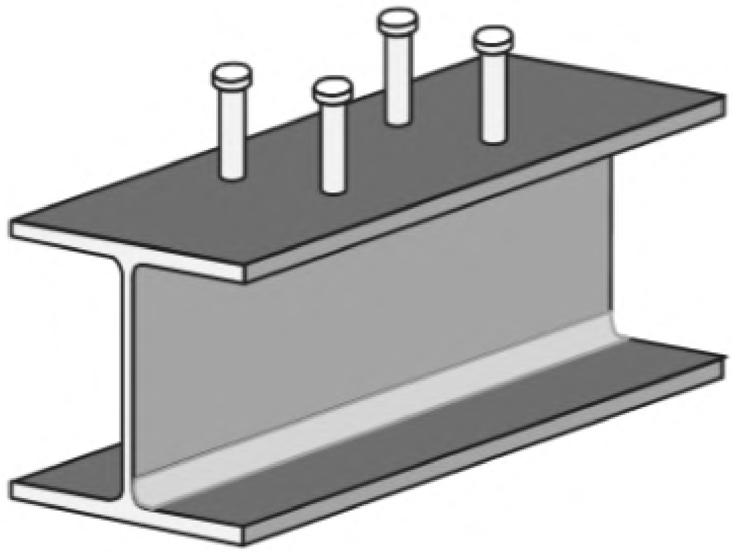
COMBEAMS - User Manual

v.1.0.0

Jorge Tamayo, Lucas Aguiar, Cristian de Campos,
Daniel Matos, Inácio Morcsh

June 24, 2024



Introduction

COMBEAMS is a computer program written in Python for the design of shear stud connectors commonly located at the interface of steel-concrete composite beams. This manual provides an overview of how the program, which comes with the same name, can be used by the user.

Contents

List of acronyms

Ass – Positive reinforcement area
Asi – Negative reinforcement area
b_f – Steel beam flange width
c – Concrete covering
d – Steel beam height
d' – Beam web height
E_a – Elastic modulus or Young's modulus of steel beam
E_{Vc} – Elastic modulus or Young's modulus of reinforcement bars
f_c – Yield strength of concrete
f_s – Yield strength of reinforcement bars
f_{ucs} – Yield strength of studs
f_y – Yield strength of steel beam
h – Distance between top and bottom flange
I_x – Inertia of the cross section in x
L – Span length
M_u – Ultimate moment
M_r – Resistance moment
Q – Uniformly distributed load
r_y – Radius of rotation
t_c – Concrete slab height
t_f – Steel beam flange width
t_w – Steel beam web width
V_r – Resistance shear
V_u – Ultimate shear
W_x – Elastic resistance modulus
ϕ – Studs diameter
γ_c – Safety factor
γ_y – Safety factor of concrete
γ_a – Safety factor of steel beam
γ_s – Safety factor of reinforcement bars

1 COMBEAMS Program Overview

The program was developed through computational code written in the Python language, version 3.8.8. The User Interface (GUI – Graphical User Interface), depicted in Figure 1, was crafted using the Tkinter library. The choice of this library was driven by its excellent portability, enabling routines implemented with Tkinter to be seamlessly transferable across Linux, Unix, Windows, and Mac operating systems.

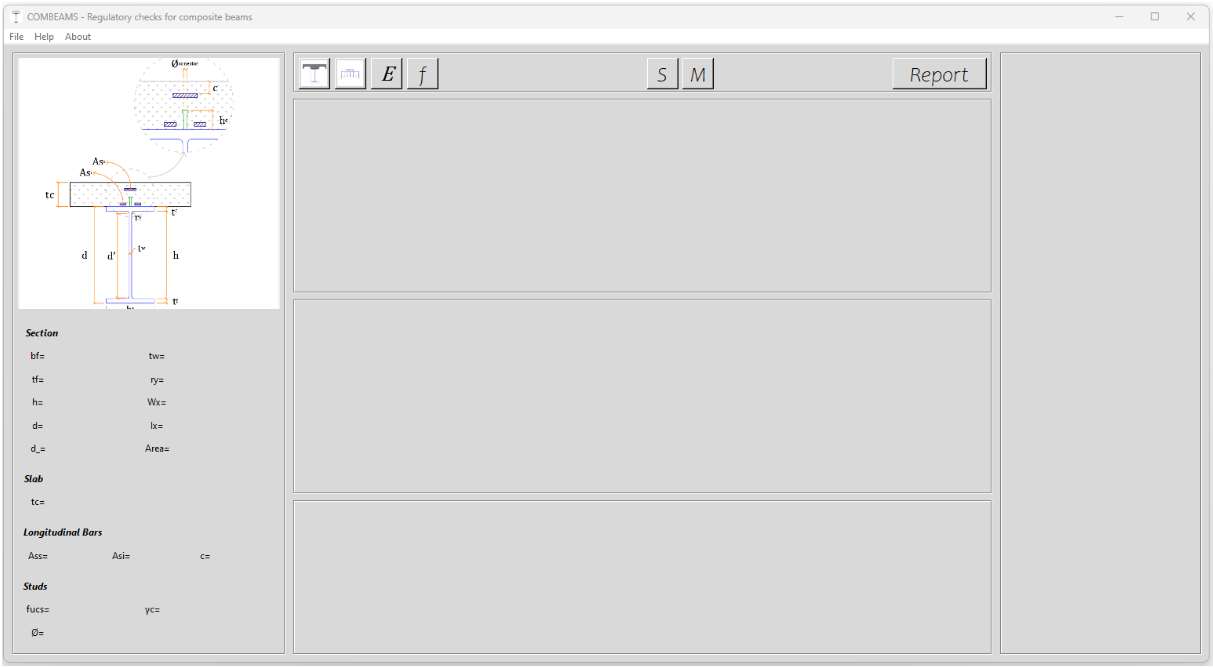


Figure 1: Overview of the main program window.

The main window has 6 different panels, as shown in Figure 2:

A **Section Details and Connectors Characteristics**
The 'A' panel exposes users to details of the section, longitudinal reinforcement, and characteristics of the considered connectors;

B **Program Functions and Controls**
The 'B' panel presents buttons that provide access to the program's functions, including data input and solution buttons;

C **Internal Forces or Moments Diagrams**
The 'C' panel is responsible for presenting users with diagrams of forces, including shear and bending moment.

D **Internal Resisting Forces or Moments Diagrams**
The 'D' panel showcases diagrams of resisting forces, already weighted and combined for shear and bending moment;

E **Graphic Display of Connector Arrangement and Forces Diagrams**
Panel 'E' visually displays the arrangement of connectors and overlays internal diagrams forces for a comprehensive visual understanding;

F **Numerical Results for Each Section**
The 'F' panel provides numerical results for each section between zero and maximum moment, including the number of connectors, spacing, calculated resisting moment, calculated moment, calculated resisting shear force, and calculated shear force.

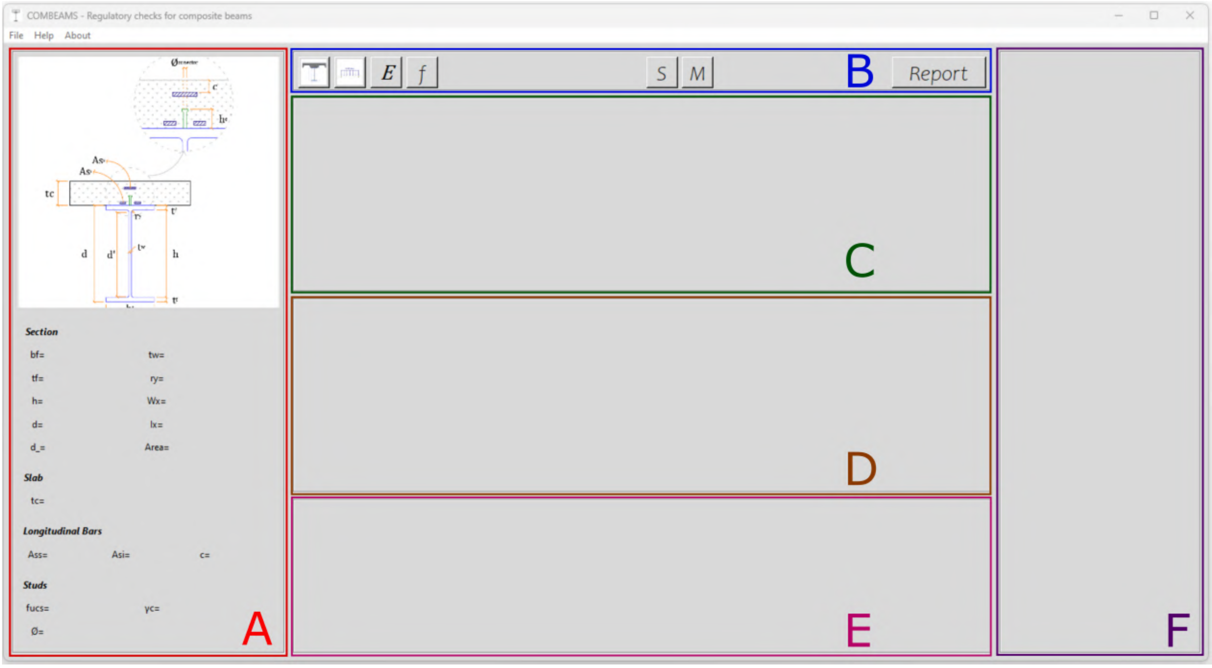


Figure 2: Main window panes.

The next section will detail how the user can use the program interface.

2 Using COMBEAMS program

2.1 Program Start-up

To run COMBEAMS Program version v1.0.0:

1. Access the Python code using a text editor or compiler;
2. Run the file in the Python code compiler.

> To execute the program, you will need a Python version 3.8.8 or higher installed and have installed the necessary Python libraries as outlined in the following sections.

2.2 Required python libraries

This program requires that some Python libraries are previously installed on the computer:

1. tkinter;
2. pandas;
3. numpy;
4. xlswriter;
5. subprocess;
6. sys;
7. matplotlib.

2.3 Initial Use

The goal of this section is to give the user an overview of the application, covering the basic aspects step-by-step to set up a model that can be used to analyze beam behavior.

2.3.1 Input Data

1. The initial step involves navigating to panel B in the main program window (see Figure 2) and clicking on the beam cross-section selection button (first button), as illustrated in Figure 3). Subsequently, a window will open, providing the option to either select a pre-defined I-section steel beam (2) or manually input the measurements for the beam cross-section (3);
2. In the secondary window, complete with information about the concrete slab, longitudinal reinforcement and connectors (Figure 4);
3. The following step is to input beam span information. To achieve this, navigate using the second button on panel B. Enter details about the number of beam spans in the secondary window, followed by the span length (L) and the corresponding distributed load (Q), as shown in Figure 5;
4. In Panel B, accessed through the third button, you can input the material properties for the concrete slab, the metal beam, and the reinforcement bars, as illustrated in Figure 6;

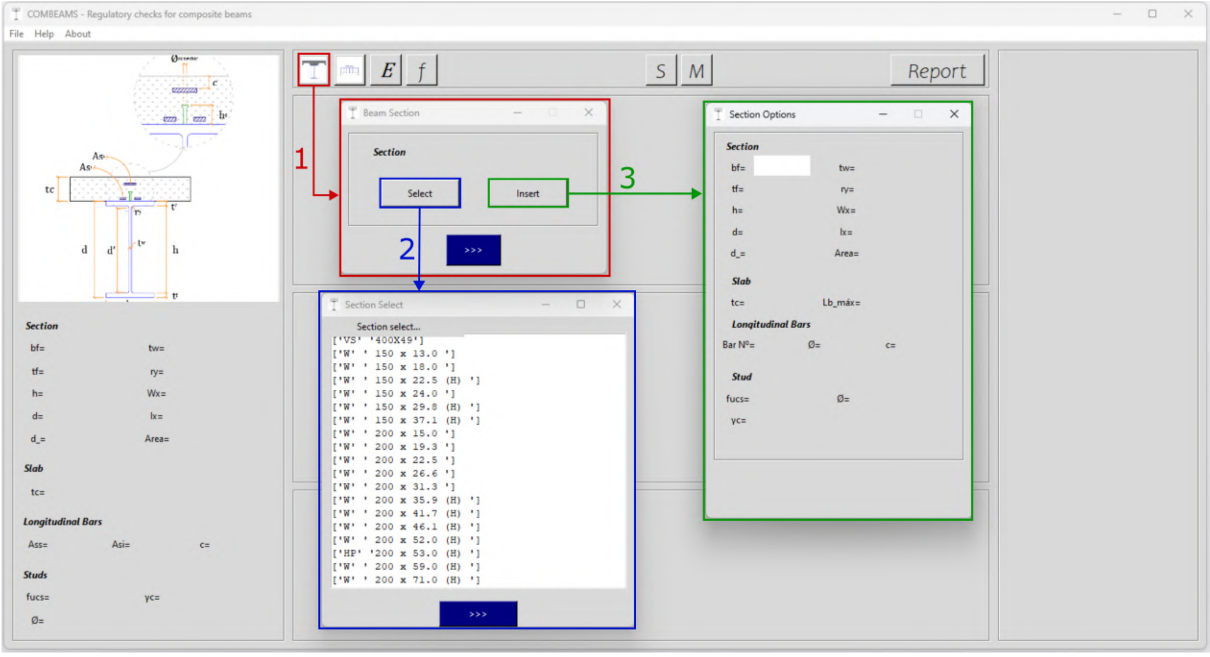


Figure 3: First stage steps.

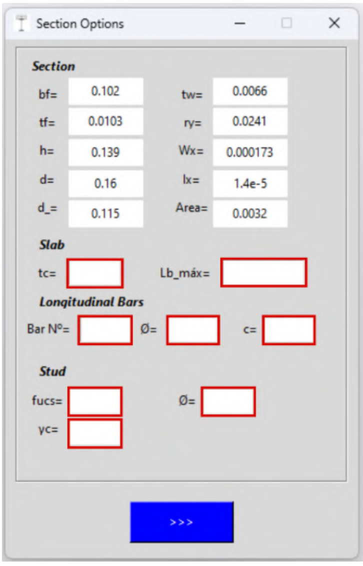


Figure 4: Second stage steps.

5. To complete the data entry process, click the fourth button on Panel B to open windows for selecting the design verification method and specifying the verification type, whether Partial or Complete, as depicted in Figure 7. If it is partial, you must enter the degree of connection;

2.3.2 Check display

After inserting the input data, Panel A within the main program window (Figure 2) displays the values entered in preceding steps. Consequently, it is crucial for the user to

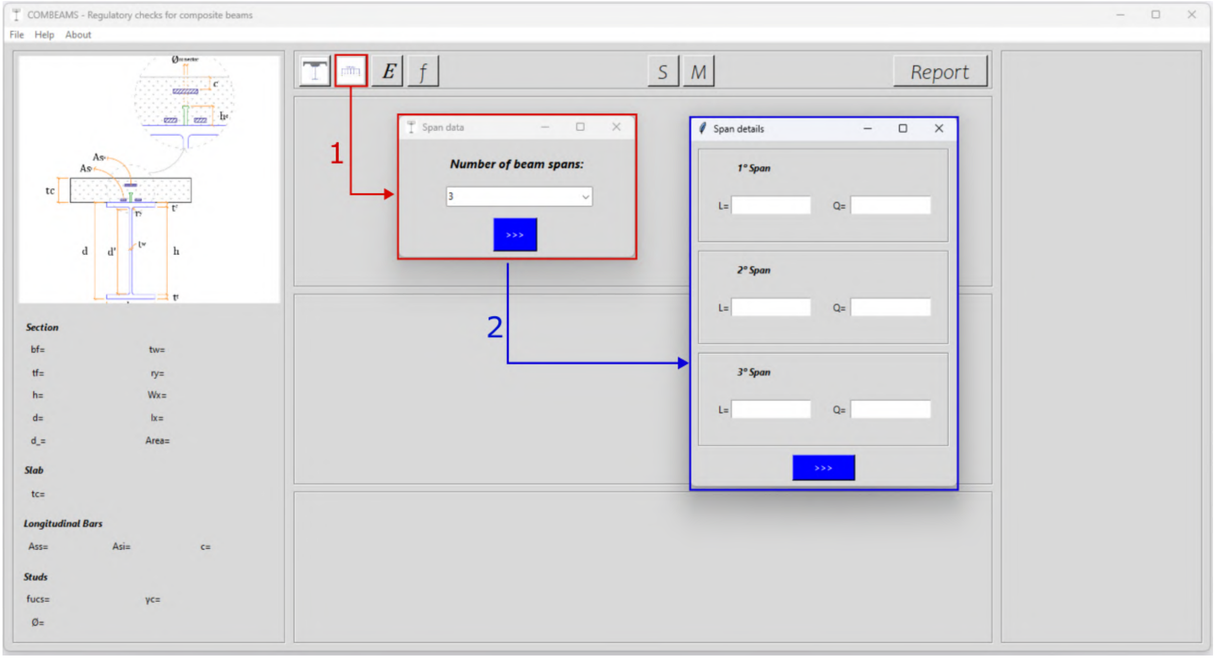


Figure 5: Third stage steps.

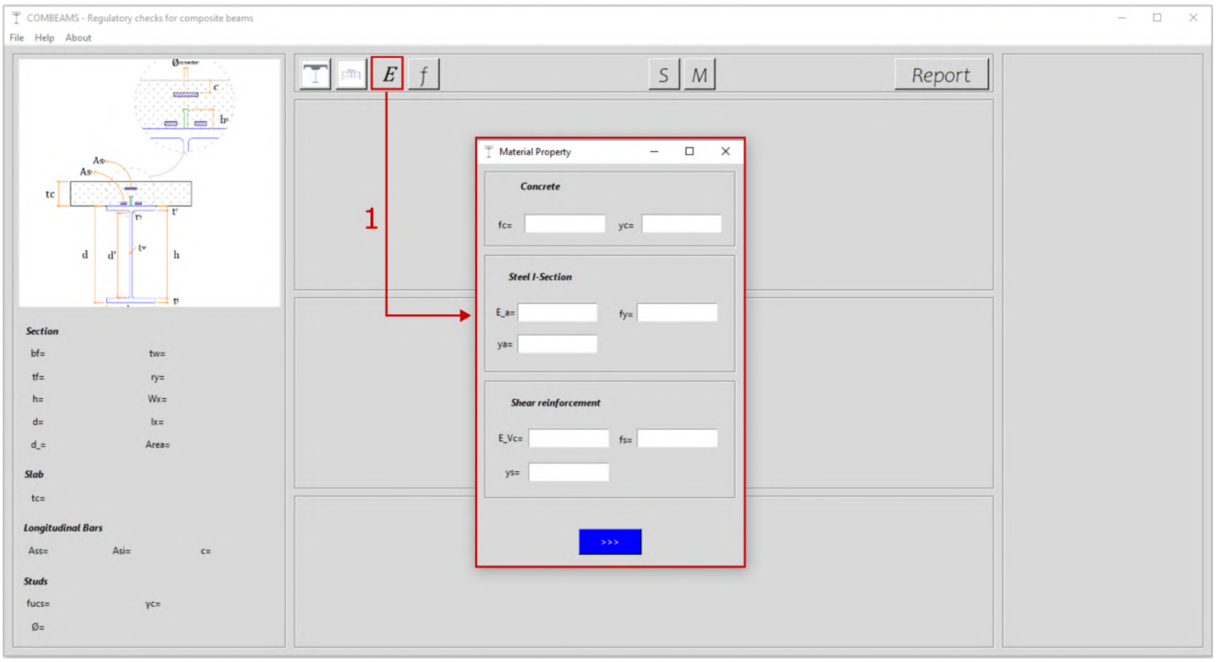


Figure 6: Fourth stage steps.

review the displayed values in this section to ensure the desired outcome during the solution process.

As an example, figure 8 shows the panel properly filled out.

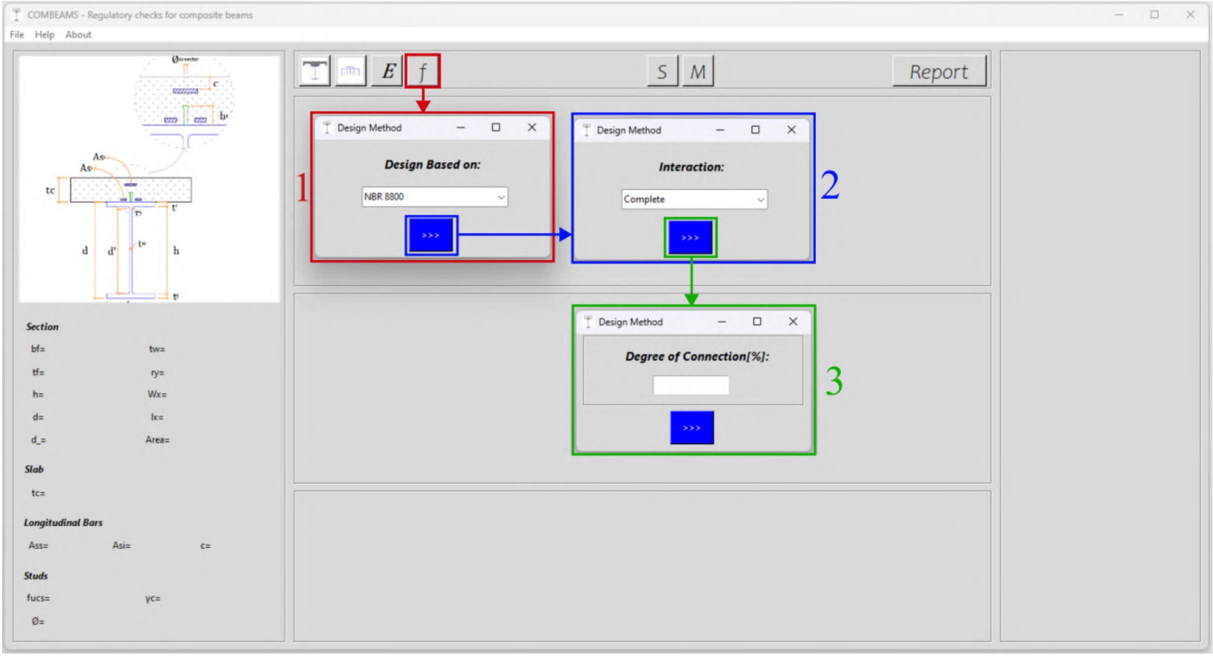


Figure 7: Fifth stage steps.

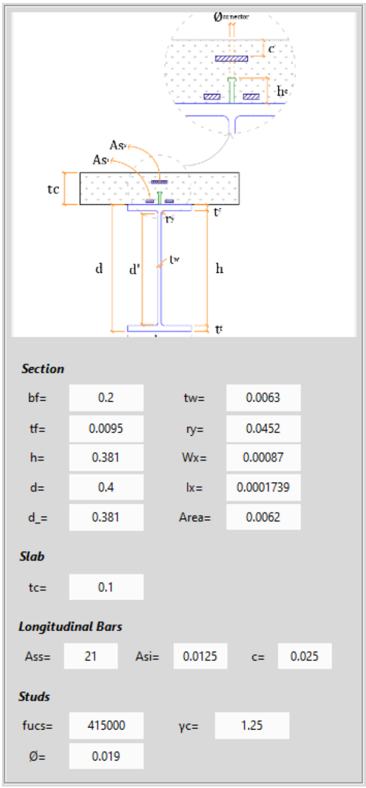


Figure 8: Initial input data display view (Panel A)

2.3.3 Applying the Solution

To initiate the solution, use the central buttons located on panel B. Specifically, press the "S" button to initiate the analysis process associated with the cutting edge. Upon activating this function, panels C, D, E, and F will simultaneously activate, displaying the comprehensive results of the analysis. For a visual representation, refer to Figure 9, which illustrates the results window following the activation of the "S" analysis button.

Figure 9: Results generated by shear force analysis

Press the "M" button to initiate the analysis process related to the bending moment. Upon pressing the second results button, panels C, D, E, and F will display their respective data, as detailed in the preceding chapter. For a practical illustration, consult Figure 10, showcasing the results of an example beam after activating the "M" solution button.

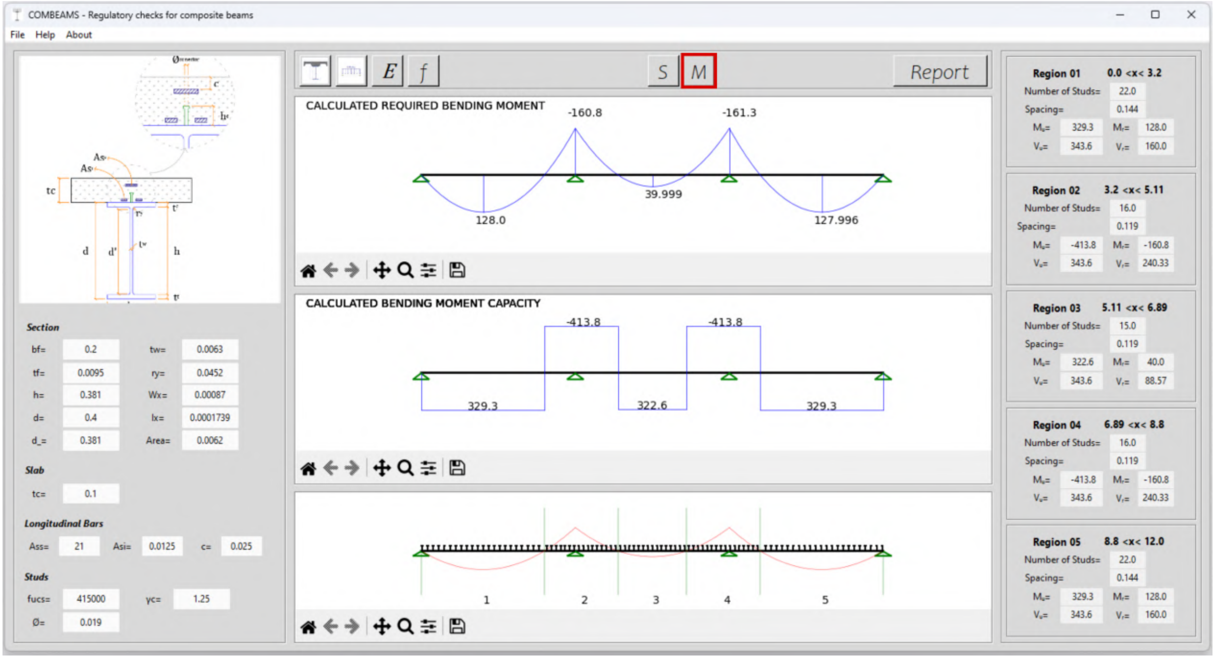


Figure 10: Results generated by moment force analysis

2.4 Quick start

The importing data into the program is facilitated by reading a file in .xlsx format. Follow these steps for a seamless process:

1. Navigate to the toolbar and select 'File';
2. Click on 'Open';
3. Enter the file name, followed by '.xlsx.'

> The file should either be located in the same folder as the program or its complete path address must be specified.

4. Verify the correct loading of the file by checking panel A in the main program window.

Refer to Figure 11 for the input data file configuration; simply input the relevant values and proceed with the aforementioned steps.

2.5 Report

The 'Report' button, located on Panel B, serves the purpose of exporting and saving the results in a .xlsx file format. This feature becomes active only after the results have been analyzed. Upon clicking this button, the software generates a .xlsx file in the same folder (path) as the main code.

This file comprehensively outlines the initial design parameters of the beam, covering crucial aspects related to the results. These include classification based on slenderness, resistance properties, shear resistance, resistance to negative and positive moments, and the distribution of connectors. In conclusion, the report provides a summarized evaluation, indicating whether the sizing is deemed adequate or not.

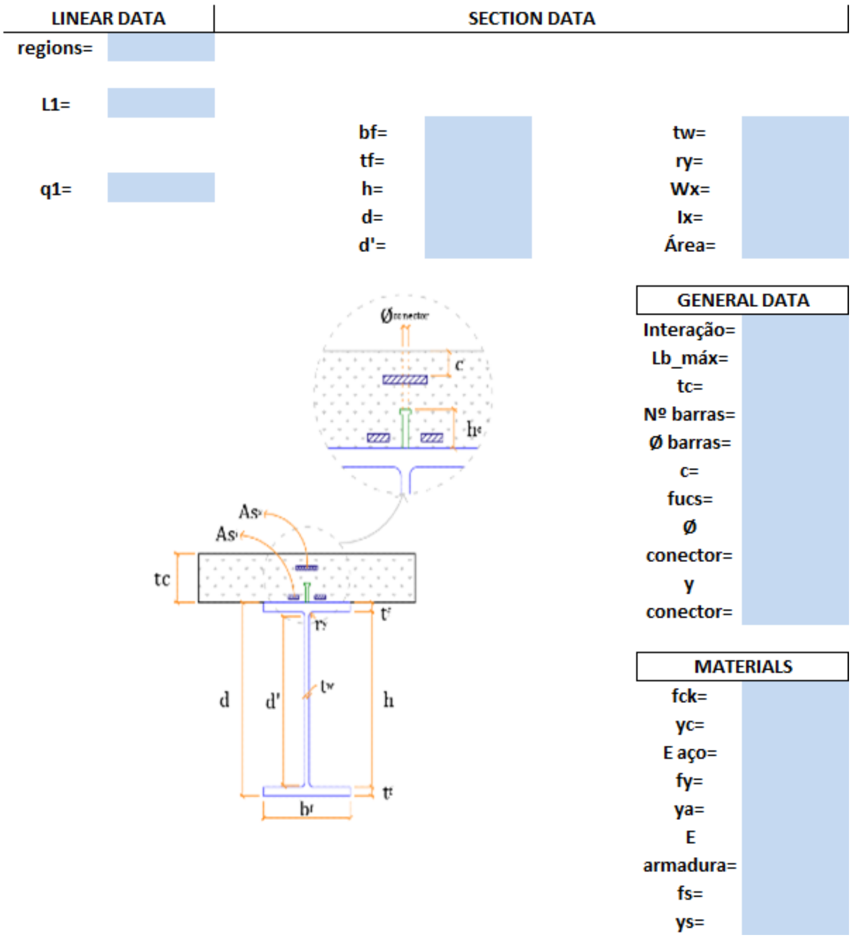


Figure 11: Input data file

3 Examples

This section will illustrate the practical application of the software through two examples. The first example showcases the analysis of a composite beam with a single span, while the second example delves into the complexities of a composite beam with three spans.

3.1 Example 01 – Single Span Beam

For our first example, we will analyze a simply supported beam with a single span of 5.9 meters under a load of 9.52 kN/m, depicted in Figure 12. To guide you through the complete analysis of this case, follow the steps outlined below:

1. Run the code using a compiler, as detailed in the subsection 2.1, using the data in the Figure 13;

2. Access the options bar, navigate to 'File' -> 'Open,' and input the name of the .xlsx file, as instructed in the subsection 2.4;

3. Initiate the analyses by heading to Panel B. Initially, examine the results relative to constant effort by selecting the 'S' button. The remaining panels in the main program window will automatically auto-fill, as demonstrated in Figure 14;

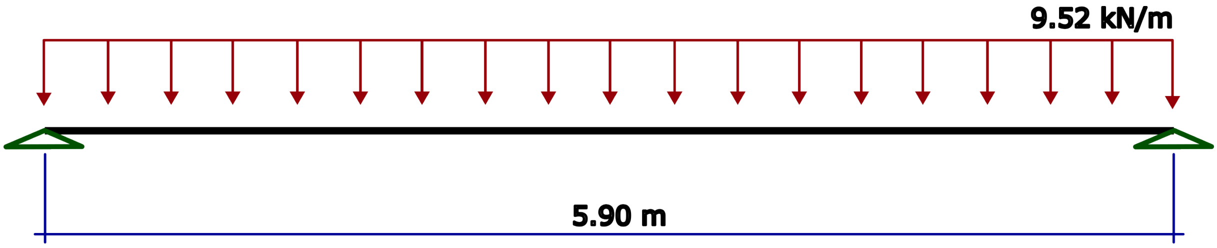


Figure 12: Initial conception - example 01

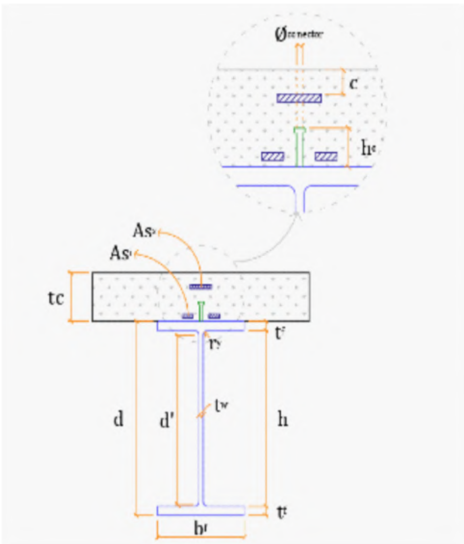
4. For the subsequent analysis concerning bending moments, click the 'M' button on Panel B. Similar to the previous step, the panels will auto-fill with results, illustrated in Figure 15;
5. Save the results either through the navigaton panel ('File' -> 'Save') or by using the 'Report' button in Panel B, as explained in the subsection 2.5;
6. Upon opening the .xlsx report file, review the final spacing conditions and connector details conveniently summarized at the document's conclusion, as shows in Figure 16;
7. When opening the final report, the user will come across the documents in .xlsx format as shown in Annex A.

LINEAR DATA	
regions=	1
L1=	5,9
q1=	9,5276

SECTION DATA

NBR 8800

bf=	0,133	tw=	0,00584
tf=	0,00782	ry=	0
h=	0,18736	Wx=	0,0002268
d=	0,203	Ix=	0,00002302
d'=0,18736		Área=	0,0031741



GENERAL DATA	
Interação=	Complete
Lb_máx=	1
tc=	0,07
Nº barras=	5
Ø barras=	0,006
c=	0,025
fucs=	415000
Ø conector=	0,0126
y conector=	1

MATERIALS	
fck=	31000
yc=	1
E aço=	200000000
fy=	265000
ya=	1
E armadura=	200000000
fs=	250000
ys=	1

Figure 13: Input data – example 01

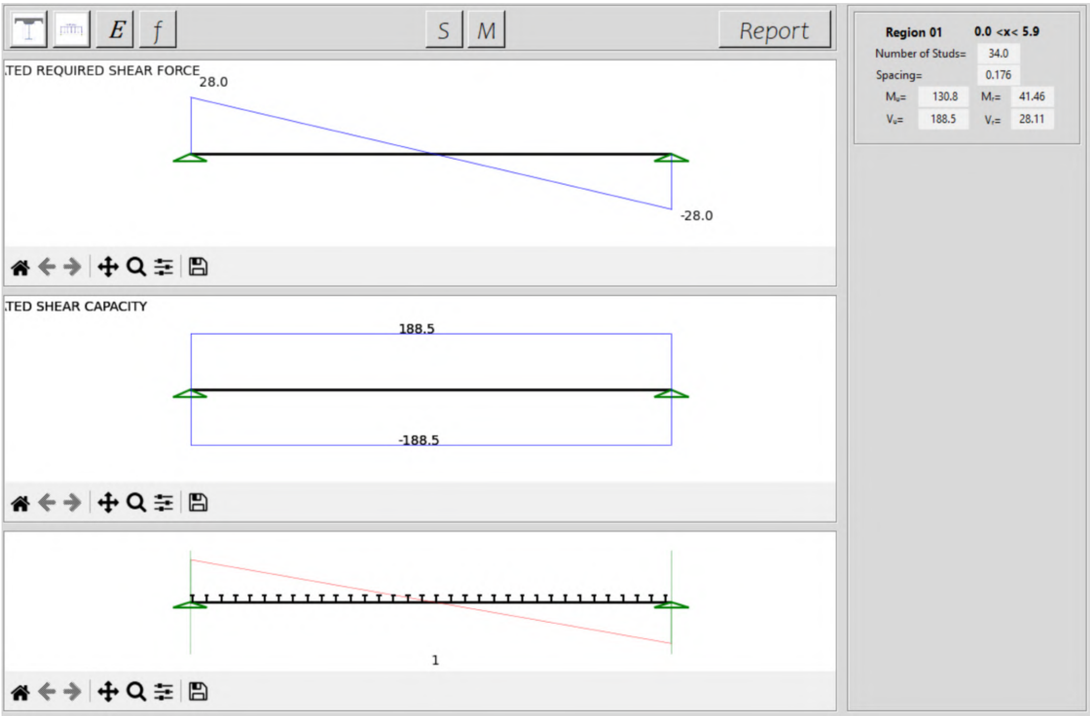


Figure 14: Exemple 01 shear force analysis

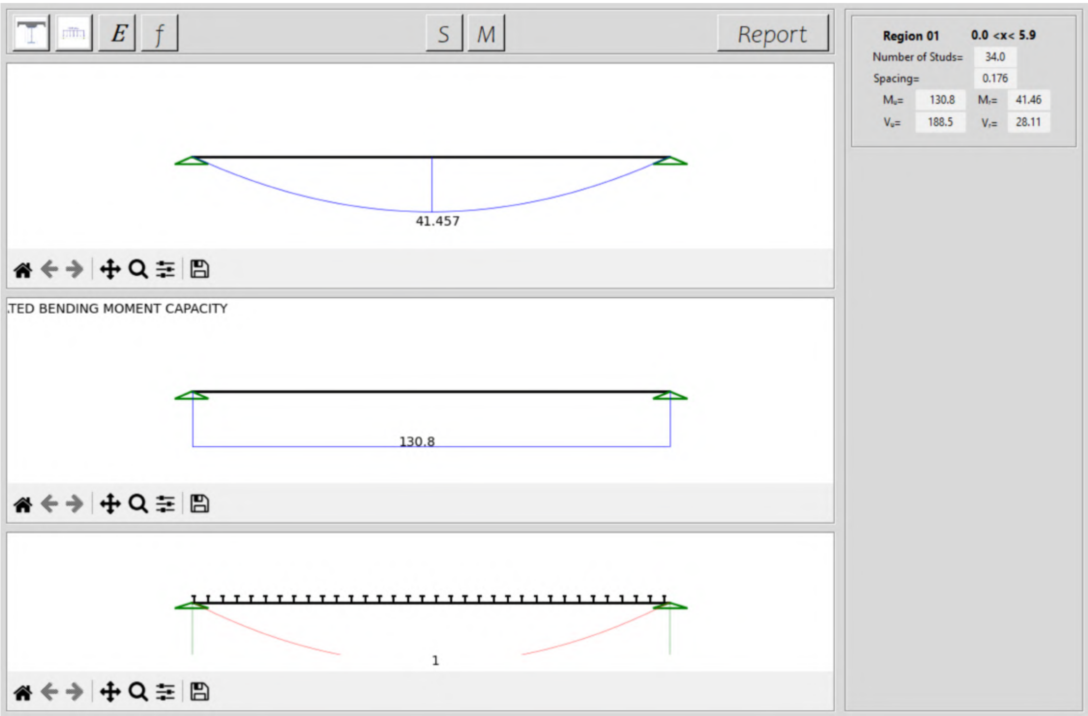


Figure 15: Example 01 moment force analysis

CHECKS

Bending Moment			
Regions	Mrd	Msd	Status
Region 0	130,83	41,46	OK!

Shear required			
Region	Vrd	Vsd	Status
Region 0	188,5	28,11	OK!

Figure 16: Final result checks of example 01

3.2 Example 02 – Three Span Beam

For our second example, we will analyze a beam with a length of 12 meters, divided into three spans of 4 meters each. All spans are subjected to a uniformly distributed load of 100 kN/m, as depicted in Figure 17. Follow the steps outlined below for this analysis:

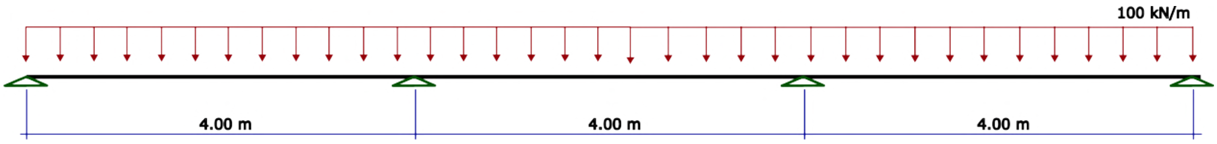


Figure 17: Initial conception – example 02

1. Run the code using a compiler, as detailed in the subsection 2.1, using the data in the Figure 18;

2. Access the options bar, navigate to 'File' -> 'Open,' and input the name of the .xlsx file, as instructed in the subsection 2.4;

> The difference with example 01 is when filling in the input data file. In example 02, the measurements of the three spans must be filled in as well as the load to which they are subjected (this procedure is highlighted in Figure18);

3. Initiate the analyses by heading to Panel B. Initially, examine the results pertaining to constant effort by selecting the 'S' button. The remaining panels in the main program window will automatically auto-fill, as demonstrated in Figure 19;

4. For the subsequent analysis concerning bending moments, click the 'M' button on Panel B. Similar to the previous step, the panels will auto-fill with results, illustrated in Figure 20;

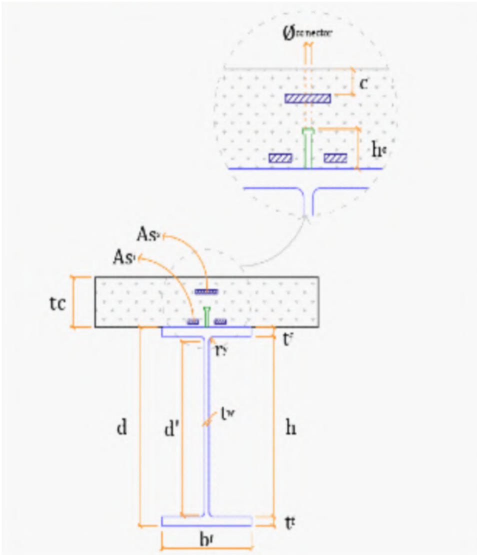
5. Save the results either through the navigation panel ('File' -> 'Save') or by using the 'Report' button in Panel B, as explained in the subsection 2.5;

6. Upon opening the .xlsx report file, review the final spacing conditions and connector details conveniently summarized at the document's conclusion, as shows in Figure 21;

7. When opening the report generated by item 6, the document will be similar to the one presented in Annex A. However, with the result information of this example.

In the event of an error, as indicated in the report in example 2, a window will appear on the screen indicating the type of error and possible measures that can be taken, as shown in the Figure 22.

LINEAR DATA		STEEL SECTION DATA			
Regions=	3				
		NBR 8800			
L1=	4				
L2=	4	bf=	0,2	tw=	0,0063
L3=	4	tf=	0,0095	ry=	0,0452
q1=	100	h=	0,381	Wx=	0,00087
q2=	100	d=	0,4	Ix=	1,74E-04
q3=	100	d'=	0,381	Área=	0,0062



GENERAL DATA	
Iteraction	Complete
Lb máx=	1
tc=	0,1
Nº of bars=	21
Ø bar=	0,0125
c=	0,025
fucs=	415000
Ø studs=	0,019
γ studs=	1,25

MATERIALS	
fck=	20000
γc=	1,4
E steel=	210000000
fy=	250000
γa=	1,1
E reinforcement=	210000000
fs=	500000
γs=	1,1

Figure 18: Input data – example 02

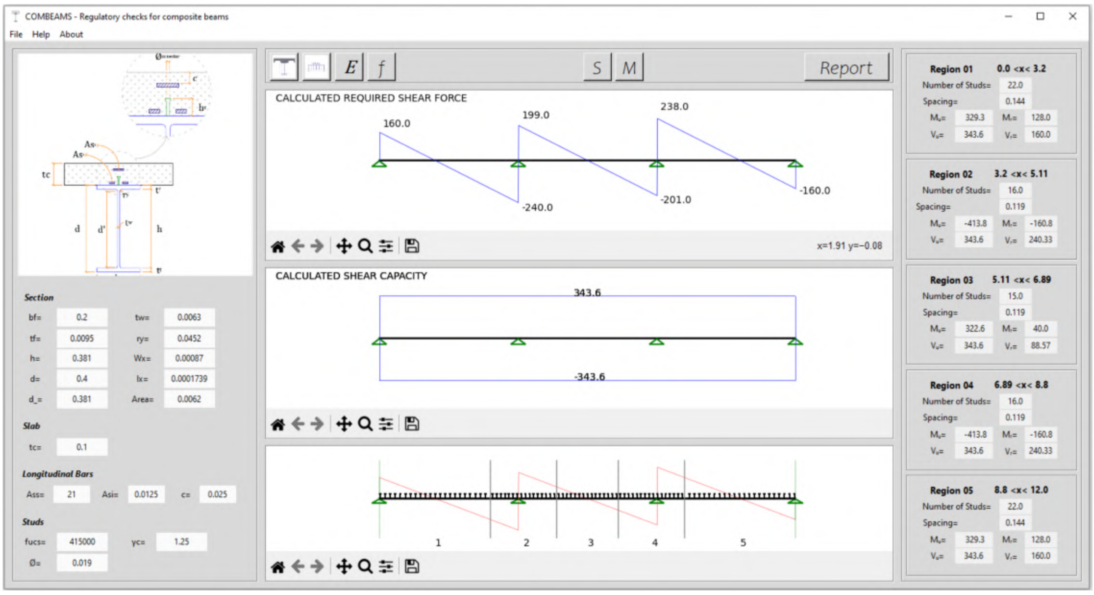


Figure 19: Exemple 02 shear force analysis

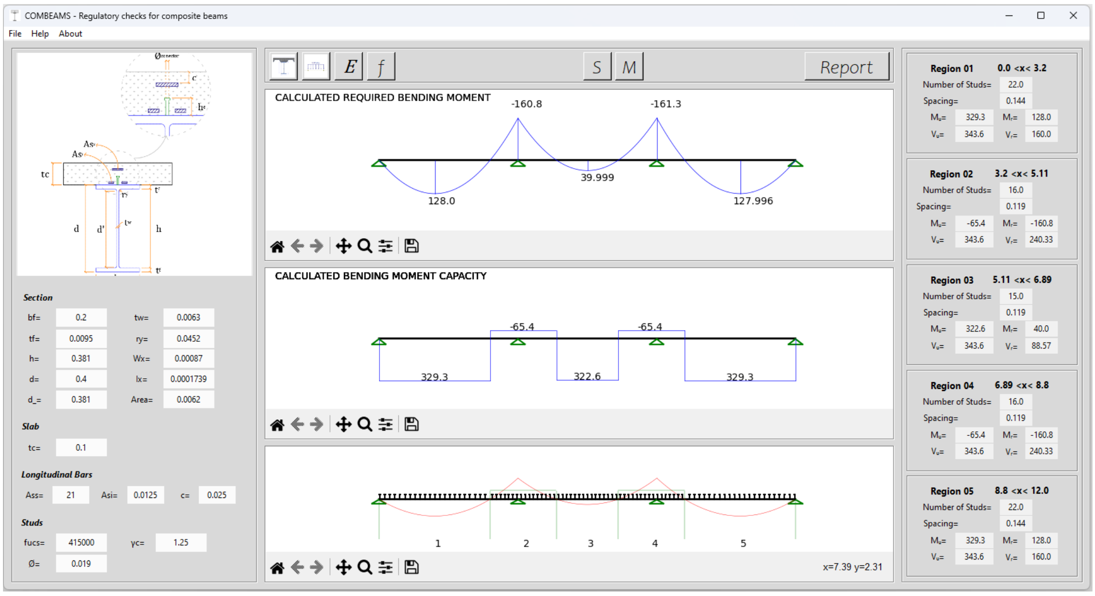


Figure 20: Example 02 moment force analysis

CHECKS

Bending Moment			
Regions	**Mrd**	**Msd**	**Status**
Region 0	329,33	128	OK!
Region 1	-65,43	-160,8	FAILURE!
Region 2	322,6	40	OK!
Region 3	-65,43	-160,8	FAILURE!
Region 4	329,33	128	OK!

Shear required			
Region	**Vrd**	**Vsd**	**Status**
Region 0	343,64	160	OK!
Region 1	343,64	240,33	OK!
Region 2	343,64	88,57	OK!
Region 3	343,64	240,33	OK!
Region 4	343,64	160	OK!

Figure 21: Final result checks of example 02

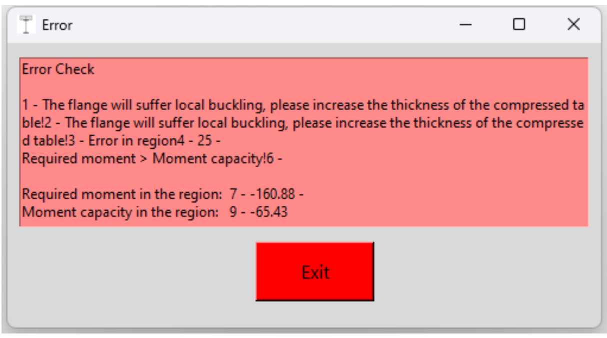
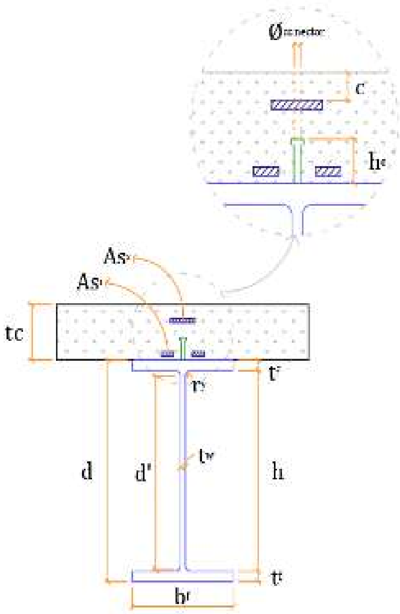


Figure 22: Example 2 error window

Annex A

LINEAR DATA		SECTION DATA			
regions=	1				
		NBR 8800			
L1=	5,9				
		bf=	0,133	**tw=**	0,00584
		tf=	0,00782	**ry=**	0
q1=	9,5276	**h=**	0,18736	**Wx=**	0,0002268
		d=	0,203	**Ix=**	0,00002302
		d'=	0,18736	**Área=**	0,0031741



GENERAL DATA	
Interação=	Complete
Lb_máx=	1
tc=	0,07
Nº barras=	5
Ø barras=	0,006
c=	0,025
fucs=	415000
Ø conector=	0,0126
y conector=	1

MATERIALS	
fck=	31000
yc=	1
E aço=	200000000
fy=	265000
ya=	1
E armadura=	200000000
fs=	250000
ys=	1

OTHERS							
hf=	0	**fcd=**	31000	**E concreto=**	26502558,4	**alpha e=**	7,54644126
fyd=	265000	**fsd=**	250000	**Asl=**	0,00014137	**alpha f=**	0,94339623

EFFECTIVE WIDTH

L1 efetivo= 5,9

Lb 1 = **1**

CLASSIFICATION

compacta

λ=32 **λp= 103** **λr= 156**

PROPERTIES

Strength of Concrete Slab		Strength of steel section	
Ccd1=	1844,5	**Tad=**	841,1365

Section elastic modulus of resistance mixed transverse in relation to the bending axis higher than CG.		Section elastic modulus of resistance mixed transverse in relation to the bending axis lower than CG.	
ws 1=	0,00101533	**wi 1=**	0,00034877

REQUIRED SHEAR

The required shear is given considering the resistance of the steel section!

k=	5	**λ=**	34,76
Vpl=	188,49768	**λp=**	67,57
Vrd=	**188,5**	**λr=**	84,16

POSITIVE MOMENT

Compressed section thickness				Moment Capacity	
Trecho 1	Neutral line in laje	**Hc=**	0,0319	**Mrd 1=**	**130,83**

NEGATIVE MOMENT

CHECKS

Bending Moment			
Regions	**Mrd**	**Msd**	**Status**
Region 0	130,83	41,46	OK!

Shear required			
Region	**Vrd**	**Vsd**	**Status**
Region 0	188,5	28,11	OK!

SHEAR STUDS

Regions	Qrd	Nº con.	Pitch	STUDS LIMATITION
Region 0	51,746	34	0,176	Plastification of steel section