Structural Analysis II.

-BMEEOTMAS42-

Syllabus for creating a model similar to the second homework assignment, using FEM-Design

Prepared by: Nagy Botond Előd

Reviewed by: Dr. Lógó János

Dr. Mohsen Ghaemi

Table of Contents

List of Figures

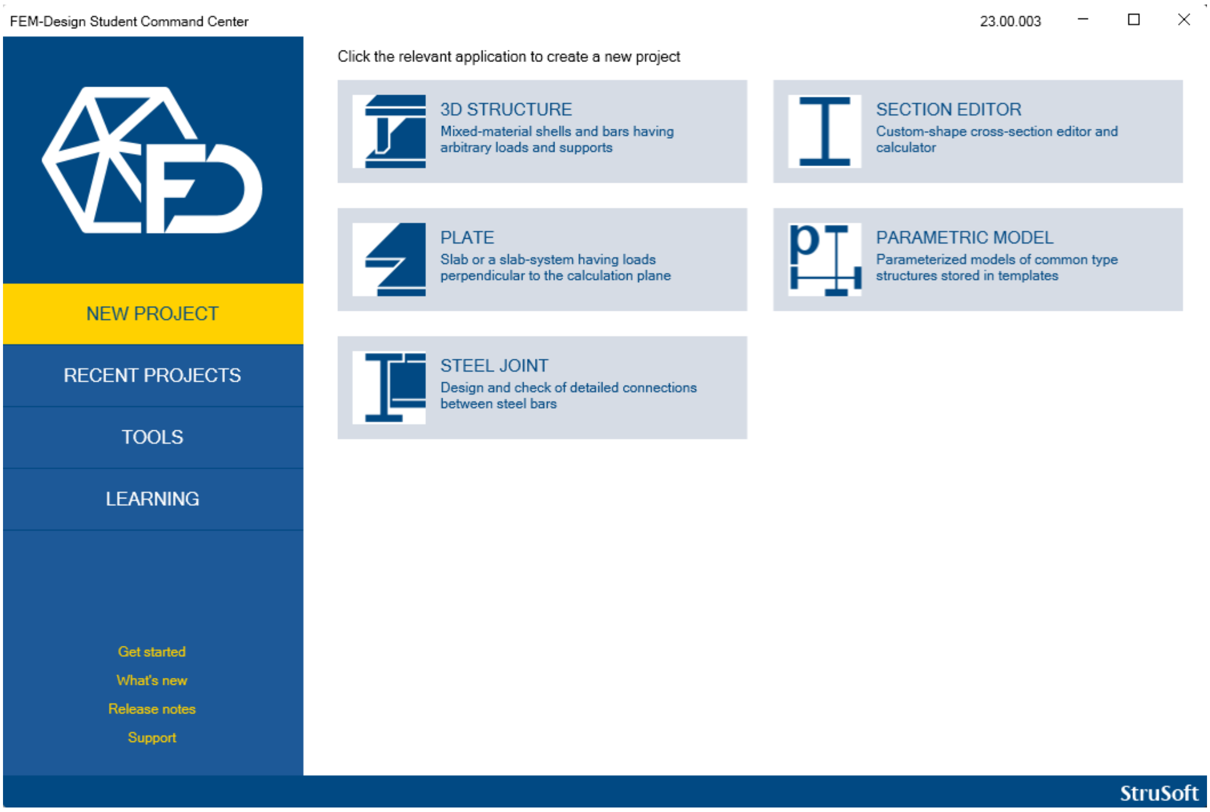
1. Installing FEM-Design

The latest student version of FEM-Design is available at the following link.

2. Getting Started

2.1 Selecting a Module

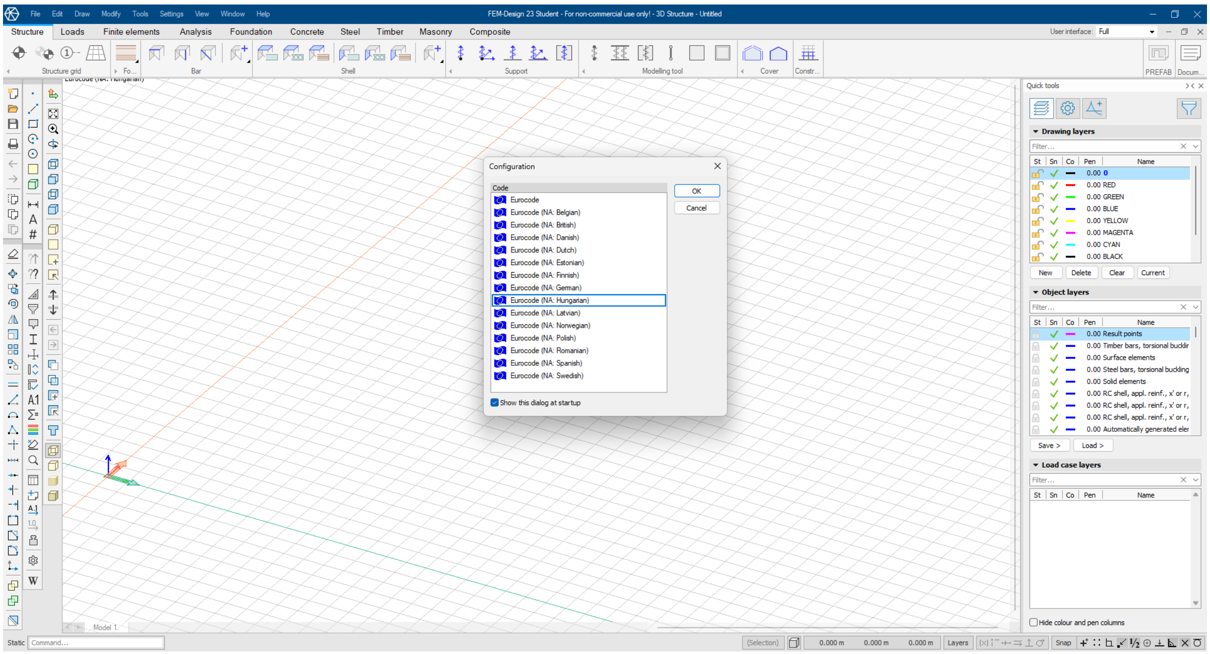
By entering the keywords "FEM-Design 23 Student" in the search menu, the following window will open. From the control panel, we can select the module we wish to use, open previously saved projects, and access useful links and educational materials that will help you easily learn how to use the program.



1. Figure: FEM-Design Control Panel - Student Version

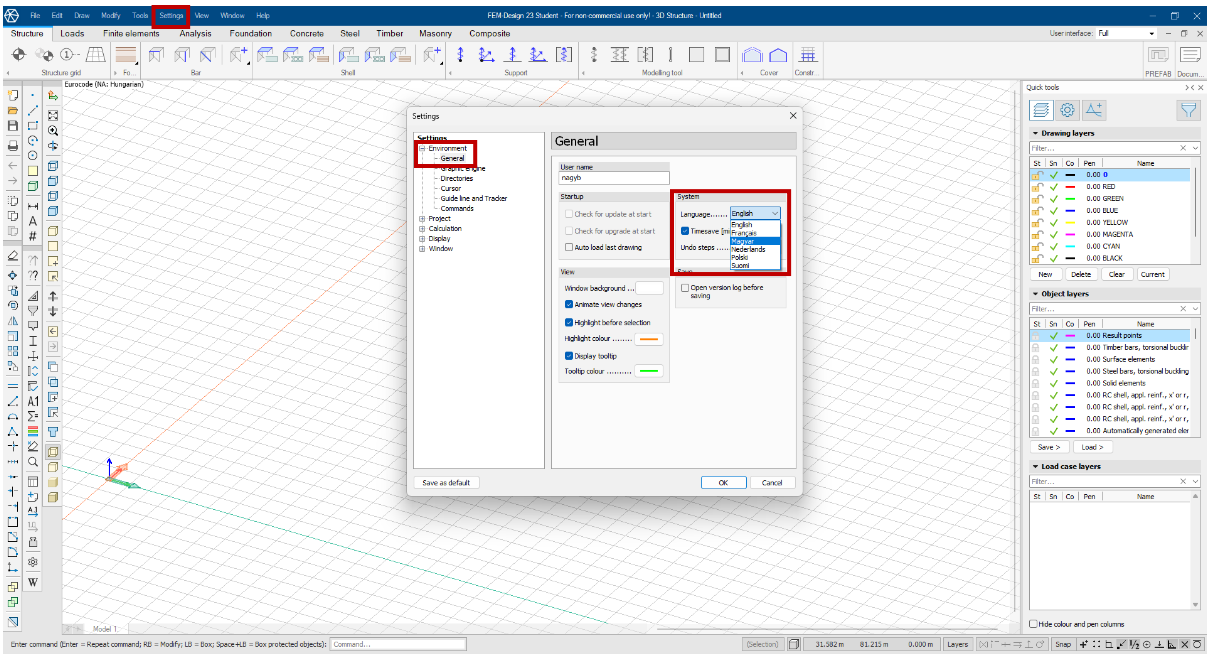
By clicking on the “3D STRUCTURE” icon, you can open the module you wish to use. As a first step, you always need to select the standard environment. Once these steps are completed, the program is ready for use.

For subsequent use, by entering the keywords "FEM-Design 3D STRUCTURE" in the search bar, you can directly open the desired module.



2. Figure: Standard Environment

2.2 Changing the Language of the Working Environment



3. Figure: Changing the Language of the Working Environment

1) Click on the “Settings” tab in the menu bar.
2) From the drop-down window, select the “Environment” option.

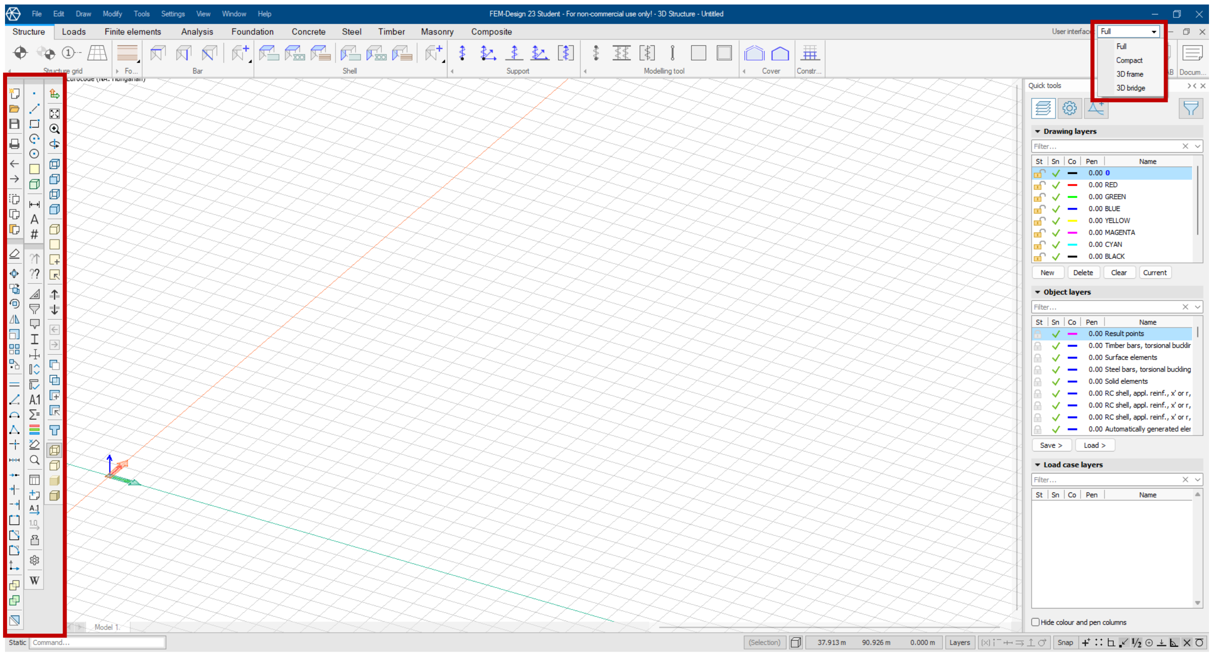
3) In the pop-up window, under the “General” section, choose the Hungarian working environment from the “Language” drop-down menu.
4) After accepting the changes, close the program, which will then reopen in the selected language.

Changing the language of the working environment is merely an option. It is advisable for everyone to use the language that is most comfortable for them.

2.3 Customization

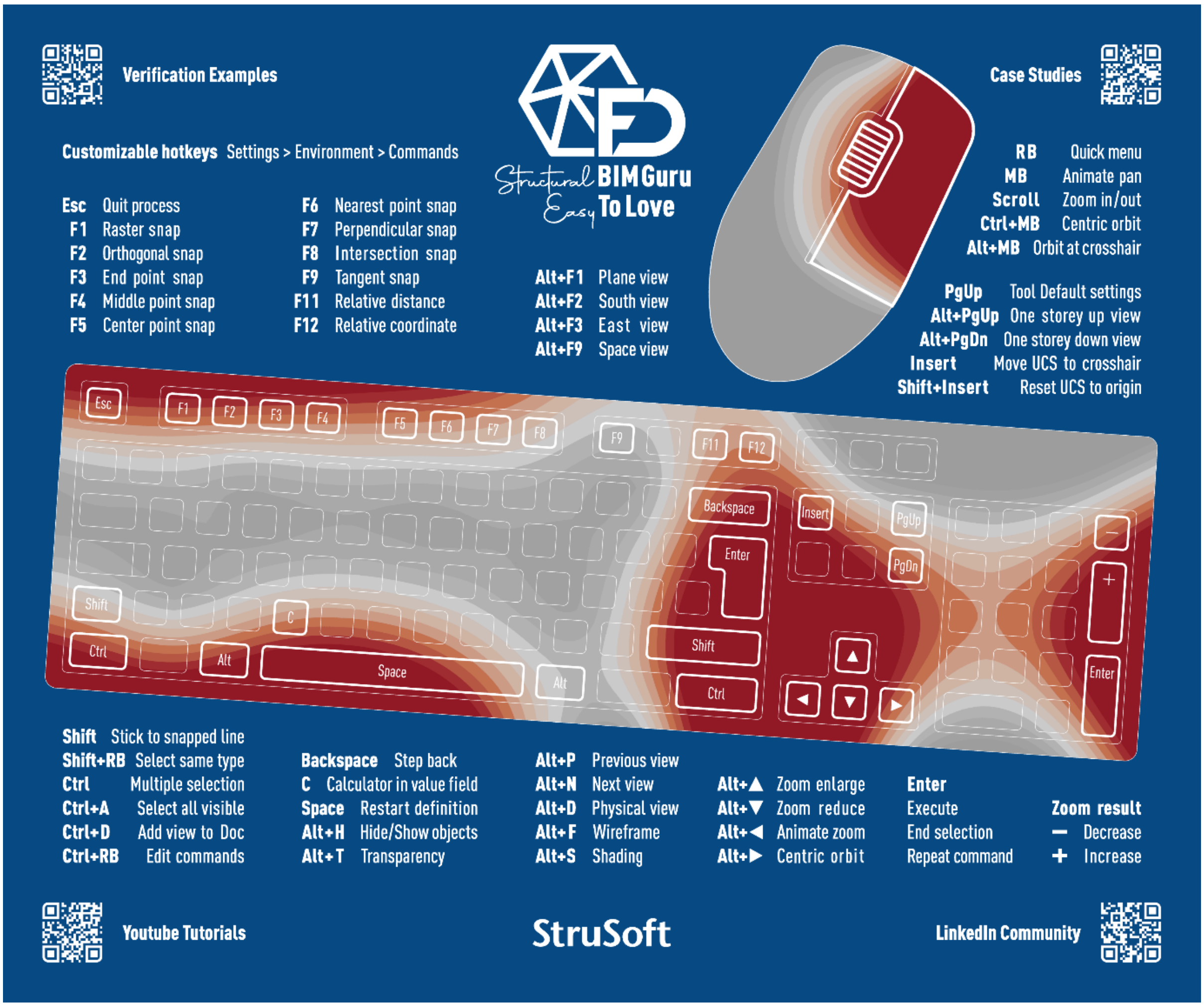
It is advisable to set the user interface to the “Full” option, so that all design possibilities are visible. The “Basic” option is useful for smaller screens, while using “3D Frame” or “3D Bridge” will hide unnecessary design options.

Further customization is possible by moving the side quick-edit options.

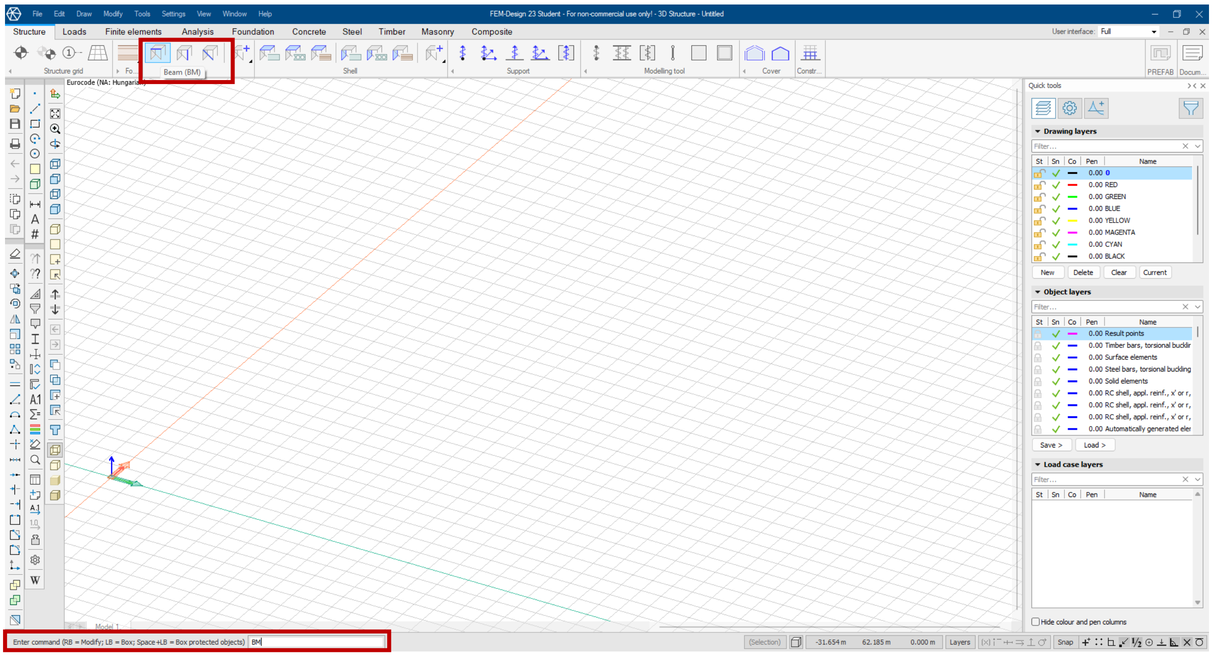


4. Figure: Customization

To facilitate modeling, various keyboard shortcuts have been integrated into the program. Additionally, functions can be accessed not only by using the icons but also by typing their short commands. These commands can be displayed by hovering the cursor over the respective icon for a few moments. For example, in the case of Figure 7, typing the command "BM" and pressing the ENTER key will open the beam function.

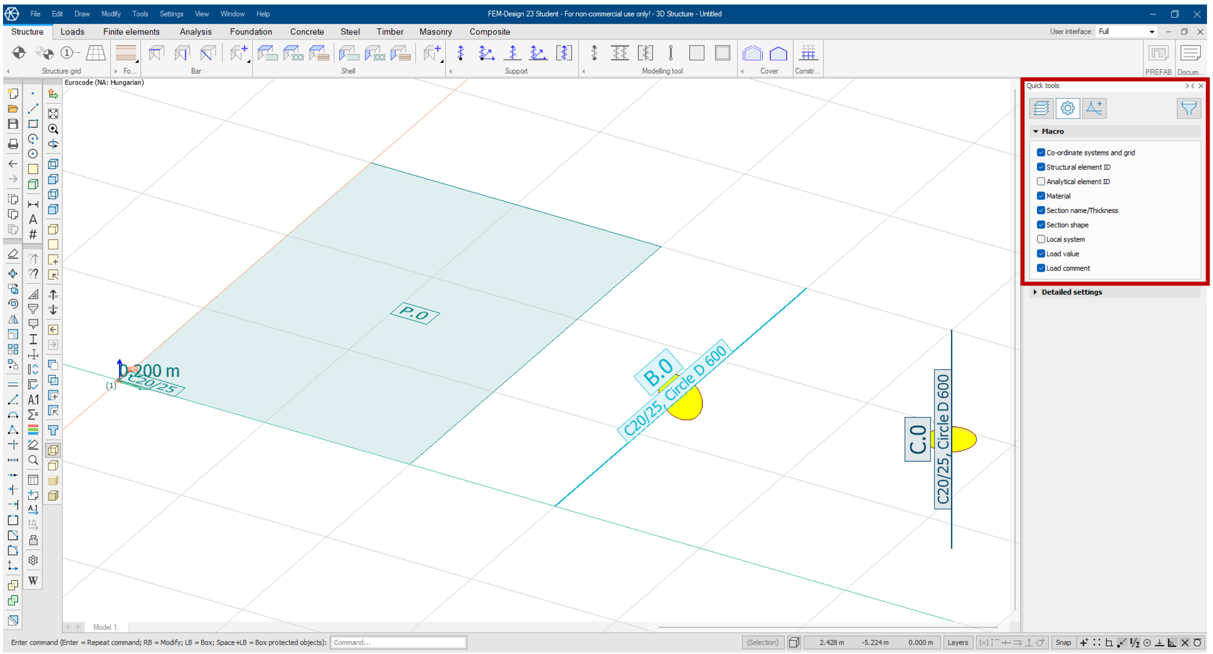


5. Figure: Keyboard Shortcuts



6. Figure: Commands

Additional customization options are available for labeling the properties of structural elements and loads. These will be useful for preparing documentation. They can be accessed under the “Quick Menu” tab in the “Display Settings” section.



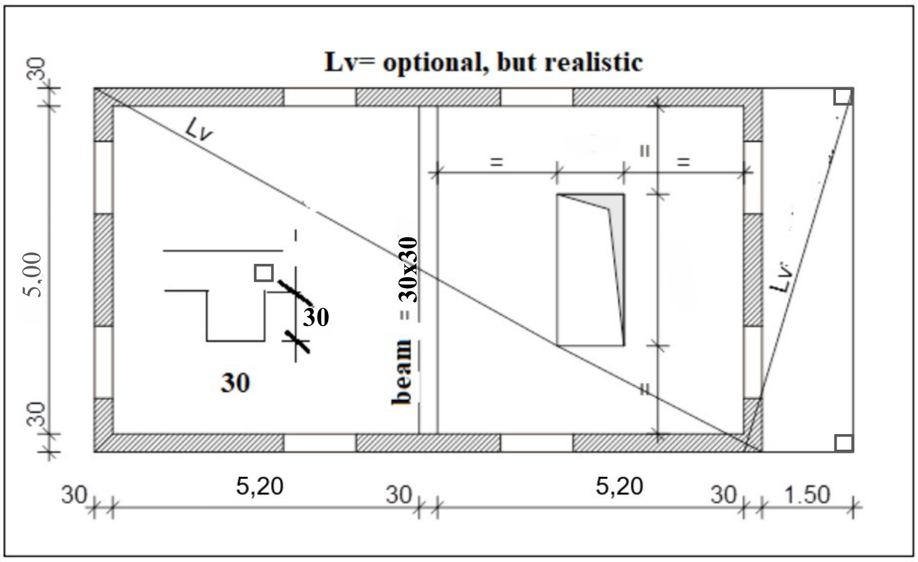
7. Figure: Display Settings

3. Sample Assignment

STRUCTURAL ANALYSIS II. – 2. HOMEWORK – SLABS

By the use of the **FEM-Design program** determine the displaments, internal forces, reaction forces of the RC slab with a master rib given the figure below.

STANDARD: **Eurocode** (HU). Qualities: concrete: **C25/30**. steel **B500B**



The thickness of the slabs should be realistic by the use of the given spans. The specified height dimension of the master beam of the bottom ribbed slab is to be understood from the lower plane of the slab.

Slab layers weight: inside:2,5 kN/m^2, balcony: 3,5 kN/m^2. Partition wall load value: 1,5 kN/m^2. Payload: indoors: 2,0 kN/m^2, balcony: 3,0 kN/m^2. The payload can also have a partial effect. At one short side edge of the staircase, the constant load of the stair is 20 kN/m and the payload of the stair is 10 kN/m. Consider the support capacity of prefabricated beams using a monolithic reinforced concrete rib above the gaps! Consider the line support of Porotherm brick walls with a realistic linear spring!

In a second model, change the height size of the master beam given in the first calculation so that you can compare the results of the two runs.

To be submitted:

- The documentation of the calculation should be prepared with the documentation editor of the FEM-Design PLATE program in the following detail:
- Static model with dimensions, profiles, loads, finite element mesh graphically; material characteristics, section data, plate thicknesses, load groups, load cases, load multipliers in tabular form.
- Drawings of the Load Cases with values.
- Standard slab deflection diagram from the **frequent SLS** load combination. (with values!)
- Standard internal force diagrams of reinforced concrete master beam and floor slab from **ULS** load combination (with values!).
- Prepare only the minimum required documentation from the second section selection!
- The effect of the change in stiffness on the displacements, the typical stresses of the master beam and the slab must be evaluated in tabular form (with actual values and percentage comparison)!
- Textual evaluation, conclusion, engineering explanation of what was seen in the previous point.
- Only the PDF file should be uploaded to Moodle. (The custom worksheet should be attached to the PDF documentation!)

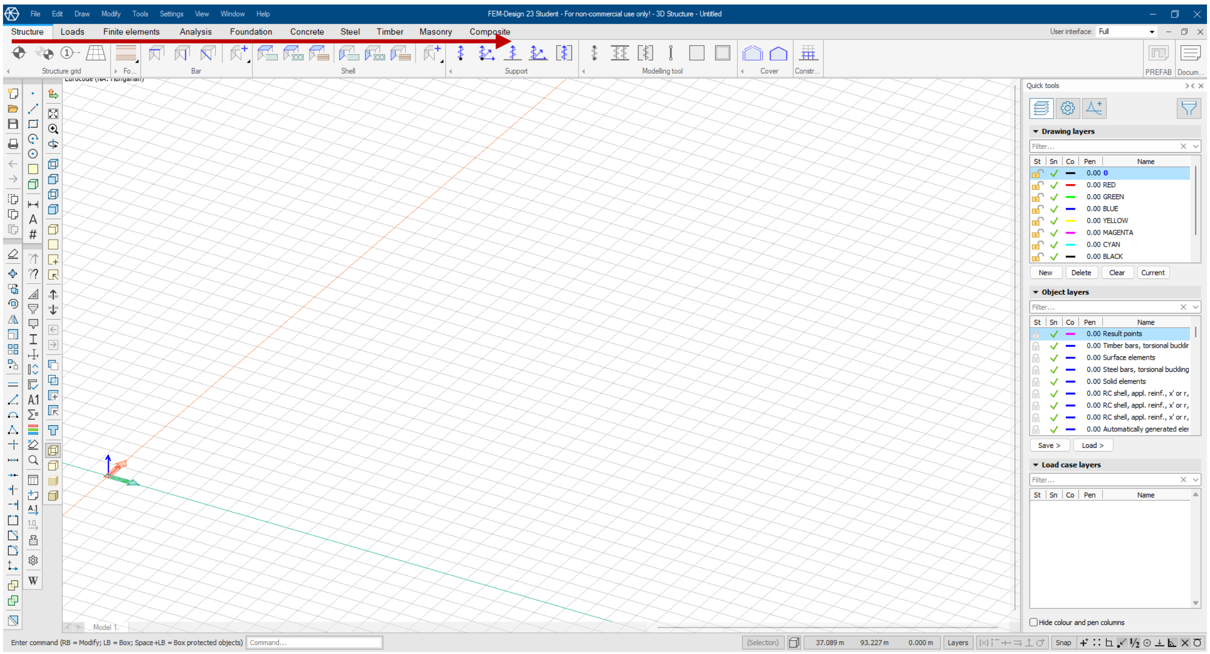
8. Figure: Sample Assignment

4. Modeling

It is important to note that the syllabus tracks the creation of a model similar to the Second Homework Assignment, so it does not fully match the model specified in the homework and, in some cases, goes beyond it. Any information provided during consultations takes precedence over the instructions in this guide.

The main concept for modelling the structure is to proceed from left to right, establishing the following order:

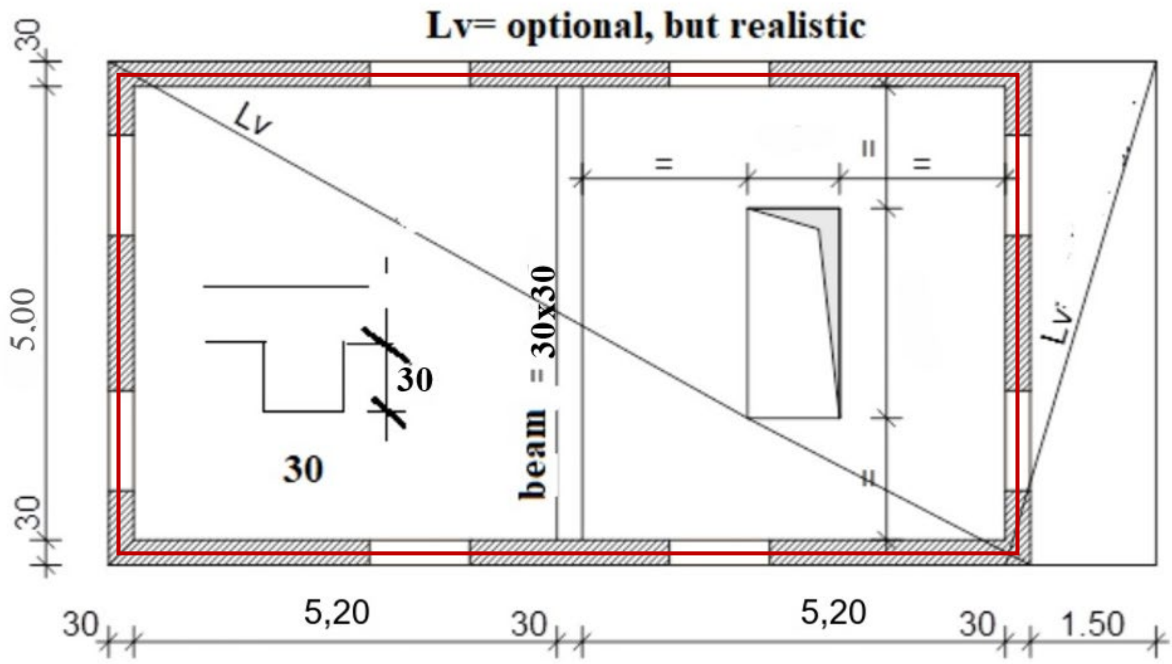
Structure → Loads → Finite Elements → Analysis and Results → Design Options



9. Figure: Modelling Logic

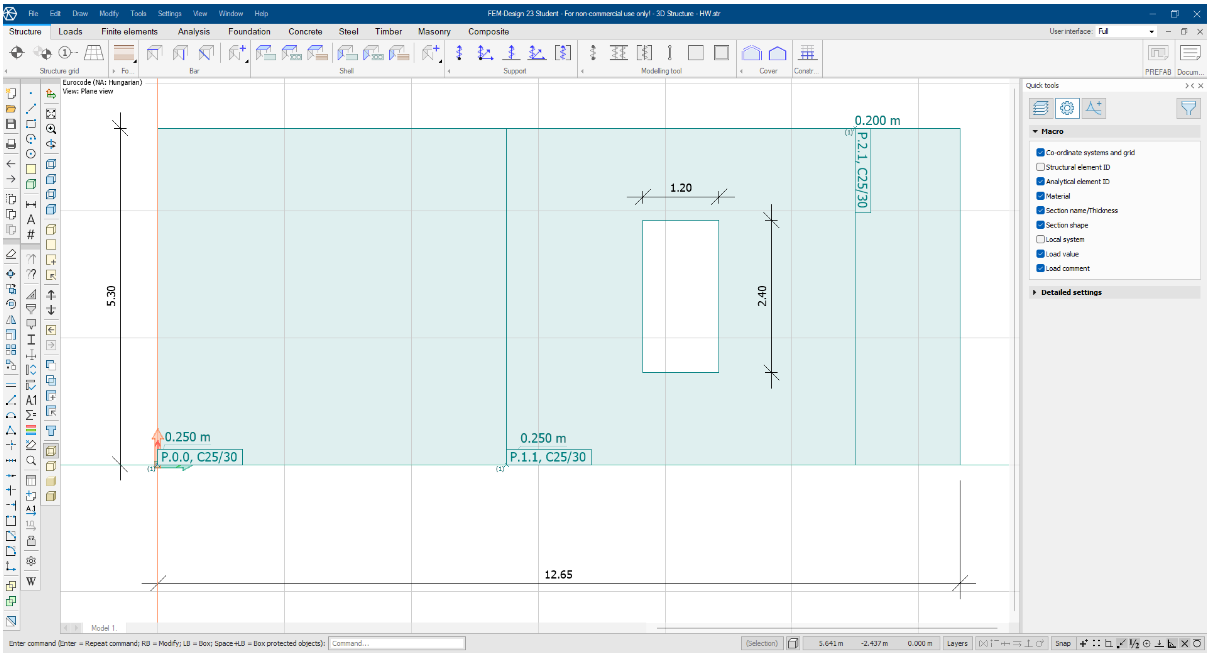
4.1 Editing Geometry

In the finite element model, supports are considered to be on the center plane of the wall, so we model the walls as supporting structures here. It is important to emphasize that the static framework and the architectural model do not match each other, neither in dimensions nor in the alignment of elements, and this should be kept in mind.



10. Figure: Support

First, we model the floor slab, as this will make it easier to place the walls that function as supports. When specifying dimensions, we take into account that we are modeling the center plane of the walls.

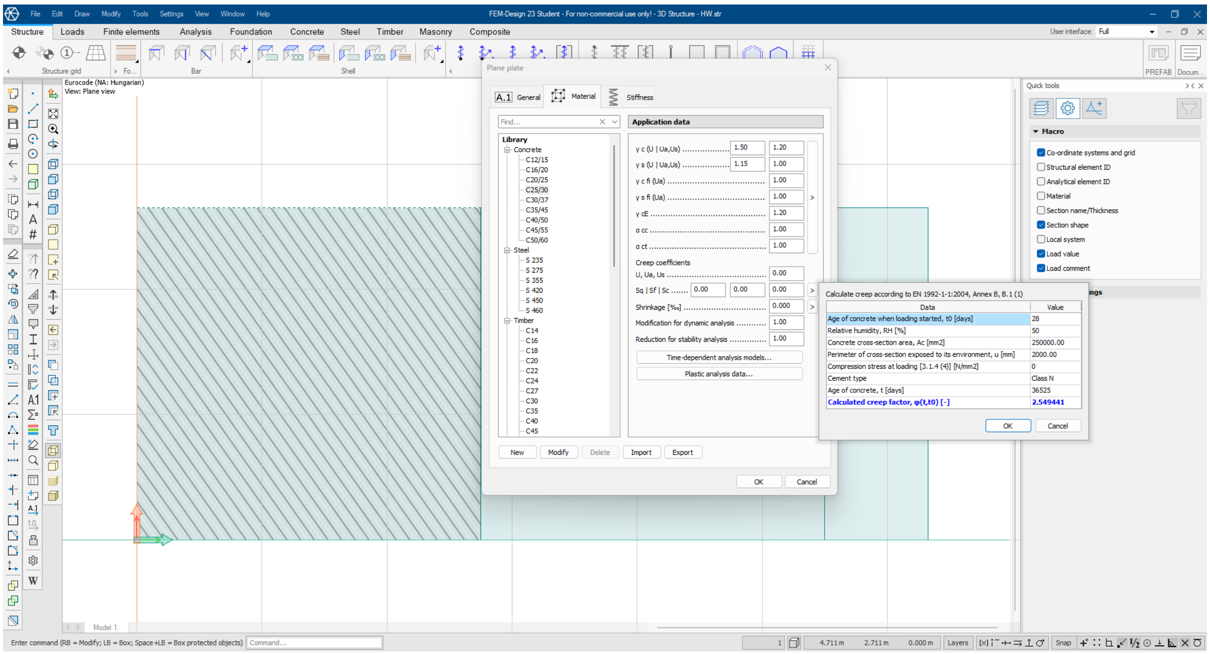


11. Figure: Modeling the Floor Slab

The slab is modeled in three separate parts to simplify the placement of loads; by selecting only the surface element, the partial placement becomes easier.

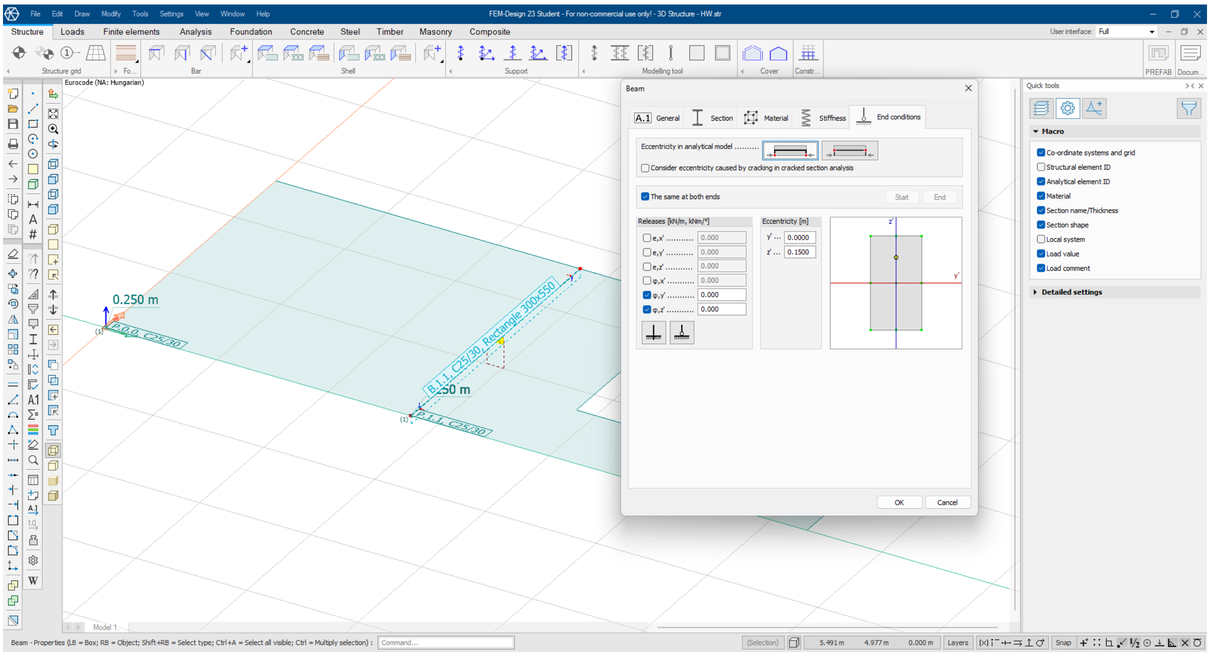
After performing quick manual calculations, we determine the location of the slab openings, which allows us to place them in the model almost instantly by pressing the F12 key. This key calculates coordinates based on the cursor's position at the moment of pressing, so it is crucial to ensure the cursor does not move.

If the task requires it, the creep factor of the structural elements can be adjusted within the material properties.

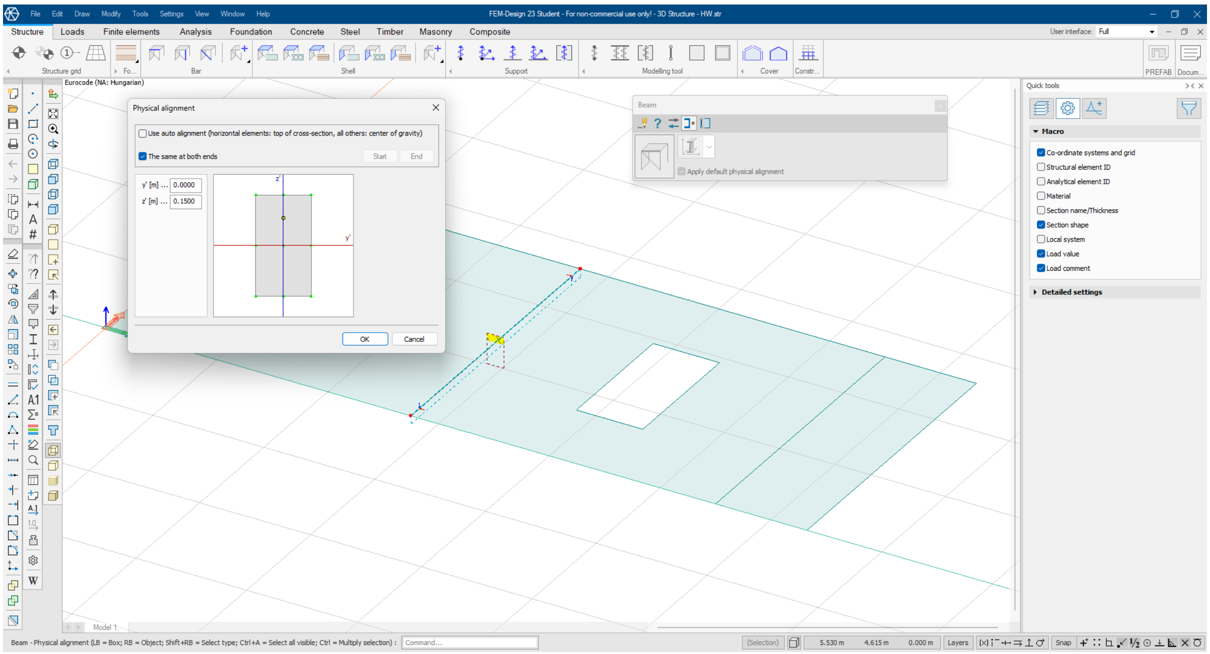


12. Figure: Setting Creep

The next step is placing the beams, for which it is important to set the correct edge precision. Since the beam size of 30x30 specified in the assignment is measured from the bottom plane of the slab, you need to add the thickness of the 25 cm slab. The top plane of the beam needs to be offset to match the top plane of the slab. To do this, subtract half of the slab thickness from half of the total height of the beam. This provides the correct edge precision, which can be set in the beam properties. Additionally, the physical model should be offset to ensure that the edge precision is also reflected in the physical model.



13. Figure: Beam Eccentricity



14. Figure: Setting Physical Alignment

We also set the edge precision for the physical model so that it appears correctly in the physical view. This does not affect the calculations in any way.

The difference between analytical and physical edge precision and when to use each is explained in the FEM-Design user manual, which can be found on the FEM-Wiki site.

First type ("End releases applied at the ends of the theoretical axis") is typically used for modeling RC bars which work together with concrete slabs (slab normal forces transfers to the bar).

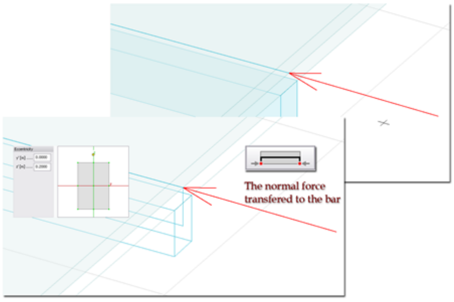


Figure: Modeling of a "ribbed concrete slab"

The second type ("End releases applied at the ends of the gravity (physical axis)") is for modeling single plate and bars which are not working together.

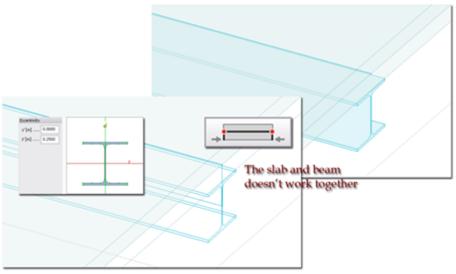
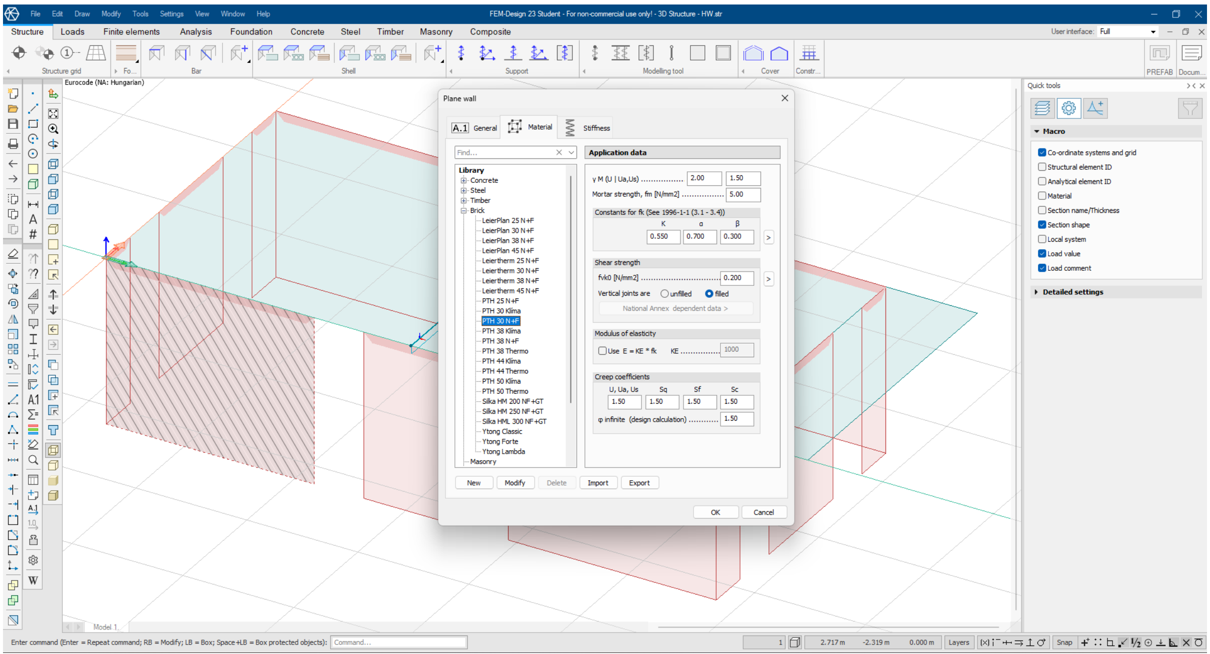


Figure: Modeling of a steel beam under slab in 3D Structure module

- **Physical model**
  Eccentricity of the physical model can be set in the Physical model tab, in a similar way as the **Eccentricity of analytical model**. It has no effect on the calculation.

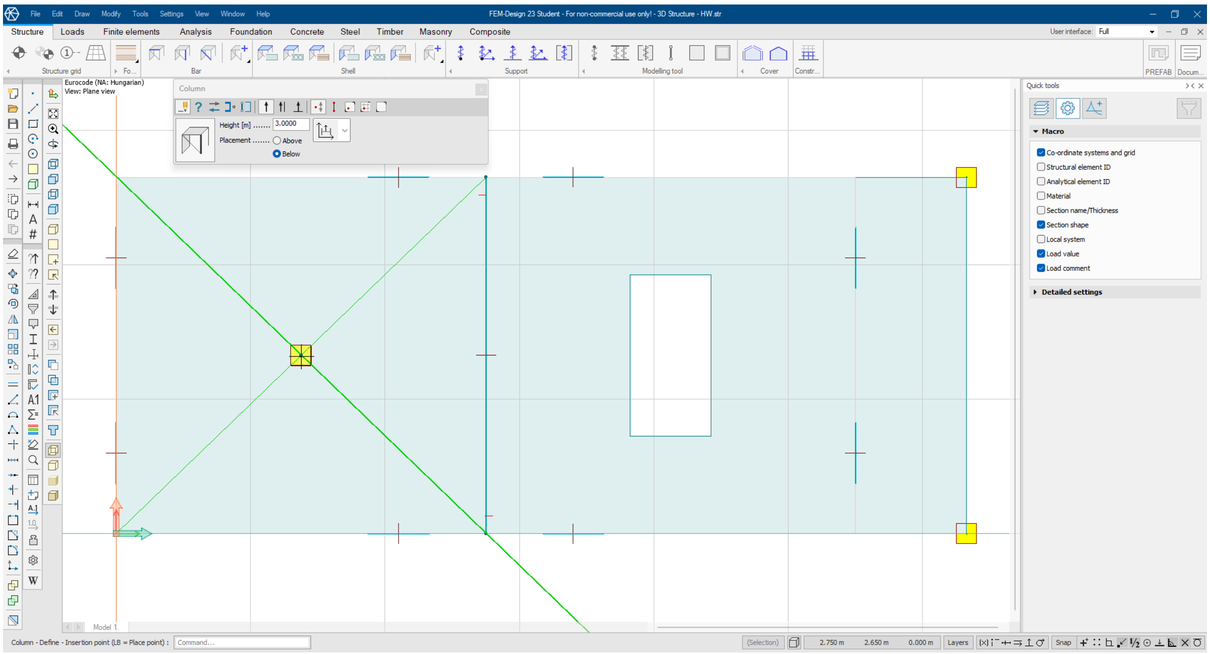
15. Figure: Explanation of Eccentricity

When modeling the walls, you need to use the specified thickness and material for the wall elements. Since the task description specifies that the wall is connected to the slab with a hinged connection, this setting must also be configured. It is important to note that different standard environments provide different options, so you should be careful to select the appropriate one.



16. Figure: Selecting a Masonry Block

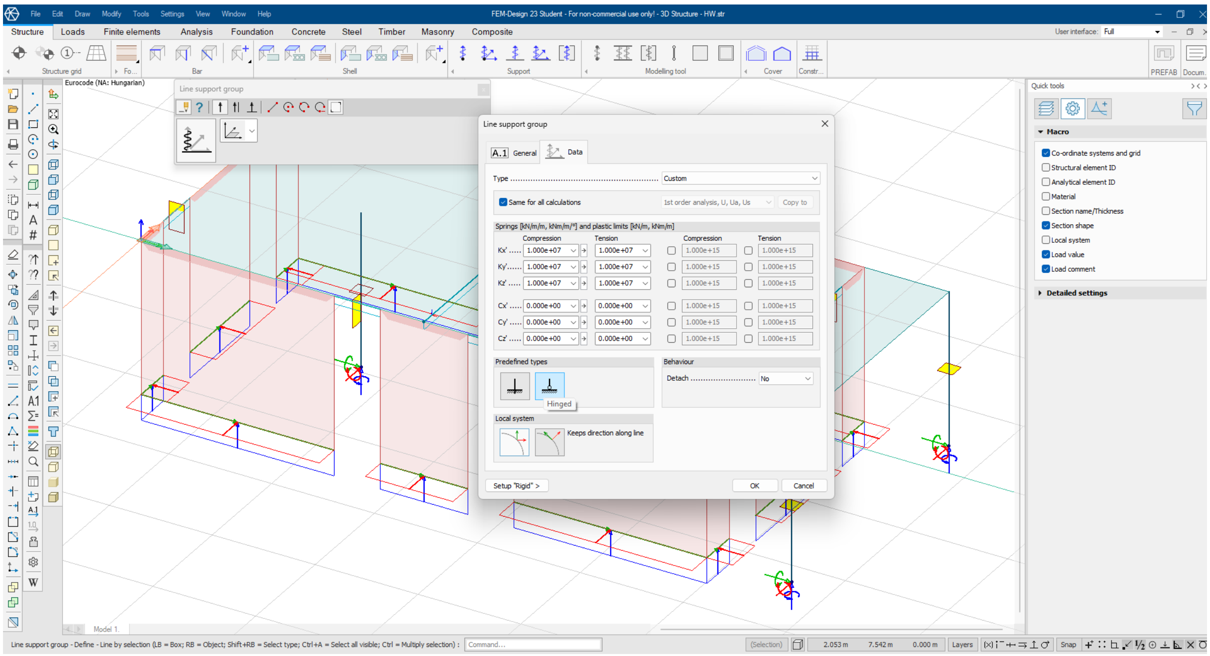
Next, we model the secondary beams, which is done similarly to the primary beams. They can be easily and quickly modeled using the copy function.



17. Figure: Construction Lines

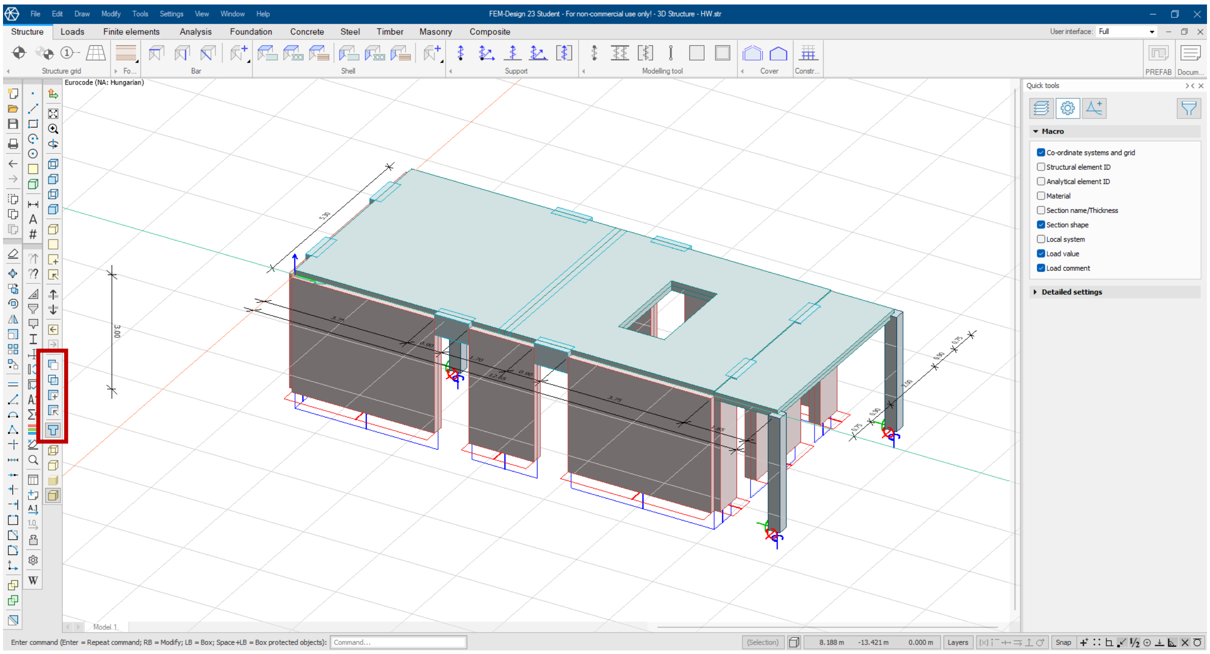
After setting the appropriate column properties, you can create editing lines by holding down the “CTRL” key and clicking with the left mouse button. If you release the key, you will get an extended editing line. If you do not release it, a green line will be drawn only between the two selected points, which can be removed using the “ESC” key.

The placement of supports defined as hinged is done by selecting lines. It is important to pay attention to the direction in which the selection window is drawn, whether from right to left or in the opposite direction. The logic is similar to the technique used in AutoCAD, where elements are selected in one direction if they are touched by the window, and fully enclosed elements are selected in the other direction.



18. Figure: Placing Supports

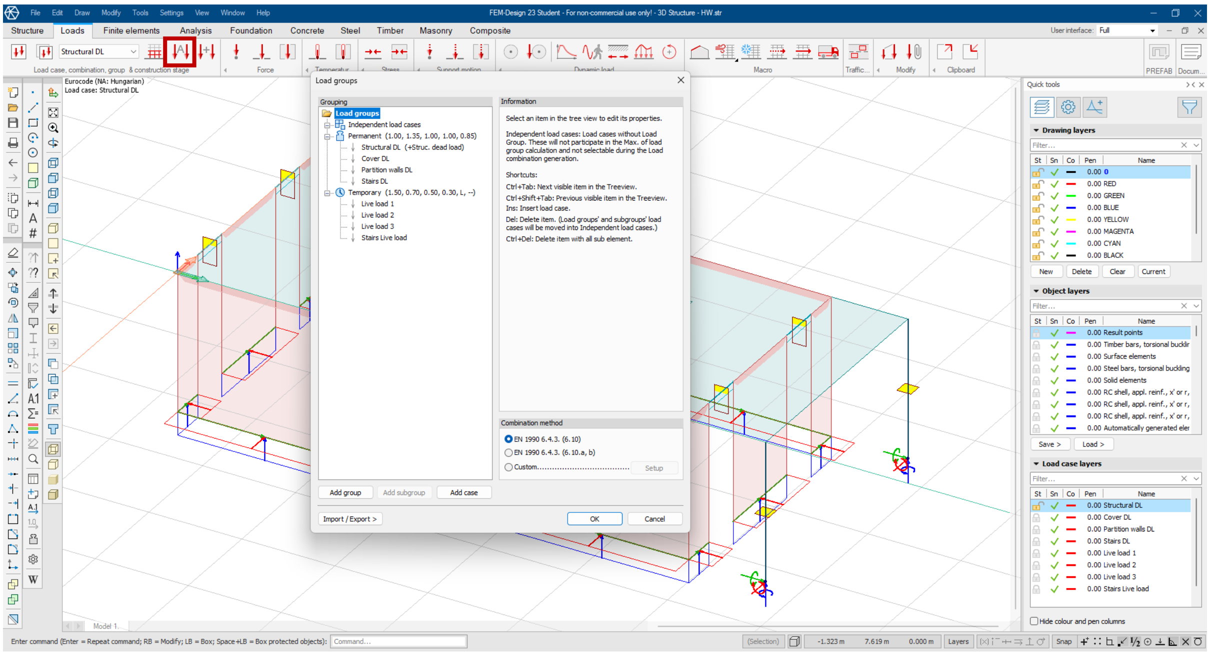
After drawing the geometry, but before starting to apply loads to the model, it is advisable to perform dimensioning. This will help ensure that the model is accurate. It is important to pay attention to any discrepancies between the architectural and structural plans.



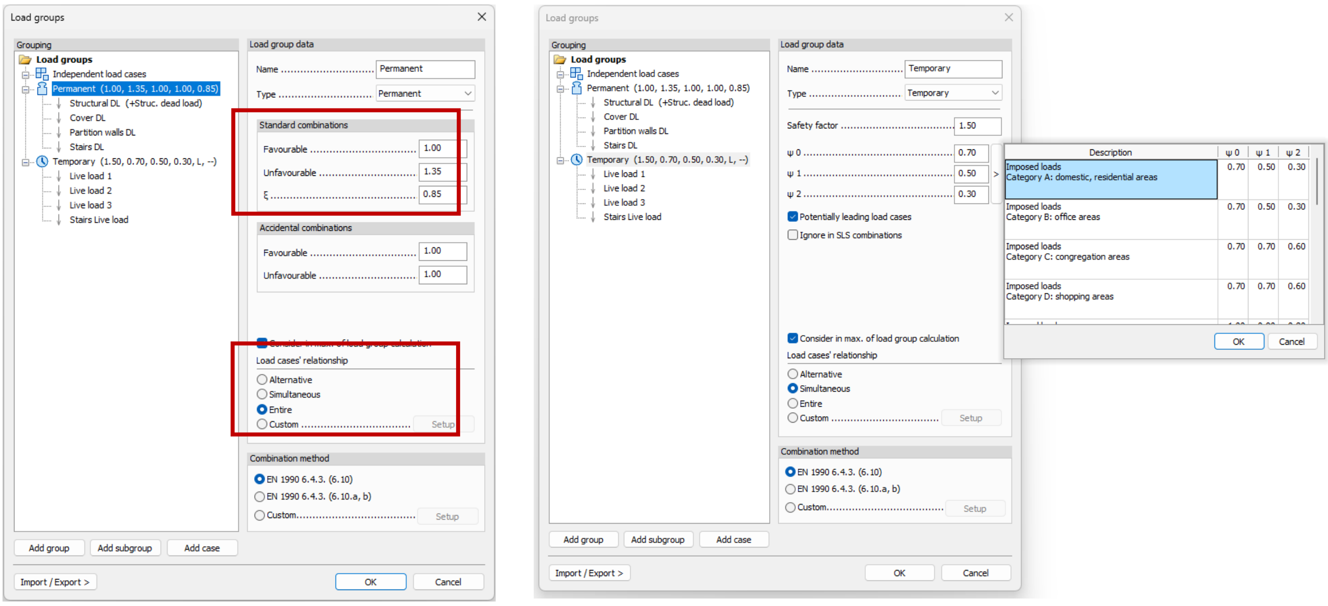
19. Figure: Cota and physical view

4.2 Modeling Loads

Within the "Loads" tab, we begin by creating load groups. Special attention must be paid to the relationships between loading cases, as this affects the generated combinations.



20. Figure: Definition of load groups

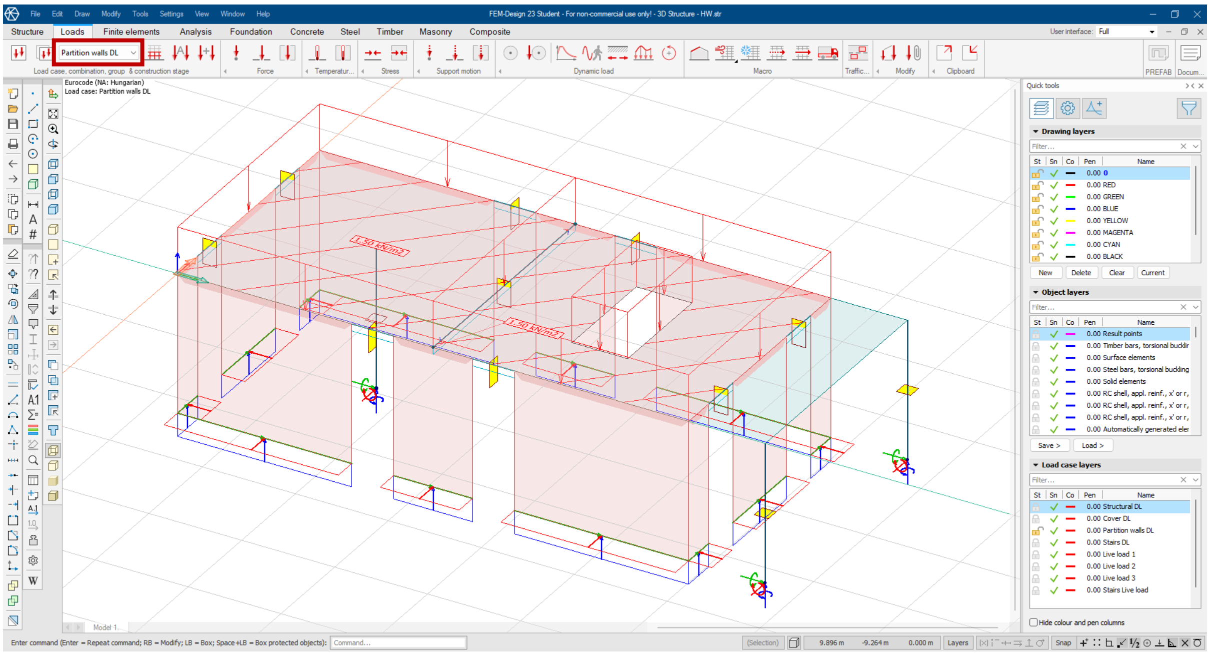


21. Figure: Setting permanent and temporary load groups

Next is the definition of the loads, where it is important to model each load according to the appropriate loading case.

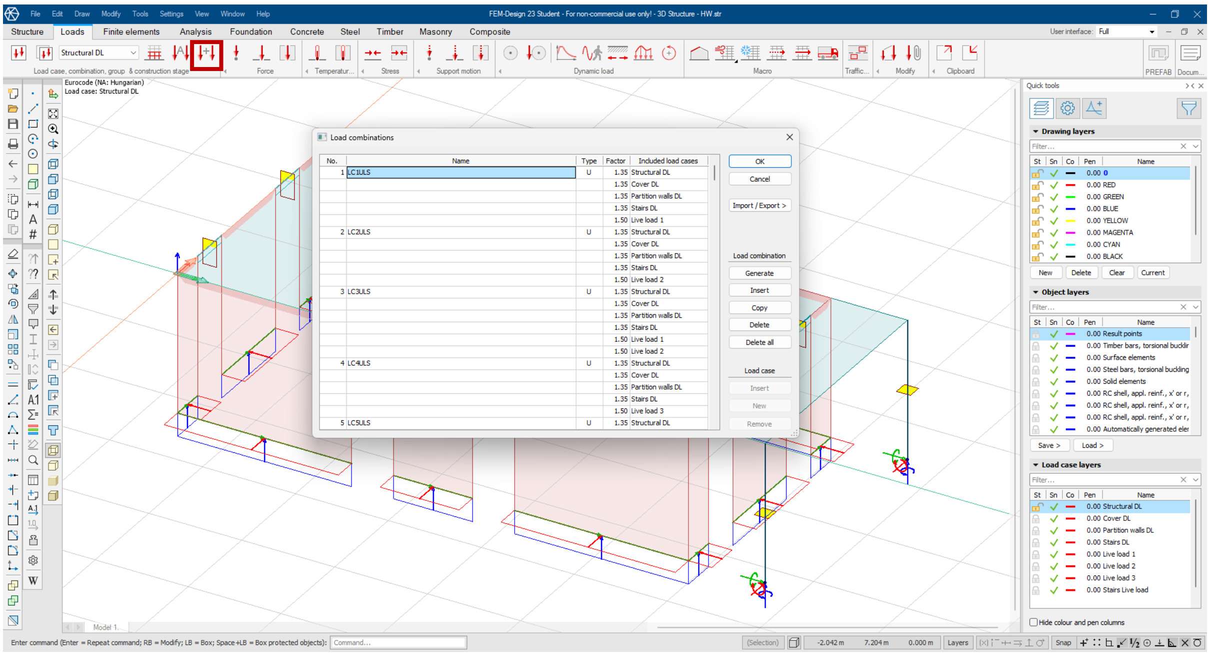
It is important to note that a partition wall is a structural element that can be moved within a given area, so we define it as a distributed load along the surface, except for cantilevers.

The label indicating the load value can be freely repositioned using the move command.



22. Figure: Definition of loads

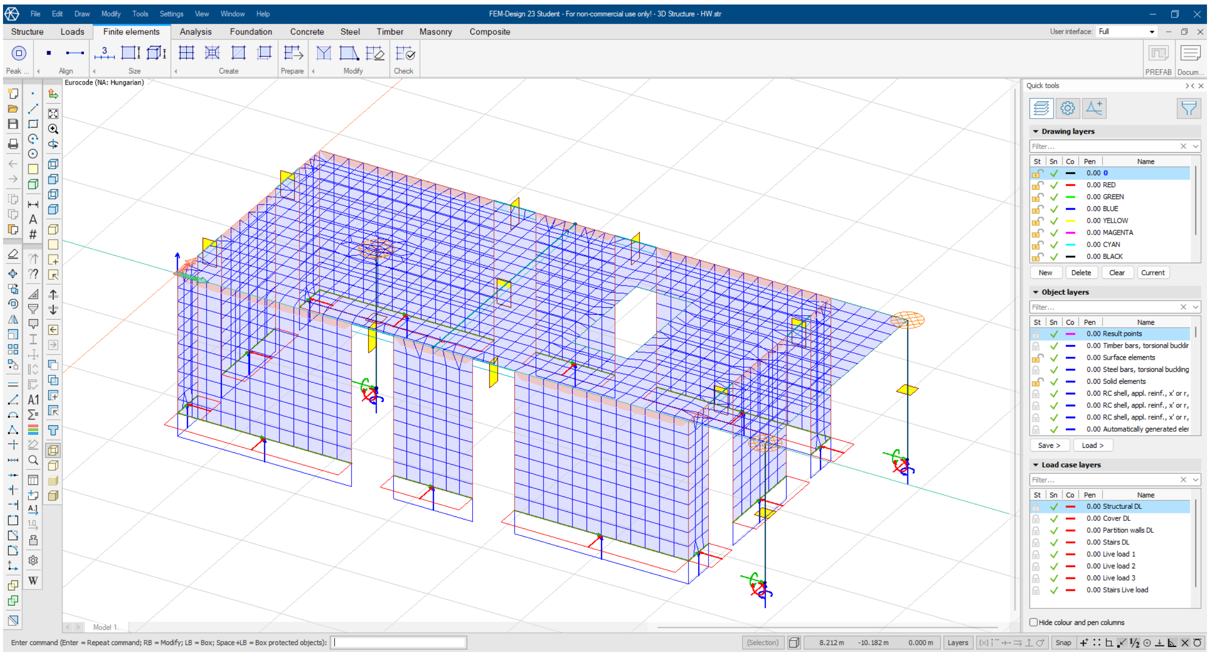
After this, we only need to prepare the load combinations. These can be generated automatically, but they can also be manually adjusted or completely newly created.



23. Figure: Defining combinations of loads

4.3 Finite Elements

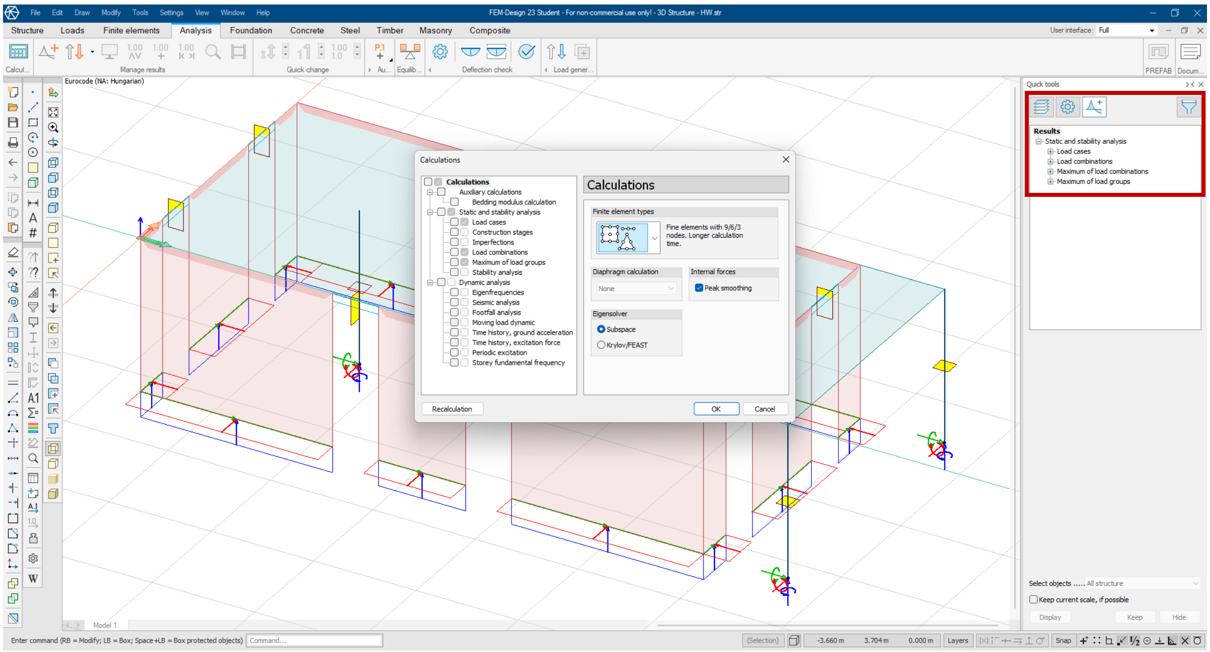
If we skip the finite elements tab, the program will automatically generate the mesh with optimal element size, aiming to create a grid with the appropriate density. We can modify this in various ways if we find it necessary.



24. Figure: Finite element mesh

4.4 Analysis and Results

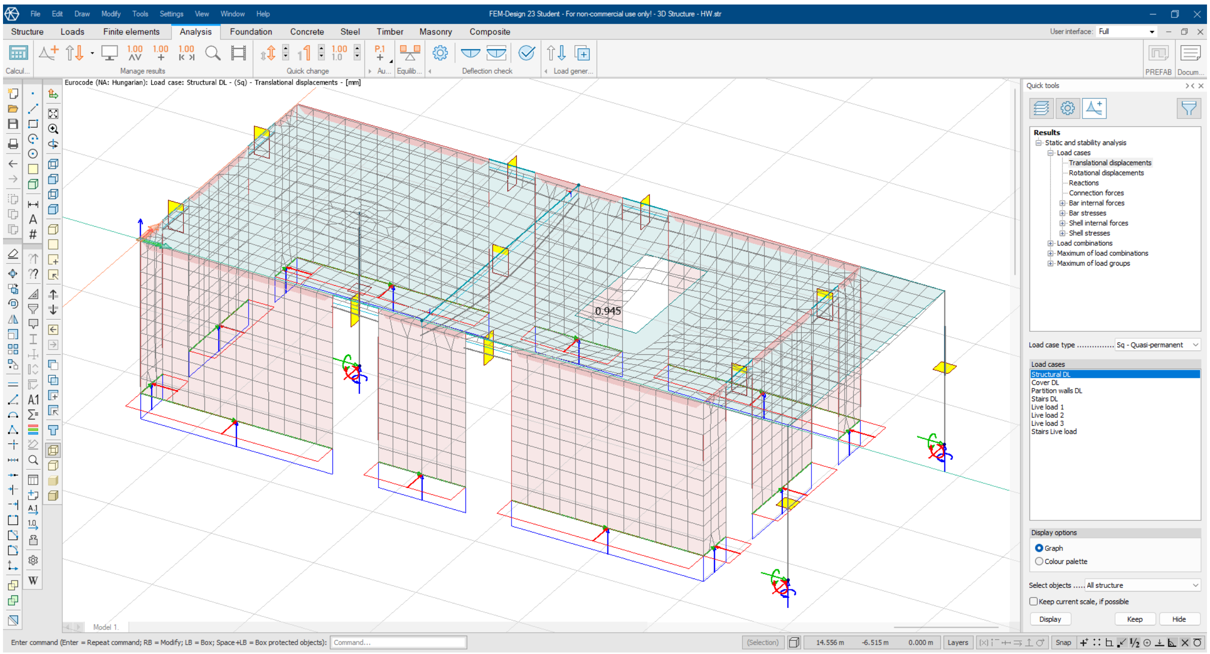
We can see that alongside the selected calculations, the program automatically computes the maximum of the load combinations, provided that we also request the load combinations to be calculated.



25. Figure: Analysis

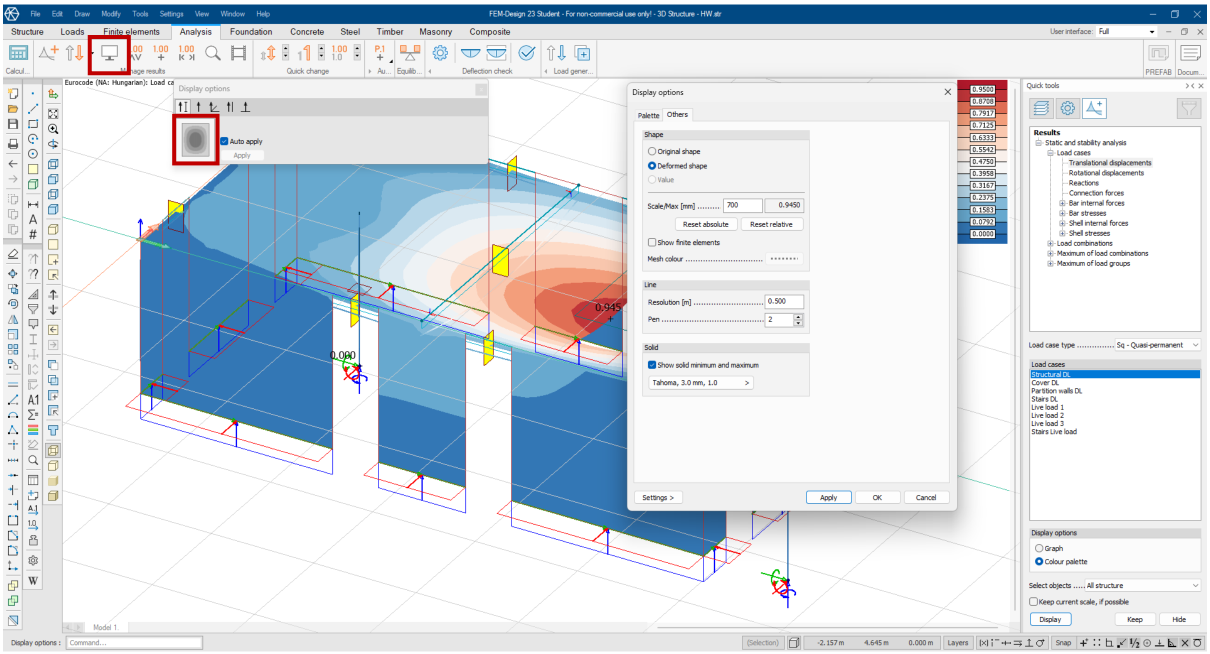
It is important to note that the analysis results can only be viewed within the “Analysis” tab by selecting the last button in the quick menu. This is similar to how the structure can only be modified within the “Structure” tab, and loads can only be adjusted within the “Loads” tab.

For loading cases, always follow this logic: first, determine the type of analysis result you want to display from top to bottom, then select which loading case it should be derived from, and finally, choose the display mode in which you want to visualize it.



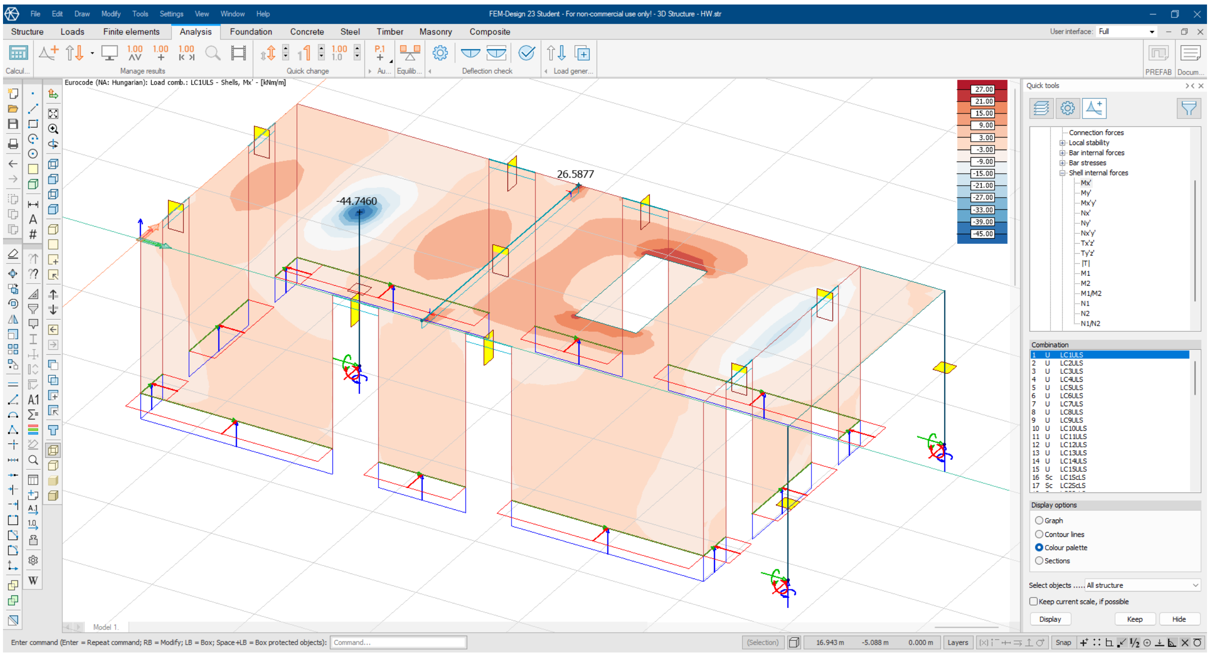
26. Figure: Results from load cases

The following image shows a color palette display with the deformed shape visualized.



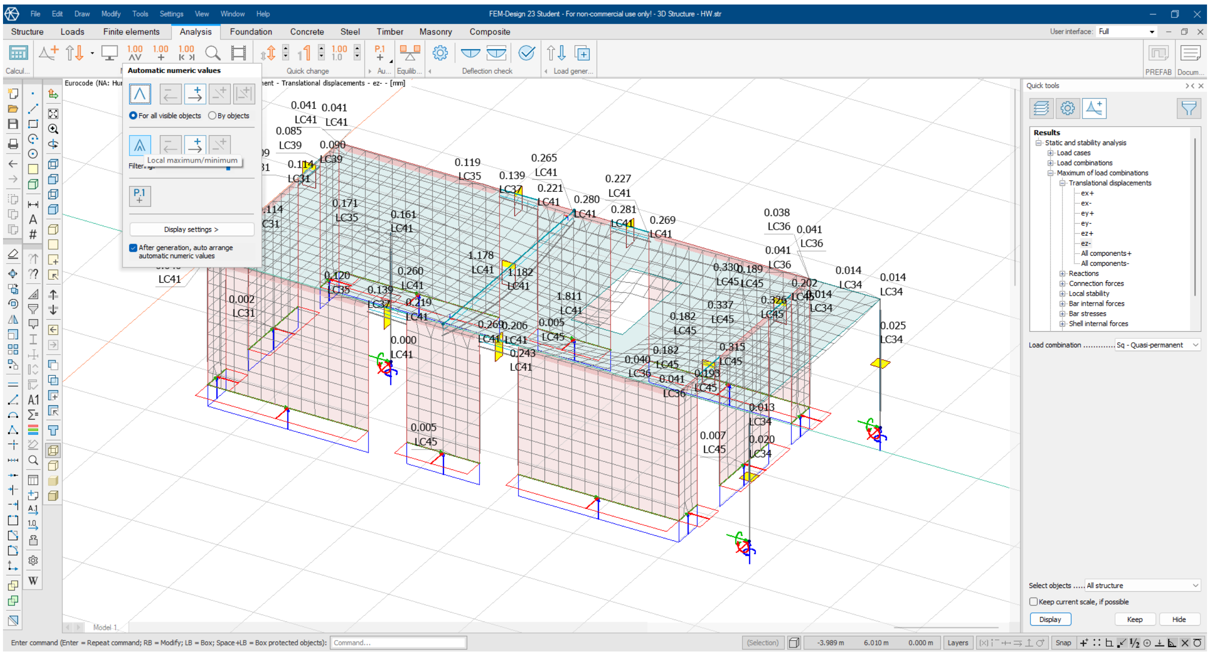
27. Figure: Display on deformed shape

For load combinations, the logic is very similar, with the slight difference that instead of selecting a loading case, you choose the desired load combination.



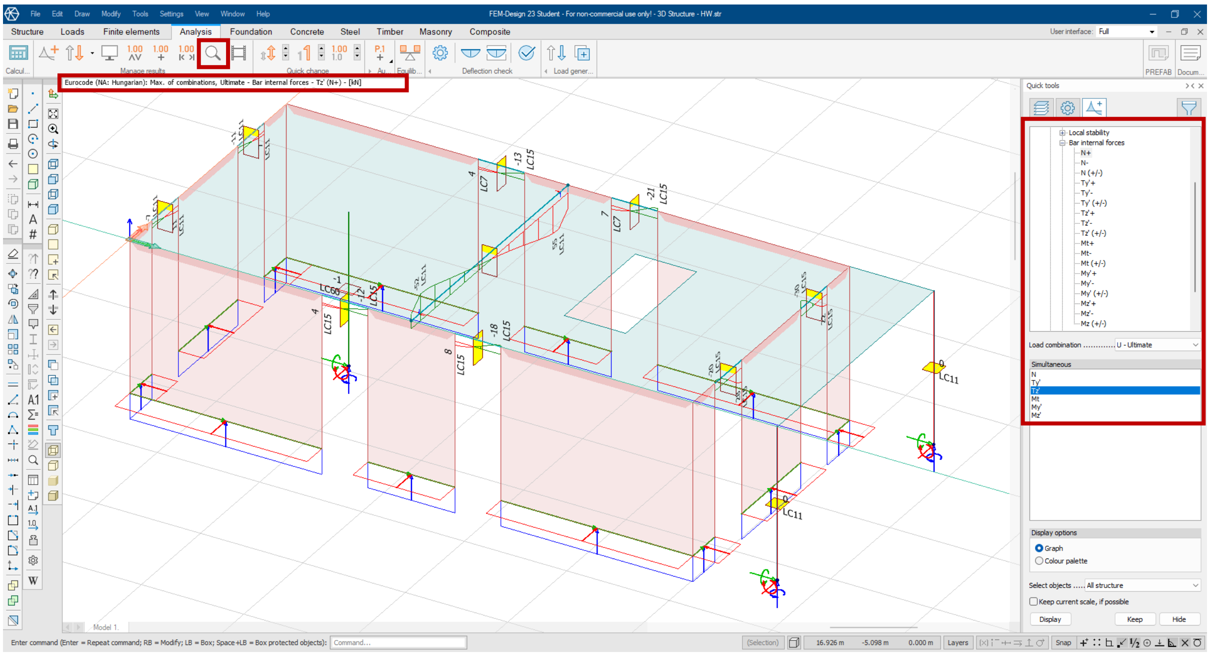
28. Figure: Results from load combinations

For the maximum of load combinations, several options are available. For example, in the case of displacements, you can select both the direction and the orientation. In this scenario, you cannot display the color palette on the deformed shape, as multiple combination results are shown, as illustrated in the image. If you are interested in the extreme values, you simply need to enable the Global/min/max option.



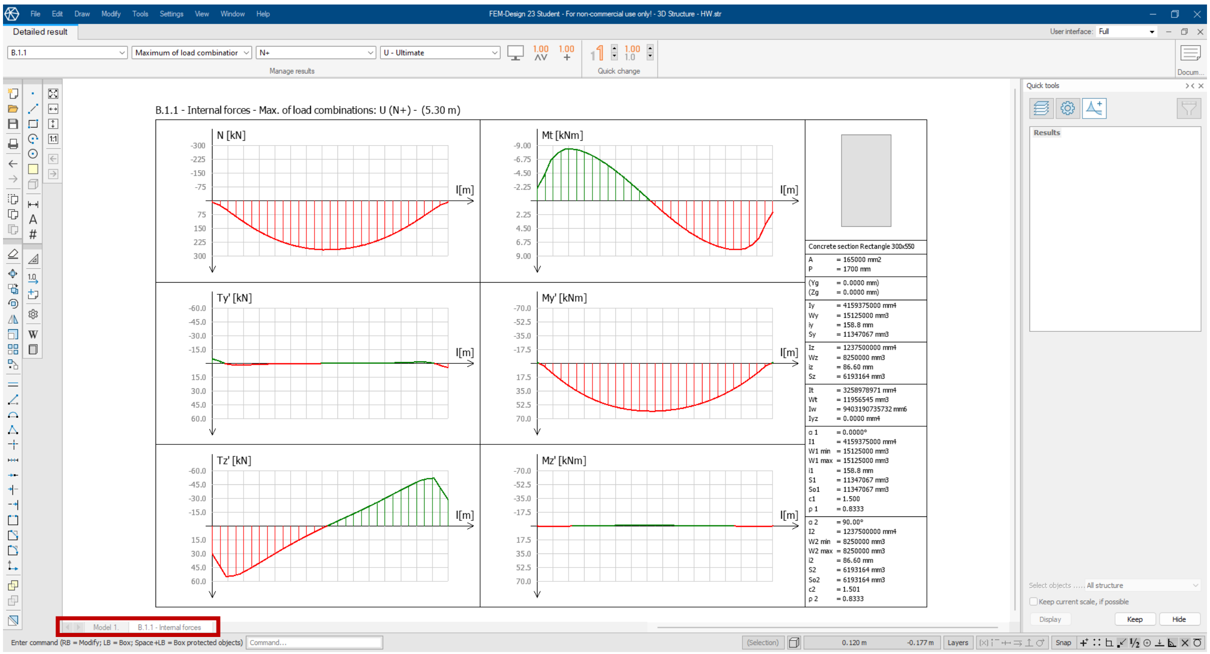
29. Figure: Displacement from maximum of z- load combinations

In the case of utilizations, when dealing with the maximum of load combinations, we need to select the so-called simultaneity. This allows us to read off the shear force from the graph that occurs when the normal force is at its maximum, as illustrated in the following example.



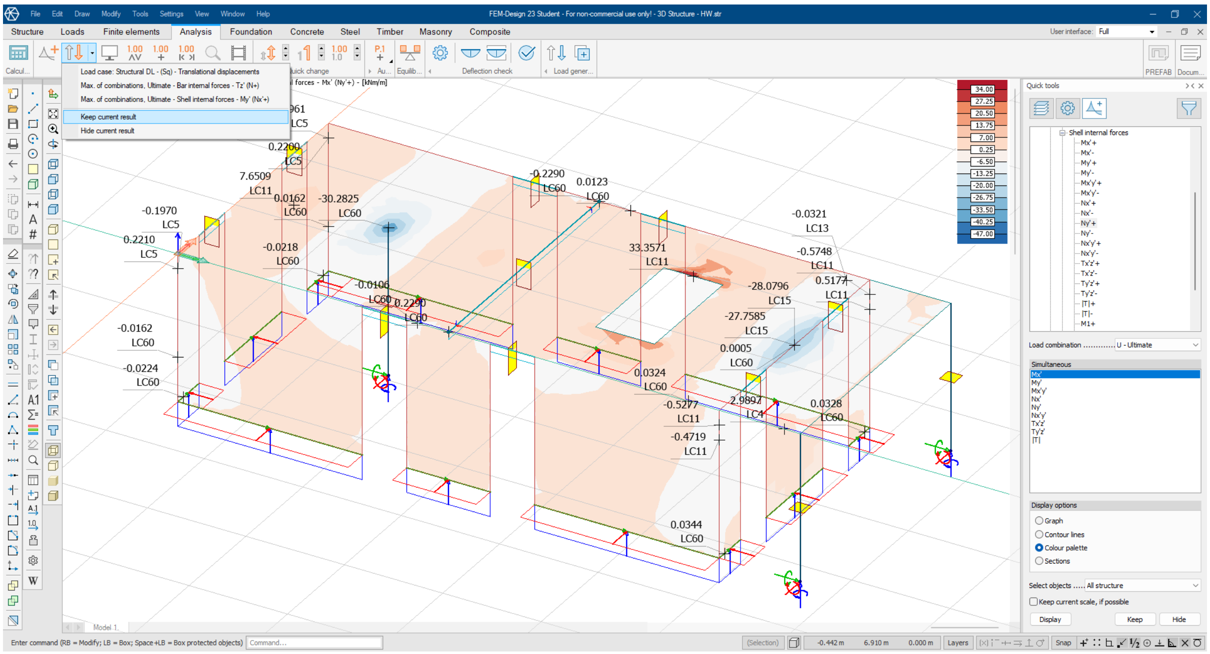
30. Figure: Internal forces from maximum load combinations

By clicking on the magnifying glass icon, we can display detailed results for individual elements. We can switch between the windows with a single click.



31. Figure: Detailed results

These utilizations can also be selected in a similar manner within the Documentation tab, but they can even be saved directly in the modeling space, allowing for just a single click to display the desired utilization diagram.

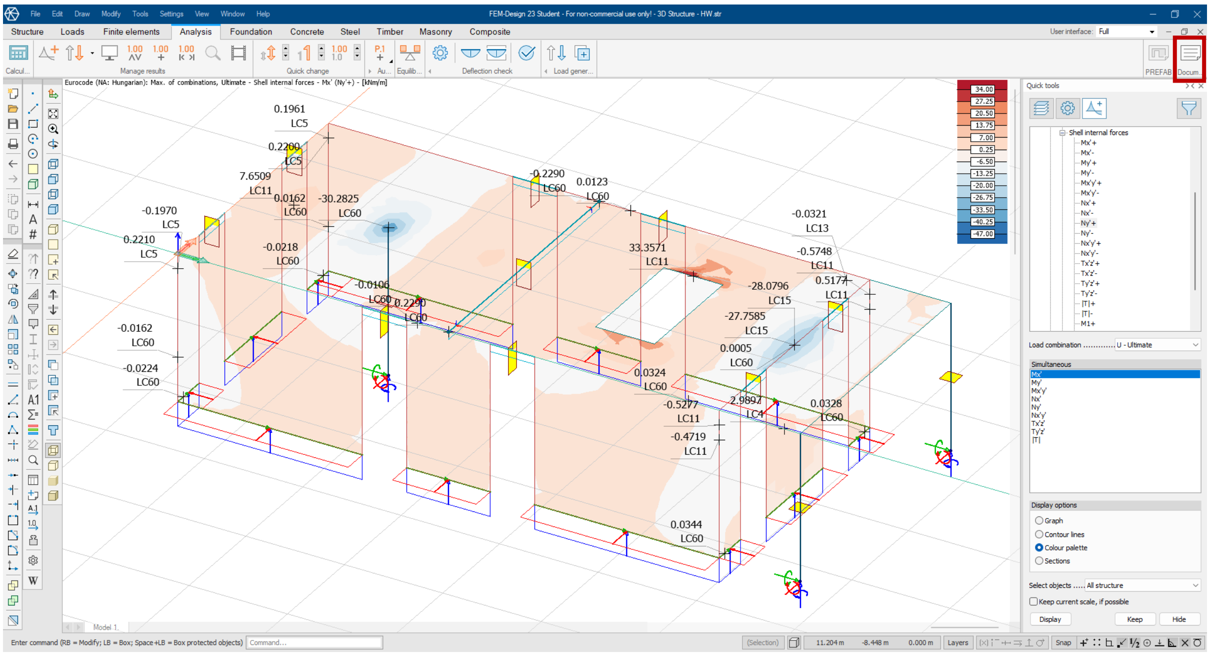


32. Figure: Saving and loading results

If, during the homework, only a few load combinations were created, rather than working with the entire generated quantity, it may be useful to work with the maximum of the load groups.

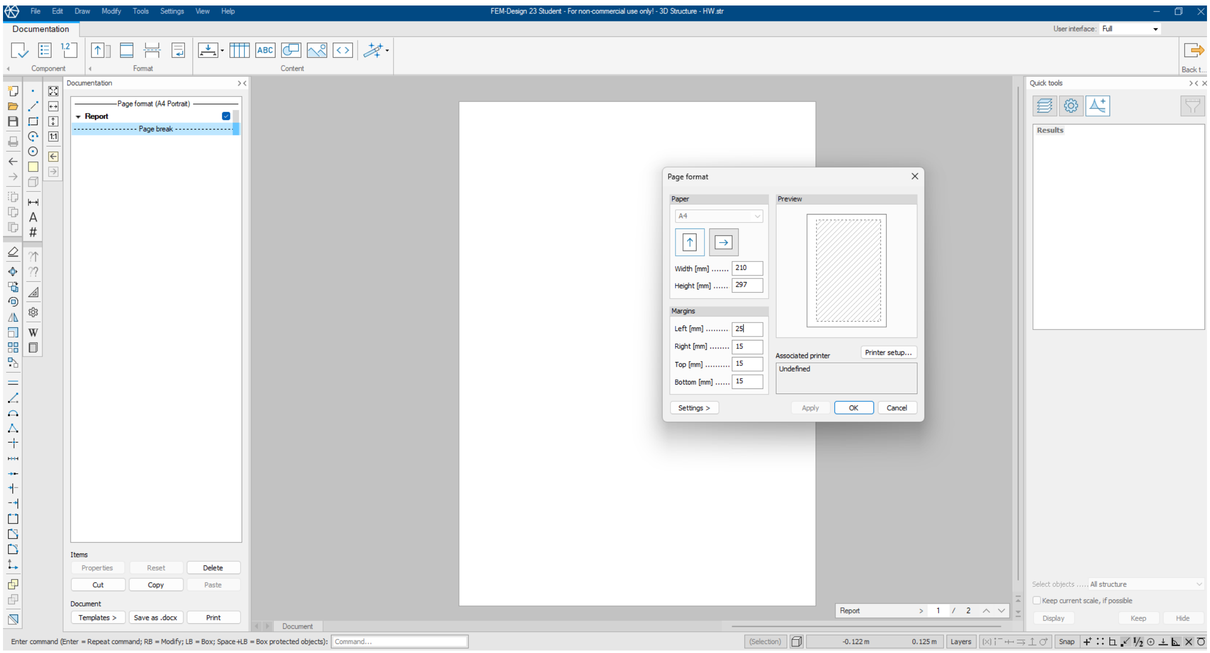
5. Preparing Documentation

It is advisable to start preparing the documentation when the model is fully completed. However, if changes are made after the documentation has been prepared, the program will automatically update the documentation in the live windows as well.

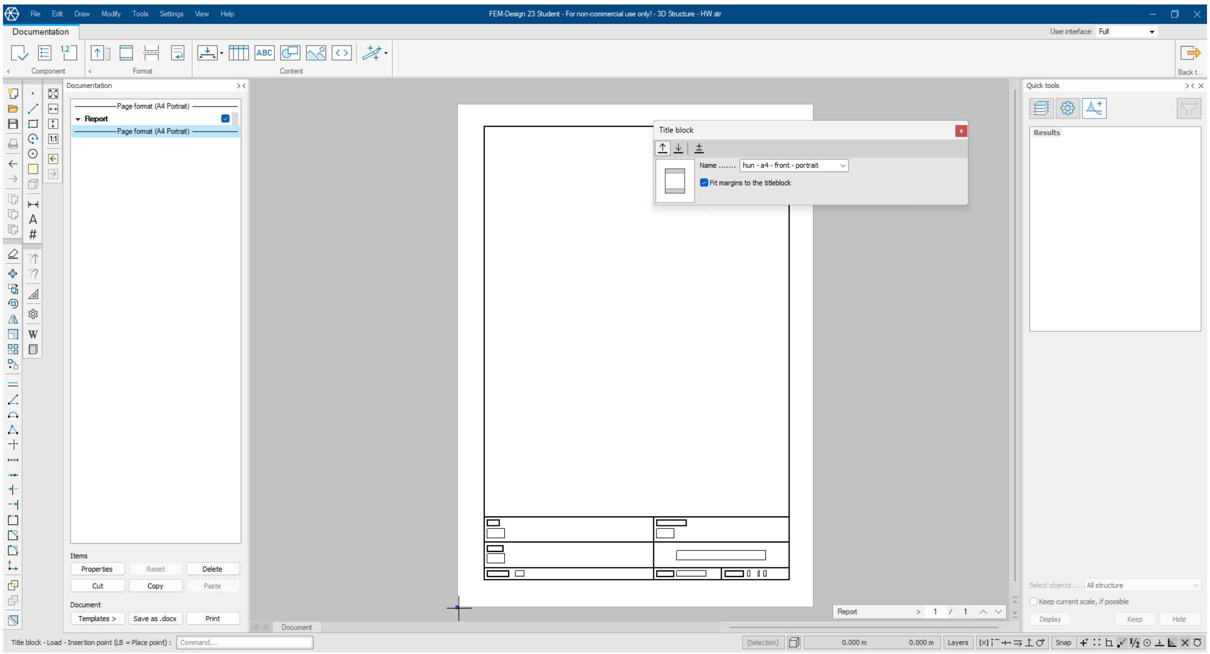


33. *Figure: Open documentation*

After setting the page format, the cover stamp should be placed on the first page.

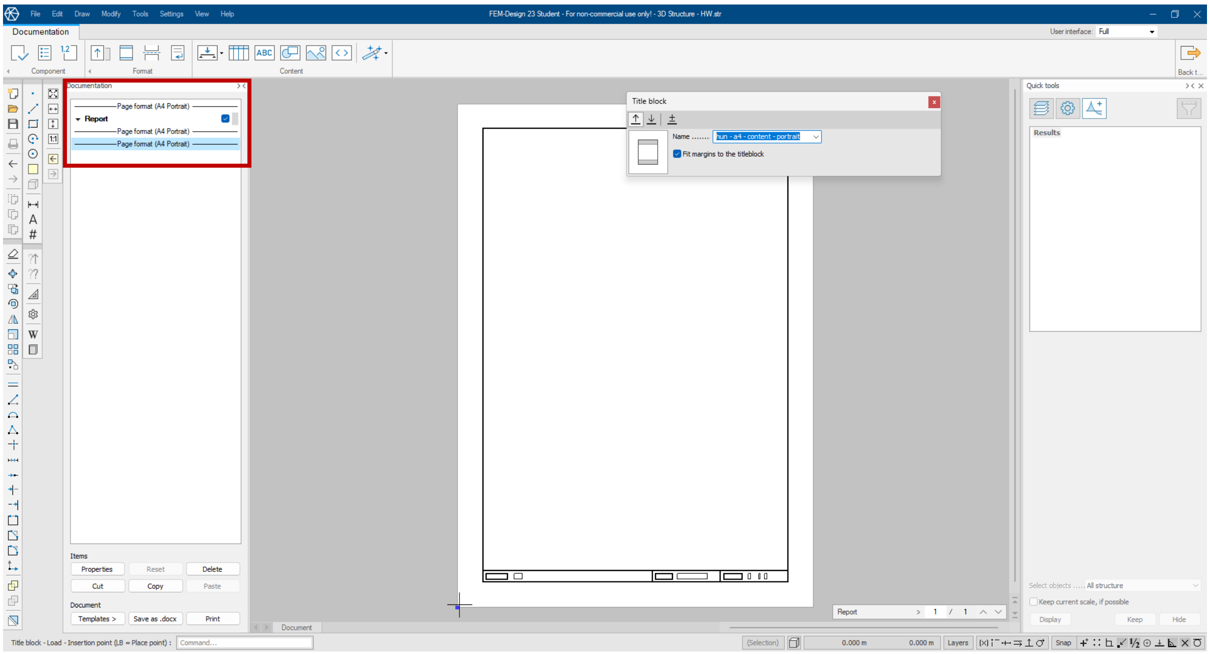


34. *Figure: Page format*



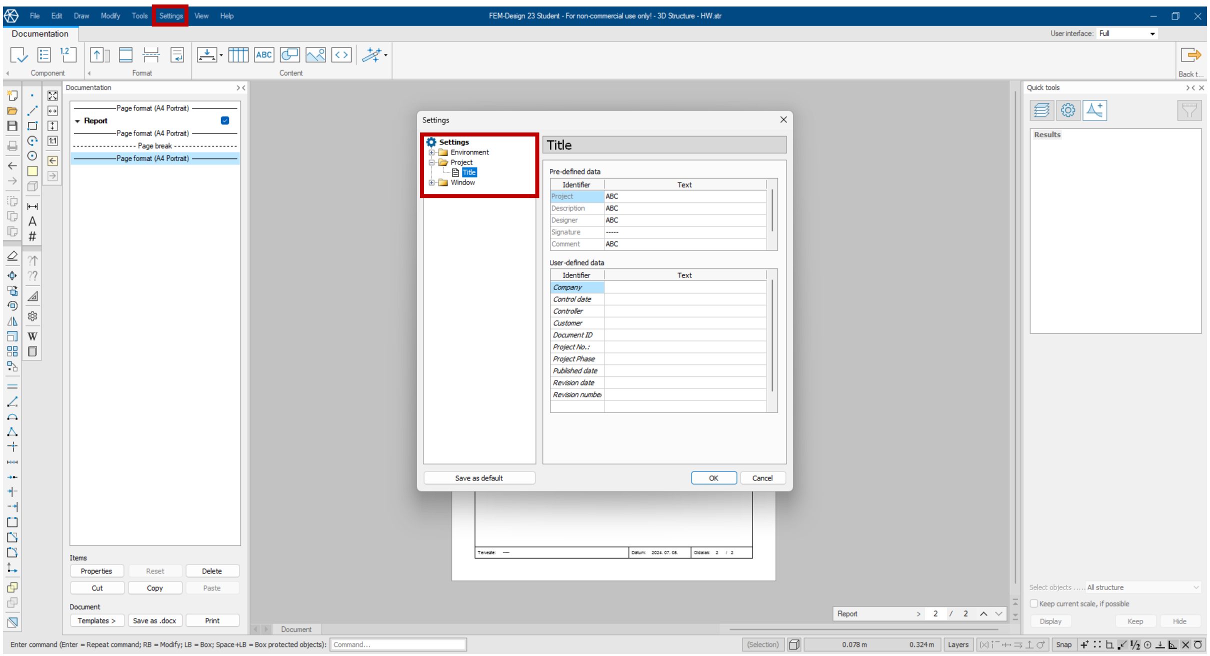
35. Figure: Title Block

If we want the cover page to differ from the other pages, we can achieve this by inserting a page break and then repeating the previous steps with a different stamp on the new page. After setting the page format and placing the stamp, the following pages will display the same stamp as before.



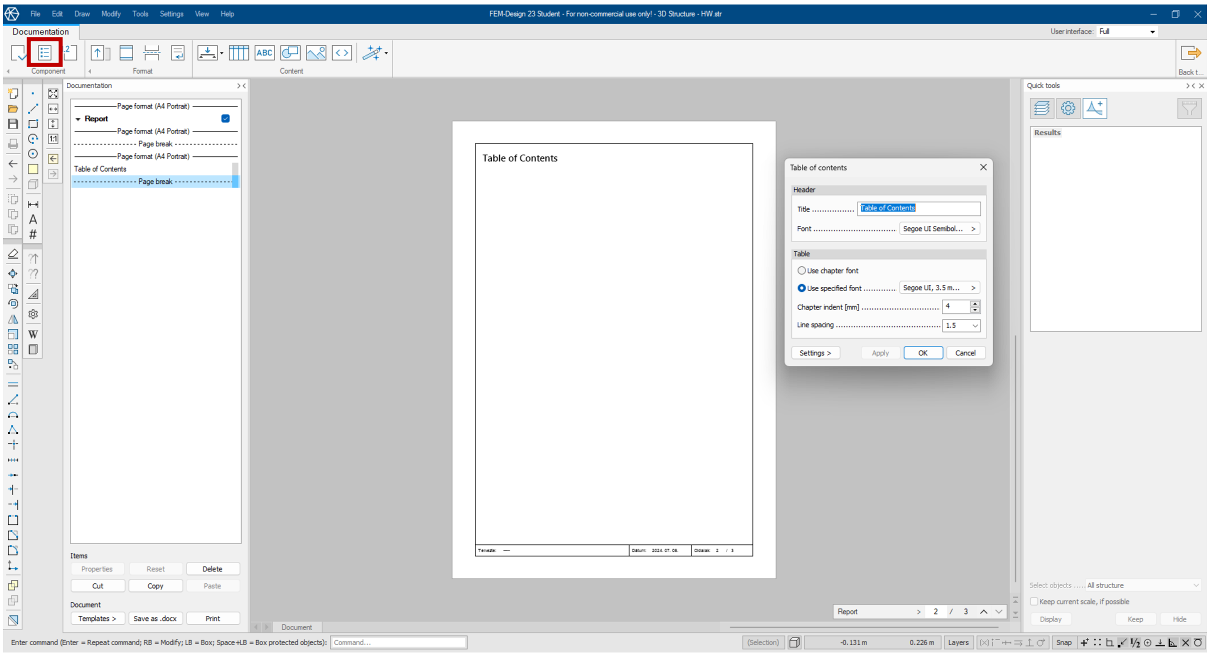
36. Figure: Set up additional pages

The information on the cover page can be set in the menu under Settings → Project → Title.

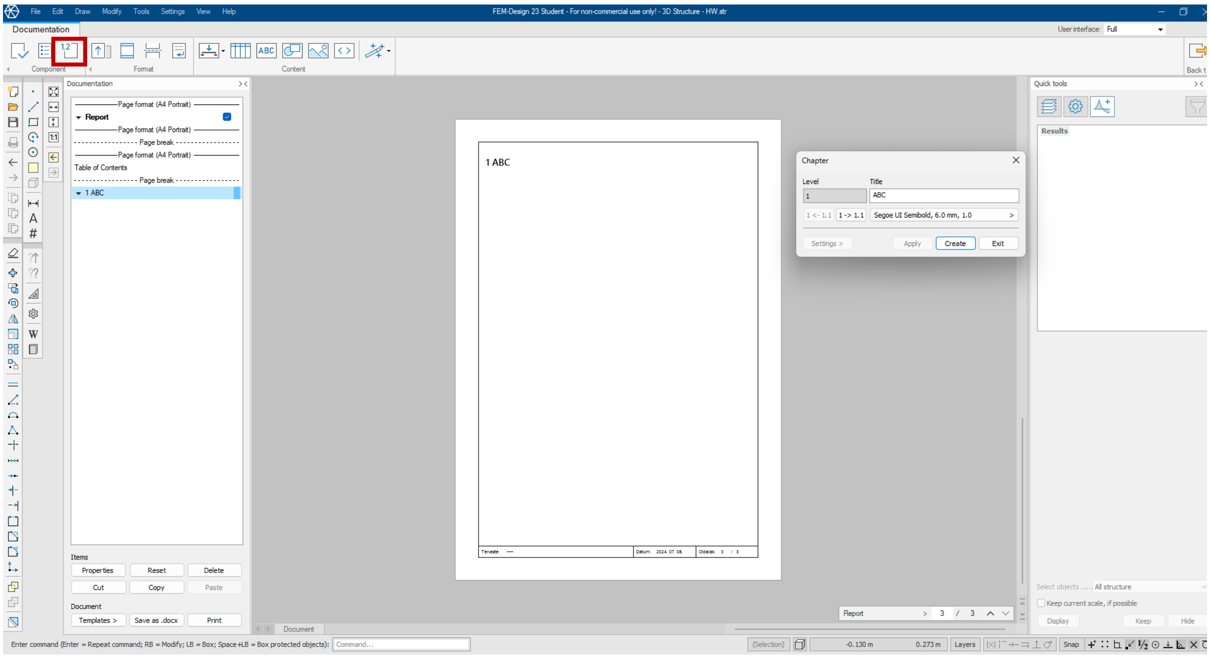


37. Figure: Upload your data

After the page break following the table of contents, we can start adding information to the documentation. It is advisable to organize this information into sections.

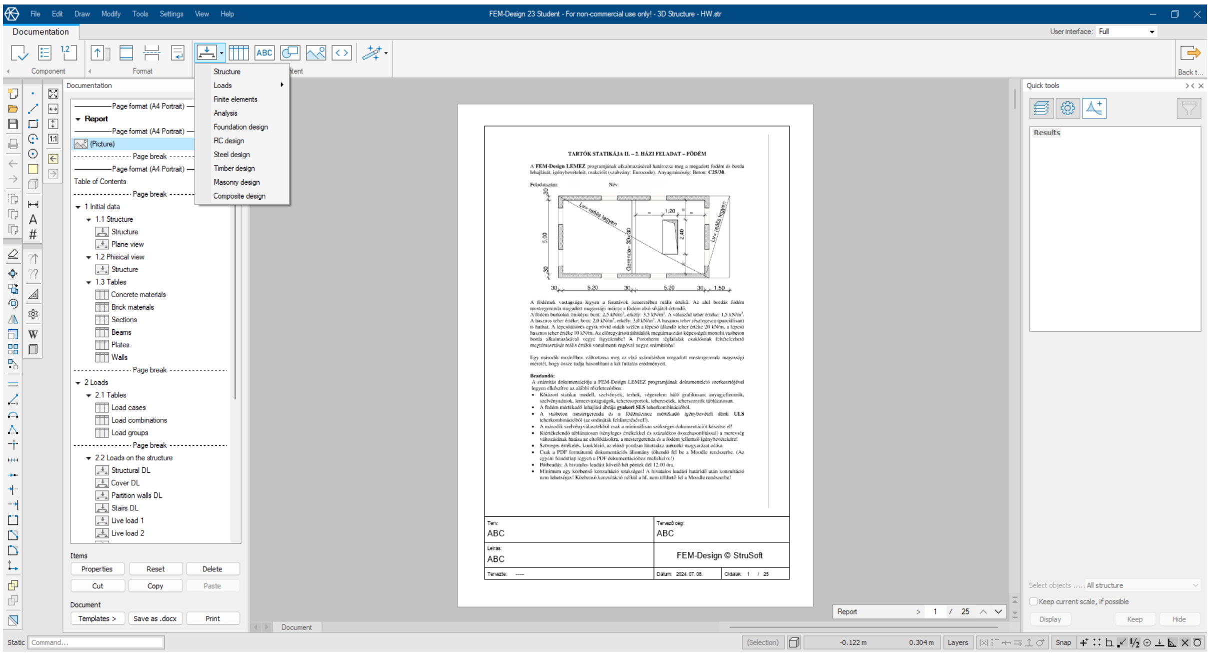


38. Figure: Table of contents



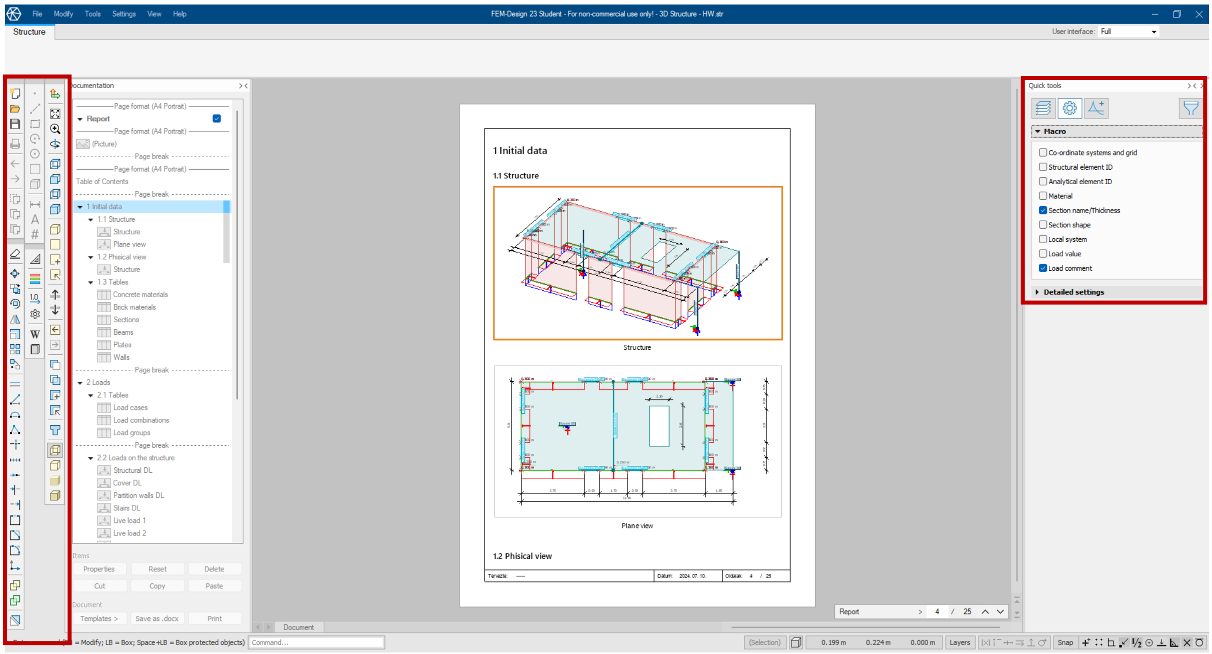
39. Figure: Chapter title

Placing live windows is extremely simple; you just need to select the type of window you want to display. Make sure to choose the appropriate window type for the desired display: select a structure window for showing the structure, a load window for loads, and a result window for displaying results.



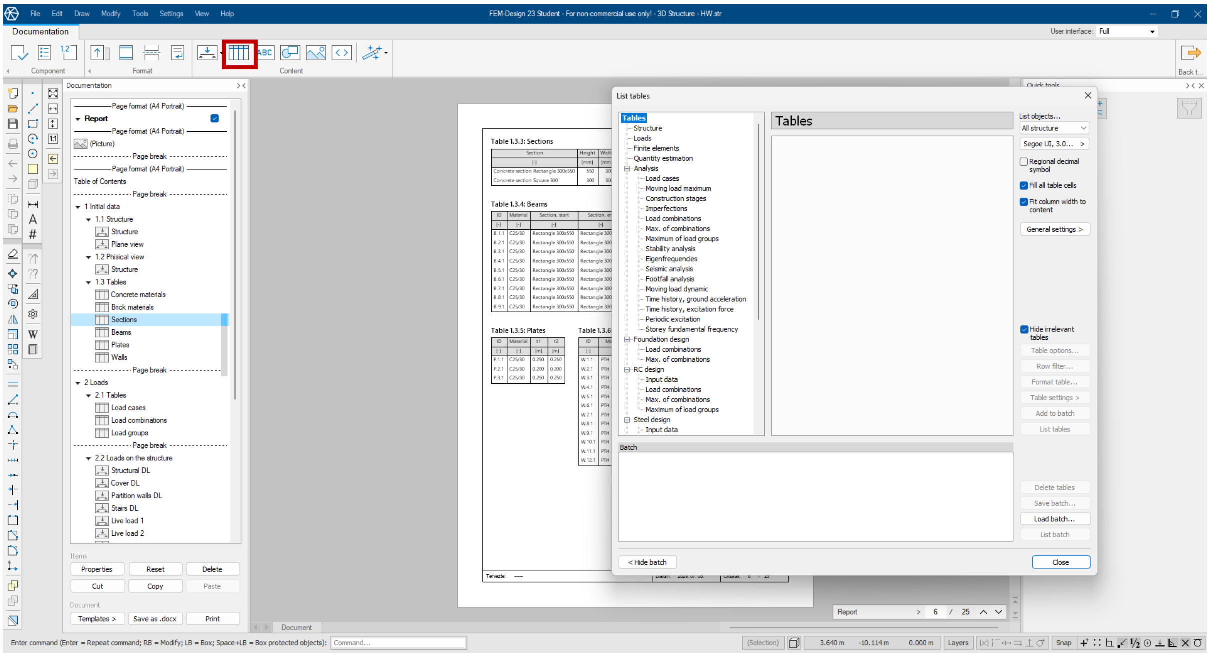
40. Figure: Inserting pictures

The settings for a given window can be managed similarly to the modeling space. These settings become available when we click on the window, indicated by the orange frame around it.



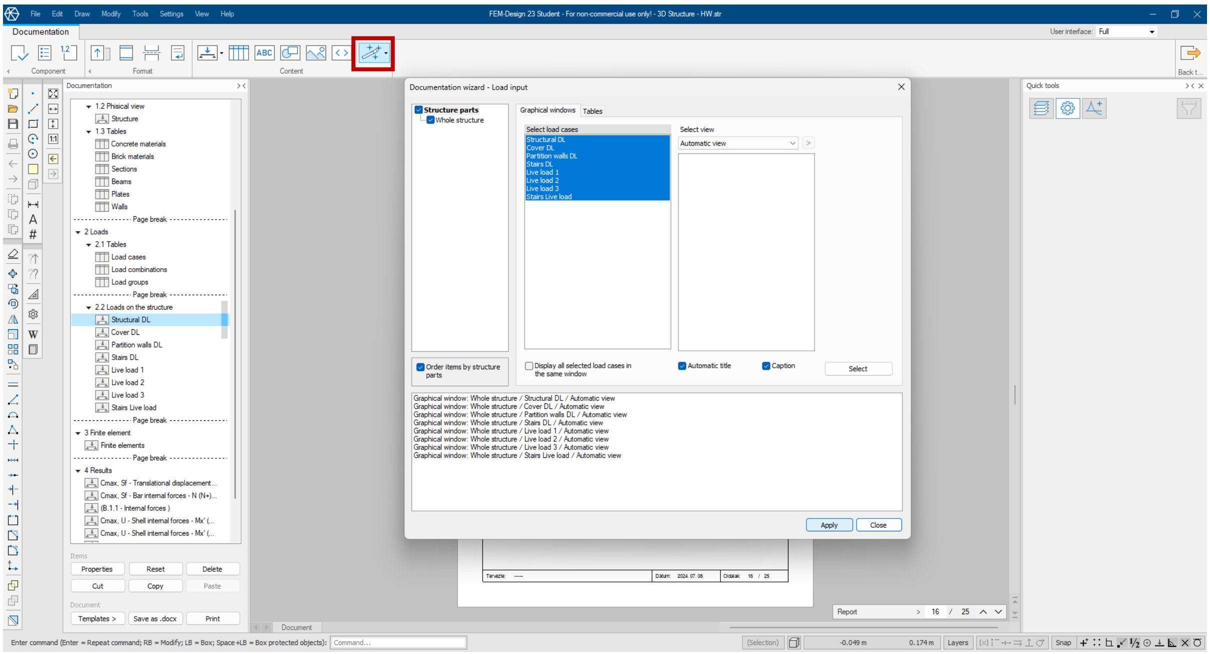
41. Figure: Settings

You can also easily insert a table using the “Table” icon.



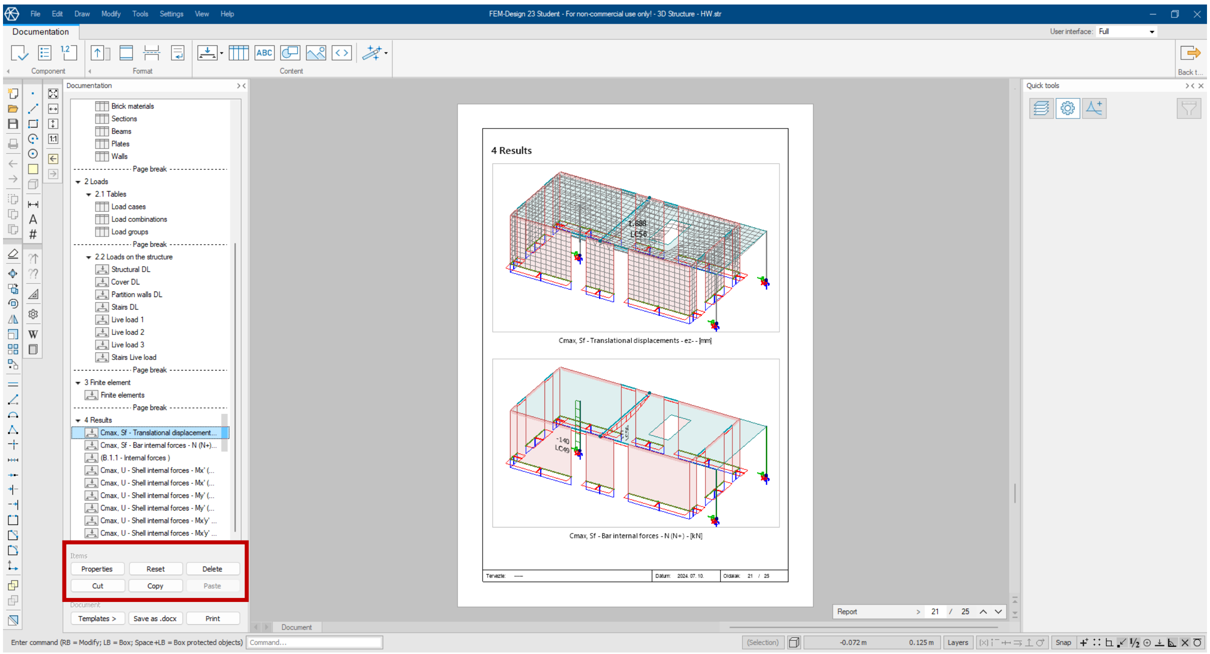
42. Figure: Insert tables

With the “Wizard” function, you can insert certain types of windows into the documentation with just a few clicks. As shown in the following illustration, by selecting the desired loads, they will be added to the documentation simultaneously. This avoids the need to add schematic parts individually.



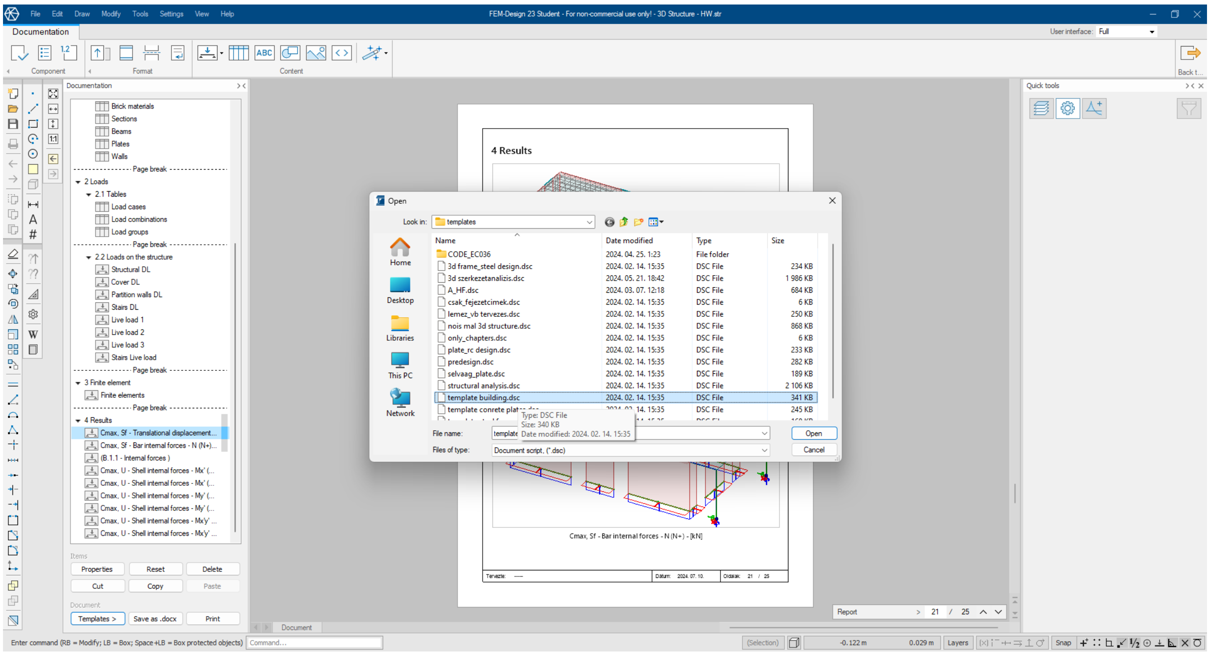
43. Figure: Wizard function

It is important to note that elements in the documentation are editable. Additionally, they can be moved using the “drag and drop” function.



44. Figure: Editing

One of the most useful tools in FEM-Design is the documentation “Templates.” With this feature, you can load previously saved templates from the library, or create your own, so that schematic tasks do not need to be documented every time. This can be particularly useful when preparing two different models for a homework assignment.



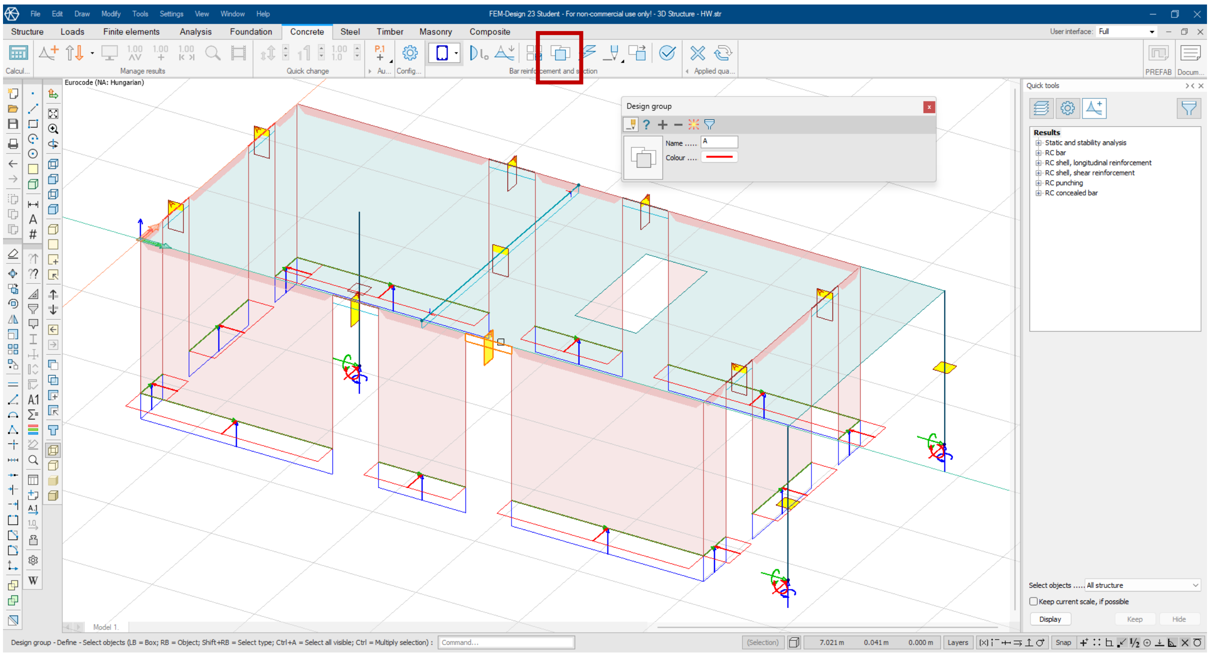
45. Figure:Documentation templates

6. Design and Verification Options

6.1 Reinforced Concrete Design

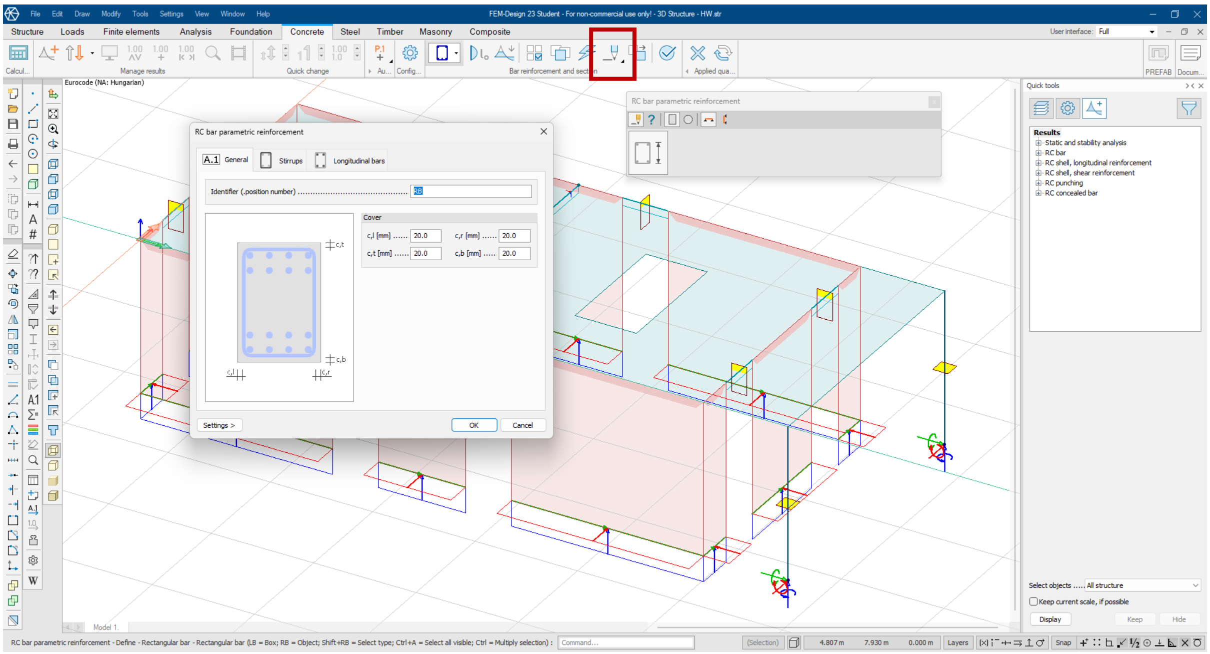
The program offers various design possibilities. In this case, we will demonstrate the design of a reinforced concrete slab.

Under the "Reinforced Concrete" tab, we start with the reinforcement of beams, where we create design groups. The purpose of these design groups is to ensure that the reinforcement within these groups is consistent during the design process. These design groups can be labeled with colors and letters.

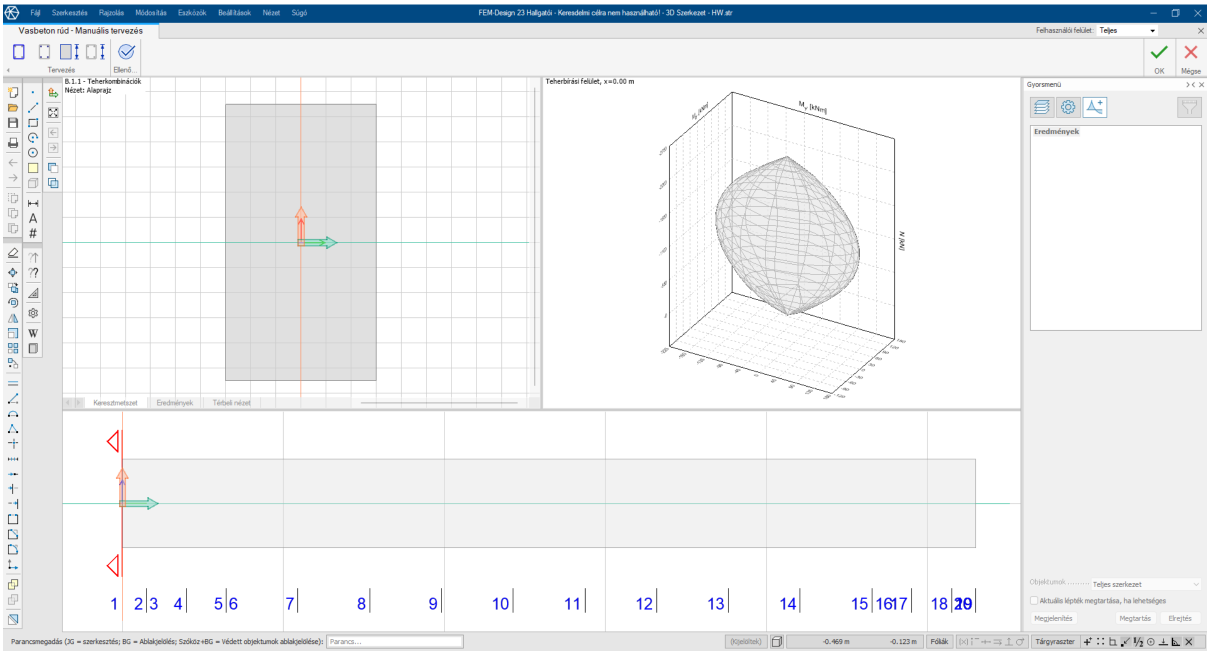


46. Figure: Design groups

During manual design, the reinforcement of the beams can be defined either parametrically or by drawing. In both cases, design groups can be particularly useful.

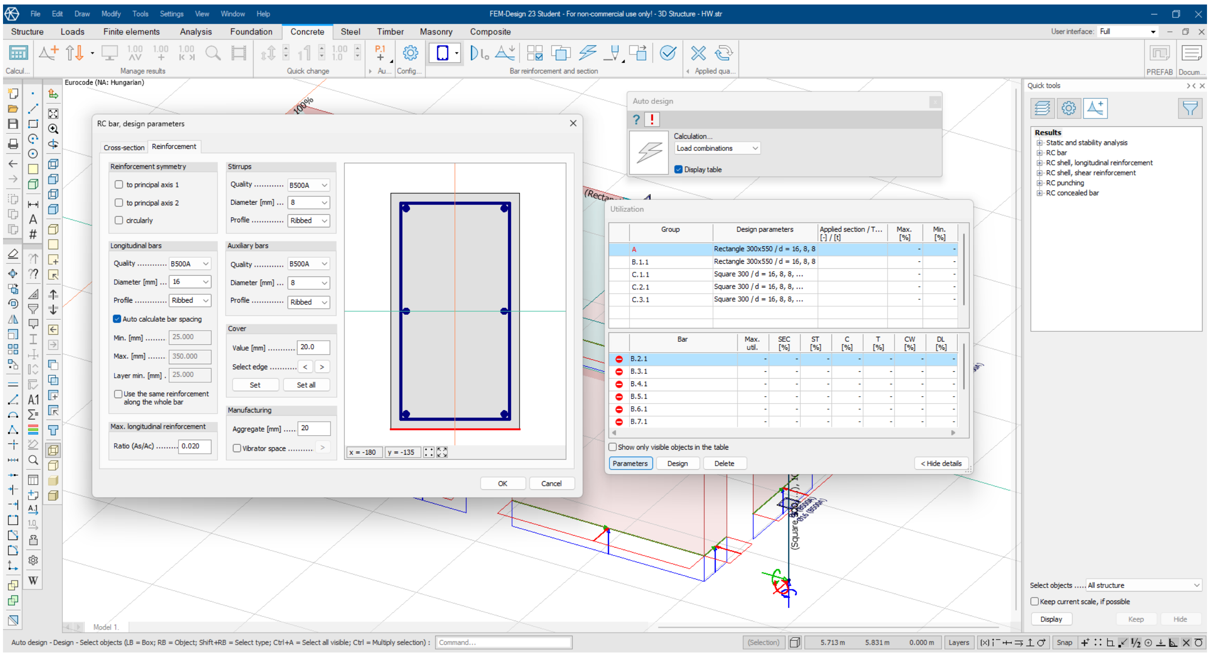


47. *Figure: Parametric reinforcement of bar elements*



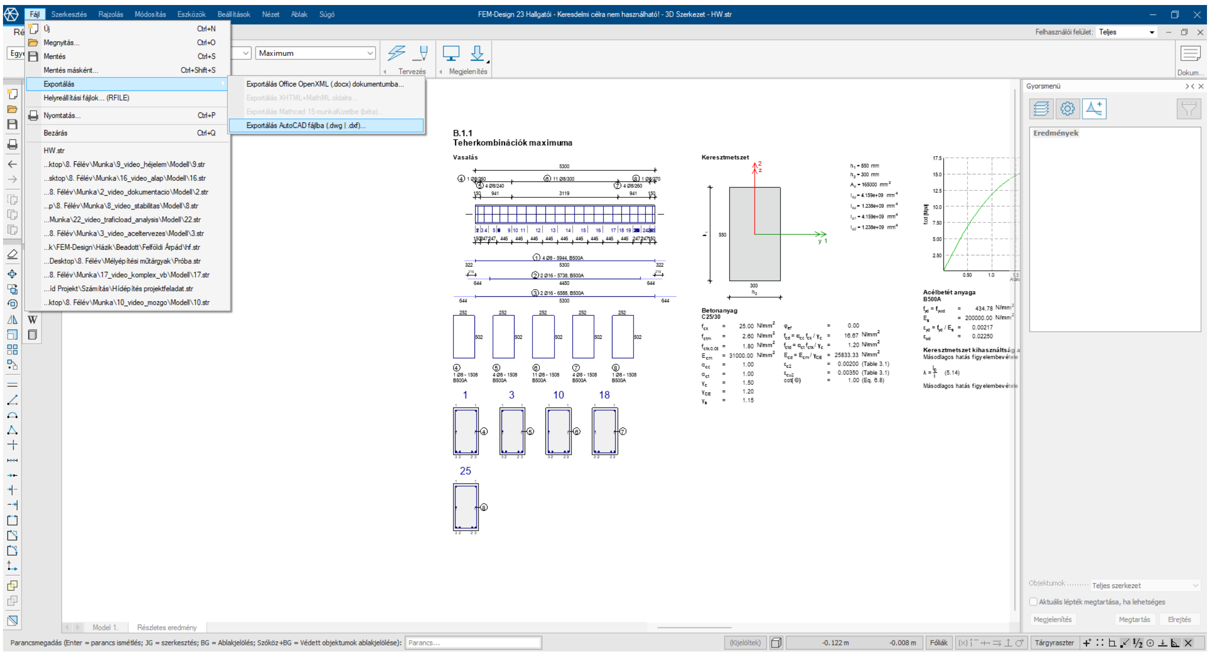
48. *Figure: Ironing of bar elements by assembly*

During automatic design, both free and constrained design options are available. This allows us not only to specify the properties of the reinforcement but also to define the cross-sectional shapes and dimensions.



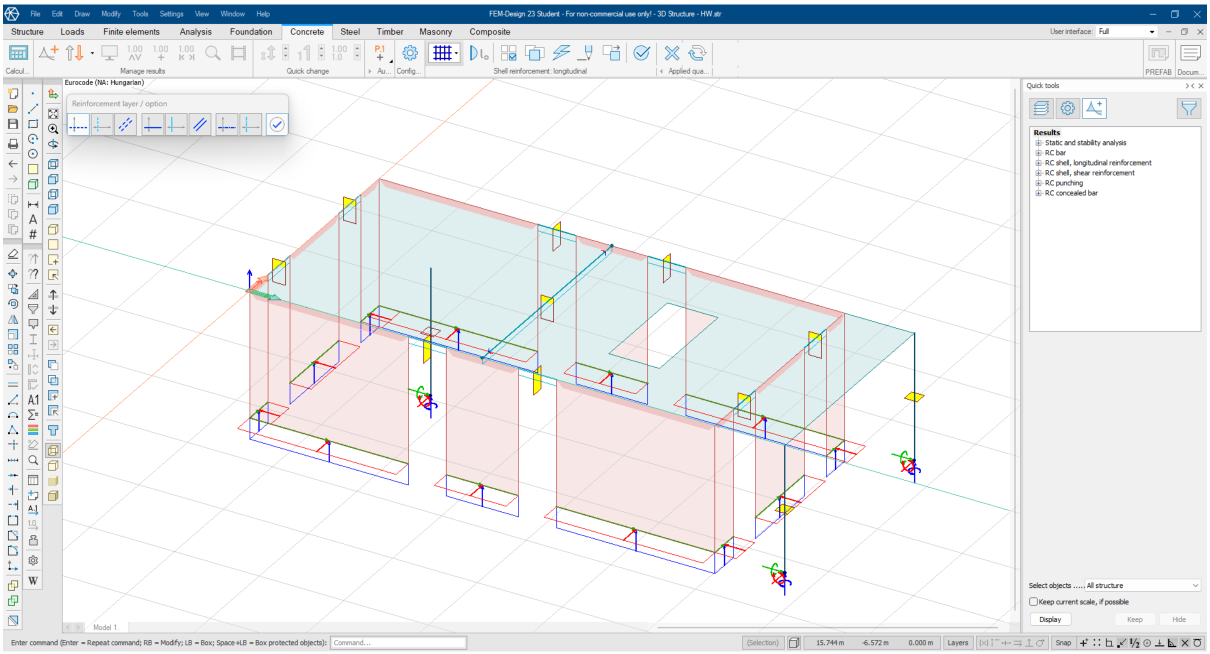
49. Figure: Automatic design of bar elements

One of the most useful features of the program is the detailed results display, which, in this case, presents the entire calculation for the reinforcement and the reinforced concrete beam, along with the reinforcement drawings. This can be used to verify manual calculations and is also suitable for creating reinforcement sketches, as the document can be exported in DWG and DXF formats.



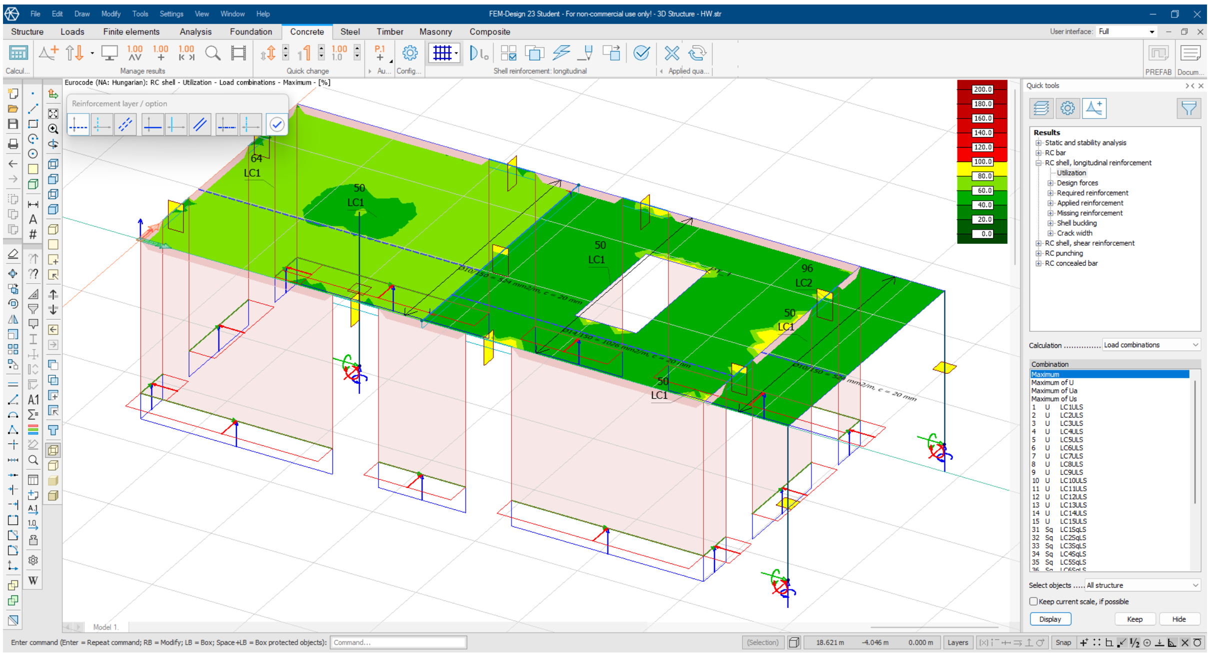
50. Figure: RC Bar element detailed results

When designing surface elements, it's possible to plan not only longitudinal reinforcement but also shear and punching reinforcement. In these cases, you can choose between manual and automatic design options, or even use a combination of both.



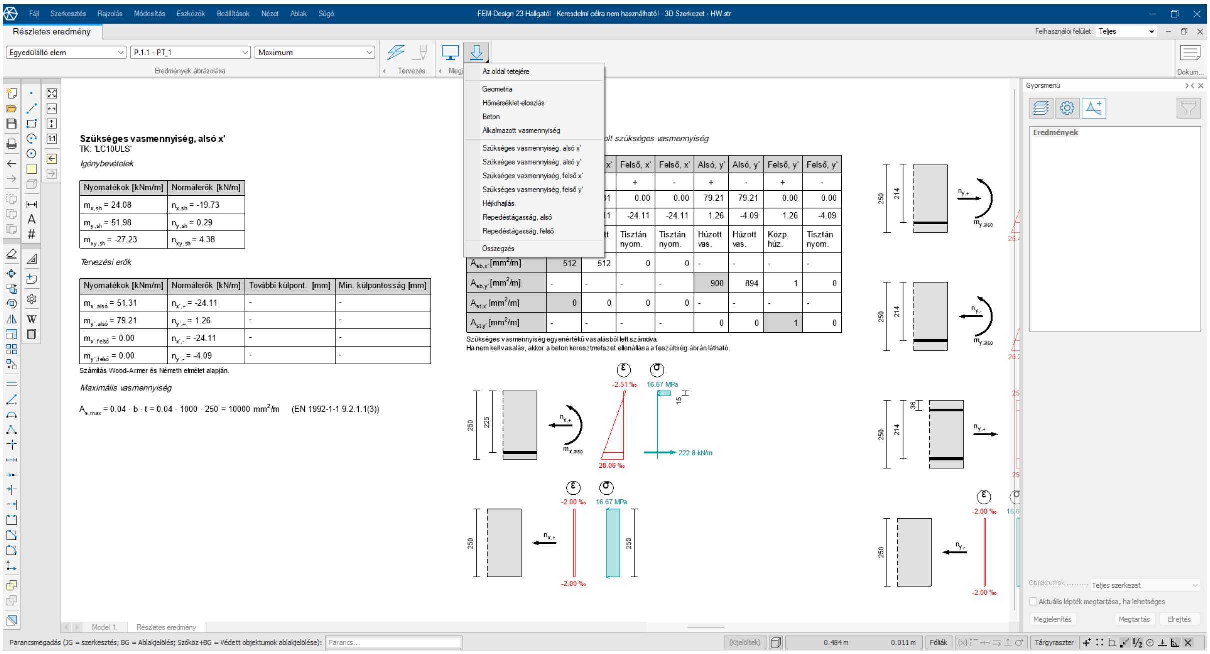
51. Figure:Longitudinal reinforcement of surface elements

For each type of reinforcement, we can also view its utilization, along with other useful values.



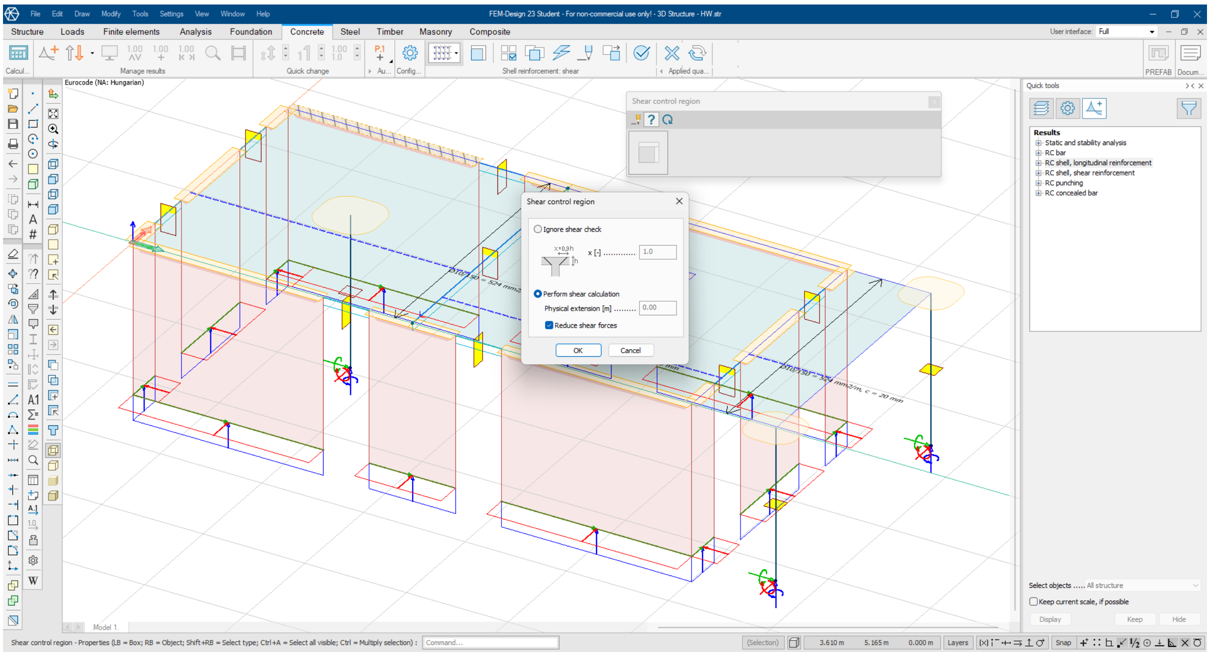
52. Figure: Utilisation

Detailed results are also available for slabs.



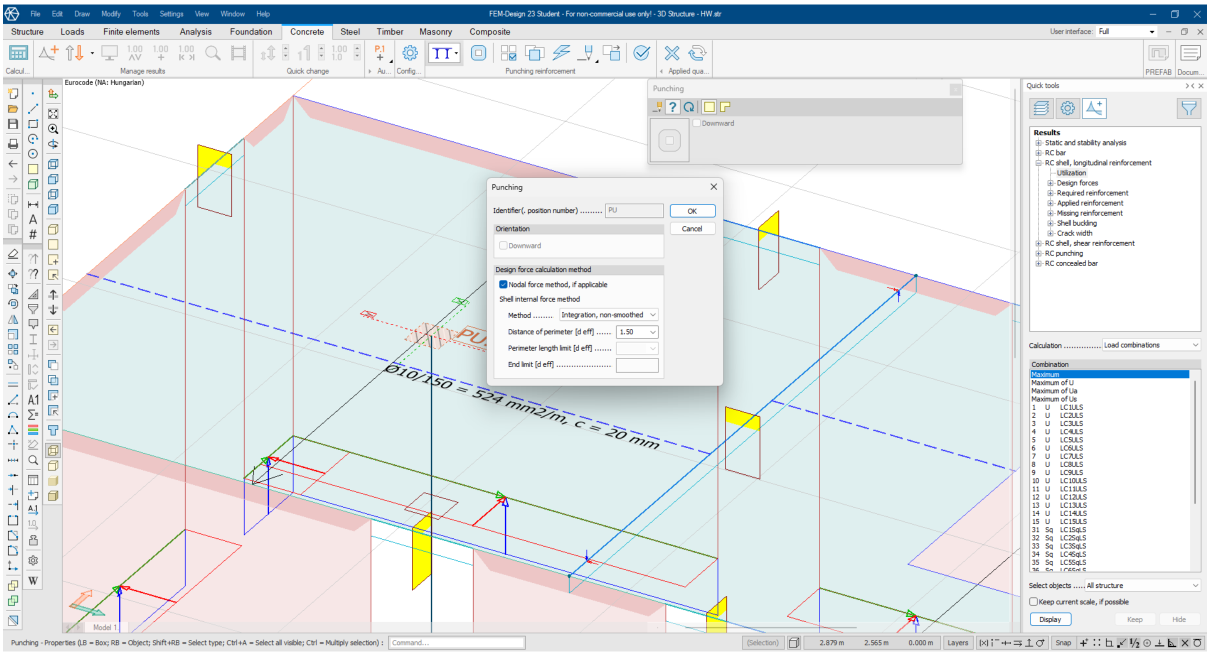
53. Figure: Detailed results of surface element longitudina reinforcement

When designing shear reinforcement, we can not only plan the reinforcement itself but also configure various options for the shear zone.

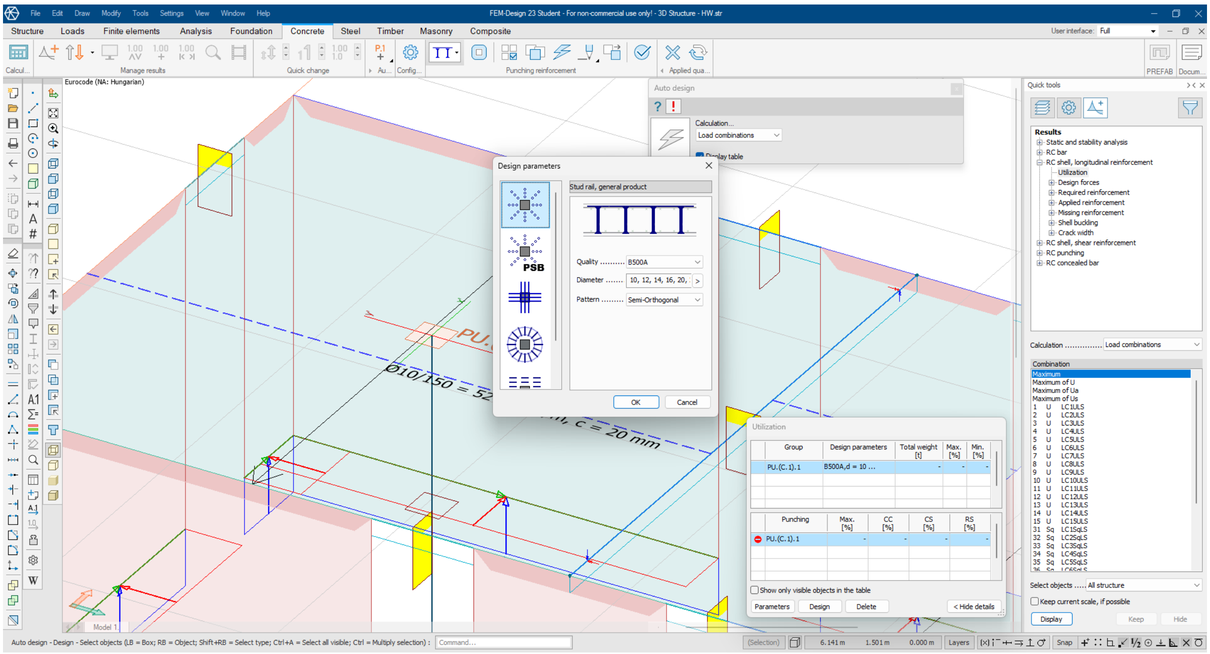


54. Figure: Shearing reinforcement

When placing punching shear reinforcement, there are also numerous configuration options available. This allows us to choose the characteristics of the region as well as the type of reinforcement.

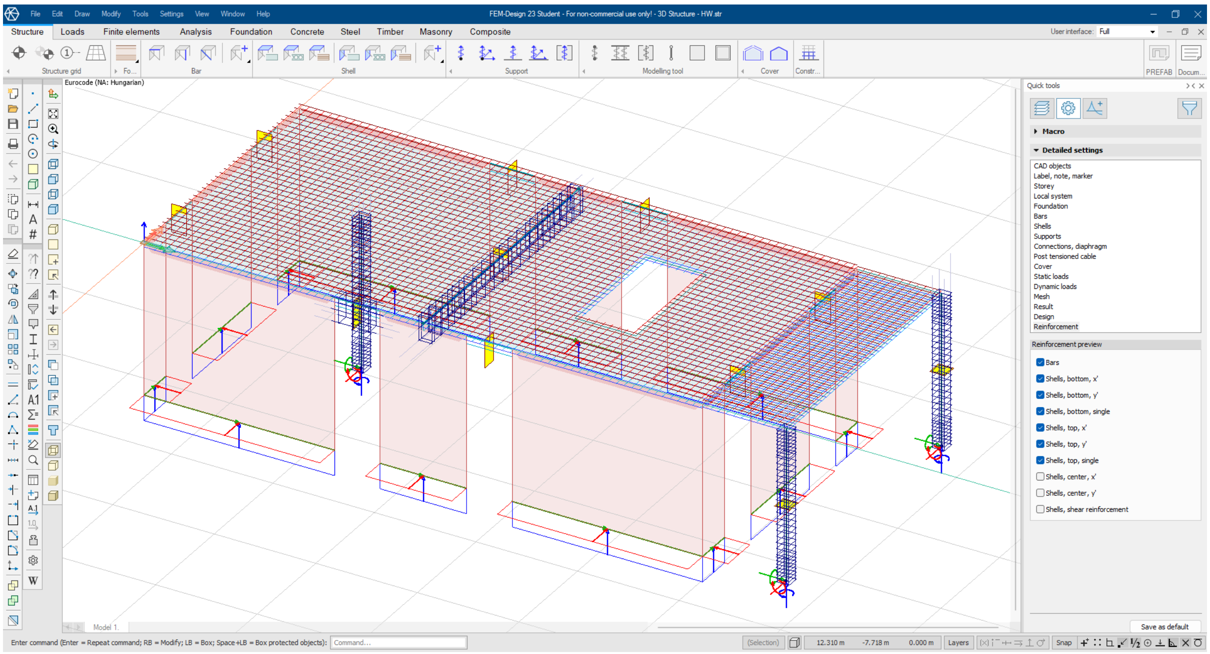


55. *Figure: Punching reinforcement*



56. *Figure: Punching reinforcement options*

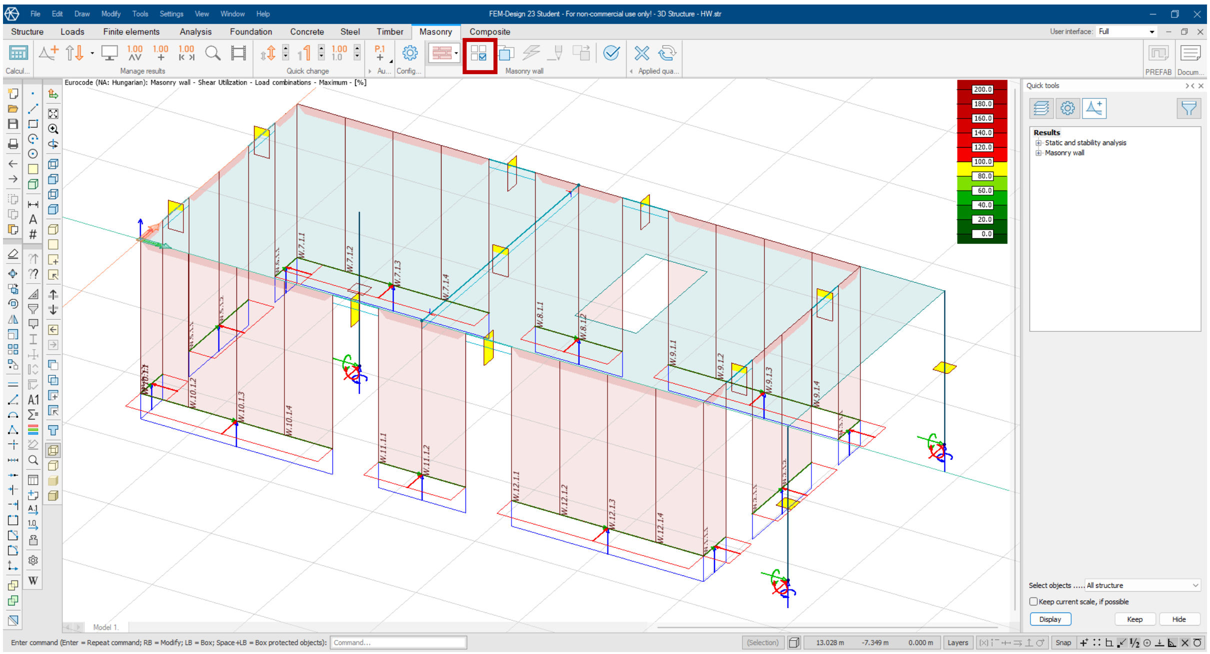
Reinforcement can be viewed not only in calculations but also in 3D format within the modeling space, as the program provides the option to display it this way.



57. Figure: Reinforcement preview

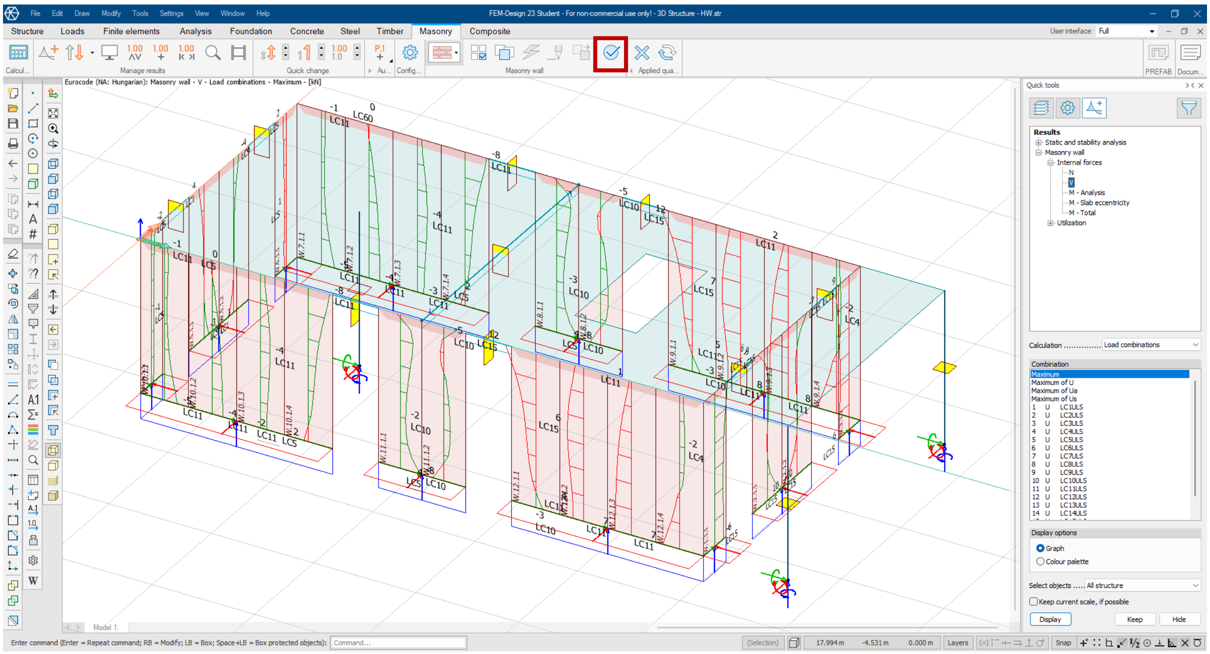
6.2 Masonry wall Check

You can create design groups and set calculation parameters.

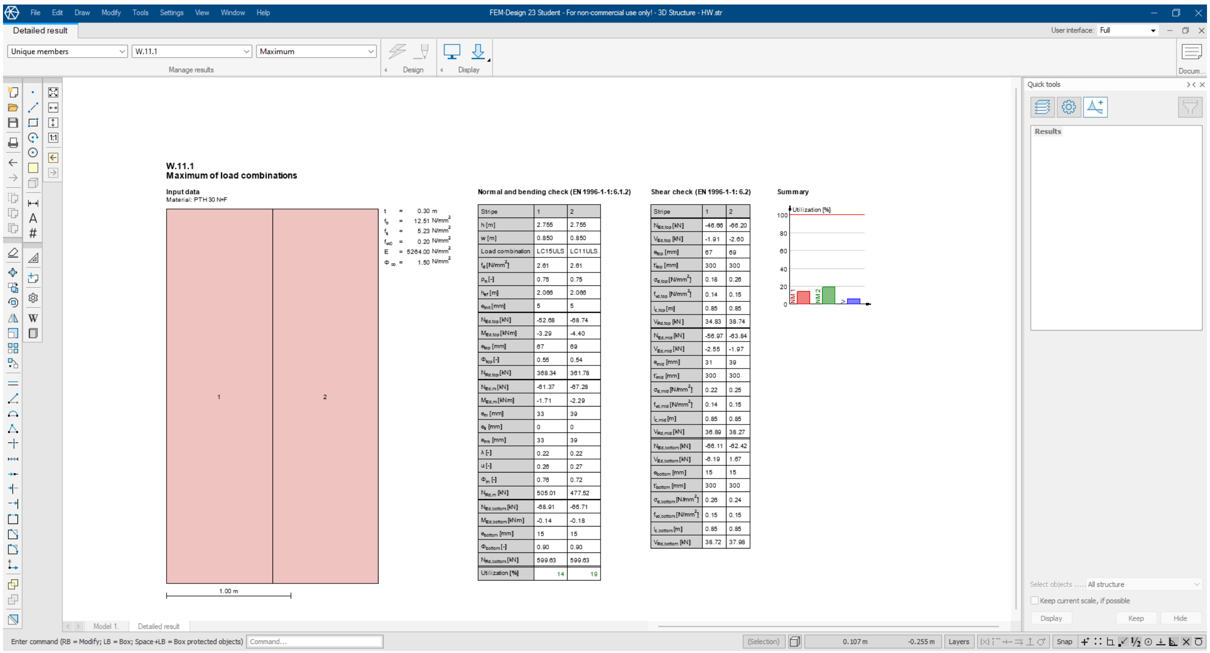


58. Figure: Masonry wall calculation parameters

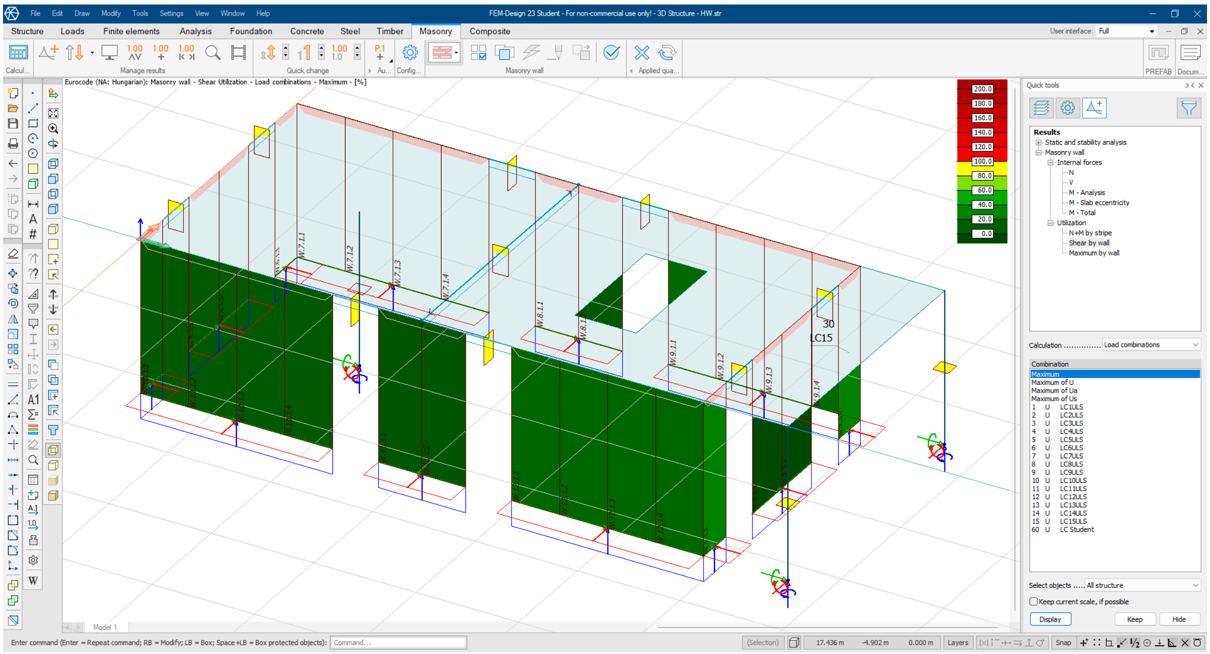
After running the verification calculation, we can immediately view the masonry stresses, along with the familiar detailed results and utilization.



59. Figure: Masonry wall internal forces



60. *Figure: Detailed results*



61. *Figure: Utilisation of masonry wall*