1.5.4 4. Local verifications on steel

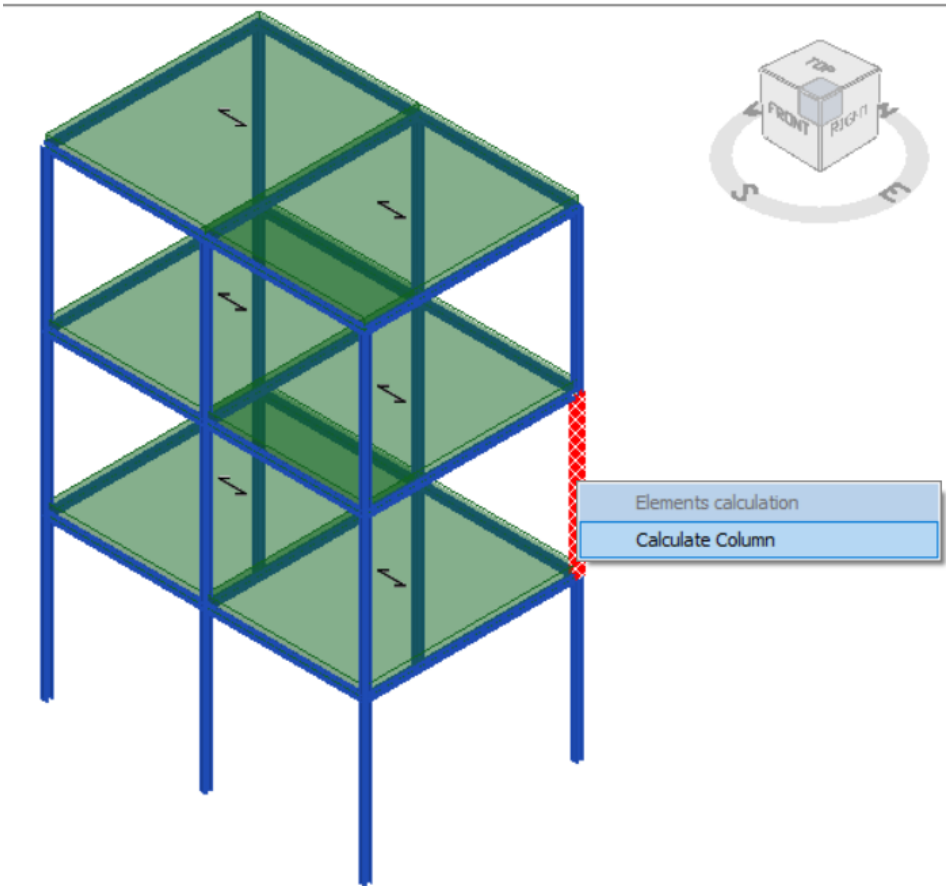
1.5.4.1 Steel Columns

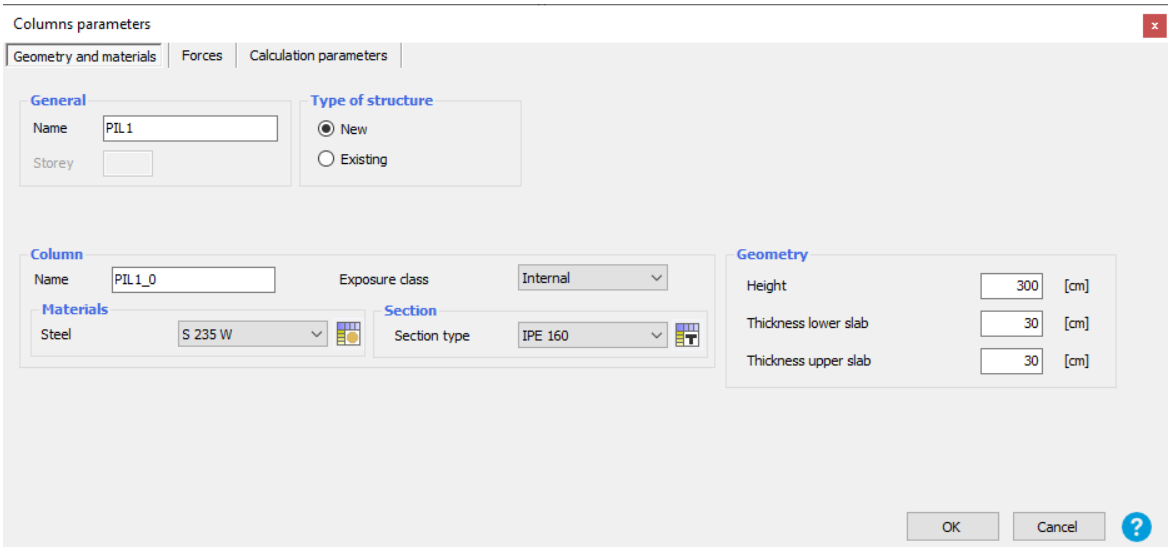
INPUT	3D MODEL	SINGLE ELEMENT
COLUMNS	✓	✓

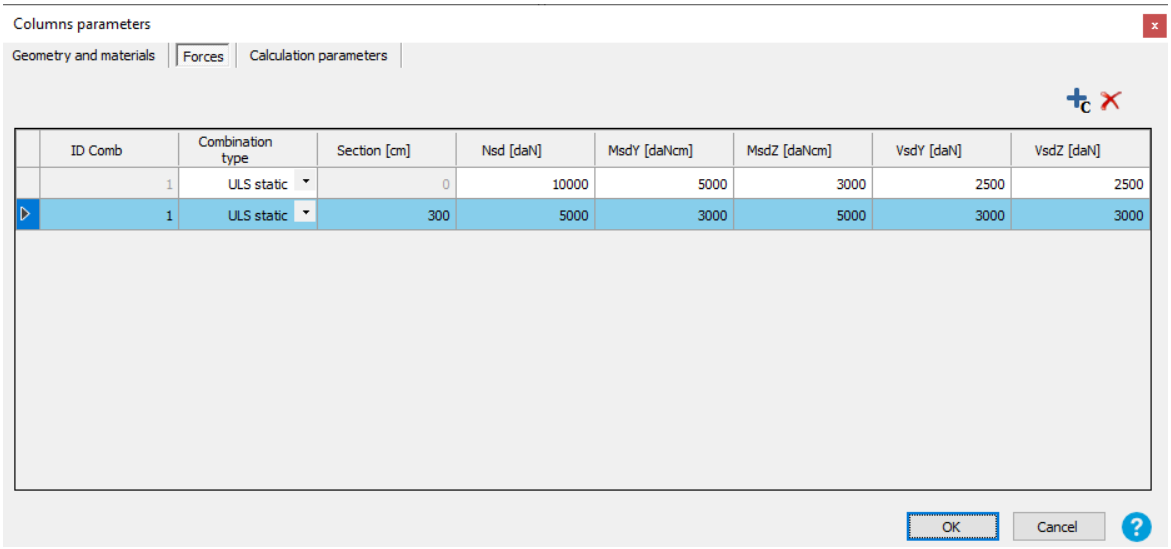
The Steel Column module allows the verification and / or design of new or existing steel columns, with the possibility to choose between different calculation methods and parameters

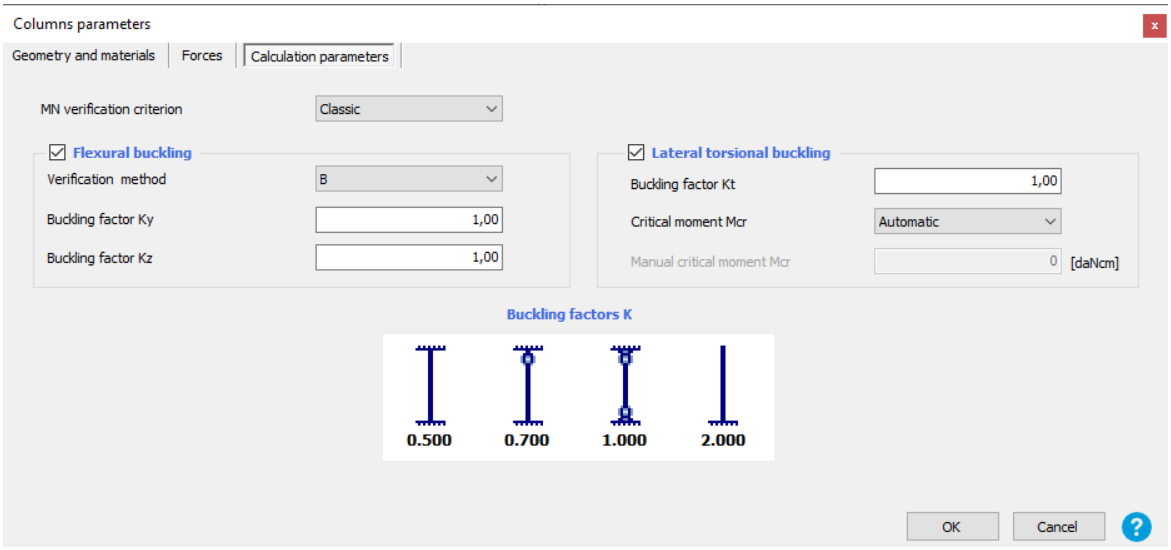
The environment dedicated to setting the parameters for the column presents the possibility to define the column's section, constraints, materials, and height, and to enter the loads applied.

All the data inserted by the user is shown in the table below which offers us a simple and clear graphic visibility. It is also possible to create a calculation tab from a column present within the model. To do this, simply right-click on the "Column" element present in the model and choose the "Calculate Column" option.







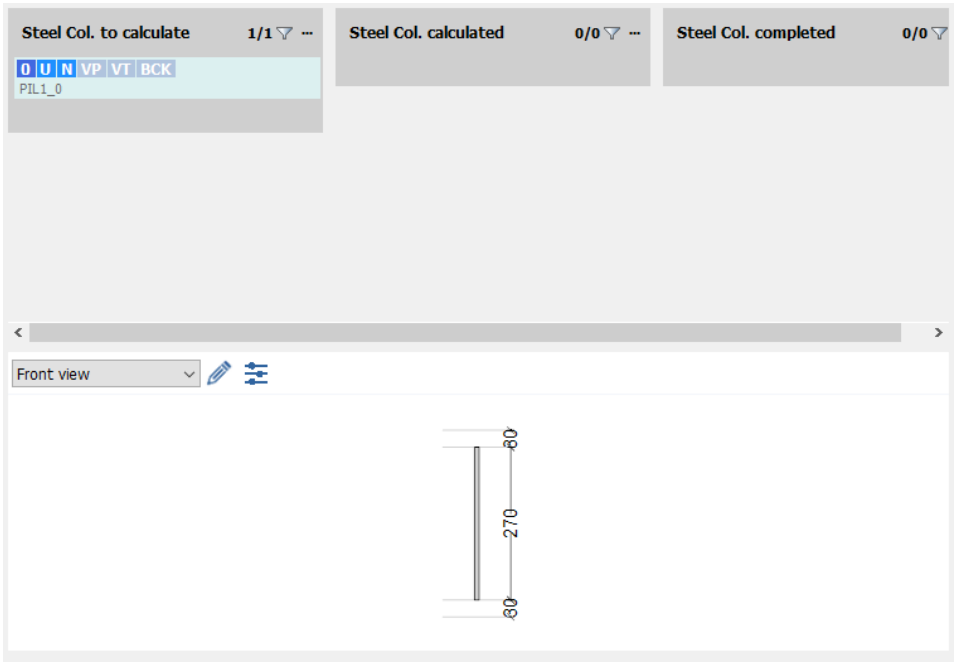


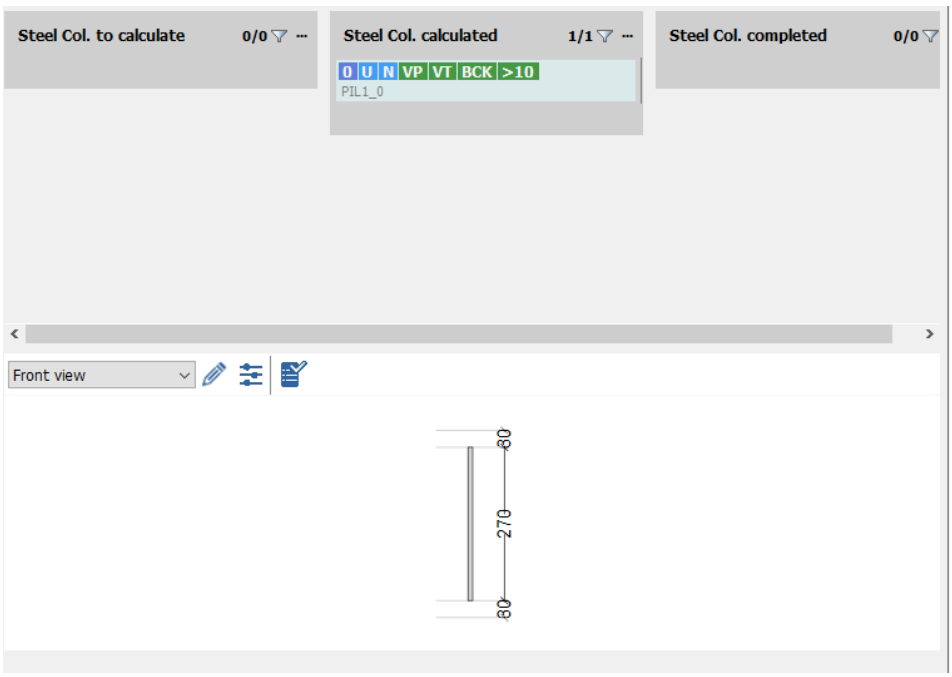
Once the column has been set, the environment is divided into 3 columns:

- Steel Columns to calculate
- Steel Columns calculated
- Steel Columns completed

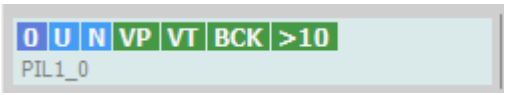
This is because the interface is based on Drag & Work which allows the user to start and control all the operations performed, by means of the action "Drag and drop".

To calculate a column inserted in the "Columns to calculate" environment, simply drag the static diagram into the "Calculated column" environment.
This will start the calculation of the structure, at the end of which the results will be shown.





The color of the icons shown represent the satisfaction or failure of the corresponding verification (green for verification satisfied, red for verification not satisfied).



The efficiency represents the safety coefficient referred to the verification in worst conditions: it will therefore be > 1 if all the verifications are satisfied, <1 if even a single verification is not satisfied.
The visualization of the results takes place in the lower part. To complete a previously calculated structure, simply drag it from the calculated beams environment to the completed beams environment.

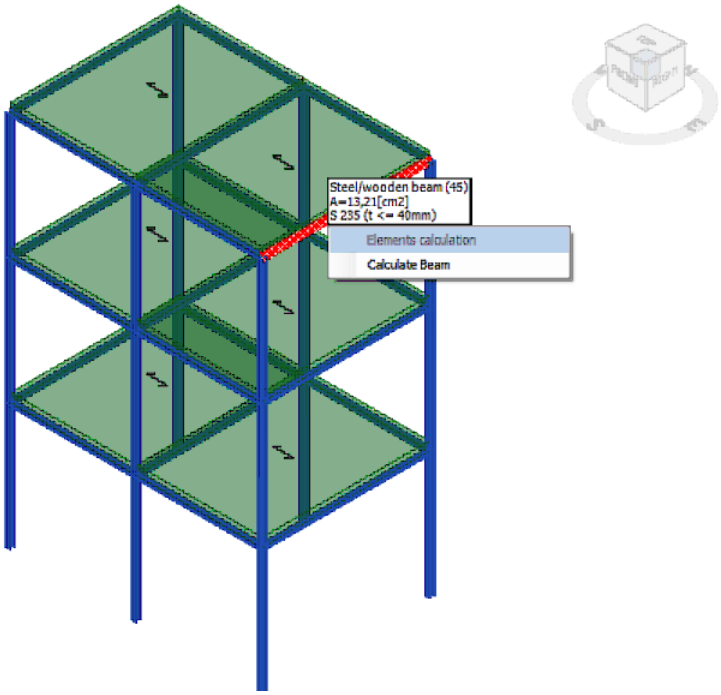
1.5.4.2 Steel Beams

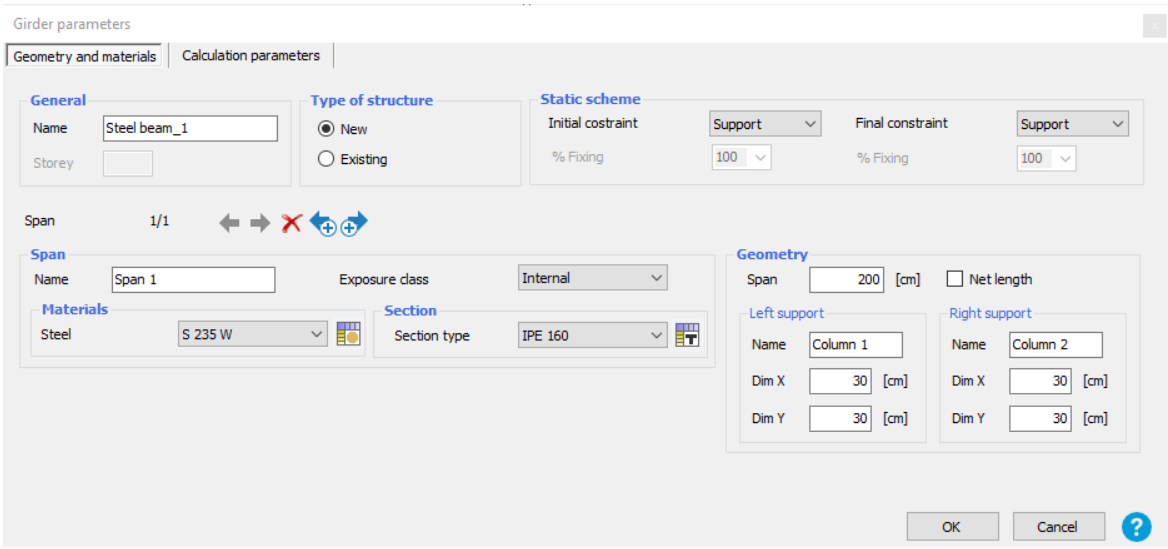
INPUT	3D MODEL	SINGLE ELEMENT
BEAMS	✓	✓

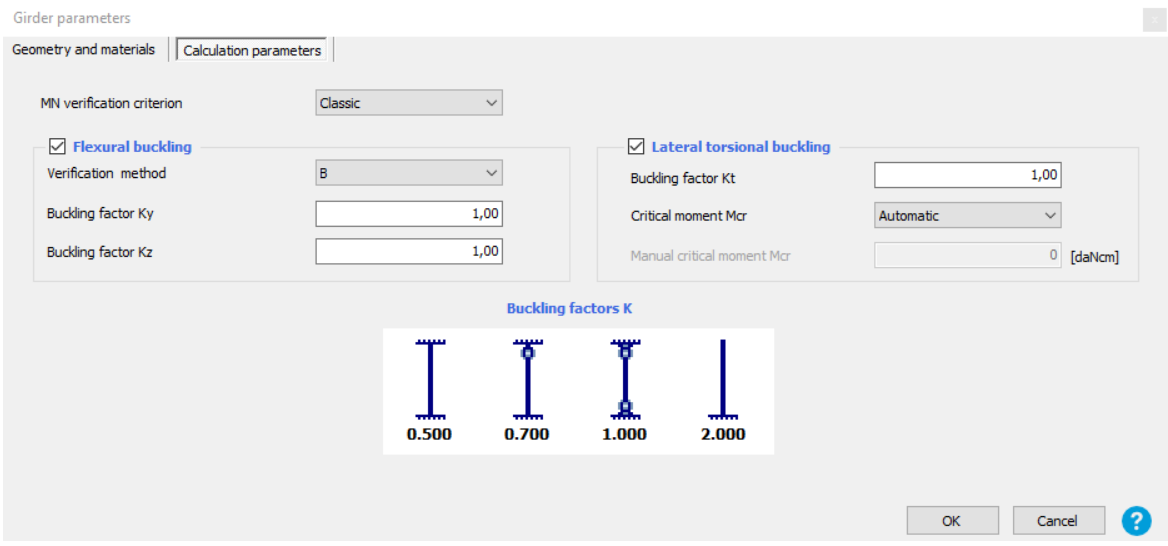
The Steel Beam module allows the verification and / or design of new or existing steel beams, with the possibility to choose between different calculation methods and parameters

The environment dedicated to setting the parameters for the beam presents the possibility to define the beam's section, constraints, materials, and span.

All the data inserted by the user is shown in the table below which offers us a simple and clear graphic visibility. It is also possible to create a calculation tab from a beam present within the model. To do this, simply right-click on the "Beam" element present in the model and choose the "Calculate Beam" option.


Steel/wooden beam (45)
A=13,21[cm2]
S 235 (t <= 40mm)
Elements calculation
Calculate Beam


Girder parameters
Geometry and materials
Calculation parameters
General
Name
Steel beam_1
Storey
Type of structure
New
Existing
Static scheme
Initial costraint
Support
% Fixing
100
Final constraint
Support
% Fixing
100
Span
1/1
Span
Name
Span 1
Exposure class
Internal
Materials
Steel
S 235 W
Section
Section type
IPE 160
Geometry
Span
200
[cm]
Net length
Left support
Name
Column 1
Dim X
30
[cm]
Dim Y
30
[cm]
Right support
Name
Column 2
Dim X
30
[cm]
Dim Y
30
[cm]
OK
Cancel


Girder parameters
Geometry and materials
Calculation parameters
MN verification criterion
Classic
Flexural buckling
Verification method
B
Buckling factor Ky
1,00
Buckling factor Kz
1,00
Lateral torsional buckling
Buckling factor Kt
1,00
Critical moment Mcr
Automatic
Manual critical moment Mcr
0
[daNcm]
Buckling factors K
0.500
0.700
1.000
2.000
OK
Cancel

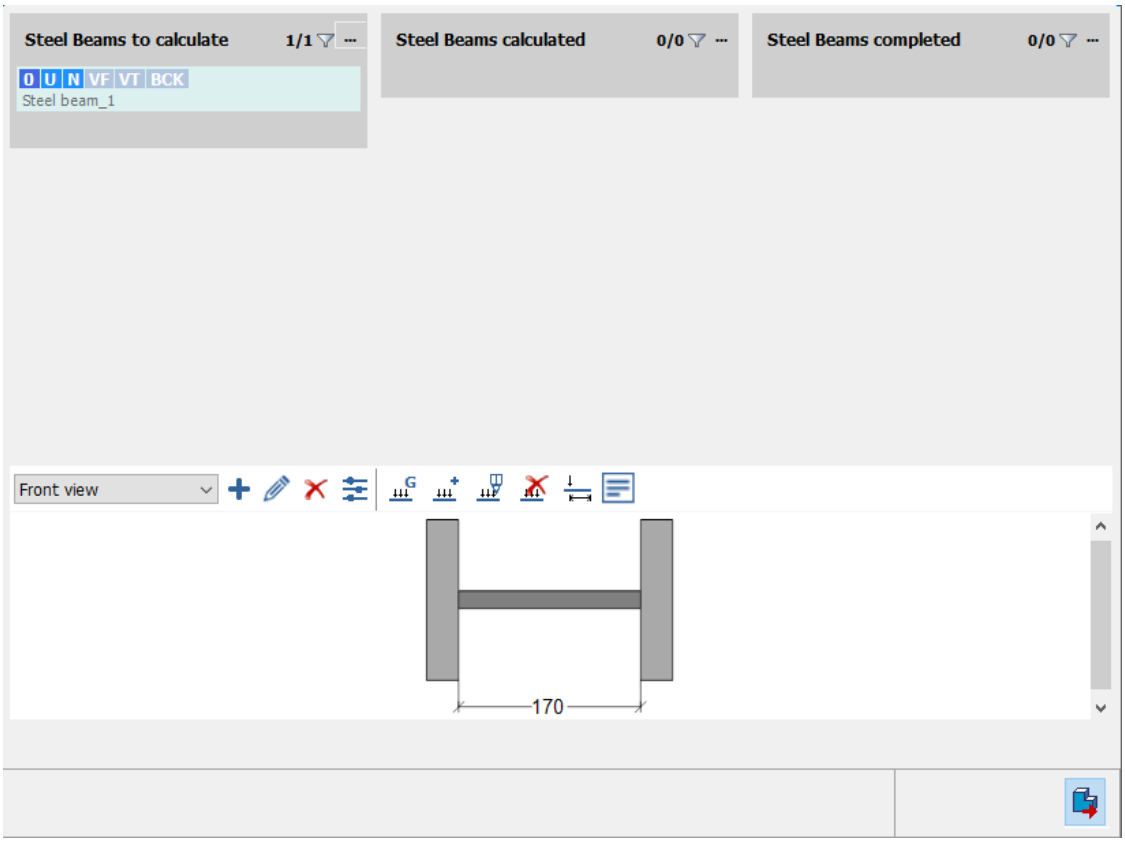
Once the beam has been set, the environment is divided into 3 columns:

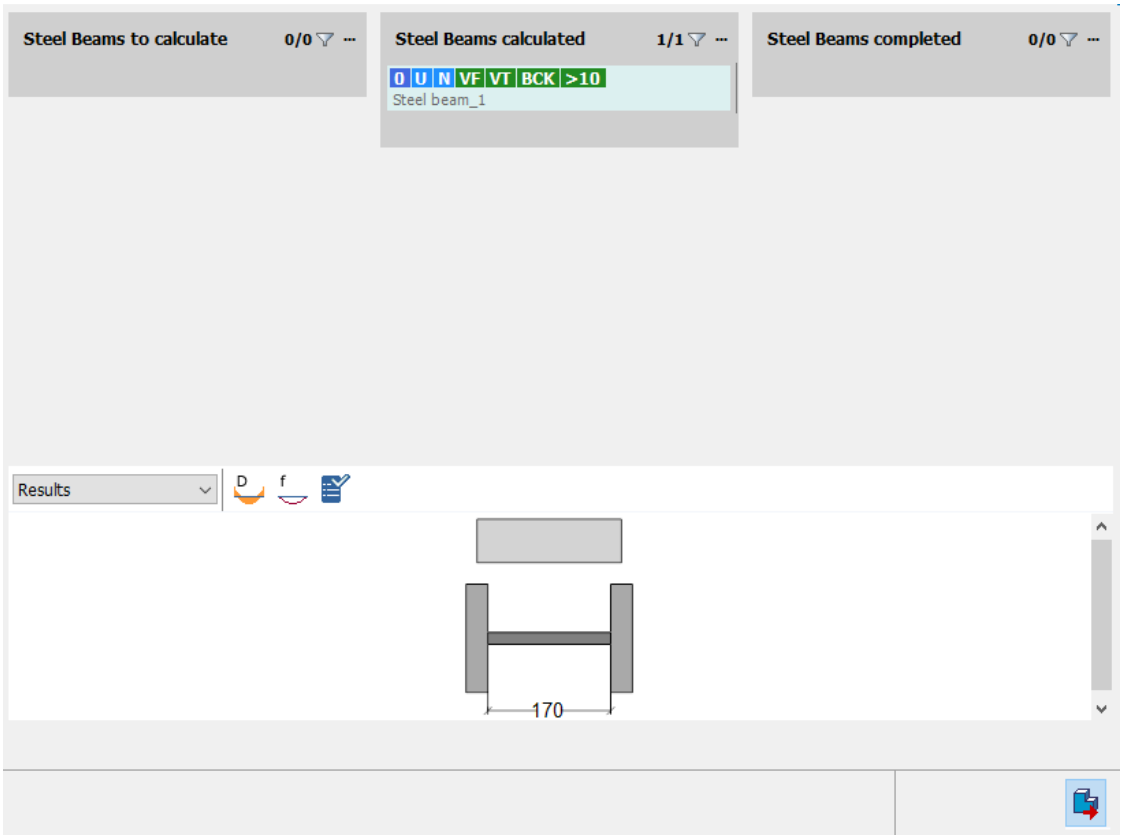
- Steel Beams to calculate
- Steel beams calculated
- Steel beams completed

This is because the interface is based on Drag & Work which allows the user to start and control all the operations performed, by means of the action "Drag and drop".

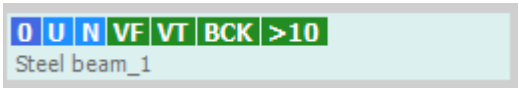
The "Steel beams to calculate" option allows to enter different types of loads to the beam, such as self-weight, linear, concentrated and surface loads.

To calculate a beam inserted in the "Beams to calculate" environment, simply drag the static diagram into the "Calculated beams" environment.
This will start the calculation of the structure, at the end of which the results will be shown.





The color of the icons shown represent the satisfaction or failure of the corresponding verification (green for verification satisfied, red for verification not satisfied).



The efficiency represents the safety coefficient referred to the verification in worst conditions: it will therefore be > 1 if all the verifications are satisfied, <1 if even a single verification is not satisfied.
The visualization of the results takes place in the lower part. To complete a previously calculated structure, simply drag it from the calculated beams environment to the completed beams environment.