

1.5.4 4. Local verifications on steel

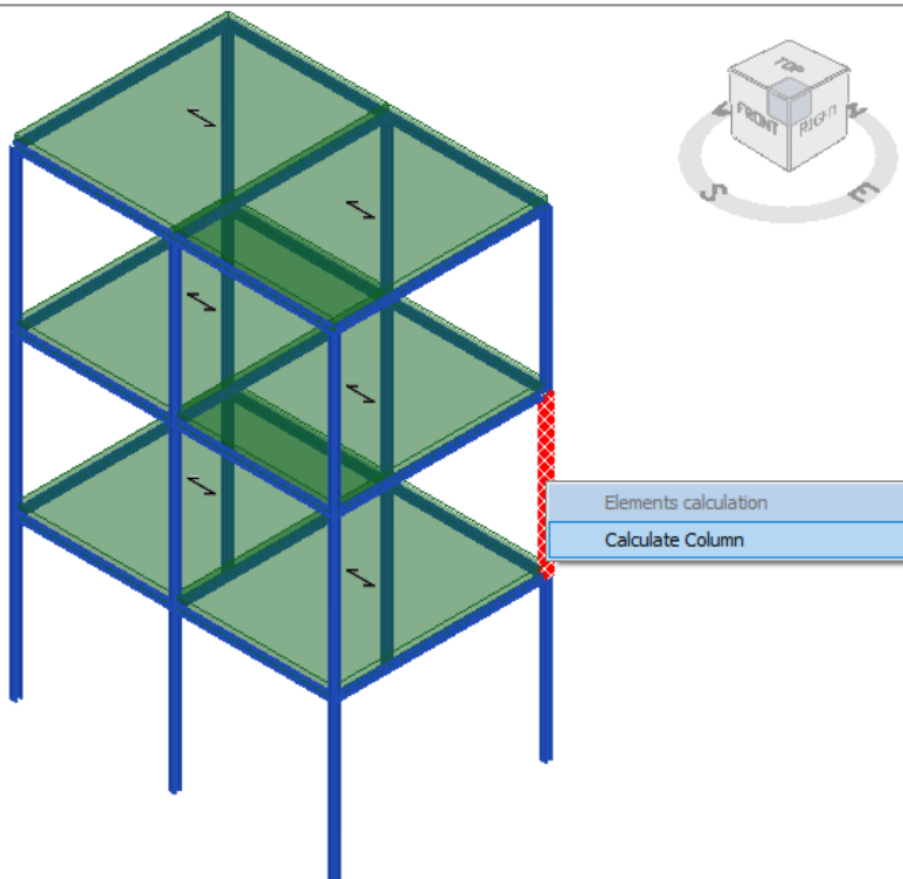
1.5.4.1 Steel Columns

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.



Columns parameters

Geometry and materials | Forces | Calculation parameters

General
 Name: PIL 1
 Storey:
Type of structure
 New
 Existing

Column
 Name: PIL_1_0
 Exposure class: Internal
Materials
 Steel: S 235 W
Section
 Section type: IPE 160

Geometry
 Height: 300 [cm]
 Thickness lower slab: 30 [cm]
 Thickness upper slab: 30 [cm]

OK Cancel ?

Columns parameters

Geometry and materials | Forces | Calculation parameters

ID Comb	Combination type	Section [cm]	Nsd [daN]	MsdY [daNcm]	MsdZ [daNcm]	VsdY [daN]	VsdZ [daN]
1	ULS static	0	10000	5000	3000	2500	2500
1	ULS static	300	5000	3000	5000	3000	3000

OK Cancel ?

Columns parameters

Geometry and materials | Forces | 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 ?

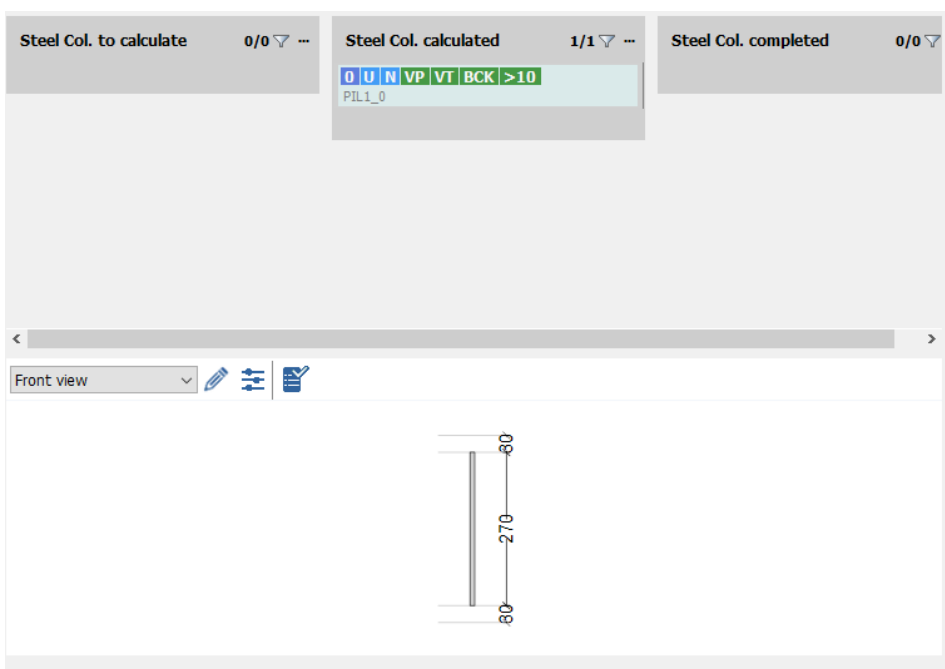
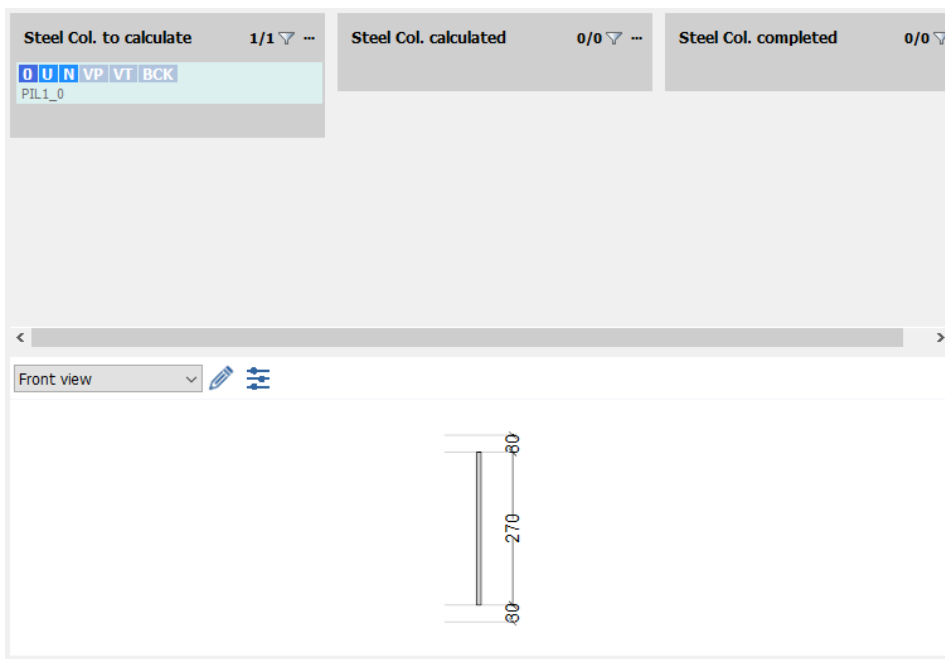
Once the column has been set, the environment is divided into 3 columns:

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

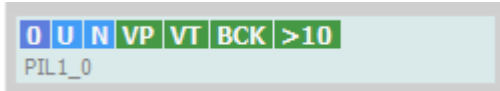
This is because the interface is based on Drag & Work which allows the user to start and control all the operations performed, by means of the action "Drag and drop".

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.

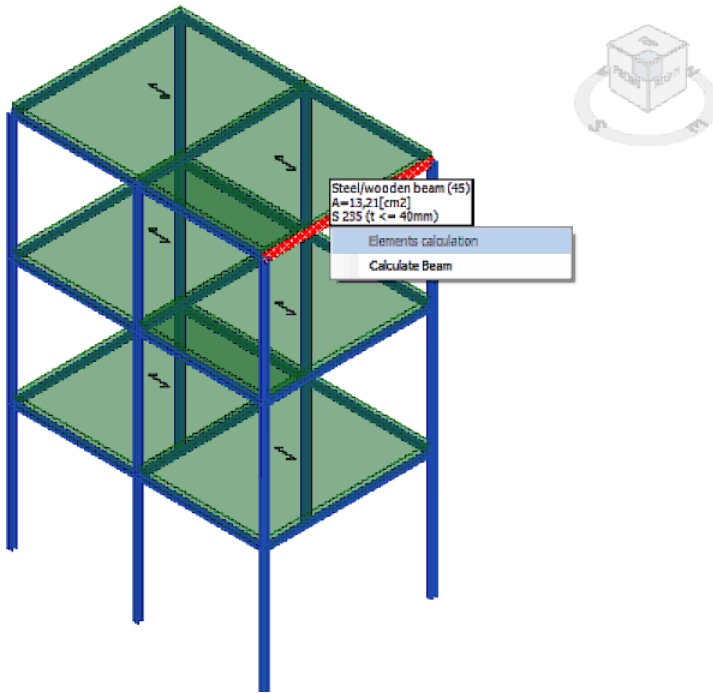
1.5.4.2 Steel Beams

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.



Girder parameters

Geometry and materials | Calculation parameters

General
 Name: Steel beam_1
 Storey:

Type of structure
 New
 Existing

Static scheme
 Initial constraint: Support
 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 K_y : 1,00
 Buckling factor K_z : 1,00

Lateral torsional buckling
 Buckling factor K_t : 1,00
 Critical moment M_{cr} : Automatic
 Manual critical moment M_{cr} : 0 [daNm]

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:

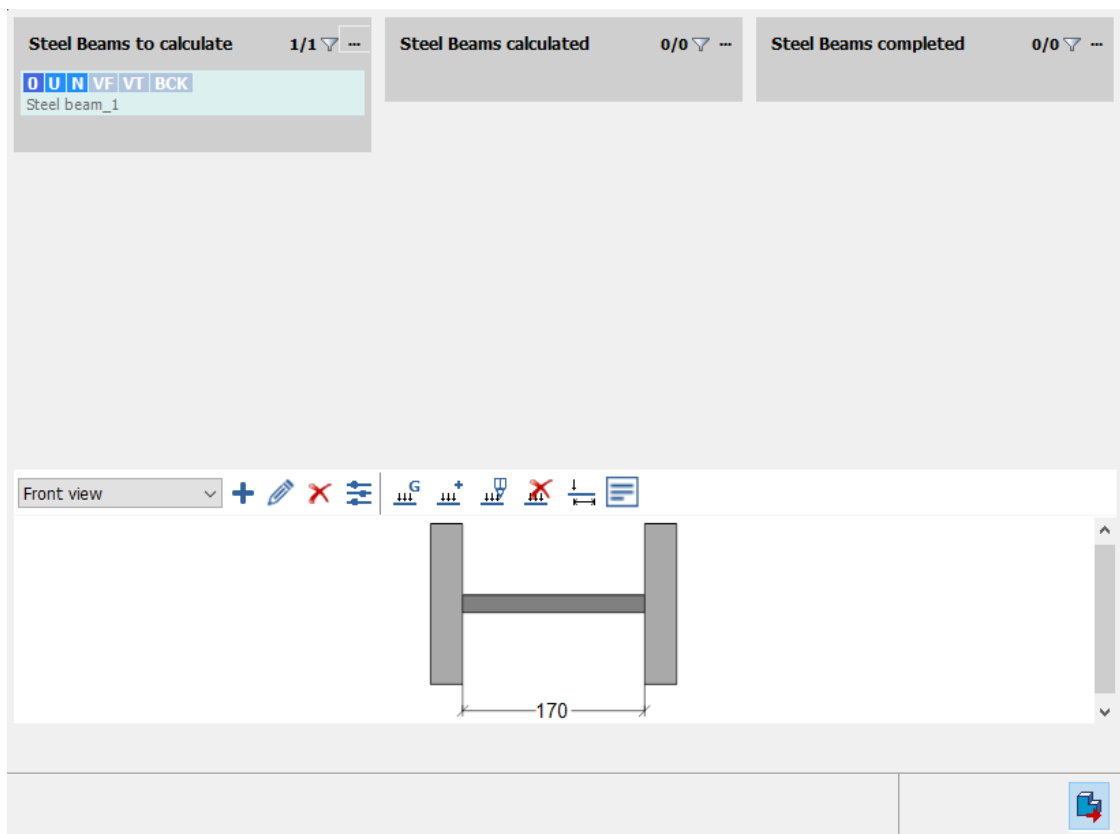
- Steel Beams to calculate
- Steel beams calculated
- Steel beams completed

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 screenshot displays a software interface for steel beam calculations. At the top, there are three panels: 'Steel Beams to calculate' (0/0), 'Steel Beams calculated' (1/1), and 'Steel Beams completed' (0/0). The 'Steel Beams calculated' panel shows a table with columns for verification types: D (orange), f (red), U (blue), N (blue), VF (green), VT (green), and BCK (green). The value for BCK is >10. Below the table is a diagram of a steel beam cross-section with a width of 170. The 'Results' panel at the bottom shows a dropdown menu and icons for D, f, and a checkmark.

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

Close-up of the verification table row for 'Steel beam_1'. The columns are: D (orange), U (blue), N (blue), VF (green), VT (green), BCK (green), and >10. The value for BCK is >10.

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.