FOUNDA MATE® User Manual

Detailing & Structural Drawing Generation in 4 Steps

Document Edition: EN350

FARASA Engineering Group

FARASA Engineering Group

Document Number: 2608143502

FOUNDA MATE
Foundation Slab Detailing Software

FOUNDA MATE
Software User Manual

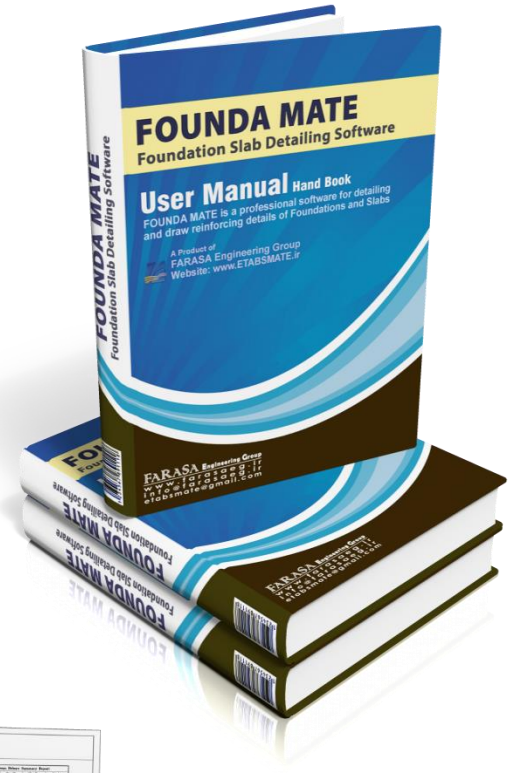
Document Edition: EN350

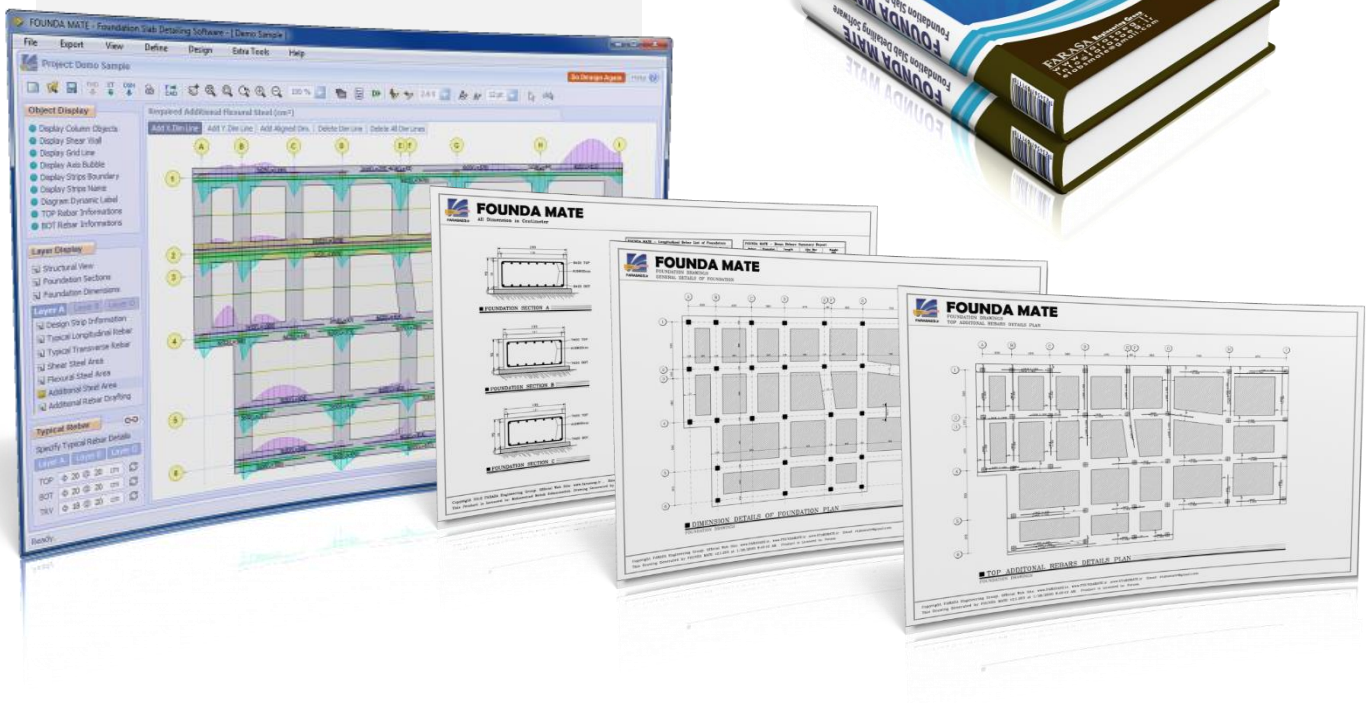
Document Language: English

Revision Date: July 2026

Publisher: FARASA Engineering Group

Document Number: EN2608213502





Copyright & Terms of Use

FOUNDA MATE
Foundation Slab Detailing Software

Table of Contents

FOUNDA MATE
Foundation Slab Detailing Software

FOUNDA MATE
Quick Start in 4 Simple Steps

The overall design philosophy of **FOUNDA MATE** is centered on simplicity, ease of use, and high-speed execution of rebar detailing calculations, while maintaining the highest possible level of user interactivity. To this end, the software's graphical interface has been designed so that all engineers familiar with other engineering software can easily connect with its tools and user interfaces. As a result, learning the software is very straightforward, and users will achieve the necessary proficiency within a short period, enabling them to complete the rebar detailing and construction drawing generation for a project in just a few minutes.

To use this software for rebar detailing, material takeoff, and the generation of construction drawings for concrete foundations, the structure must first be modeled, analyzed, and designed in SAFE. Fortunately, importing the structural model into this software does not require any specific naming conventions for elements or particular foundation geometry drawing methods, and you can model your project in SAFE as usual. Once the modeling and design in SAFE is complete and all design results are satisfactory, you can easily proceed with the rebar detailing and construction drawing generation process in **FOUNDA MATE**.

The entire process of rebar detailing and structural drawing generation using **FOUNDA MATE** is accomplished in just four simple steps. In this guide, you will become familiar with these four simple steps in a very short time.

To achieve the best results in detail design and drawing generation, please carefully study and follow the four simple steps below:

1. **Exporting the Model from SAFE**
2. **Importing the Structure into FOUNDA MATE**
3. **Configuring Design Parameters and Detailing the Reinforcement**
4. **Generation of Structural Drawings**

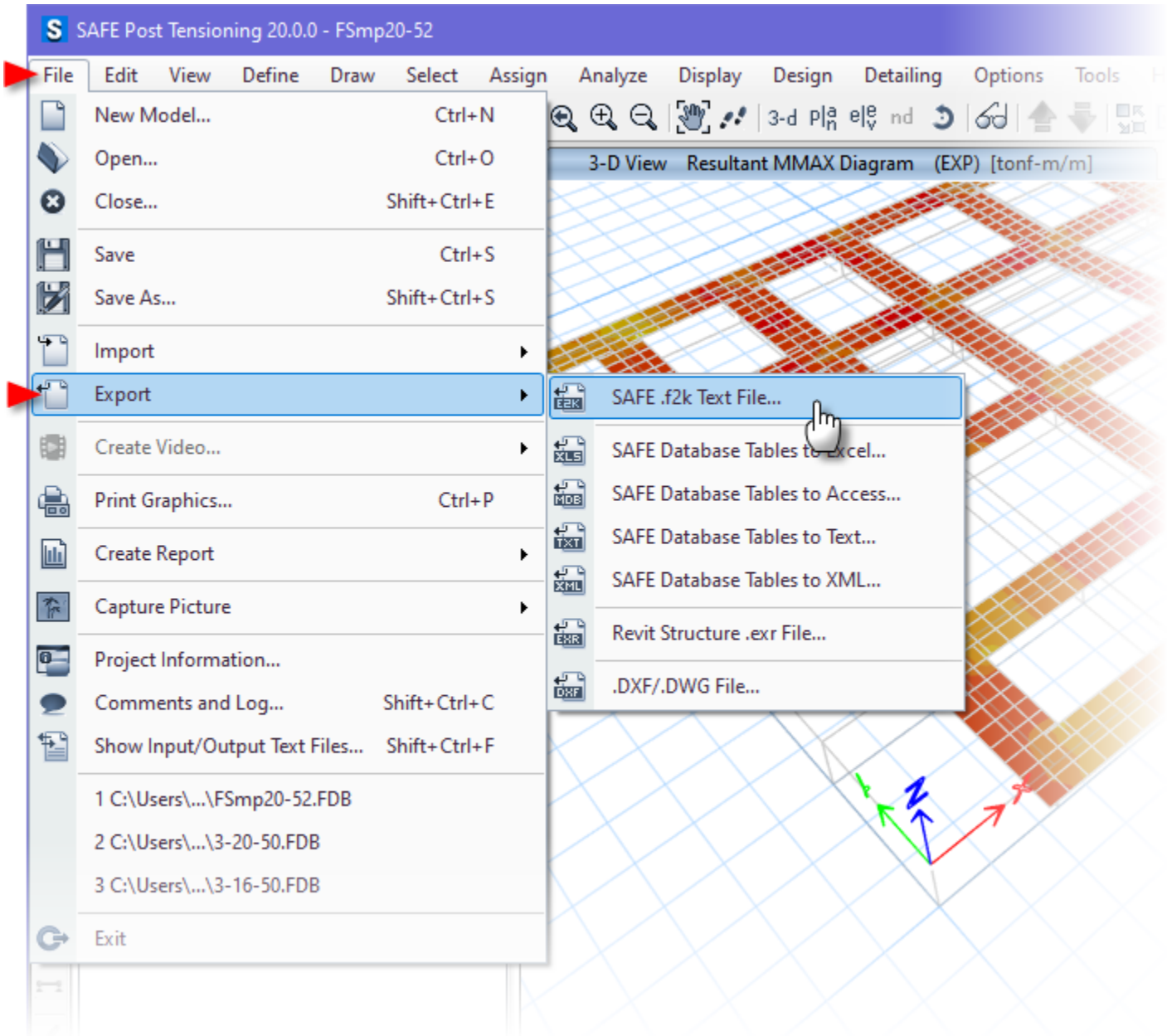
In the following sections, detailed and illustrated explanations of these four simple steps are provided. After studying this article, you will be able to complete the entire process of rebar detailing, material takeoff, and high-quality construction drawing generation quickly and easily using this software.

FOUNDA MATE
Foundation Slab Detailing Software

1 Exporting the Model from SAFE

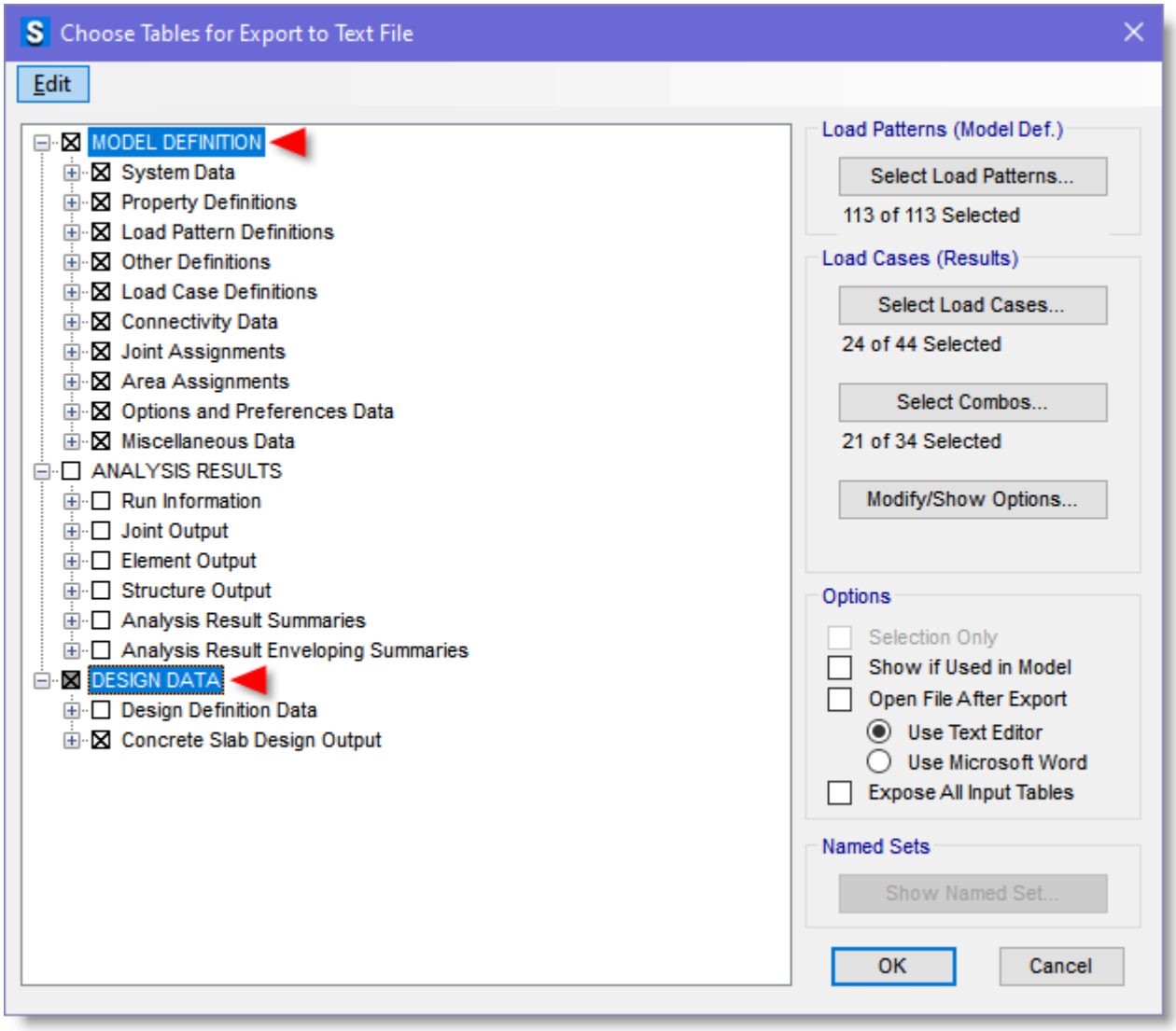
After completing the modeling in SAFE, performing the analysis and design processes, and achieving satisfactory results, you must transfer the model data to **FOUNDA MATE**. To do this, set the units in SAFE to **Kgf, cm** and then proceed as follows from the File menu:

File Menu > Export > SAFE .f2k Text File...



Executing this command opens the table selection window, as shown in the image below. Use this window to export and save the tables containing the information required by the software, as described below:

1. Select the **MODEL DEFINITION** table.
2. Select the **DESIGN DATA** table (or **DESIGN RESULTS** in earlier versions).
3. Press the **OK** button to save the file containing the model information with **Kgf, cm, C** units at your desired location.



Note: The data export from **SAFE** must be performed using the **Kgf, cm** unit system. For large projects, to reduce file size and increase saving speed, you may select only the **Concrete Slab Design Output** tables instead of the entire **DESIGN DATA** tables.

Several Critical Notes

To achieve high quality stgructural drawings, please observe the following points during modeling and model transfer:

1. The more care taken during modeling, particularly in drawing the design strips in SAFE, the higher the accuracy of the results and the better the quality of the structural drawings generated by the software.

2. To transfer model data to **FOUNDA MATE**, the unit system in SAFE must be set to **Kgf, cm, C**.

3. For special cases, you may save the **MODEL DEFINITION** and **DESIGN DATA** tables in two separate files and import them into the software in two stages, as described in the model import section.

4. To reduce the output file size and increase saving speed, you may select only the **Concrete Slab Design Output** table (or the **Concrete Design** table in earlier versions of SAFE) in the **DESIGN DATA** section.

5. The **DESIGN DATA** tables (or **DESIGN RESULTS** in earlier versions) become active only after the model has been analyzed and designed. Therefore, analyze and design the model before saving the data.

6. In SAFE, you may use the various layers **A, B**, and **Other** without any restrictions to separately define design strips in different directions. However, it is recommended to define horizontal or nearly horizontal design strips in layer **A**, and vertical or nearly vertical design strips in layer **B**, so they can be managed independently.

7. The software can detail all design strips in layers **A, B**, and **Other** collectively on a single plan, or draw each layer separately on individual structural drawings.

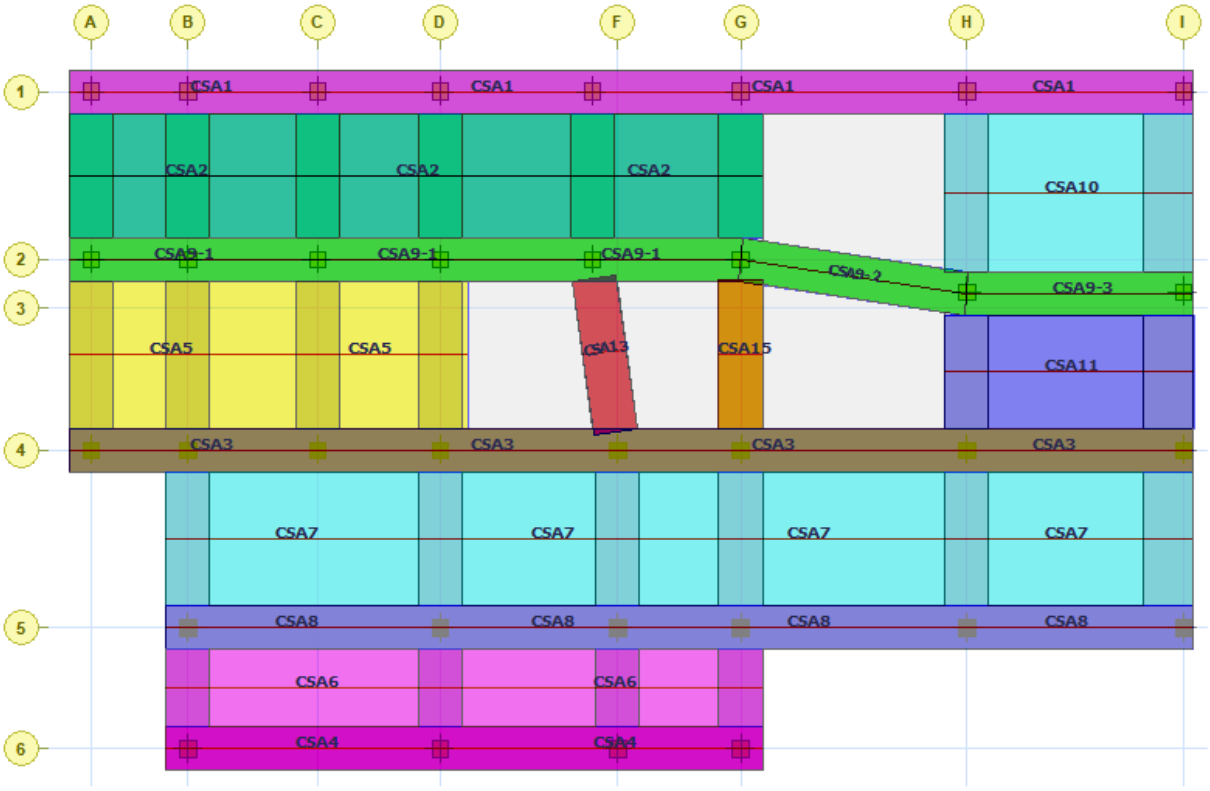
8. All parts of the foundation must be covered by design strips in both the direction of the foundation strips and the direction perpendicular to them. In other words, as shown in the images on the following page, the layer **A** design strips, which run horizontally, cover the entire foundation, and the layer **B** design strips, which run vertically, also cover the entire foundation. This is also illustrated graphically on the following page.

   Please note that if this point is not observed when drawing the design strips, the transverse foundation rebars will not be drawn, and consequently, they will not be included in the rebar schedule or the rebar weight calculation. For further explanation, please refer to the example on the following page.

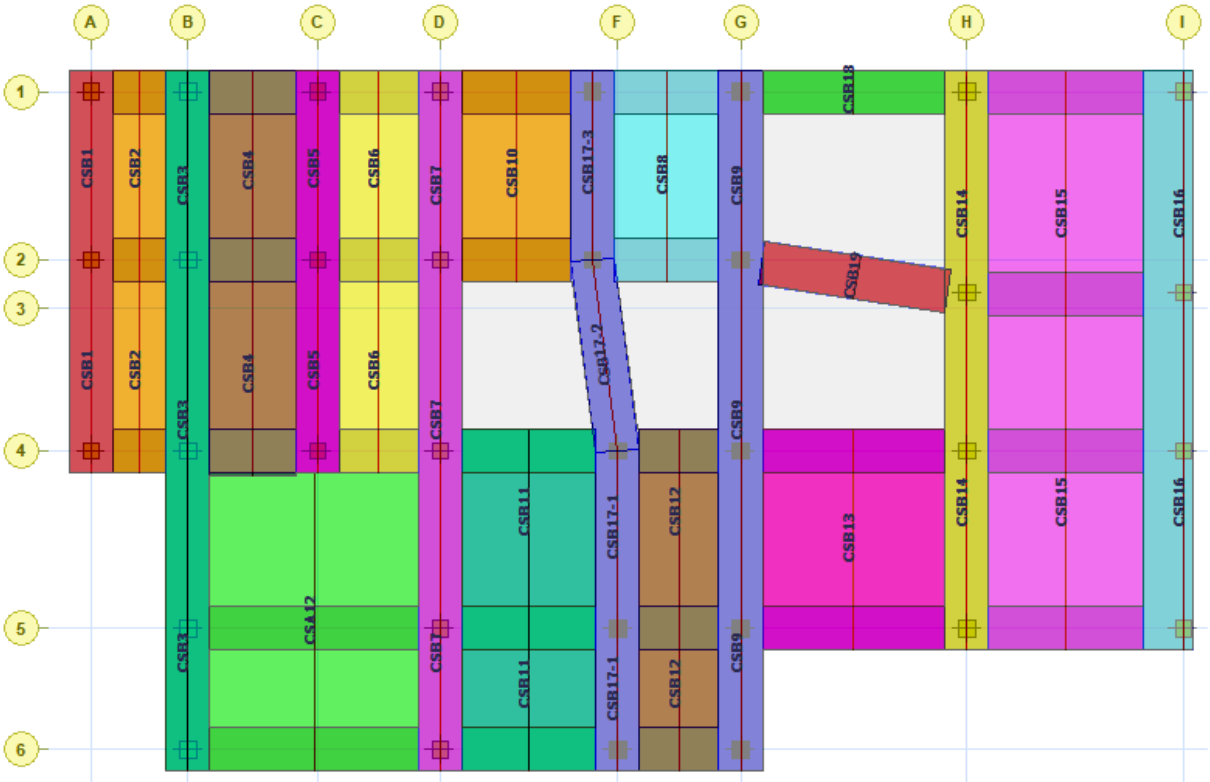
FOUNDA MATE
Foundation Slab Detailing Software

For example, design strip **CSA1** covers the horizontal foundation strip on **Axis 1**, and design strip **CSA2** covers the vertical foundation strips located between **Axes 1 and 2**. As can be seen, these strips also pass over the openings.

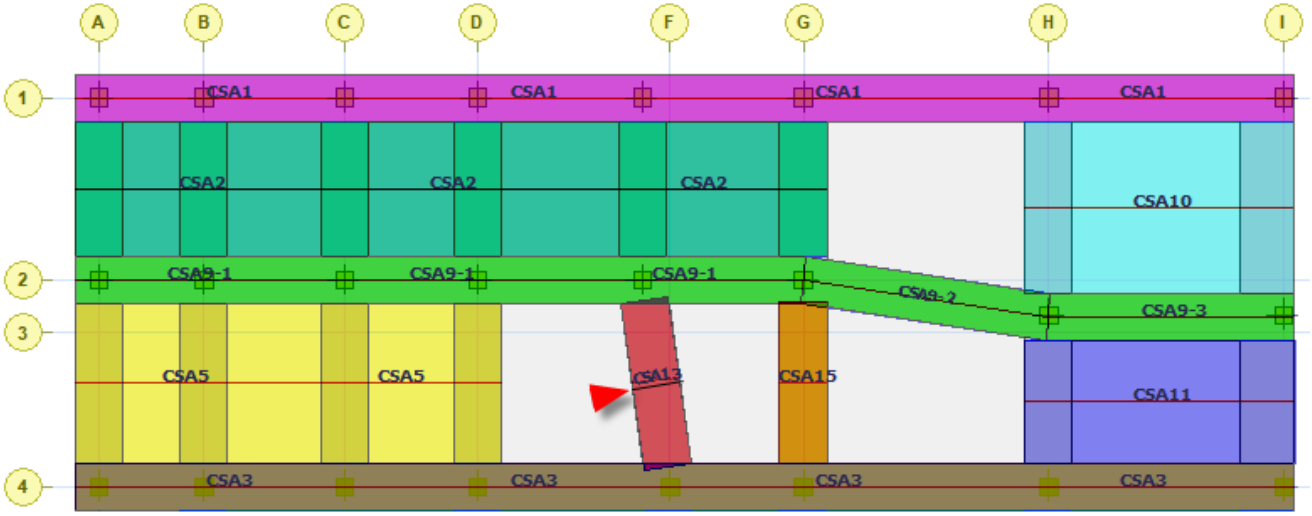
Design Strips in **Layer A**



Design Strips in **Layer B**



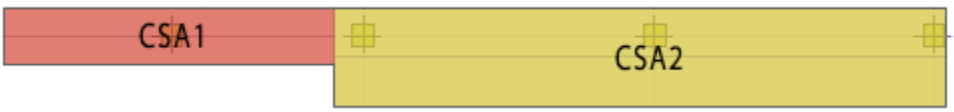
9. For more accurate calculations and better detail drawings, it is recommended that the design strips be drawn along the axis of the strip or perpendicular to the foundation strip. For example, in the image below, design strip **CSA9** is drawn along the foundation strip direction in all its segments, and design strip **CSA13** is drawn perpendicular to the foundation strip.



10. If the width of a foundation strip changes in a section, do not divide the design strips along the length of the strip into multiple parts. For example, if your foundation is shaped as follows:

☑ **The correct method for drawing the design strips is as follows:**

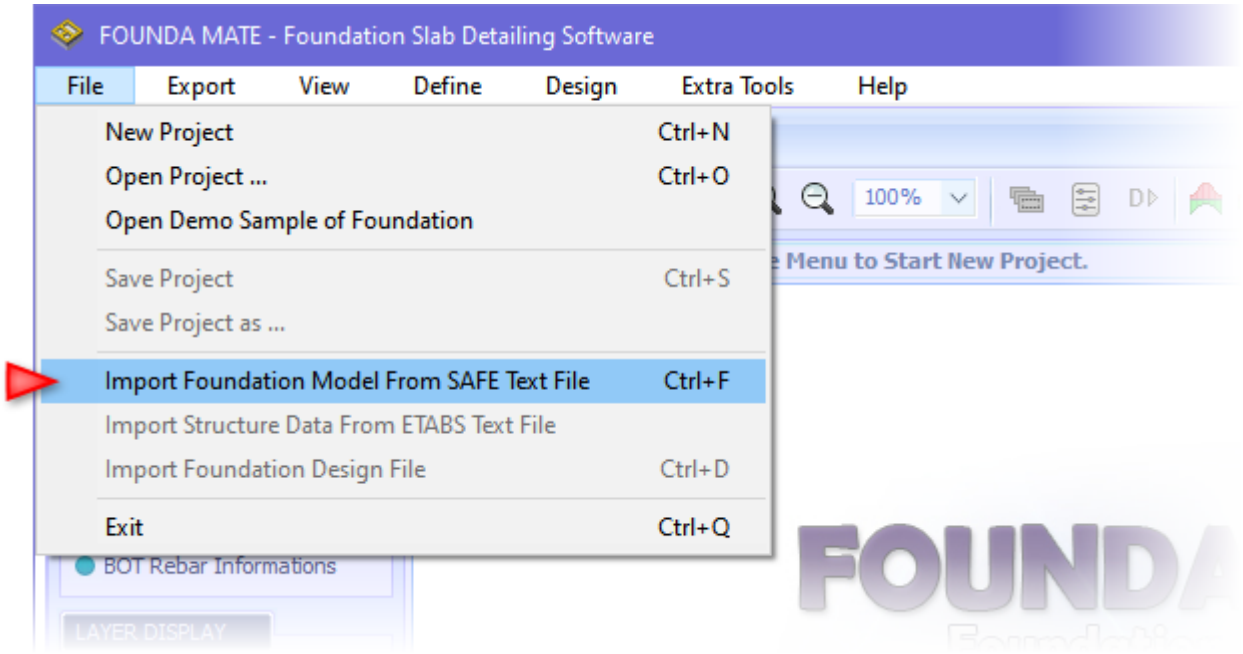
✕ **The incorrect method for drawing the design strips is as follows:**



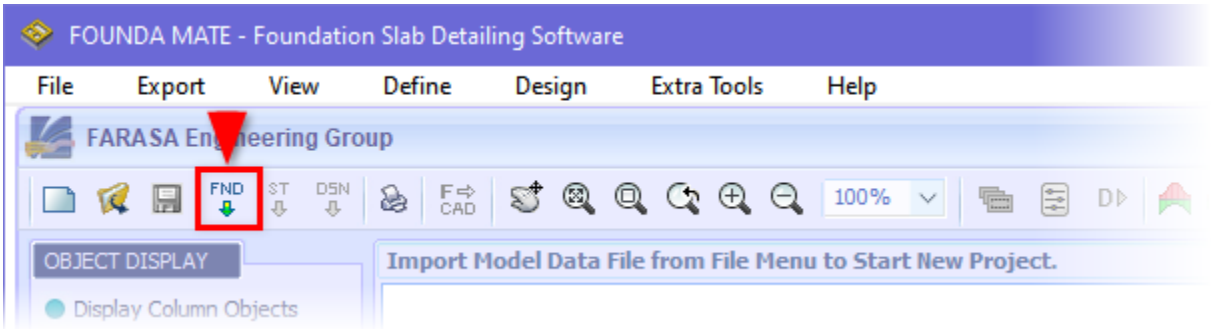
2 Importing the Model into FOUNDA MATE

To import the model into **FOUNDA MATE**, import the text file, i.e., the **F2K** model file generated in the previous step, into the program via the File menu as follows. This file may contain both the model geometry data and the design results, or solely the model geometry data. However, it is recommended to import all geometry and design data together into the software.

File Menu > Import Model From SAFE Text File (Ctrl + F)



As shown in the image below, for greater convenience and speed, you may also use the FND icon in the main toolbar at the top of the software to import the model.

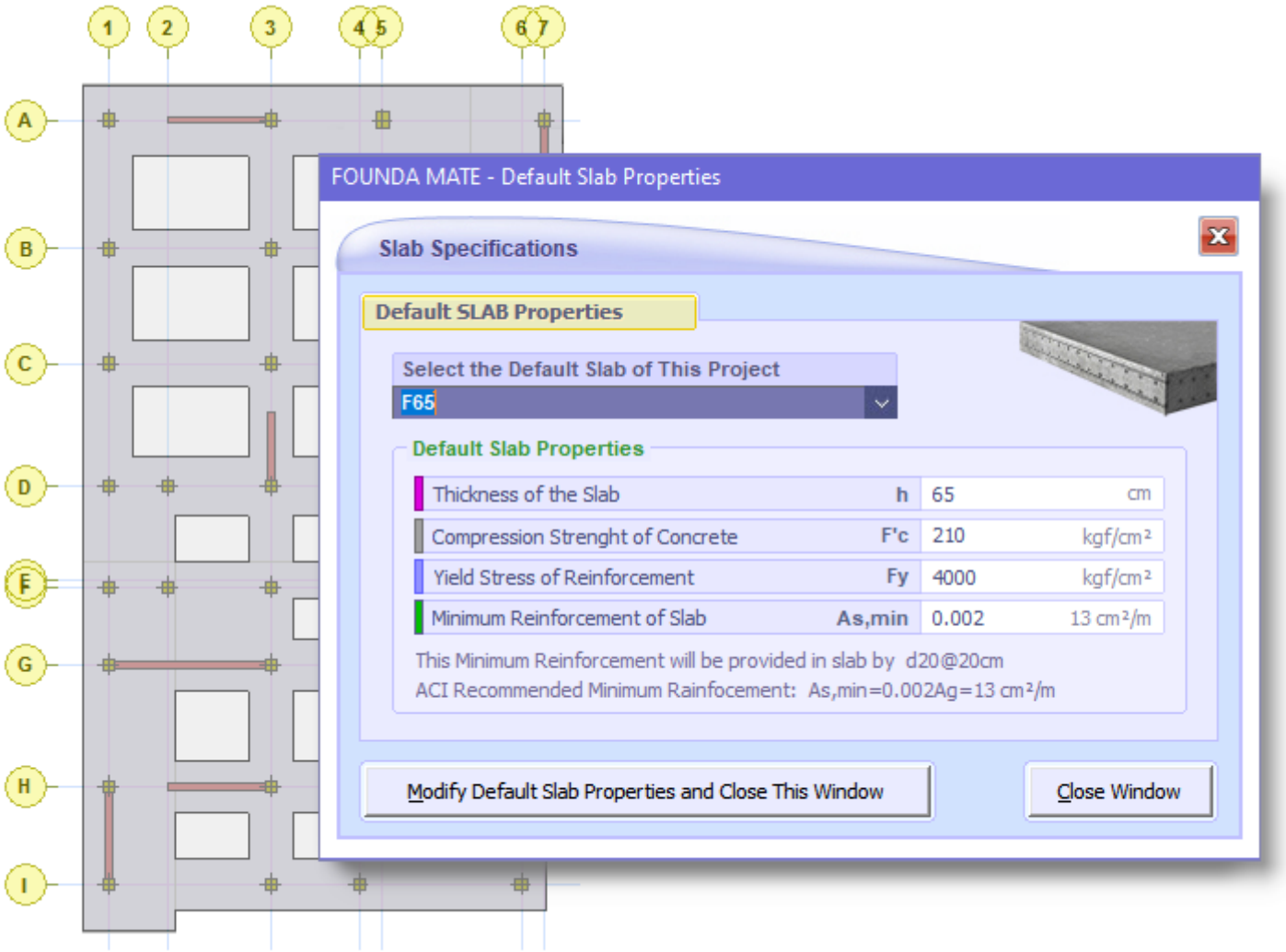


FOUNDA MATE
Foundation Slab Detailing Software

Specifying the Concrete Slab Properties

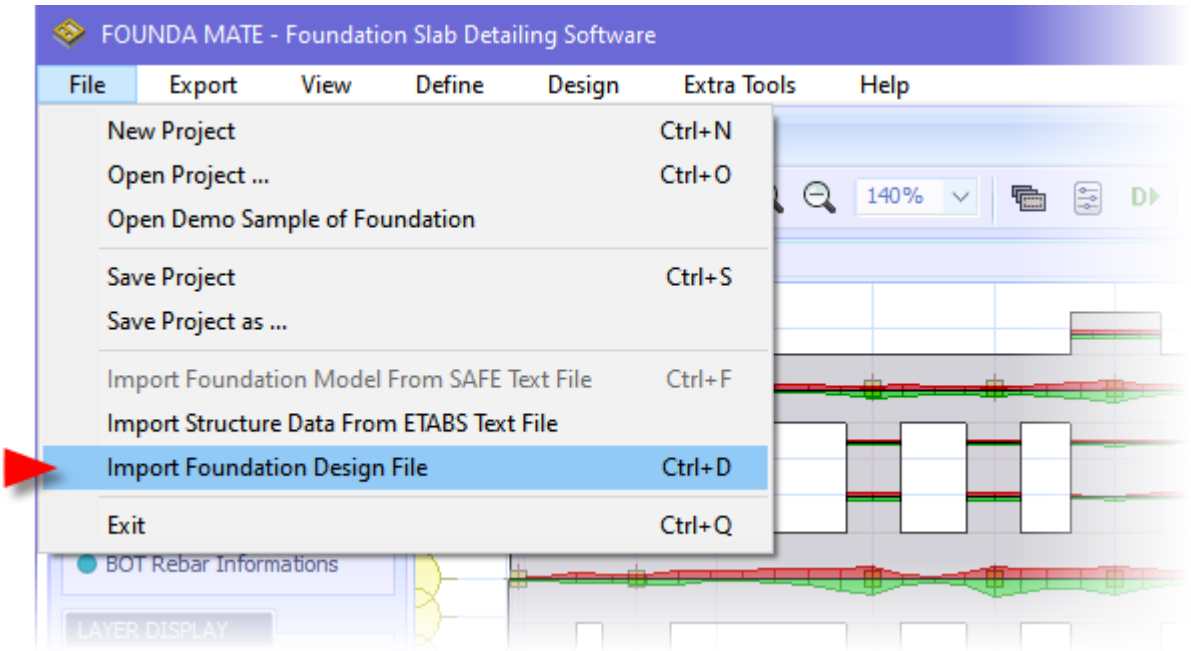
After importing the model into the software, its geometry will be visible in the program's graphical environment. Simultaneously with the display of the model, the slab properties user interface will appear, as shown in the figure below. In this interface, you must select the default Slab from the dropdown list at the top of the window. As shown in the image below, upon selecting the default Slab, the properties defined for this item in SAFE will be displayed in the **Default Slab Properties** section.

Additionally, the minimum required reinforcement percentage will be calculated by the software based on the material specifications and in accordance with the code, and will be displayed in the **Minimum Reinforcement of Slab** section. Furthermore, in the lower part of the window, the minimum continuous rebar details considered based on the minimum reinforcement percentage will also be displayed. Users may either accept the default properties shown in this interface or edit these parameters according to their own preferences.

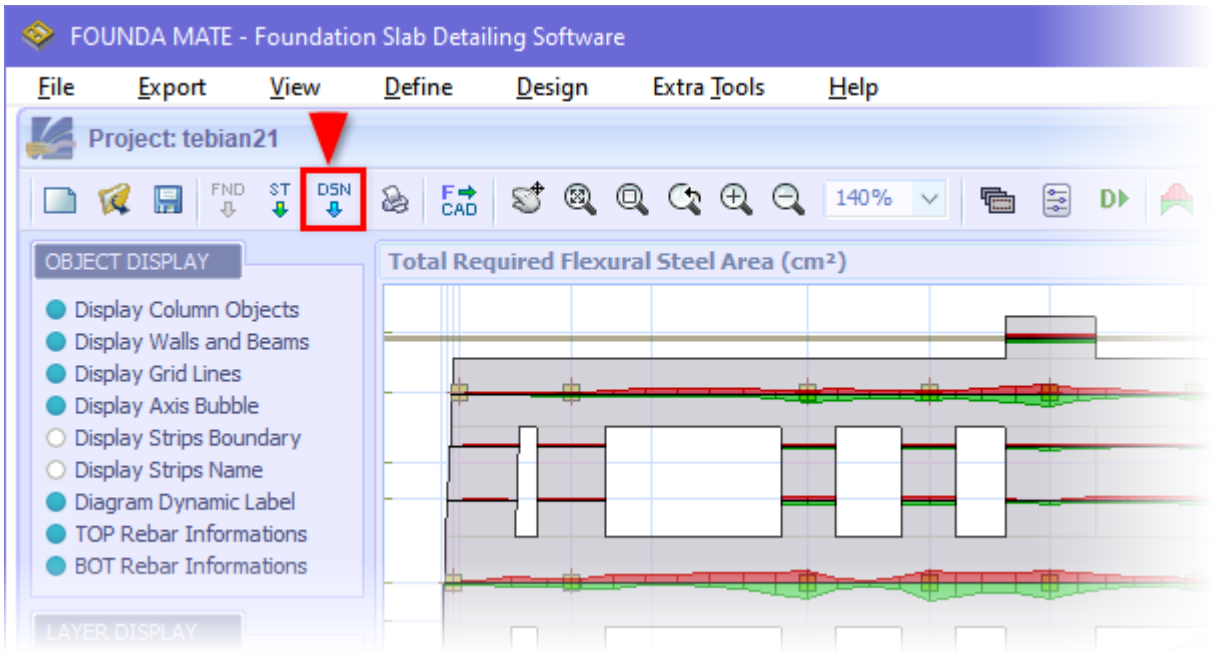


The software provides the capability to import the model design data separately from the model geometry data, if required. However, please note that importing any new design file will delete all previously imported design data and replace it with the new design information. To do this, import the file containing the design data, i.e., the **DESIGN DATA** table (or the **Design Result Table** in earlier versions), into the program as follows:

File Menu > Import Foundation Design File (Ctrl + D)

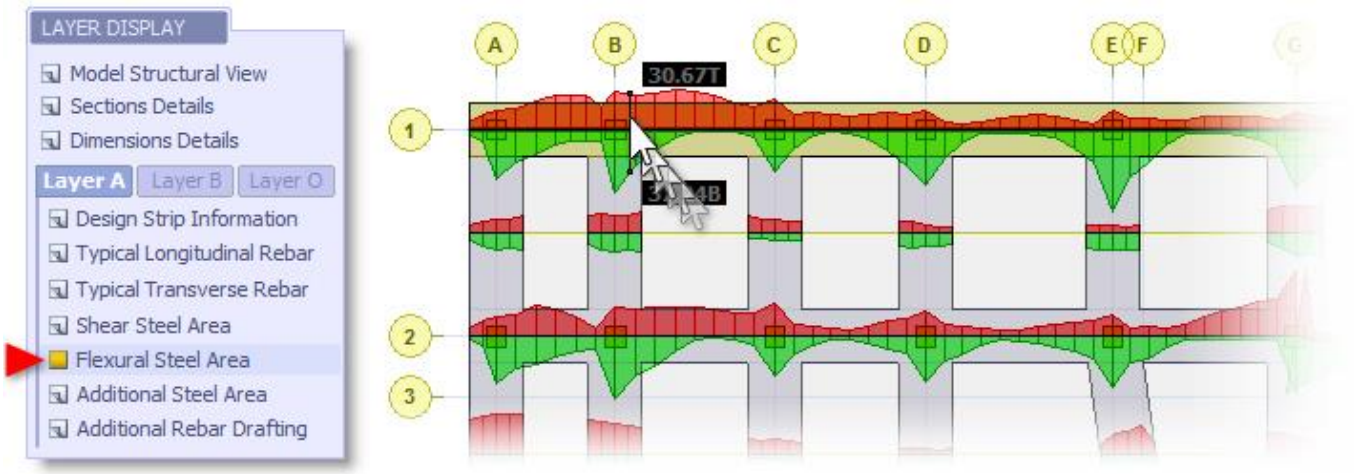


You may also use the icon DSN in the main toolbar at the top of the software to import the strip design results file.



After completing the model import step, the model geometry and its design results are imported into the software, and this information can be viewed in the program's graphical environment.

As shown in the image below, by selecting the **Flexural Steel Area** option from the **Layer Display** section in the software's side panel, the diagrams of the total required steel area are drawn on the design strips. By moving the mouse over the design strips, the exact total flexural steel values for each position can be viewed. To view information for the design strips in other layers, use the buttons Layer A | Layer B | Layer O to select your desired layer.



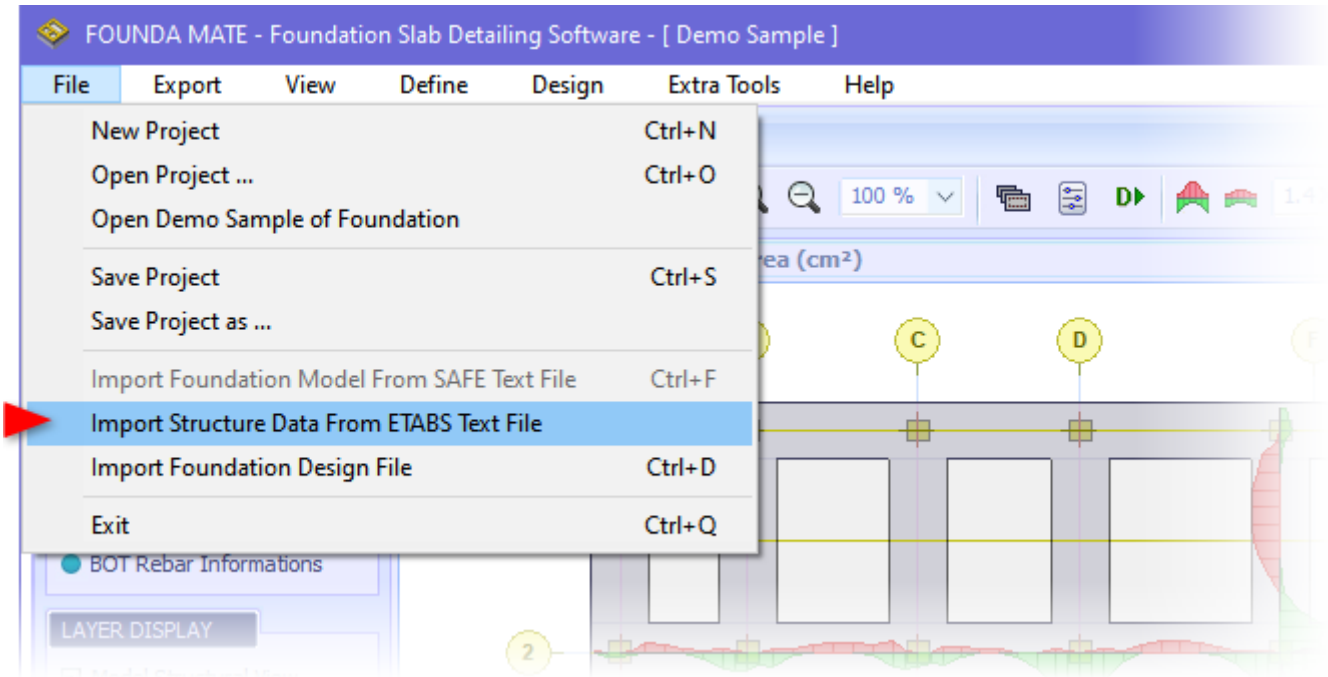
As you move the mouse over the design strips, in addition to the steel values displayed on the diagrams, other highly useful information can also be seen in the status bar at the bottom of the software.

Furthermore, for a more detailed view of the diagrams, you can use the tools identified in the image below to increase, decrease, or change the scale of these diagrams.

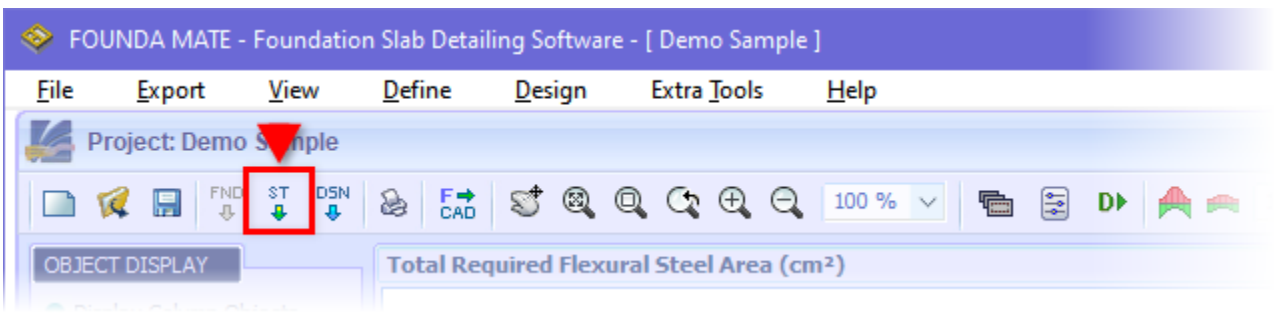
Correcting Column Rotation in Plan

SAFE does not save certain model information, such as column rotation, in the F2K text file. As a result, if the local axis of a column has a non-zero angle, the column orientation in plan will not be drawn correctly and in accordance with the created model. To resolve this issue, the software provides the capability to import, in addition to the SAFE file, an e2k text file from ETABS containing the precise structural model data. In this case, the local axis angle of the columns is extracted from the ETABS file, and the column orientation in plan is corrected accordingly. Therefore, if necessary, generate the structural model text file, i.e., the e2k file, using ETABS and import it into the software as follows:

File Menu > Import Structure Data From ETABS Text File

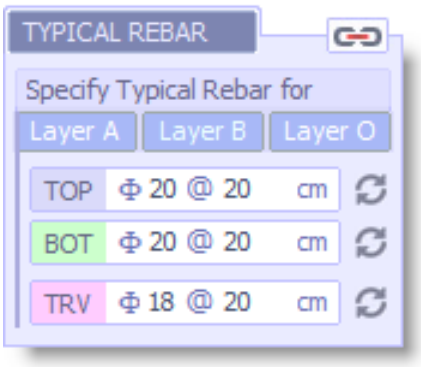


You may also use the icon  in the main toolbar at the top of the software to import the model file from ETABS.



3 Configuring Design Parameters and Detailing the Reinforcement

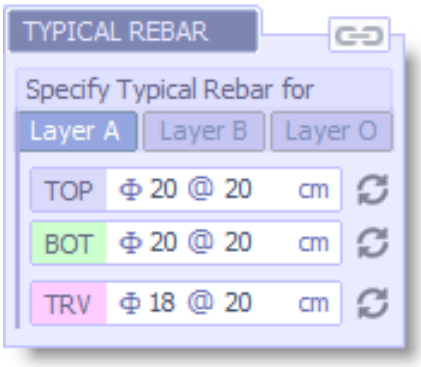
After importing the design results file into the software, you must define the continuous foundation rebars if necessary. The software automatically calculates the continuous rebars based on the code, slab thickness, and material properties, and assigns them to all model strips. However, you can also view and edit these values through the **Typical Rebar** section in the software's side panel, as shown in the image below.



As shown in the image above, this panel allows you to easily view or edit the continuous rebar details for the top and bottom positions, as well as the transverse rebars of the design strips, independently.

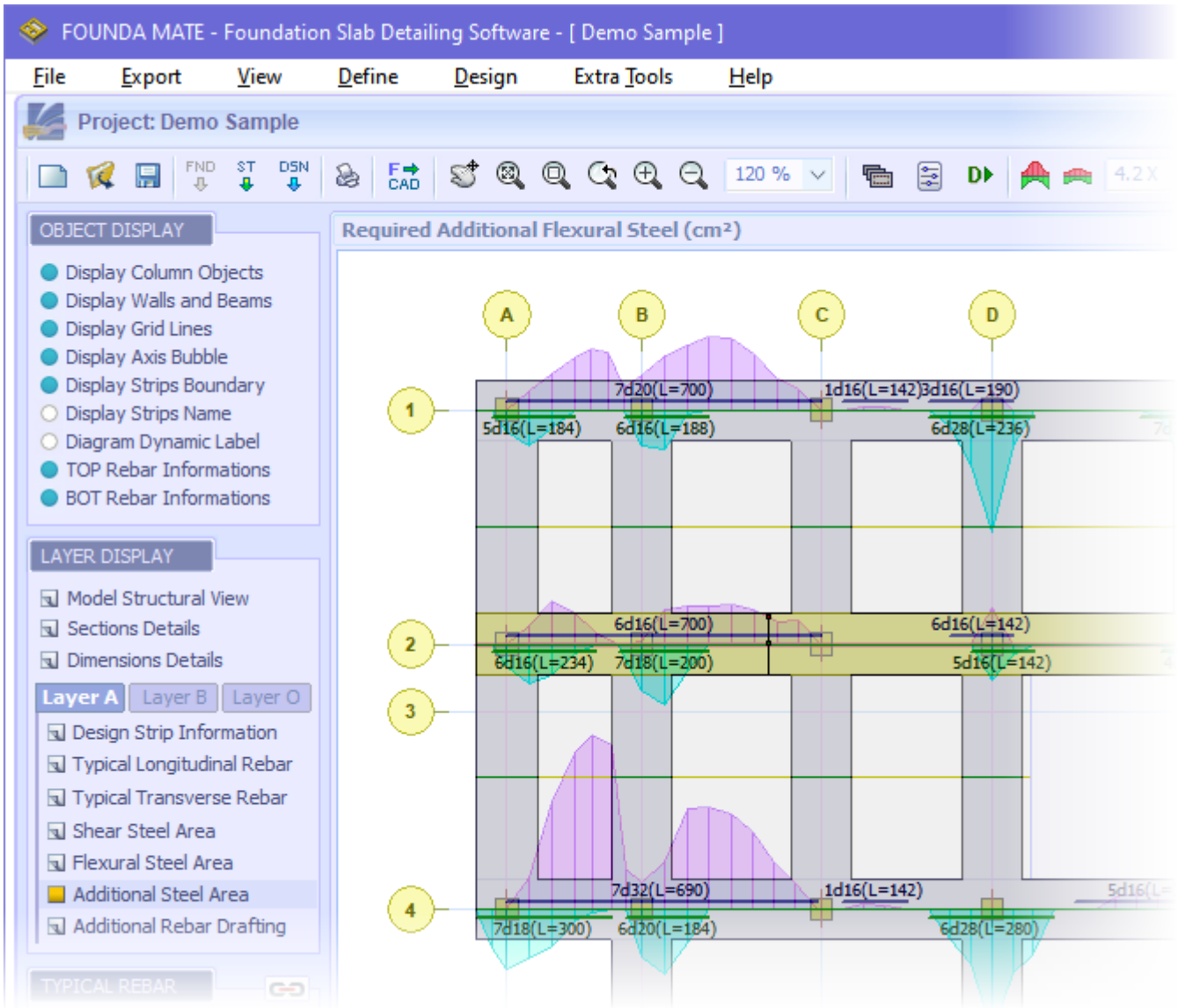
By default, the **Constrain Typical Rebar Detail for All Layers** option is enabled, and the software applies the same typical rebars to layers **A, B** and **Other**. However, the software provides the capability to define different typical rebars for each of the layers **A, B** and **Other**.

If you wish to edit the typical rebars for the design strips in different layers separately, click the icon to disable the **Constrain Typical Rebar Detail for All Layers** mode. In this mode, the fields for all layers become active, and you can click each of the buttons Layer A Layer B Layer O to select the desired layer and view or edit the typical rebar details for the design strips in each layer independently.



After modifying any of the typical rebar details, press any of the icons shown in the image above or press the Enter key. The additional rebars will then be recalculated by the software based on the new continuous rebars and the theoretical cutoff points, and displayed on the model design strips.

As shown in the image below, by selecting the **Additional Steel Area** option from the **Layer Display** section in the software's side panel, the calculated additional reinforcement and the corresponding diagrams of the required additional steel area are drawn on the design strips. The additional reinforcement diagrams are obtained by subtracting the existing continuous reinforcement area from the total steel diagrams, and represent the steel quantity required in addition to the existing continuous rebars.

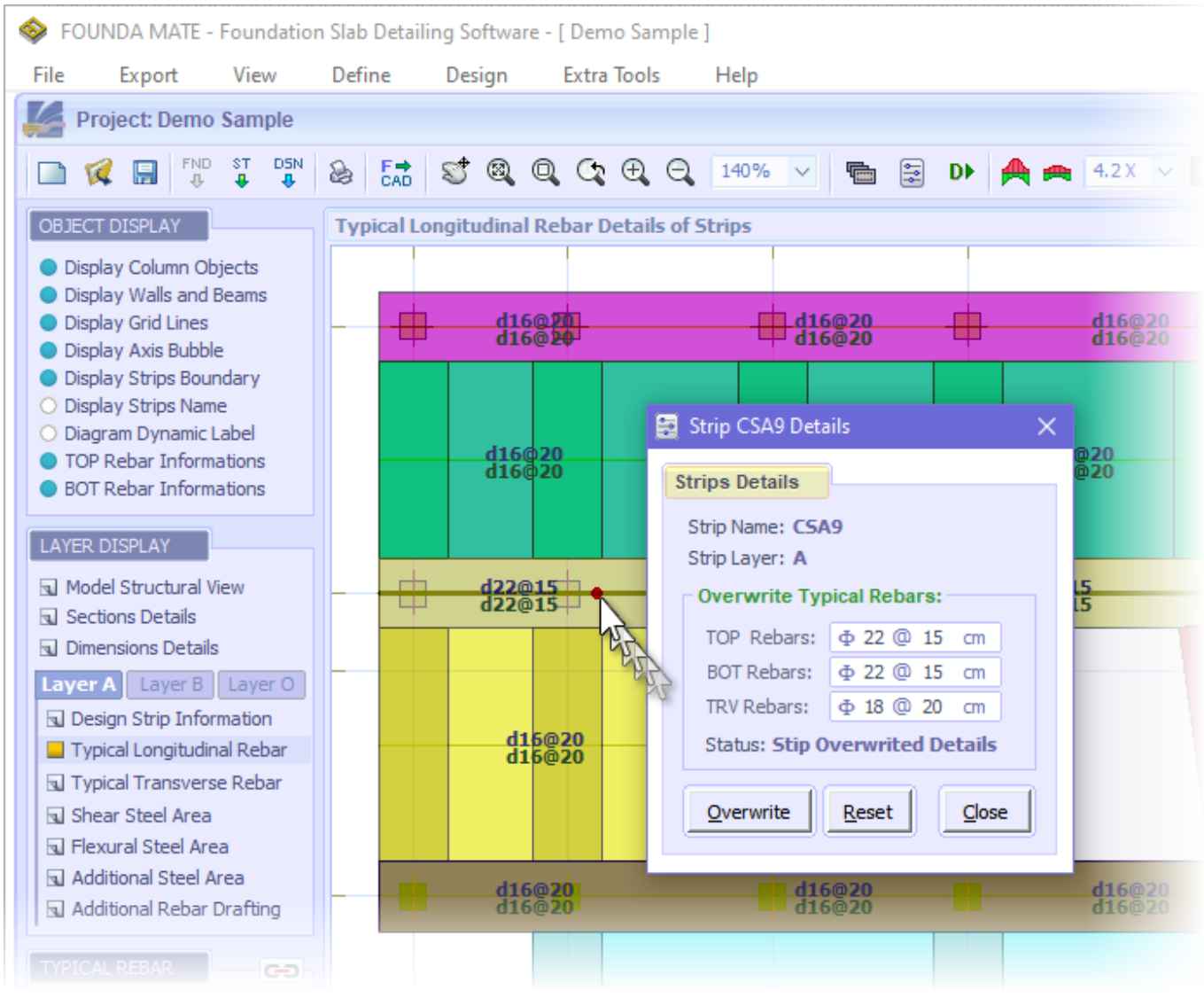


As you move the mouse over the design strips, the exact additional reinforcement values for each position can be viewed. Additionally, highly useful information is displayed in the status bar at the bottom of the software, allowing users to thoroughly examine the details.

The software provides the capability to define separate continuous rebars for each design strip. To do this, select the **Typical Longitudinal Rebar** or **Typical Transverse Rebar** option from the **Layer Display** section in the side panel. Upon selecting either of these layers, the software displays the typical steel details on the design strips, and to better distinguish the design strips, each strip is shown in a different color so that the strip widths are clearly identifiable.

Right-clicking on any design strip opens the new detail assignment window, as shown in the figure below, through which new details can be assigned to the selected design strip. Pressing the **Reset** button restores the assigned typical rebar details to the general details.

Furthermore, if you wish to restore the overwritten details of all design strips to the general details defined for the entire model, you can use the **Reset All Strip's Overwrites** option from the **Design** menu.

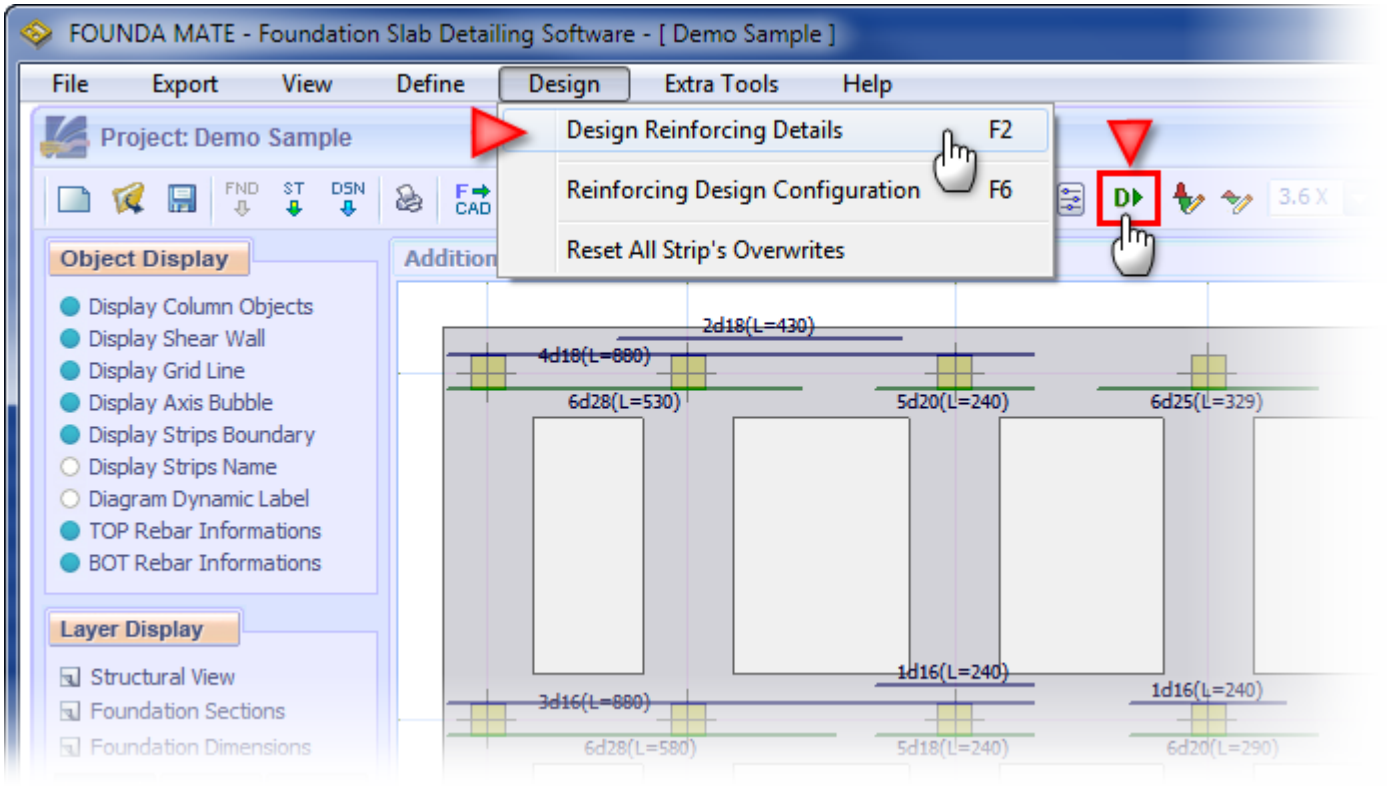


FOUNDA MATE
Foundation Slab Detailing Software

Reinforcement Detailing

After defining the typical rebars, the reinforcement detailing must be designed. To do this, use the icon D▸ in the main toolbar or proceed from the **Design** menu:

Design Menu > Design Reinforcing Details (F2)



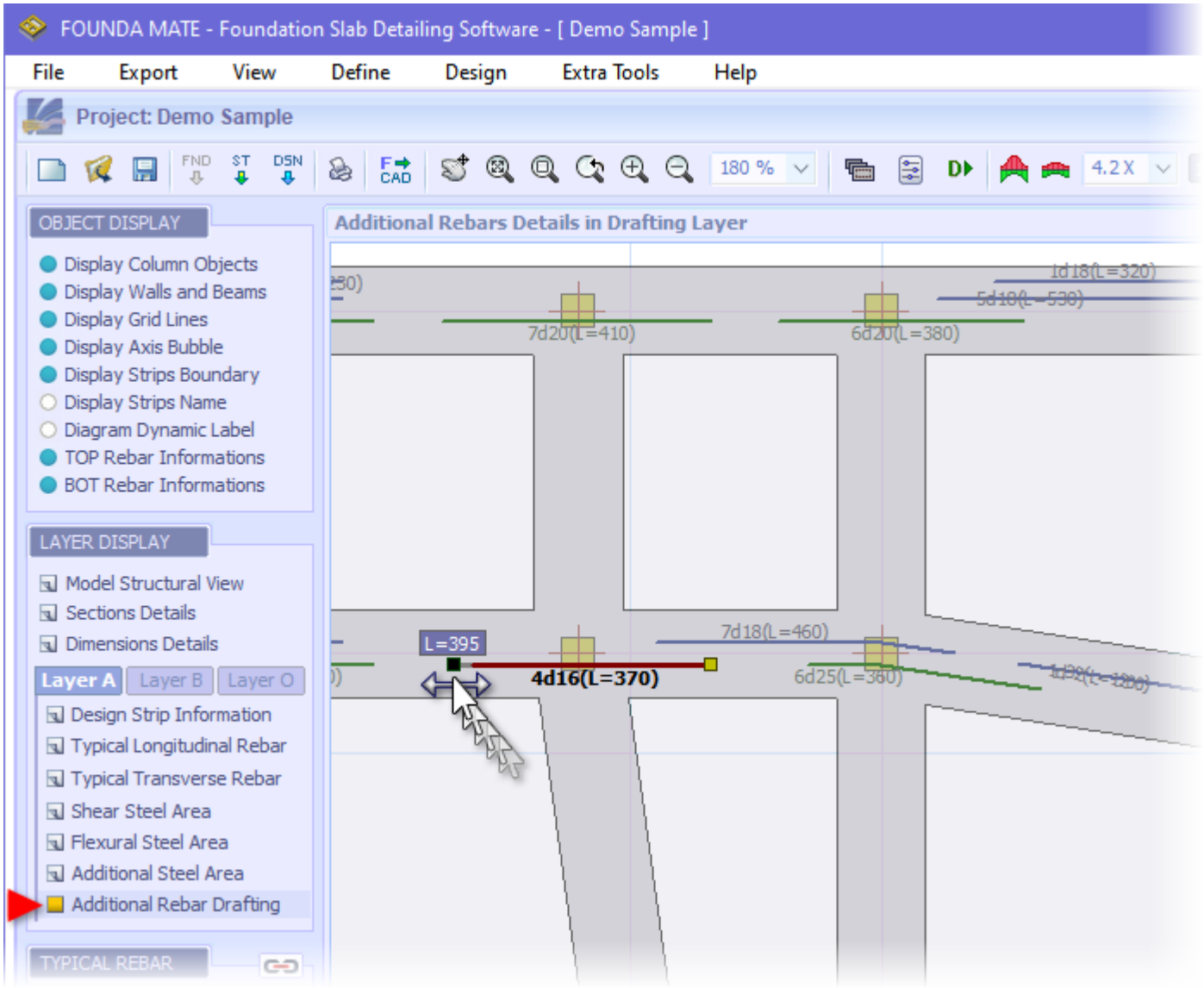
Upon executing the reinforcement detailing command, the details of the additional rebar quantity, diameter, and length are calculated based on the practical cutoff points and the reinforcement design configuration. These details are then displayed on the model in the software's graphical environment. The details are calculated according to the current design configuration and the typical rebars defined at the time of executing the **Design** command, and are transferred to the **Additional Rebar Drafting** layer. Therefore, if the reinforcement design configuration settings or the typical rebars are modified, the **Design** command must be executed again to recalculate the details under the new conditions. To this end, whenever the design parameters are changed, the icon Do Design Again appears at the top of the software to alert the user to the need to re-execute the reinforcement design command. On the other hand, the additional reinforcement details remain available in the **Additional Rebar Drafting** layer at all times and remain unchanged until a new **Design** command is issued, allowing the user to compare results or make the necessary modifications.

Note: If the typical rebars or the design configuration are modified, the Design command must be executed again to repeat the reinforcement design process so that the drafting layer is updated accordingly.

FOUNDA MATE
Foundation Slab Detailing Software

Upon executing the **Design** command, the drafting layer becomes active and the design results are displayed in this layer. As shown in the image below, by selecting the **Additional Rebar Drafting** layer from the **Layer Display** section in the software's side panel, the construction additional rebars are drawn on the design strips based on the practical cutoff points and the design configurations.

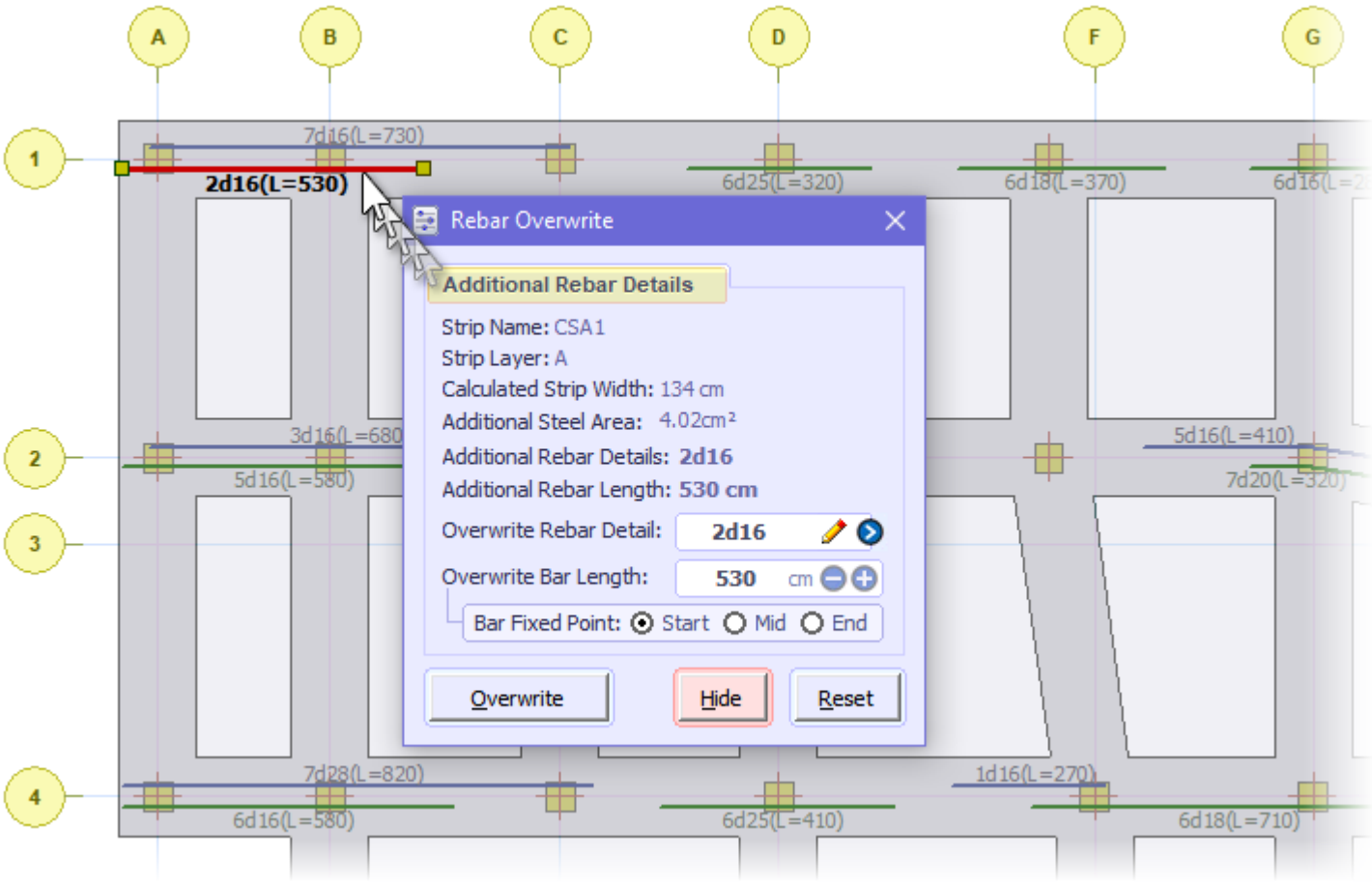
In the **Additional Rebar Drafting** layer, you can easily edit the additional reinforcement details. To do this, click on the desired rebar to display two **End Grip Squares** on the additional rebar. Then, use the mouse to grab the desired **Grip Square** and Drag it to the desired position. While dragging the **End Grip**, the new rebar length is displayed on the selected additional rebar for greater user convenience.



Furthermore, in the **Additional Rebar Drafting** layer, right-clicking on any rebar opens the additional reinforcement details editing interface, as shown in the image below. Through this interface, you can edit the number and diameter of the additional rebars either by selecting from the software's suggested list, which appears by clicking the icon next to the additional rebar detail text, or by typing the desired details directly in the **Overwrite Rebar Detail** field.

You can also edit the length of the additional rebars in this window. To do so, as shown in the image, first type the new additional rebar length in the **Overwrite Bar Length** field, or increase/decrease it using the buttons. Then, specify a fixed point for the length change in the **Bar Fixed Point** section. For example, selecting the **Mid** point extends the additional rebar length from both ends, while selecting **Start** as the fixed point extends the rebar length from its end only.

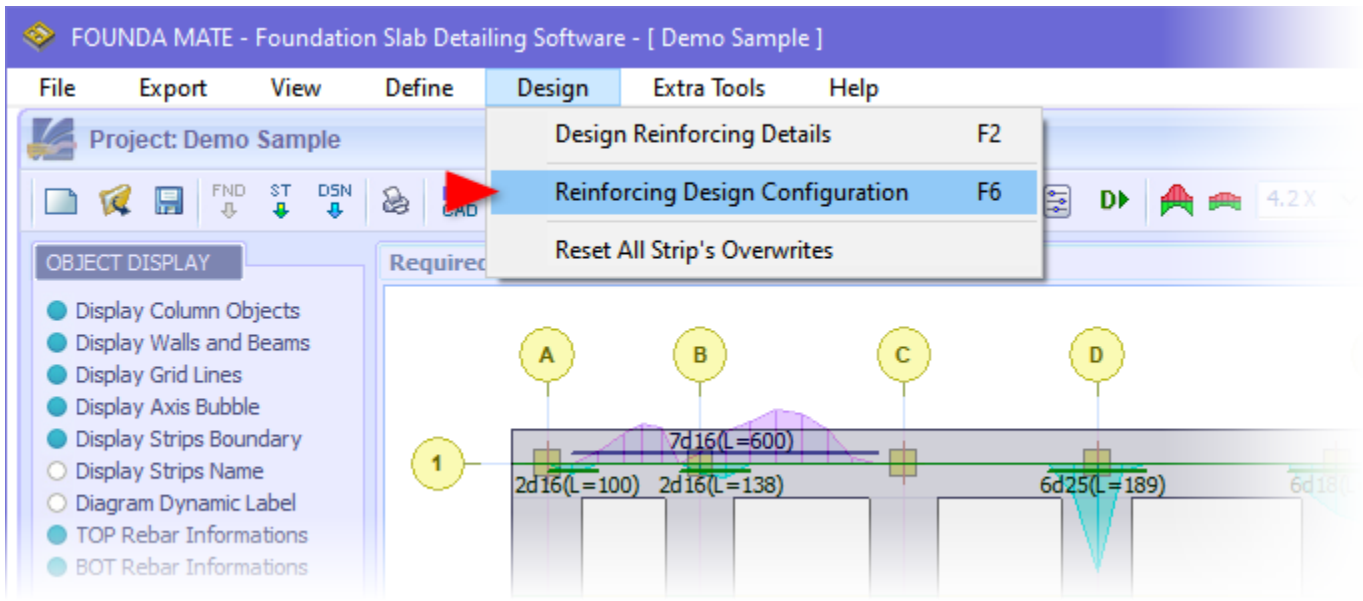
After making the necessary changes, press the Overwrite button to apply the user modifications. Note that these modifications will govern the drawing generation. Pressing the Reset button restores the user-applied changes for this rebar to the software-calculated details. Additionally, the Hide button allows you to hide the selected rebar so that it is not drawn in the drawings; this action can be reversed using the same button.



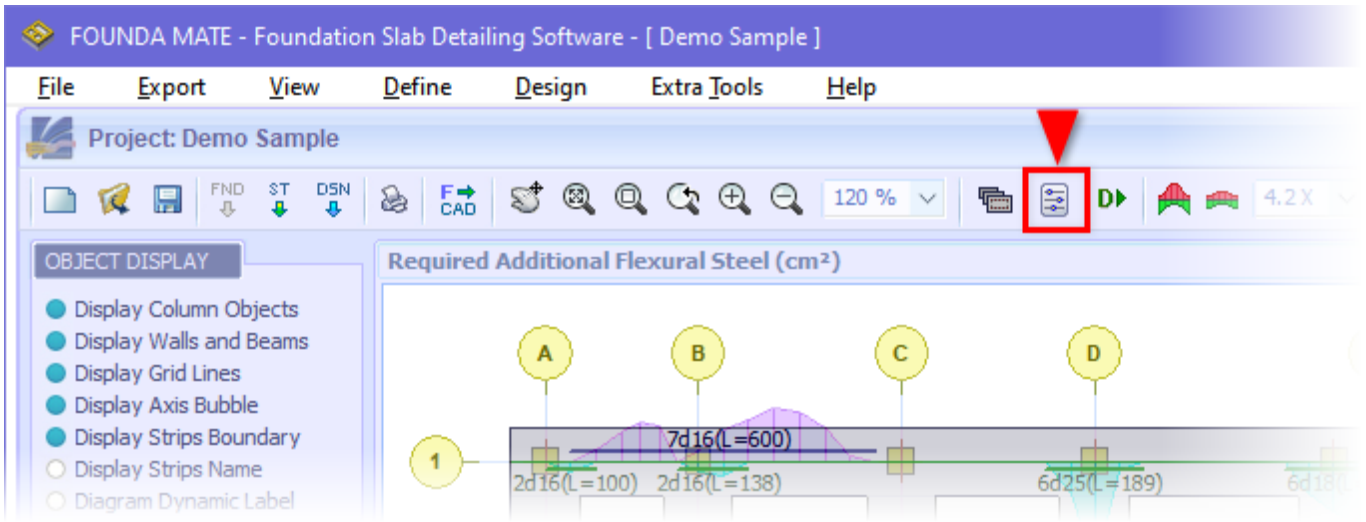
Configuring Reinforcement Detailing Design Parameters

To configure the design parameters, proceed as follows from the **Design** menu:

Design Menu > Reinforcing Design Configuration (F6)

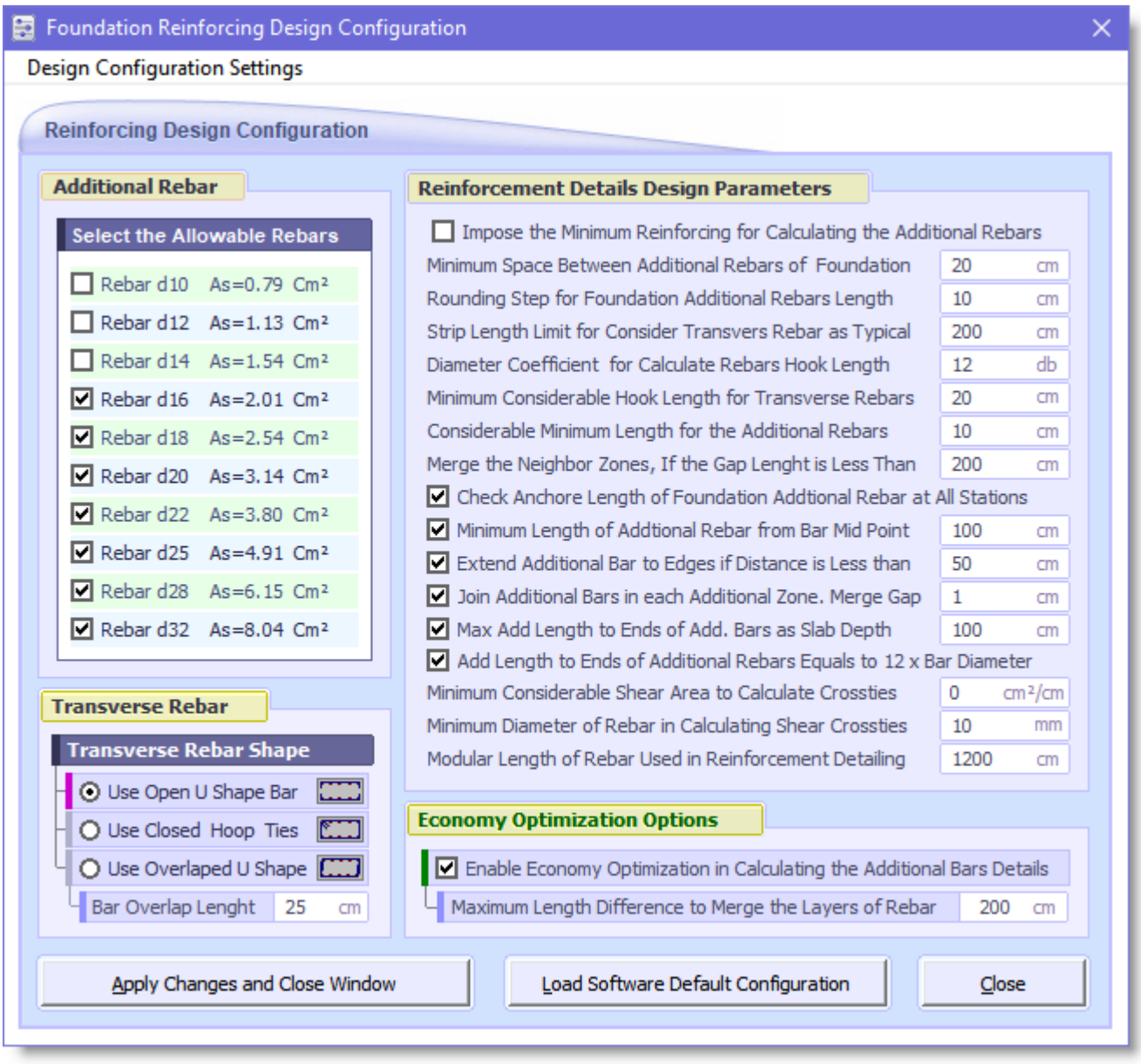


You may also use the icon in the main toolbar at the top of the software, or use the shortcut key F6, as shown in the image below.

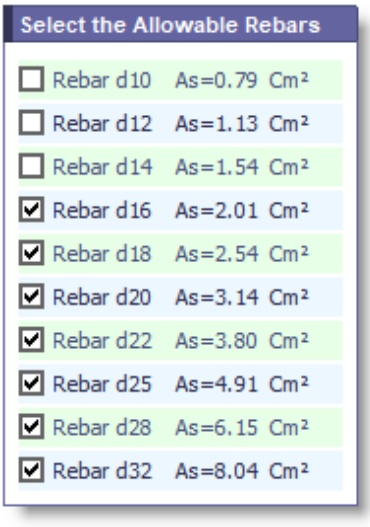


Executing this command opens the reinforcement detailing design parameters configuration interface, as shown below. Through this interface, you can configure the general parameters that will be used by the software in the additional reinforcement design. The various parameters in this interface are described in the following.

FOUNDA MATE
Foundation Slab Detailing Software



Permissible Additional Rebar Sizes:

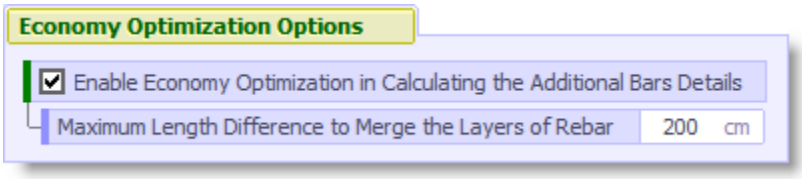


Select Additional Rebar Size

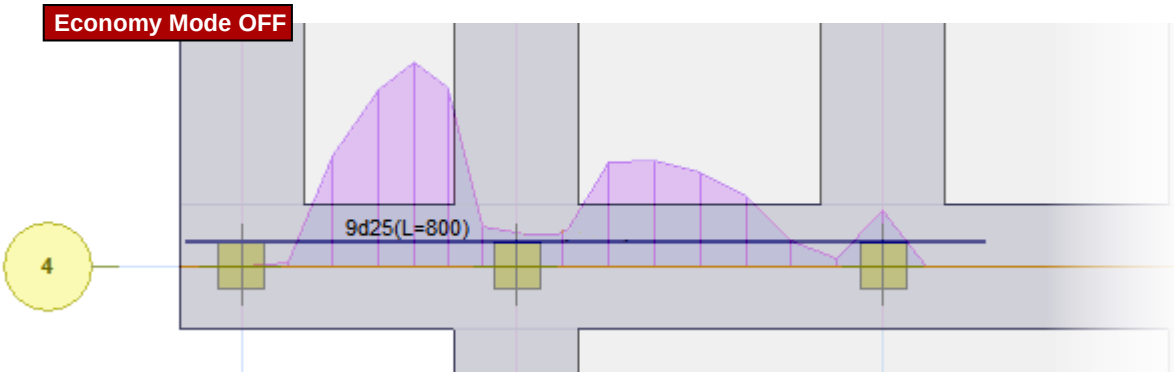
As shown in the adjacent image, in this section you can specify, from among the available rebars, the permissible sizes for the software to use in the additional reinforcement design process. In other words, the software will only use the sizes selected in this section for the additional rebars.

Economy Optimization Parameters for Additional Reinforcement Design

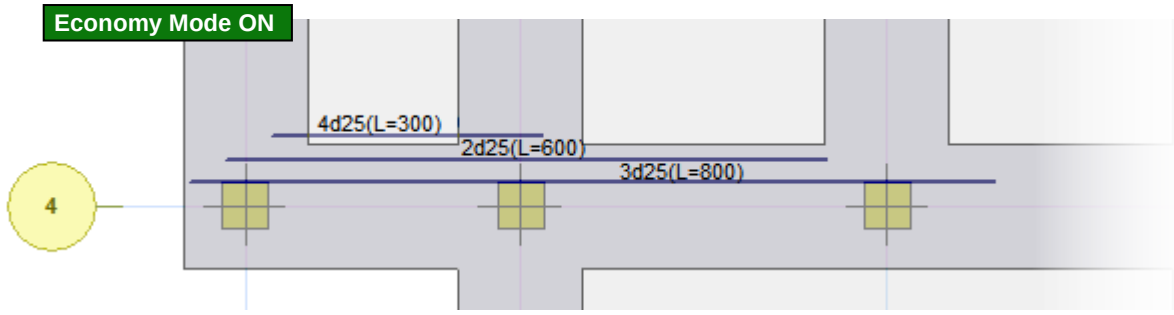
One of the prominent features of the software is the design of additional reinforcement using an intelligent economic optimization algorithm. In many cases, this capability can reduce additional steel consumption by up to 30%. As shown in the image below, you can configure the parameters related to this mode in the **Economy Optimization Option** section.



When this option is enabled, the additional reinforcement design is not based solely on the maximum required steel in each region. Instead, the software employs an intelligent economic optimization algorithm to perform the design based on the actual shape of the steel diagram. For example, the additional reinforcement configuration in a foundation region based on theoretical cutoff points is shown in the figure below. As can be seen, if the design were based solely on the maximum diagram steel, **9 Ø25** rebars with a length of **800 cm** would be required, whereas in many parts of the diagram, less steel is needed.



However, when the **Economy Optimization** option is Enabled, the design is performed based on the shape of the diagram. As shown in the figure below, instead of using **7200 cm** of additional reinforcement, a total of **4800 cm** of additional reinforcement is used, representing a **33% Reduction** in steel consumption.



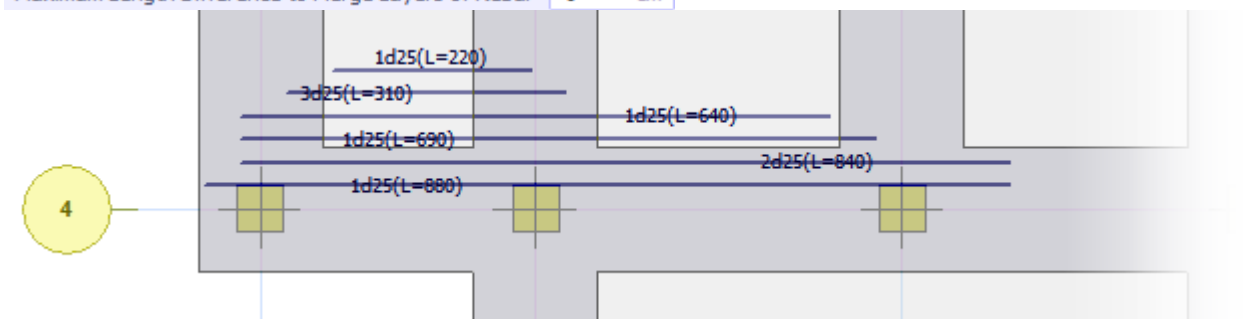
Economy Mode OFF **Rebar Length = 9*800 = 7200 cm**
Economy Mode ON **Rebar Length = 3*800 + 2*600 + 4*300 = 4800 cm**

Maximum Length Difference to Merge Layers of Rebar 200 cm

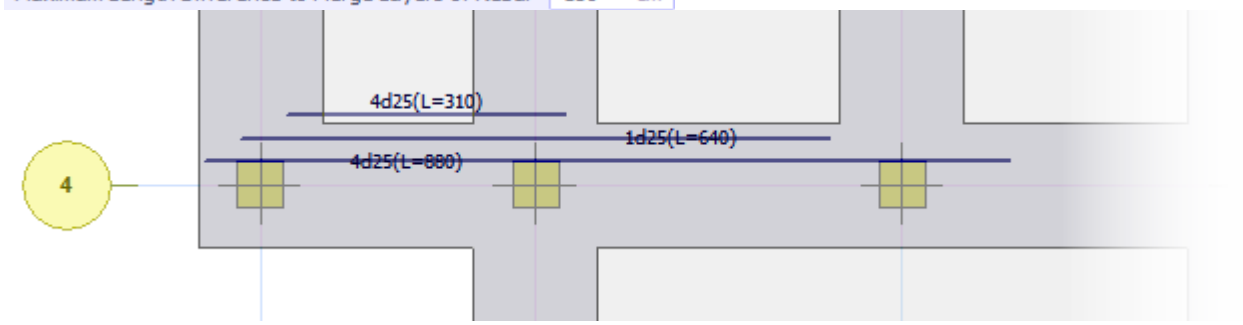
This parameter, which represents the maximum permissible length difference between additional rebars, is one of the most influential factors in the economic design of additional reinforcement. Using this parameter, you specify that rebars with a length difference less than the set value be grouped together to avoid an excessive variety of additional reinforcement lengths in the drawings.

As shown in the images below, if this parameter is set to a small value, such as zero, the software calculates numerous additional rebars with varying lengths, which is not very suitable for construction. However, as the value of this parameter is increased, the variety of additional rebar lengths is reduced, resulting in more practical detailing. Correspondingly, the percentage of economic savings decreases, such that if a large value is set for this parameter, the effect of the intelligent economic design algorithm is completely eliminated, and the calculations are performed solely based on the maximum steel diagram. Therefore, the design engineer must adjust this parameter to achieve the best possible result.

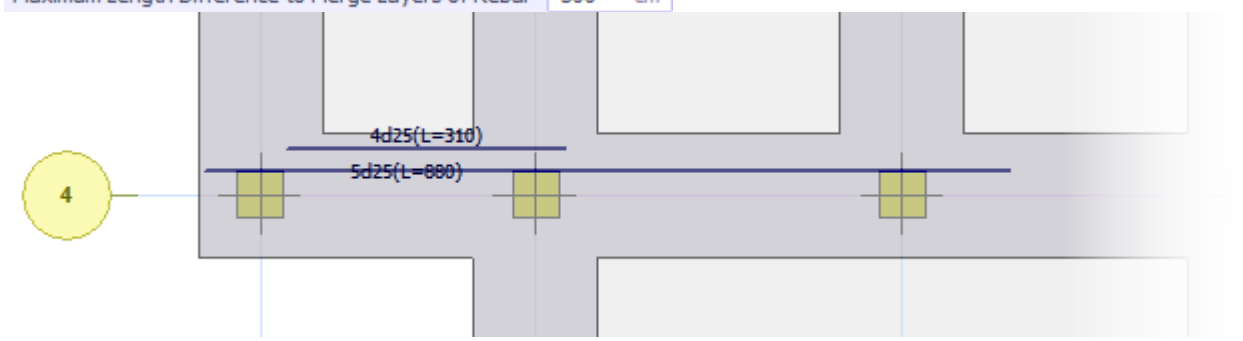
Maximum Length Difference to Merge Layers of Rebar 0 cm



Maximum Length Difference to Merge Layers of Rebar 150 cm



Maximum Length Difference to Merge Layers of Rebar 300 cm



FOUNDA MATE
Foundation Slab Detailing Software

Reinforcement Detailing Design Parameters

☐ Impose Minimum Reinforcing for Calculating the Additional Rebars

When this option is enabled, **FOUNDA MATE**, like **SAFE**, also controls the minimum reinforcement quantity and compensates for any deficiency by means of additional rebars in locations where the steel quantity falls below the minimum required value.

Minimum Space Between Additional Rebars of Foundation 20 cm

This parameter allows you to specify the minimum spacing between additional rebars. Since the software considers a default spacing of 20 cm between typical rebars, this parameter is also set to 20 cm by default, ensuring that the spacing between additional and typical rebars does not fall below 10 cm.

Rounding Step for Foundation Additional Rebars Length 10 cm

This parameter allows you to define the rounding step for additional rebar lengths. In other words, the additional rebar lengths will always be a multiple of the value defined for this parameter.

Strip Length Limit for Consider Transvers Rebar as Typical 200 cm

This parameter allows you to set a threshold for the foundation segment length, based on which the software decides whether to use the transverse rebar details or the longitudinal rebar details for the typical steel. In other words, if the foundation segment length along its defined design strip axis is less than this parameter, the transverse rebar details are used as the typical steel in the calculations and drawing generation. If the length exceeds this parameter, the longitudinal rebar details are used as the typical steel.

Diameter Coefficient for Calculate Rebars Hook Length 12 db

This parameter allows you to specify the rebar diameter coefficient for calculating the length of the 90-degree end hooks. In other words, the end hook length is obtained by multiplying this parameter by the rebar diameter.

Minimum Considerable Hook Length for Transverse Rebars 20

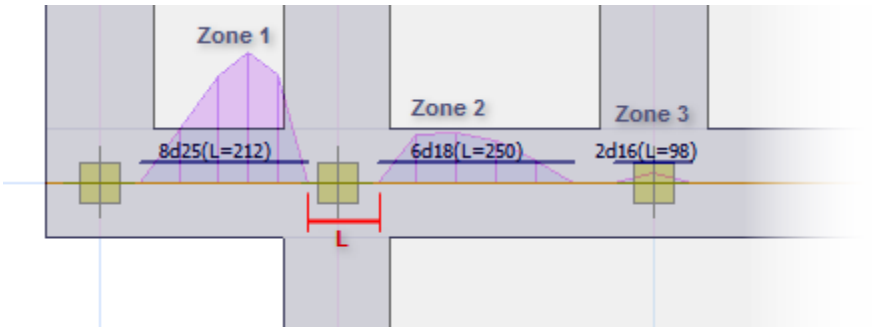
This parameter allows you to specify the minimum desired length for the end hooks of transverse rebars. In other words, the end hook length is initially obtained by multiplying the previous parameter by the rebar diameter. However, if the resulting length is less than the value defined in this parameter, the value defined here governs the 90-degree end hook length. In other words, the end hook length will never be less than the value of this parameter.

Considerable Minimum Length for Additional Rebars 10 cm

This parameter allows you to specify the minimum length for considering additional rebars. In other words, if the calculated additional rebar length based on the theoretical cutoff is less than this value, the rebar will be disregarded, and thus rebars shorter than this parameter value will be eliminated.

Merge the Neighbor Zones, If the Gap Lenght is Less Than 200 cm

This parameter allows you to specify that the software should consider rebars in adjacent regions as continuous if the distance between the two regions is less than the value defined in this parameter. The distance between two adjacent regions is shown as (L) in the figure below. It is recommended to set this parameter to at least twice the value defined as the foundation depth for determining the practical cutoff point in this same interface.



☑ Check Anchore Length of Foundation Addtional Rebar for All Stations

This parameter allows you to specify that the software should check the required development length for the additional rebars at all stations. In other words, when this parameter is enabled, the software automatically checks whether the required development length is provided from each peak. If the development length is not satisfied, the software extends the rebar length to ensure the development length is met.

☑ Minimum Length of Addtional Rebar from Bar Mid Point 100 cm

This parameter allows you to specify the minimum length from the midpoint of the rebar to both ends of the additional rebars. When enabled, a minimum length is always maintained for short additional rebars, resulting in better-quality drawings.

☑ Extend Additional Rebar to Edge if Distance is Less than 50 cm

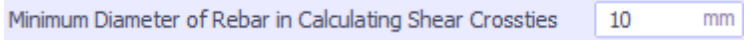
This section allows you to specify whether additional rebars whose end distance to the foundation edge is less than the value defined for this parameter should be extended to the foundation edge. You can also specify the value of this parameter.

This option allows you to specify whether the software should join additional rebars together and apply identical details to all of them if the distance between them is less than the value defined for this parameter.

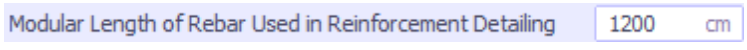
This parameter allows you to specify the maximum length that the software automatically adds to the additional rebar length from the theoretical cutoff point to determine the practical cutoff point. Based on the code, the value of this parameter should be taken as the foundation depth. It is recommended to set the adjacent region joining length, previously described, to at least twice the value defined for this parameter. Note that if the value of this parameter is greater than the foundation depth, the entered value is added to the additional rebar length from the theoretical cutoff point; otherwise, the foundation depth is added to the additional rebar length from the theoretical cutoff point.

This parameter allows you to specify whether the software should automatically check that at least 12 times the rebar diameter is added from the theoretical cutoff point of the additional rebars to determine the practical cutoff point.

This parameter allows you to define a threshold or minimum shear steel value for calculating shear crossties. In other words, if the shear steel quantity at the critical shear section is less than this parameter value, the software will omit the shear crosstie calculation. This allows shear crossties to be omitted in locations where only a small amount of shear steel is required.



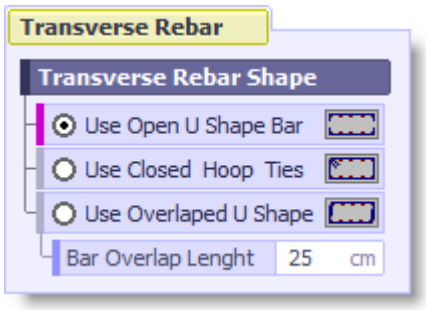
This parameter allows you to specify the minimum diameter of the foundation shear crossties. In other words, during the transverse shear steel detailing process of the foundation beam, the crosstie diameter will always be equal to or greater than the value of this parameter.



This parameter allows you to specify the standard modular length of the reinforcing bars used.

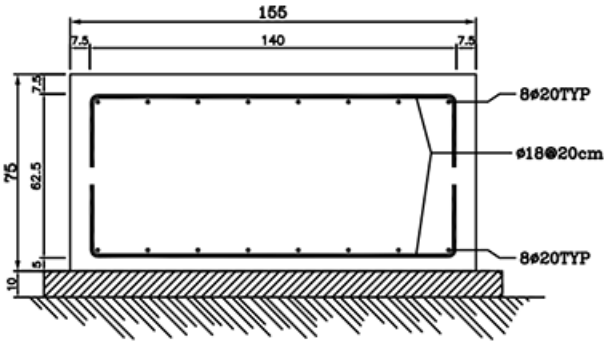
Transverse Rebar Shape Settings

Using the parameters in this section, you can specify the shape of the transverse rebars. As shown in the adjacent image, through the settings available in this section, you can define the transverse rebar shape as two U-bars at the top and bottom of the slab, a single closed tie, or two U-bars at the top and bottom with a specified overlap length. These configurations are described below:



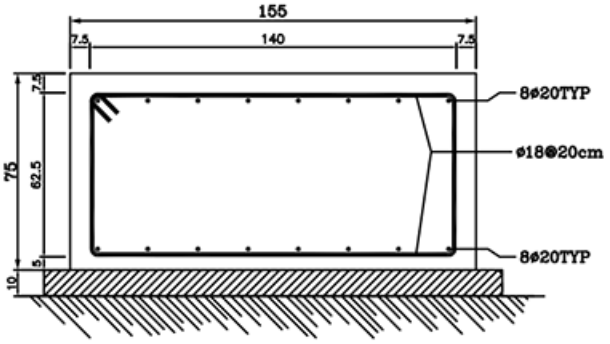
1. Use Open U Shape Bar

In this configuration, the transverse rebars are drawn as one U-bar at the top and one U-bar at the bottom. In this case, the area of these legs is not included in the beam shear calculations.



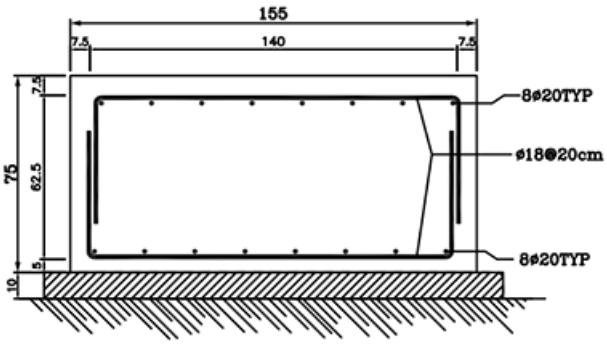
2. Use Closed Hoop Ties

In this configuration, the transverse rebars are drawn as a single closed tie. In this case, the area of both legs is included in the beam shear steel calculations.

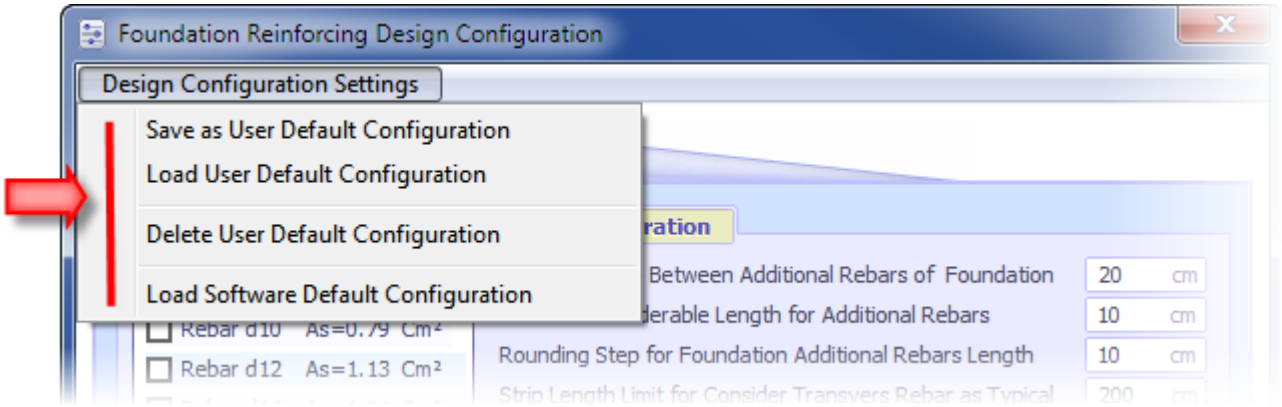


3. Use Overlaped U Shape

In this configuration, the transverse rebars are drawn as one U-bar at the top and one U-bar at the bottom, with overlapping legs. Since the legs overlap, the area of both legs is included in the beam shear calculations. The overlap length of the legs can be adjusted using the **Overlap Length** parameter. If you do not wish for the area of these legs to be included in the calculations, use the **Open U Shape Bar** option.



As shown in the image below, using the **Design Configuration Settings** menu and its available options, you can save the settings and configurations made through this interface for use in other projects. The items available in this menu are described below:



Design Configuration Settings > 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 settings as its own default configuration.

Design Configuration Settings > Load User Default Configuration

This option allows you to reload your previously saved settings.

Design Configuration Settings > Delete User Default Configuration

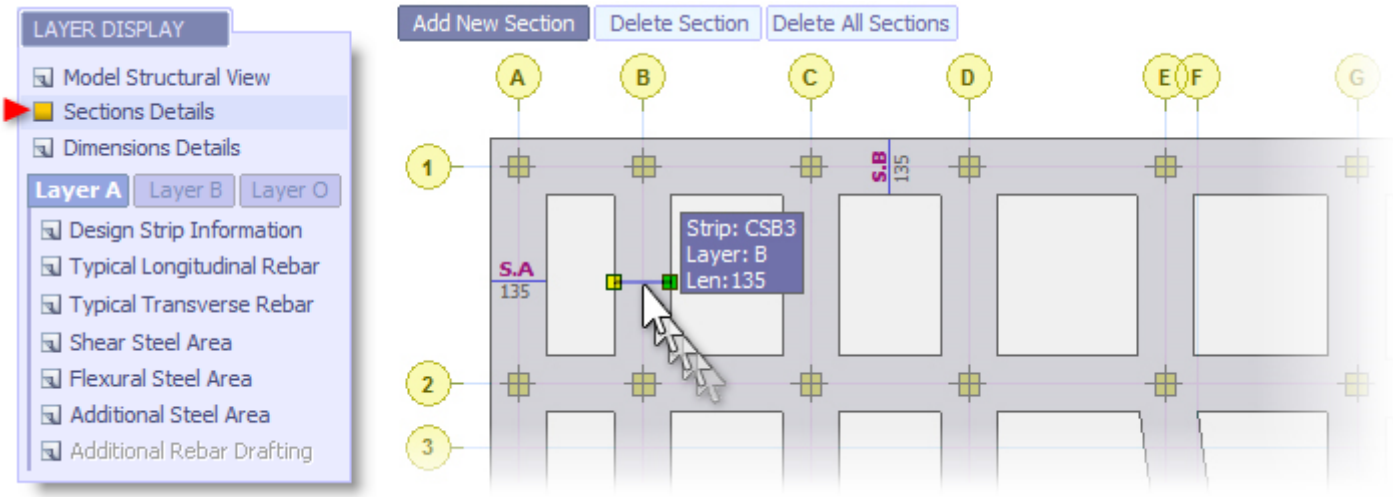
This option allows you to delete the user default settings. In this case, the software will launch with its own factory default settings in subsequent sessions.

Design Configuration Settings > Load Software Default Configuration

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

Defining and Managing Cross Sections

To create or manage cross sections, use the **Sections Details** option from the **Layer Display** section in the software's side panel, as shown in the image below. You may also use the **Define** menu and the **Foundation Section** option. In this mode, previously defined cross sections are displayed, and tools for defining new sections or managing them are also made available. These are described in the following.



When the **Sections Details** layer is activated, the Add New Section | Delete Section | Delete All Sections tools for adding or managing sections appear at the top of the software's graphical environment, as described below:

Add New Section

By activating this tool, you can define a new cross section at any desired location. To do this, move the mouse cursor to the desired position. As you move the mouse, the software automatically detects the foundation edges, and as shown in the image, the section width and certain other details are displayed on the foundation strips as the mouse moves. Once the cursor is positioned at the desired location, each click creates a new cross section.

Note that the section indicator line is always perpendicular to the yellow square shown in the image. You can determine perpendicularity to a specific edge by moving the cursor close to that edge, which changes the color of the end square of the section line to yellow. This point is only applicable in areas of the foundation where the edges are not parallel.

Delete Section

By activating this tool, you can delete previously defined sections. To do this, move the cursor close to the indicator line of a previously defined section until it turns red, then click on it to delete the selected section.

Delete All Sections

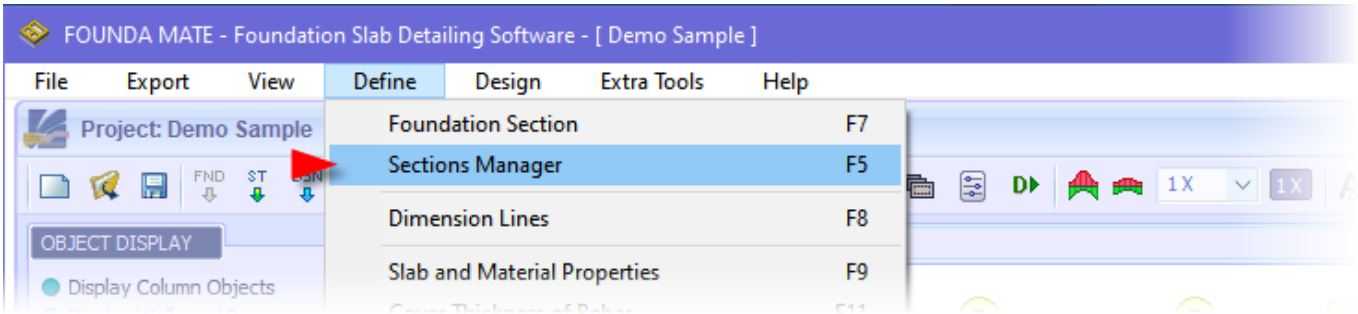
Using this tool, you can delete all defined cross sections.

FOUNDA MATE
Foundation Slab Detailing Software

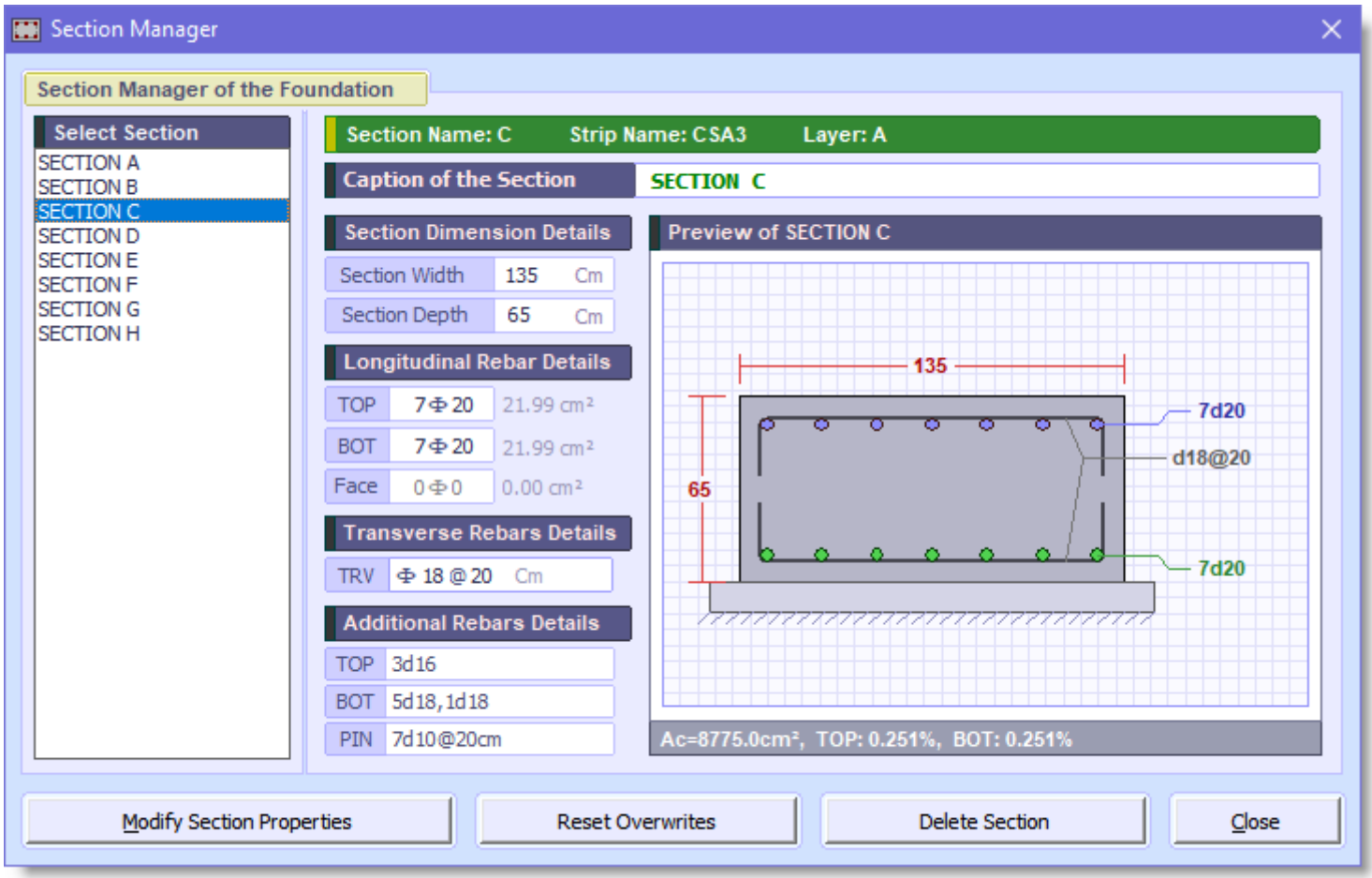
Editing Cross Section Properties

To view or edit cross section properties, you may use the  icon in the main toolbar at the top of the software, the shortcut key F5, or proceed as follows from the **Define** menu:

Define Menu > Section Manager

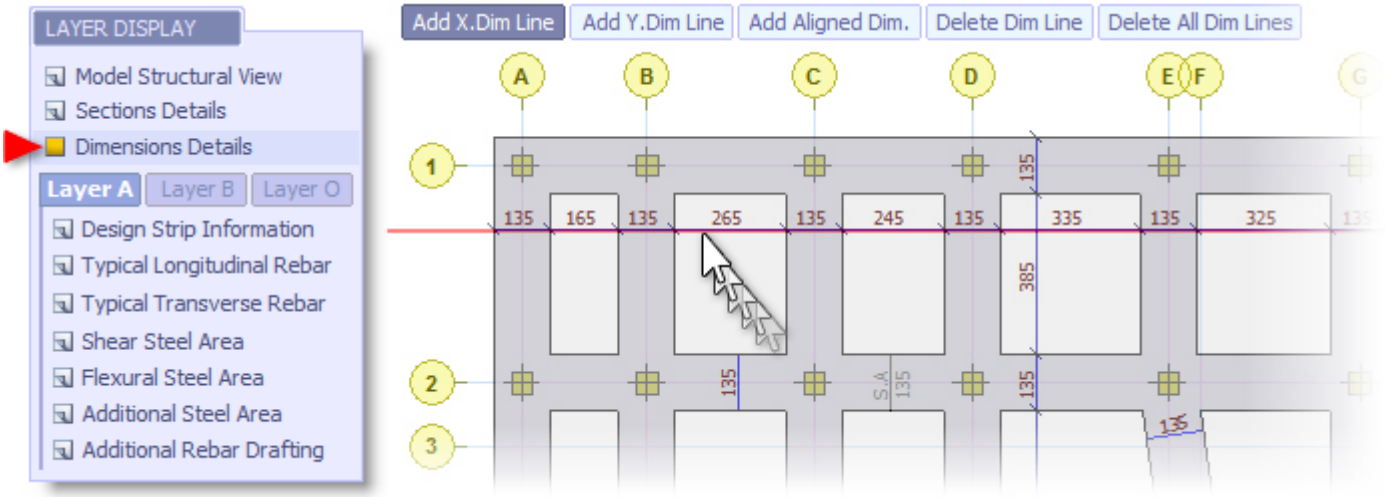


In this mode, the section properties user interface appears, as shown in the image below. Through this interface, the user can view or edit all section properties, as well as the section name, if necessary. Note that if the name of any section is changed, the name defined by the user in the **Section Caption** field will be inserted in the structural drawings. Please note that modifying the section details does not affect the foundation design details and only impacts the display of the cross sections.



Defining and Managing Dimension Lines

To create or manage dimension lines, use the **Dimensions Details** layer from the **Layer Display** section in the software's side panel, as shown in the image below. In this mode, previously defined dimension lines are displayed, and a set of control tools for adding or managing dimension lines also becomes available.



When this layer is activated, the Add X.Dim Line | Add Y.Dim Line | Add Aligned Dim. | Delete Dim Line | Delete All Dim Lines buttons for adding or managing dimensions appear at the top of the software's graphical environment, as described below:

Add Y.Dim Line | Add X.Dim Line

By activating these tools, you can define new dimension lines in the X or Y direction at any desired location. To do this, move the mouse cursor to the desired position. A horizontal or vertical line will follow the mouse cursor. When this line is positioned at your desired location, click the mouse to set the dimension line position. The software automatically detects the foundation edges, and as shown in the image above, the dimension lines are drawn on the model.

Add Aligned Dim.

Using this tool, you can draw individual dimension lines on diagonal strips by moving the mouse over the strips.

Delete Dim Line

By activating this tool, you can delete previously defined dimension lines. To do this, move the mouse cursor close to one of the previously defined dimension lines until it turns red, then click on it to delete the selected dimension line.

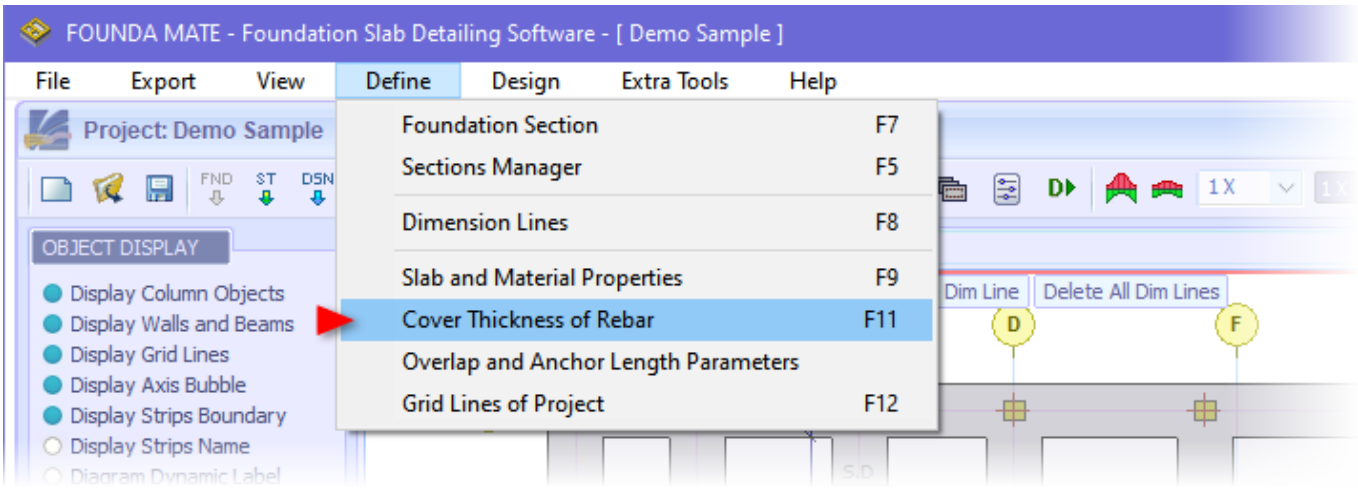
Delete All Dim Lines

Selecting this button deletes all previously defined dimension lines.

Defining Concrete Cover of Rebars

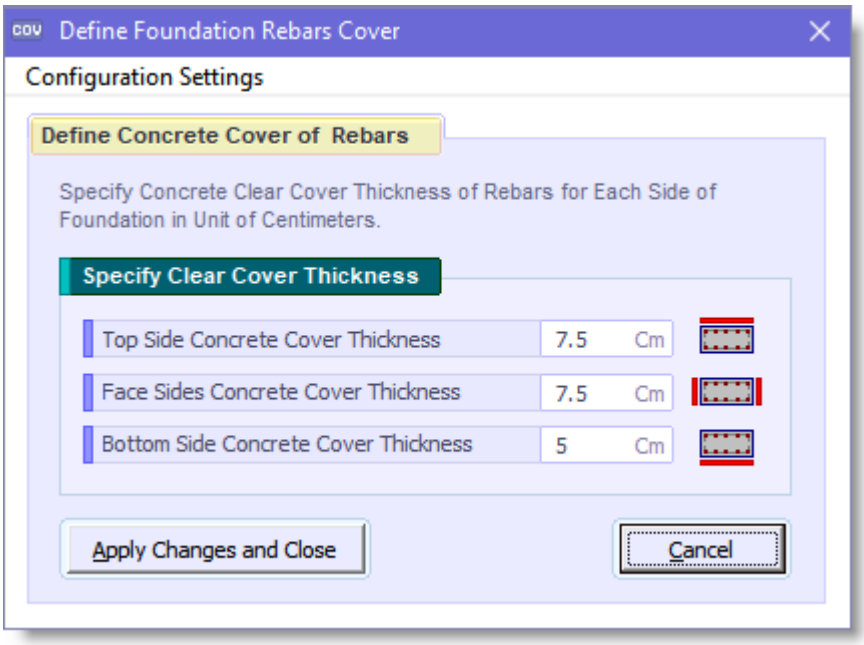
To define the rebar cover for the model, proceed as follows from the **Define** menu, as shown in the image. You may also use the shortcut key F5 on the keyboard:

Define Menu > Cover Thickness of Rebar



In this mode, the concrete cover thickness interface appears, as shown in the image below. Also through this interface, you can define the concrete cover thickness of the rebars separately for different positions.

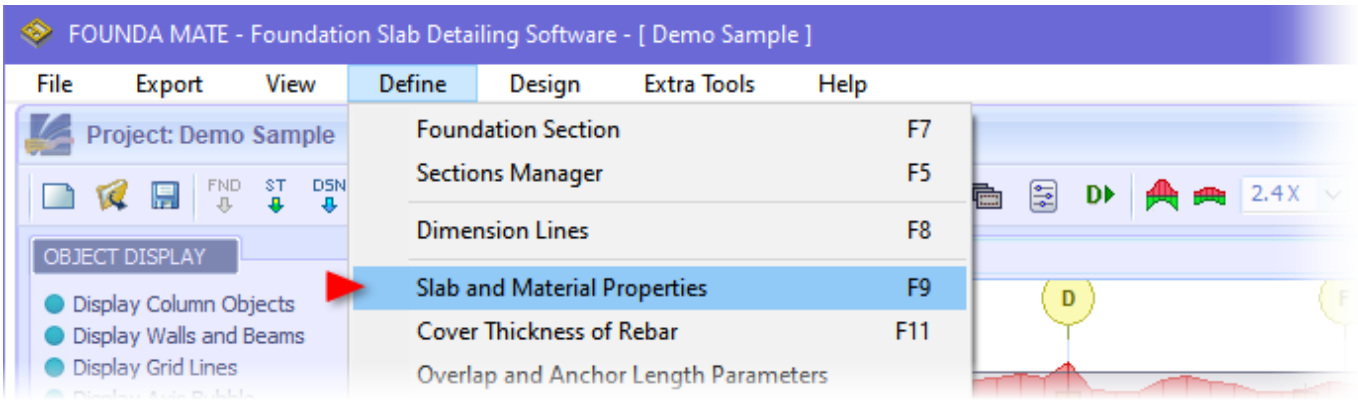
Note that you can save these settings for future projects via the **Configuration Settings** menu.



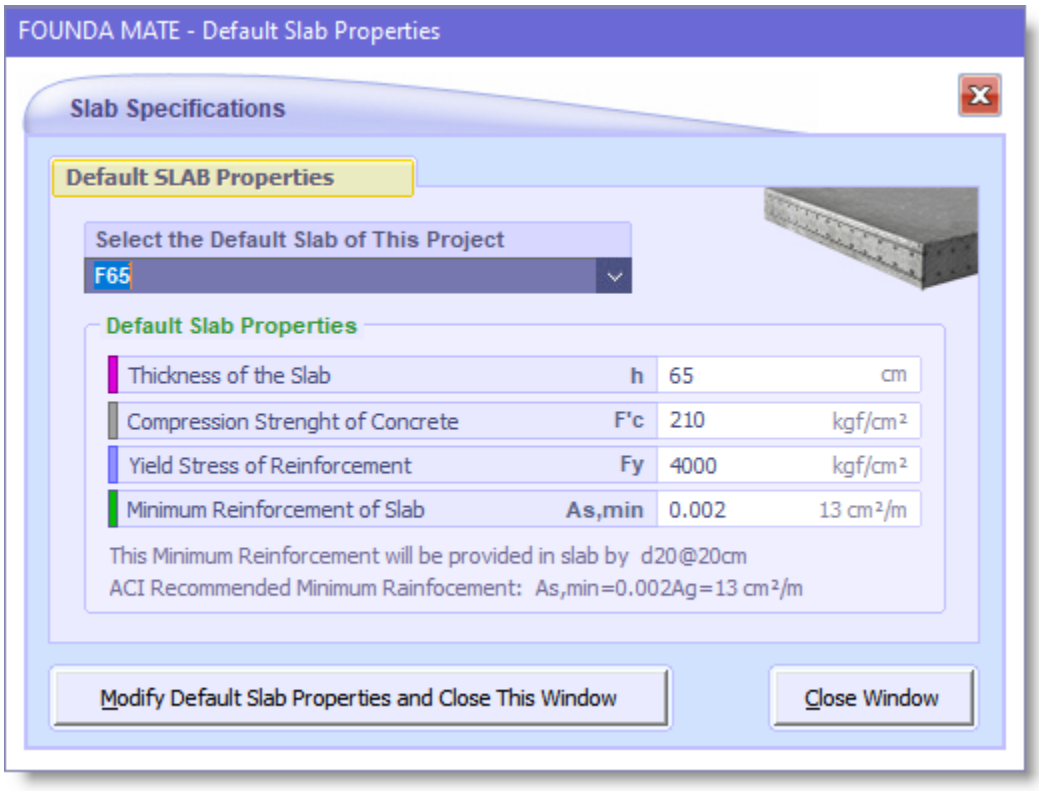
Specifying the Default Slab Properties of the Project

To specify the default Slab properties, proceed as follows from the **Define** menu, as shown in the image:

Define Menu > Slab and Material Properties



In this interface, you must select the **Default Slab** from the dropdown list at the top of the window. As shown in the image below, upon selecting the **Default Slab**, the properties defined for this item in **SAFE** are displayed in the **Default Slab Properties** section. Additionally, the minimum required reinforcement percentage is calculated by the software based on the material specifications and the relevant code, and is displayed in the **Minimum Reinforcement of Slab** section. Note that all of these parameters are user-editable.



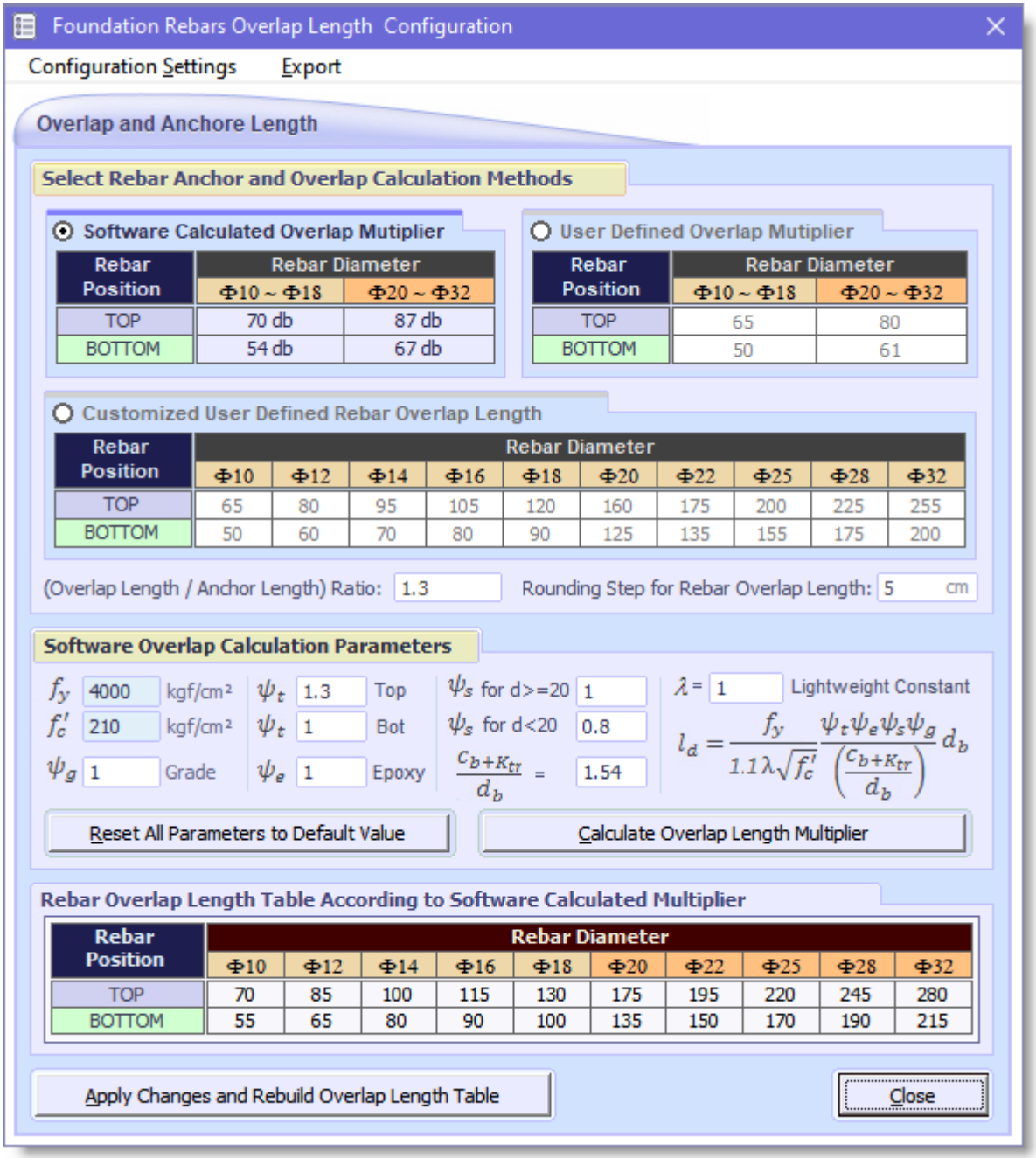
FOUNDA MATE
Foundation Slab Detailing Software

Configuring Splice and Development Length Settings of Rebars

To configure splice and development length settings, proceed as follows from the Define menu:

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 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 in the following.



Rebar Position	Rebar Diameter Φ10 ~ Φ18	Rebar Diameter Φ20 ~ Φ32
TOP	70 db	87 db
BOTTOM	54 db	67 db

Rebar Position	Rebar Diameter Φ10 ~ Φ18	Rebar Diameter Φ20 ~ Φ32
TOP	65	80
BOTTOM	50	61

Rebar Position	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
TOP	65	80	95	105	120	160	175	200	225	255
BOTTOM	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 λ = 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 $\frac{c_b+K_{tr}}{d_b}$ = 1.54

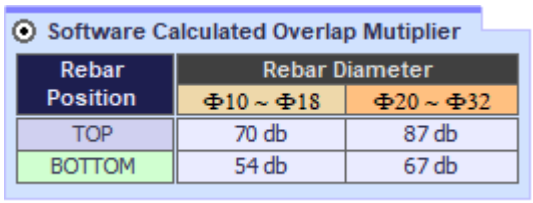
$$l_d = \frac{f_y}{1.1\lambda\sqrt{f'_c}} \frac{\psi_t\psi_e\psi_s\psi_g}{\left(\frac{c_b+K_{tr}}{d_b}\right)} d_b$$

Rebar Position	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
TOP	70	85	100	115	130	175	195	220	245	280
BOTTOM	55	65	80	90	100	135	150	170	190	215

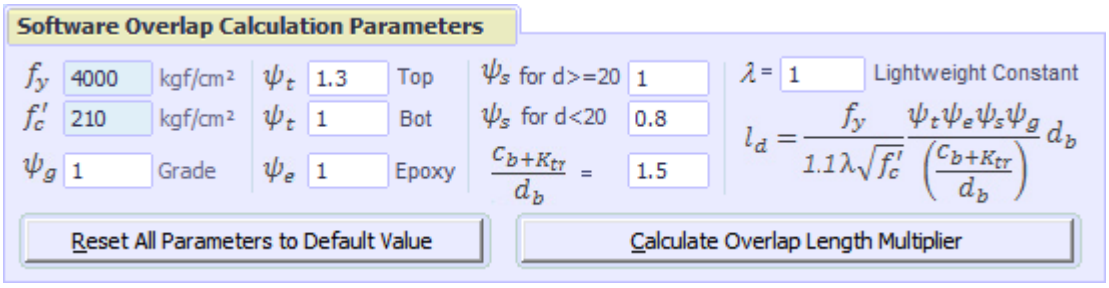
In this user interface, you are presented with three different options for the method of calculating the development and splice lengths:

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.

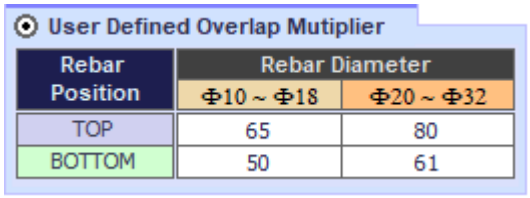


In this mode, the software calculates the development length and splice length factors for various rebar sizes and positions based on the formula shown in the image below, using the user-configurable parameters in the **Software Overlap Calculation Parameters** section, as well as the compressive strength 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.

Customized User Defined Rebar Overlap Length

Rebar Position	Rebar Diameter									
	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
TOP	55	80	95	105	120	160	175	200	225	255
BOTTOM	50	60	70	80	90	125	135	155	175	200

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 **View** menu and the **Rebar Anchor / Overlap Length Table** option, and can also be exported to AutoCAD.

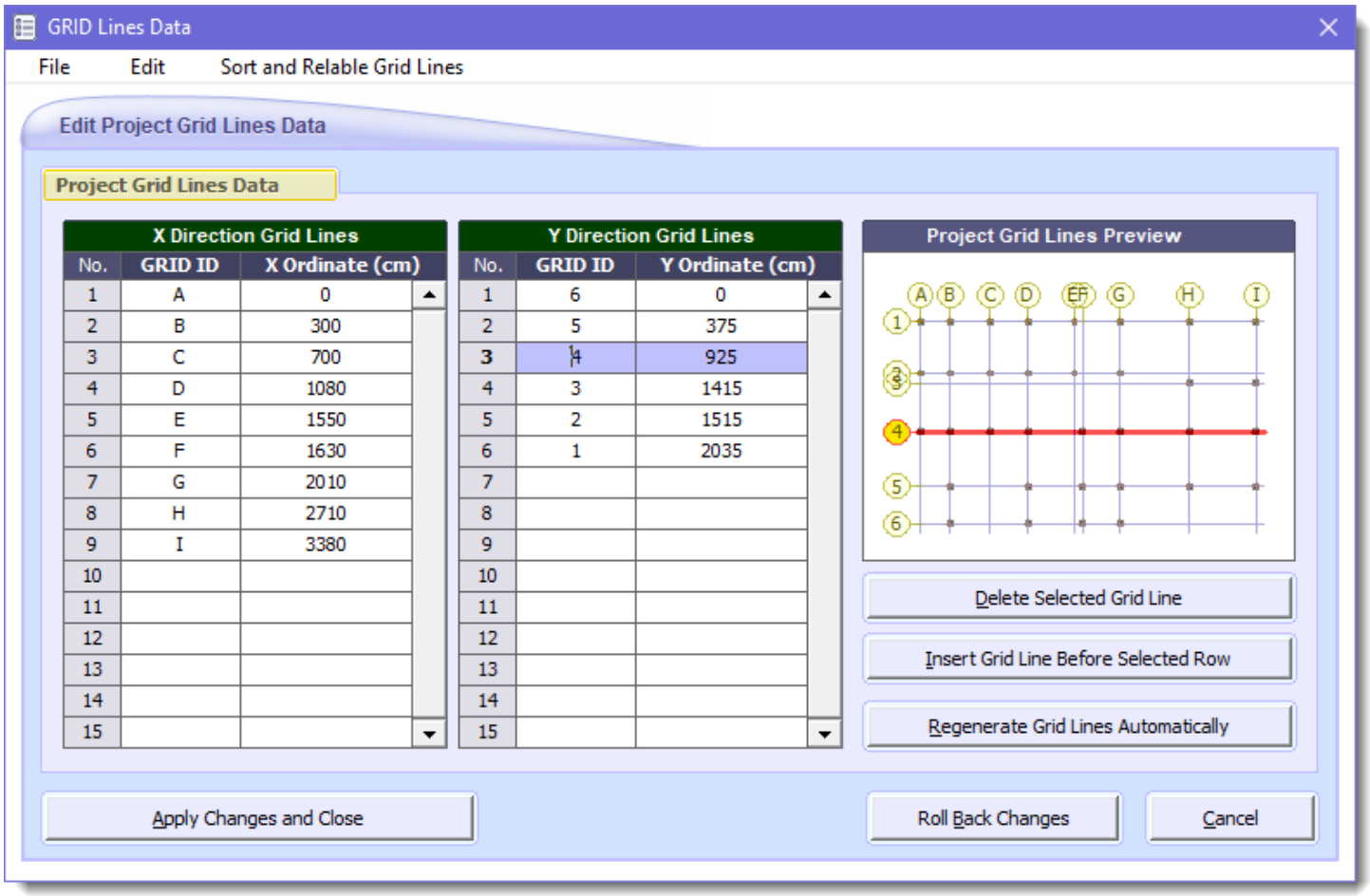
FOUNDA MATE
Foundation Slab Detailing Software

Grid Lines Configuration

The **GRID Line Data** interface allows you to view the grid line information defined in the structural model and, if necessary, edit this information, which includes the names and coordinates of the project grid lines. To access this interface, proceed as follows from the **Define** menu:

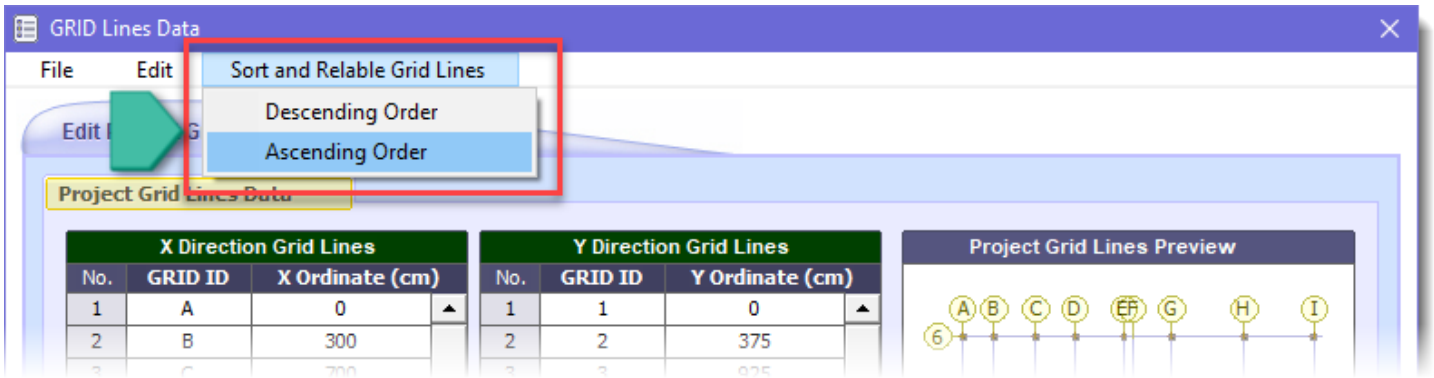
Define Menu > Grid Lines of Project (F12)

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 **FOUNDA 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
  ....
```

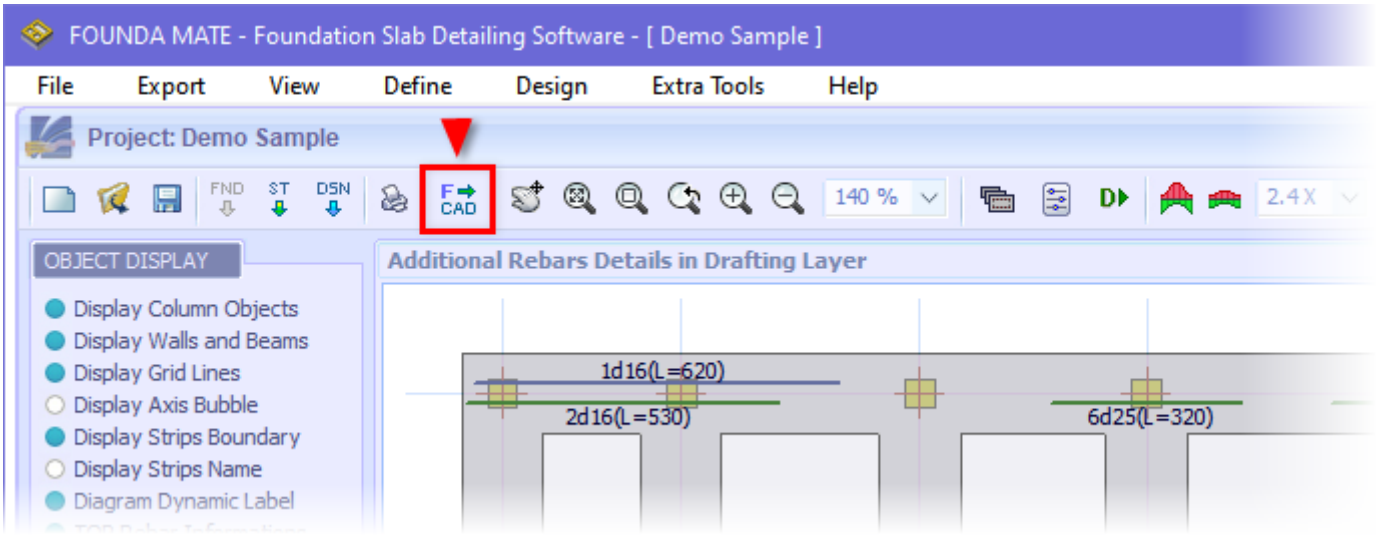
At the bottom of this interface, pressing the **Apply Changes and Close** button saves and applies the modifications made. The **Roll Back Changes** button allows you to discard the changes and revert to the initial settings.

FOUNDA MATE
Foundation Slab Detailing Software

4 Generating Structural Drawings

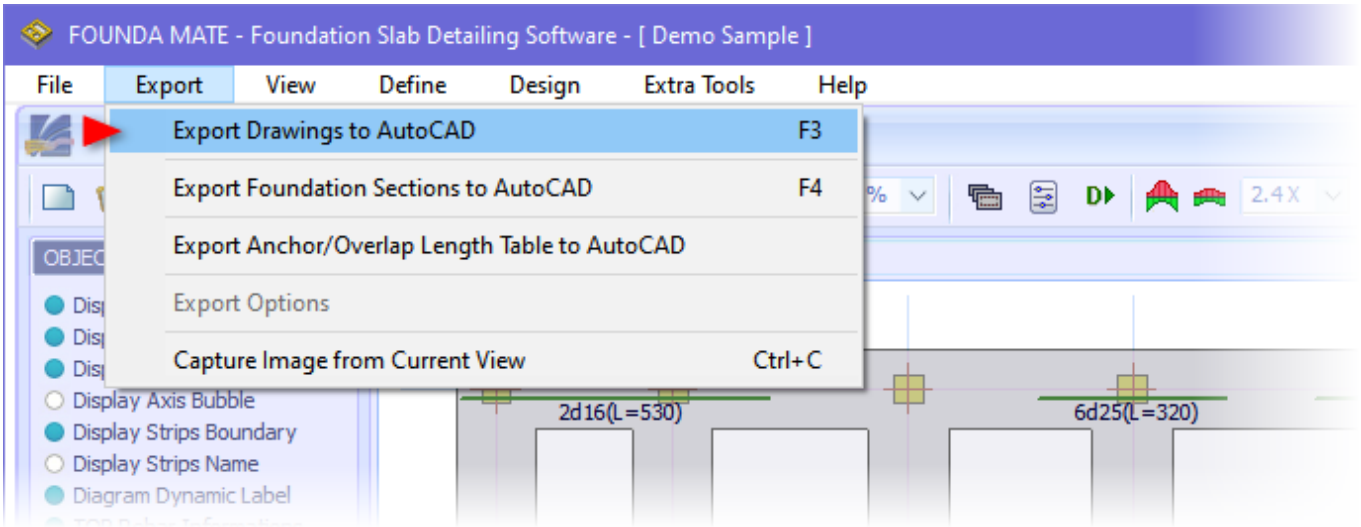
After designing the additional reinforcement, performing the necessary edits, and defining the cross sections and dimension lines, you can issue the command to generate the structural drawings. The software will then draw and produce the structural drawings, along with the corresponding rebar schedules, in the standard **AutoCAD** drawing format in a fraction of a second.

These drawings include the dimensioning and cross section location plan, the bottom mesh additional reinforcement plan, the top mesh additional reinforcement plan, the bottom mesh typical reinforcement plan, the top mesh typical reinforcement plan, the defined cross sections, and the model rebar schedules. To generate the drawings, you may use the F CAD icon in the main toolbar at the top of the software, or use the F3 shortcut key.



You may also proceed as follows from the **Export** menu:

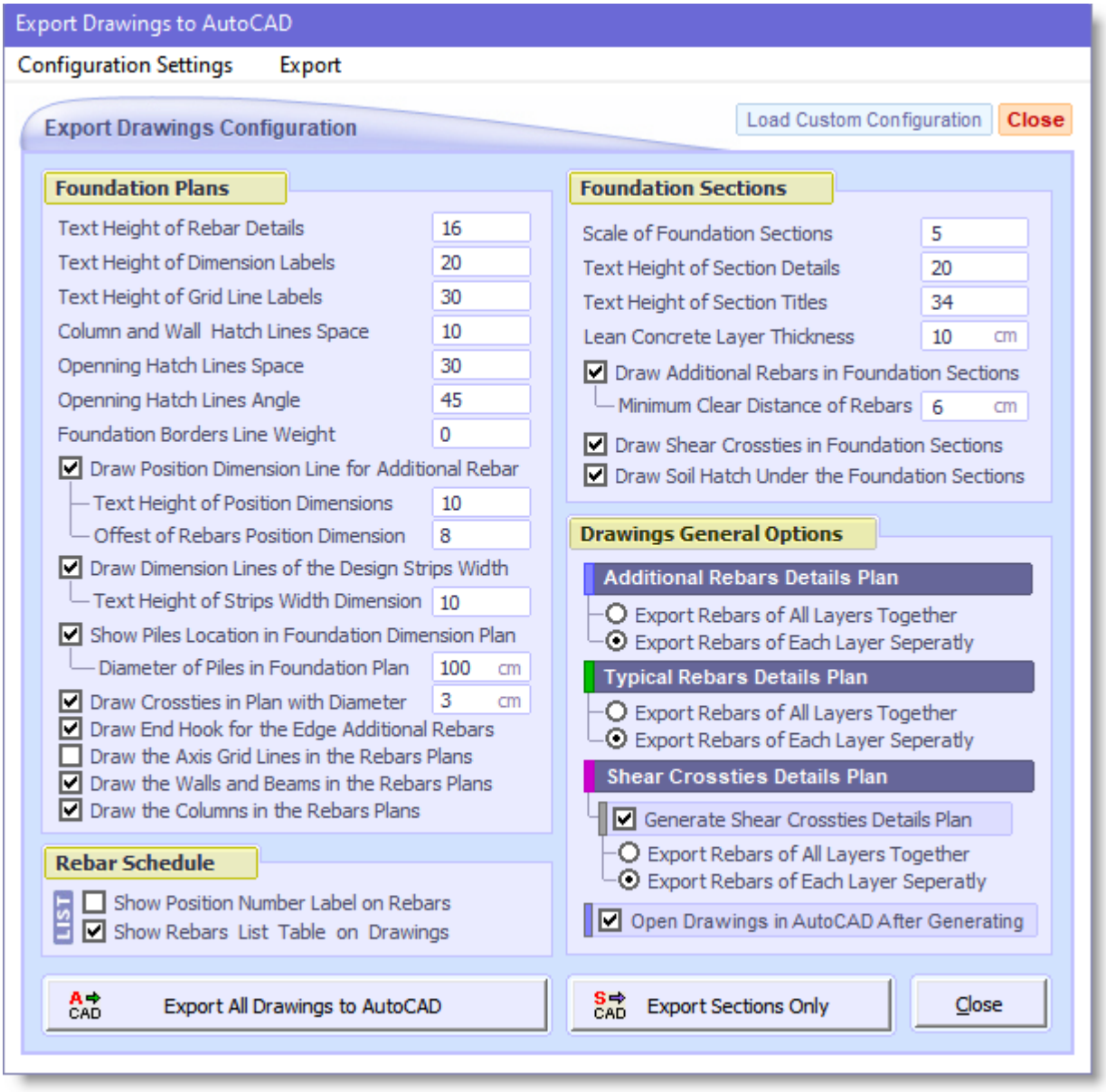
Export Menu > Export Drawings to AutoCAD (F3)



FOUNDA MATE
Foundation Slab Detailing Software

Structural Drawing Configuration

As explained above, upon issuing the drawing generation command, the configuration interface for the structural drawing generation settings appears, as shown below.



This interface consists of four separate sections, as described below. The parameters available in each of these sections will be explained in the following:

Foundation Plans
Foundation Sections
Drawings General Options
Rebar Schedule

Foundation Plans

This part of the **Export Drawing to AutoCAD** interface includes the configuration and settings for drawing the foundation plans. The description of the parameters available in this section is provided below:

Text Height of Rebar Detail
The font size of the detail annotations in the plans.

Text Height of Dimension Labels
The font size of the dimension line annotations in the plans.

Text Height of Grid Line Labels
The font size of the grid line labels in the plans.

Column and Wall Hatch Lines Space
The spacing between the hatch lines of shear walls and columns in the plan.

Opening Hatch Lines Space
The spacing between the hatch lines of openings in the plan.

Opening Hatch Lines Angle
The angle of the opening hatch lines in the plan.

Foundation Borders Line Weight
The drawn line weight of the foundation edges.

Draw Position Dimension Line for Additional Rebar
Option to draw or hide the position dimension lines for additional rebars.

Text Height of Position Dimensions
The font size of the additional rebar position dimension lines.

Offset of Rebars Position Dimensions
The offset distance of the additional rebar position dimension lines.

Draw Dimension Line of the Design Strip Width
Option to draw or hide the dimension lines for the design strip widths.

Text Height of Strip Width Dimension
The font size of the design strip width dimension annotations.

Show Piles Location in Foundation Dimension Plan
Option to draw or hide the pile locations in the foundation dimensioning plan.

Diameter of Piles in Foundation Plan
The diameter of the circles representing the piles in the foundation dimensioning plan.

Draw Crossties in Plan Diameter

Option to draw or hide the shear crossties in the plan as hollow circles with a diameter specified by this parameter.

Draw End Hook for the Edge Additional Rebars

Option to draw or hide the end hooks for additional rebars that are close to the foundation edge.

Draw the Axis Grid Lines in the Rebar Plans

Option to draw or hide the axis grid lines in the rebar detail plans other than the dimensioning plan.

Draw the Walls and Beams in the Rebar Plans

Option to draw or hide the walls and beams in the rebar detail plans other than the dimensioning plan.

Draw the Column in the Rebar Plans

Option to draw or hide the columns in the rebar detail plans other than the dimensioning plan.

Foundation Sections

This part of the **Export Drawing to AutoCAD** interface includes the configuration and settings for drawing the foundation cross sections, comprising the following parameters:

Scale of Foundation Sections

The drawing scale of the foundation cross sections.

Text Height of Sections Details

The font size of the foundation cross section detail annotations.

Text Height of Sections Title

The font size of the foundation cross section titles.

Lean Concrete Layer Thickness

The thickness of the lean concrete layer beneath the foundation.

Draw Additional Rebars in Foundation Sections

Option to draw or hide the additional rebars in the foundation cross sections.

Minimum Clear Distance of Rebars

The minimum acceptable clear distance between rebars in the cross sections. If the distance between rebars in the cross sections falls below this value, the software automatically transfers some of the rebars to another layer to ensure the rebar spacing never falls below the value specified by this parameter.

Draw Shear Crossties in Foundation Sections

Option to draw or hide the shear crossties in the foundation cross sections.

Draw Soil Hatch Under the Foundation Sections
Option to draw or hide the soil hatch beneath the foundation.

Column and Wall Hatch Lines Space
The spacing between the hatch lines of shear walls and columns in the plan.

Drawings General Options

This part of the **Export Drawing to AutoCAD** interface contains the general drawing generation settings, including the methods for generating the structural drawings. It comprises the following parameters:

Additional Rebars Details Plan
The method for drawing the additional reinforcement plans, which includes two methods:

1. **Export Rebars of All Layers Together**
   The additional rebars for all layers **A**, **B** and **Other** are drawn together in a single plan.
2. **Export Rebars of Each Layer Separately**
   The additional rebars for each of the layers **A**, **B** and **Other** are drawn in separate plans.

Typical Rebars Details Plan
The method for drawing the typical reinforcement plans, which includes two methods:

1. **Export Rebars of All Layers Together**
   The typical rebars for all layers **A**, **B** and **Other** are drawn together in a single plan.
2. **Export Rebars of Each Layer Separately**
   The typical rebars for each of the layers **A**, **B** and **Other** are drawn in separate plans.

Shear Crossties Details Plan
The options and methods for drawing the beam shear reinforcement plans.

Generate Shear Crossties Details Plan
Option to draw or hide the shear crosstie details plan.

1. **Export Rebars of All Layers Together**
   The beam shear crossties for all layers **A**, **B** and **Other** are drawn together in a single plan.
2. **Export Rebars of Each Layer Separately**
   The beam shear crossties for each of the layers **A**, **B** and **Other** are drawn in separate plans.

Open Generated Drawings in AutoCAD
Specifies whether the structural drawings are automatically opened in AutoCAD upon completion of the drawing generation.

Rebar Schedule

This part of the **Export Drawing to AutoCAD** interface pertains to the rebar schedule drawing settings and rebar numbering, and includes the following parameters:

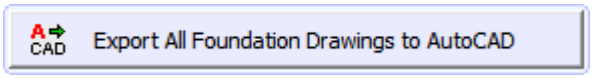
Show Position Number Label on Rebars

Option to display or hide the position number label on each drawn rebar.

Show Rebar List Table on Drawings

Option to draw or hide the rebar schedule tables alongside the structural drawings.

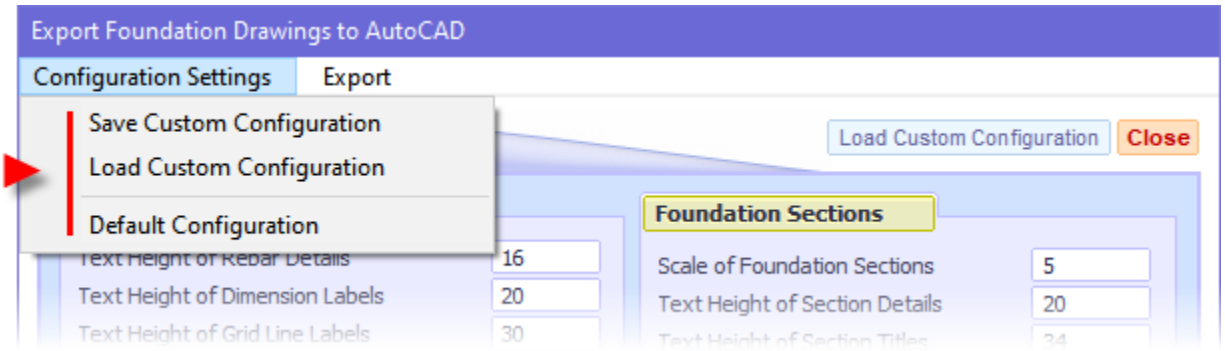
This interface also includes two buttons for generating the structural drawings, as follows:



By using this button, all structural drawings are generated and saved at your desired location. Samples of these drawings are provided on the following pages.

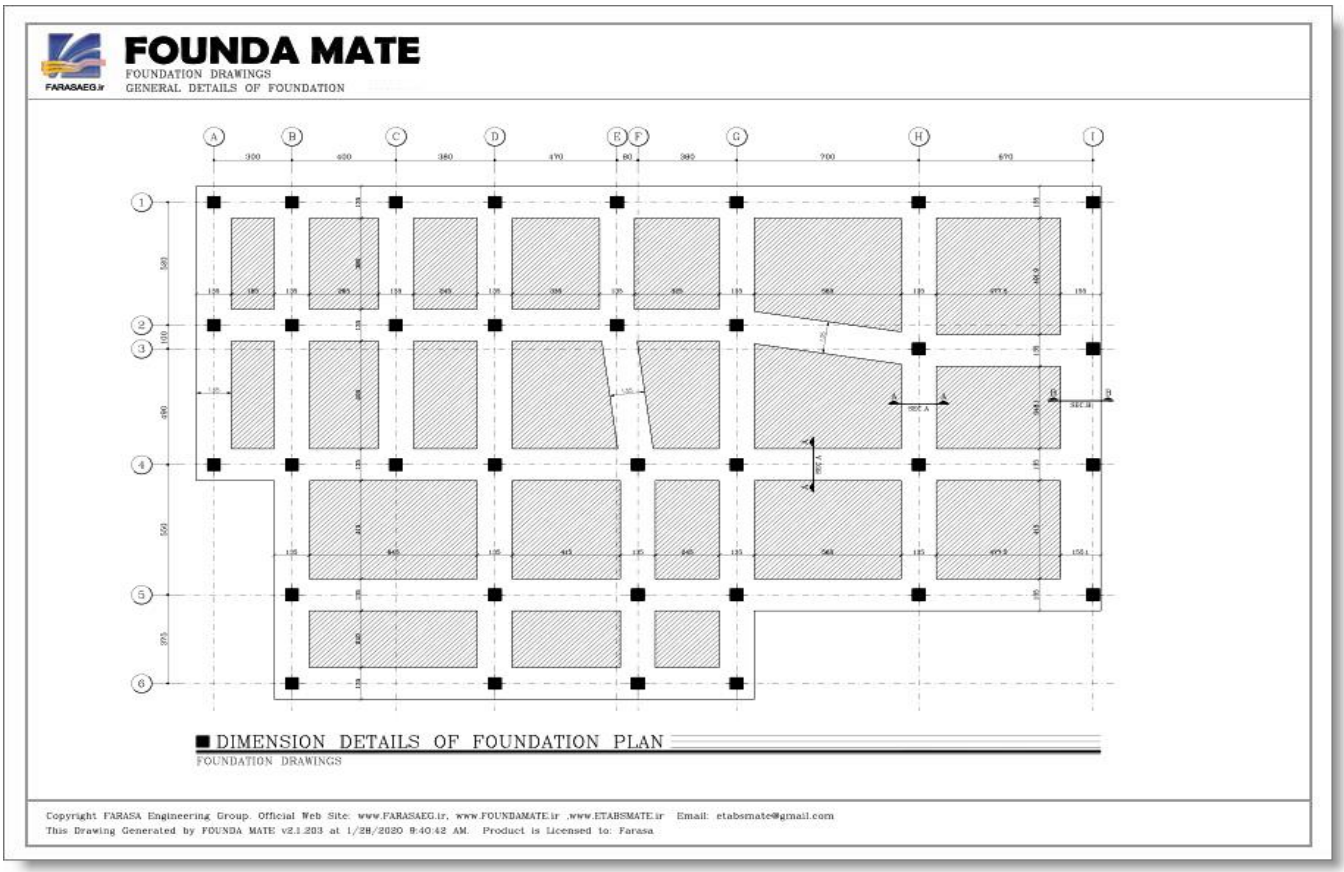
By using this button, only the cross section drawings are generated and saved at your desired location.

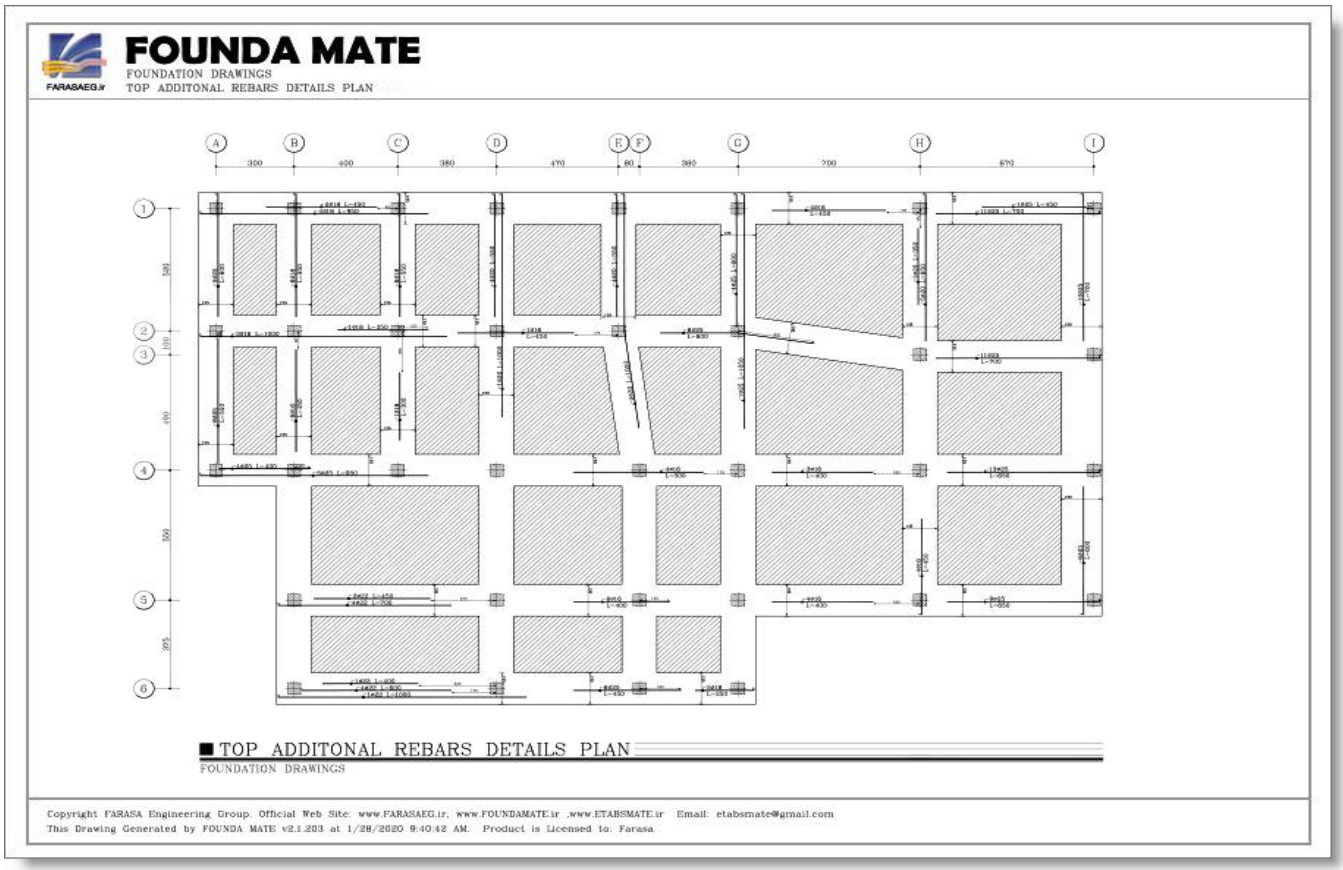
Note that through the **Configuration Settings** menu at the top of this interface, you can save your customized configurations for use in future projects.



Samples of the structural drawings generated by the software are provided in the following.

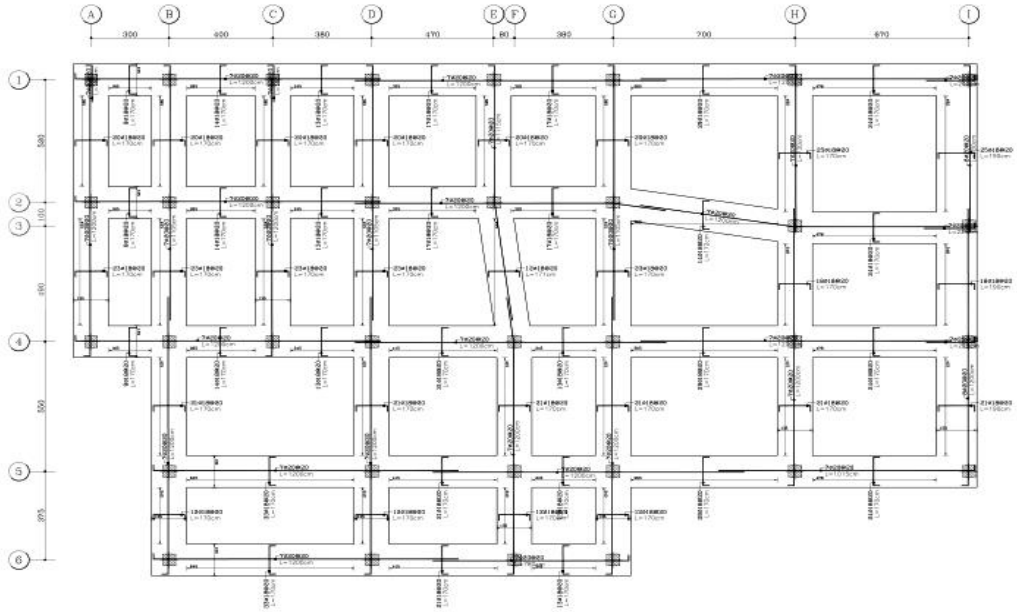
FOUNDA MATE
Foundation Slab Detailing Software





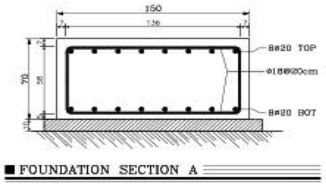
FOUNDA MATE
Foundation Slab Detailing Software

FOUNDA MATE
FOUNDATION DRAWINGS
BOTTOM TYPICAL REBARS DETAILS

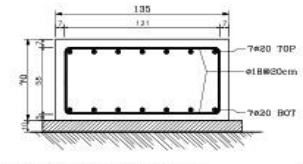


■ BOTTOM TYPICAL REBARS DETAILS PLAN
FOUNDATION DRAWINGS

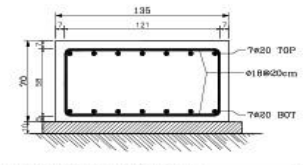
FOUNDA MATE
All Dimension in Centimeter



■ FOUNDATION SECTION A



■ FOUNDATION SECTION B



■ FOUNDATION SECTION C

FOUNDA MATE – Longitudinal Rebar List of Foundation

Position Number	Rebar Shape (schematic)	Dia. (mm)	Length (cm)	Weight (kg)	Number (pcs)	Weight Total (kg)
1f		Φ16	400	6.3	79	495.8
2f		Φ16	430	6.8	18	122.2
3f		Φ20	470	11.6	11	127.5
4f		Φ16	470	7.4	2	14.8
5f		Φ16	430	6.8	5	42.9
6f		Φ18	400	8	10	79.9
7f		Φ16	520	8.2	10	82.1
8f		Φ20	530	13.1	5	65.4
9f		Φ25	530	20.4	6	122.5
10f		Φ16	450	7.1	3	21.3
11f		Φ16	350	5.5	1	5.5
12f		Φ16	430	6.8	1	6.8
13f		Φ20	1200	29.6	184	5445.3
14f		Φ20	190	4.7	7	32.8
15f		Φ20	171	4.2	1096	4622
16f		Φ20	1110	27.4	35	958.1
17f		Φ20	740	18.2	15	273.7
18f		Φ20	1200	29.6	98	2900.2
19f		Φ20	220	5.4	28	151.9
20f		Φ16	171	3.4	974	3327.1
21f		Φ18	186	3.7	128	475.6
22f		Φ20	790	19.5	7	136.4
23b		Φ20	1020	25.2	7	176.1
24f		Φ20	1150	28.4	35	992.6
25f		Φ20	770	19	15	284.8
26f		Φ20	320	7.9	21	165.7
27f		Φ20	820	20.2	7	141.6
28f		Φ20	1090	26.9	7	188.2
Foundation Longitudinal Rebars Summation					2815	21462 kg

FOUNDA MATE – Beam Rebars Summary Report

Rebar Size	Diameter (mm)	Length (m)	12m Bar Number	Weight (kg)
Φ16	16	478.1	40	751
Φ18	18	1965.1	184	3925
Φ20	20	6756.4	563	16962
Φ25	25	31.8	3	123
Foundation Rebars Total Weigth = 21462 kg (21.462 ton)				

Overlap Length Table for Foundation Rebars in cm
According to Rebar Position, Rebar Size and Properties of (F'c=210 Kg/cm², Fy=4200 Kg/cm²)

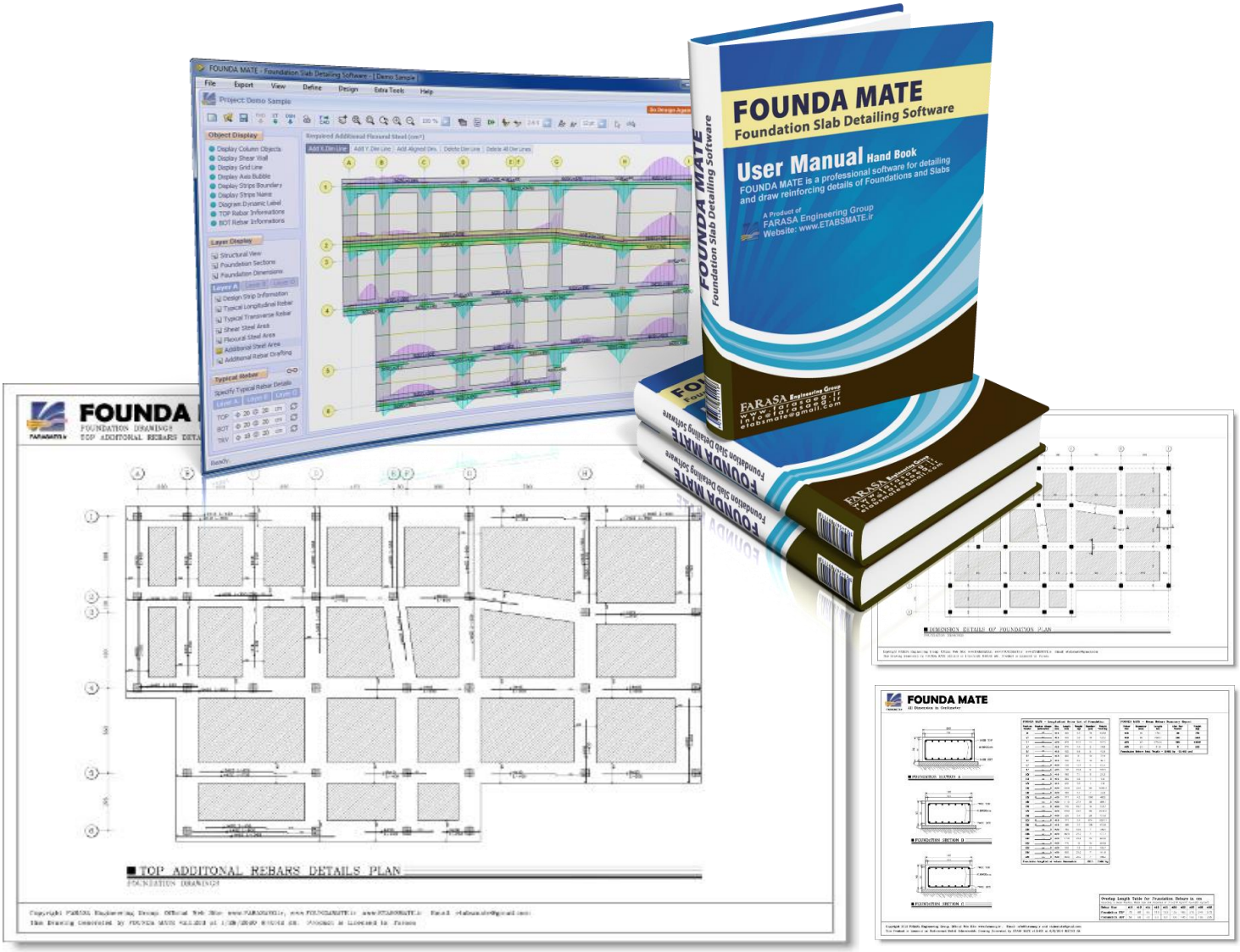
Rebar Size	Φ10	Φ12	Φ14	Φ16	Φ18	Φ20	Φ22	Φ25	Φ28	Φ32
Foundation TOP	70	80	95	110	120	135	185	210	240	270
Foundation BOT	50	60	70	80	90	100	145	160	180	205

FOUNDA MATE
Foundation Slab Detailing Software

Final Remarks and Recommendations

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 the 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 **FOUNDA 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 Instagram page of the software.



FOUNDA MATE
Foundation Slab 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	(+98) 09173171373 (+98) 09301325576
WhatsApp, Telegram and Other Messaging App	+989173171373 @etabsmate
Fax/Tel Number	(+98) 07132323810 (+98) 07132323811 (+98) 07132323812 (+98) 07132323813
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