User Guide

FRP RC Studio

Platform User Guide

Platform	https://frprcstudio.net/
Document	Platform user guide
Version	1.0
Date	07 July 2026

Table of contents

1. General features 2
2. Creating a project 2
2.1 Defining project information and settings 3
2.2 Defining materials 4
2.3 Defining sections 8
2.3.1 Defining the section for a “Beam” element 8
2.3.2 Defining the section for a “Slab” element 13
2.4 Verifications and viewing the results 16
3. Layout 18
3.1 General layout management 18
3.2 Save your favorite layout(s) 19

All rights reserved. Reproduction prohibited - www.asdea.net/

1. General features

The platform present a side menu that gives access to the Dashboard, where you will have your projects and works, and a useful contact page.

If the user is an ERC member, the side menu is extended with an additional page, where they can provide ASDEA with details on the number of licenses they are interested in purchasing and other details we would like to see in the final version of the application.

There is also a top bar with buttons that allow you to view the licenses plans for using the application, download the documentation (manuals and videotutorials), submit bug reports, feedbacks and suggestions, view notifications, switch the language, and change the application theme.

The main page, i.e. the dashboard, allows you to browse all the projects created, duplicate and delete them, as well as create a new one.

2. Creating a project

The steps for creating a project are the following:

- Decide and entering the project name, information and settings;
- Defining the materials that make up the structural element to be analyzed;
- Defining the sections to be analyzed (entering the geometry and the FRP flexural reinforcement, optionally customizing the environmental conditions, and the ULS and SLS loads).

These steps are explained into details in the next paragraphs.

Defining the section provides a real time rendering of the entered element, so you can check that the inputs are correct. The details of the defined sections can always be viewed and edited from the Input window, which opens after clicking the section entry in the work tree.

Everything is then ready to proceed with the verifications. By clicking the “Verify Selected Section” or “Verify All Sections” button, you can view the summary of the various checks for the different structural elements.

You can also view the details of each verification performed by clicking the open-details button for each section.

In case of any errors, a popup will appear on the top with the details of the issue, and a notification with the same information will be stored in the relative notifications settings, so the user can review it in a second moment without losing it.

All rights reserved. Reproduction prohibited - www.asdea.net/

2.1 Defining project information and settings

The project information required are the name, which will allow searching within the dashboard, and some general details that will be printed in the calculation report.

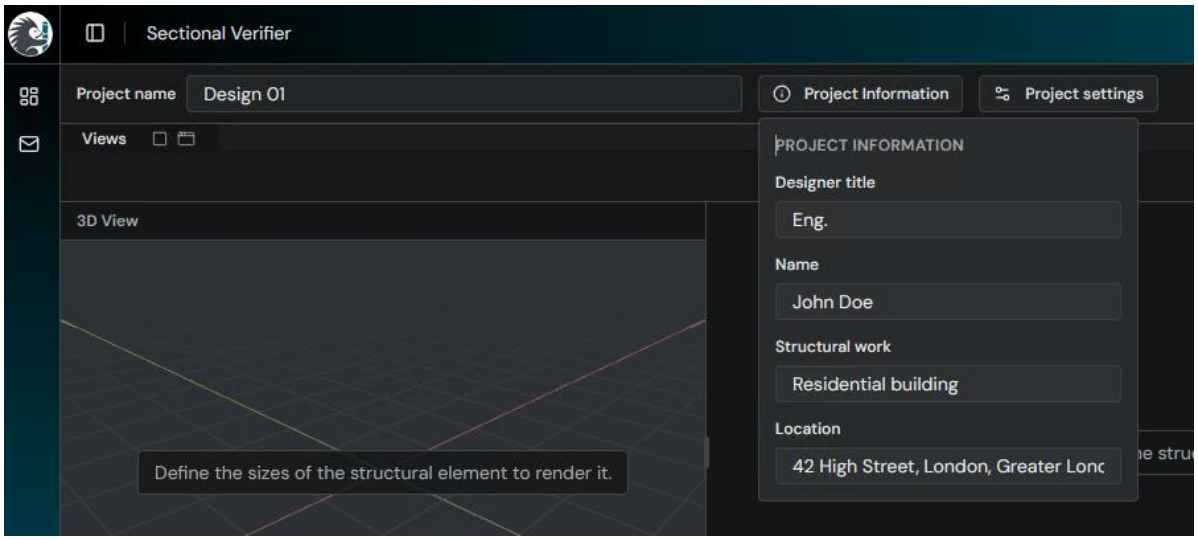


Figure 1 *— Project information panel*

The project settings, on the other hand, are relevant to the verification of the structural elements that will be defined later. The designer must define the exposure type of the structure, specify the associated exposure class, and — in the case of an external environment — indicate whether or not the structure is directly exposed to sunlight. The environment definition follows the specifications required by CNR-DT 203 R1, section 4.6.1.

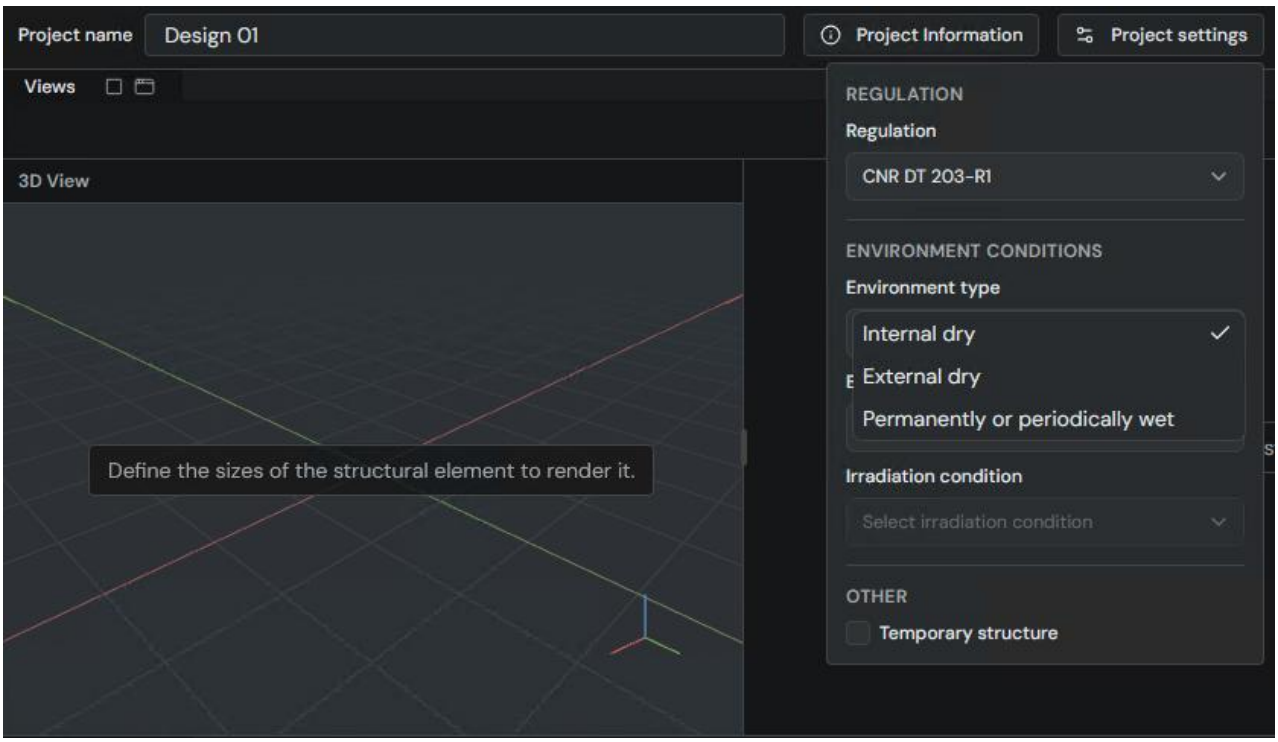


Figure 2 *— Project settings — environmental conditions*

All rights reserved. Reproduction prohibited - www.asdea.net/

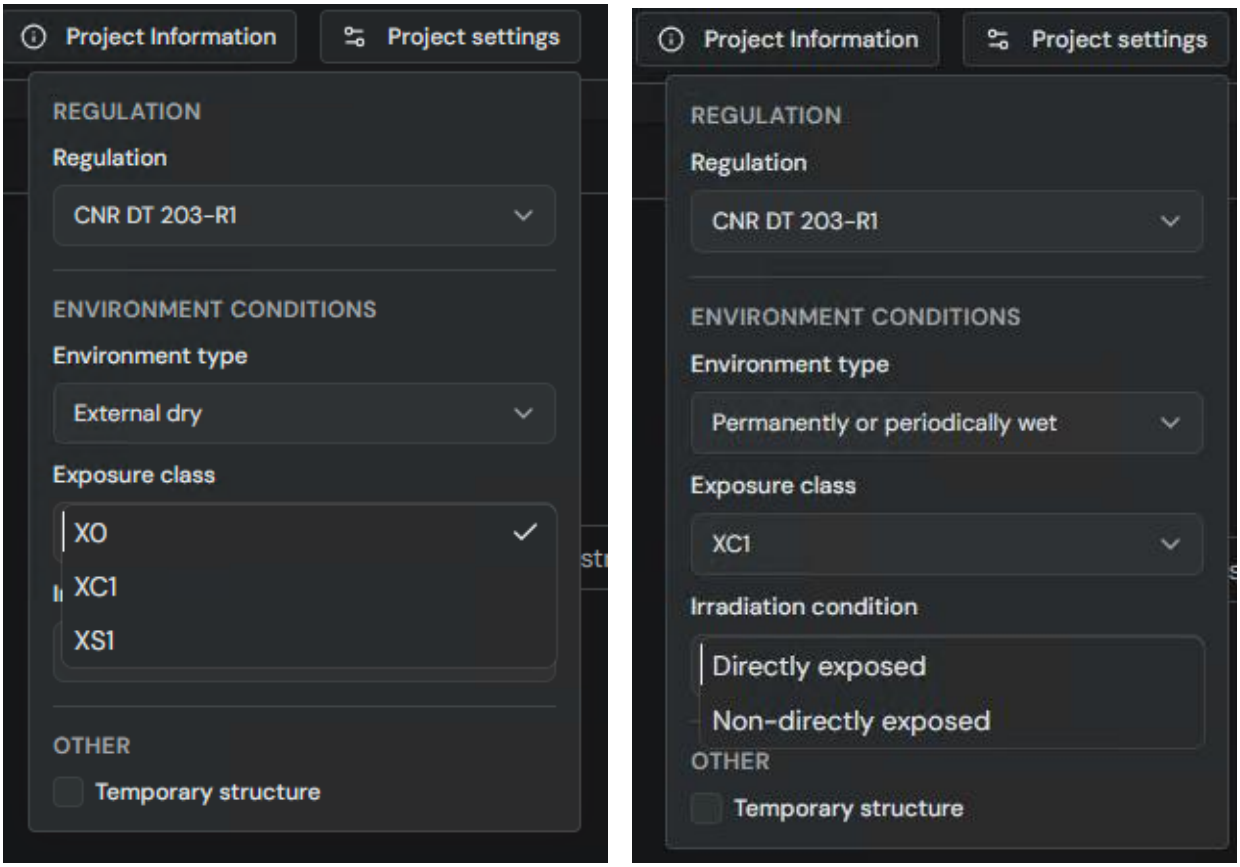


Figure 3 *— Exposure type and exposure class selection*

The designer is nevertheless allowed to perform the design for temporary structures , following the indications given in CNR-DT 203 R1, section 4.4.4. In that case, the relevant check-box must be ticked.

2.2 Defining materials

To proceed with the project, it is necessary to define the materials that make up the various sections to be analyzed. The designer must therefore enter the data for the concrete and the GFRP bars they intend to use, through the “Add Material” command, or right clicking the “Materials” node in the work tree and selecting “+ Add”.

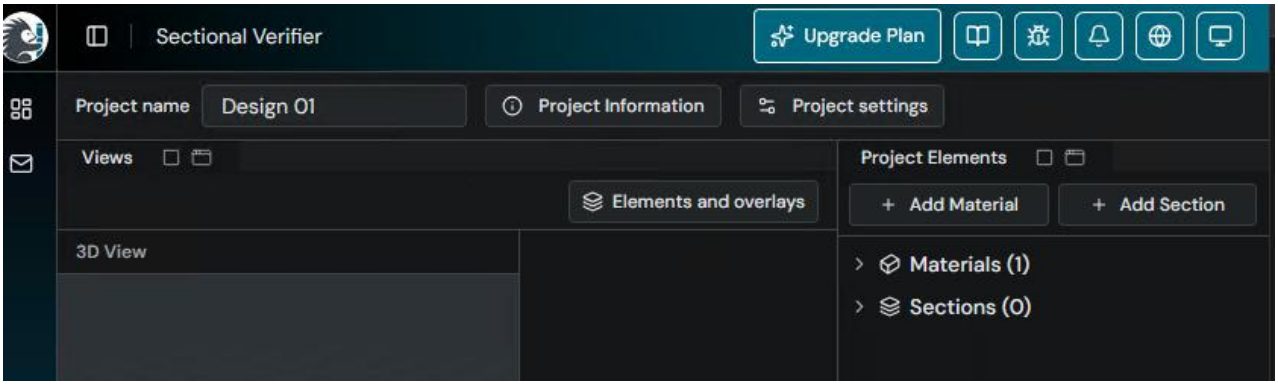


Figure 4 *— The Add Material command*

In the material input area, you must assign a name, indicate the material type — whether Concrete or FRP Rebars — and choose whether to use catalogue data or create a custom one.

For concrete, if the catalogue mode is chosen, the designer only needs to identify the strength class and indicate the creep coefficient of the concrete, which will be used in the SLS verifications; if instead a specific concrete is to be analyzed, all the quantities required to define must be provided.

All rights reserved. Reproduction prohibited - www.asdea.net/

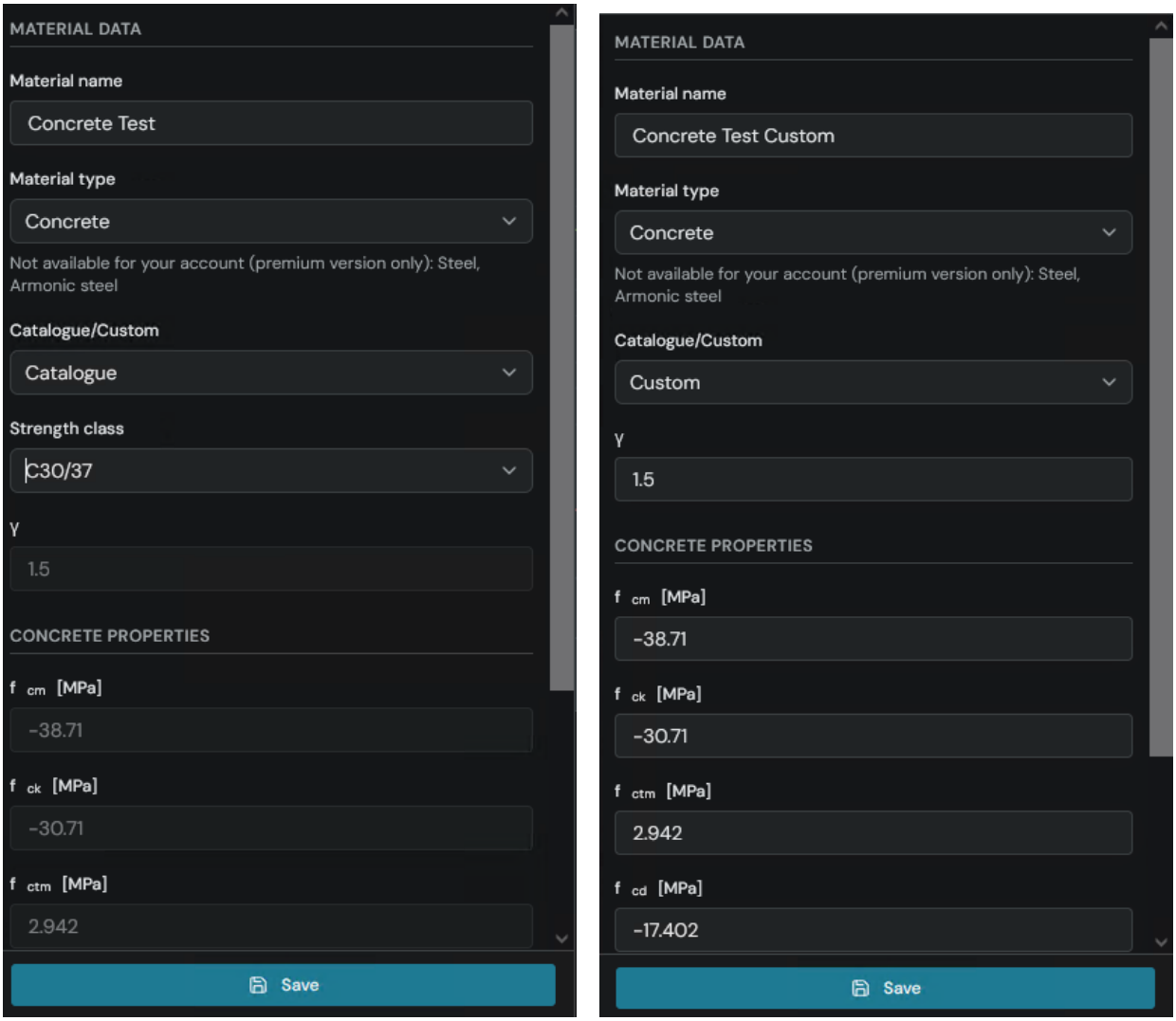


Figure 5 *— Concrete definition — catalogue and custom modes*

Defining the GFRP material is analogous to concrete definition: you must set a name, indicate the type, and choose whether to use a catalogue or a custom definition.

The catalogue loaded in the platform includes the materials characterized by the strength and bond classes available on the market, defined in accordance with the code minimums required by CNR-DT 203 R1, section 3.3.2 (table 3-3) and section 5.3. Specifically, the available designations are:

- E45/850 (C1) — ftk0 = 850 MPa for diameters db < 24 and 750 MPa for db ≥ 24; bond class 1, τb > 15 MPa;
- E50/1000 (C1) — ftk0 = 1000 MPa for diameters db < 24 and 900 MPa for db ≥ 24; bond class 1, τb > 15 MPa;
- E60/1100 (C1) — ftk0 = 1100 MPa for diameters db < 24 and 1000 MPa for db ≥ 24; bond class 1, τb > 15 MPa;
- E45/850 (C2) — ftk0 = 850 MPa for diameters db < 24 and 750 MPa for db ≥ 24; bond class 2, 7 ≤ τb ≤ 15 MPa;
- E50/1000 (C2) — ftk0 = 1000 MPa for diameters db < 24 and 900 MPa for db ≥ 24; bond class 2, 7 ≤ τb ≤ 15 MPa;
- E60/1100 (C2) — ftk0 = 1100 MPa for diameters db < 24 and 1000 MPa for db ≥ 24; bond class 2, 7 ≤ τb ≤ 15 MPa.

The associated diameters are the most common on the market and can be viewed, together with their properties, through the view button.

All rights reserved. Reproduction prohibited - www.asdea.net/

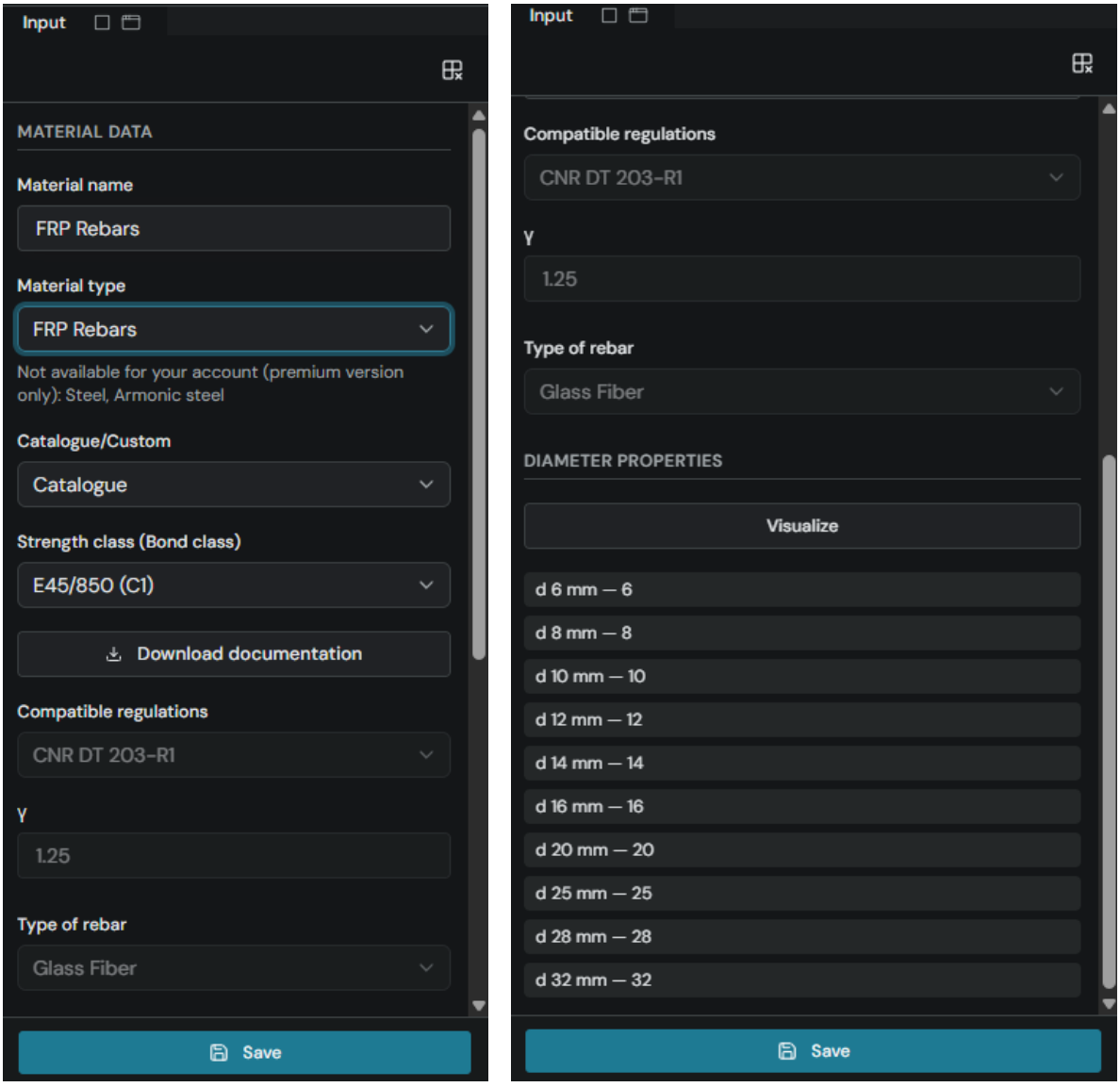
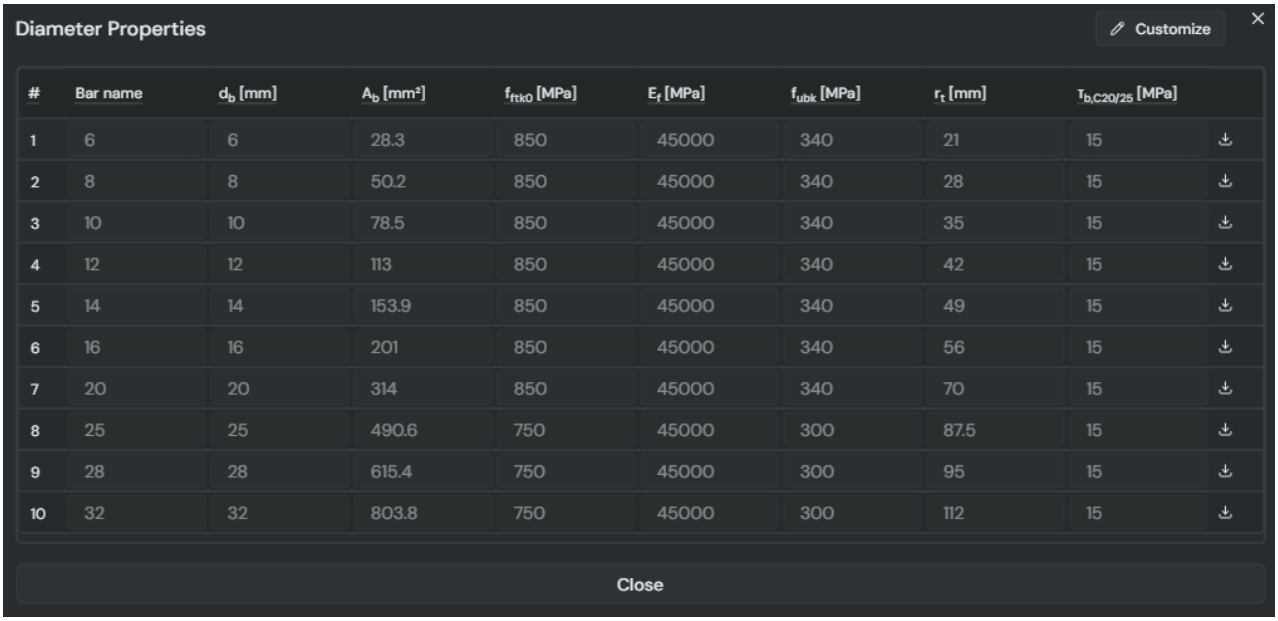


Figure 6 *— Viewing and customizing the bar diameters*



#	Bar name	d_b [mm]	A_b [mm²]	f_{frk0} [MPa]	E_f [MPa]	f_{ubk} [MPa]	r_t [mm]	$\tau_{b,C20/25}$ [MPa]
1	6	6	28.3	850	45000	340	21	15
2	8	8	50.2	850	45000	340	28	15
3	10	10	78.5	850	45000	340	35	15
4	12	12	113	850	45000	340	42	15
5	14	14	153.9	850	45000	340	49	15
6	16	16	201	850	45000	340	56	15
7	20	20	314	850	45000	340	70	15
8	25	25	490.6	750	45000	300	87.5	15
9	28	28	615.4	750	45000	300	95	15
10	32	32	803.8	750	45000	300	112	15

Figure 7 *— FRP catalogue designations*

Therefore, if the designer wishes to use the catalogue definition, they only need to select the material type. To modify the properties viewable through the “View” button, or to add diameters starting from a

All rights reserved. Reproduction prohibited - www.asdea.net/

custom material, they can click the “Customize” button and edit the table, or choose the custom definition. However in the Standard plan version, no checks are performed on the performance minimums required by CNR-DT 203 R1.

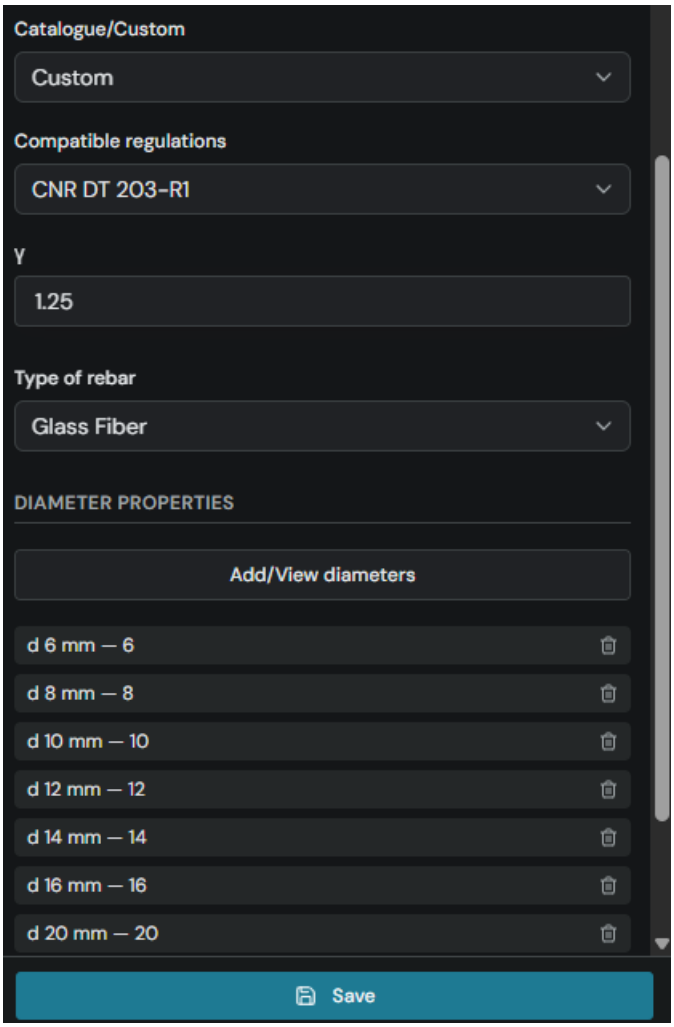


Figure 8 *— Customizing bar diameters and properties*

Once saved, the materials are listed in the “Project Elements” work tree under the Materials node.

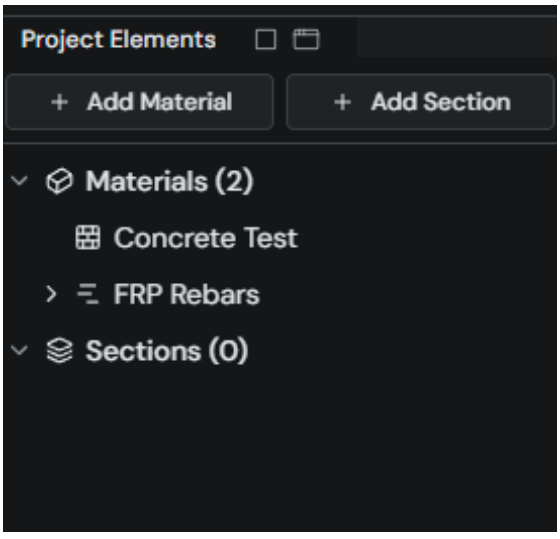


Figure 9 *— Materials listed in the Project Elements work tree*

It is always possible to view or edit the defined materials. To do so, left-clicking the specific material entry in the “Project Elements” work tree reopens the corresponding input screen, through which the operations described can be carried out.

All rights reserved. Reproduction prohibited - www.asdea.net/

Individual materials can be, eventually, deleted or copied by right-clicking the entry of the relevant item in the “Project Elements” work tree.

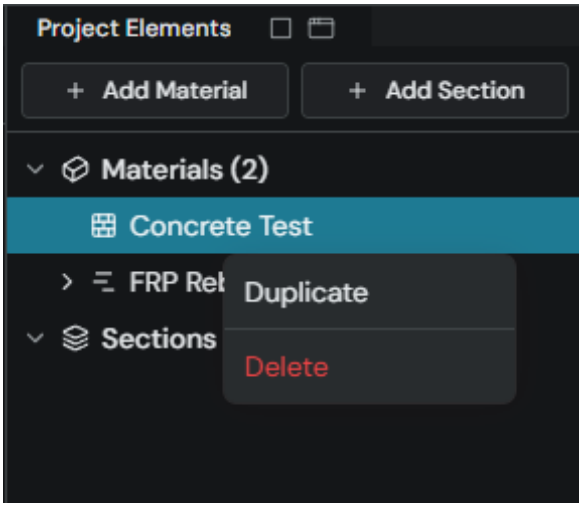


Figure 10 *— Editing, copying and deleting a material*

2.3 Defining sections

To define sections, the designer must click the “Add Section” button (or right clicking the “Sections” node in the work tree and clicking “+ Add”) and enter, on the input area, the name, the section shape (in the Standard version only the rectangular shape is available), and the type of structural element — whether “Beam” or “Slab”. Selecting the beam type requires only the reinforcement arranged along the element’s dimension; in the case of a slab, the reinforcement must be introduced along the two main, mutually orthogonal directions.

2.3.1 Defining the section for a “Beam” element

To define the section for the “Beam” structural element, the designer must indicate the width and height of the section, the length of the structural element, and the related concrete-type material. The length information is used by the application to render the structural element and to compute the deflection in the Premium version.

All rights reserved. Reproduction prohibited - www.asdea.net/

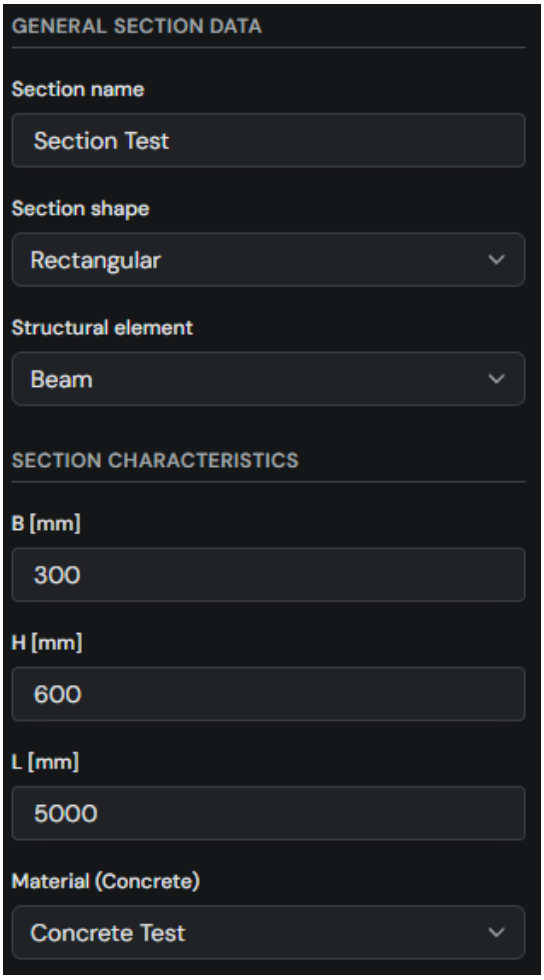


Figure 11 *— Beam section — geometry input*

Another required data is the location and identification of the FRP bars. To do this, from the “Input” screen press the “Add FRP Rebars” button, choose the previously defined FRP material, and add the various bar orders by clicking “Add Rebar”.

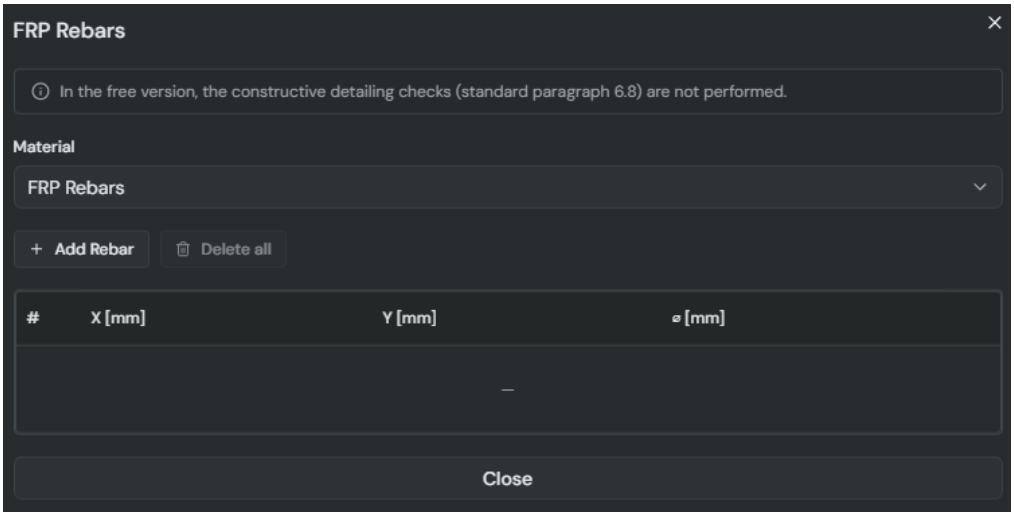


Figure 12 *— Add FRP Rebars*

Insertion can be done in three ways:

- Automatic arrangement — this will provide an helper that will arrange automatically the bars knowing the net concrete cover, the number of bars on the 4 sides, and the bar diameter;

All rights reserved. Reproduction prohibited - www.asdea.net/

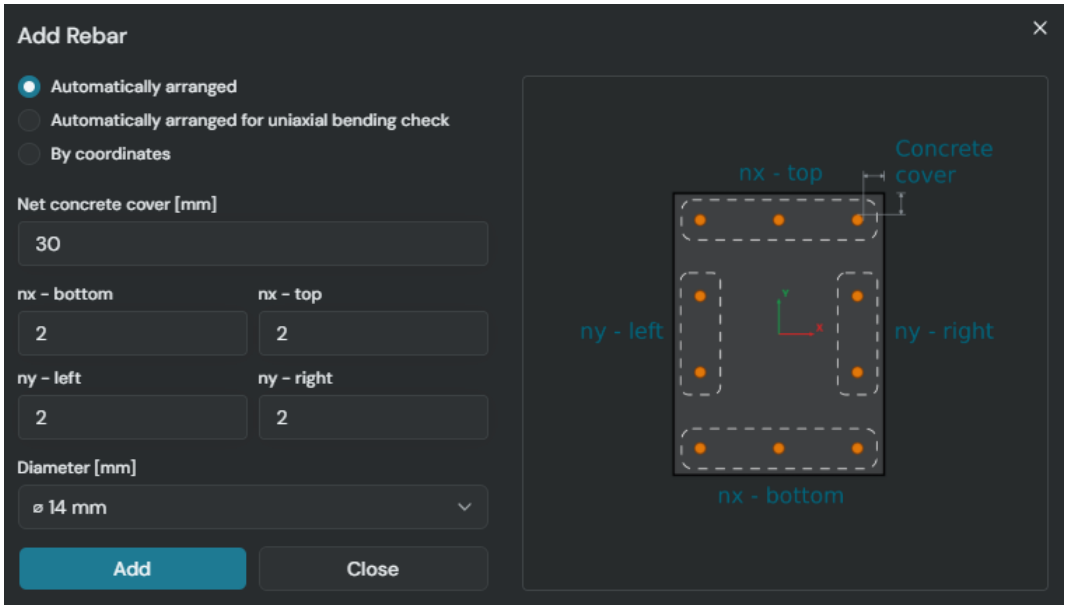


Figure 13 *— Automatic arrangement*

– Automatic arrangement for uniaxial bending — works similarly to the previous helper, but will ask only the parameters for the uniaxial bending verification, i.e. the net concrete cover, the number of bars on the bottom and top faces, and the bar diameter;

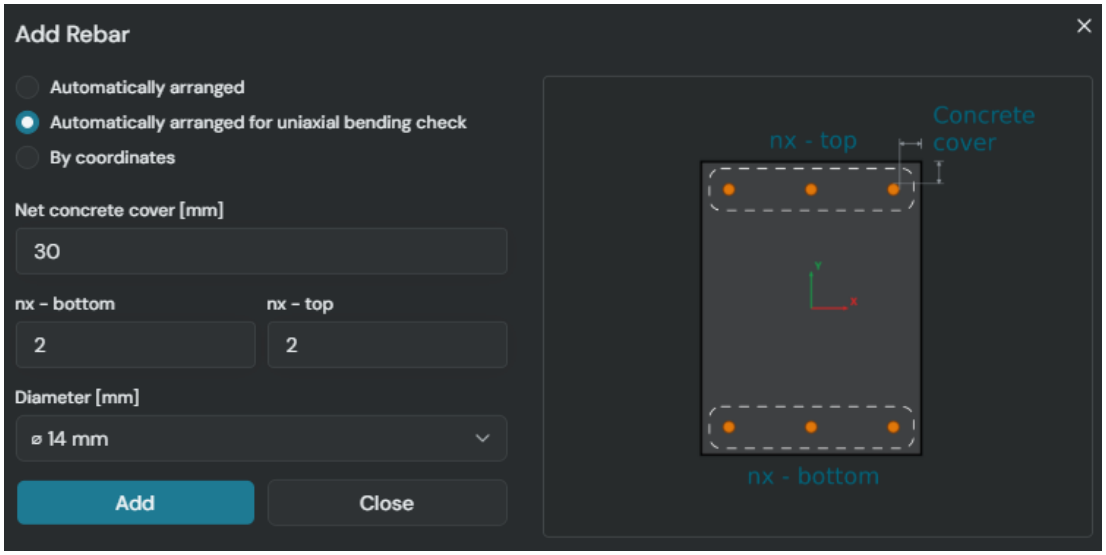


Figure 14 *— Automatic arrangement for uniaxial bending*

– Arrangement by coordinates — it will allow the user to manually define the bars position, asking the x and y coordinates in mm and the bar diameter for each entry.

All rights reserved. Reproduction prohibited - www.asdea.net/

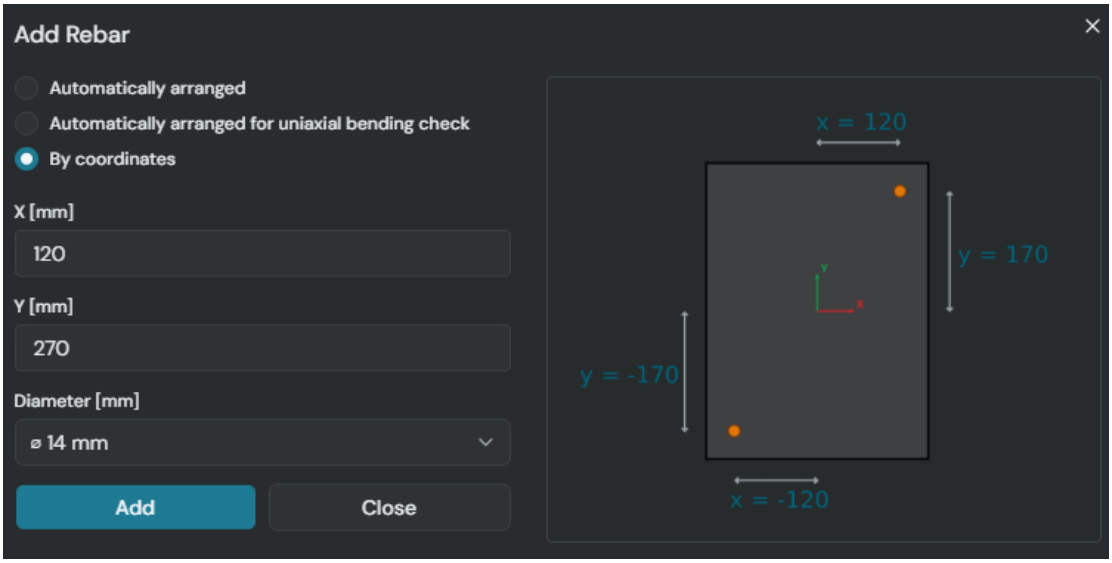


Figure 15 *— Arrangement by coordinates*

Once the reinforcement has been defined as described, a detail table is shown with the chosen coordinates and diameters. This data can still be modified, both in terms of coordinates and diameter. Note that the available diameters are only those defined for the previously chosen FRP material. Closing the defined-bars detail renders the created section in the application's main view.

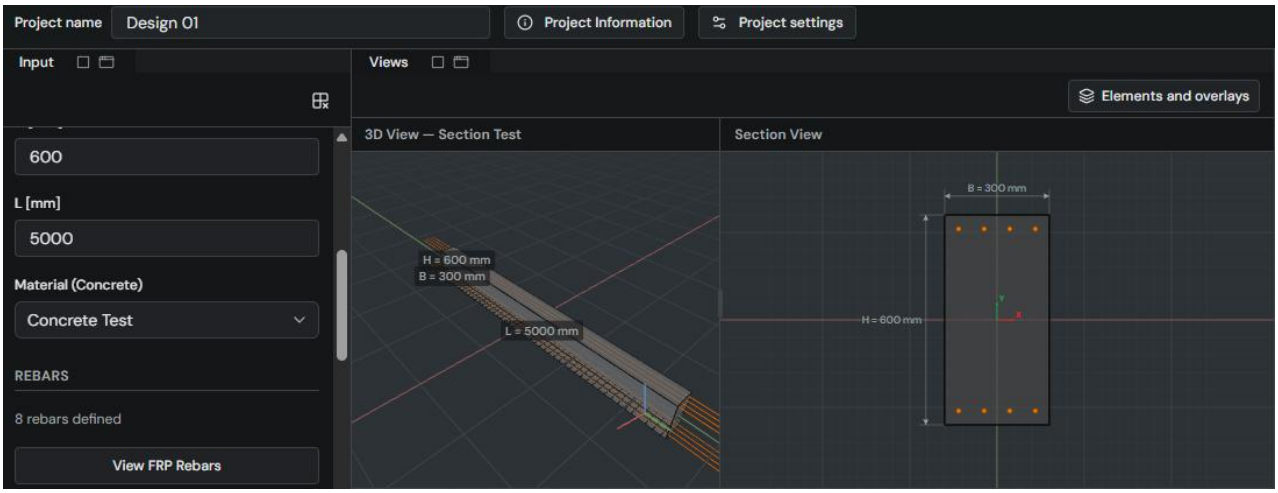


Figure 16 *— Rendered beam section*

At this point, the designer can indicate whether the section verification should use the same environmental conditions entered in the project information, or customize them for the selected element.

In the Premium version, the definition of the stirrups and the settings for the deflection and shear verifications are required.

The last step in defining the section is entering the loads; to do this, always from the input screen, click the "Add Loads" button. The designer can choose whether to perform only the ULS verification, only the SLS ones, or both. For each load case, the value of the applied bending moment defined about the local x axis must be entered, and the load type must be specified — whether short-term, long-term, or cyclic.

All rights reserved. Reproduction prohibited - www.asdea.net/

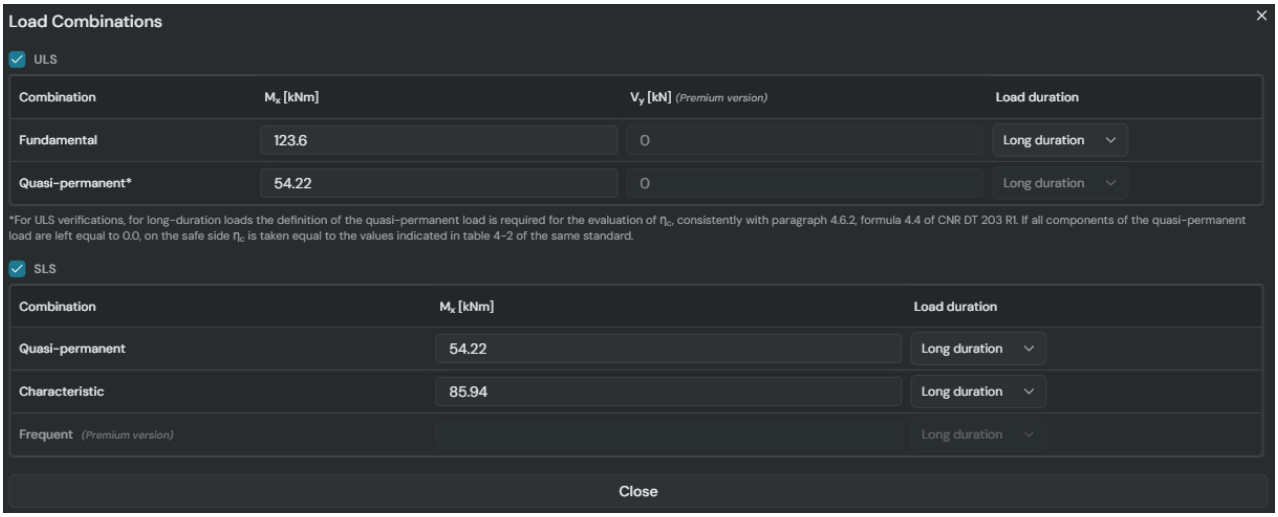


Figure 17 — Adding loads (ULS / SLS)

Finally, click the save button. As with the materials, the defined section is listed in the “Project Elements” work tree under the “Sections” entry.

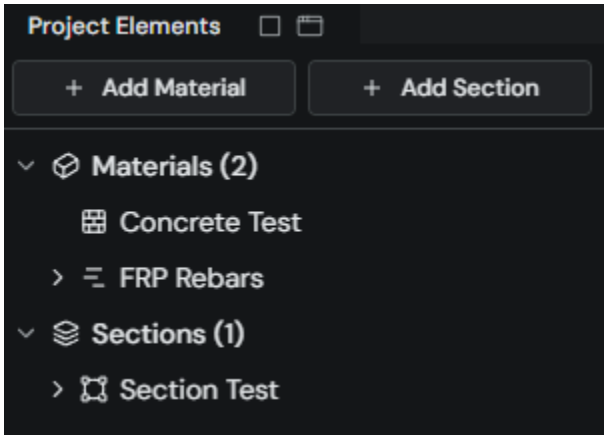


Figure 18 — Section listed in the work tree

2.3.2 Defining the section for a “Slab” element

The definition of “Slab” structural element is similar to the previous structural element. The main difference is the designer must indicate the slab span in directions 1 and 2 and the section height, since this is a “double section” element. The slab-span information in the two directions is used by the application to render the structural element and to compute the deflection in the Premium version.

All rights reserved. Reproduction prohibited - www.asdea.net/

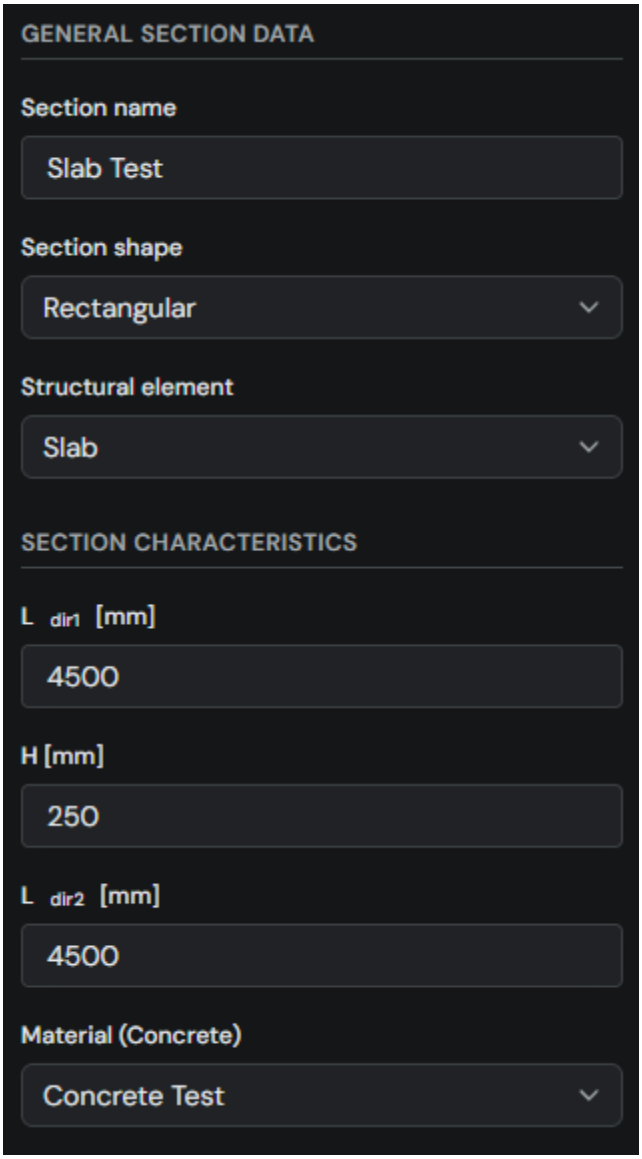


Figure 19 *— Slab section — geometry input*

As for the previous structural element, it is required to identify location and specs of FRP bars. To do this, from the “Input” panel, press the “Add FRP Rebars” button, choose the previously defined FRP material, indicate the bar diameter, and add the bar layers using the “Add Rebar” button for the two directions.

All rights reserved. Reproduction prohibited - www.asdea.net/

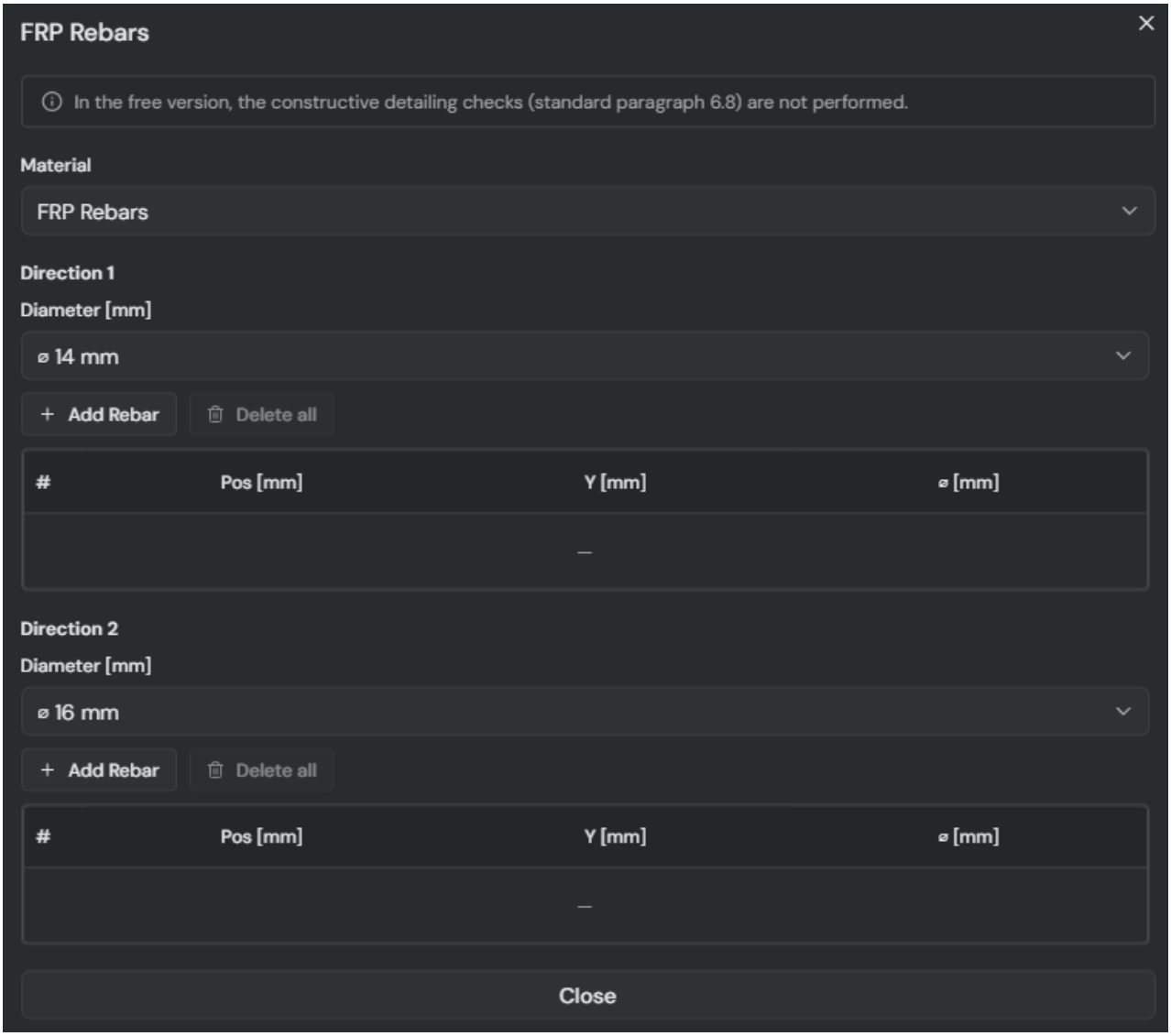
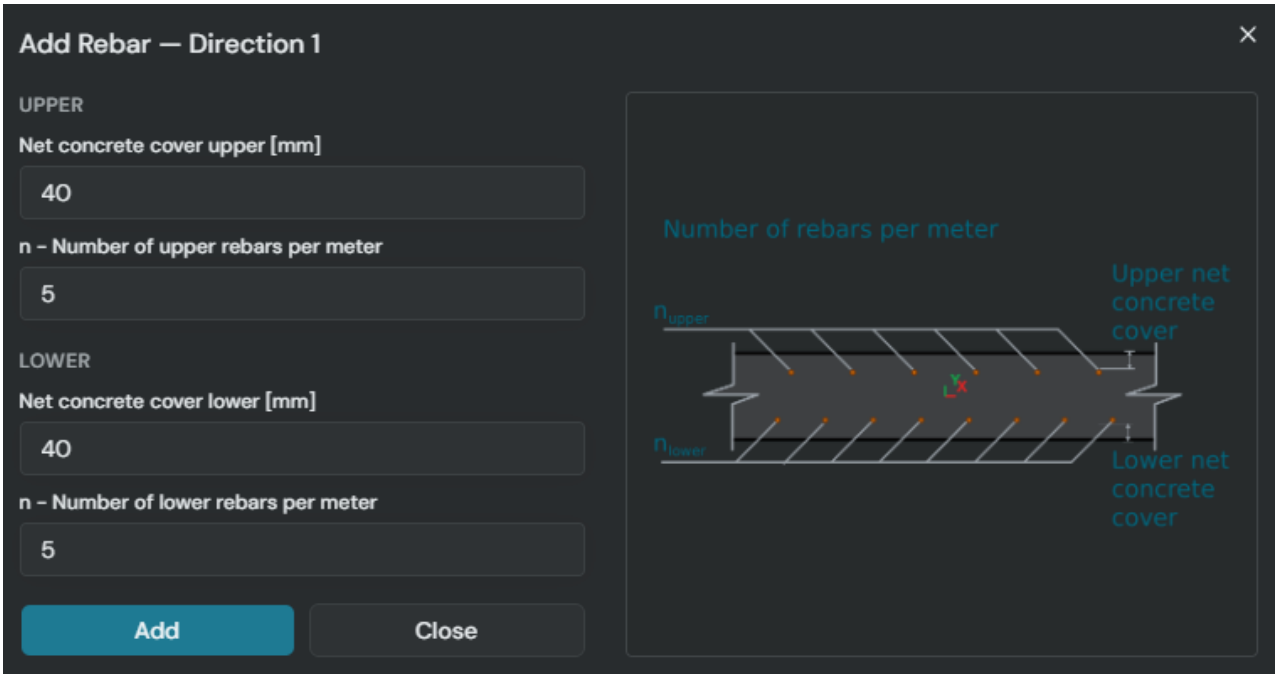


Figure 20 — Slab — Add FRP Rebars

For each layer, the designer must indicate the top and bottom reinforcement, specifying the net concrete cover and the number of bars per metre.



All rights reserved. Reproduction prohibited - www.asdea.net/

Figure 21 — Reinforcement layers per direction

Once the reinforcement has been defined as described, a detail table is shown with the chosen coordinates and diameters. Closing the defined-bars detail renders the created section in the application's main view.

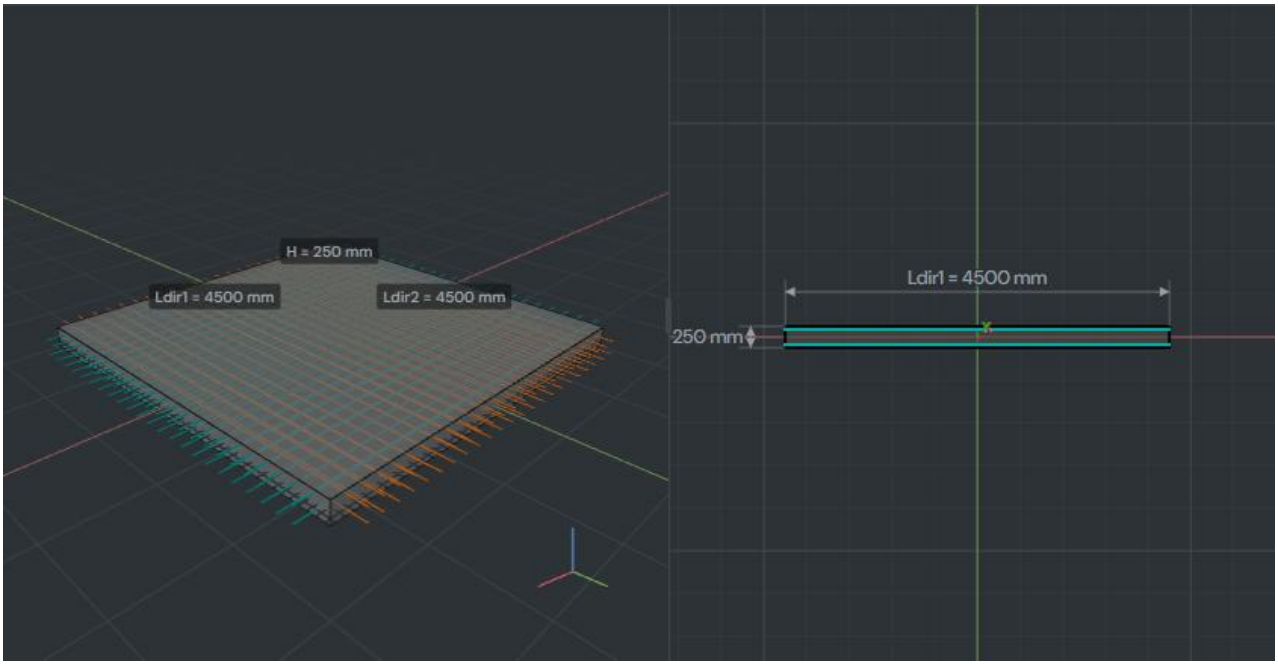


Figure 22 — Rendered slab section

The designer can indicate whether the section verification should use the same environmental conditions entered in the project information, or whether they should be customized.

In the Premium version, some data required for the deflection and shear verifications will be requested.

The last step of section definition is entering the loads; to do this, always from the input area, click the "Add Loads" button. The designer can choose whether to perform only the ULS verification, only the SLS ones, or both. For each load case, the value of the applied bending moment defined about the dir1 and dir2 axes must be entered, and the load type must be specified — whether short-term, long-term, or cyclic.

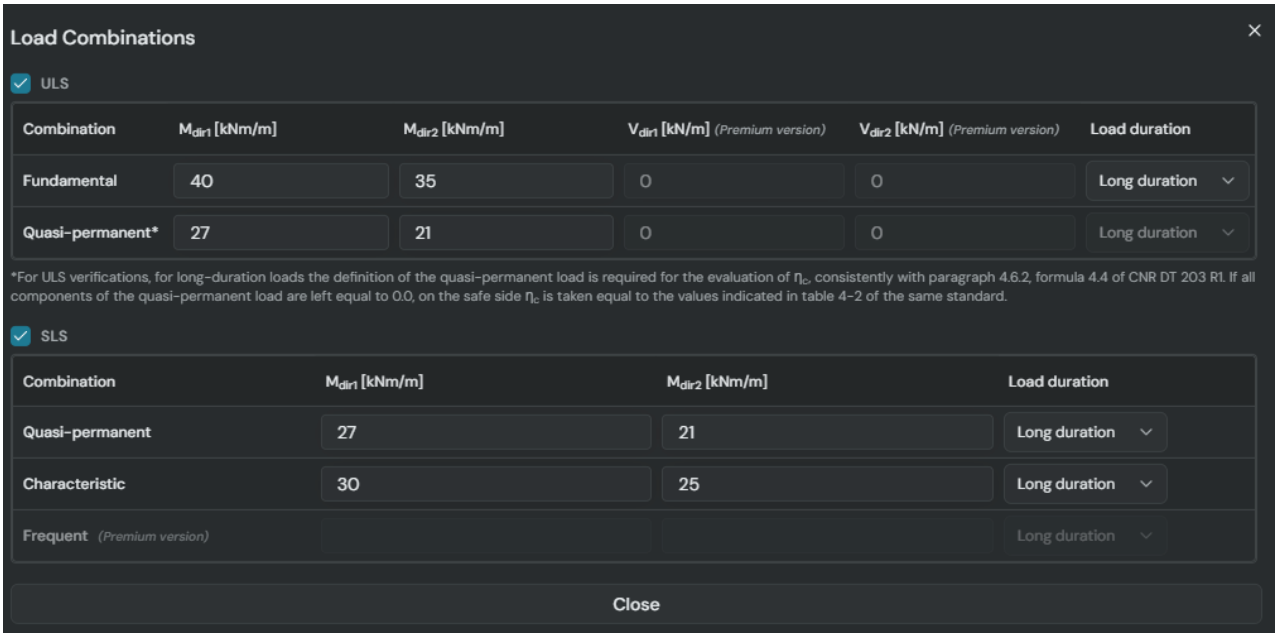


Figure 23 — Slab — adding loads

All rights reserved. Reproduction prohibited - www.asdea.net/

Finally, click the save button. As with the materials, the defined section is listed in the “Project Elements” work tree under the “Sections” entry.

2.4 Verifications and viewing the results

To perform the verifications for the various sections defined within the project, click the “Verify All Sections” button on the bottom right, or run the verification of the current focused section using the “Verify Selected Section” button.

Once the analysis is launched, the summary table of the performed verifications is filled in with all the worst outcomes for each analysis.

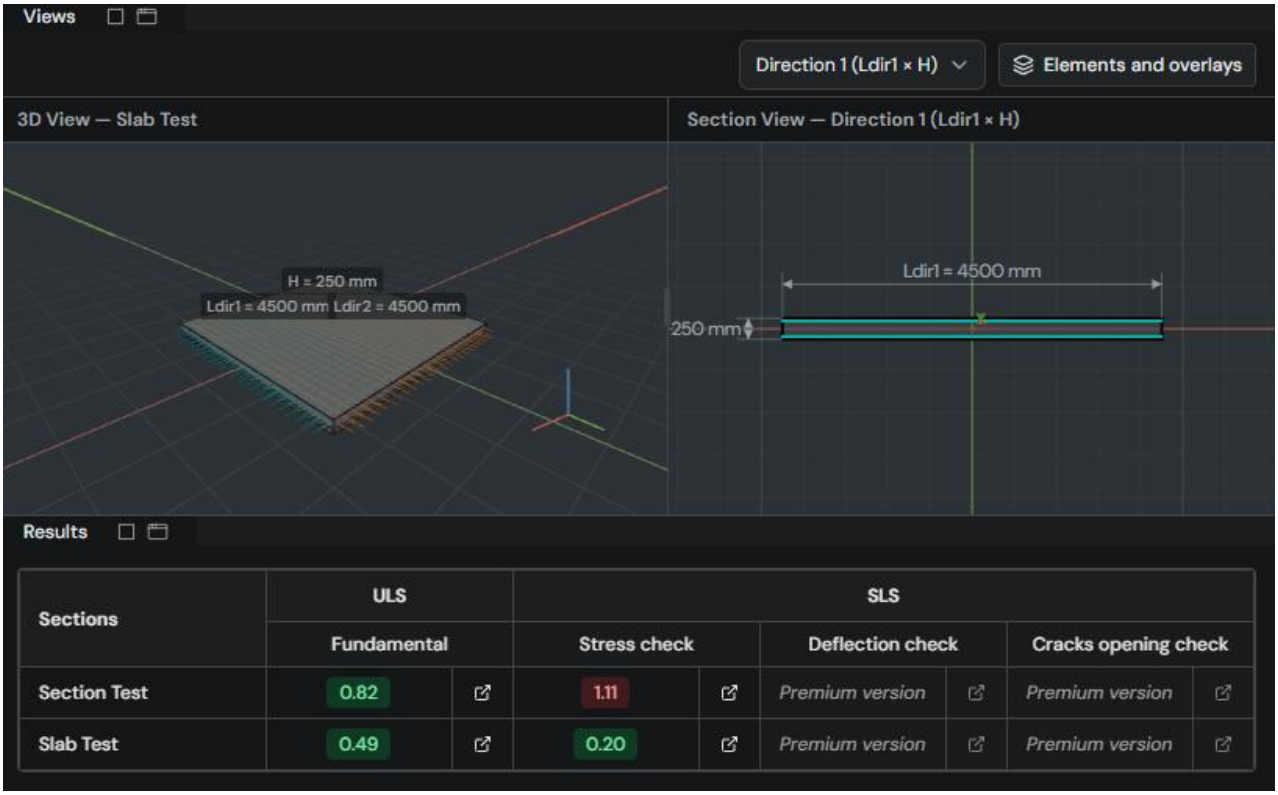
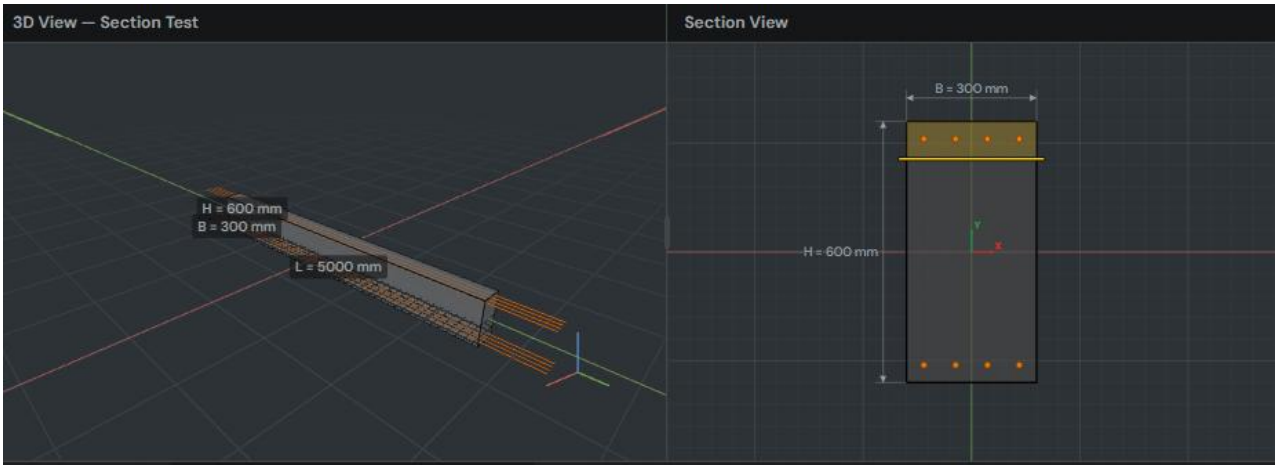


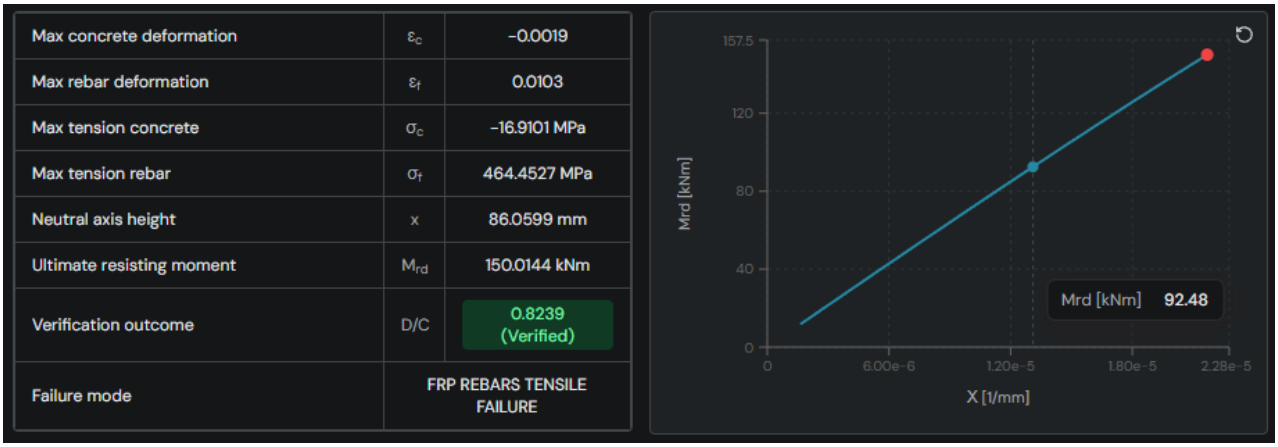
Figure 24 *— Verification summary table*

The results are divided by limit states. To view the detail of the various verifications performed, click the “Open details” icon shown in the individual rows of the table.

In the ULS verification detail, you can see the resisting moment, the related stress and strain states, and the moment–curvature relationship.



All rights reserved. Reproduction prohibited - www.asdea.net/



Max concrete deformation	ε_c	-0.0019
Max rebar deformation	ε_f	0.0103
Max tension concrete	σ_c	-16.9101 MPa
Max tension rebar	σ_f	464.4527 MPa
Neutral axis height	x	86.0599 mm
Ultimate resisting moment	M_{rd}	150.0144 kNm
Verification outcome	D/C	0.8239 (Verified)
Failure mode	FRP REBARS TENSILE FAILURE	

Figure 25 — *ULS verification detail*

In the main view, the position of the neutral axis is shown at the same time.

In the SLS verification detail, you can see the position of the neutral axis, the maximum stress in the concrete and in the FRP bars, and the ratio between the latter and the code limits set for the quasi-permanent and rare combinations.

The neutral axis is relative to the worst outcome between the two stress check's combinations.

STRESS CHECK

Characteristic combination		
Neutral axis	x	53.7279 mm
Compressive stress in concrete (- compression; + Tension)	σ_c	-20.0395 MPa
Compressive stress in FRP rebars (- compression; + Tension)	σ_f	258.7732 MPa
Verification outcome (Demand/Capacity)	FRP rebars	0.4477 (Verified)
	concrete	1.1133 (Not verified)
Quasi-permanent combination		
Neutral axis	x	93.1378 mm
Compressive stress in concrete (- compression; + Tension)	σ_c	-7.3662 MPa
Compressive stress in FRP rebars (- compression; + Tension)	σ_f	166.9813 MPa
Verification outcome (Demand/Capacity)	FRP rebars	0.4622 (Verified)
	concrete	0.5456 (Verified)

Figure 26 — *SLS verification detail*

For "Slab" elements, there is the possibility to analyze the results of both directions: as default, the worst outcome is shown, but the user can always switch between directions using the dropdown input located at the top of each result detail tab.

Finally, you can export a detailed report through the "Generate Report" → "Basic report" button.

There is not an auto-save system, so it is important not to forget save the project using one of the two "Save" buttons located in the bottom right of the page, so they will be stored in the dashboard page.

All rights reserved. Reproduction prohibited - www.asdea.net/

3. Layout

This application presents a user-friendly layout that remembers the style of the major engineering softwares for structural analysis and modelling, despite the fact it is a web-application.

In particular, inside the project page, user will be able to resize freely all the panels and menus. Also, each panel can be moved around the page, attached to all the areas available, detached and kept as floating window and maximized in full screen.

3.1 General layout management

To move and attach automatically a panel in an anchor area of the page, just hold the panel name and move the mouse to the desired location.

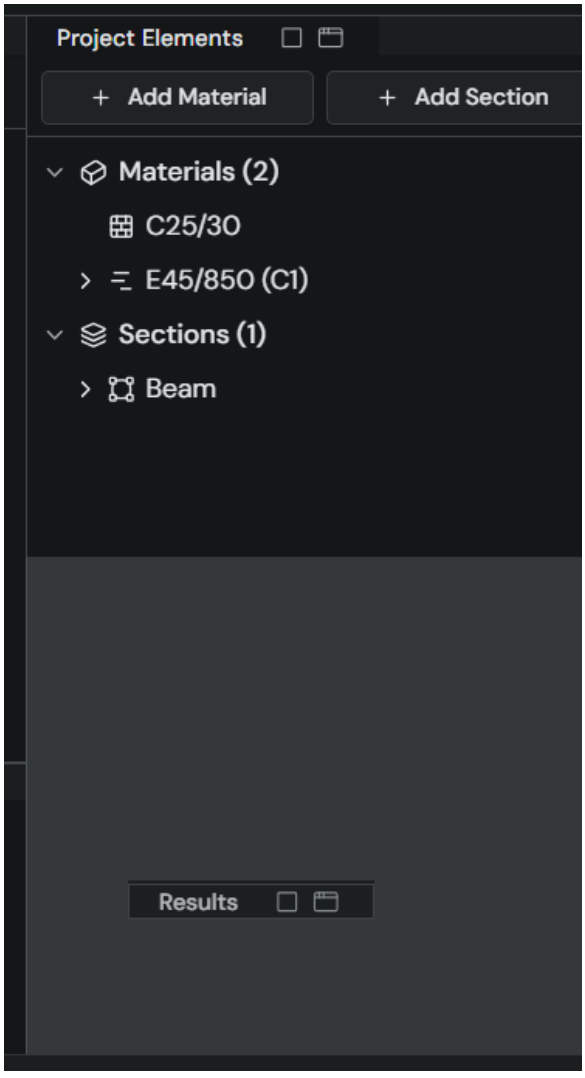


Figure 27 *— Reorganize layout with drag and drop of panels*

To deattach a panel using it as a floating window, use the detach button located at the left of the panel's title. Press the same button to force the panel anchor back to the starting location.

To maximize the panel, click the "Maximize" button located to the left of the panel's title. Press it back to minimize it.

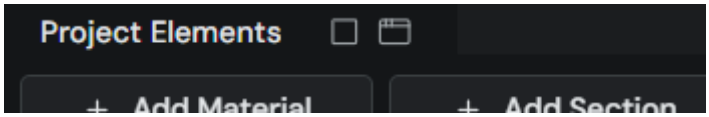


Figure 28 *— Focus on panel title, maximize and detach buttons*

All rights reserved. Reproduction prohibited - www.asdea.net/

3.2 Save your favorite layout(s)

The application allows the user to save their layouts, so it is not required each time you enter the platform to restore it manually. You can save up to 5 different layouts.



Figure 29 — User's layouts in bottom bar

When the user wants to save the current layout, press the "+" button located in the bottom bar of the page, near the "☆" button. The application will ask a custom name for the layout and the information will be saved.

It is possible to freely switch between layouts in any moment.

If the layout was changed by mistake, it is always possible to go back to the default one clicking the star "☆" button on the bottom bar. This will restore immediately the base layout. All the previously saved ones will be kept.

The user's layouts can be deleted, renamed or overwritten using the "3 points" action attached to the layout selection button in the bottom bar.

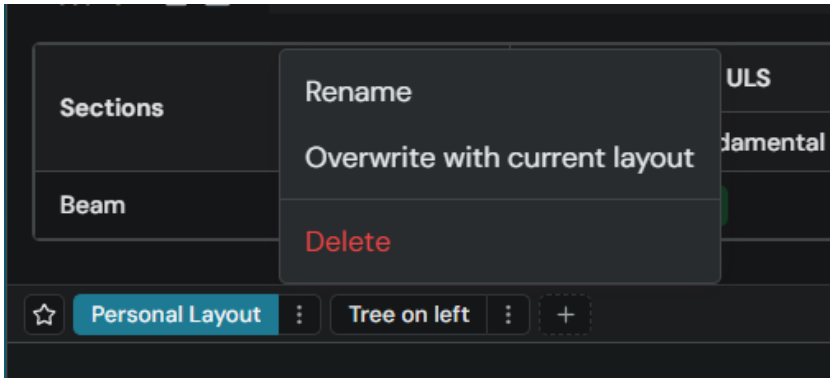


Figure 30 — Modify a saved layout

All rights reserved. Reproduction prohibited - www.asdea.net/