6 Graphical User Interface – Program HLS

In the following, the Graphical User Interface (GUI) of the software HLS is presented. The main objective of the GUI is the ease of use of the design methods developed in the framework of HOLLOSSTAB. Therefore, it has been decided to create a simple interface that allows the user to define his/her design case step by step with as few manipulations as possible.

Fig. 28 represents the main window of HLS with the tab opened after the user has launched the program. In the upper part of the main window, the user has access to the general "file" options "New", "Open", "Save", "Save as" and "Quit" by clicking on "File" (see Fig. 28 – 1)).

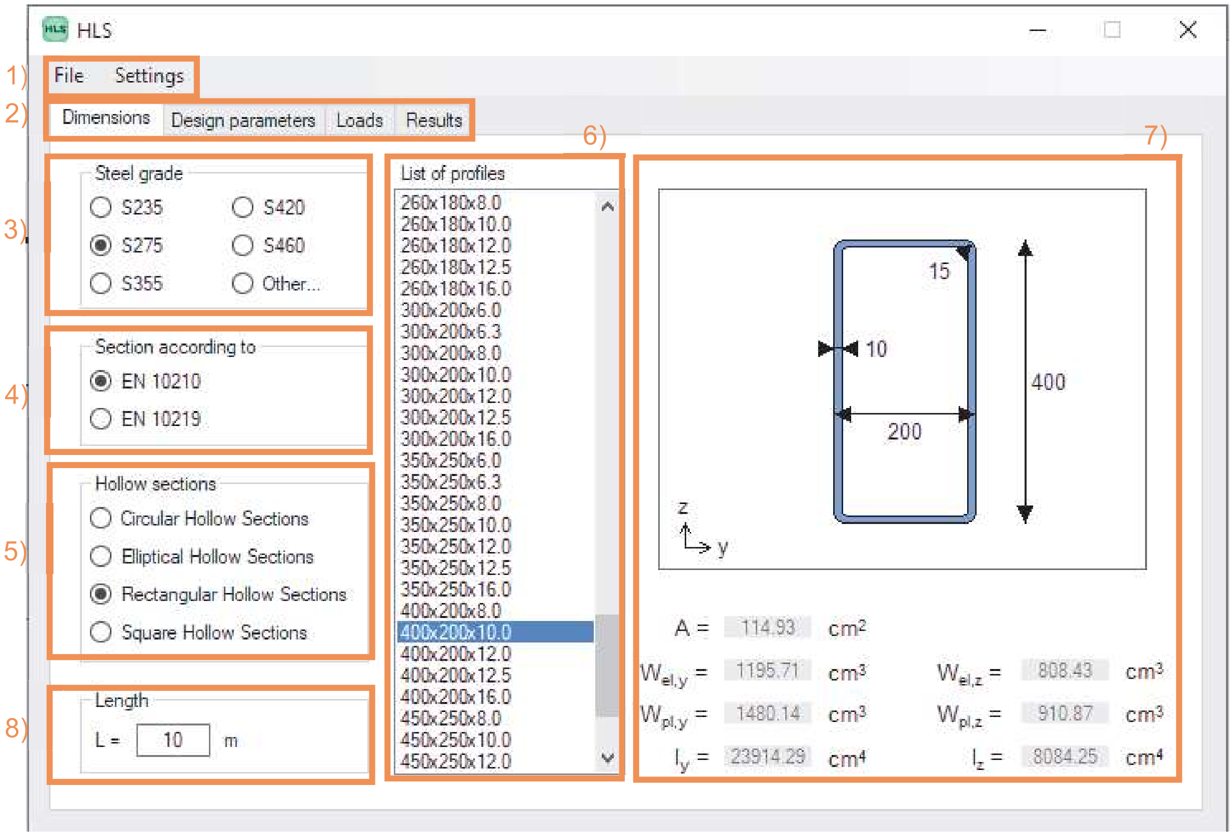

Fig. 28 Tab "Dimensions"

By clicking on "Settings" in zone 1) of Fig. 28, the user can define the language used in the GUI (the GUI is available in French and in English) as well as the calculation parameters (used for the execution of the programs presented in paragraphs 2 to 5 – see Fig. 29). Default values for the calculation parameters are provided and they should only be modified by experienced users. In the window "Calculation_parameters", the user has also the choice of the design method for circular and elliptical hollow sections (strength based or deformation based – see deliverables D.8.3 of HOLLOSSTAB for more information).

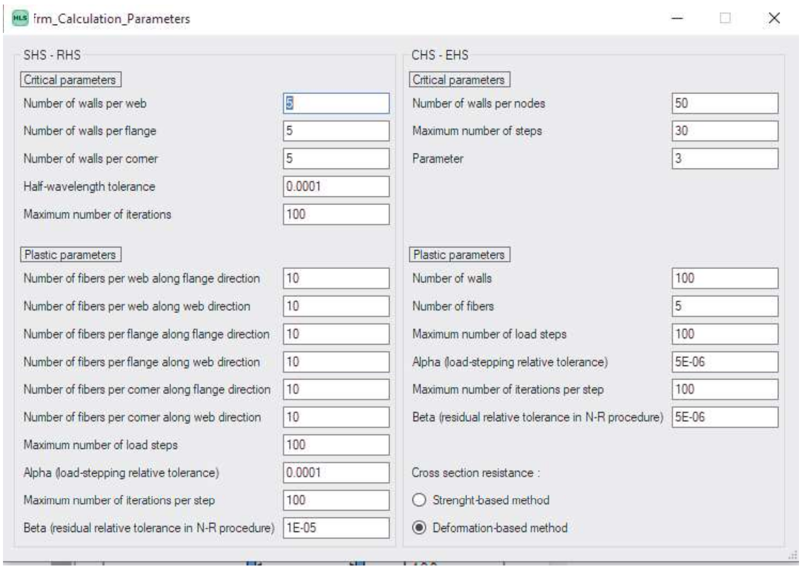

Fig. 29 Calculation parameters

The different tabs used for the definition of the design case and its calculations are then shown in zone 2) of Fig. 28. After the program has been launched, the first tab “Dimensions” is automatically activated. This first tab is organised in 6 zones (3) to 8) in Fig. 28), that allow the user to define:

- Steel grade – zone 3);
- Product standard (used to load the catalogue of sections shown in 6)) – zone 4);
- Shape of the section – zone 5);
- List of profiles according to the chosen product standard and the shape of the section – zone 6);
- Representation of the chosen section and its main mechanical properties – zone 7);
- Length of the member – zone 8).

The user has the possibility to define a customised section with the “Profile_Creator” that is accessible in the top of the “list of profiles” (see Fig. 30).

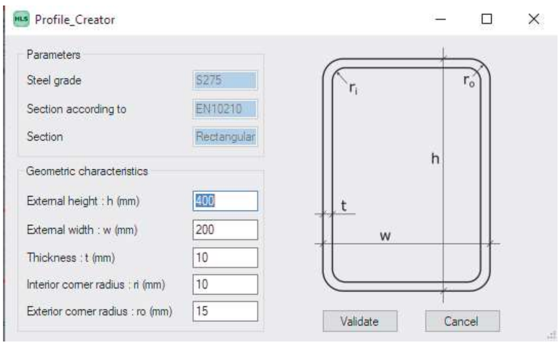

Fig. 30 Profile Creator

Additionally, the user can define steel grades higher than S460 by choosing "Other" in zone 3) of Fig. 28. In this case the "Steel_grade_creator" is opened and the user may enter a specific yield strength f_y and ultimate strength f_u.

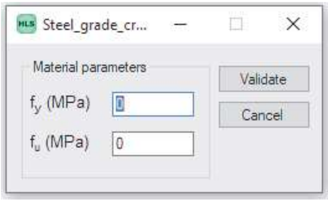

Fig. 31 Steel Grade Creator

The tab "Design parameters" recalls the automatically generated load combinations used to determine the design internal forces and moments (see zone 1) – Fig. 32). The partial factors for the loads, γ_G and γ_Q, as well as the combination factors, $\psi_{0,1}$ and $\psi_{0,2}$, can be defined by the user in zone 2) and 3) of the "Design Parameters" Tab. Unlike the partial factors for the loads, the user cannot modify the partial factors for the resistance. In this case, the value determined in Work Package 7 is used.

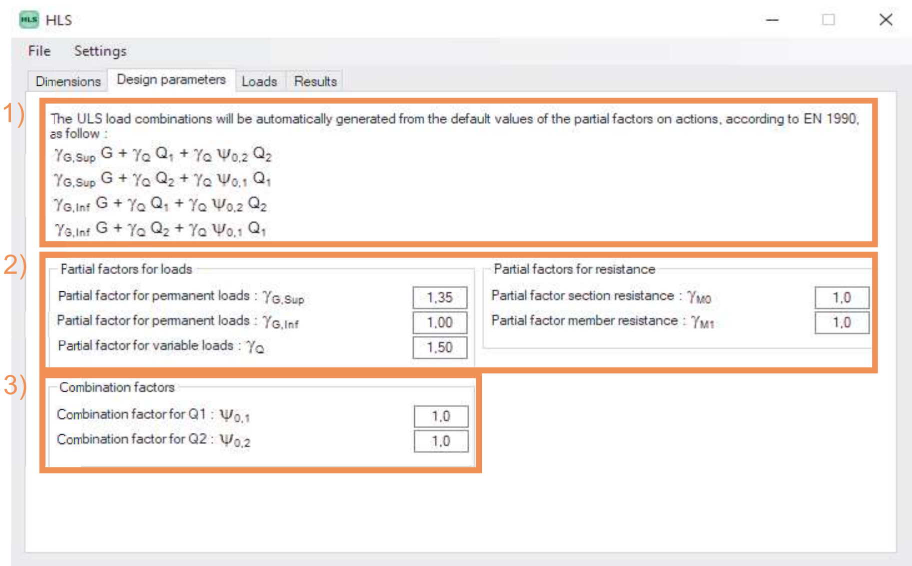

Fig. 32 Tab "Design Parameters"

Next, Fig. 33 and Fig. 34 represent the "Loads" tab. In zone1), the user can chose the load case for which the loads are defined. The load cases are combined according to the load combinations defined in the "Design parameters" tab. For each load case, the use can define:

- an axial force (positive indicates compression);
- loads generating a major-axis bending moment (acting about y-y axis): uniformly distributed load, point load at mid-span and end moments. The major-axis bending moment diagram is represented in zone 3) of the "Loads" tab as shown in Fig. 33. If

the sign of the moment M_y is positive, compression stresses are induced in the upper flange (z-positive – see zone 7) of Fig. 28);

- loads generating a minor-axis bending moment (acting about z-z axis): uniformly distributed load, point load at mid-span and end moments. The minor-axis bending moment diagram is represented in zone 4) of the "Loads" tab as shown in Fig. 33. If the sign of the moment M_z is positive, compression stresses are induced in the web at the right-hand side (y-positive – see zone 7) of Fig. 28).

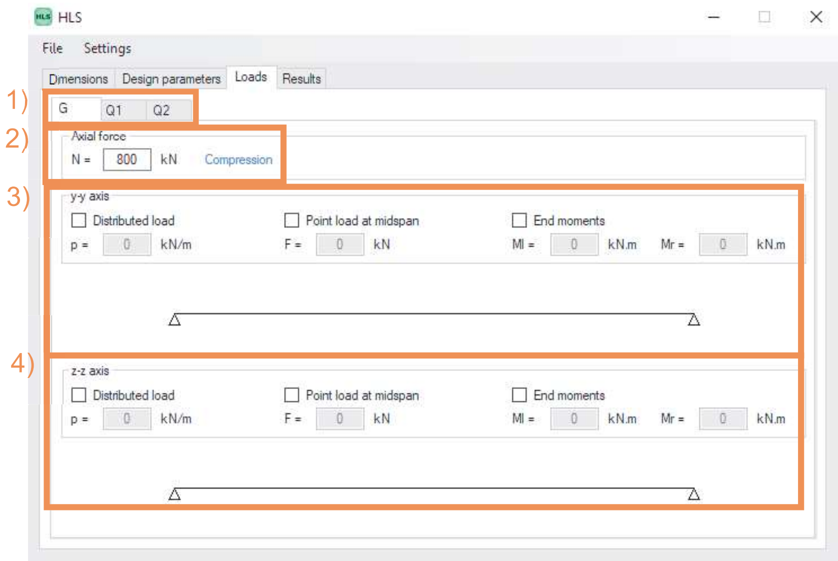

Fig. 33 Tab "Loads" – Permanent loads "G"

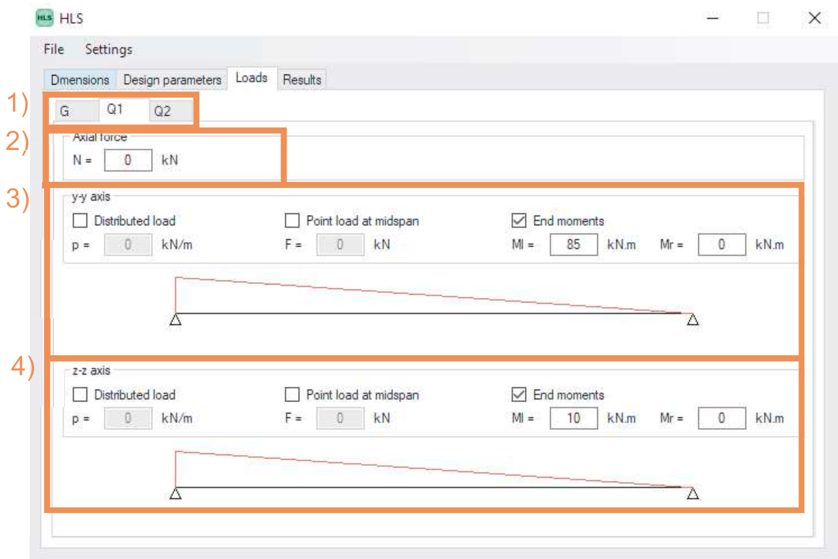

Fig. 34 Tab "Loads" – Variable loads "Q_1"

Finally, after having defined the design case (cross-section dimensions, material properties, member length, load combinations and loads), the user can launch the calculations and display the results in the

“Results” tab. Before the calculations are finished, only the button “Calculation” is active in this tab as shown in Fig. 35.

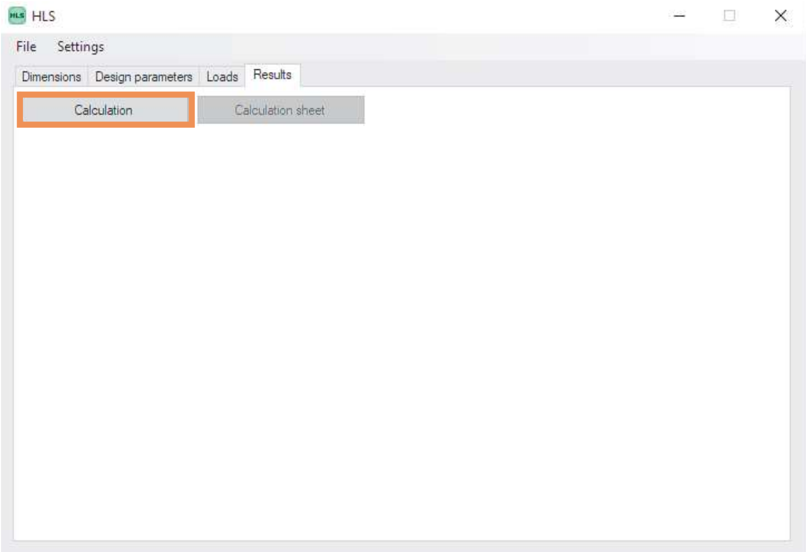

Fig. 35 Tab “Results”

After the execution of the calculations, the main results are summarised in the “Results” tab represented in Fig. 36. By default, the design verification is displayed for the most severe situation (combination, most severe position along the member for local resistance) in zone 5) and zone 6) of the “Results” tab. Nonetheless, the user may display the main results obtained for the other load combinations and positions along the member by modifying the selection in zone 2). In zone 4) of the “Results” tab, the interface proposes, according to the user’s choice, a graphical representation of the mode shape for local instability (see Fig. 36), the elastic stress distribution (see Fig. 37) or the plastic stress distribution (see Fig. 38). It should be noted that the stresses indicated in the elastic stress distribution are those induced by the design loads of the chosen load combination. Consequently, the maximum stress is in general not equal to the yield strength. Conversely, the stresses displayed in plastic stress distribution are equal to $\pm f_y$ apart from small zones around the plastic neutral axis that are not yielded. The extension of these not yielded zones mainly depends on the calculation tolerance fixed in the “Calculation parameters” window (see Fig. 29).

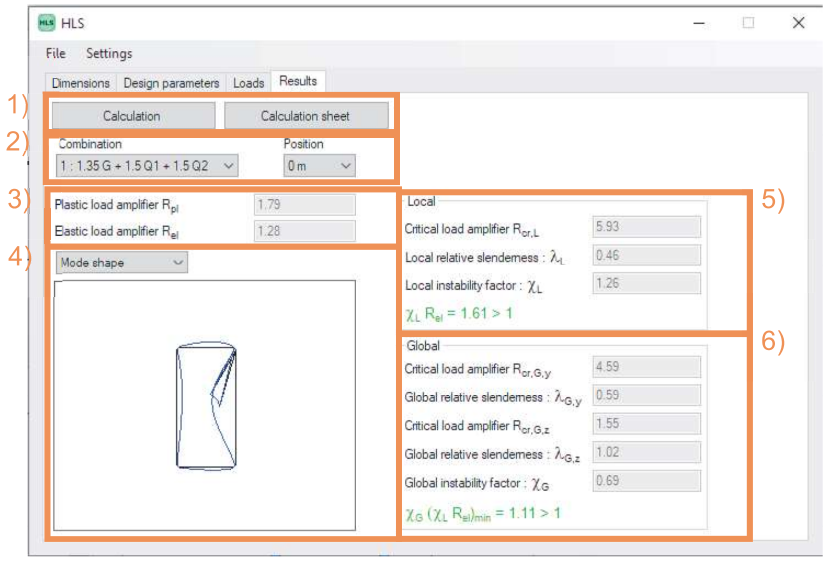

Fig. 36 Tab "Results" – Summary of the results

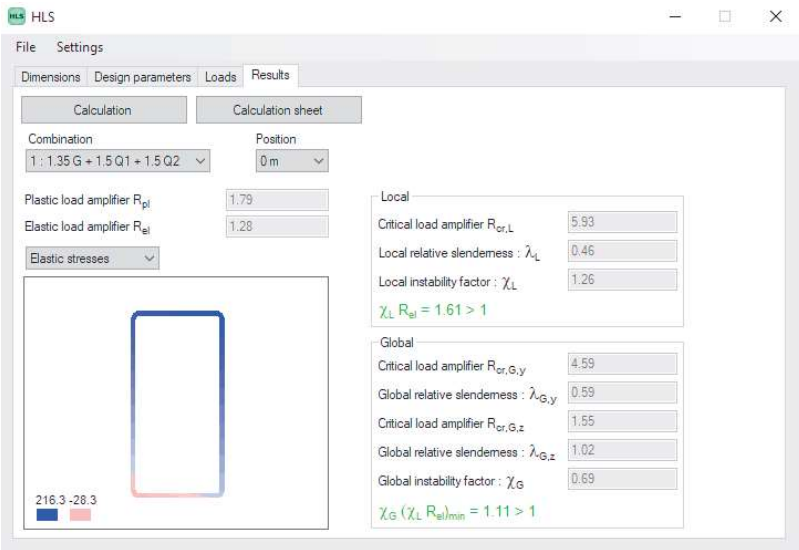

Fig. 37 Tab "Results" – Representation of elastic stress distribution

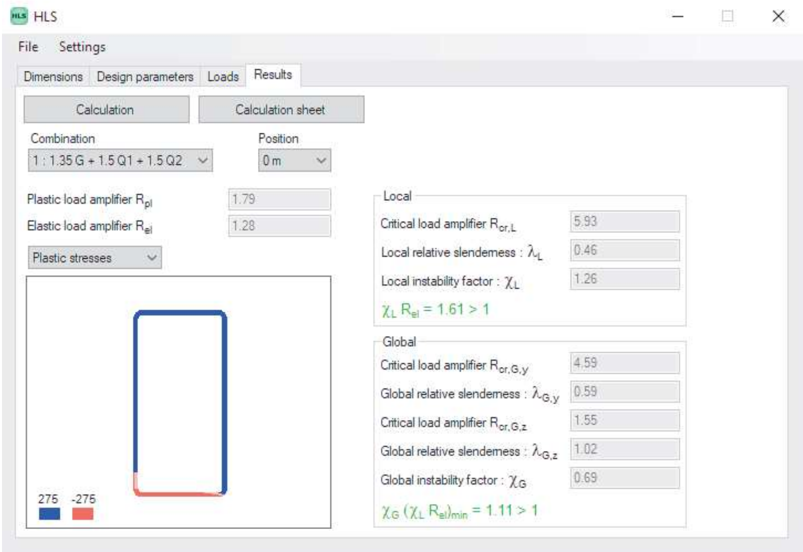

Fig. 38 Tab “Results” – Representation of plastic stress distribution

Finally, the user may open and print a calculation sheet by clicking on the button “Calculation sheet” (zone 1) of Fig. 36. This calculations sheet summarises the main input parameters (cross-section, material parameters, loads, etc.) and the results obtained for the most severe load combination.