tie calculation.

As shown in the image below, in the portions of the beam identified as high shear, the tie spacing has been reduced to the minimum of (**d/4 and 300 mm**). Furthermore, to ensure the spacing between legs across the section does not exceed the minimum of (**d/2 and 300 mm**), the number of crossties at the given location has also been automatically increased.



A sample AutoCAD output generated by the software for the detailing of the above beam is shown in the image below, in which the beam dimensions and their tie details are clearly presented.



Hooked Bar Development Length Control

The software calculates the development length of hooked bars for end beams of spam in accordance with ACI 318 provisions, using Sections 25.4.3.1, 25.4.3.2, and 18.8.5.1, based on the maximum values obtained from the following relationships.

$$l_{dh} = \frac{\psi_e \psi_r \psi_o \psi_c}{\lambda} \frac{0.043 f_y}{\sqrt{f_c'}} d_b^{1.5}$$

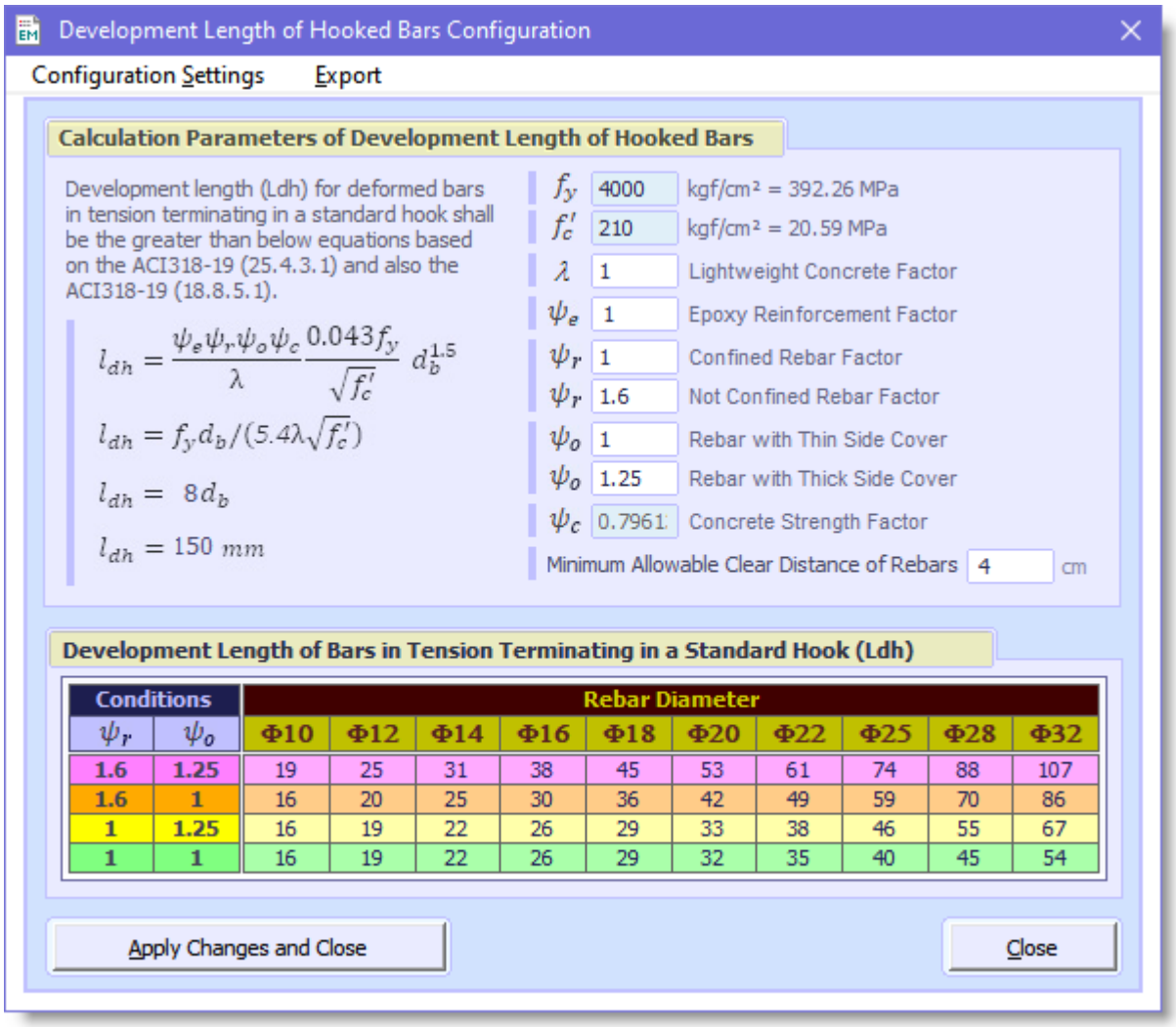
$$l_{dh} = f_y d_b / (5.4\lambda\sqrt{f_c'})$$

$$l_{dh} = 8d_b \quad (10d_b \text{ for lightweight concrete})$$

$$l_{dh} = 150mm \quad (190mm \text{ for lightweight concrete})$$

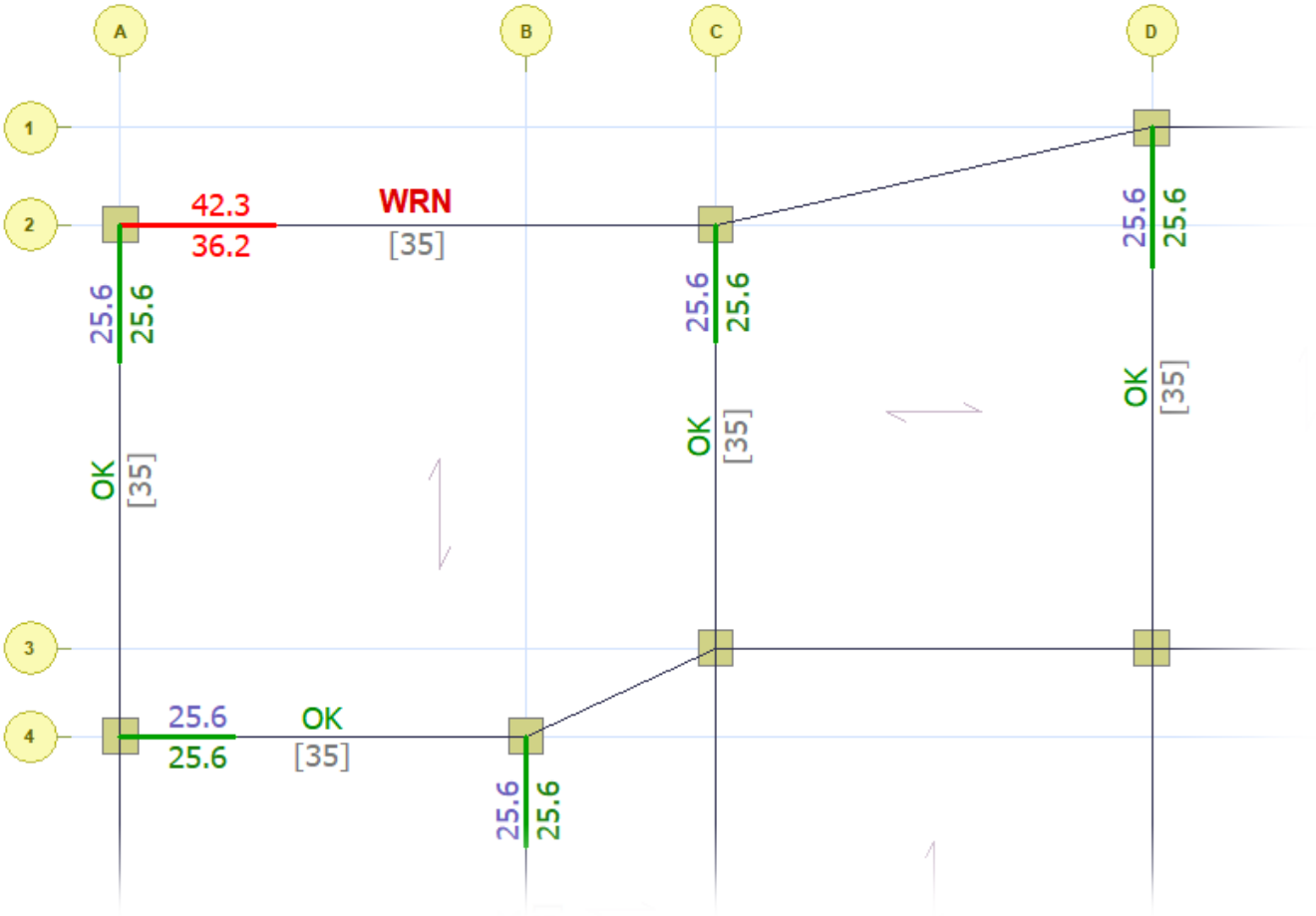
Modification factor	Condition	Value of factor
Lightweight λ	Lightweight concrete	0.75
	Normalweight concrete	1.0
Epoxy ψ_e	Epoxy-coated or zinc and epoxy dual-coated reinforcement	1.2
	Uncoated or zinc-coated (galvanized) reinforcement	1.0
Confining reinforcement ψ_r	For No. 36 and smaller bars with $A_{tr} \geq 0.4A_{hs}$ or s[1] $\geq 6d_b$[2]	1.0
	Other	1.6
Location ψ_o	For No. 36 and smaller diameter hooked bars: (1) Terminating inside column core with side cover normal to plane of hook ≥ 65 mm, or (2) With side cover normal to plane of hook ≥ $6d_b$	1.0
	Other	1.25
Concrete strength ψ_c	For $f_c' < 42$ MPa	$f_c'/105 + 0.6$
	For $f_c' \geq 42$ MPa	1.0

You can configure all parameters for calculating the development length of hooked bars in end beams. To do so, select **Development Length of Hooked Bars Parameter** from the **Design** menu to open the corresponding interface, as shown in the image below. As can be seen, all parameters in this interface are configured by default based on ACI 318, in accordance with the table above, and no modification is required. However, they can be edited if necessary.



Based on the relationships mentioned above and using the parameters defined in this interface, the software automatically calculates the development length of hooked bars in end beams of frame. To view the calculated details, simply select the **Ldh of Beam Hooked Rebar** option from the **LAYER DISPLAY** panel, and the calculated required development length values for the top and bottom hooked bars of the end beams will be displayed on the plan, as shown in the image below.

As shown in the below image, for each beam located at the end of the frame, the required development length for the top hooked end bars is displayed at the top of the beam, and the required development length for the bottom hooked end bars is displayed at the bottom of the beam. Additionally, the development length provided by the column dimension in the beam direction is displayed in the middle of the beam, enclosed within [XX] brackets in gray.



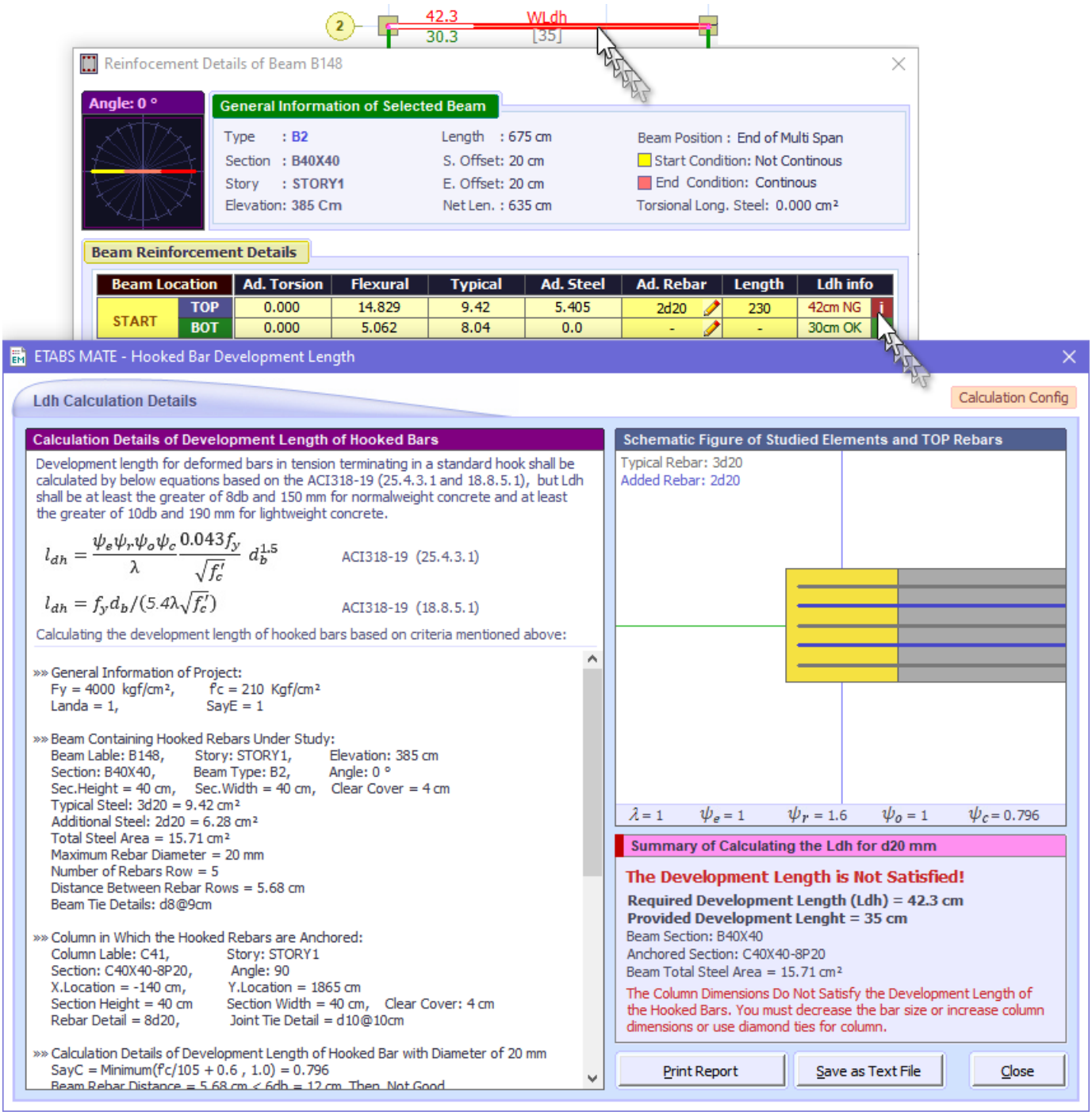
If the calculated development length is less than the length provided by the column dimension in the beam direction, the calculated required development length is displayed in blue for the top bars and in green for the bottom bars, indicating that the development length of the hooked bar is satisfied. Otherwise, the calculated required development length is displayed in red, signifying that the required development length for the hooked end bars of the beam is not satisfied within the column, and a **WRN** warning is also displayed in the middle portion of the beam. Note that by right-clicking on any beam, you can view the detailed calculations of the development length for the hooked end bars, as described in the following.

To view the detailed calculations of the development length for hooked bars in end beams, select **Ldh of Beam Hooked Rebar** or **Total Beam Rebar Details** from the **LAYER DISPLAY** panel, then click on the desired beam to open the beam reinforcement detailing interface, as shown in the image below.

As shown in the image, in the column of **Ldh info**, the calculated development length for the hooked rebars of end beam, on the discontinuous side, is displayed for both the top and bottom beam rebars. Additionally, an icon **i** is activated for each part where the calculation has been performed. Clicking on each icon opens the corresponding interface for viewing the development length details.

It should be noted that that if an Ldh calculation is not required, for example, if the rebar is not an end rebar, the phrase N/A is displayed in this field, indicating that the calculation of this parameter is not applicable.

Clicking on the icon i opens the corresponding interface for viewing the detailed calculation of the hooked bar development length, as shown in the image below. These details include the beam information, the typical and additional beam rebars, the beam ties and dimensions, the information of the column connected to the beam, the calculation details of the coefficients $\psi_e, \psi_r, \psi_o, \psi_c$, the method of calculating A_{th} , A_{hs} and the entire computational procedure leading to the calculation of the hooked bar development length, all of which can be reviewed by scrolling through the interface.



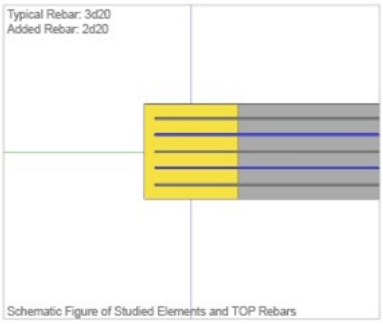
It should be noted that you can print the detailed calculations of the hooked bar development length as a complete report, and use it for verification or inclusion in the calculation book. To do so, use the **Print Report** button in the hooked bar development length details interface to generate the graphical report shown below, which is printed directly by the software.

ETABS MATE
Sample Structure > Development Length of Hooked Bars Calculation Report

Development length for deformed bars in tension terminating in a standard hook shall be calculated by below equations based on ACI318-19 (25.4.3.1 and 18.8.5.1) but Ldh shall be at least the greater of 8db and 150 mm for normalweight concrete and at least the greater of 10db and 190 mm for lightweight concrete.

$$l_{dh} = \frac{\psi_e \psi_r \psi_o \psi_c}{\lambda} \frac{0.043 f_y}{\sqrt{f_c'}} d_b^{1.5} \qquad \text{ACI318-19 (25.4.3.1)}$$

$$l_{dh} = f_y d_b / (5.4\lambda\sqrt{f_c'}) \qquad \text{ACI318-19 (18.8.5.1)}$$



Details of calculating the development length of hooked rebar 20mm in TOP position of beam:

»» General Information of Project:

Fy = 4000 kgf/cm², f'c = 210 Kgf/cm²
Landa = 1, SayE = 1

»» Beam Containing Hooked Rebars Under Study:

Beam Lable: B148, Story: STORY1, Elevation: 385 cm
Section: B40X40, Beam Type: B2, Angle: 0 °
Sec.Height = 40 cm, Sec.Width = 40 cm, Clear Cover = 4 cm
Typical Steel: 3d20 = 9.42 cm²
Additional Steel: 2d20 = 6.28 cm²
Total Steel Area = 15.71 cm²
Maximum Rebar Diameter = 20 mm
Number of Rebars Row = 5
Distance Between Rebar Rows = 5.68 cm
Beam Tie Details: d8@9cm

»» Column in Which the Hooked Rebars are Anchored:

Column Lable: C41, Story: STORY1
Section: C40X40-8P20, Angle: 90
X.Location = -140 cm, Y.Location = 1865 cm
Section Height = 40 cm Section Width = 40 cm, Clear Cover: 4 cm
Rebar Detail = 8d20, Joint Tie Detail = d10@10cm

»» Calculation Details of Development Length of Hooked Bar with Diameter of 20 mm

SayC = Minimum(f'c/105 + 0.6 , 1.0) = 0.796
Beam Rebar Distance = 5.68 cm < 6db = 12 cm Then Not Good
Number of Ties d10@10cm in Hook 15db (30cm); N.Ties = 3 >= 2 Then OK
Tie Space = 10 cm < 8db = 16 cm Then OK
Number of Acceptable Leg; N.Legs = 2
Ath = (N.Ties) x (N.Legs) x (Tie Area) = 3 x 2 x 78.54 = 471.24 mm²
0.4Ahs = 0.4 x (Total Steal Area) = 0.4 x 1570.77 = 628.31 mm²
Ath < 0.4Ahs Then Not Good
{ If [(Beam Rebar Distance) >= 6db] OR
[(N.Ties) >= 2 AND (Tie Space) <= 8db AND Ath >= 0.4Ahs]
Then SayR = 1 Else SayR = 1.6 }
Therefore SayR = 1.6
Rebar Side cover = 7 cm
{ If [Side Cover >= Minimum(65mm, 6db)] Then SayO = 1 Else SayO = 1.25 }
Side Cover = 7 cm >= 6.5 cm Then SayO = 1
So in summary we will have:
SayE = 1 , SayR = 1.6 , SayO = 1 , SayC = 0.796
Therefore, according to ACI318-19 (25.4.3.1 and 18.8.5.1):
Required Development Length (Ldh) = 42.3 cm
Based on the geometry of the model:
Provided Development Lenght = 35 cm

Required Development Length > Provided Development Lenght

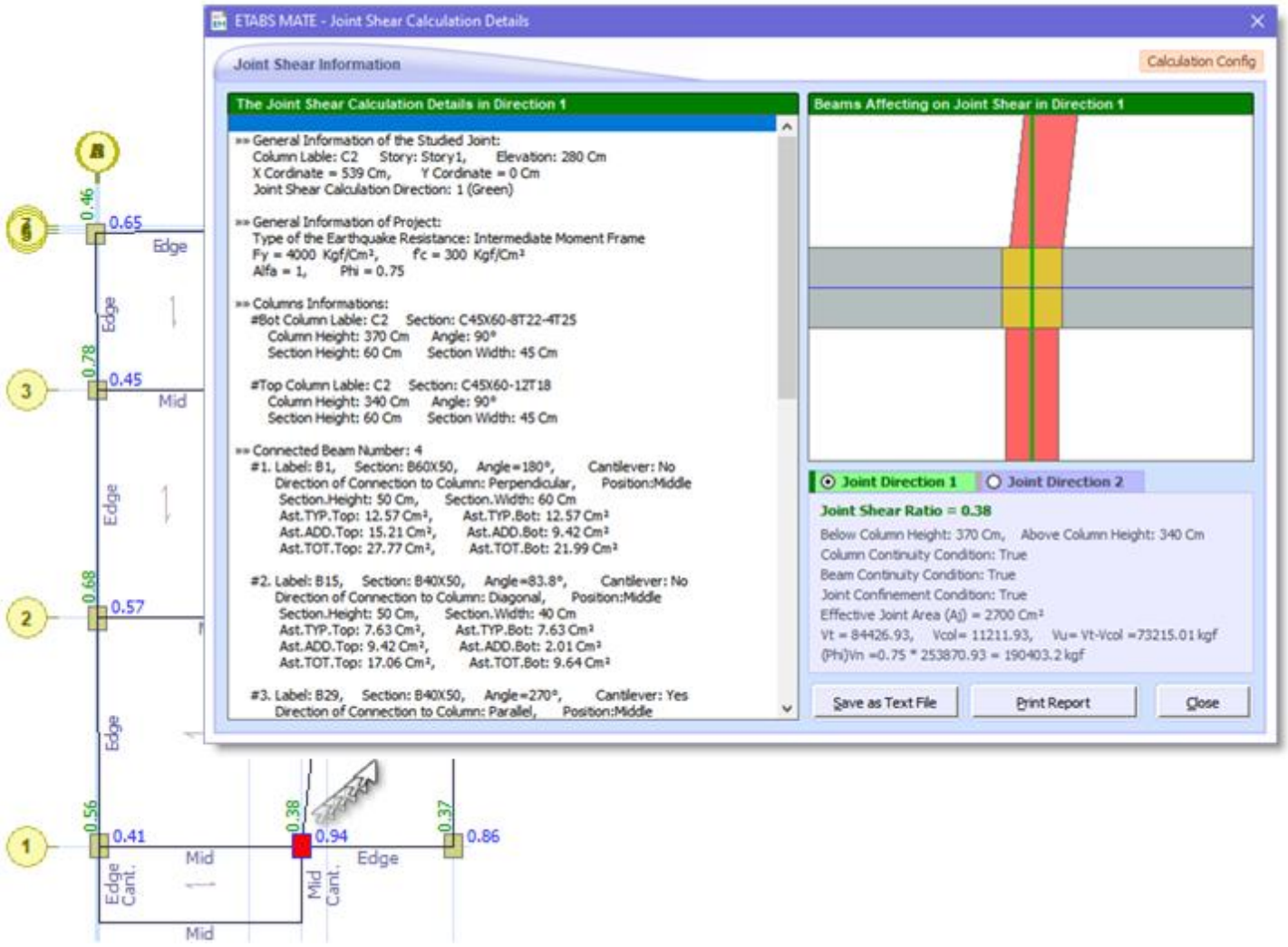
Thus » The Development Length is Not Satisfied!

ETABS MATE
Concrete Structure Detailing Software

Joint Shear Ratio Control

ETABS MATE automatically calculates the joint shear ratio parameter for all joints of the structure without any limitations regarding the column orientation angle in plan, the number of beams, or the angles of the beams entering the joint. This calculation considers the continuity conditions of the beams and columns entering the joint, the confinement condition of the joint, and is based on the actual detailing reinforcement drawn in the structural drawings. It also accounts for whether the beams are cantilevered or non-cantilevered, as well as the positioning of the beams relative to the column.

A sample of the software's joint shear calculations is shown in the image below. As can be seen, the software automatically calculates the joint shear ratio for both directions of the column and displays it on all joints. Furthermore, clicking on any joint provides the detailed calculations.



Joint Shear Ratio Calculation Methodology

The joint shear ratio in the software is calculated using the following relationships:

$$\textbf{Joint Shear Capacity} = V_u/\emptyset V_n$$

$$V_u^h = T_L + T_R - V_u$$

$$V_u = \frac{M_u^L + M_u^R}{H}$$

In calculating V_u, the probable flexural strength of the beams, M_{pr}, is determined using the following relationship, based on the detailing reinforcement at the top and bottom of each beam connected to the joint. These values for each beam are then projected onto the direction for which the joint shear is being calculated.

$$M_{pr} = A_s d \alpha f_y \left(1 - \frac{0.59 \alpha f_y A_s}{d b f'_c}\right)$$

Where:

A_s: Cross-sectional area of the detailing reinforcement

d: Effective depth of the beam

b: Width of the beam

f_y: Yield strength of the rebar

f'_c: Compressive strength of the concrete

α: 1.25 for special moment frame, and 1.0 for intermediate moment frame

Additionally, the V_n parameter is calculated based on the table below, according to the various continuity conditions of the beam and column, as well as the confinement condition of the joint, which will be described in the following.

Table 18.8.4.3—Nominal joint shear strength V_n

Column	Beam in direction of V_u	Confinement by transverse beams according to 15.2.8	V_n, N[1]
Continuous or meets 15.2.6	Continuous or meets 15.2.7	Confined	$1.7\lambda\sqrt{f'_c}A_j$
		Not confined	$1.2\lambda\sqrt{f'_c}A_j$
	Other	Confined	$1.2\lambda\sqrt{f'_c}A_j$
		Not confined	$1.0\lambda\sqrt{f'_c}A_j$
Other	Continuous or meets 15.2.7	Confined	$1.2\lambda\sqrt{f'_c}A_j$
		Not confined	$1.0\lambda\sqrt{f'_c}A_j$
	Other	Confined	$1.0\lambda\sqrt{f'_c}A_j$
		Not confined	$0.7\lambda\sqrt{f'_c}A_j$

[1]λ shall be 0.75 for lightweight concrete and 1.0 for normalweight concrete. A_j shall be calculated in accordance with 15.4.2.4.

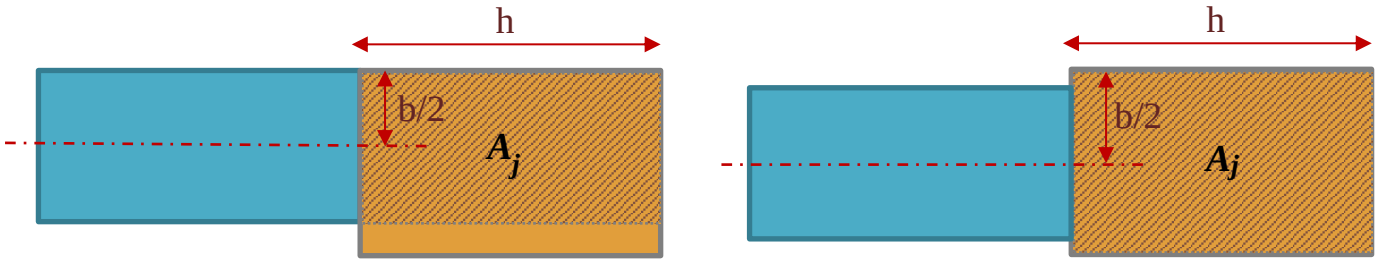
In this relationships A_j is the Effective Joint Area, the calculation of which is provided in the following.

The Effective Joint Area, A_j, is defined in accordance with ACI provisions as follows:

15.4.2.4 Effective cross-sectional area within a joint, A_j, shall be calculated as the product of joint depth and effective joint width. Joint depth shall be the overall depth of the column, ***h***, in the direction of joint shear considered. Effective joint width shall be the overall width of the column where the beam is wider than the column. Where the column is wider than the beam, effective joint width shall not exceed the lesser of (a) and (b):

(a) Beam width plus joint depth

(b) Twice the perpendicular distance from longitudinal axis of beam to nearest side face of the column



The continuity conditions of the columns and beams connected to the joint are defined in accordance with ACI provisions as follows:

15.2.7 A beam extension assumed to provide continuity through a beam-column joint in the direction of joint shear considered shall satisfy (a) and (b):

(a) The beam extends at least one beam depth ***h*** beyond the joint face.

(b) Longitudinal and transverse reinforcement from the beam on the opposite side of the joint is continued through the extension.

15.2.8 A beam-column joint shall be considered to be confined for the direction of joint shear considered if two transverse beams satisfying (a), (b), and (c) are provided:

(a) Width of each transverse beam is at least three-quarters of the width of the column face into which the beam frames

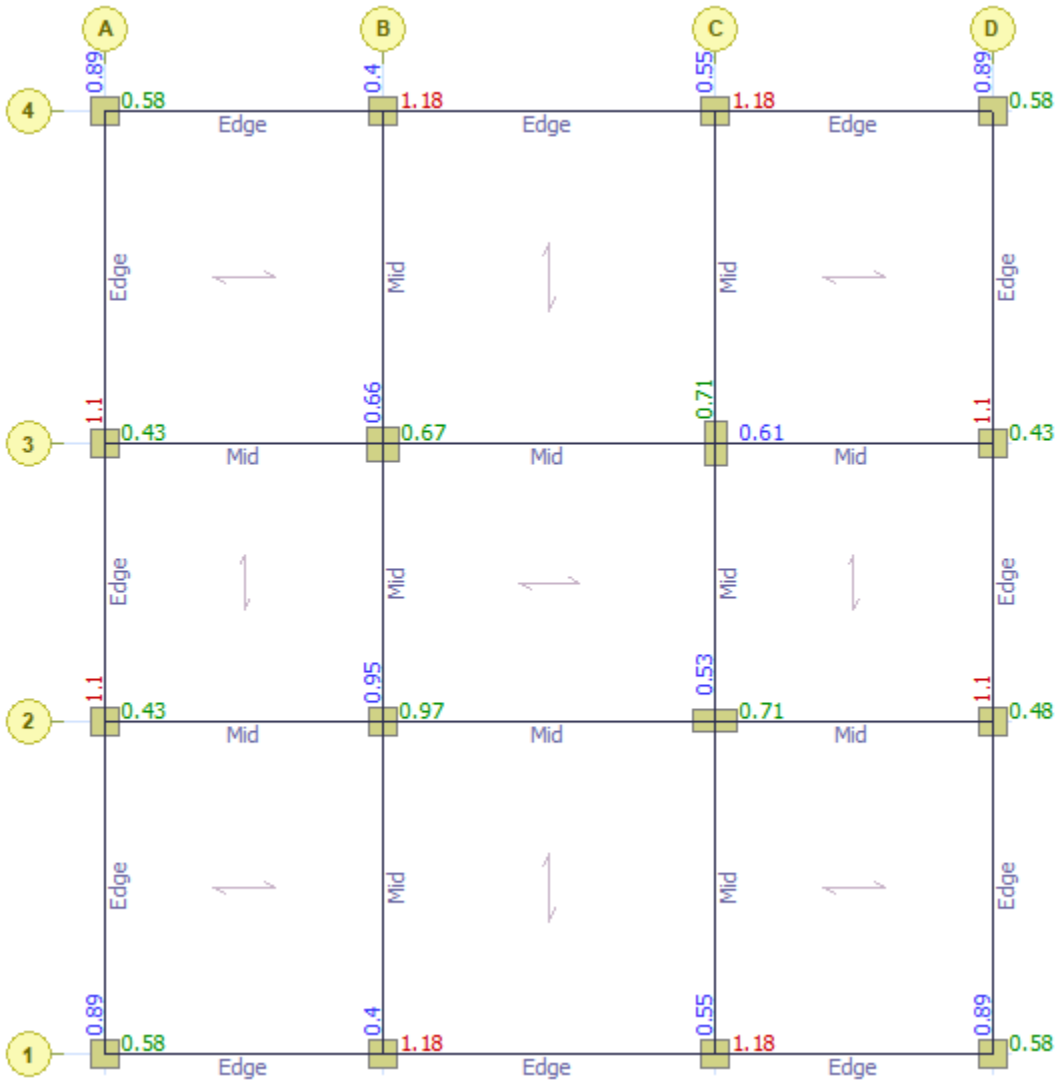
(b) Transverse beams extend at least one beam depth ***h*** beyond the joint faces

(c) Transverse beams contain at least two continuous top and bottom bars satisfying 9.6.1.2 and No. 10 or larger stirrups satisfying 9.6.3.4 and 9.7.6.2.2

Reviewing the Joint Shear Ratio

The software automatically calculates the joint shear ratio for all joints of the structure. To view the joint shear ratio, simply select the **Joint Shear Information** layer from the **LAYER DISPLAY** section in the side panel of the software. The joint shear ratio for both column directions will then be displayed on all joints, as shown in the image below.

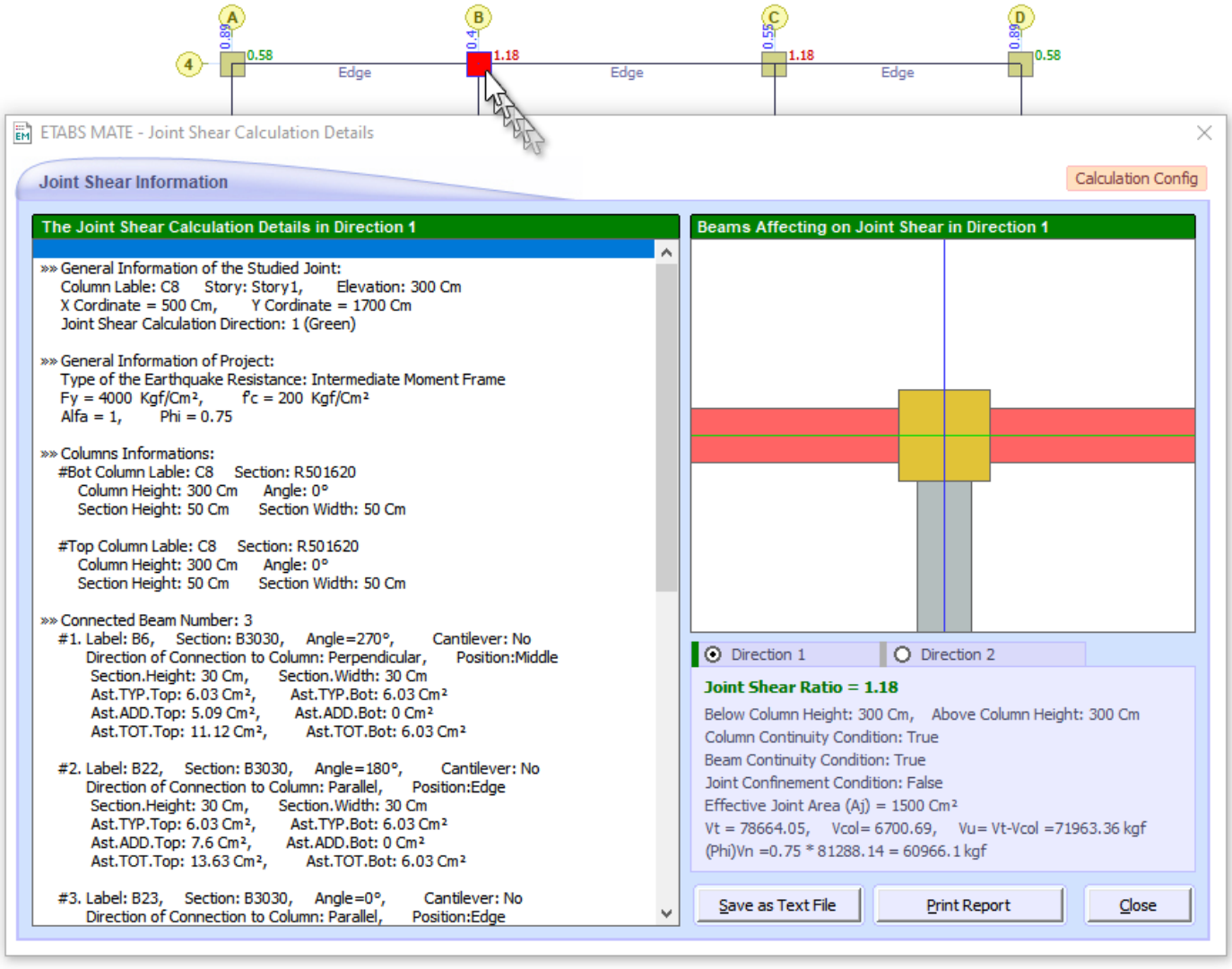
The Joint Shear Ratio in direction 1 of the joint is displayed in green, and in direction 2 of the joint is displayed in blue, on each joint. If the joint shear ratio for a joint exceeds one, the software displays this value in red to distinguish it and alert the user that the joint shear ratio at that node is not satisfied.



Reviewing Calculation Details of Joint Shear Ratio

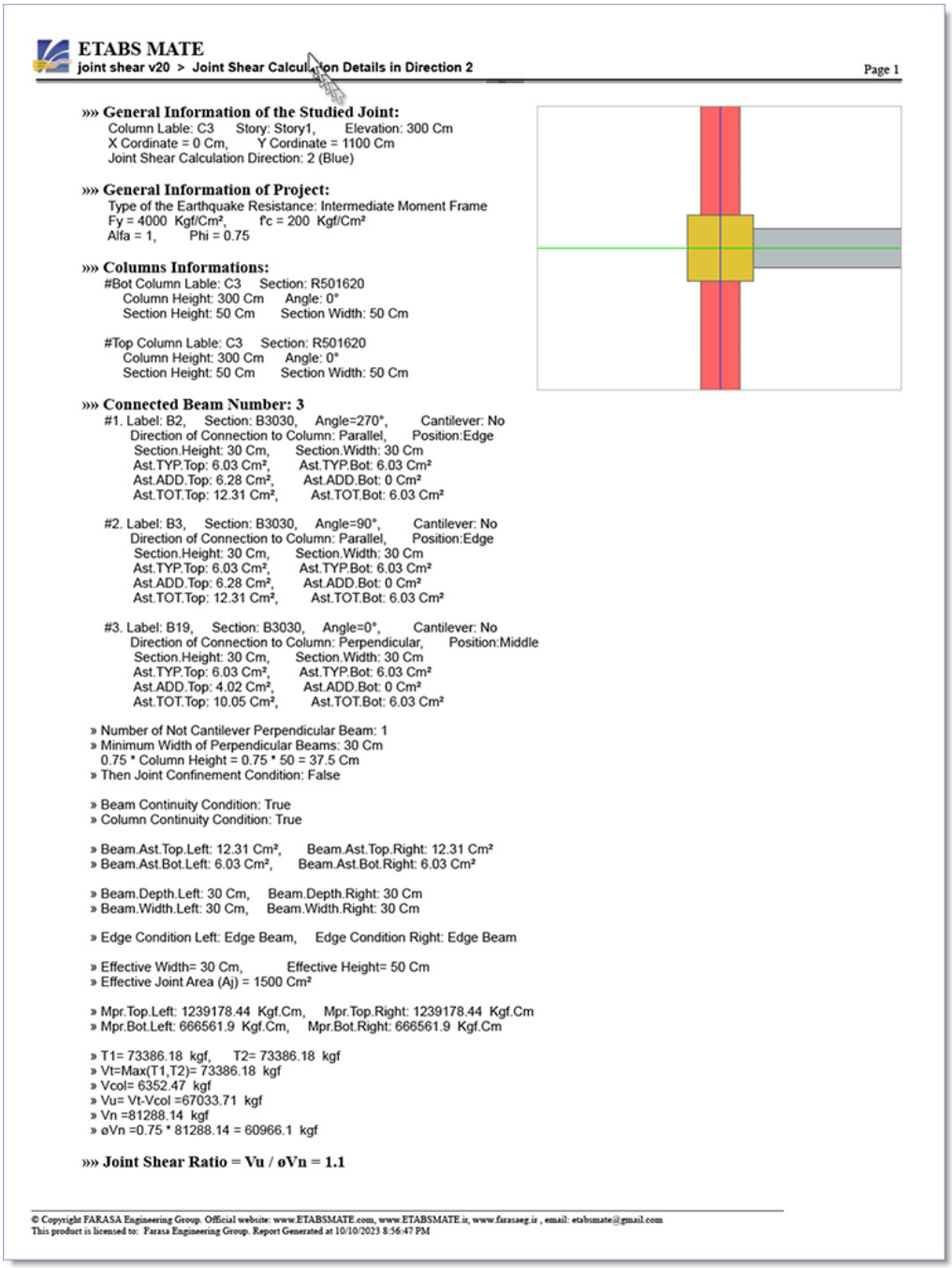
To review the full details of the joint shear ratio calculations, first activate the **Joint Shear Information** layer from the **LAYER DISPLAY** section in the side panel of the software. With this layer active, clicking on any column opens the joint shear calculation details interface, as shown in the image below. By selecting the desired direction, you can view all the conditions applied to the joint, as well as the detailed calculations of the joint shear ratio for that joint.

Note that in the graphical part of this interface, the beams and columns connected to the joint are displayed. The beams that contribute to the joint shear calculation in the selected direction are highlighted in a red color spectrum, with a more intense red indicating a greater contribution of the beam in that direction. In other words, the contribution range of a beam in the given direction, from zero to one hundred percent, is indicated by a color spectrum ranging from gray to red, as shown in the image below.



Printing the Joint Shear Ratio Calculation Report

The software provides highly practical reports of all the conditions applied to the joint, as well as the detailed calculation procedure of the joint shear ratio. You can use these reports for submission or inclusion in the project calculation book. To do so, first activate the **Joint Shear Information** layer from the **LAYER DISPLAY** section in the side panel of the software. Then, clicking on any column opens the joint shear calculation details interface. In this interface, pressing the **Print Report** button allows you to print the joint shear ratio calculation report, or you can save the report as a text file using the **Save as Text File** button.



ETABS MATE
joint shear v20 > Joint Shear Calculation Details in Direction 2 Page 1

»» General Information of the Studied Joint:
Column Lable: C3 Story: Story1, Elevation: 300 Cm
X Cordinate = 0 Cm, Y Cordinate = 1100 Cm
Joint Shear Calculation Direction: 2 (Blue)

»» General Information of Project:
Type of the Earthquake Resistance: Intermediate Moment Frame
Fy = 4000 Kgf/Cm², f'c = 200 Kgf/Cm²
Alfa = 1, Phi = 0.75

»» Columns Informations:
#Bot Column Lable: C3 Section: R501620
Column Height: 300 Cm Angle: 0°
Section Height: 50 Cm Section Width: 50 Cm

#Top Column Lable: C3 Section: R501620
Column Height: 300 Cm Angle: 0°
Section Height: 50 Cm Section Width: 50 Cm

»» Connected Beam Number: 3
#1. Label: B2, Section: B3030, Angle=270°, Cantilever: No
Direction of Connection to Column: Parallel, Position:Edge
Section.Height: 30 Cm, Section.Width: 30 Cm
Ast.TYP.Top: 6.03 Cm², Ast.TYP.Bot: 6.03 Cm²
Ast.ADD.Top: 6.28 Cm², Ast.ADD.Bot: 0 Cm²
Ast.TOT.Top: 12.31 Cm², Ast.TOT.Bot: 6.03 Cm²

#2. Label: B3, Section: B3030, Angle=90°, Cantilever: No
Direction of Connection to Column: Parallel, Position:Edge
Section.Height: 30 Cm, Section.Width: 30 Cm
Ast.TYP.Top: 6.03 Cm², Ast.TYP.Bot: 6.03 Cm²
Ast.ADD.Top: 6.28 Cm², Ast.ADD.Bot: 0 Cm²
Ast.TOT.Top: 12.31 Cm², Ast.TOT.Bot: 6.03 Cm²

#3. Label: B19, Section: B3030, Angle=0°, Cantilever: No
Direction of Connection to Column: Perpendicular, Position:Middle
Section.Height: 30 Cm, Section.Width: 30 Cm
Ast.TYP.Top: 6.03 Cm², Ast.TYP.Bot: 6.03 Cm²
Ast.ADD.Top: 4.02 Cm², Ast.ADD.Bot: 0 Cm²
Ast.TOT.Top: 10.05 Cm², Ast.TOT.Bot: 6.03 Cm²

» Number of Not Cantilever Perpendicular Beam: 1
» Minimum Width of Perpendicular Beams: 30 Cm
0.75 * Column Height = 0.75 * 50 = 37.5 Cm
» Then Joint Confinement Condition: False

» Beam Continuity Condition: True
» Column Continuity Condition: True

» Beam.Ast.Top.Left: 12.31 Cm², Beam.Ast.Top.Right: 12.31 Cm²
» Beam.Ast.Bot.Left: 6.03 Cm², Beam.Ast.Bot.Right: 6.03 Cm²

» Beam.Depth.Left: 30 Cm, Beam.Depth.Right: 30 Cm
» Beam.Width.Left: 30 Cm, Beam.Width.Right: 30 Cm

» Edge Condition Left: Edge Beam, Edge Condition Right: Edge Beam

» Effective Width= 30 Cm, Effective Height= 50 Cm
» Effective Joint Area (Aj) = 1500 Cm²

» Mpr.Top.Left: 1239178.44 Kgf.Cm, Mpr.Top.Right: 1239178.44 Kgf.Cm
» Mpr.Bot.Left: 666561.9 Kgf.Cm, Mpr.Bot.Right: 666561.9 Kgf.Cm

» T1= 73386.18 kgf, T2= 73386.18 kgf
» Vt=Max(T1,T2)= 73386.18 kgf
» Vcol= 6352.47 kgf
» Vu= Vt-Vcol =67033.71 kgf
» Vn =81288.14 kgf
» øVn =0.75 * 81288.14 = 60966.1 kgf

»» Joint Shear Ratio = Vu / øVn = 1.1

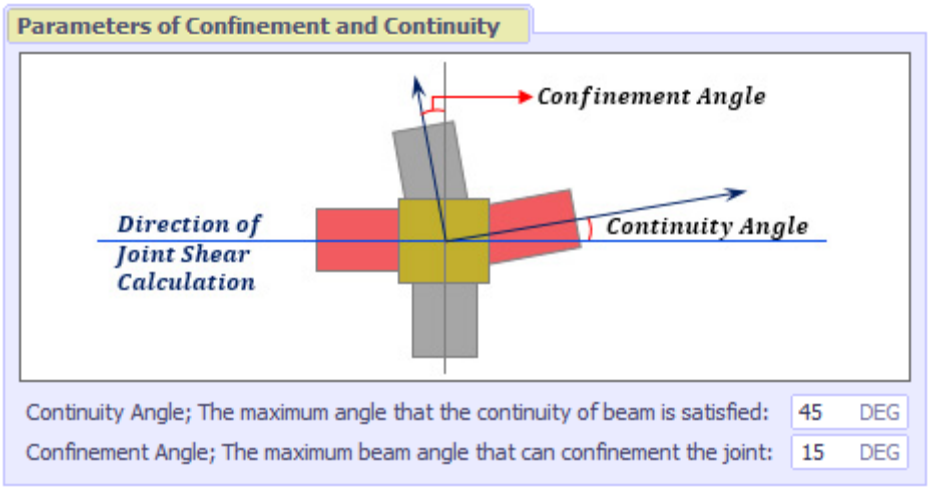
© Copyright FARASA Engineering Group. Official website: www.ETABSMATE.com, www.ETABSMATE.ir, www.farasaeg.ir , email: etabsmate@gmail.com
This product is licensed to: Farasa Engineering Group. Report Generated at 10/10/2023 8:56:47 PM

Configuring Joint Shear Ratio Calculation Parameters

The joint shear ratio calculation parameters configuration interface allows you to configure the parameters governing the joint shear calculation. To access this interface, proceed as follows using the **Define** menu:

Define Menu > Joint Shear Calculation Parameters

The upper section of this interface contains the parameters for defining the continuity and confinement angles, which are described below:



Continuity Angle:

This parameter defines the maximum angle of the beams relative to the joint shear calculation direction for a beam entering the joint to be considered continuous. In other words, if the angle of any beam entering the joint is less than this angle, the software assumes that beam to satisfy the continuity condition. This parameter is set to 45 degrees in the software, and users can easily adjust it to their desired value.

Confinement Angle:

This parameter defines the maximum angle of the beams relative to the direction perpendicular to the joint shear calculation direction for the possibility of providing confinement to the joint by the beams entering the joint. In other words, if the angle of any beam entering the joint is less than this parameter, the software assumes that beam to satisfy the confinement condition for the joint. This parameter is set to 15 degrees in the software, and users can easily adjust it to their desired value.

Note that the values of these parameters are not specified in any reference, and users must set them to appropriate values based on their own engineering judgment.

As shown in the image below, the lower section of the joint shear ratio calculation parameters configuration interface contains the parameters for calculating V_n under various joint conditions.

Since the coefficients for calculating V_n have undergone changes in code revisions, these coefficients have been made user-editable, allowing the user to easily configure the method of joint shear calculation. By default, these parameters are set based on ACI 318-19, as shown in the image below.

The λ parameter, which pertains to lightweight concrete, is also editable at the end of this section. As shown in the image below, the λ parameter is set to 1 by default, which corresponds to normal-weight concrete.

Parameters for Calculation of the Vn

Column	Beam in direction of V_u	Joint Confinement	V_n, N
Continuous	Continuous	Confined	1.7 $\lambda\sqrt{f'_c}A_j$
		Not confined	1.2 $\lambda\sqrt{f'_c}A_j$
	Not Continuous	Confined	1.2 $\lambda\sqrt{f'_c}A_j$
		Not confined	1 $\lambda\sqrt{f'_c}A_j$
Not Continuous	Continuous	Confined	1.2 $\lambda\sqrt{f'_c}A_j$
		Not confined	1 $\lambda\sqrt{f'_c}A_j$
	Not Continuous	Confined	1 $\lambda\sqrt{f'_c}A_j$
		Not confined	0.7 $\lambda\sqrt{f'_c}A_j$

λ; shall be 0.75 for lightweight concrete and 1.0 for normalweight concrete: 1

Note that using the **Configuration Setting** menu at the top of the joint shear ratio calculation parameters configuration interface, the user can save all parameters in this interface after applying the desired changes. This allows the software to automatically use the user's customized parameters in subsequent sessions, eliminating the need to reconfigure these parameters for future projects.

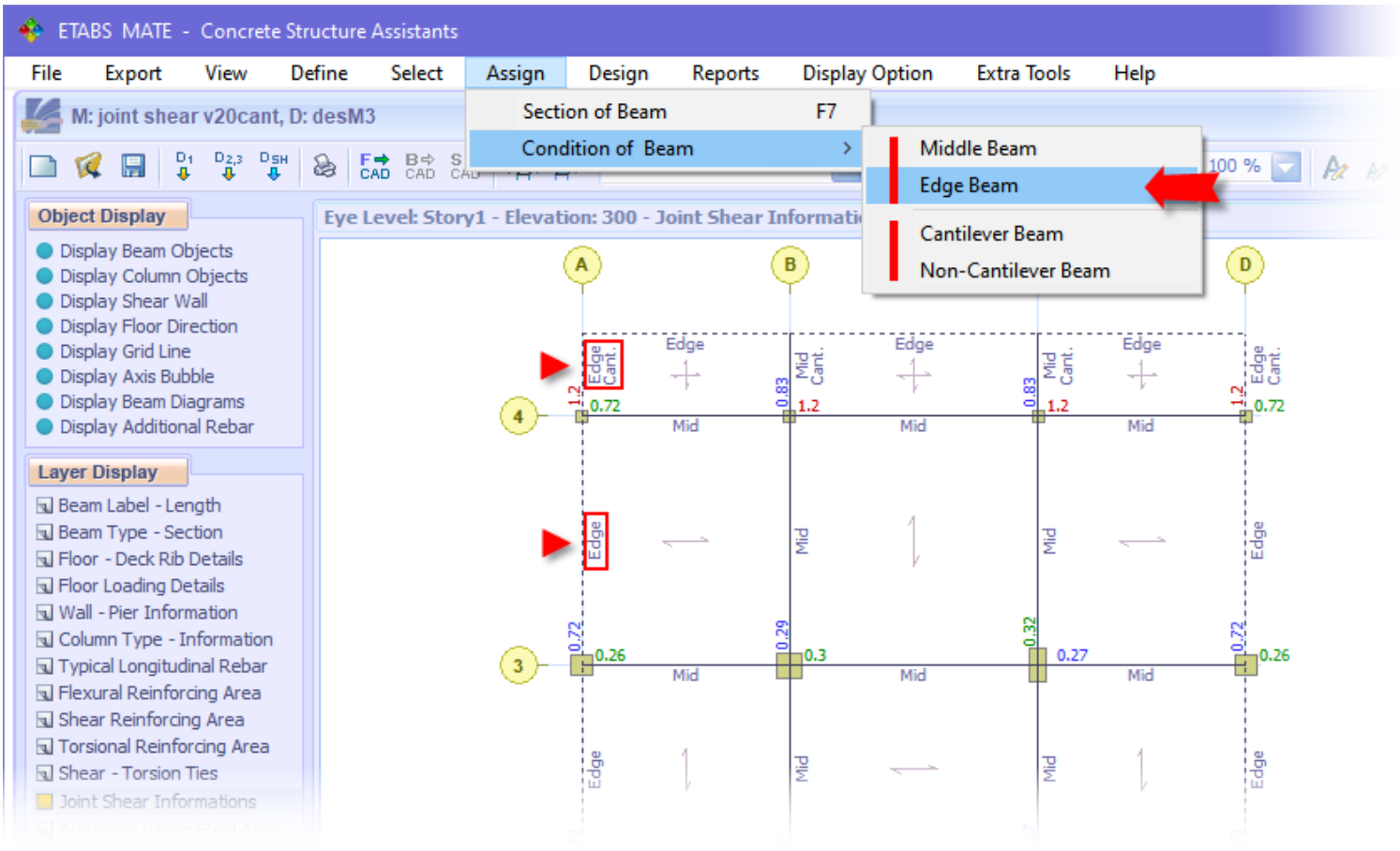
One of the most critical parameters in calculating the joint shear ratio is the positioning of the beams relative to the column section, which must be specified by the user to ensure the joint shear ratio is correctly calculated by the software. This topic is described in the following.

ETABS MATE
Concrete Structure Detailing Software

Defining Edge or Cantilever Beams for Joint Shear Calculation

Given that the position of beams relative to the column section significantly influences the calculation of the effective joint area, A_j, and also that whether a beam is cantilevered or not affects the calculation of M_{pr} due to the fact that a cantilever beam is not part of the seismic frame, the software provides the ability for users to easily change the beam condition to Intermediate Beam, Edge Beam, Cantilever Beam, or Non-Cantilever Beam.

By default, the software assumes the **Middle** condition for all beams, and you must change the condition of side beams to **Edge**. For this purpose, as shown in the figure, first select the edge beams, then using the **Assign** menu and the **Beam Condition** option, select **Edge Beam** so that the beam-to-column configuration is set to edge beam. Then, again select the cantilever beams and select **Cantilever Beam to** define the beam as cantilever. It should be noted that the software automatically identifies cantilever beams and displays them on the beams as shown in the figure below. However, since the automatic identification of cantilever beams is performed through an intelligent routine, it is recommended that the user performs the necessary checks to ensure the software has correctly identified them. The software also displays the assigned beam conditions in the **Joint Shear Information** layer, as illustrated below.

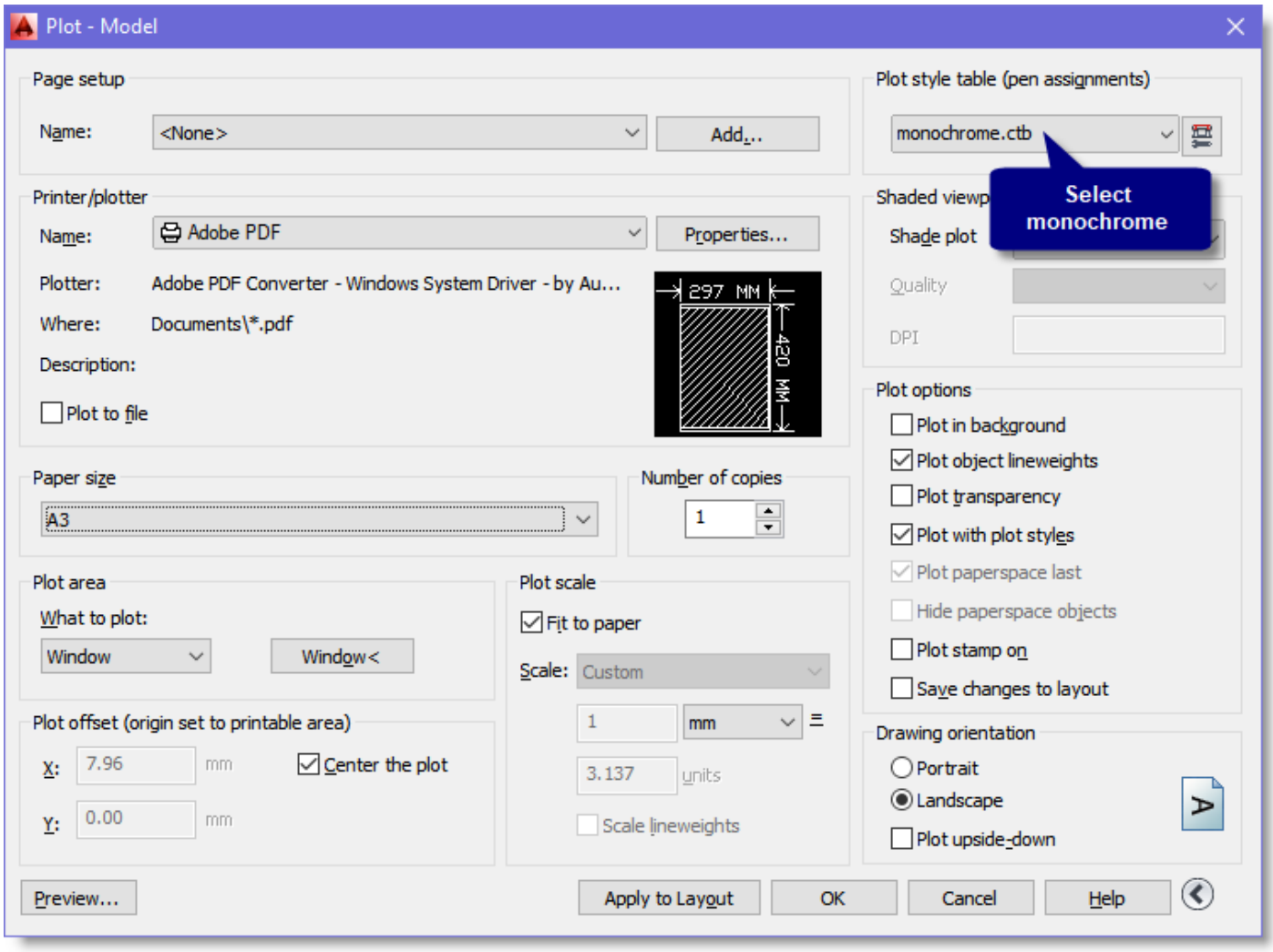


ETABS MATE
Concrete Structure Detailing Software

Plot Settings for Drawings in AutoCAD

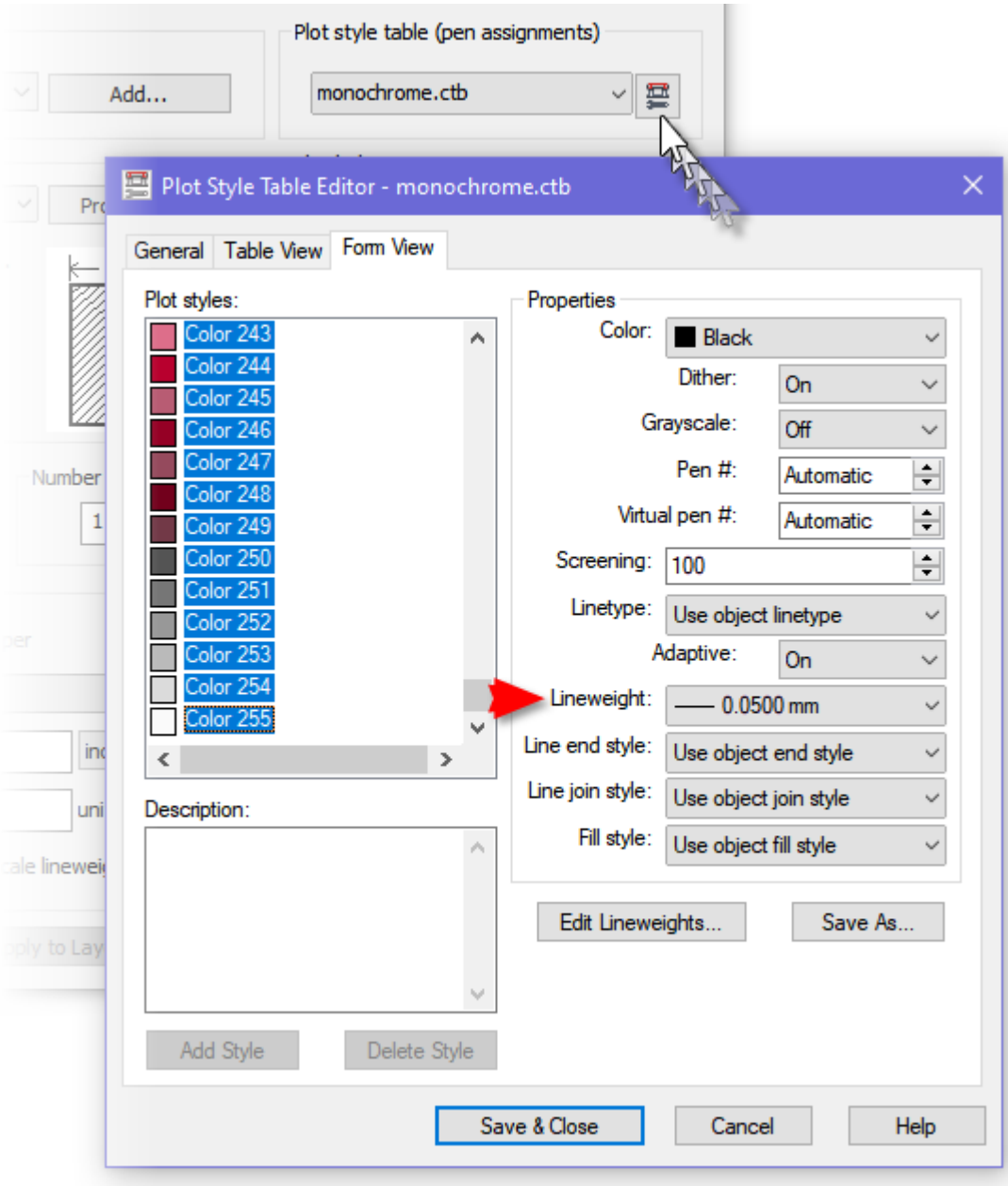
The generation of construction drawings by **ETABS MATE** is performed entirely based on professional drafting principles, enabling you to present high quality construction drawings. For this purpose, various components are drawn in separate layers. Additionally, to utilize different lineweights when printing the drawings, which significantly contributes to the professionalism, clarity, and legibility of the drawings, different colors have been used to filter lineweights for printing. These colors are not assigned to the components themselves but rather to the layer containing those components, so that the user can easily change the color of all components within a layer by modifying the layer color.

For AutoCAD to print components with different colors using varying lineweights, after issuing the print command in AutoCAD, as shown in the figure below, you must first set the plot style to **Monochrome.ctb** from the **Plot Style** drop down list. Then, click the icon  next to the same drop down list to display the table for assigning lineweights to different colors. Subsequently, assign a lineweight for printing to each color, as explained below.



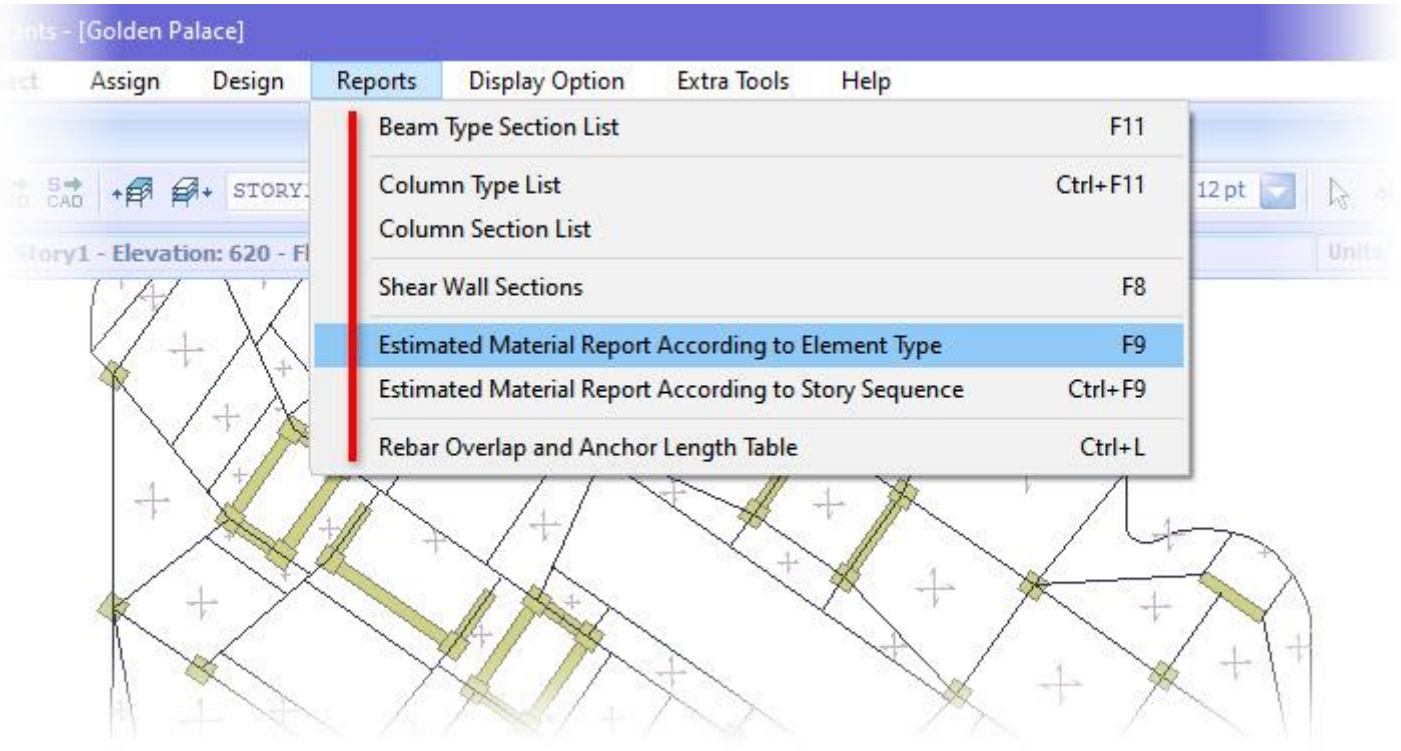
After clicking the icon , the lineweight assignment interface for each color will appear, as shown in the image below. First, select all colors by holding down the Shift key and clicking on the last color. Then, in the **Lineweight** section, set the lineweight to **0.05** for **all colors**. Next, for the colors **Yellow (Color 2), White (Color 7), and Black (Color 255)**, set the lineweight to **0.09** so they are printed with a greater thickness. Additionally, if you have used other colors in the drawing, specify their lineweights as well. Then, press the **Save & Close** button to close this interface.

You may now proceed with the remaining drawing print steps to produce high-quality plots of the drawings.



Generating Reports by the Software

In addition to the structural drawings, highly useful reports can also be generated by the software. As shown in the image below, these reports can be created using the commands available in the **Report** menu of the software.



These reports include the following items, which are described below:

Beam Section List Report

Report Menu > Beam Type Section List (F11)

This option allows you to generate a report listing the beam sections used in the structure.

Column Type Details Report

Report Menu > Column Type List (Ctrl + F11)

This option allows you to generate a report of the column types used, along with the information for each column type.

Column Section List Report

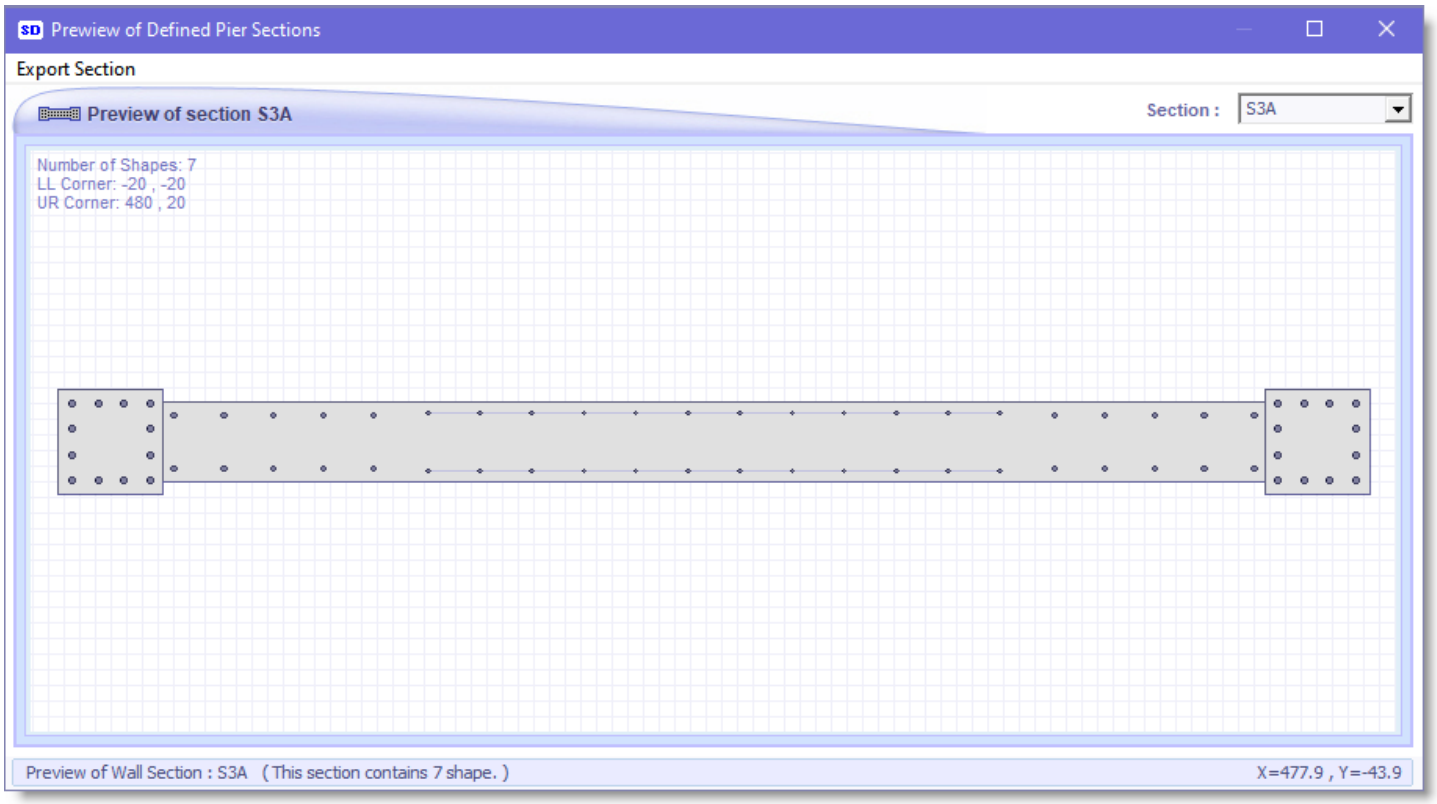
Report Menu > Column Section List

This option allows you to generate a report listing the column sections used in the structure.

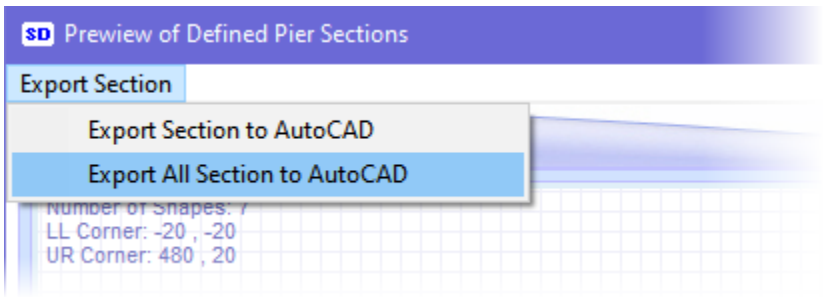
Shear Wall Section List Report

Report Menu > Shear Wall Section (F8)

Graphical display of the structural shear wall sections.



Using the dropdown list at the top of this form, you can view the wall sections defined in the structure. Additionally, at the top of this interface, an **Export Section** menu is available, through which you can export the shear wall sections as an AutoCAD drawing.



■ Export Section to AutoCAD

Exports the selected section as a drawing file in the AutoCAD drawing format.

■ Export All Section to AutoCAD

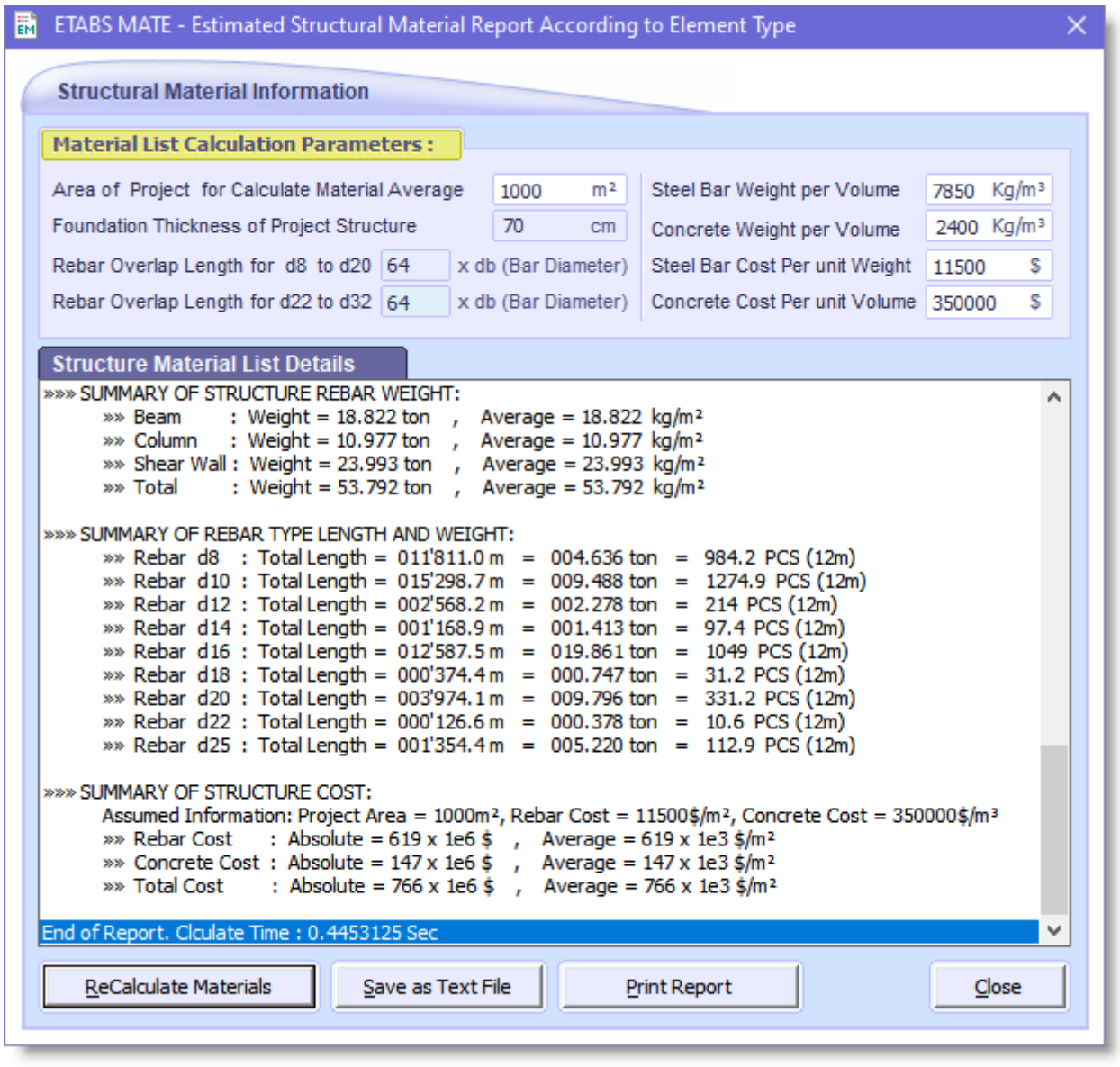
Exports all sections defined in the structural model as a drawing file into the AutoCAD.

Estimated Material Weight and Volume Report

Export > Estimated Material Report According to Element Type (F9)

One of the most useful reports provided by the software is the estimated weight of rebars used throughout the project and the concrete volume of the structure, which can serve as a sound basis for many engineering judgments.

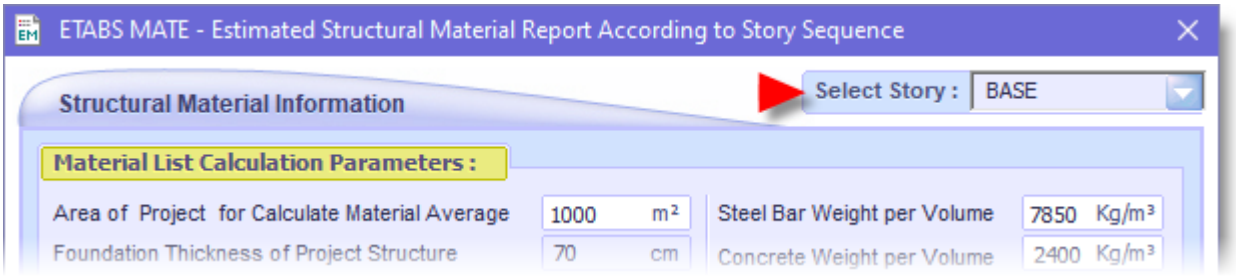
Using this tool, the software automatically calculates the volume and weight of the materials used in the project based on the designs performed by the program, as well as the modifications made by the user, and then presents the results as a printable report. This report includes detailed and summarized breakdowns by element type and by rebar size used in the project. If the project area is specified by the user in the **Area of Project** field, the software calculates and reports the average steel consumption per unit area for the beams, columns, and shear walls. Furthermore, if the rebar and concrete unit prices are correctly specified by the user, the software will also provide an approximate calculation of the total project steel and concrete cost, as well as the cost per unit length.



Estimated Material Weight and Volume Report per Story

Export > Estimated Material Report According to Story Sequence (Ctrl + F9)

This report provides the estimated rebars weight and the concrete volume, separately for each story.



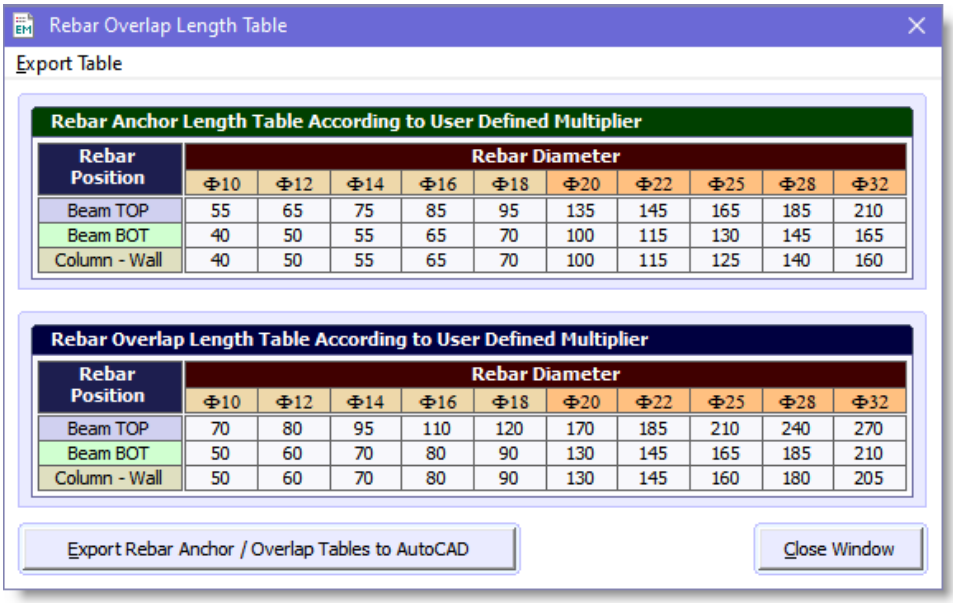
The difference between this report and the previous one is that here, the software calculates the estimated steel weight and concrete volume only for the story selected by the user, whereas the previous report covers the entire structure. After generating these reports, you can print the report directly through the software in its specific format by pressing the **Print Report** button, or save it as a text file using the **Save as Text File** button.

Note: The steel weight and concrete volume calculated in these reports are approximate and suitable only for preliminary engineering judgments. However, the rebar schedules provided alongside the drawings are highly accurate and calculated in full compliance with all provisions. Therefore, it is recommended to use only the rebar schedules drawn alongside the structural drawings for final submission.

Rebar Splice and Development Length Report

Export > Rebar Overlap and Anchor Length table (Ctrl + L)

Displays the development length and splice length tables of the rebars, with the option to export them as a drawing to AutoCAD.

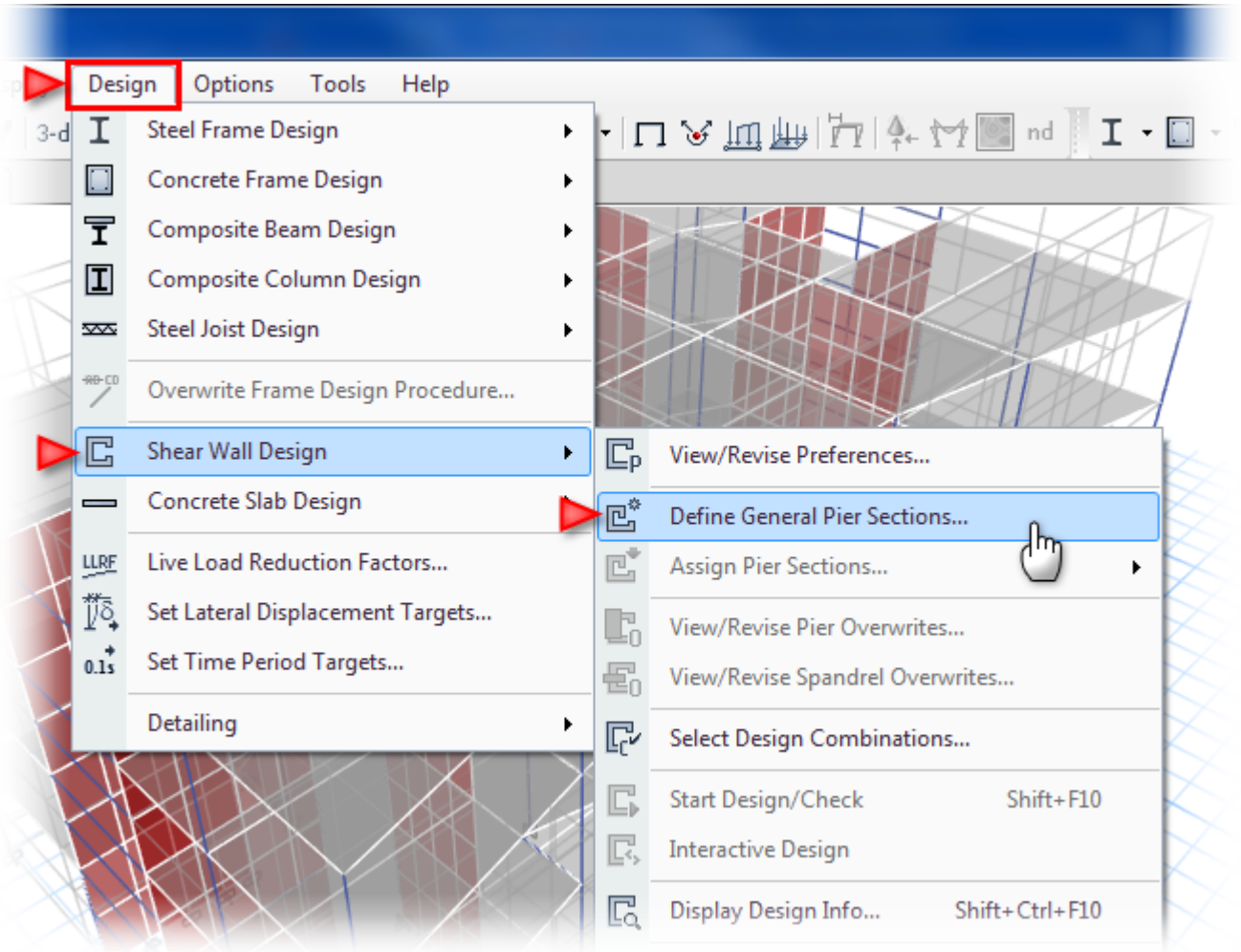


ETABS MATE
Concrete Structure Detailing Software

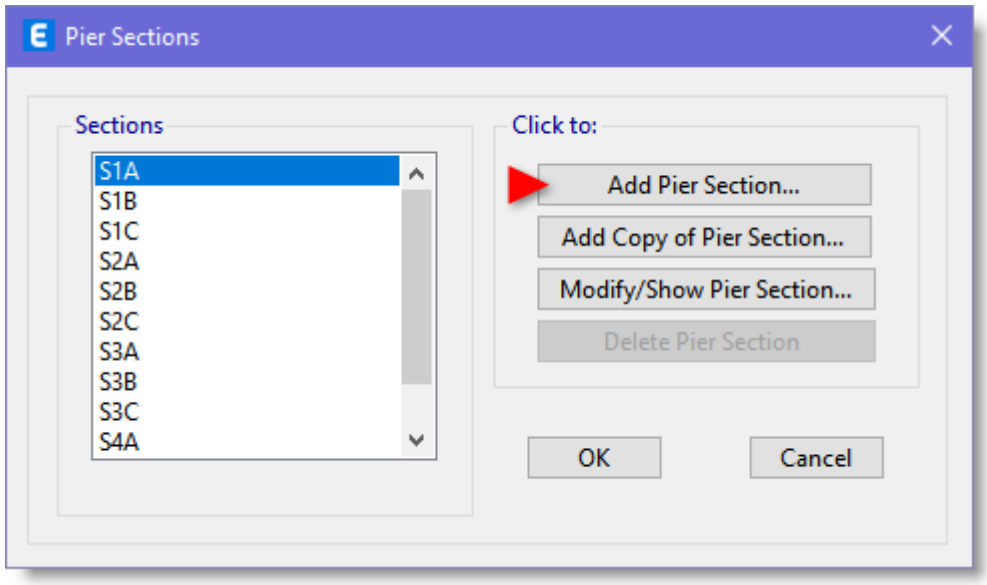
Defining Shear Wall Sections in ETABS

ETABS MATE provides highly suitable outputs for shear walls and also supports complex reinforcement configurations for wall sections. To generate correct and high-quality structural drawings using **ETABS MATE**, please carefully study the material in this section and define the shear wall sections in the **Section Designer** of the ETABS software exactly in accordance with the instructions provided herein to achieve the desired result.

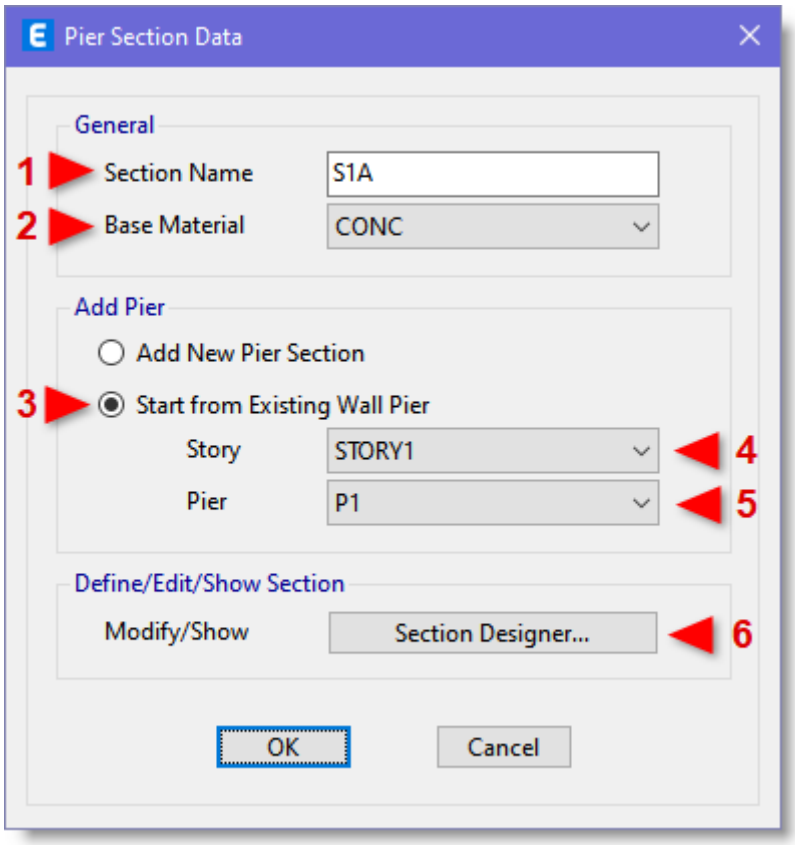
To define the shear wall sections in such a way that you have full control over the reinforcement of all wall components and obtain high-quality outputs from **ETABS MATE**, after completing the modeling and assigning pier names to the walls, define the wall sections using the **General** method. In the ETABS software, from the **Design** menu, select **Shear Wall Design**, and then select **Define General Pier Section**, as shown in the image below.



Selecting **Define General Pier Section** opens the window shown below.

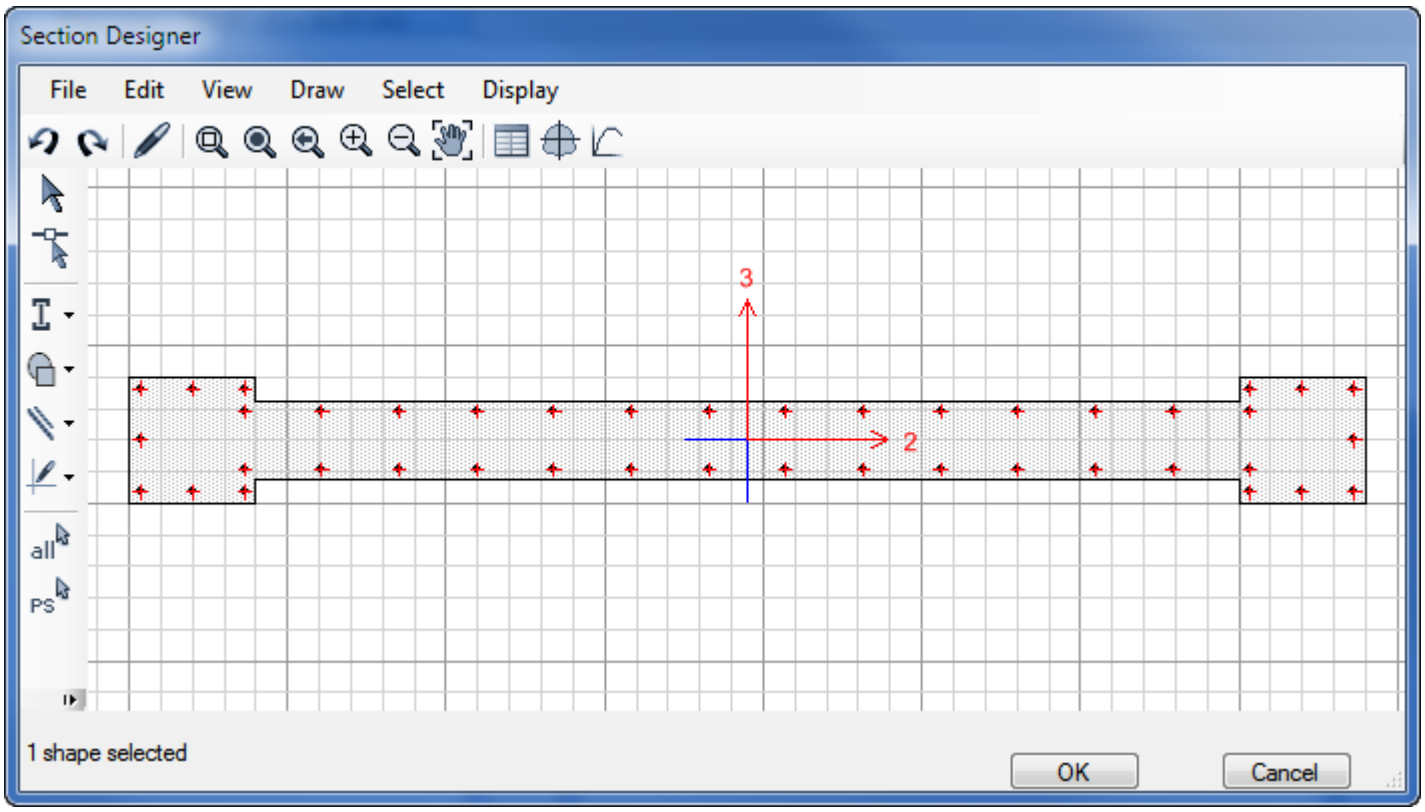


In the **Pier Section** window, as shown in the figure above, click the **Add Pier Section** button to open the window shown below.



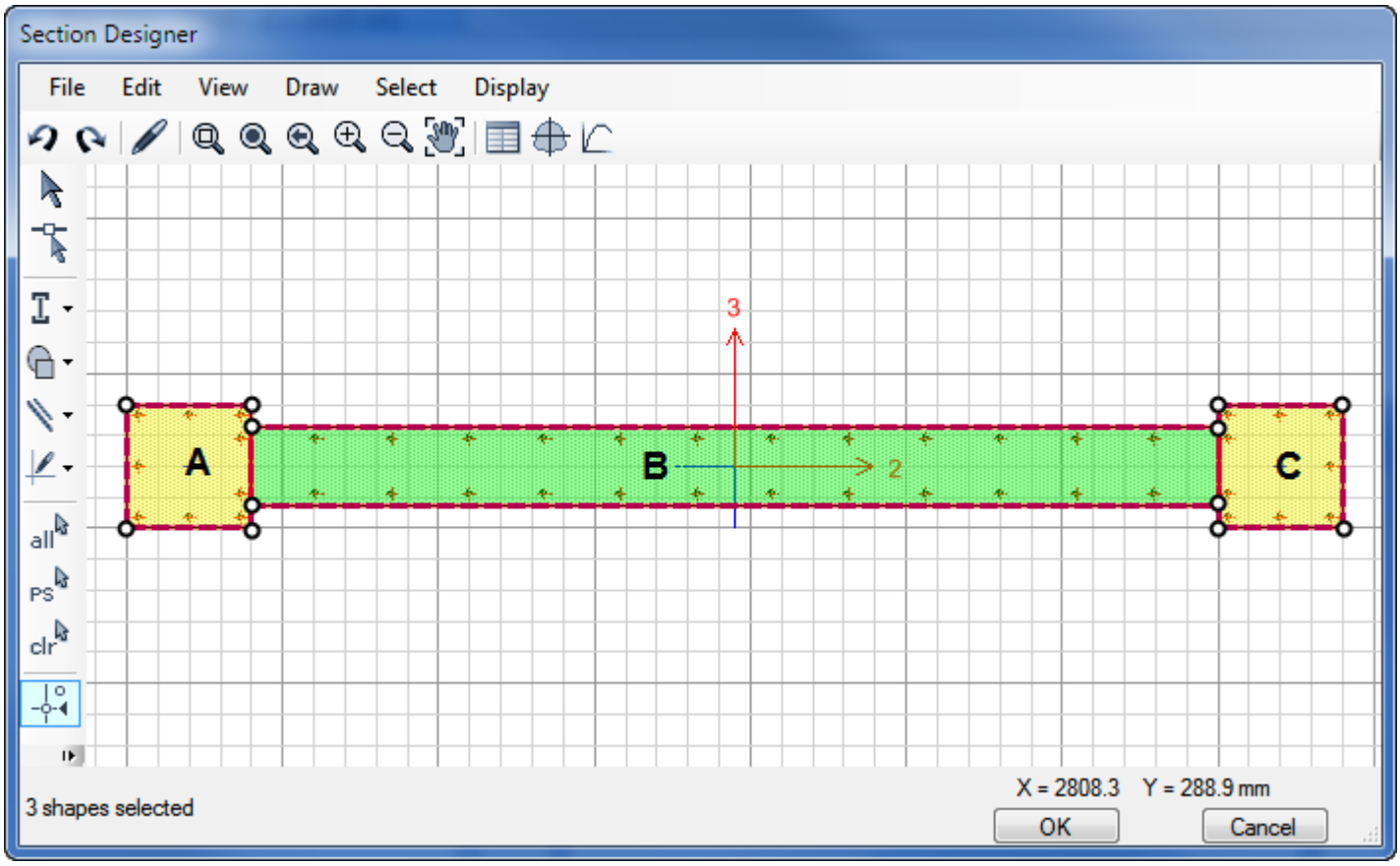
In this window, as shown in the image above, first specify the section name in the **General** section, and then define the base material of the wall. Note that when naming wall sections, and in all naming within the ETABS software, illegal characters, particularly the comma character, must never be used.

Then, in the **Add Pier** section, select the **Start from Existing Wall Pier** option, choose the desired **Story** and **Pier**, and then press the **Section Designer** button to automatically generate the shear wall section by the program. The **Section Designer** window will then appear, as shown in the figure below.

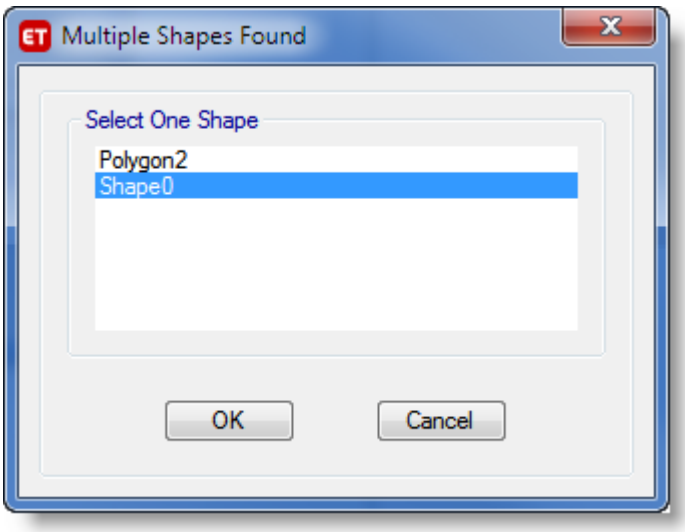


In ETABS version 22.5 and earlier, an issue exists where the section drawn by the software, as shown in the image above, is a single unified piece. This makes it impossible to independently control the reinforcement of the section in the end columns and the wall web. Furthermore, in this state, distinguishing the columns and the wall web is also impossible for **ETABS MATE** software. To address this, a new section must be drawn using the polygon drawn by ETABS, as will be described in the following. Note that in ETABS version 22.5 and later, these polygons are automatically defined as separate components, and performing this step and redrawing them is no longer necessary.

To do this, first click on icon , and then select the Draw Polygon tool Draw Polygon. Next, as shown in the figure below, activate the vertex snap and, by clicking on the vertices of the original polygon drawn by the software, draw the three rectangles **A**, **B**, and **C** over the existing polygon. Alternatively, the Draw Rectangle tool Draw Rectangle may also be used to draw the rectangles.

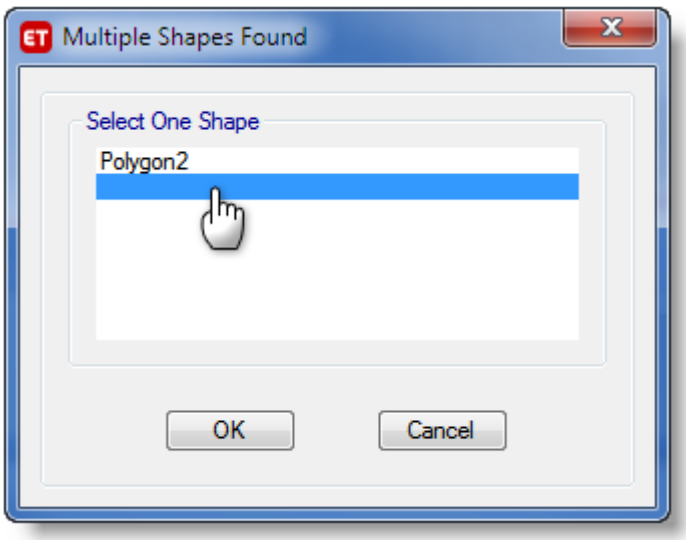


Since the initial polygon still exists beneath these three rectangles, the initial section must first be deleted. To do this, hold down the **Ctrl** key on the keyboard, then click on the drawn section. A window will appear, as shown in the figure below, listing the names of the overlapping shapes. Since the underlying section is the first drawn shape, its name is **Shape0**. Select **Shape0** from the list and delete it using the **Delete** key.

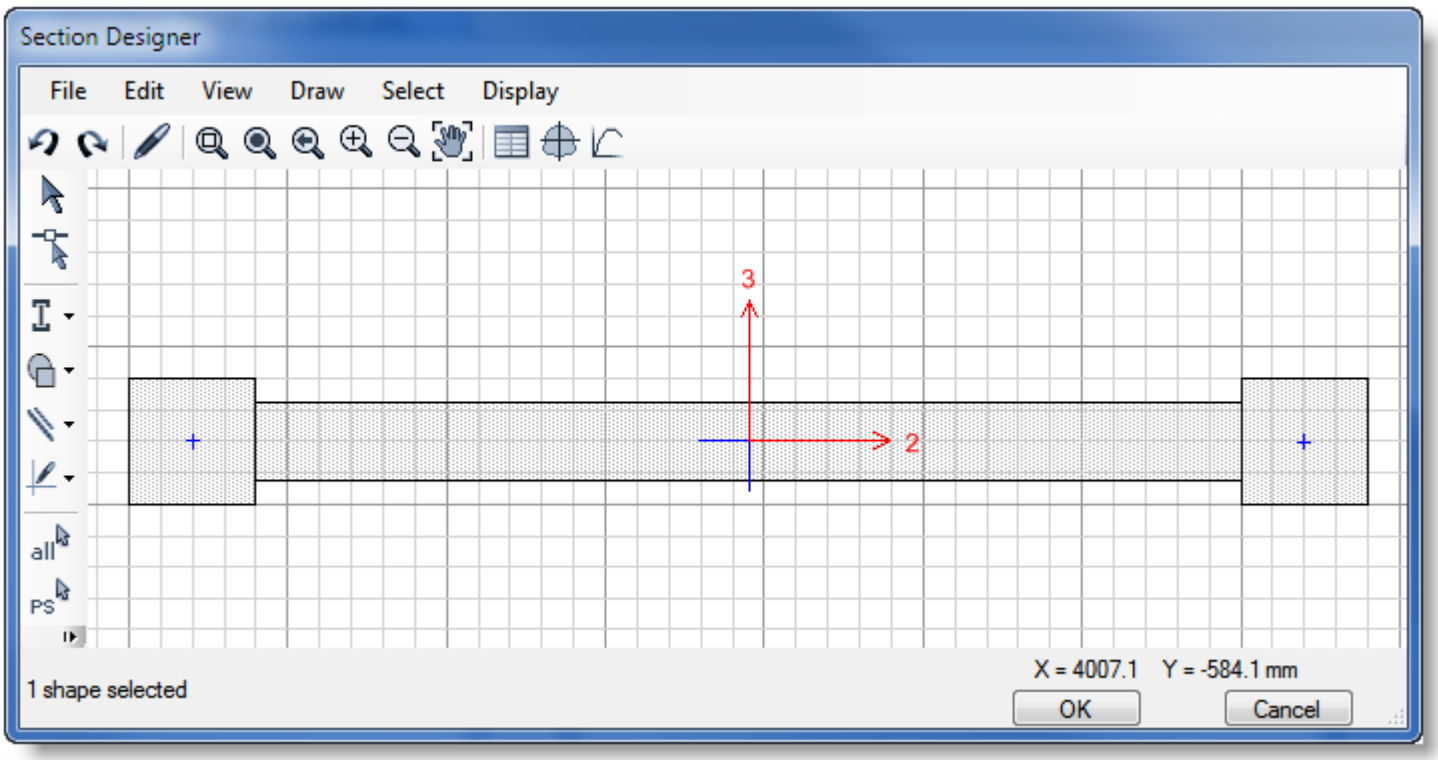


However, if the underlying section does not appear in the list, proceed as follows.

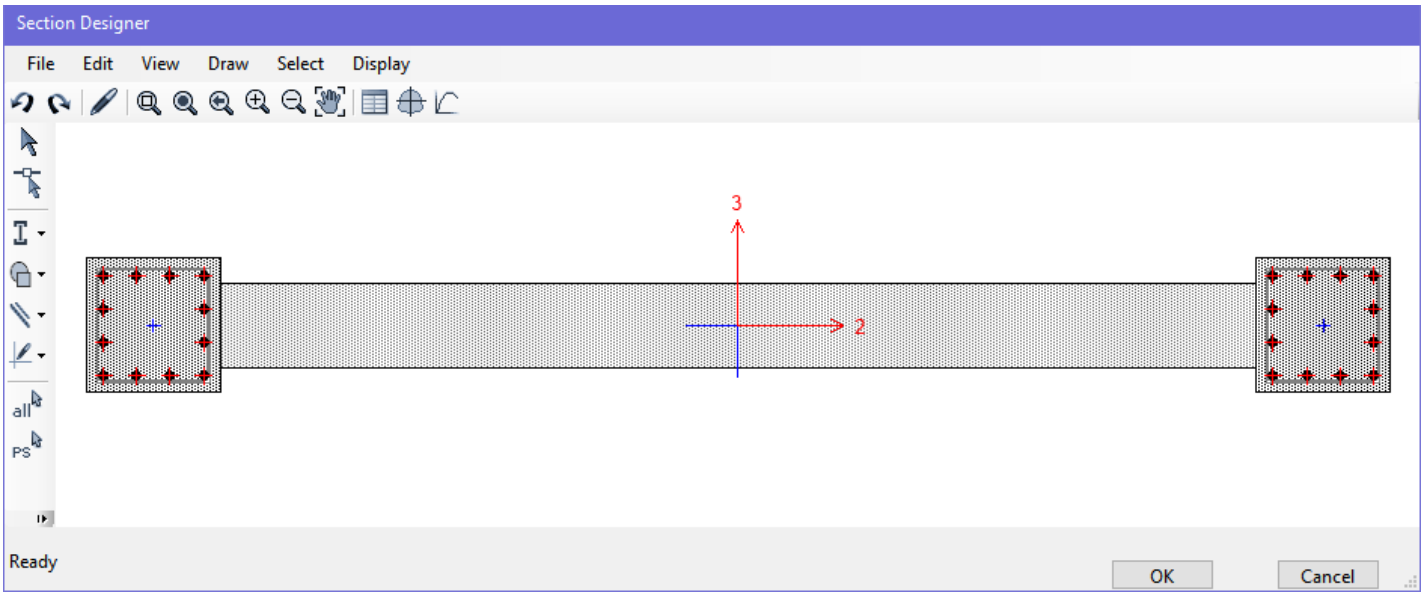
In some cases, the software does not assign a name to the original section that was automatically drawn, and therefore, the name of this shape is not visible in the list, although it is present. To resolve this issue, as shown in the image below, click on the first empty space below the existing section name in the list to select it, as shown in the figure below. Then, press the **OK** button to select the original shape, and subsequently delete it using the **Delete** key.



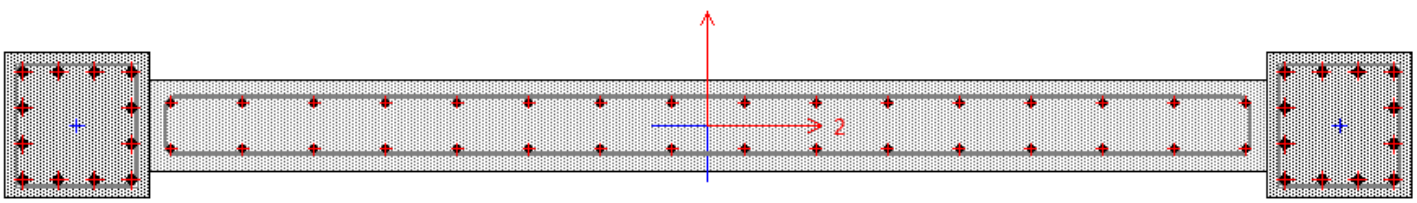
After deleting the original polygon, as shown in the image below, only the three rectangles drawn by you remain. These currently have no reinforcement, and their reinforcement details must now be defined, which will be explained in the following.



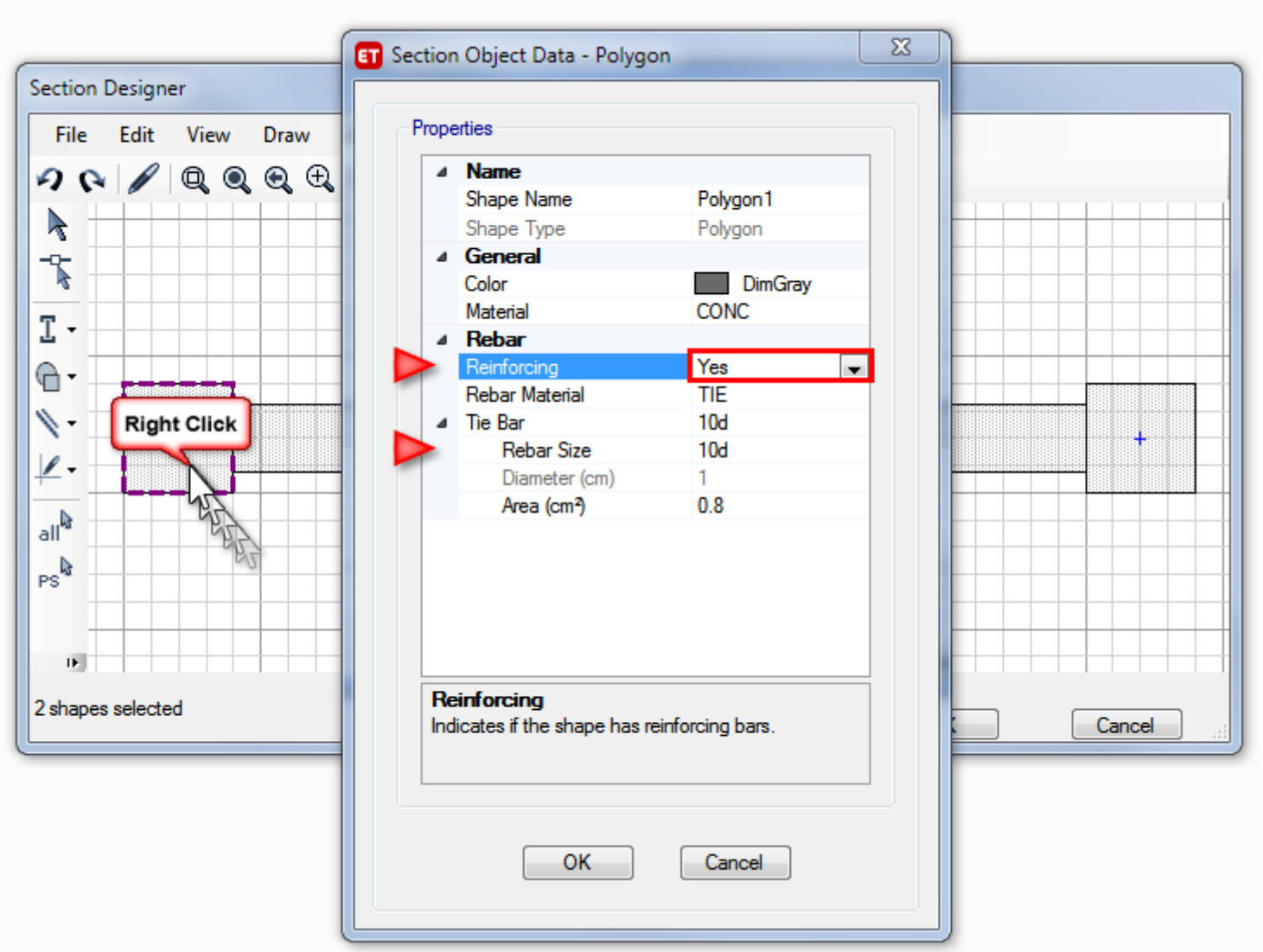
Note: As previously mentioned, in **ETABS version 22.5 and later**, changes have been made to the automatic definition of wall sections in the Section Designer. Specifically, ETABS now automatically defines the columns and the wall web segments as separate polygons, rather than defining a single polyline for the entire wall. This change ensures that the wall section defined by ETABS is more compatible with **ETABS MATE**, eliminating the need for the user to modify or redraw the wall sections, thereby significantly increasing the speed of section definition and reinforcement assignment in ETABS. Therefore, if you are using ETABS version 22.5 or later, the time-consuming steps described above are no longer necessary, and ETABS automatically generates the wall section, as shown in the image below, fully compatible with **ETABS MATE**.



As shown in the image, ETABS draws the columns as separate components and automatically sets their reinforcement details according to what has been defined for the columns piered with the wall. Therefore, it is only necessary to right-click on the wall web, and by setting the **Reinforcing** parameter to **Yes**, define the reinforcement details of the wall mid-portion, which will be explained in the following. The only point to note is that the defined section must be checked, as in some versions, the middle rectangle is drawn twice and the necessary corrections must be made.



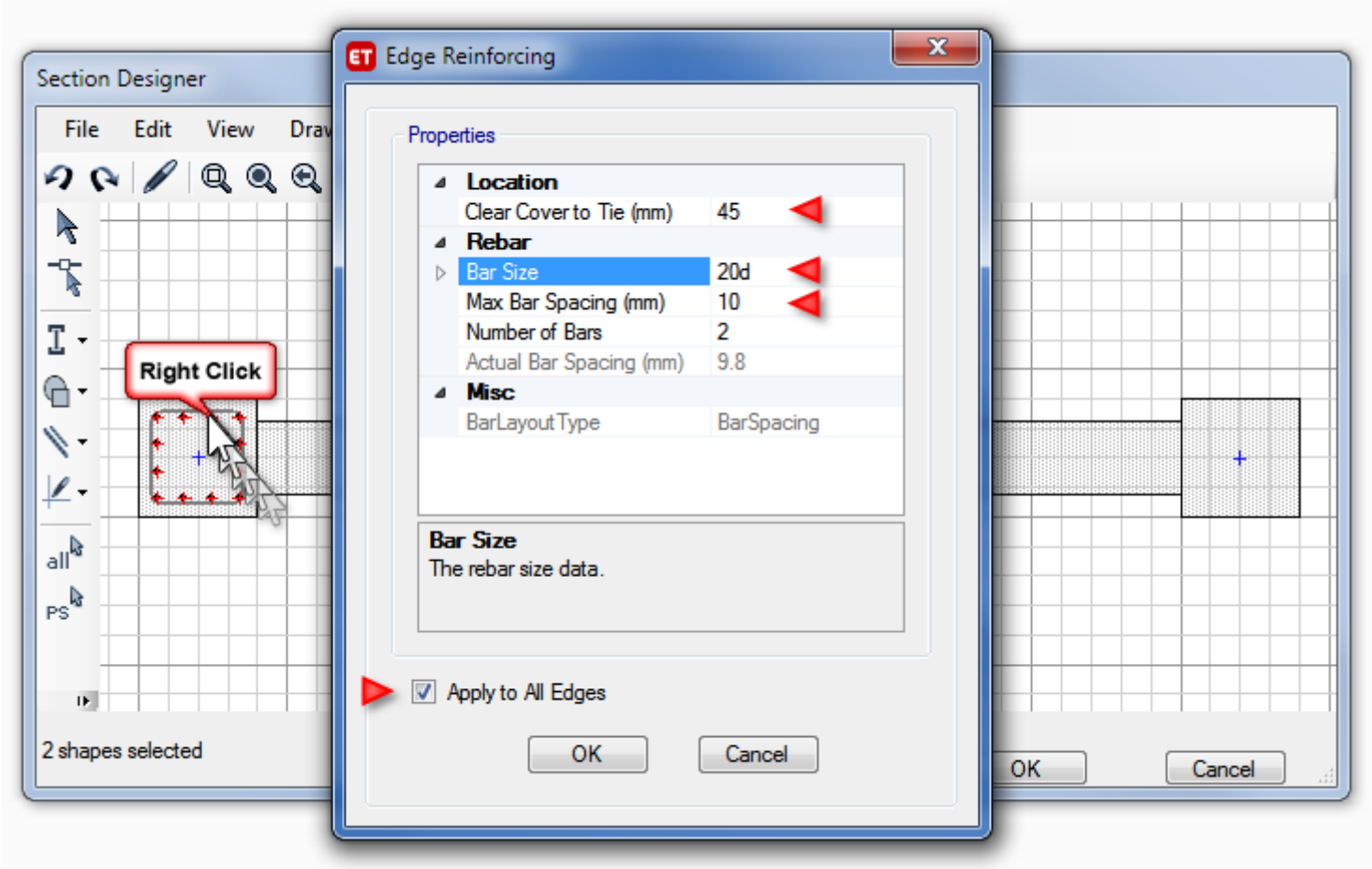
At this stage, the reinforcement details each parts of the wall section must be defined. To do this, right-click on each of the rectangles you have drawn to open the dialog box of Section Object Data. In this dialog box set the **Reinforcing** parameter to **Yes**, as indicated in the image below.



Given that in the Section Designer, the rebar cover is the code-specified cover, i.e., the distance from the concrete surface to the outer face of the tie, specify the tie diameter for the end columns, as the tie diameter, in addition to the cover, is important in calculating the centerline of the longitudinal rebars. Since the default tie in **ETABS MATE** is 10d, it is recommended to enter 10d for the tie diameter in this interface.

Upon completing this step, the section rebars are displayed on the selected rectangle, as shown in the figure below, and the precise reinforcement detailing of this shape must now be carried out.

To do this, right-click on the edge rebars to open the **Edge Reinforcing** interface, as shown below. Through this dialog box, you can specify the number of edge rebars, the spacing between rebars, and the rebar cover.



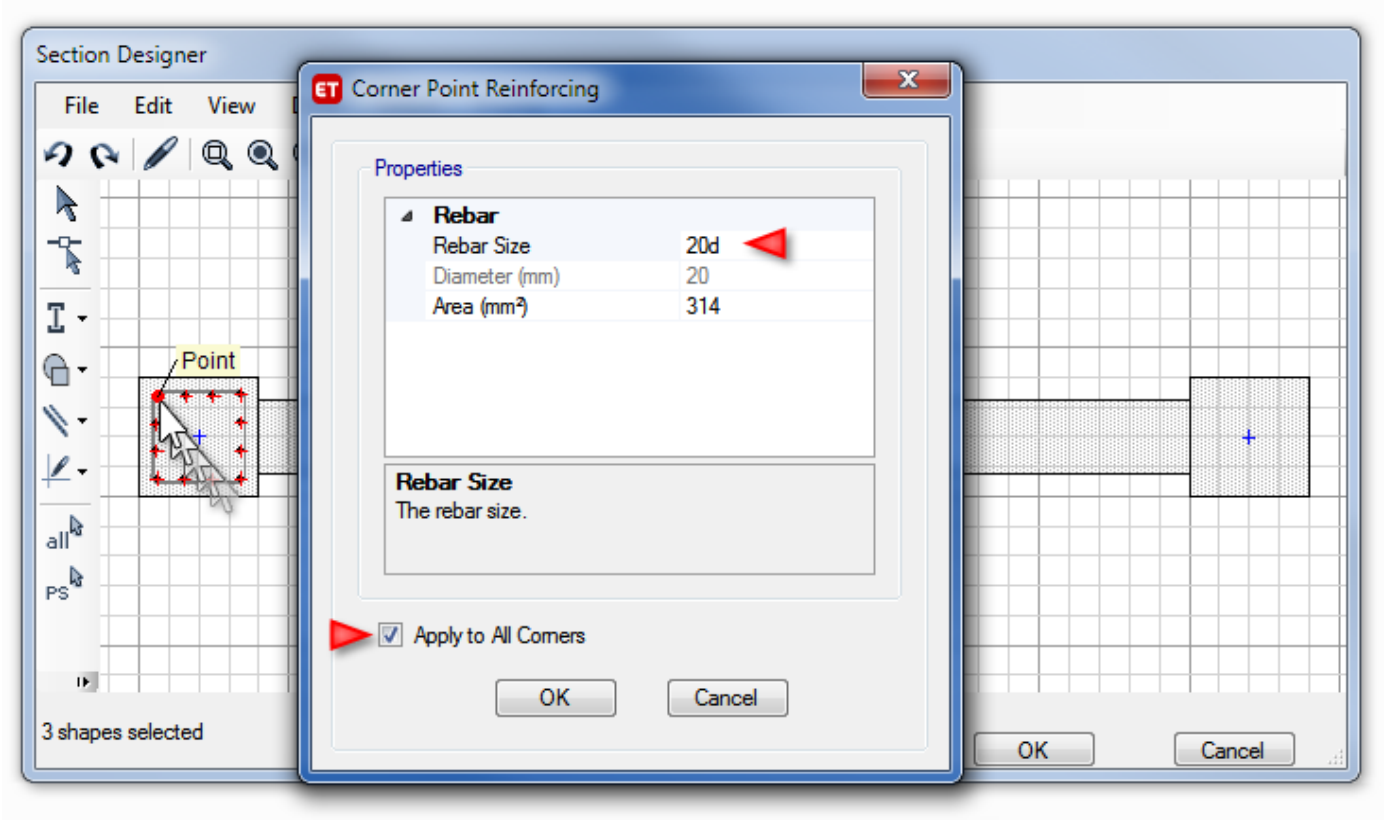
Note that the cover parameter in the **Clear Cover to Tie** field refers to the code-specified cover, i.e., the distance from the concrete surface to the outer face of the tie. Also, ensure that the cover entered here is identical to the cover defined in **ETABS MATE**, as the number of rebars is determined based on the clear length over which the rebars are distributed. If these values are not precisely set, the number of drawn rebars may differ between the two software programs.

To specify the rebar diameter in the **Bar Size** field, ensure that you select rebars in the format 16d, 18d, 20d, ….

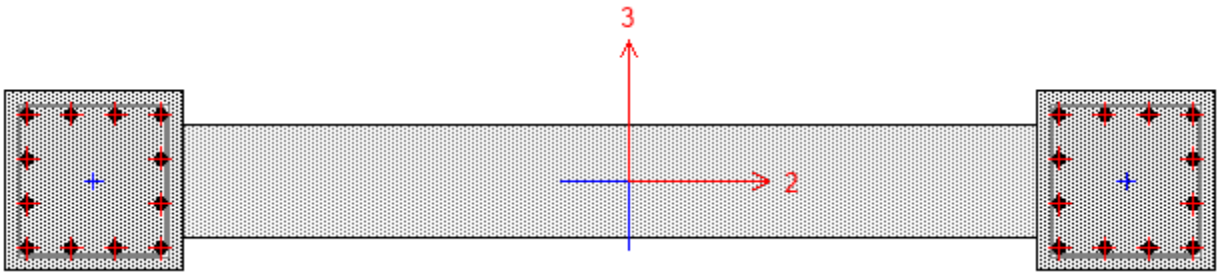
Similarly, specify the spacing between rebars in the **Max Bar Spacing** field. For the end columns, since the number of rebars on each face may vary with changes in the cover, you can specify the number of rebars in the **Number of Bars** field instead of defining the spacing. After specifying all details, also select the **Apply to All Edge** option and press the **OK** button to apply the settings to the reinforcement of all edges of the shape.

Then, to define the corner rebars, right-click on one of the corner rebars. The **Corner Point Reinforcing** interface for defining the corner rebars will appear, as shown in the image below.

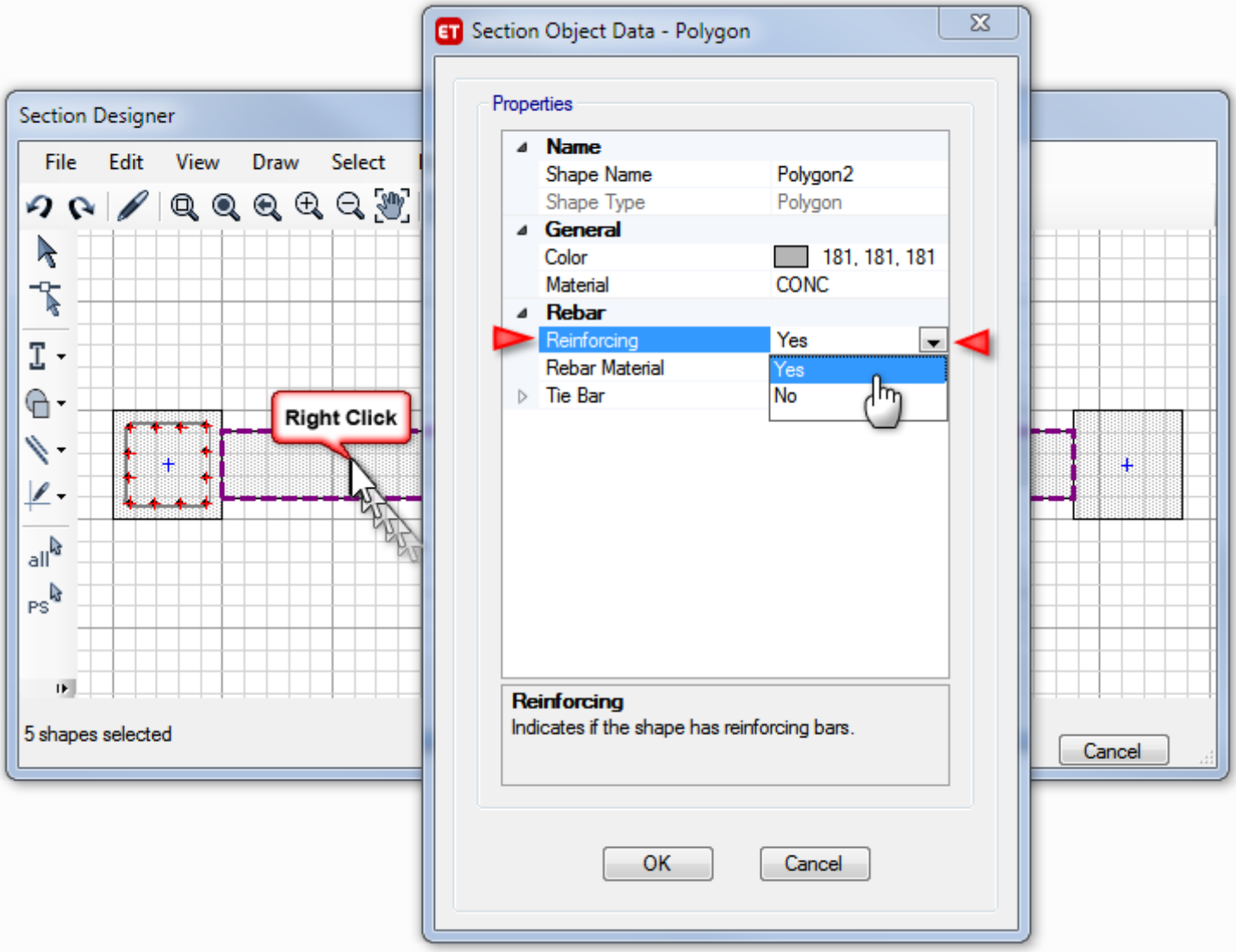
After selecting the desired rebar in the **Rebar Size** field, also select the **Apply to All Corner** option, and then click the **OK** button to apply the changes to all corners of the section.



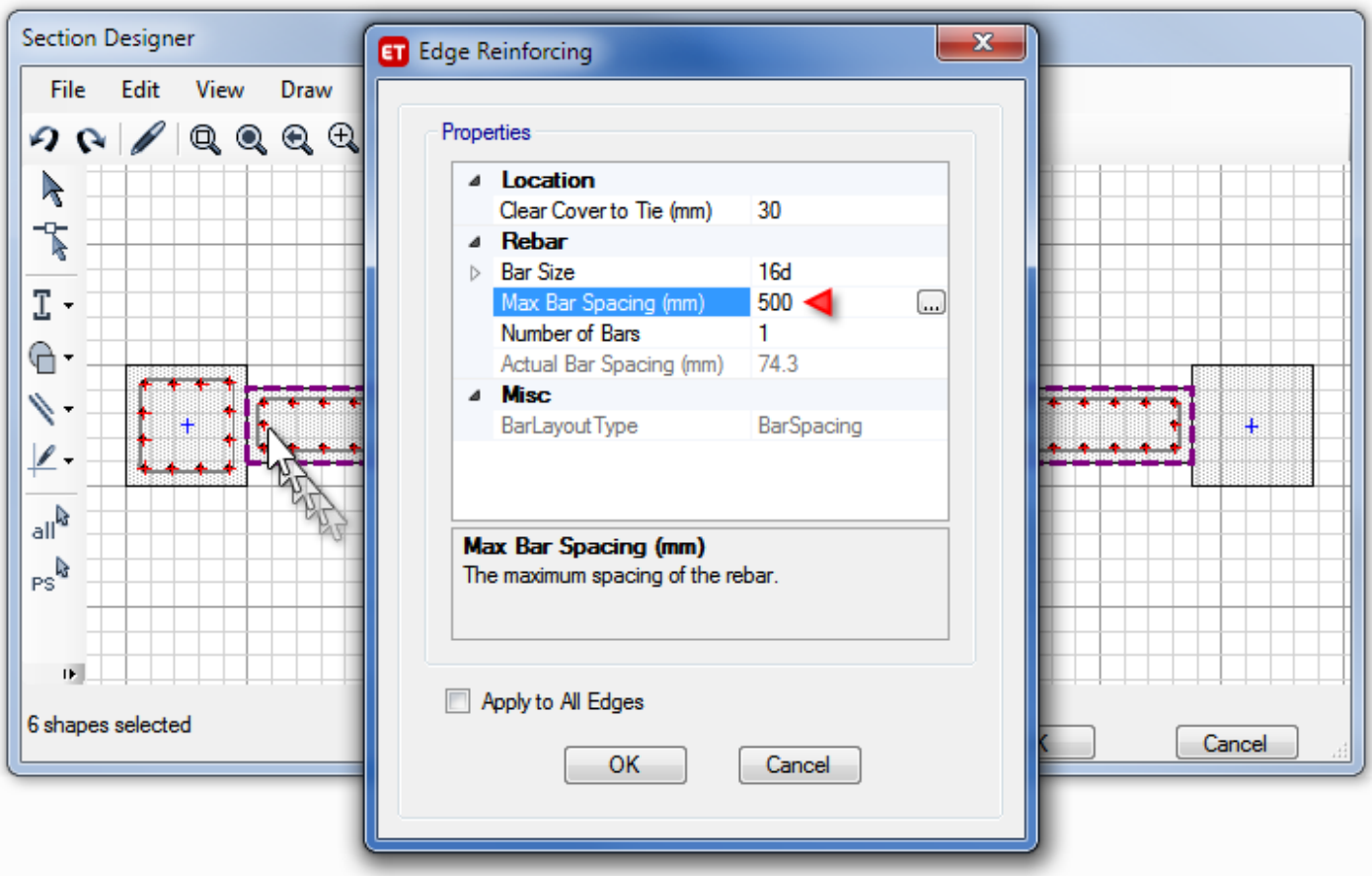
Similarly, perform the reinforcement detailing of the second column so that the section is defined as shown below. The wall web reinforcement must then be carried out, which will be explained in the following.



To define the reinforcement of the wall mid-portion, similarly to the method described above, right-click on the mid-portion of the wall to open a dialog box, as shown in the figure below. In this dialog box, as shown in the image below, select **Yes** for the **Reinforcing** option.

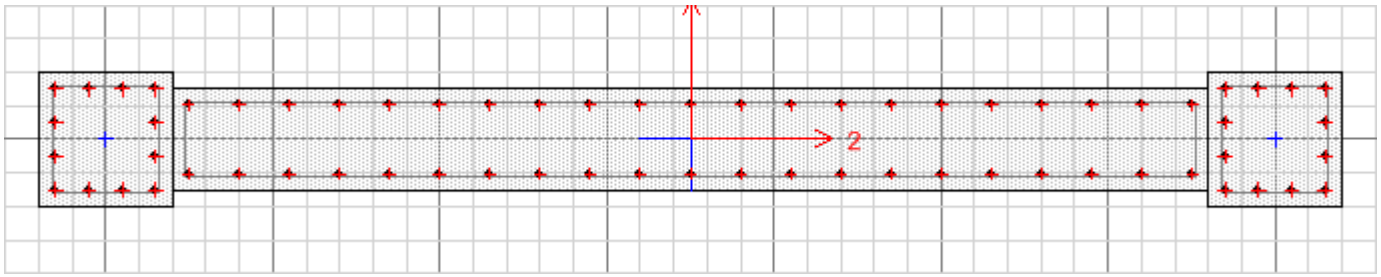


Then, as described for the first rectangle, right-click on one of the edge rebars of the wall mid-portion to precisely define the cover, rebar size, and edge rebar spacing parameters. Next, right-click on one of the corner rebars to specify the corner rebar diameter, and enable the **Apply to All** option to apply the settings to all corners and edges. Upon completing this step, the reinforcement of the wall web is also defined. However, as shown in the image below, there may be unwanted rebars present within the thickness of the wall web. To remove these rebars, proceed as follows.



To remove the unwanted rebars present within the wall thickness, right-click on the desired rebar, as shown in the figure above, to open the **Edge Reinforcing** editing dialog box. Then, for option of the **Max Bar Spacing**, as shown in the image above, enter a value greater than the wall thickness to remove these rebars.

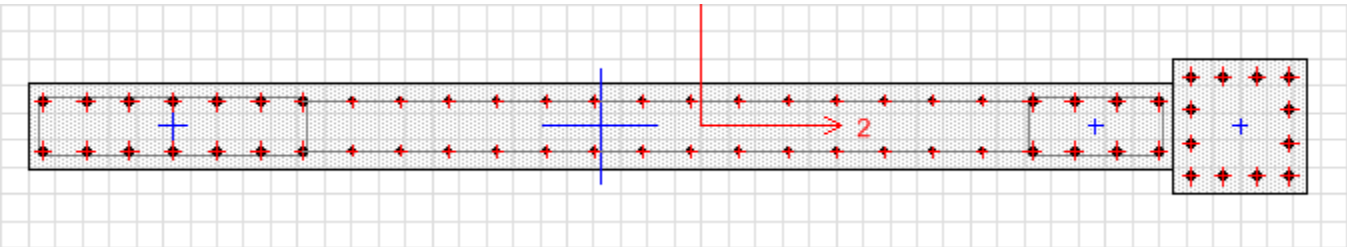
Finally, the shear wall section definition is completed, as shown in the image below. However, if you wish to use more complex reinforcement configurations, the method for defining the section is described in the next part.



Note: The basis for distinguishing a column from the wall web in **ETABS MATE** is the length-to-width ratio of the rectangles drawn in the section. In other words, if the length-to-width ratio of any defined rectangle exceeds **2.4**, the software considers that rectangle as the wall web; otherwise, it is considered a column. Note that if an element is identified as a column, the software applies the stringent column confinement provisions, including closed ties and crossties, for its detailing.

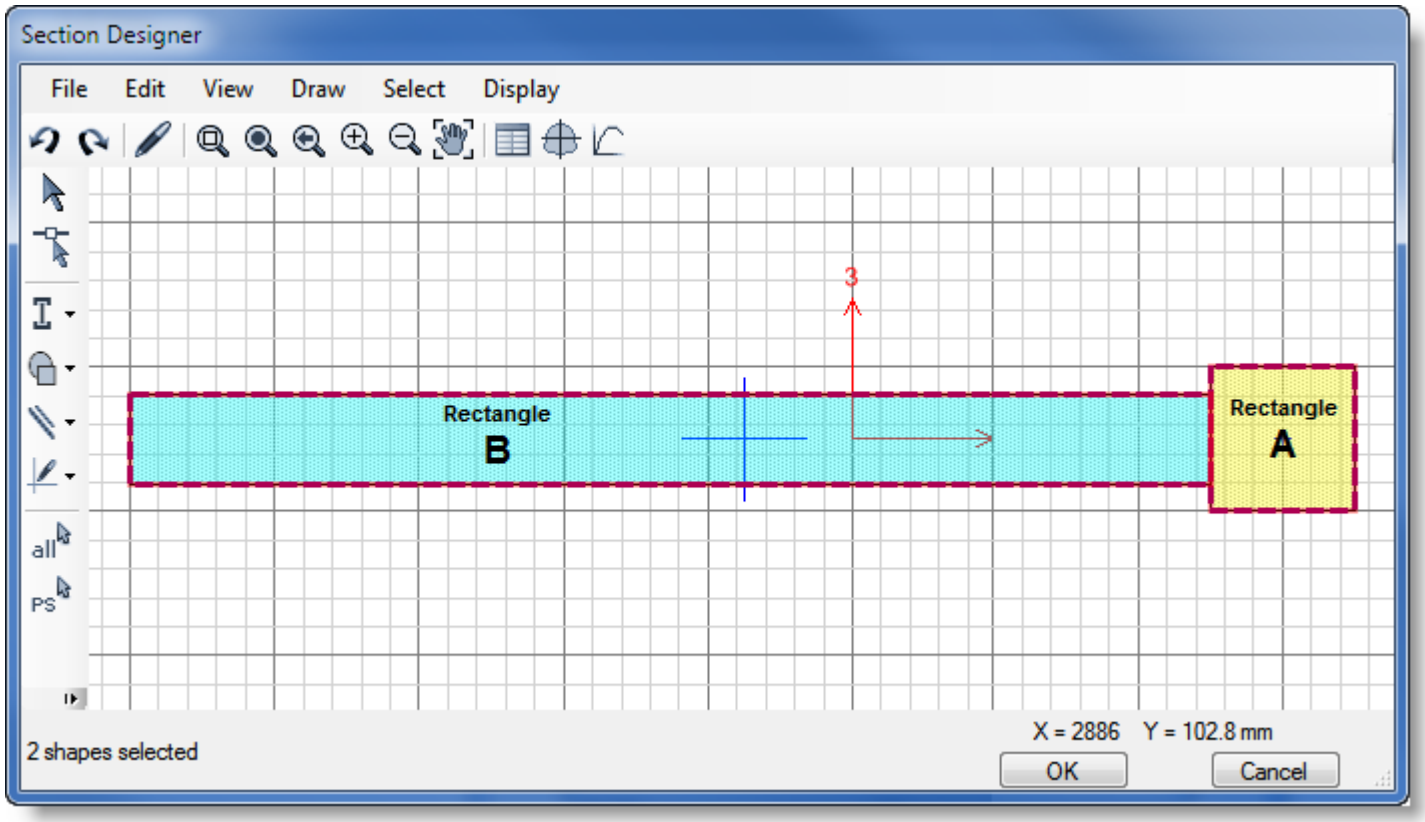
Defining Shear Wall Sections with More Complex Reinforcement

If you wish to define more complex reinforcement details in the wall section, for example, placing different rebars at the two ends of the wall web as shown in the image below, use the following method.



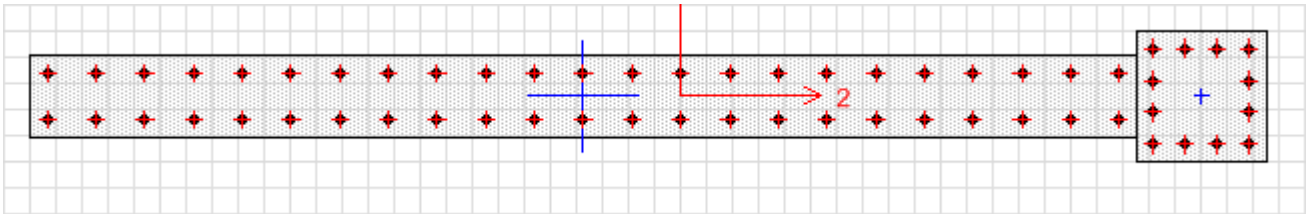
As in the previous case, a new section must first be drawn using the polygon generated by the software. Then click on the tool , and select the Draw Polygon tool Draw Polygon. Next, activate the vertex snap and, by clicking on the vertices of the existing polygon that was automatically drawn by the software, draw the two rectangles **A** and **B** over the original shape, as shown in the figure below. Then, delete the inital polygon as described in the previous part.

Note: if **ETABS version 22.5 or later** is used for modeling, these polygons are automatically defined as separate components, and performing this step and separating them is no longer necessary.

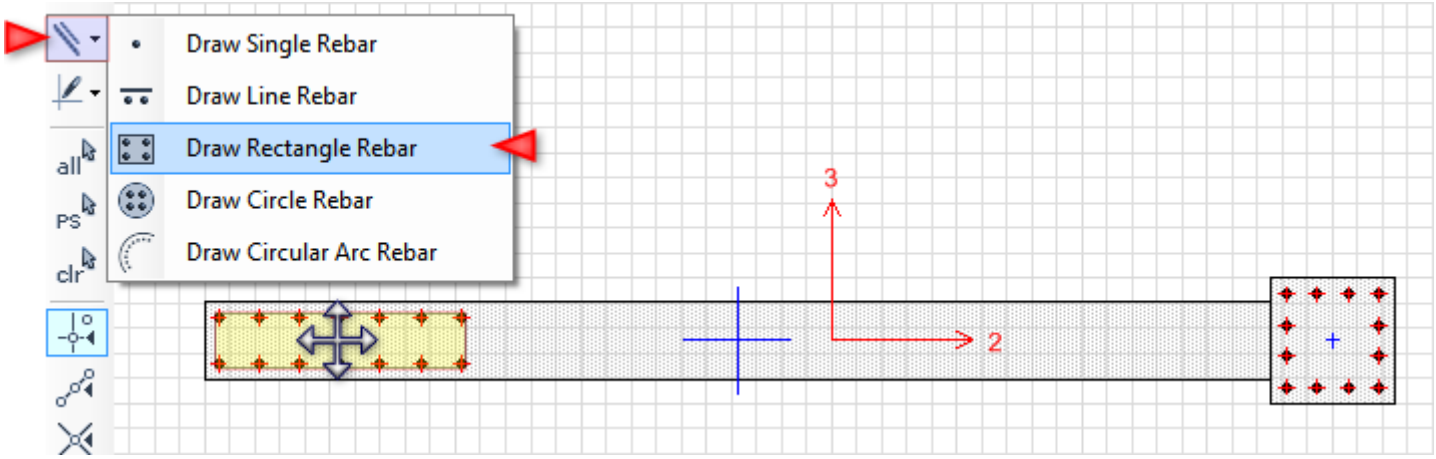


Then, for column part **A**, as in the previous case, right-click on the column and, in the dialog box that appears, set the **Reinforcing** parameter to **Yes**. Once the column reinforcement is displayed, define its rebars.

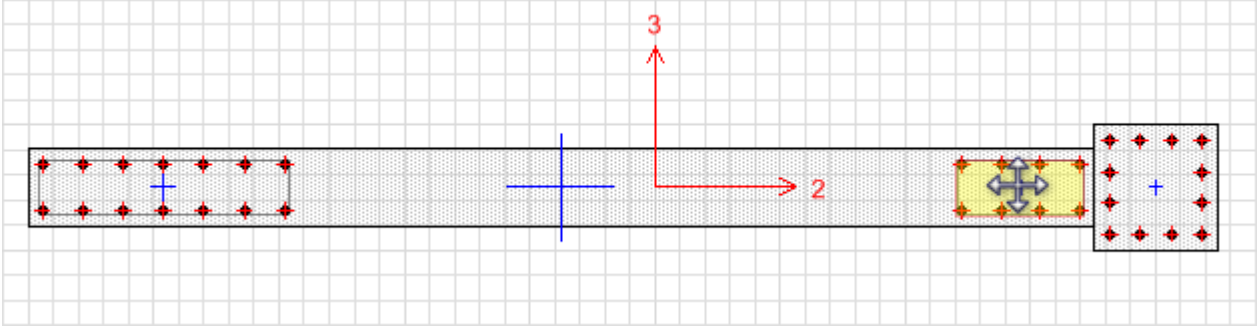
If uniform reinforcement is also desired for part **B**, right-click on it and again select **Yes** for the **Reinforcing** parameter. Once the rebars appear, define them. In this case, if the end rebars located in the boundary zone, **ETABS MATE** automatically draws a closed tie around them, or in other words, provides them with transverse reinforcement similar to a column element.



However, if you wish to concentrate more rebars at the end portions of the wall section, from the **Draw Rebar Shape** tools , use the **Draw Rectangle Rebar** tool Draw Rectangle Rebar to draw the rebars at the left end of the wall. Then, carefully adjust their placement and reinforcement as shown in the image below.

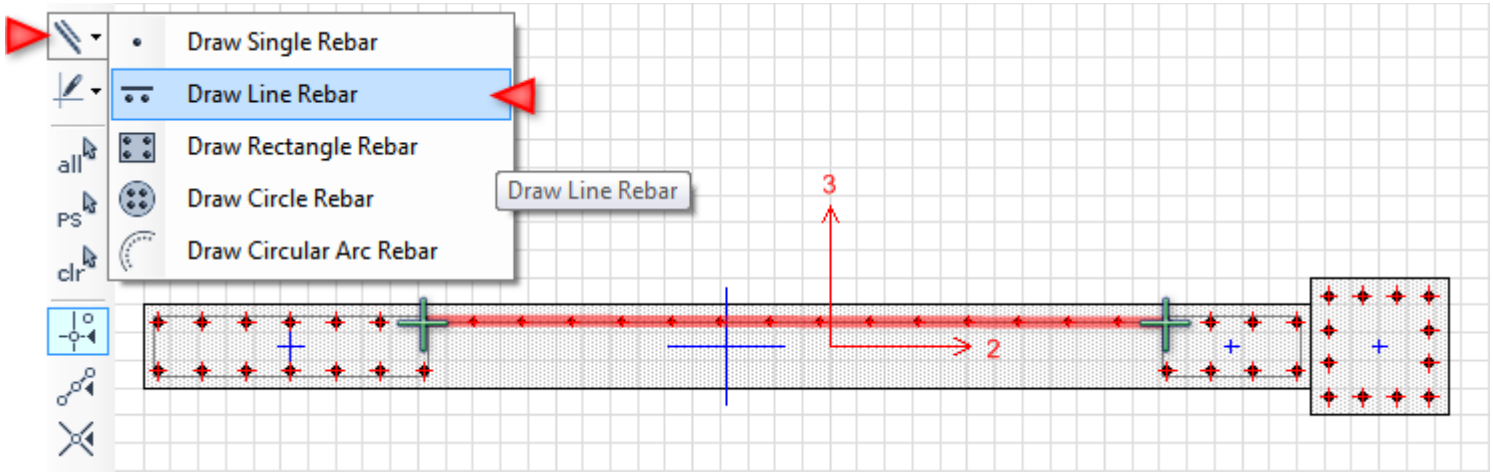


Again, using the tool Draw Rectangle Rebar, draw the rebars at the right end of the wall, and then carefully adjust their placement and reinforcement configuration as shown in the image below.

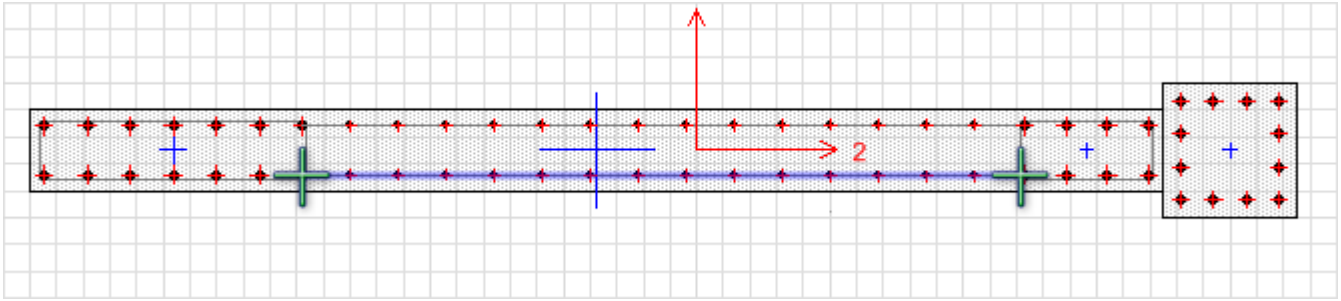


Now, the reinforcement configuration at the end regions of the wall has been defined, and only the rebar arrangement in the mid-portion of the wall remains. To define the wall rebars in the mid-portion, proceed as follows:

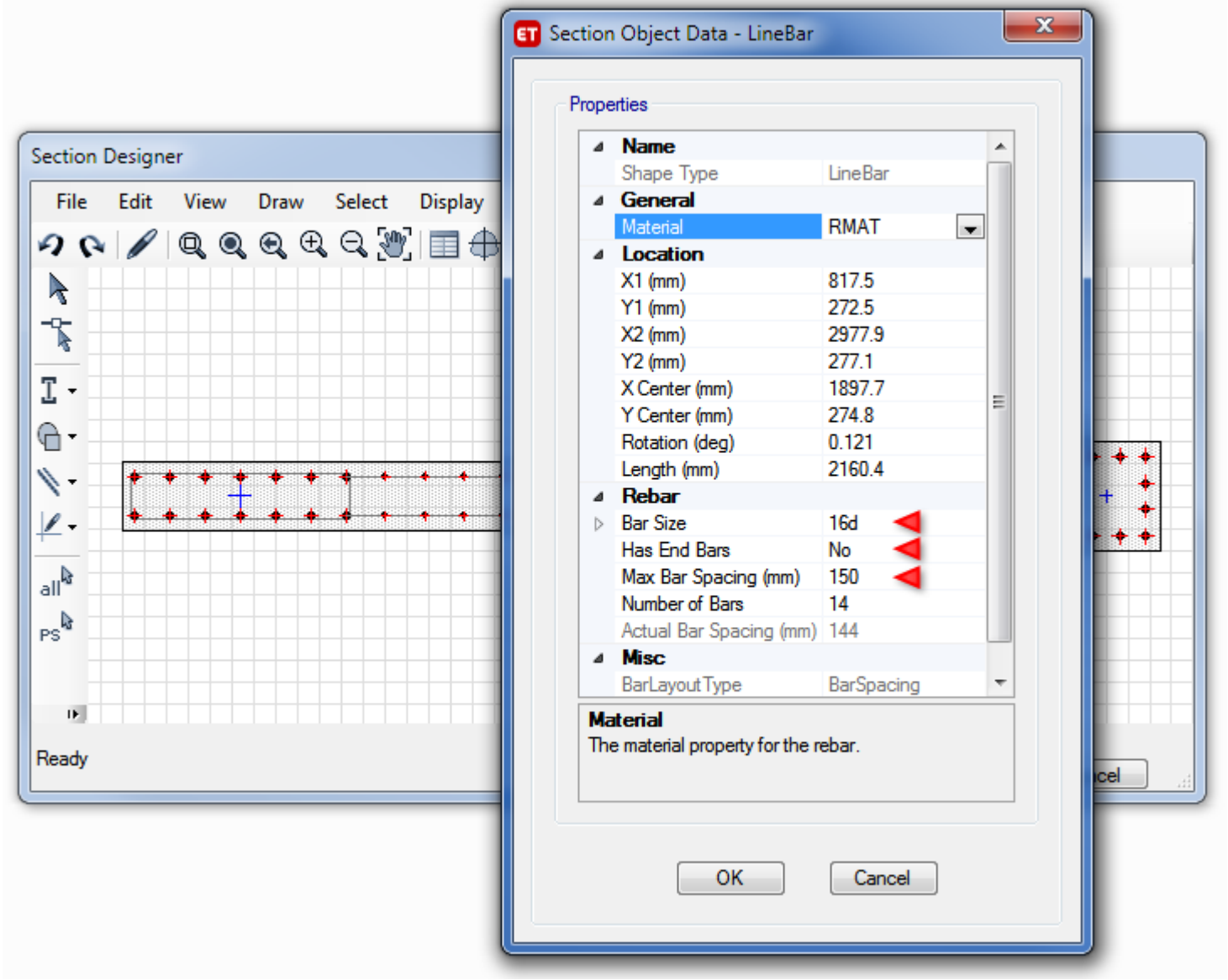
From the **Draw Rebar Shape** tools , use the **Draw Line Rebar** tool Draw Line Rebar to draw the rebars in the upper mid-portion of the wall. To do this, using the point snap, click on the endpoints of the last rebars placed within the rectangles, as shown in the figure below, to draw the mid-portion rebars.



Again, using the **Draw Line Rebar** tool Draw Line Rebar, draw the rebars in the lower mid-portion of the wall. To do this, using the point snap, click on the endpoints of the last rebars placed within the rectangles, as shown in the figure below, to draw the mid-portion rebars as illustrated.

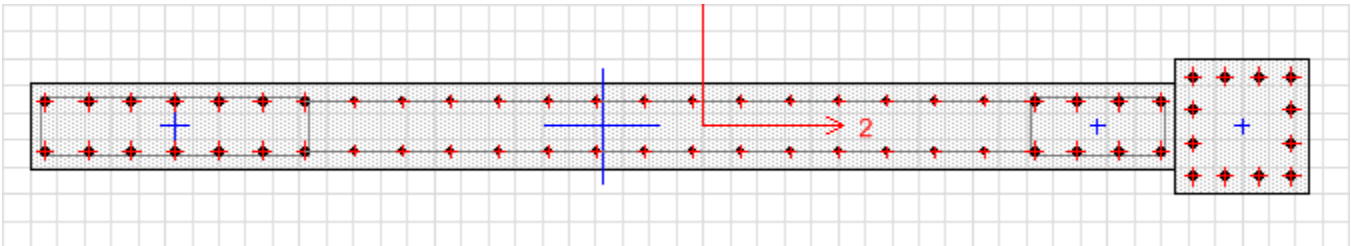


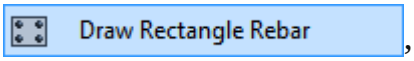
Afterward, right-clicking on one of the **Line Rebar** in mid-portion of section at the top and bottom of the wall, to opens the linear reinforcement settings dialog box **Section Object Data - LineBar**, as shown below.



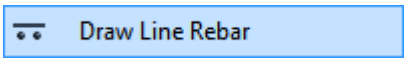
Using the parameters available in this dialog box, specify the **Bar Size** and the **MAX Bar Spacing** of Line Rebars. Then, set the **Has End Bars** parameter to **No** to remove the end bars of the linear reinforcement, preventing overlapping rebars. Now, the linear reinforcement details for the mid-portion of the wall at the top and bottom are fully defined.

The final defined shear wall section will appear as shown below.

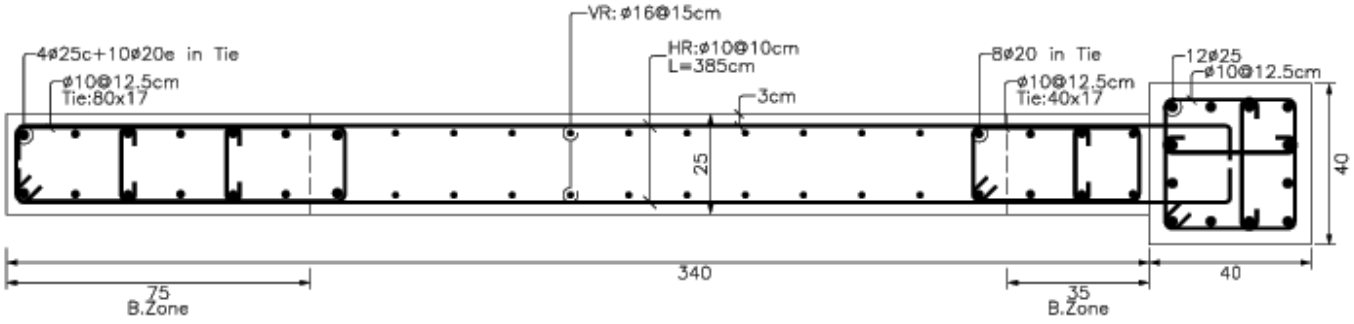


Note: **ETABS MATE** uses a perimeter closed tie and intermediate crossties to restrain the rebars drawn by using the **Draw Rectangle Rebar** tool 

, similar to a column, as shown in the image below.

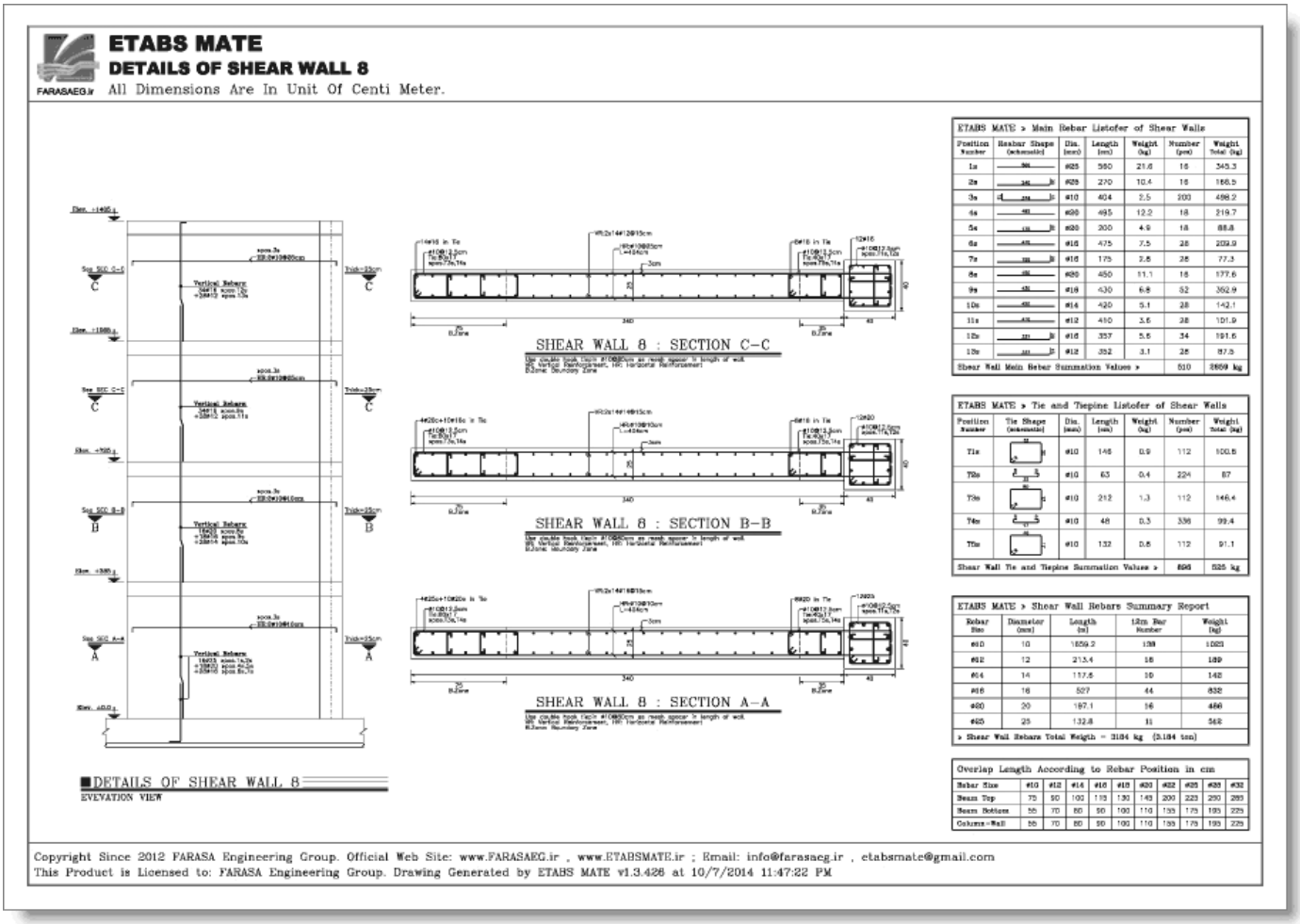
And rebars that are drawn by using the **Draw Line Rebar** tool 

are restrained by crossties within the boundary zone, as shown in the image below.



SHEAR WALL 8 : SECTION A–A

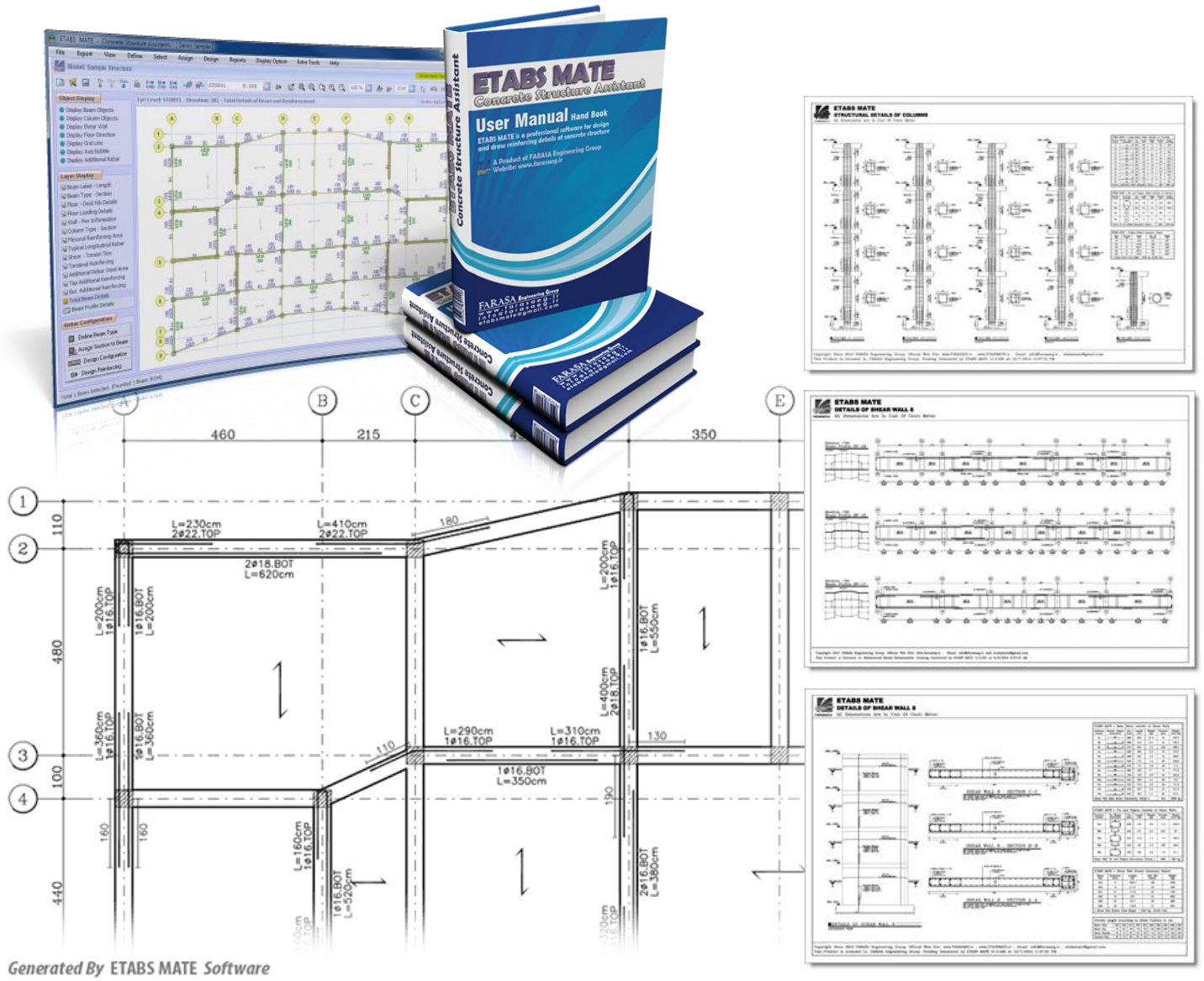
Use double hook tiepin ø10@80cm as mesh spacer in length of wall.
VR: Vertical Reinforcement, HR: Horizontal Reinforcement
B.Zone: Boundary Zone



ETABS MATE
Concrete Structure Detailing Software

Given the new features introduced in each software update and the continuous revisions to this guide, it is recommended to always download the latest version of the software guide from the **download** section of our official website and replace the previous file in the software installation folder. Furthermore, highly useful supplementary material is regularly provided in the **Articles** section of the website, and it is recommended to download and study these materials as well.

To obtain a trial version of **ETABS MATE**, sample drawings generated by the software in AutoCAD format, and to access educational articles and further information, please visit the official website and the social media pages of the software.



ETABS MATE
Concrete Structure Detailing Software

FARASA Engineering Group Contact Information

Official Web Sites Address	www.etabsmate.com www.etabsmate.ir www.foundamate.ir www.farasaeg.ir
Email Address	etabsmate@Gmail.com
Main Office Address	Unit Number 119 and 120, Second Floor, Hashemi Building, Enghelab Street, Shiraz, Iran
Phone Number	+989173171373 +989301325576
WhatsApp, Telegram and Other Messaging App	+989173171373 @etabsmate
Fax/Tel Number	+987132323810 +987132323811 +987132323812 +987132323813
Instagram Page	instagram.com/etabsmate **@etabsmate**
Rubika Page	rubika.ir/etabsmate **@etabsmate**
Aparat Channel	aparat.com/etabsmate **@etabsmate**
You Tube Channel	youtube.com/@etabsmate **@etabsmate**
Telegram Channel	telegram.me/etabsmate **@etabsmate**

FARASA Engineering Group
Contact information QR Code