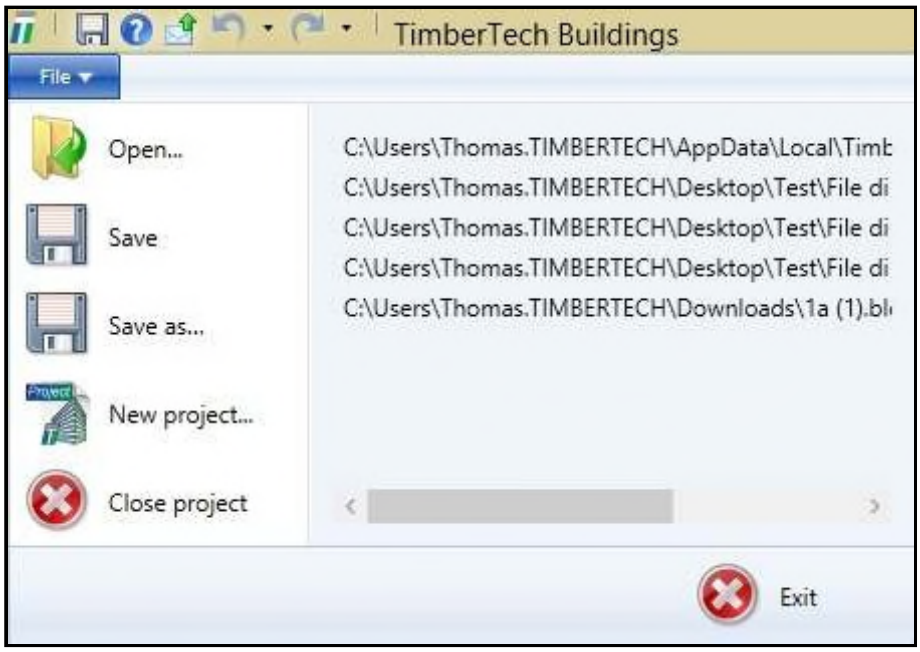
TimberTech Buildings

Guide

1 Tools in the File menu

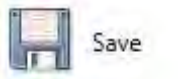
The following operations are provided by the File menu.



1.1 Open

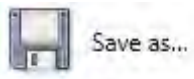
The *Open* command provides the opening of an existing file.

1.2 Save



The command saves the project on the existing file.

1.3 Save as

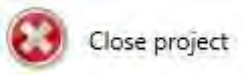


The command saves the project on a new file.

1.4 New project

Creates a new empty file and a new project.

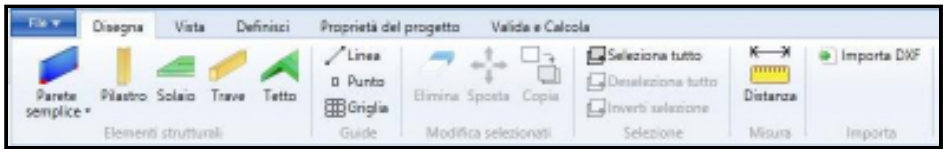
1.5 Close project



The command quits the project. If the project has not been saved the software gives the choice to save it.

2 Tools of Draw menu

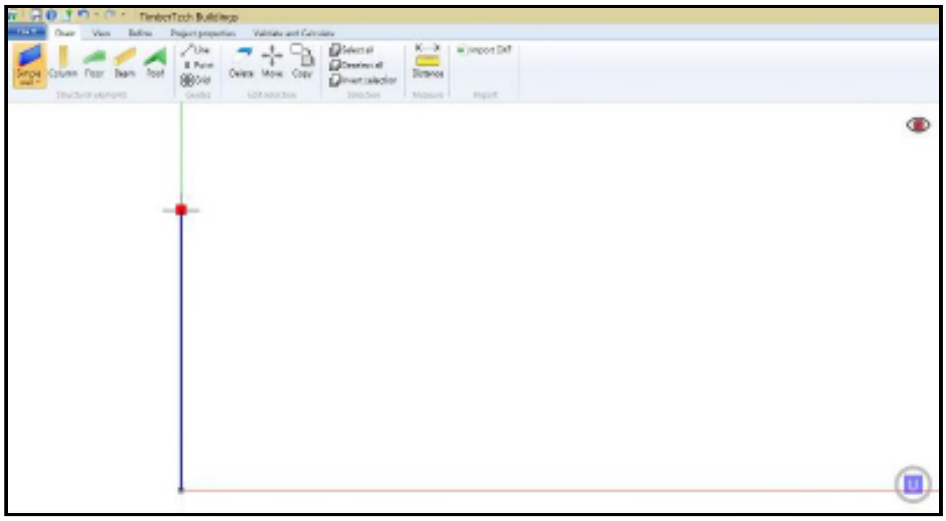
The commands in the *Draw* menu allow the user to draw the structural model. Other useful informations can be found in ***Chapter 8***.



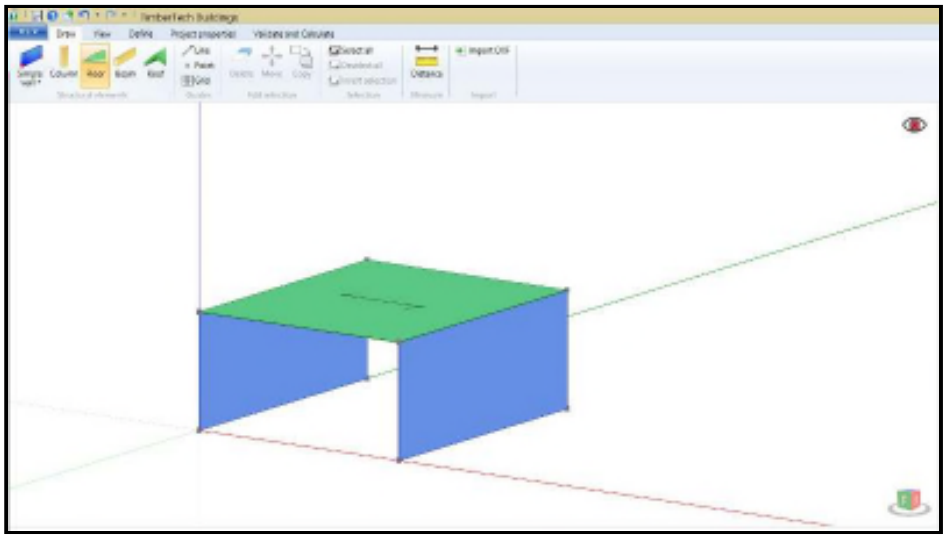
2.1 Structural elements

2.1.1 Simple wall

The *Simple wall* element can be drawn selecting two points on the screen. The software automatically assigns to the wall an height equal to 2.80 m; the user can modify it later (see ***Chapter 8***.).

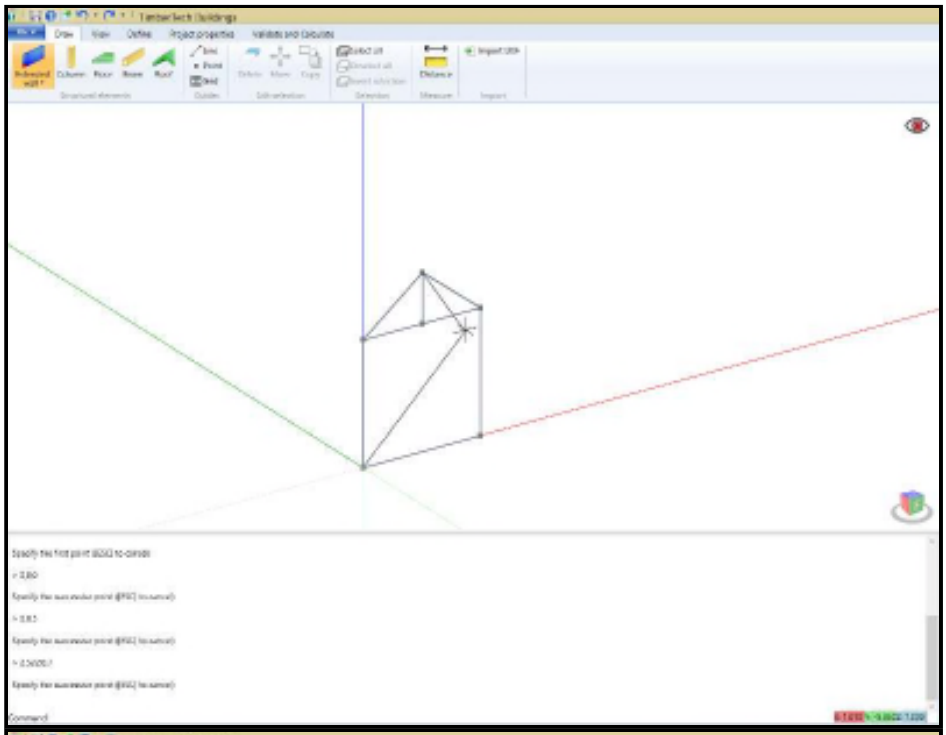


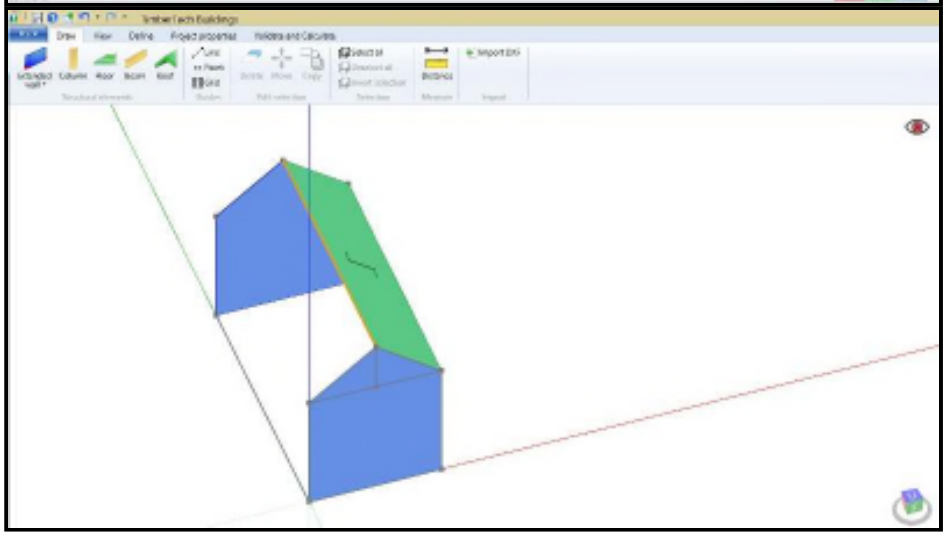
The wall element can also be defined typing ***Wall*** in the command window and thereafter the points coordinates. The first point is defined by the *absolute coordinates*, i.e. in the global system (for instance: 4,4,0). The second point coordinates can be defined in two ways using the absolute or the relative coordinates. If the user prefers the second way, he must put the symbol @ before the coordinates (for instance: @4,4,0 ; the values represent the distances along the three directions x, y and z).



2.1.1 Extended wall

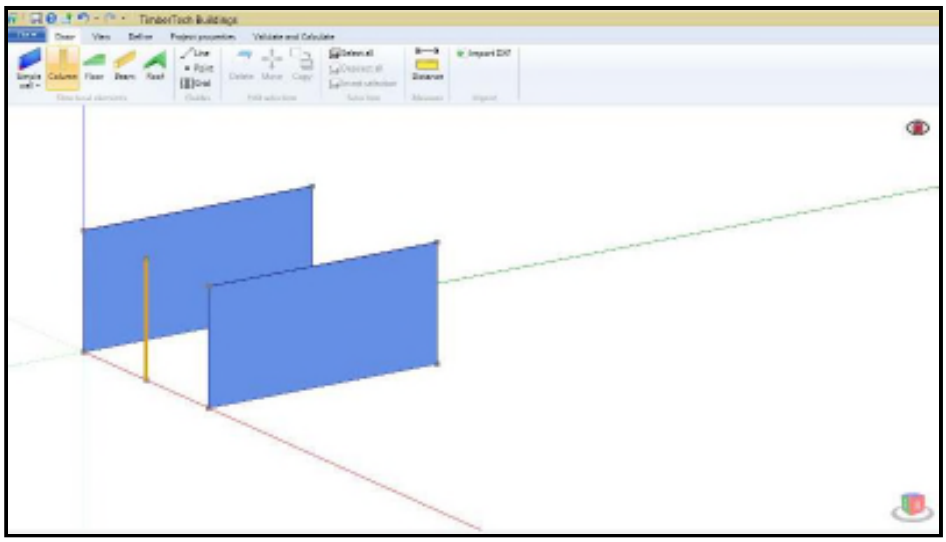
The *Extended wall* element can be defined drawing the wall vertexes on the screen. This command is used to define non rectangular walls or walls supporting a beam like the ridges one. The Extended walls can be also used to design rectangular walls: in this case the supported walls are considered non primary elements.





2.1.2 Pillar

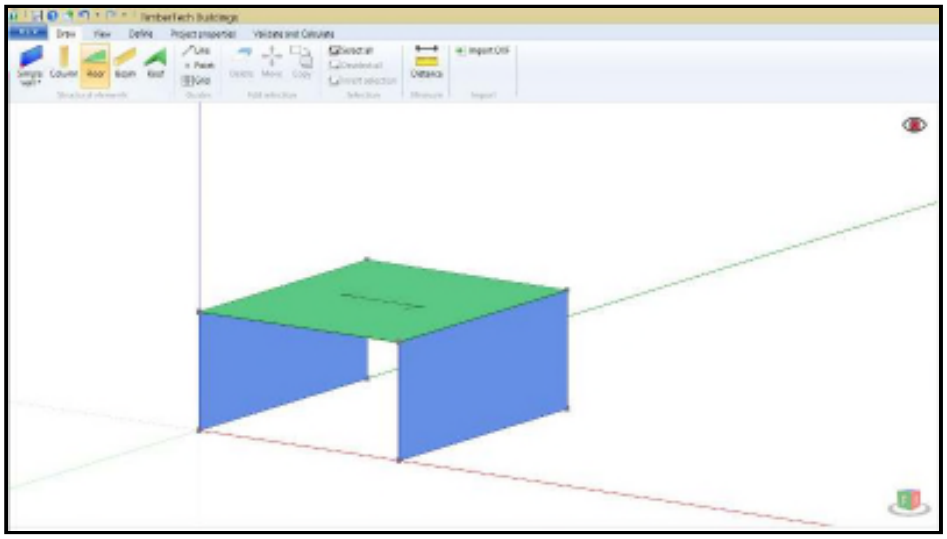
The *Pillar* element can be defined by a single point on the screen: the software assigns automatically to it an height equal to 2.80 m.



The user can use the command window to create a new element, typing ***Pillar*** and then the first point coordinates.

2.1.3 Floor element

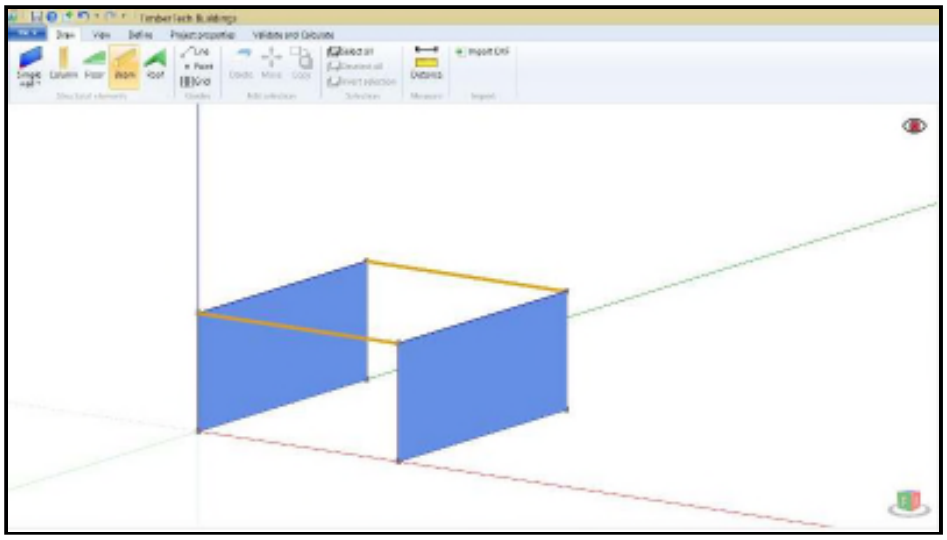
The *Floor* element can be straight designed on the screen defining at least three points. The points must be coplanar, with the same z-coordinate and belonging to a vertex of an any form polygon.



The user can define the Floor element directly in the command window, typing ***Floor*** and then the points coordinates.

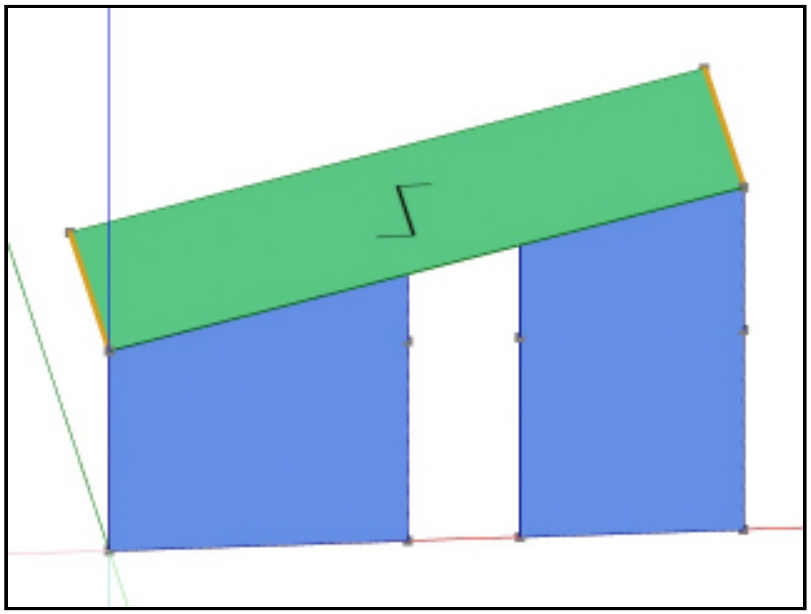
2.1.4 Beam element

The Beam element can be drawn on the screen selecting two points.



The user can define the Beam element directly in the command window, typing ***Beam*** and then inputting the points coordinates.

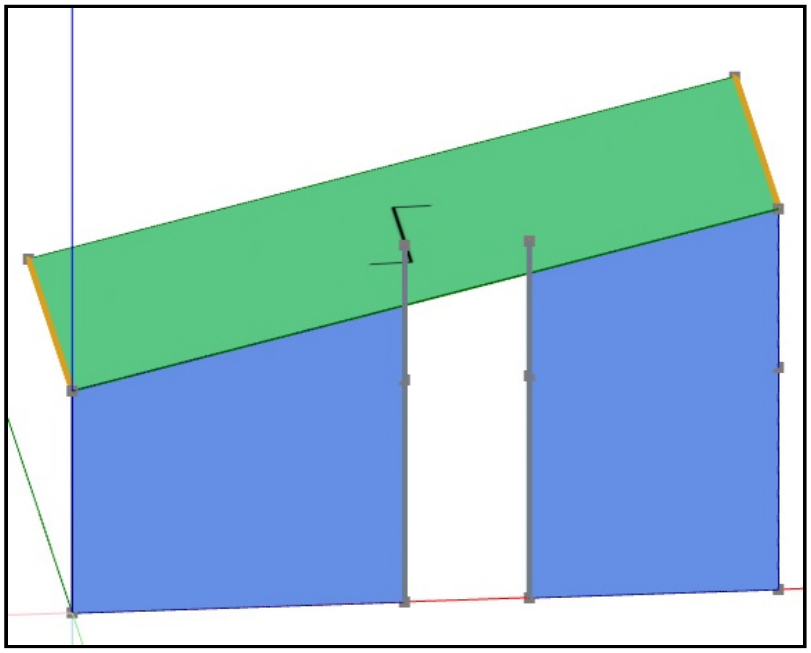
Advice: It could happen to design a beam, above a wall opening, supporting an inclinded roof.



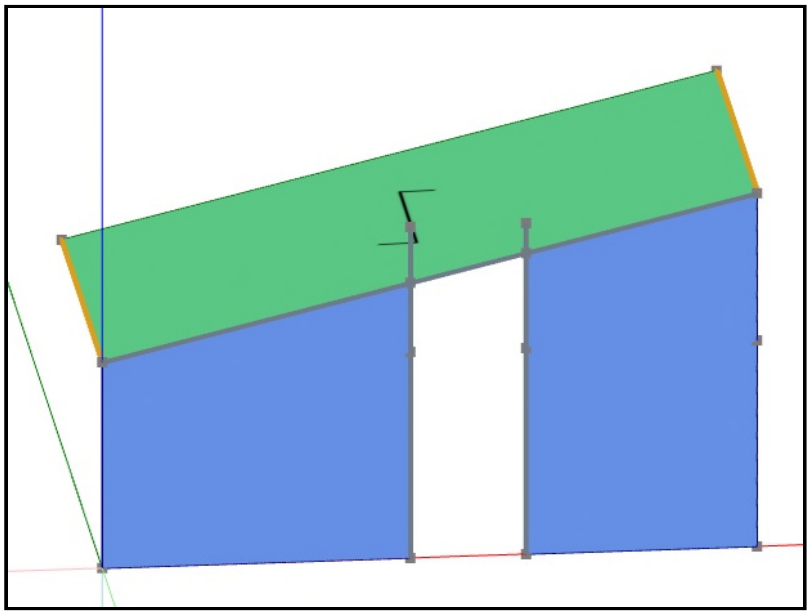
The procedure to define the beam is described in the following steps. If the user designs a new line intersecting another one, the software will automatically create a node.

This does not happen if the user moves an existing line on another one.

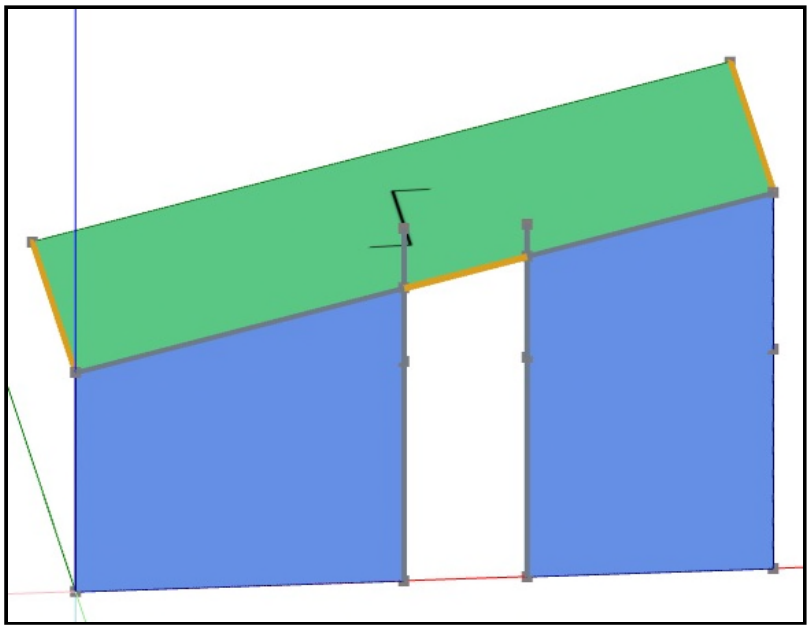
- Design two vertical lines (see paragraph 2.2.1): the lower point can be designed left-clicking the mouse, the upper one can be defined typing the relative coordinates, for instance: @0,0,5



- Design the line above the walls. This line belongs to the intersection between the plane of the roof and the plane of the walls. Two nodes will be created; they will be useful to define the new beam.

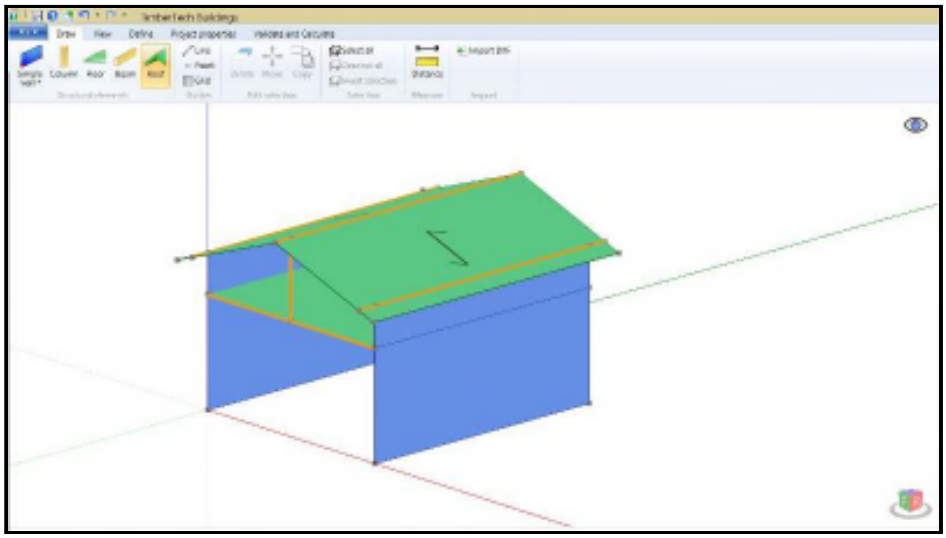


- Use the new nodes to design the beam.



2.1.5 Roof element

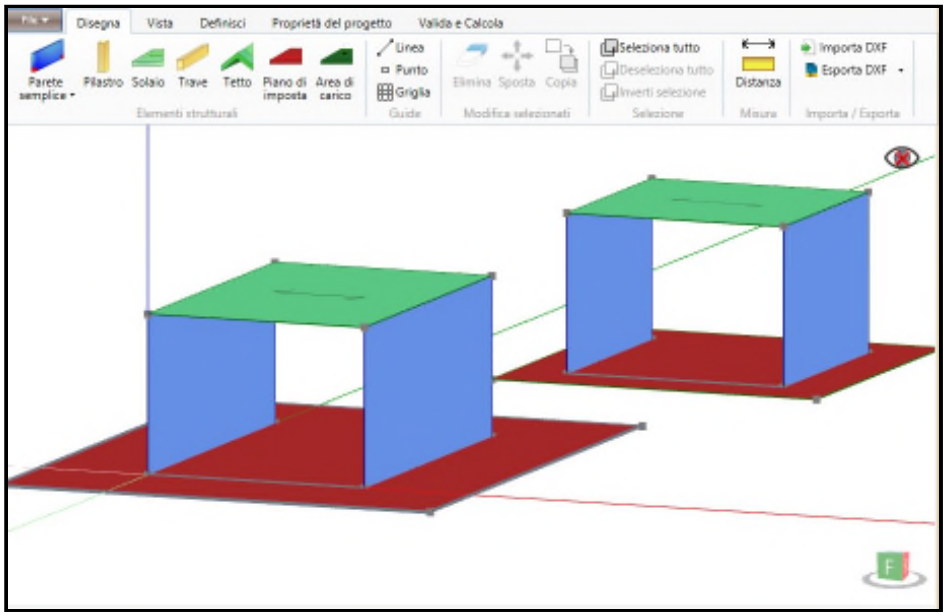
The Roof element is defined in the same way as the Floor element (see ***Floor element***). The Roof command provides the input of a pitched roof with a grade to the horizontal at no higher than 80 degree.



In the same way as the other elements, the Roof can be defined typing **Roof** and inputting the points coordinates in the command window.

2.1.6 Base element

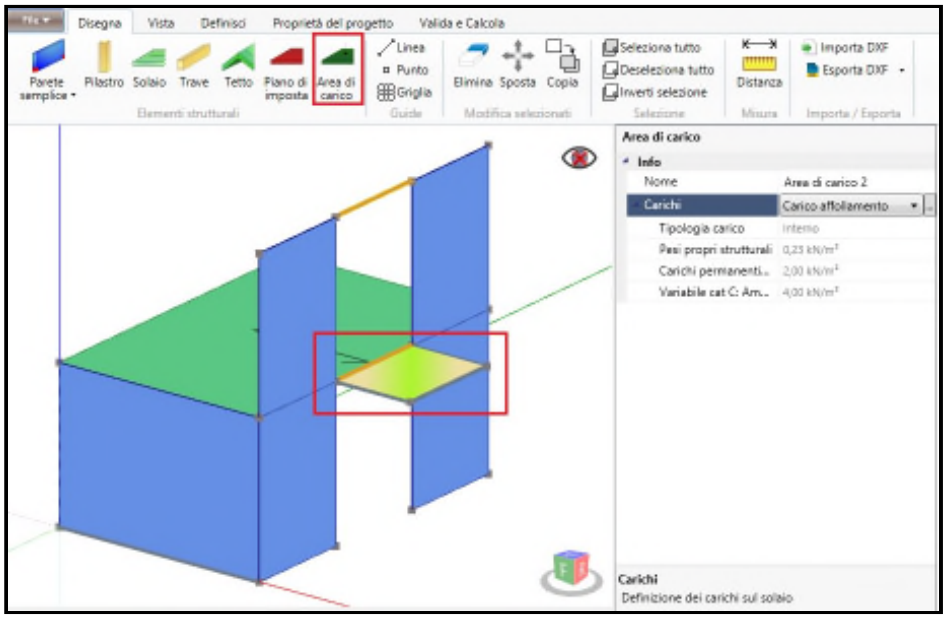
The *Base* element can be straight designed on the screen defining at least three points. The points must be coplanar, with the same z-coordinate and belonging to a vertex of an any form polygon. The Base element is very useful when there are buildings with bodies belonging to several levels.



The user can define the Base element directly in the command window, typing *Base* and then the points coordinates.

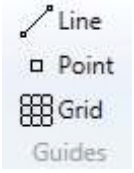
2.1.6 Load element

The *Load* element can be straight designed on the screen putting at least three points on a floor element or a roof element. The points must belong to a vertex of an any form polygon. The Load element is very useful in the case of not uniform load, like a floor with a terrace.



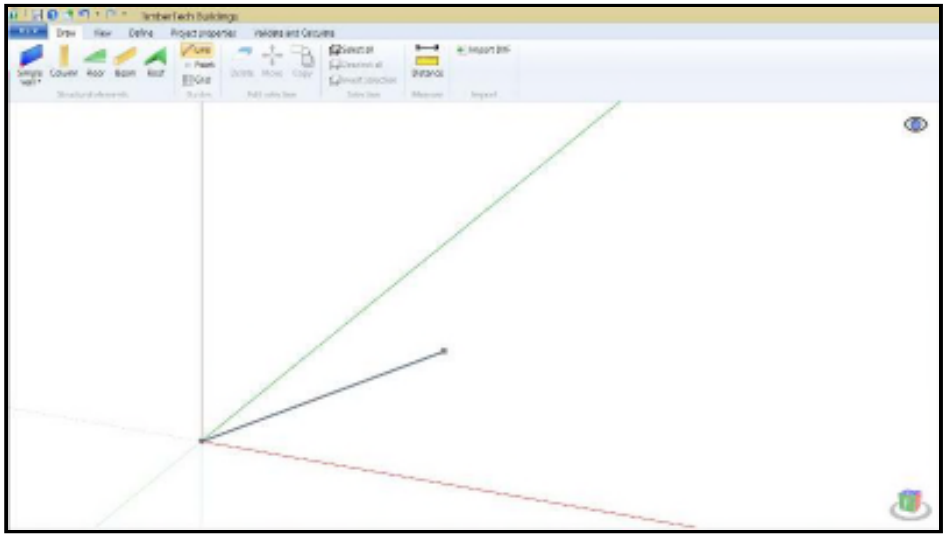
The user can define the Load element directly in the command window, typing Load and then the points coordinates.

2.2 Guides

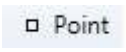


2.2.1 Line

The Line command is used to drawn a line in any direction. Thanks to the lines, the user can draw easily the structural elements in the model. The line can be defined selecting two point on the screen or typing ***Line*** in the command window and then inputting the points coordinates.



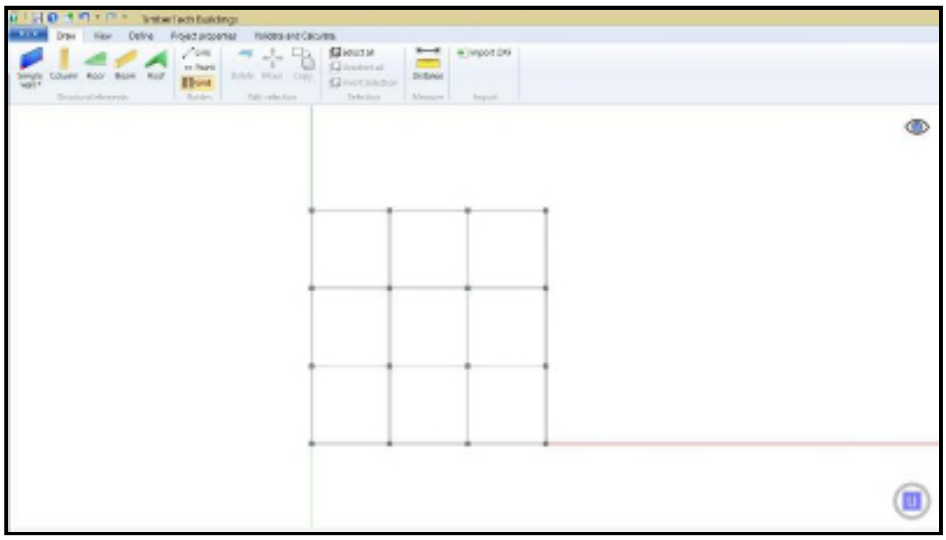
2.2.2 Point



The command is used to define directly on the screen a point. It is useful to draw and to place a structural element. The point can be defined typing ***Point*** in the command window and then inputting the points coordinates.

2.2.3 Grid

The *Grid* command defines, in the x-y plane, a grid of equidistant points belonging to perpendicular directrices. The user can select two points on the screen or type ***Grid*** in the command window and the points coordinates.



2.3 Edit selection

2.3.1 Delete

The *Delete* command deletes all the selected elements. The user can type ***Delete*** on the command window or press *Del* on the keyboard.

2.3.2 Move

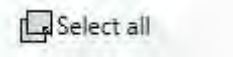
The command moves in the model all the selected elements. To move the elements the user has to input the initial and the final points directly on the screen or writing their coordinates in the command window. In the same way as the other commands, the user can also type ***Move*** in the commands line.

2.3.3 Copy

This command is used to copy the selected graphical elements. To copy the elements the user has to input the initial and the final points directly on the screen or inputting the coordinates in the command window. The user can also type ***Copy*** in the line command.

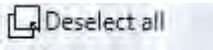
2.4 Selection

2.4.1 Select all



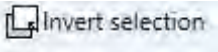
The command *Select all* is used to select all the graphical elements of the model. The user can use directly the command line typing ***Select All*** in it.

2.4.2 Deselect all



The command deselects all the selected graphical elements. The user can use directly the command line typing ***Select None*** in it.

2.4.3 Invert selection



The command is used to invert the selection. The command could be invoked by typing ***Select Invert*** in the command window.

Note: the user can select the objects left-clicking the mouse. Moving the cursor from the left to the right all objects which lie entirely within the window will be selected; moving the cursor from the right to the left all objects which lie entirely within the window and those which cross the window border will be selected.

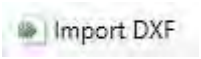
2.5 Distance

2.5.1 Distance (*Distance*)

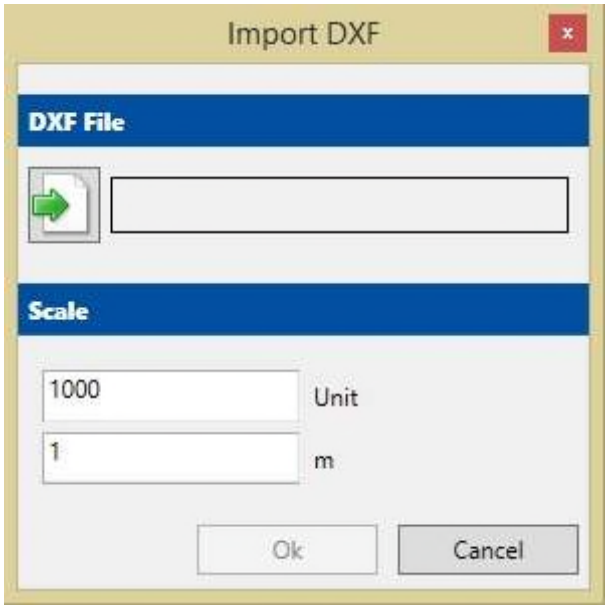
The *Distance* command measures the distance between two selected points. The measurement data along the axis (x, y and z) are displayed on the command line (for instance: Distance: 4,53 m; X: 3,21 m; Y: 7,10 m; Z: 0,86 m;). The command could be invoked by typing ***Distance*** in the command window.

2.6 Import

2.6.1 Import Dxf



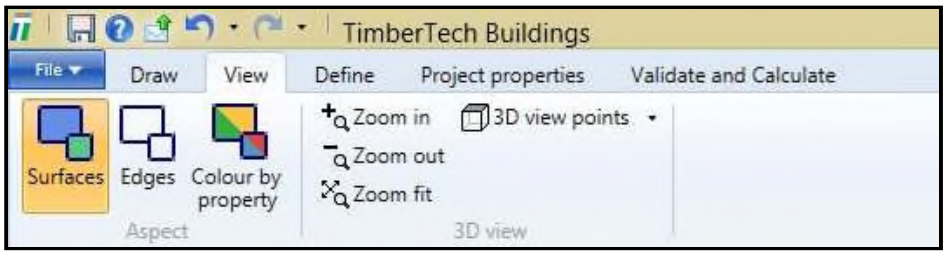
The command allows the user to import a dxf file within points, lines and polylines, useful to define, for instance, a pitched roof.



The command is linked to a window in which the user can select the dfx file and specify the importing scale. If the scale is too much small, the software will alert the user.

3 Tools of the View Menu

The View tools help the user to display the structural model.



3.1 Appearance

3.1.1 Surfaces

The *Surfaces* command allows the user to display the surfaces and the definition points of the structural elements.

3.1.2 Edges

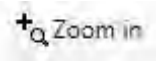
The *Edges* command allows the user to display the edges of the structural elements.

3.1.3 Colour by property

The command allows the user to display the elements properties: walls, floors, roofs, columns, beams. The user can easily display the materials, the cross-sections, the types and even the connections and the loads.

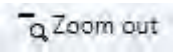
3.2 3D View

3.2.1 Zoom in



The command allows the user to zoom in the current view.

3.2.2 Zoom out

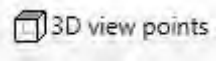


The command allows the user to zoom out the current view.

3.2.3 Zoom fit;

The command allows the user to display all the graphical elements in the current view.

3.3 3D Point of view



The menu allows to choose a type of view (orthogonal or axonometric) to display the current view.

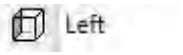
3.3.1 Up

The command sets a view from above.

3.3.2 Down

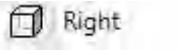
The command sets a view from below.

3.3.3 Left



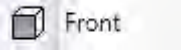
The command sets a view from the left.

3.3.4 Right



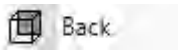
The command sets a view from the right.

3.3.5 Front



The command sets a frontal view.

3.3.6 Back



The command sets a back view.

3.3.7 SW

The command sets an axonometric view: South-West.

3.3.8 SE

The command sets an axonometric view: South-East.

3.3.9 NW

The command sets an axonometric view: North-West.

3.3.9 NE

The command sets an axonometric view: South-East.

Note:

The user can perform zoom and pan with a wheel mouse:

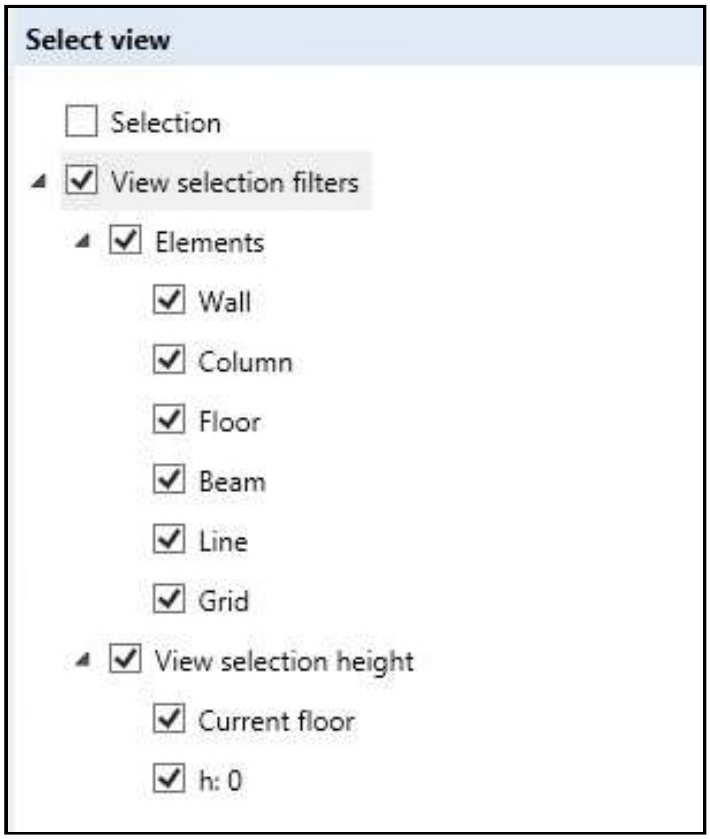
- zoom: scrolls up and down to zoom out and zoom in the view;
- pan: press the mouse wheel button and drag the mouse;
- point of view: rigth-click the mouse and drag it.

3.4 Other tools

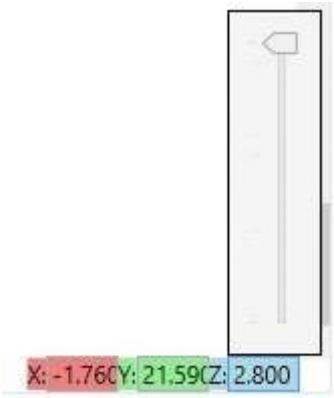
3.4.1 Select view

If the user selects the icon on the upper right side of the screen the checkbox will appear. The user can select the elements he wants to display or the floor levels to display the elements belonging to it.

The non selected elements can not be displayed.



If the user select *Current floor* it will appear a cursor (on the lower right side of the console) which allows to choose the level to display.



3.4.2 Point of view

The icon, in the lower right side of the visualization window, allows the user to set the orthogonal views (see: **3D Point of view**) double-clicking on the desired cube face.

4 Tools in Define Menu

The tools in the *Define* menu allow the user to:

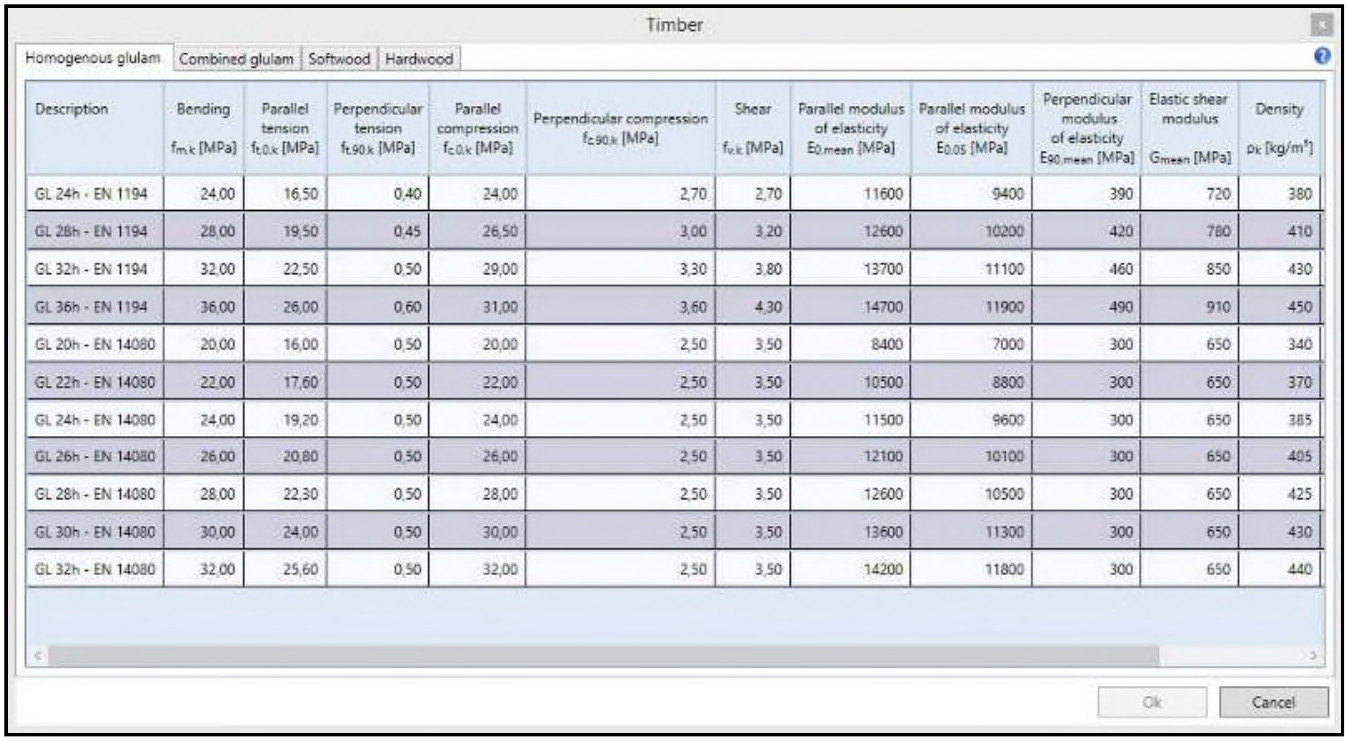
- consult the materials and joints catalogues;
- define the structural elements type;
- define the loads on the structural elements;

4.1 Materials and joints catalogue

4.1.1 Timber

The *Timber* command allows to consult the performance data of:

4.4.1.1 Homogenous and combined glulam:

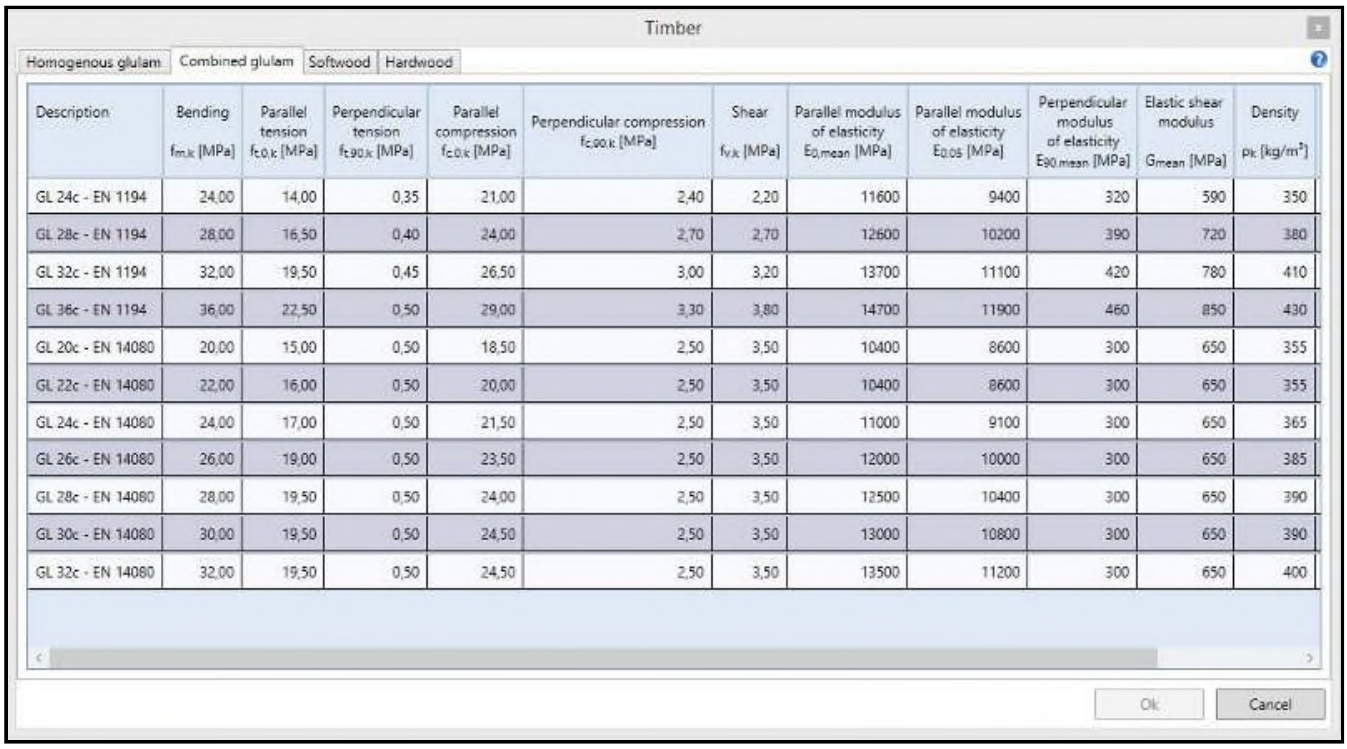


Description	Bending $f_{m,k}$ [MPa]	Parallel tension $f_{t,0,k}$ [MPa]	Perpendicular tension $f_{t,90,k}$ [MPa]	Parallel compression $f_{c,0,k}$ [MPa]	Perpendicular compression $f_{c,90,k}$ [MPa]	Shear $f_{v,k}$ [MPa]	Parallel modulus of elasticity $E_{0,mean}$ [MPa]	Parallel modulus of elasticity $E_{0,05}$ [MPa]	Perpendicular modulus of elasticity $E_{90,mean}$ [MPa]	Elastic shear modulus G_{mean} [MPa]	Density ρ_k [kg/m³]
GL 24h - EN 1194	24,00	16,50	0,40	24,00	2,70	2,70	11600	9400	390	720	380
GL 28h - EN 1194	28,00	19,50	0,45	26,50	3,00	3,20	12600	10200	420	780	410
GL 32h - EN 1194	32,00	22,50	0,50	29,00	3,30	3,80	13700	11100	460	850	430
GL 36h - EN 1194	36,00	26,00	0,60	31,00	3,60	4,30	14700	11900	490	910	450
GL 20h - EN 14080	20,00	16,00	0,50	20,00	2,50	3,50	8400	7000	300	650	340
GL 22h - EN 14080	22,00	17,60	0,50	22,00	2,50	3,50	10500	8800	300	650	370
GL 24h - EN 14080	24,00	19,20	0,50	24,00	2,50	3,50	11500	9600	300	650	385
GL 26h - EN 14080	26,00	20,80	0,50	26,00	2,50	3,50	12100	10100	300	650	405
GL 28h - EN 14080	28,00	22,30	0,50	28,00	2,50	3,50	12600	10500	300	650	425
GL 30h - EN 14080	30,00	24,00	0,50	30,00	2,50	3,50	13600	11300	300	650	430
GL 32h - EN 14080	32,00	25,60	0,50	32,00	2,50	3,50	14200	11800	300	650	440

The tables provide the mechanical properties values of different *strength classes*:

- $f_{m,k}$: bending strength;
- $f_{t,0,k}$: parallel tension strength;
- $f_{t,90,k}$: perpendicular tension strength;
- $f_{c,0,k}$: parallel compression strength;
- $f_{c,90,k}$: perpendicular compression strength;
- $f_{v,k}$: shear strength;
- $E_{0,mean}$: mean value of the parallel modulus of elasticity;
- $E_{0,05}$: characteristic value of parallel modulus of elasticity;
- $E_{90,mean}$: mean value of the perpendicular modulus of elasticity;
- $G_{,mean}$: mean value of the elastic shear modulus;
- $\rho_{,k}$: density (characteristic value);
- $\rho_{,m}$: density (mean value);
- γ: volumetric weight;

4.1.1.2 Softwood and hardwood:



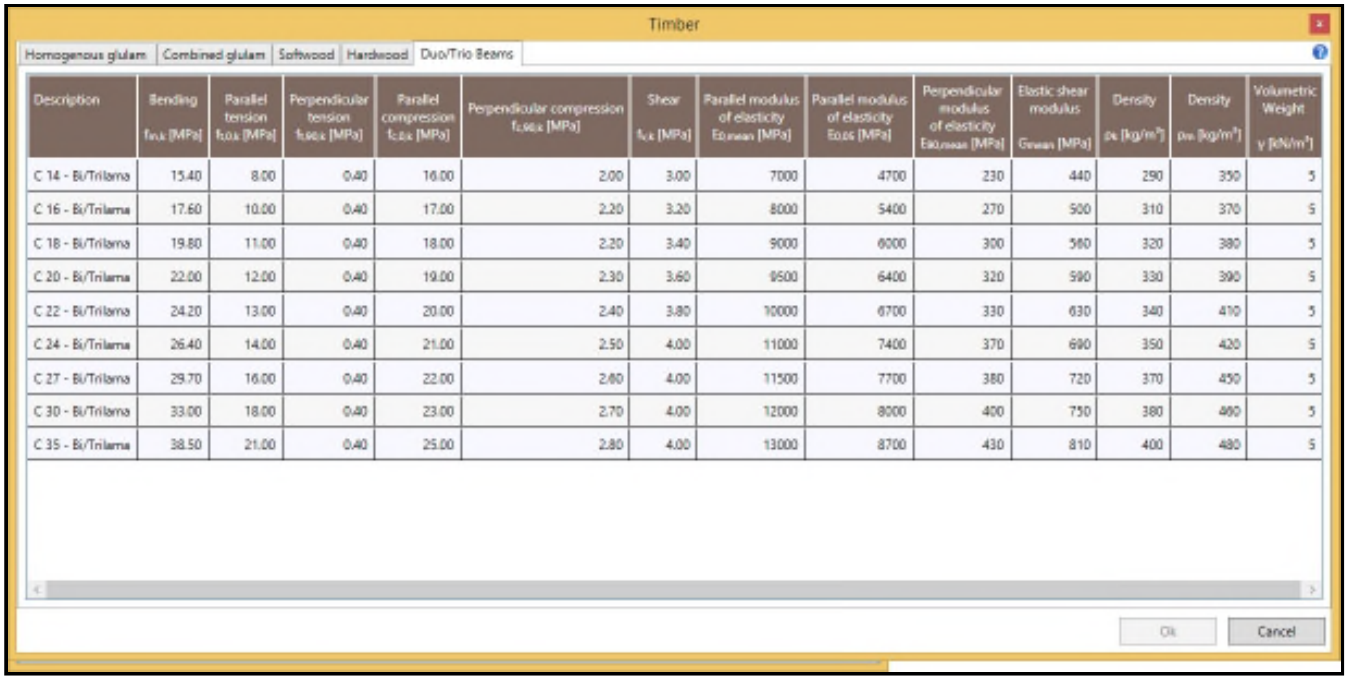
Description	Bending $f_{m,k}$ [MPa]	Parallel tension $f_{t,0,k}$ [MPa]	Perpendicular tension $f_{t,90,k}$ [MPa]	Parallel compression $f_{c,0,k}$ [MPa]	Perpendicular compression $f_{c,90,k}$ [MPa]	Shear $f_{v,k}$ [MPa]	Parallel modulus of elasticity $E_{0,mean}$ [MPa]	Parallel modulus of elasticity $E_{0,05}$ [MPa]	Perpendicular modulus of elasticity $E_{90,mean}$ [MPa]	Elastic shear modulus G_{mean} [MPa]	Density ρ_k [kg/m³]
GL 24c - EN 1194	24,00	14,00	0,35	21,00	2,40	2,20	11600	9400	320	590	350
GL 28c - EN 1194	28,00	16,50	0,40	24,00	2,70	2,70	12600	10200	390	720	380
GL 32c - EN 1194	32,00	19,50	0,45	26,50	3,00	3,20	13700	11100	420	780	410
GL 36c - EN 1194	36,00	22,50	0,50	29,00	3,30	3,80	14700	11900	460	850	430
GL 20c - EN 14080	20,00	15,00	0,50	18,50	2,50	3,50	10400	8600	300	650	355
GL 22c - EN 14080	22,00	16,00	0,50	20,00	2,50	3,50	10400	8600	300	650	355
GL 24c - EN 14080	24,00	17,00	0,50	21,50	2,50	3,50	11000	9100	300	650	365
GL 26c - EN 14080	26,00	19,00	0,50	23,50	2,50	3,50	12000	10000	300	650	385
GL 28c - EN 14080	28,00	19,50	0,50	24,00	2,50	3,50	12500	10400	300	650	390
GL 30c - EN 14080	30,00	19,50	0,50	24,50	2,50	3,50	13000	10800	300	650	390
GL 32c - EN 14080	32,00	19,50	0,50	24,50	2,50	3,50	13500	11200	300	650	400

The tables provide the mechanical properties values of different *strength classes*>:

- $f_{m,k}$: Bending strength;
- $f_{t,0,k}$: Parallel tension strength;
- $f_{t,90,k}$: Perpendicular tension strength;
- $f_{c,0,k}$: Parallel compression strength;
- $f_{c,90,k}$: Perpendicular compression strength;
- $f_{v,k}$: Shear strength;

- $E_{0,mean}$: Mean value of the parallel modulus of elasticity;
- $E_{0,05}$: Characteristic value of parallel modulus of elasticity;
- $E_{90,mean}$: Mean value of the perpendicular modulus of elasticity;
- $G_{,mean}$: Mean value of the elastic shear modulus;
- $\rho_{,k}$: Density (characteristic value);
- $\rho_{,m}$: Density (mean value);
- γ: Volumetric weight;

4.1.1.3 Duo/Trio Beams:



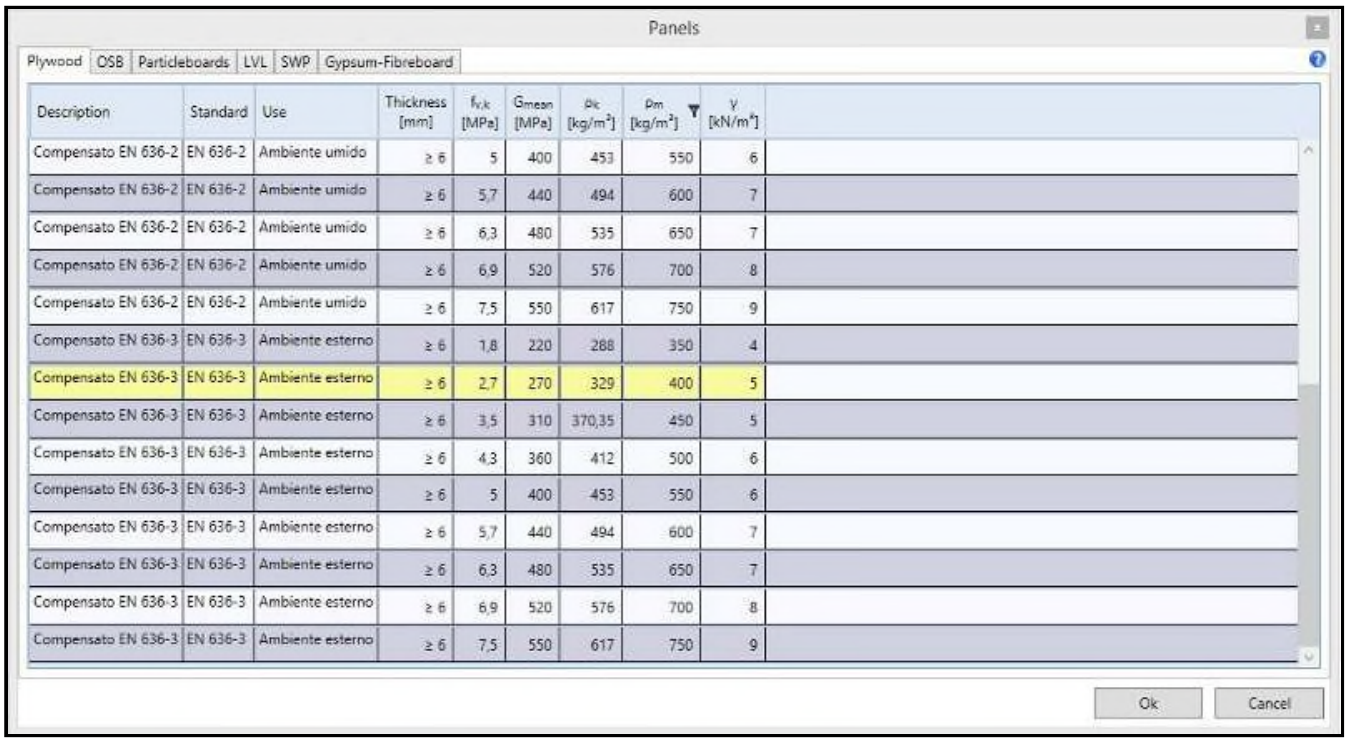
The tables provide the mechanical properties values of different *strength classes*>:

- $f_{m,k}$: Bending strength;
- $f_{t,0,k}$: Parallel tension strength;
- $f_{t,90,k}$: Perpendicular tension strength;
- $f_{c,0,k}$: Parallel compression strength;
- $f_{c,90,k}$: Perpendicular compression strength;
- $f_{v,k}$: Shear strength;
- $E_{0,mean}$: Mean value of the parallel modulus of elasticity;
- $E_{0,05}$: Characteristic value of parallel modulus of elasticity;
- $E_{90,mean}$: Mean value of the perpendicular modulus of elasticity;
- $G_{,mean}$: Mean value of the elastic shear modulus;
- $\rho_{,k}$: Density (characteristic value);
- $\rho_{,m}$: Density (mean value);
- γ: Volumetric weight;

4.1.2 Panels

The *Panels* command allows the user to consult the performance data of different panel types:

4.1.2.1 Plywood

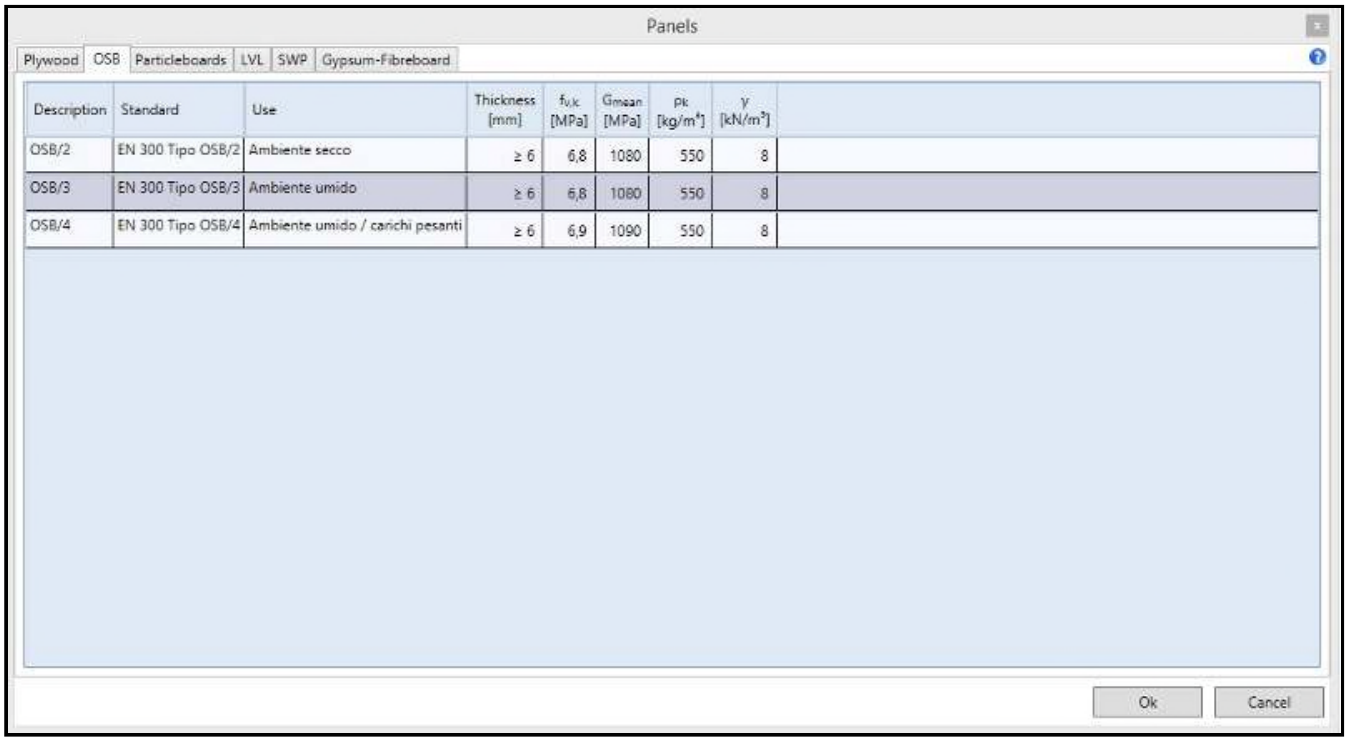


Description	Standard	Use	Thickness [mm]	$f_{v,k}$ [MPa]	G_{mean} [MPa]	ρ_k [kg/m³]	ρ_m [kg/m³]	γ [kN/m³]
Compensato EN 636-2	EN 636-2	Ambiente umido	≥ 6	5	400	453	550	6
Compensato EN 636-2	EN 636-2	Ambiente umido	≥ 6	5,7	440	494	600	7
Compensato EN 636-2	EN 636-2	Ambiente umido	≥ 6	6,3	480	535	650	7
Compensato EN 636-2	EN 636-2	Ambiente umido	≥ 6	6,9	520	576	700	8
Compensato EN 636-2	EN 636-2	Ambiente umido	≥ 6	7,5	550	617	750	9
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	1,8	220	288	350	4
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	2,7	270	329	400	5
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	3,5	310	370,35	450	5
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	4,3	360	412	500	6
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	5	400	453	550	6
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	5,7	440	494	600	7
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	6,3	480	535	650	7
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	6,9	520	576	700	8
Compensato EN 636-3	EN 636-3	Ambiente esterno	≥ 6	7,5	550	617	750	9

The tables provide the mechanical properties values of different *use classes* (dry conditions, humid conditions and exterior conditions), according to EN 636-1, EN 636-2 and EN 636-3:

- Thickness;
- $f_{v,k}$: Shear strength;
- G_{mean}: Mean value of Shear modulus;
- ρ_k: Characteristic density;
- ρ_m: Mean density;
- γ: Volumetric weight;

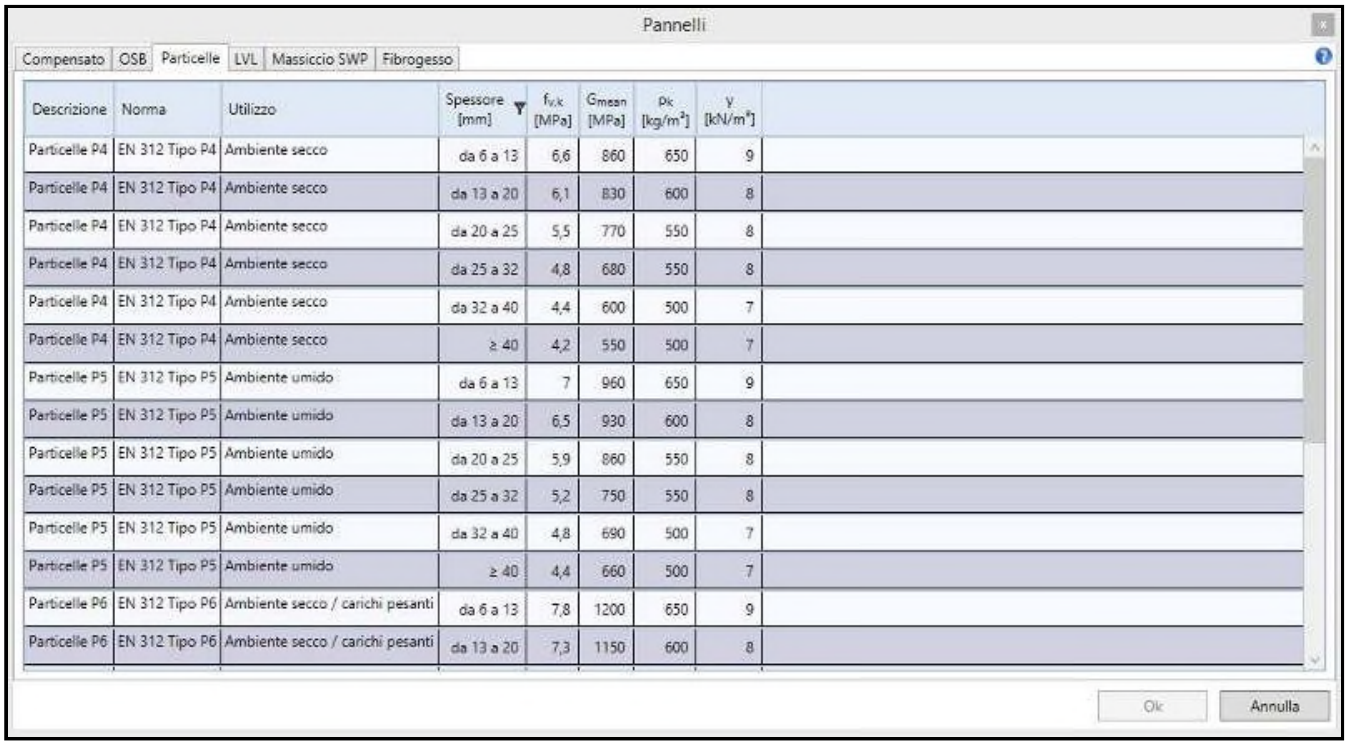
4.1.2.2 OSB



The tables provide the mechanical properties values of different *use classes*: panels OSB/2 in dry conditions, panels OSB/3 in humid conditions and panels OSB/4 for humid conditions and heavy loads according to EN 300:

- Thickness;
- $f_{v,k}$: Shear strength;
- G_{mean}: Mean value of Shear modulus;
- ρ_k: Characteristic density;
- ρ_m: Mean density;
- γ:Volumetric weight;

4.1.2.3 Particleboards

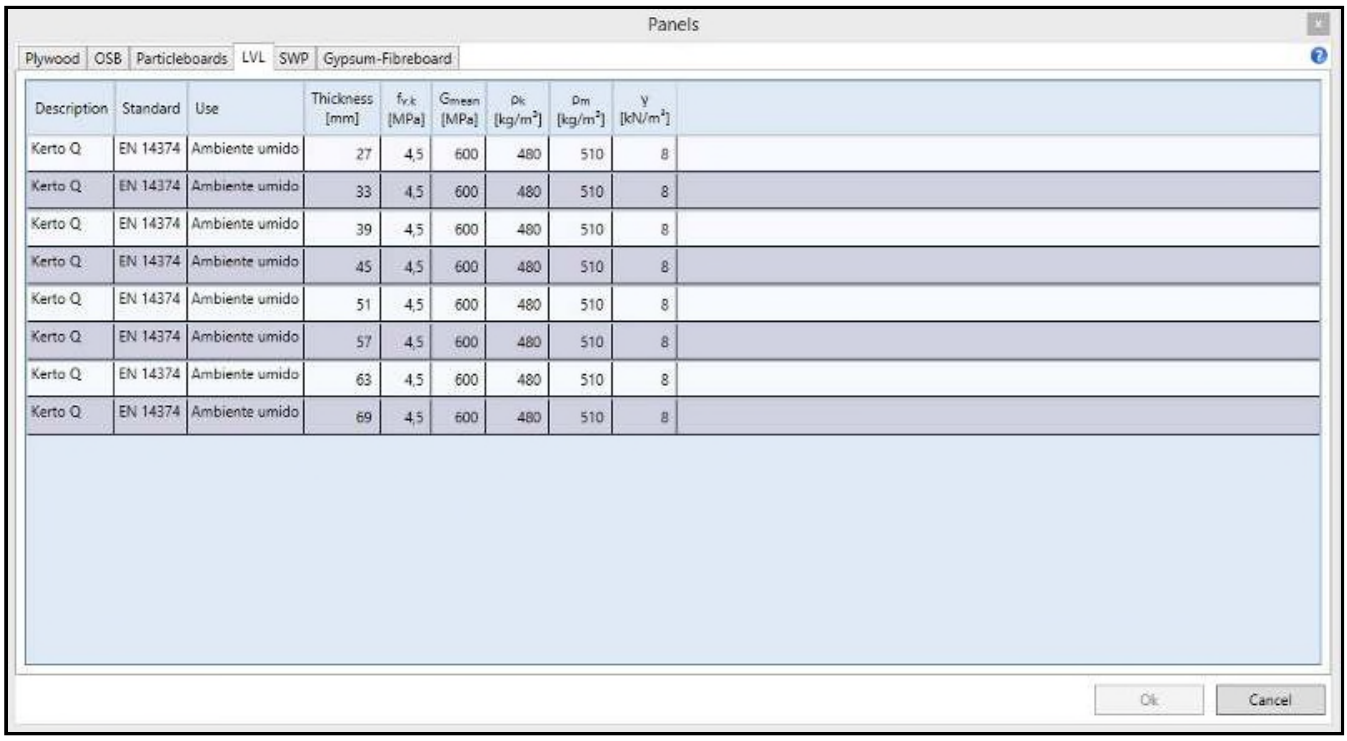


Descrizione	Norma	Utilizzo	Spessore [mm]	$f_{v,k}$ [MPa]	G_{mean} [MPa]	ρ_k [kg/m³]	γ [kN/m³]
Particelle P4	EN 312 Tipo P4	Ambiente secco	da 6 a 13	6,6	860	650	9
Particelle P4	EN 312 Tipo P4	Ambiente secco	da 13 a 20	6,1	830	600	8
Particelle P4	EN 312 Tipo P4	Ambiente secco	da 20 a 25	5,5	770	550	8
Particelle P4	EN 312 Tipo P4	Ambiente secco	da 25 a 32	4,8	680	550	8
Particelle P4	EN 312 Tipo P4	Ambiente secco	da 32 a 40	4,4	600	500	7
Particelle P4	EN 312 Tipo P4	Ambiente secco	≥ 40	4,2	550	500	7
Particelle P5	EN 312 Tipo P5	Ambiente umido	da 6 a 13	7	960	650	9
Particelle P5	EN 312 Tipo P5	Ambiente umido	da 13 a 20	6,5	930	600	8
Particelle P5	EN 312 Tipo P5	Ambiente umido	da 20 a 25	5,9	860	550	8
Particelle P5	EN 312 Tipo P5	Ambiente umido	da 25 a 32	5,2	750	550	8
Particelle P5	EN 312 Tipo P5	Ambiente umido	da 32 a 40	4,8	690	500	7
Particelle P5	EN 312 Tipo P5	Ambiente umido	≥ 40	4,4	660	500	7
Particelle P6	EN 312 Tipo P6	Ambiente secco / carichi pesanti	da 6 a 13	7,8	1200	650	9
Particelle P6	EN 312 Tipo P6	Ambiente secco / carichi pesanti	da 13 a 20	7,3	1150	600	8

The tables provide, according to EN 312,the mechanical properties values of different particleboards: P4 type for the use in dry conditions, P5 in humid conditions, P6 for heavy loads and humid conditions:

- Thickness;
- $f_{v,k}$: Shear strength;
- G_{mean}: Mean value of Shear modulus;
- ρ_k: Characteristic density;
- ρ_m: Mean density;
- γ:Volumetric weight;

4.1.2.4 LVL (*Laminated Veneer Lumber*)

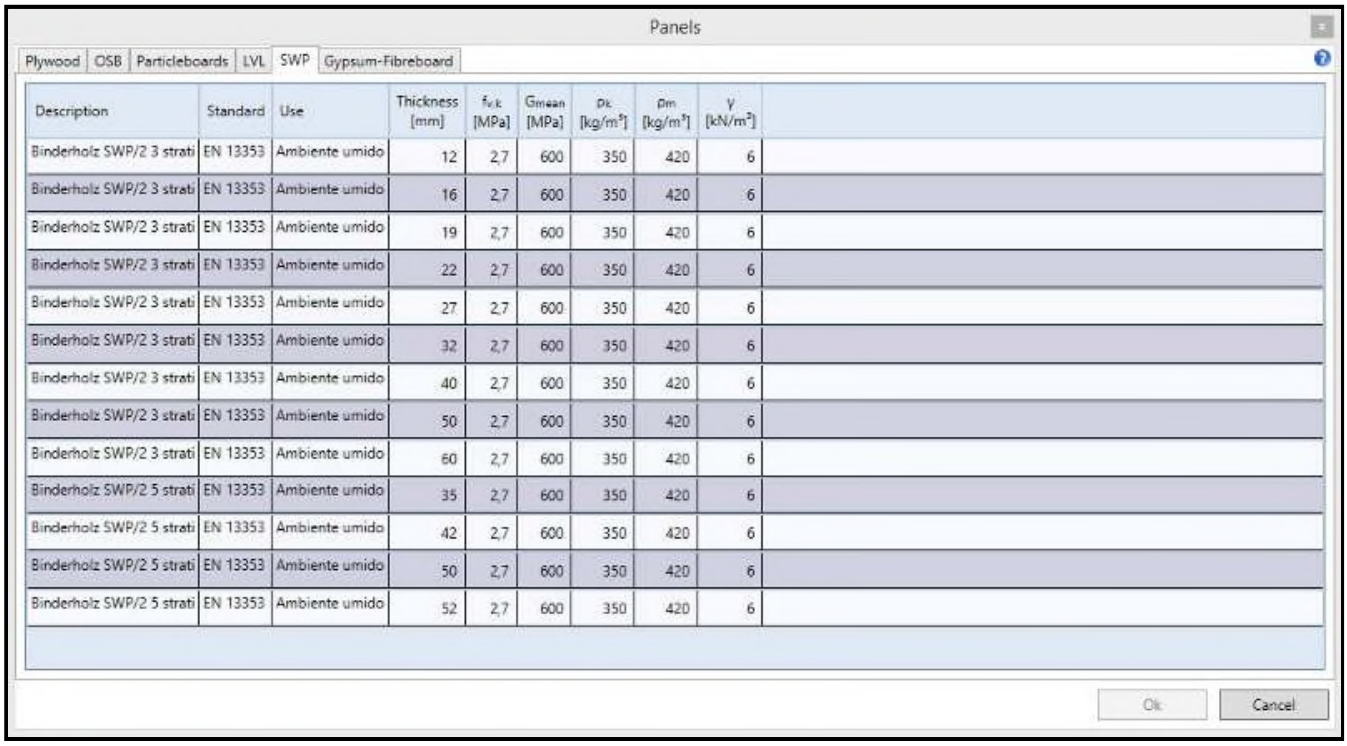


Description	Standard	Use	Thickness [mm]	$f_{v,k}$ [MPa]	G_{mean} [MPa]	ρ_k [kg/m³]	ρ_m [kg/m³]	γ [kN/m³]
Kerto Q	EN 14374	Ambiente umido	27	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	33	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	39	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	45	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	51	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	57	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	63	4,5	600	480	510	8
Kerto Q	EN 14374	Ambiente umido	69	4,5	600	480	510	8

The table provides, in addition to the product description and the use conditions, according to EN 14374, the following mechanical and physical properties values:

- Thickness;
- $f_{v,k}$: Shear strength;
- G_{mean}: Mean value of Shear modulus;
- ρ_k: Characteristic density;
- ρ_m: Mean density;
- γ:Volumetric weight;

4.1.2.5 SWP (*Solid Wood Panels*)

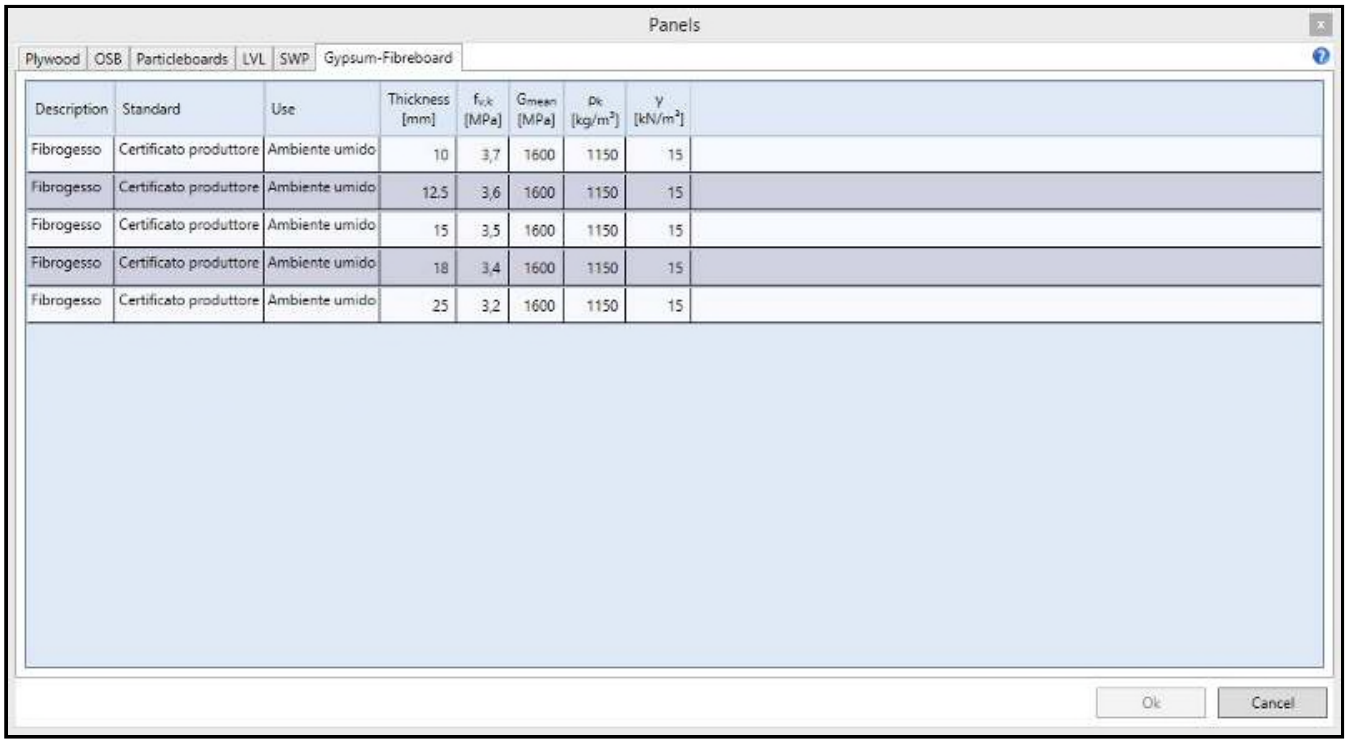


Description	Standard	Use	Thickness [mm]	$f_{v,k}$ [MPa]	G_{mean} [MPa]	ρ_k [kg/m³]	ρ_m [kg/m³]	γ [kN/m³]
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	12	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	16	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	19	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	22	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	27	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	32	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	40	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	50	2,7	600	350	420	6
Binderholz SWP/2 3 strati	EN 13353	Ambiente umido	60	2,7	600	350	420	6
Binderholz SWP/2 5 strati	EN 13353	Ambiente umido	35	2,7	600	350	420	6
Binderholz SWP/2 5 strati	EN 13353	Ambiente umido	42	2,7	600	350	420	6
Binderholz SWP/2 5 strati	EN 13353	Ambiente umido	50	2,7	600	350	420	6
Binderholz SWP/2 5 strati	EN 13353	Ambiente umido	52	2,7	600	350	420	6

The table provides, in addition to the product description and the use conditions, according to EN 13353, the following mechanical and physical properties values:

- Thickness;
- $f_{v,k}$: Shear strength;
- G_{mean}: Mean value of Shear modulus;
- ρ_k: Characteristic density;
- ρ_m: Mean density;
- γ:Volumetric weight;

4.1.2.6 Gypsum-Fibreboard



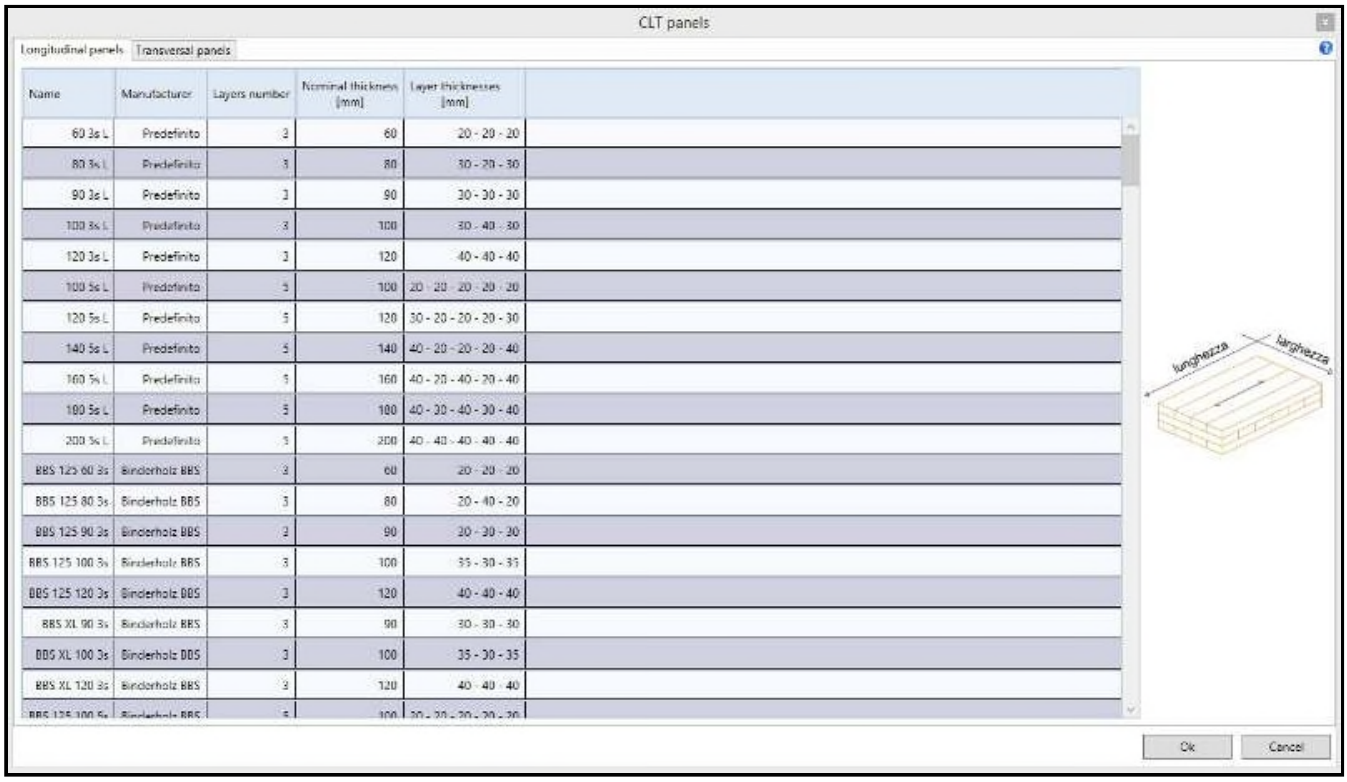
The table provides, in addition to the product description and the use conditions, the following mechanical and physical properties values:

- Thickness;
- $f_{v,k}$: Shear strength;
- G_{mean}: Mean value of Shear modulus;
- ρ_k: Characteristic density;
- ρ_m: Mean density;
- γ:Volumetric weight;

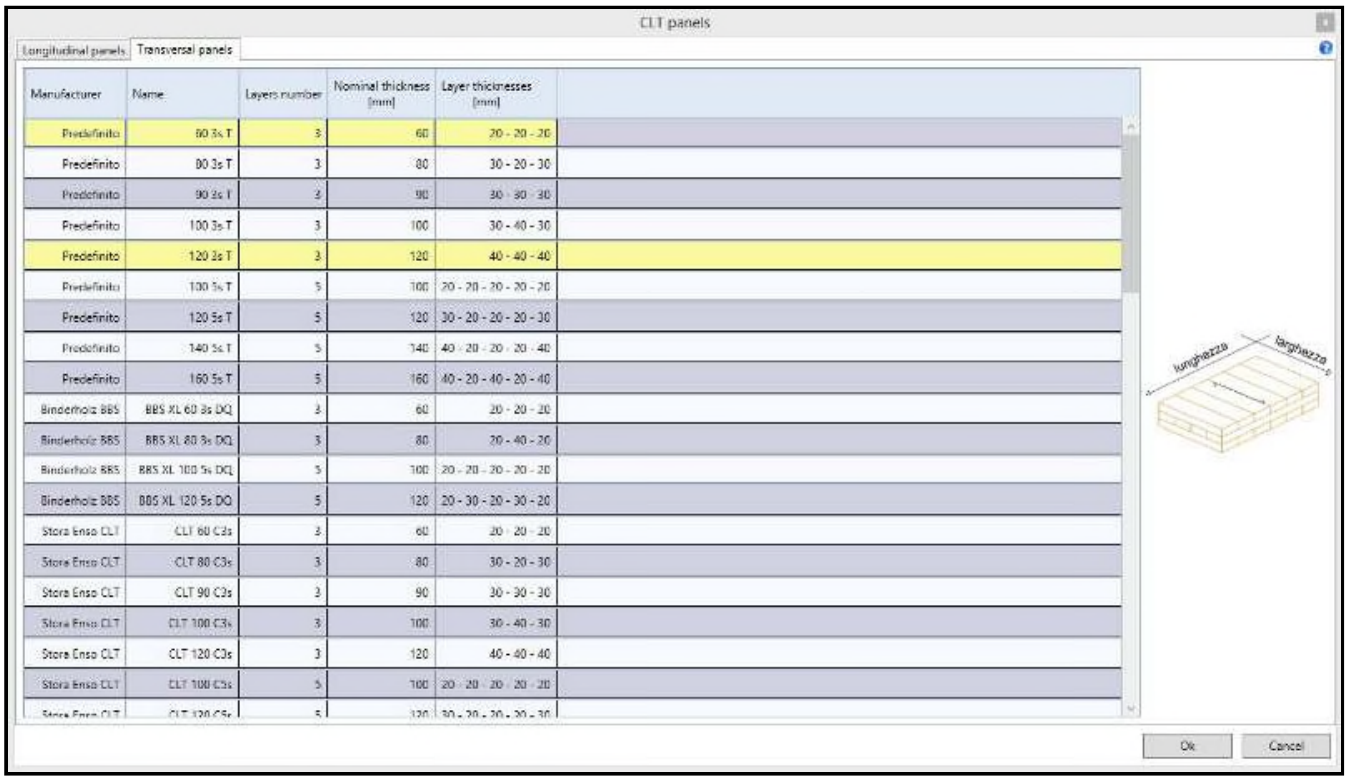
4.1.3 CLT

The *CLT command* provides the access to the performance table of the CLT panels (Cross laminated timber). There are two types of panels depending on the external layers orientation:

External layers parallel to the longitudinal direction



External layers parallel to the transversal direction



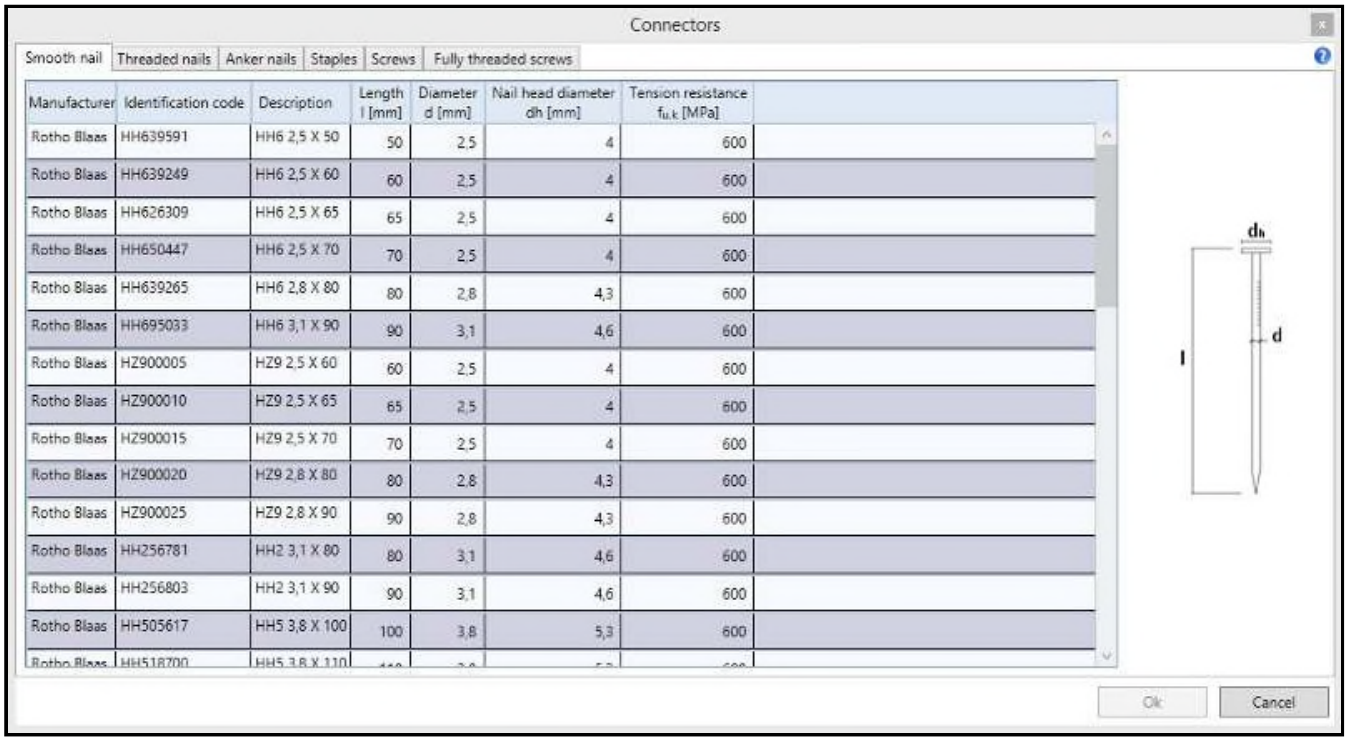
The table provide the mean product data:

- Manufacturer;
- Code describing the product (for instance.: 90 3s L) nominal thickness, the number of layers, the external layers orientation (L: longitudinal orientation; T: transversal orientation);
- Nominal thickness;
- Layers stratigraphy.

4.1.4 Fasteners

The *Fasteners* command allows the user to consult the performance tables with the technical data of the following products:

4.1.4.1 Smooth nail

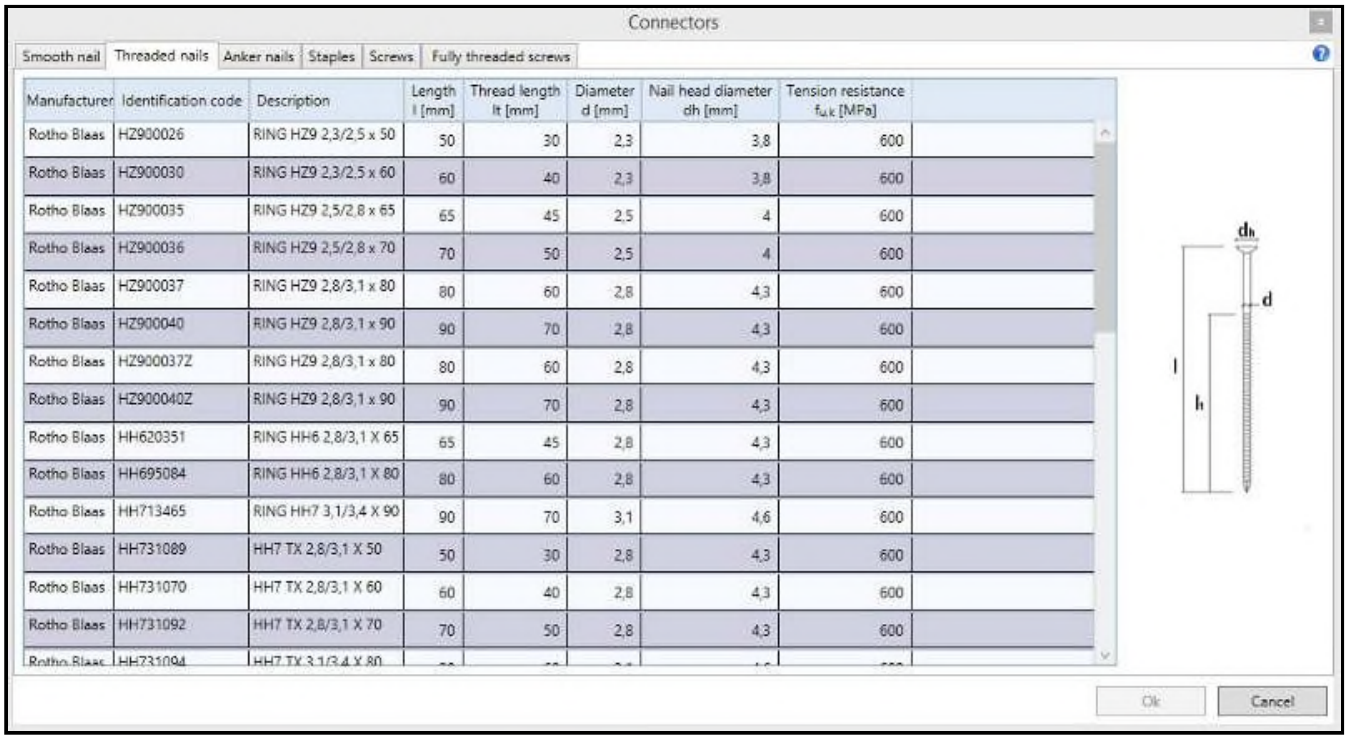


Manufacturer	Identification code	Description	Length l [mm]	Diameter d [mm]	Nail head diameter dh [mm]	Tension resistance $f_{u,k}$ [MPa]
Rotho Blaas	HH639591	HH6 2,5 X 50	50	2,5	4	600
Rotho Blaas	HH639249	HH6 2,5 X 60	60	2,5	4	600
Rotho Blaas	HH626309	HH6 2,5 X 65	65	2,5	4	600
Rotho Blaas	HH650447	HH6 2,5 X 70	70	2,5	4	600
Rotho Blaas	HH639265	HH6 2,8 X 80	80	2,8	4,3	600
Rotho Blaas	HH695033	HH6 3,1 X 90	90	3,1	4,6	600
Rotho Blaas	HZ900005	HZ9 2,5 X 60	60	2,5	4	600
Rotho Blaas	HZ900010	HZ9 2,5 X 65	65	2,5	4	600
Rotho Blaas	HZ900015	HZ9 2,5 X 70	70	2,5	4	600
Rotho Blaas	HZ900020	HZ9 2,8 X 80	80	2,8	4,3	600
Rotho Blaas	HZ900025	HZ9 2,8 X 90	90	2,8	4,3	600
Rotho Blaas	HH256781	HH2 3,1 X 80	80	3,1	4,6	600
Rotho Blaas	HH256803	HH2 3,1 X 90	90	3,1	4,6	600
Rotho Blaas	HH505617	HH5 3,8 X 100	100	3,8	5,3	600
Rotho Blaas	HH518700	HH5 3,8 X 110				

The table provides the manufacturer, the identification code, the description and the following geometrical and mechanical values:

- l: Length;
- d: Diameter;
- d_h: Head diameter;
- $f_{u,k}$: Tension strength;

4.1.4.2 Threaded nails

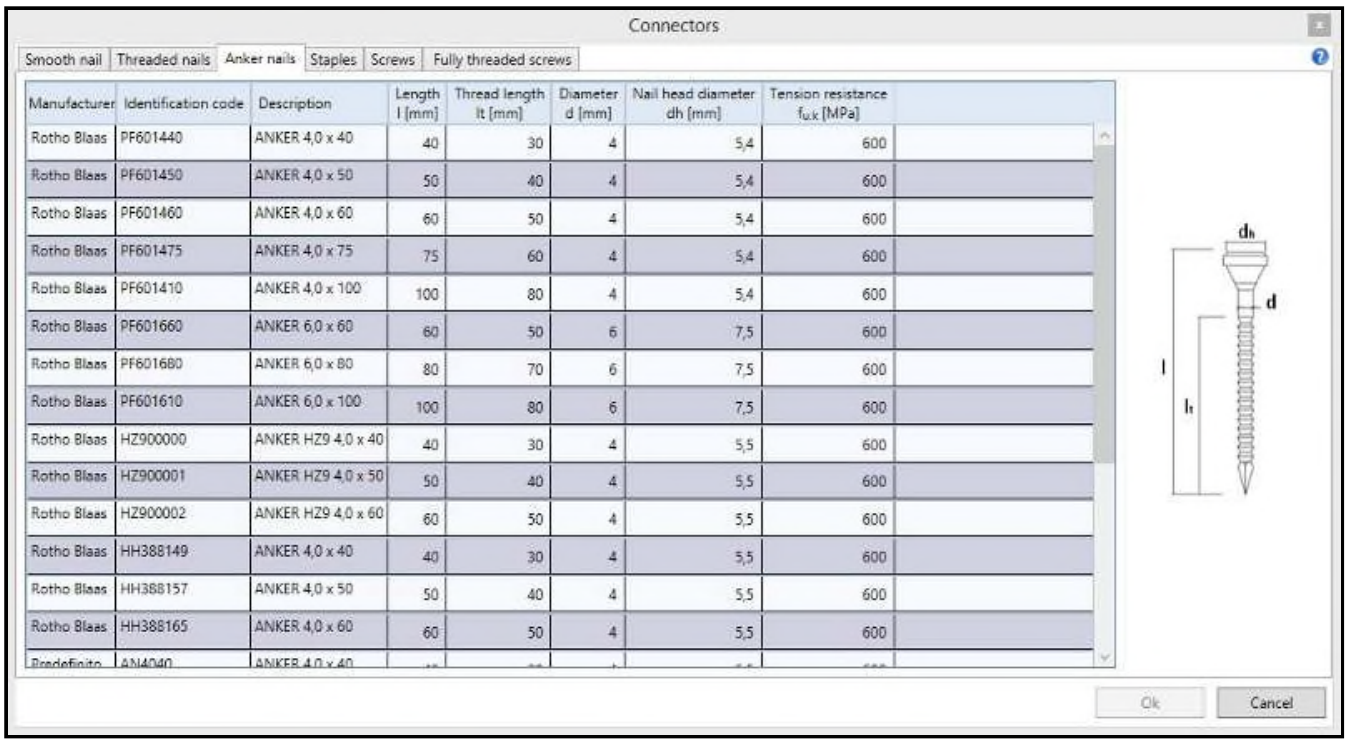


Manufacturer	Identification code	Description	Length l [mm]	Thread length lt [mm]	Diameter d [mm]	Nail head diameter dh [mm]	Tension resistance $f_{u,k}$ [MPa]
Rotho Blaas	HZ900026	RING HZ9 2,3/2,5 x 50	50	30	2,3	3,8	600
Rotho Blaas	HZ900030	RING HZ9 2,3/2,5 x 60	60	40	2,3	3,8	600
Rotho Blaas	HZ900035	RING HZ9 2,5/2,8 x 65	65	45	2,5	4	600
Rotho Blaas	HZ900036	RING HZ9 2,5/2,8 x 70	70	50	2,5	4	600
Rotho Blaas	HZ900037	RING HZ9 2,8/3,1 x 80	80	60	2,8	4,3	600
Rotho Blaas	HZ900040	RING HZ9 2,8/3,1 x 90	90	70	2,8	4,3	600
Rotho Blaas	HZ900037Z	RING HZ9 2,8/3,1 x 80	80	60	2,8	4,3	600
Rotho Blaas	HZ900040Z	RING HZ9 2,8/3,1 x 90	90	70	2,8	4,3	600
Rotho Blaas	HH620351	RING HH6 2,8/3,1 X 65	65	45	2,8	4,3	600
Rotho Blaas	HH695084	RING HH6 2,8/3,1 X 80	80	60	2,8	4,3	600
Rotho Blaas	HH713465	RING HH7 3,1/3,4 X 90	90	70	3,1	4,6	600
Rotho Blaas	HH731089	HH7 TX 2,8/3,1 X 50	50	30	2,8	4,3	600
Rotho Blaas	HH731070	HH7 TX 2,8/3,1 X 60	60	40	2,8	4,3	600
Rotho Blaas	HH731092	HH7 TX 2,8/3,1 X 70	70	50	2,8	4,3	600
Rotho Blaas	HH731094	HH7 TX 3,1/3,4 X 80					

The table provides the manufacturer, the identification code, the description and the following geometrical and mechanical values:

- l: Length;
- l_t: Threaded nails;
- d: Diameter;
- d_h: Head diameter;
- $f_{u,k}$: Tension resistance.

4.1.4.3 Anker nails

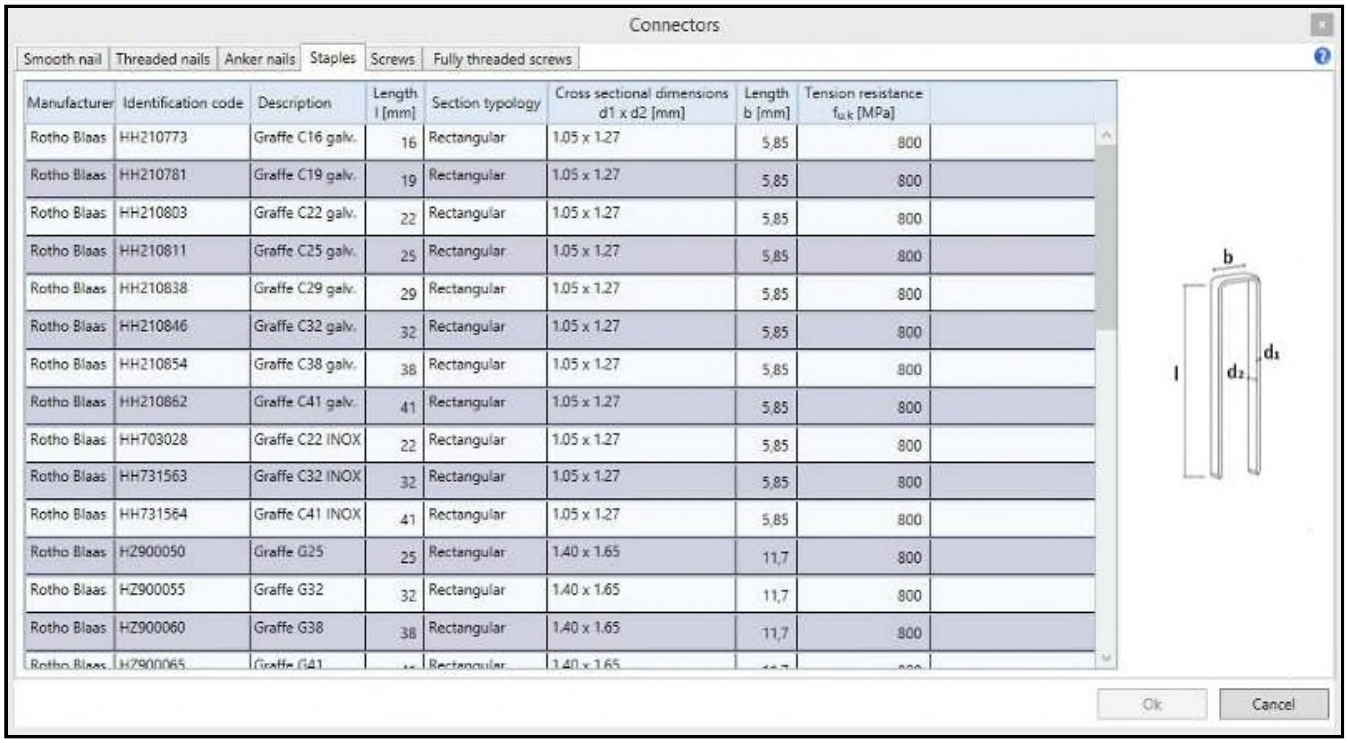


Manufacturer	Identification code	Description	Length l [mm]	Thread length lt [mm]	Diameter d [mm]	Nail head diameter dh [mm]	Tension resistance $f_{u,k}$ [MPa]
Rotho Blaas	PF601440	ANKER 4,0 x 40	40	30	4	5,4	600
Rotho Blaas	PF601450	ANKER 4,0 x 50	50	40	4	5,4	600
Rotho Blaas	PF601460	ANKER 4,0 x 60	60	50	4	5,4	600
Rotho Blaas	PF601475	ANKER 4,0 x 75	75	60	4	5,4	600
Rotho Blaas	PF601410	ANKER 4,0 x 100	100	80	4	5,4	600
Rotho Blaas	PF601660	ANKER 6,0 x 60	60	50	6	7,5	600
Rotho Blaas	PF601680	ANKER 6,0 x 80	80	70	6	7,5	600
Rotho Blaas	PF601610	ANKER 6,0 x 100	100	80	6	7,5	600
Rotho Blaas	HZ900000	ANKER HZ9 4,0 x 40	40	30	4	5,5	600
Rotho Blaas	HZ900001	ANKER HZ9 4,0 x 50	50	40	4	5,5	600
Rotho Blaas	HZ900002	ANKER HZ9 4,0 x 60	60	50	4	5,5	600
Rotho Blaas	HH388149	ANKER 4,0 x 40	40	30	4	5,5	600
Rotho Blaas	HH388157	ANKER 4,0 x 50	50	40	4	5,5	600
Rotho Blaas	HH388165	ANKER 4,0 x 60	60	50	4	5,5	600
Predefinito	AN4040	ANKER 4,0 x 40					

The table provides the manufacturer, the identification code, the description and the following geometrical and mechanical values:

- l: Length;
- l_t: Threaded nails;
- d: Diameter;
- d_h: Head diameter;
- $f_{u,k}$: Tension resistance.

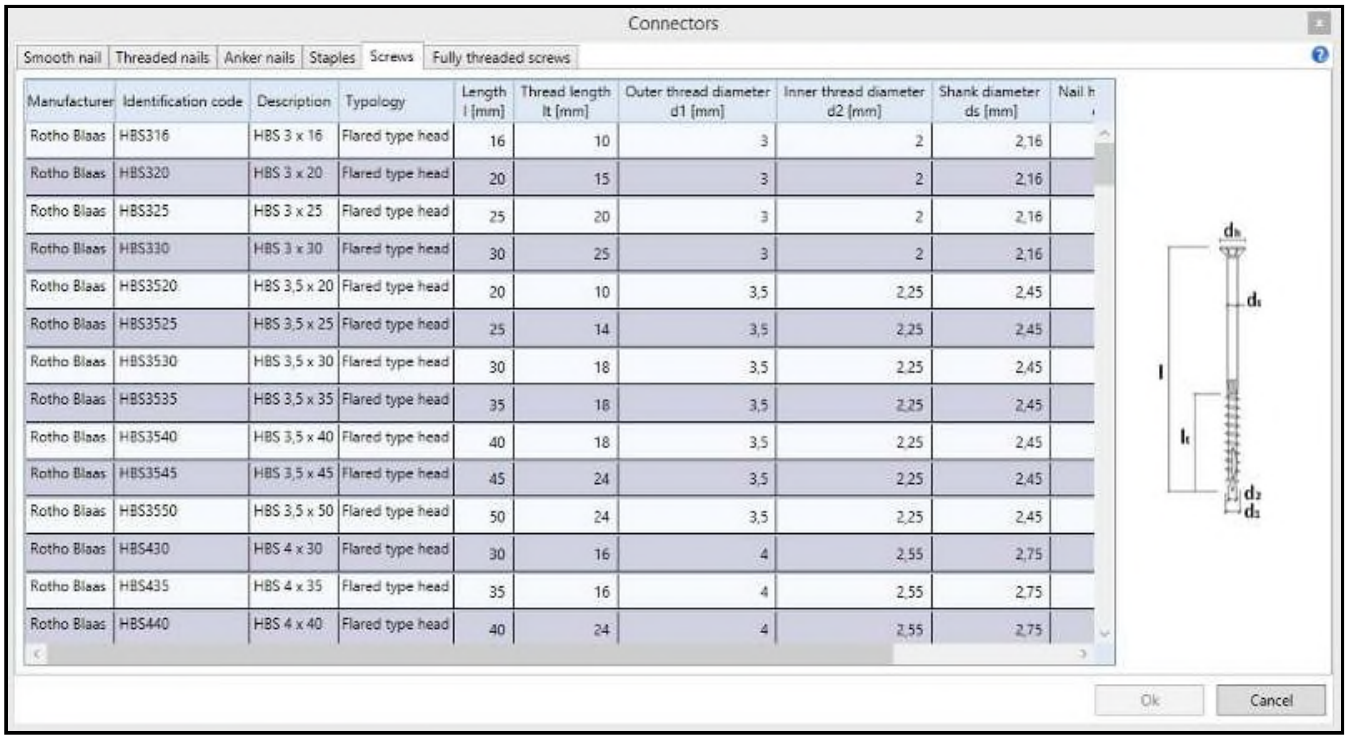
4.1.4.4 Staples



The table provides the manufacturer, the identification code, the description and the following geometrical and mechanical values:

- l: Length;
- Section typology;
- Cross sectional dimensions d_1 x d_2;
- b: length;
- $f_{u,k}$: tension strength;

4.1.4.5 Screws

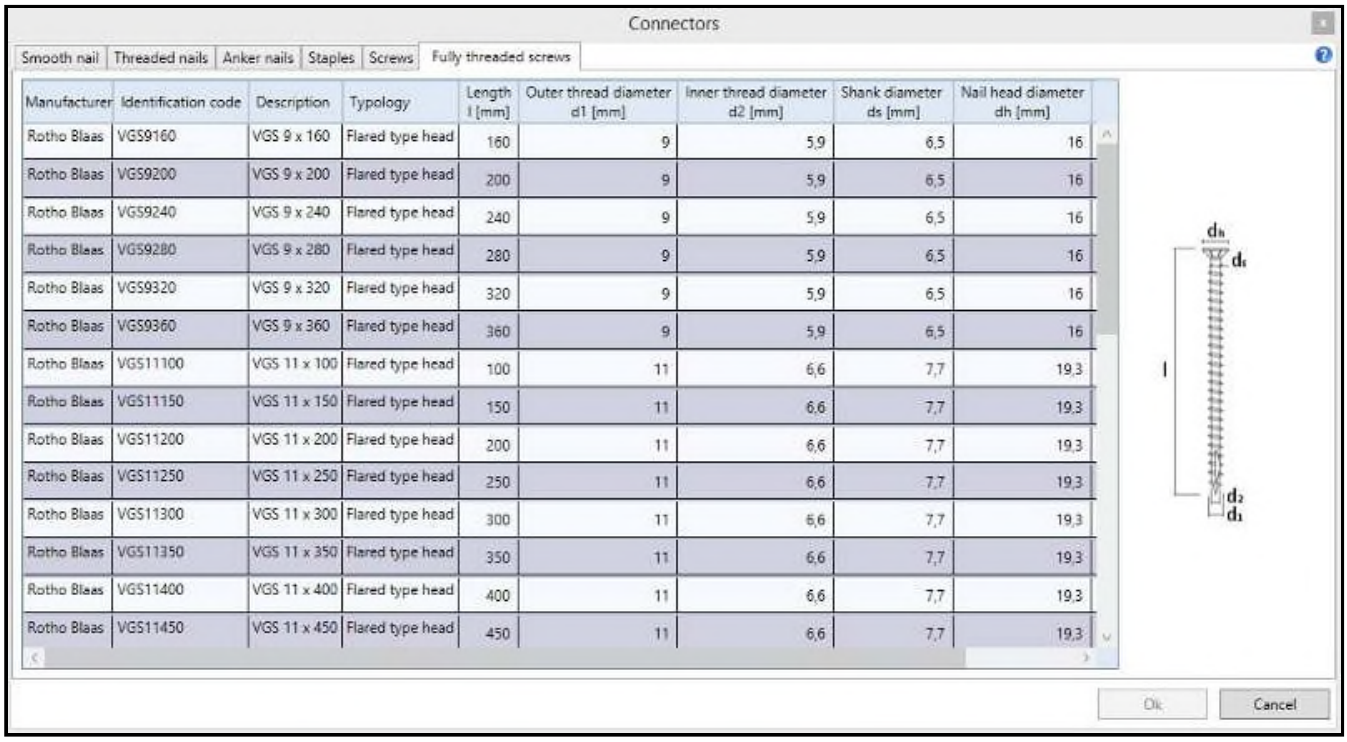


Manufacturer	Identification code	Description	Typology	Length l [mm]	Thread length lt [mm]	Outer thread diameter d1 [mm]	Inner thread diameter d2 [mm]	Shank diameter ds [mm]	Nail h
Rotho Blaas	HBS316	HBS 3 x 16	Flared type head	16	10	3	2	2,16	
Rotho Blaas	HBS320	HBS 3 x 20	Flared type head	20	15	3	2	2,16	
Rotho Blaas	HBS325	HBS 3 x 25	Flared type head	25	20	3	2	2,16	
Rotho Blaas	HBS330	HBS 3 x 30	Flared type head	30	25	3	2	2,16	
Rotho Blaas	HBS3520	HBS 3,5 x 20	Flared type head	20	10	3,5	2,25	2,45	
Rotho Blaas	HBS3525	HBS 3,5 x 25	Flared type head	25	14	3,5	2,25	2,45	
Rotho Blaas	HBS3530	HBS 3,5 x 30	Flared type head	30	18	3,5	2,25	2,45	
Rotho Blaas	HBS3535	HBS 3,5 x 35	Flared type head	35	18	3,5	2,25	2,45	
Rotho Blaas	HBS3540	HBS 3,5 x 40	Flared type head	40	18	3,5	2,25	2,45	
Rotho Blaas	HBS3545	HBS 3,5 x 45	Flared type head	45	24	3,5	2,25	2,45	
Rotho Blaas	HBS3550	HBS 3,5 x 50	Flared type head	50	24	3,5	2,25	2,45	
Rotho Blaas	HBS430	HBS 4 x 30	Flared type head	30	16	4	2,55	2,75	
Rotho Blaas	HBS435	HBS 4 x 35	Flared type head	35	16	4	2,55	2,75	
Rotho Blaas	HBS440	HBS 4 x 40	Flared type head	40	24	4	2,55	2,75	

The table provides the manufacturer, the identification code, the description and the following geometrical and mechanical values:

- Type (Flared type head, large type head);
- l: length;
- d_1: Outer thread diameter;
- d_2: Inner thread diameter;
- d_s: Shank diameter;
- d_h: Nail head diameter;
- $f_{u,k}$: Tension strength;

4.1.4.6 Fully threaded screws



Manufacturer	Identification code	Description	Typology	Length l [mm]	Outer thread diameter d1 [mm]	Inner thread diameter d2 [mm]	Shank diameter ds [mm]	Nail head diameter dh [mm]
Rotho Blaas	VGS9160	VGS 9 x 160	Flared type head	160	9	5,9	6,5	16
Rotho Blaas	VGS9200	VGS 9 x 200	Flared type head	200	9	5,9	6,5	16
Rotho Blaas	VGS9240	VGS 9 x 240	Flared type head	240	9	5,9	6,5	16
Rotho Blaas	VGS9280	VGS 9 x 280	Flared type head	280	9	5,9	6,5	16
Rotho Blaas	VGS9320	VGS 9 x 320	Flared type head	320	9	5,9	6,5	16
Rotho Blaas	VGS9360	VGS 9 x 360	Flared type head	360	9	5,9	6,5	16
Rotho Blaas	VGS11100	VGS 11 x 100	Flared type head	100	11	6,6	7,7	19,3
Rotho Blaas	VGS11150	VGS 11 x 150	Flared type head	150	11	6,6	7,7	19,3
Rotho Blaas	VGS11200	VGS 11 x 200	Flared type head	200	11	6,6	7,7	19,3
Rotho Blaas	VGS11250	VGS 11 x 250	Flared type head	250	11	6,6	7,7	19,3
Rotho Blaas	VGS11300	VGS 11 x 300	Flared type head	300	11	6,6	7,7	19,3
Rotho Blaas	VGS11350	VGS 11 x 350	Flared type head	350	11	6,6	7,7	19,3
Rotho Blaas	VGS11400	VGS 11 x 400	Flared type head	400	11	6,6	7,7	19,3
Rotho Blaas	VGS11450	VGS 11 x 450	Flared type head	450	11	6,6	7,7	19,3

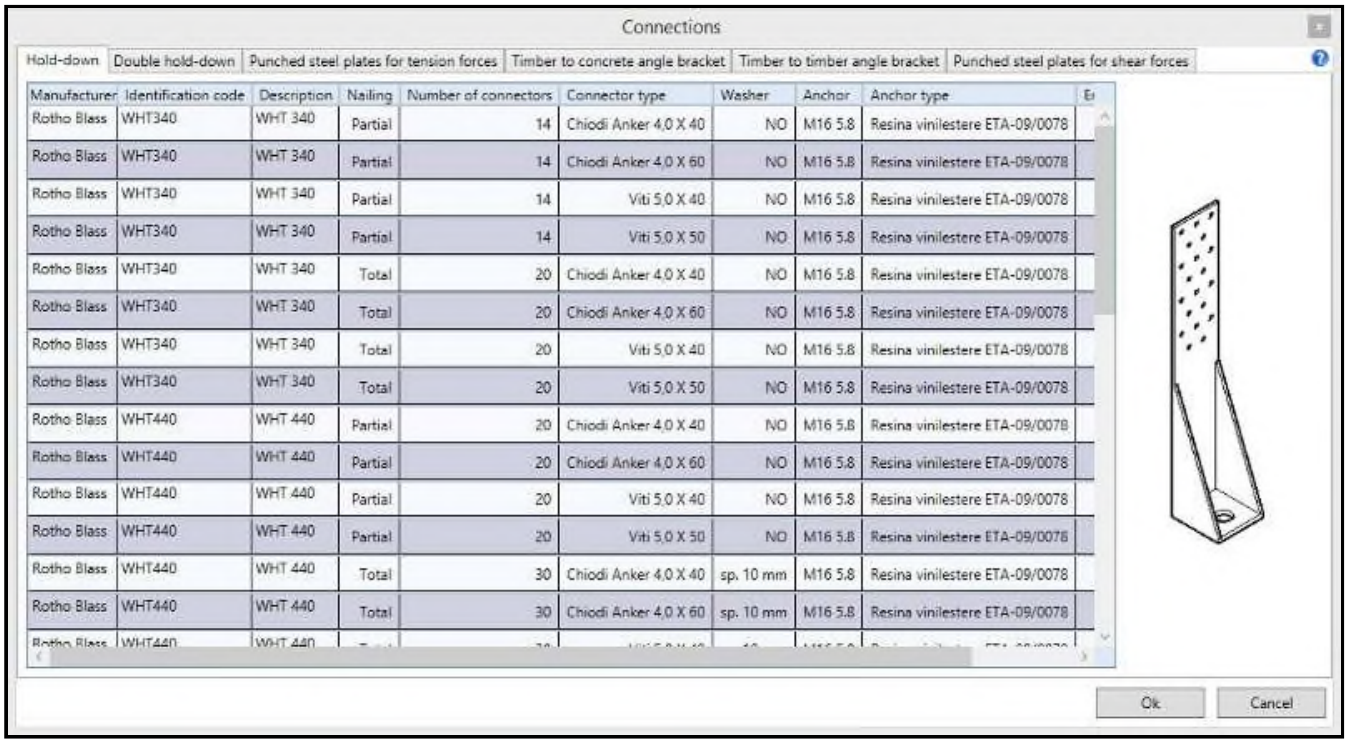
The table provides the manufacturer, the identification code, the description and the following geometrical and mechanical values:

- Type (Flared type head, large type head);
- l: length;
- d_1: Outer thread diameter;
- d_2: Inner thread diameter;
- d_s: Shank diameter;
- d_h: Nail head diameter;
- $f_{u,k}$: Tension strength;

4.1.5 Anchors

The *Connectors elements* command provides the access to the performance table of anchors:

4.1.5.1 Hold-Down

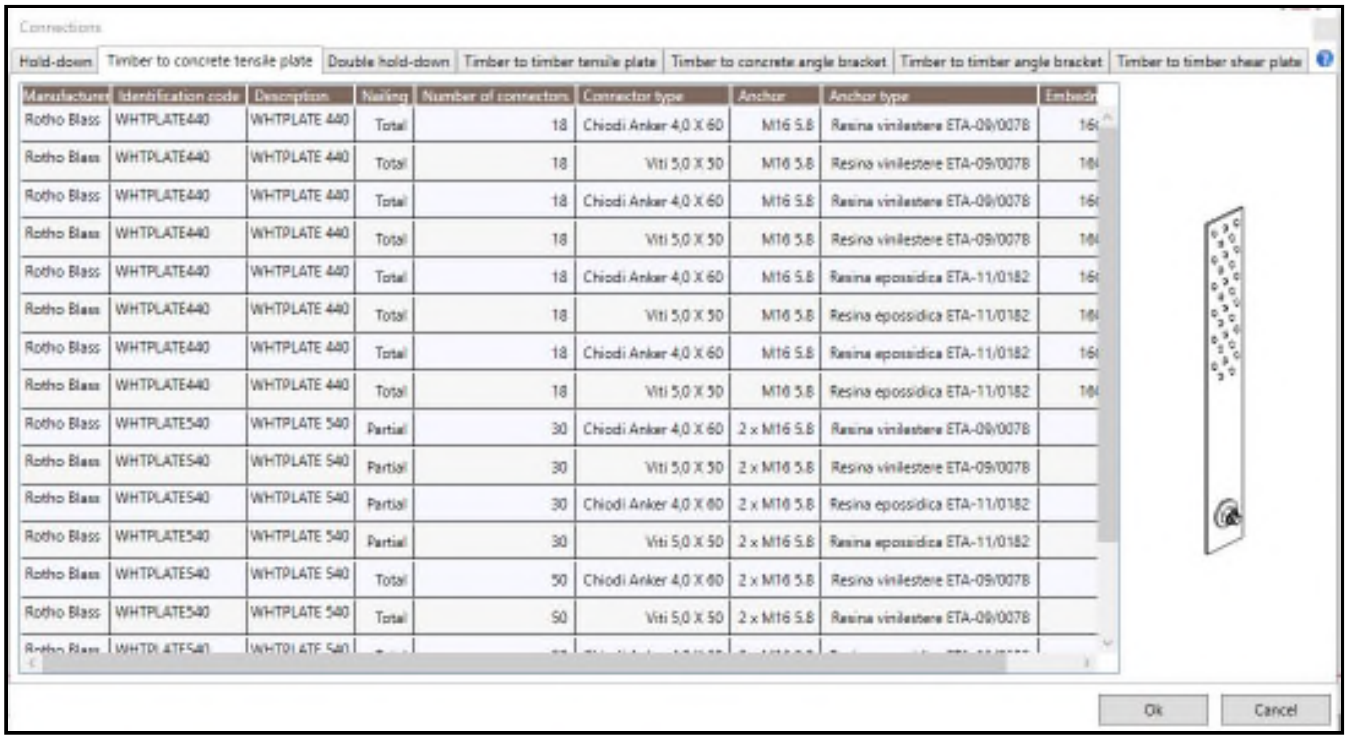


Manufacturer	Identification code	Description	Nailing	Number of connectors	Connector type	Washer	Anchor	Anchor type	Em
Rotho Blass	WHT340	WHT 340	Partial	14	Chiodi Anker 4,0 X 40	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Partial	14	Chiodi Anker 4,0 X 60	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Partial	14	Viti 5,0 X 40	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Partial	14	Viti 5,0 X 50	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Total	20	Chiodi Anker 4,0 X 40	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Total	20	Chiodi Anker 4,0 X 60	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Total	20	Viti 5,0 X 40	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT340	WHT 340	Total	20	Viti 5,0 X 50	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT440	WHT 440	Partial	20	Chiodi Anker 4,0 X 40	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT440	WHT 440	Partial	20	Chiodi Anker 4,0 X 60	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT440	WHT 440	Partial	20	Viti 5,0 X 40	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT440	WHT 440	Partial	20	Viti 5,0 X 50	NO	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT440	WHT 440	Total	30	Chiodi Anker 4,0 X 40	sp. 10 mm	M16 5.8	Resina vinilestere ETA-09/0078	
Rotho Blass	WHT440	WHT 440	Total	30	Chiodi Anker 4,0 X 60	sp. 10 mm	M16 5.8	Resina vinilestere ETA-09/0078	

The table provides the manufacturer, the identification code, the description, the geometrical and mechanical values, and the following implementing informations:

- Nailing: partial or total;
- Connectors number: number of connectors used;
- Connector type: nailed or screwed;
- Washer: if the washer are present they are identified by the thickness;
- Anchor: anchor type used in the anchoring system;
- Anchoring type: type used to fix the anchor;
- Embedment depth;

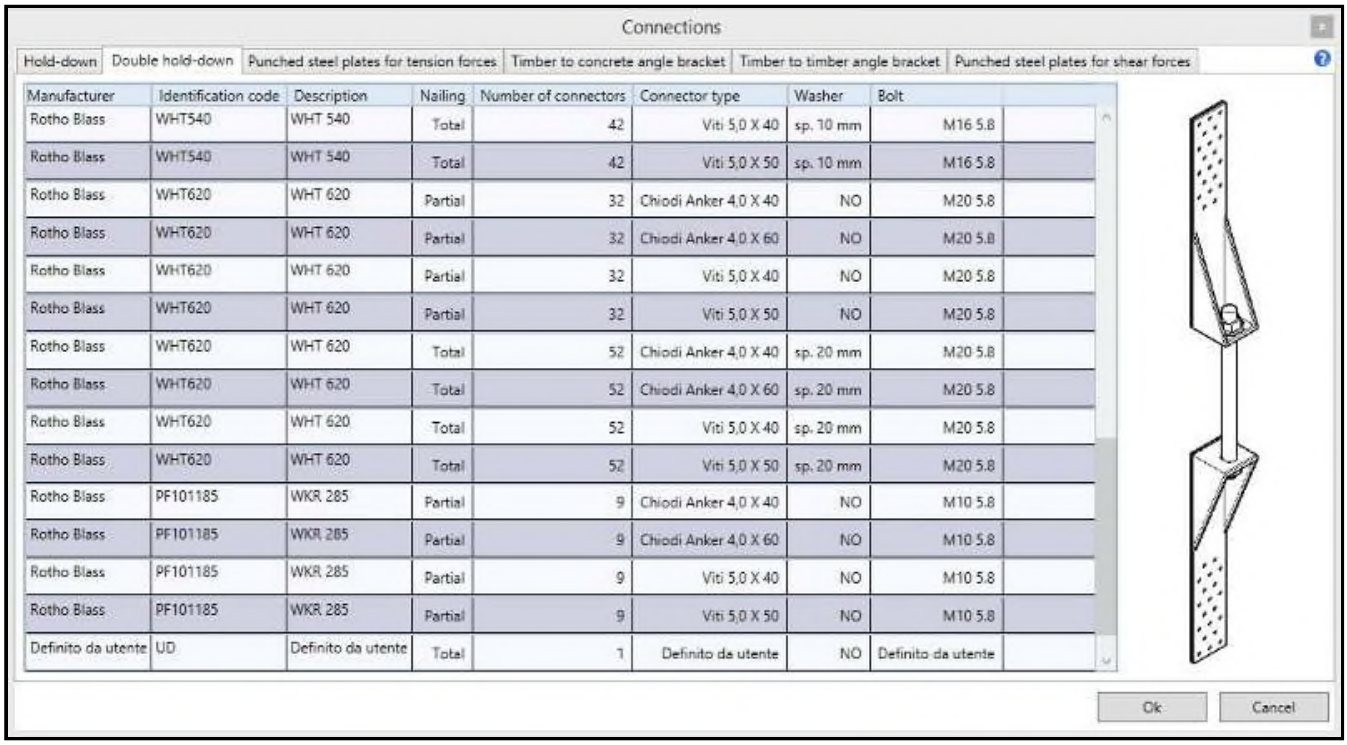
4.1.5.2 Timber to concrete tensile plate



The table provides the manufacturer, the identification code, the description, the geometrical and mechanical values, and the following implementing informations:

- Nailing: partial or total;
- Connectors number: number of connectors used;
- Connector type: nailed or screwed;
- Anchor: anchor type used in the anchoring system;
- Anchoring type: type used to fix the anchor;
- Embedment depth;

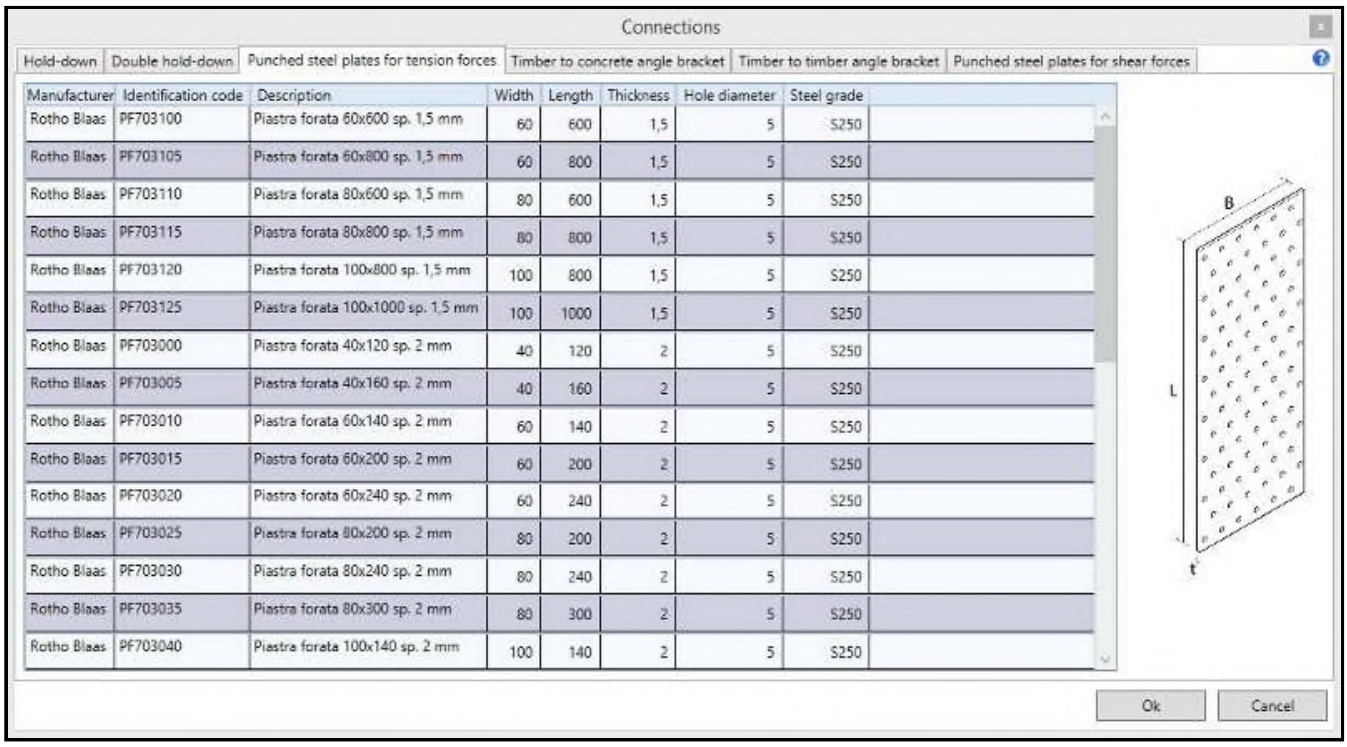
4.1.5.3 Double hold-down



The table provides the manufacturer,the identification code, the description, the geometrical and mechanical values, and the following implementing informations:

- Nailing: partial or total;
- Connectors number: number of connectors used;
- Connector type: nailed or screwed;
- Washer: if the washer are present they are identified by the thickness;
- Anchor: anchor type used in the anchoring system;

4.1.5.4 Timber to timber punched steel plates for tension forces

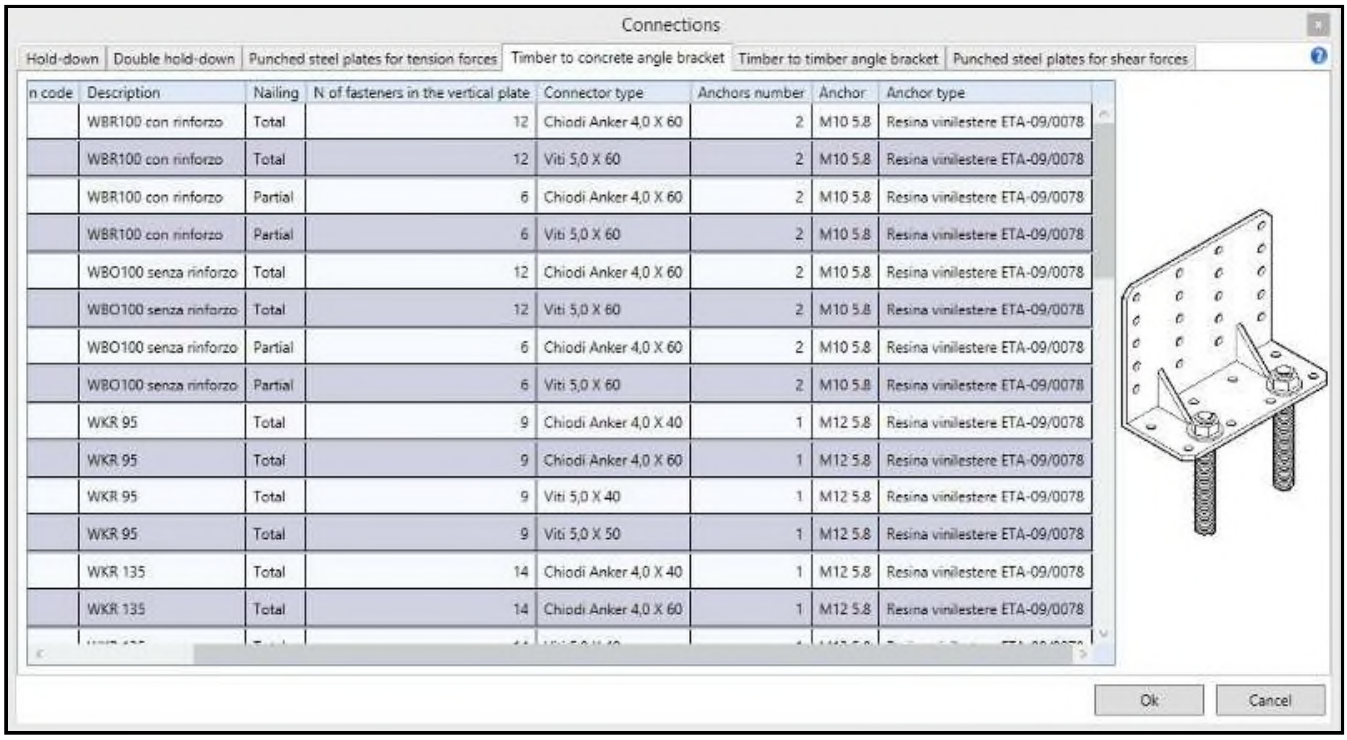


Manufacturer	Identification code	Description	Width	Length	Thickness	Hole diameter	Steel grade
Rotho Blaas	PF703100	Piastra forata 60x600 sp. 1,5 mm	60	600	1,5	5	S250
Rotho Blaas	PF703105	Piastra forata 60x800 sp. 1,5 mm	60	800	1,5	5	S250
Rotho Blaas	PF703110	Piastra forata 80x600 sp. 1,5 mm	80	600	1,5	5	S250
Rotho Blaas	PF703115	Piastra forata 80x800 sp. 1,5 mm	80	800	1,5	5	S250
Rotho Blaas	PF703120	Piastra forata 100x800 sp. 1,5 mm	100	800	1,5	5	S250
Rotho Blaas	PF703125	Piastra forata 100x1000 sp. 1,5 mm	100	1000	1,5	5	S250
Rotho Blaas	PF703000	Piastra forata 40x120 sp. 2 mm	40	120	2	5	S250
Rotho Blaas	PF703005	Piastra forata 40x160 sp. 2 mm	40	160	2	5	S250
Rotho Blaas	PF703010	Piastra forata 60x140 sp. 2 mm	60	140	2	5	S250
Rotho Blaas	PF703015	Piastra forata 60x200 sp. 2 mm	60	200	2	5	S250
Rotho Blaas	PF703020	Piastra forata 60x240 sp. 2 mm	60	240	2	5	S250
Rotho Blaas	PF703025	Piastra forata 80x200 sp. 2 mm	80	200	2	5	S250
Rotho Blaas	PF703030	Piastra forata 80x240 sp. 2 mm	80	240	2	5	S250
Rotho Blaas	PF703035	Piastra forata 80x300 sp. 2 mm	80	300	2	5	S250
Rotho Blaas	PF703040	Piastra forata 100x140 sp. 2 mm	100	140	2	5	S250

The table provides the manufacturer,the identification code, the description and the following geometrical and mechanical values:

- B: width;
- L: length;
- t : thickness;
- hole diameter;
- Steel grade;

4.1.5.5 Timber to concrete angle bracket

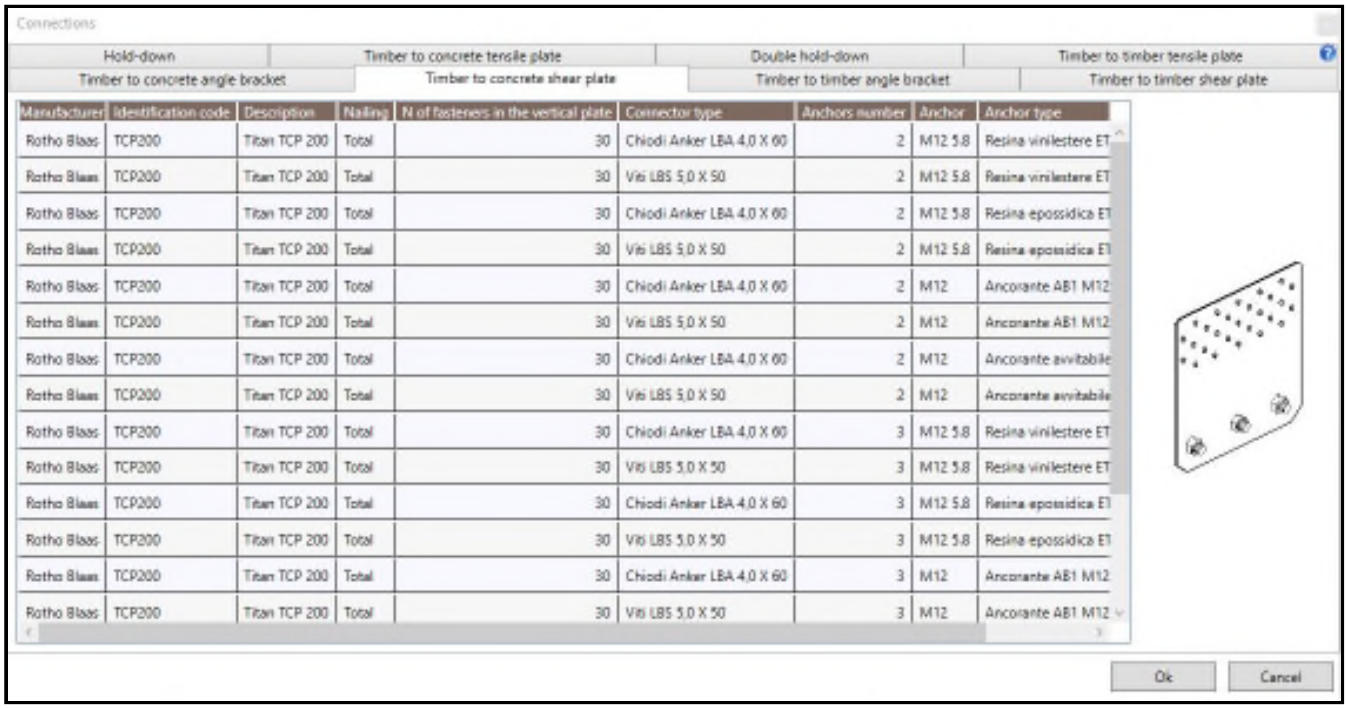


n code	Description	Nailing	N of fasteners in the vertical plate	Connector type	Anchors number	Anchor	Anchor type
	WBR100 con rinforzo	Total	12	Chiodi Anker 4,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBR100 con rinforzo	Total	12	Viti 5,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBR100 con rinforzo	Partial	6	Chiodi Anker 4,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBR100 con rinforzo	Partial	6	Viti 5,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBO100 senza rinforzo	Total	12	Chiodi Anker 4,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBO100 senza rinforzo	Total	12	Viti 5,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBO100 senza rinforzo	Partial	6	Chiodi Anker 4,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WBO100 senza rinforzo	Partial	6	Viti 5,0 X 60	2	M10 5.8	Resina vinilestere ETA-09/0078
	WKR 95	Total	9	Chiodi Anker 4,0 X 40	1	M12 5.8	Resina vinilestere ETA-09/0078
	WKR 95	Total	9	Chiodi Anker 4,0 X 60	1	M12 5.8	Resina vinilestere ETA-09/0078
	WKR 95	Total	9	Viti 5,0 X 40	1	M12 5.8	Resina vinilestere ETA-09/0078
	WKR 95	Total	9	Viti 5,0 X 50	1	M12 5.8	Resina vinilestere ETA-09/0078
	WKR 135	Total	14	Chiodi Anker 4,0 X 40	1	M12 5.8	Resina vinilestere ETA-09/0078
	WKR 135	Total	14	Chiodi Anker 4,0 X 60	1	M12 5.8	Resina vinilestere ETA-09/0078

The table provides the manufacturer,the identification code, the description and the following implementing informations:

- Nailing: partial or total;
- Connectors number: number of connectors used;
- Connector type: nailed or screwed;
- Number of anchors;
- Anchor: anchor type used;
- Anchoring type: type used to fix the anchor;

4.1.5.6 Timber to concrete shear plate

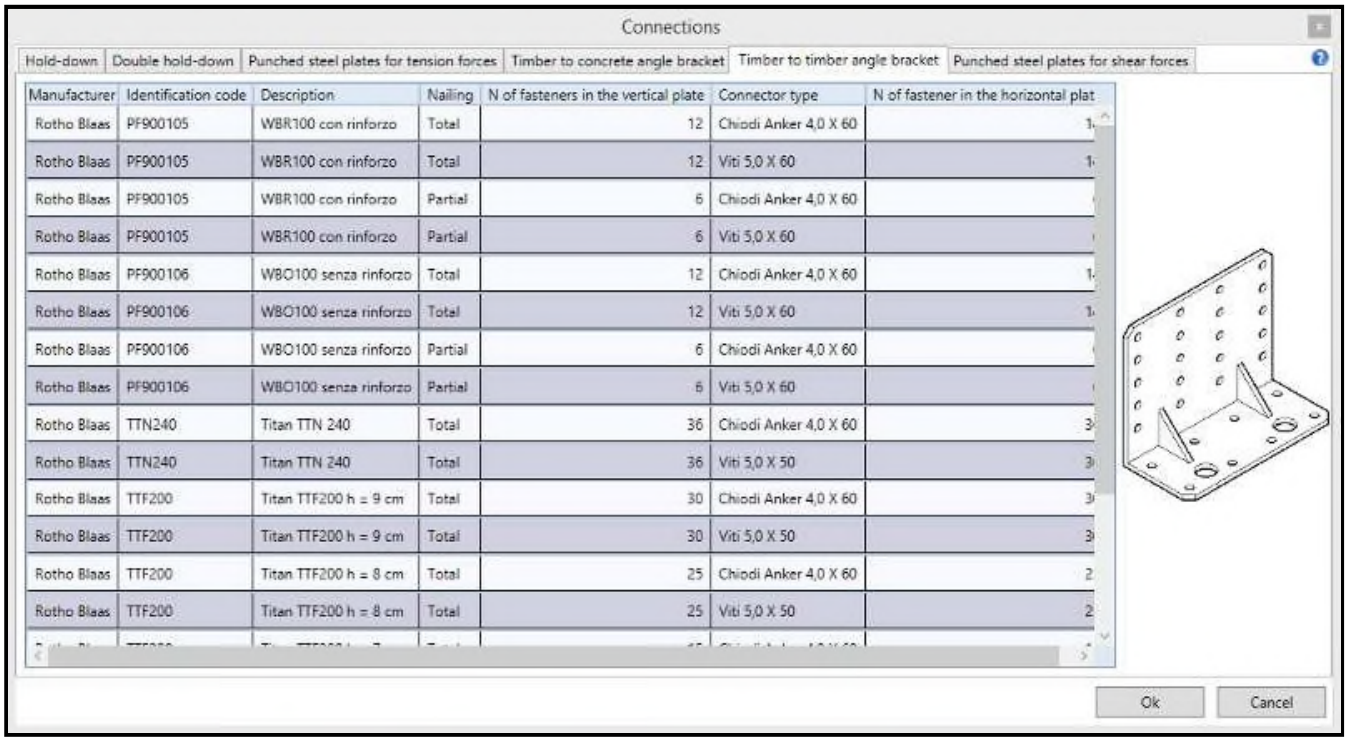


Manufacturer	Identification code	Description	Nailing	N of fasteners in the vertical plate	Connector type	Anchors number	Anchor	Anchor type
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	2	M12 5.8	Resina vinilestere ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	2	M12 5.8	Resina vinilestere ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	2	M12 5.8	Resina epossidica ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	2	M12 5.8	Resina epossidica ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	2	M12	Ancorante AB1 M12
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	2	M12	Ancorante AB1 M12
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	2	M12	Ancorante avvitabile
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	2	M12	Ancorante avvitabile
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	3	M12 5.8	Resina vinilestere ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	3	M12 5.8	Resina vinilestere ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	3	M12 5.8	Resina epossidica ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	3	M12 5.8	Resina epossidica ET
Rothoblaas	TCP200	Titan TCP 200	Total	30	Chiodi Anker LBA 4,0 X 60	3	M12	Ancorante AB1 M12
Rothoblaas	TCP200	Titan TCP 200	Total	30	Viti LBS 5,0 X 50	3	M12	Ancorante AB1 M12

The table provides the manufacturer,the identification code, the description and the following implementing informations:

- Nailing: partial or total;
- Connectors number: number of connectors used;
- Connector type: nailed or screwed;
- Number of anchors;
- Anchor: anchor type used;
- Anchoring type: type used to fix the anchor;

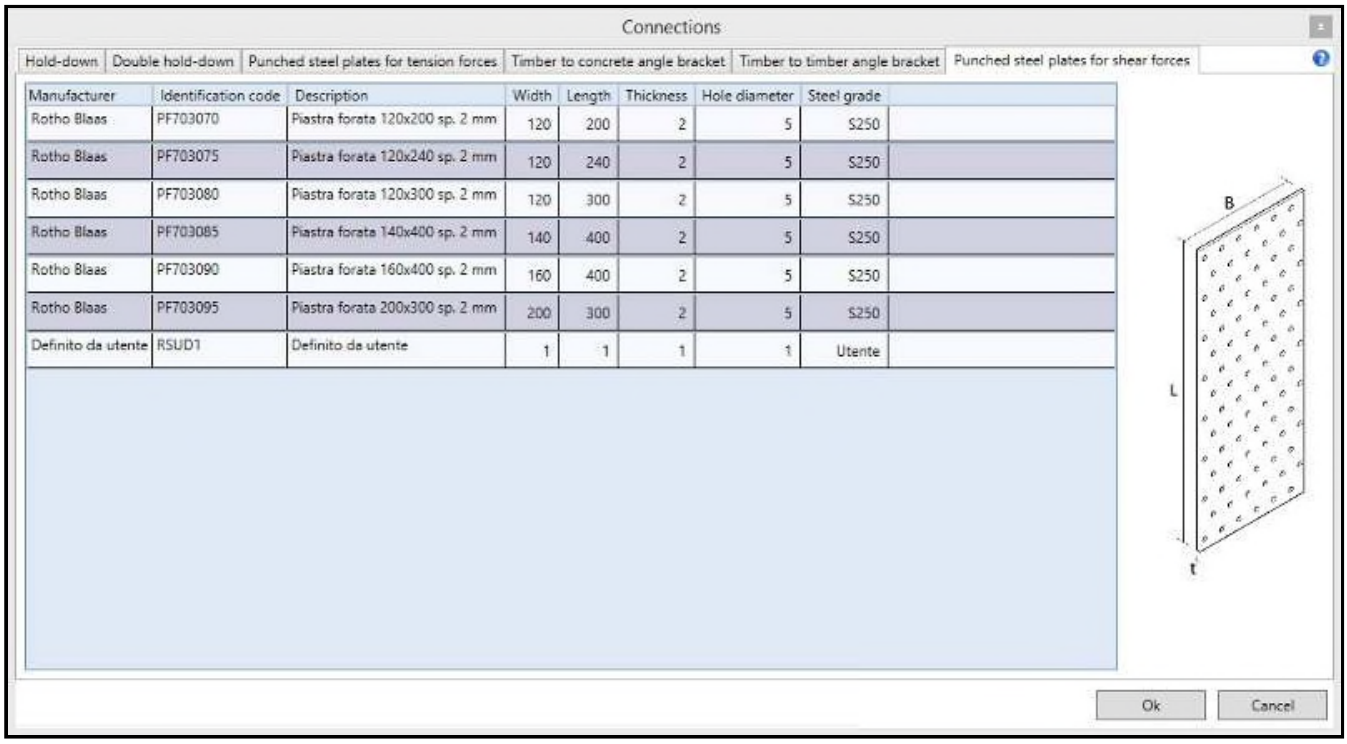
4.1.5.7 Timber to timber angle bracket



The table provides the manufacturer,the identification code, the description and the following implementing informations:

- Nailing: partial or total;
- Number of fasteners in the vertical plate;
- Connector type: nailed or screwed according to the anchoring system sizes;
- Number of fasteners in the horizontal plate;

4.1.5.8 Timber to timber punched steel plates for shear forces

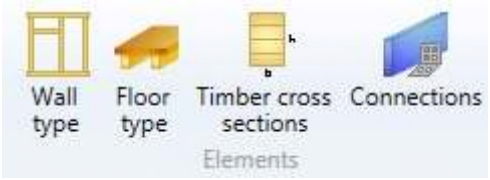


Manufacturer	Identification code	Description	Width	Length	Thickness	Hole diameter	Steel grade
Rotho Blaas	PF703070	Piastra forata 120x200 sp. 2 mm	120	200	2	5	S250
Rotho Blaas	PF703075	Piastra forata 120x240 sp. 2 mm	120	240	2	5	S250
Rotho Blaas	PF703080	Piastra forata 120x300 sp. 2 mm	120	300	2	5	S250
Rotho Blaas	PF703085	Piastra forata 140x400 sp. 2 mm	140	400	2	5	S250
Rotho Blaas	PF703090	Piastra forata 160x400 sp. 2 mm	160	400	2	5	S250
Rotho Blaas	PF703095	Piastra forata 200x300 sp. 2 mm	200	300	2	5	S250
Definito da utente	RSUD1	Definito da utente	1	1	1	1	Utente

The table provides the manufacturer,the identification code, the description and the mechanical and geometrical proprieties:

- B: width;
- L: length;
- t : thickness;
- Hole diameter;
- Steel grade;

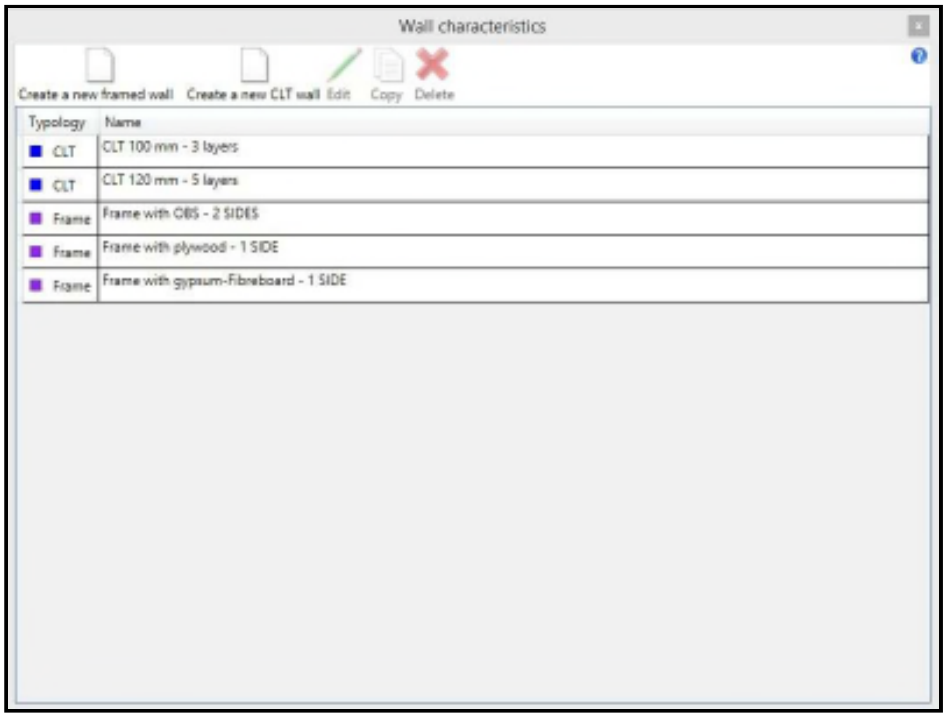
4.2 Elements



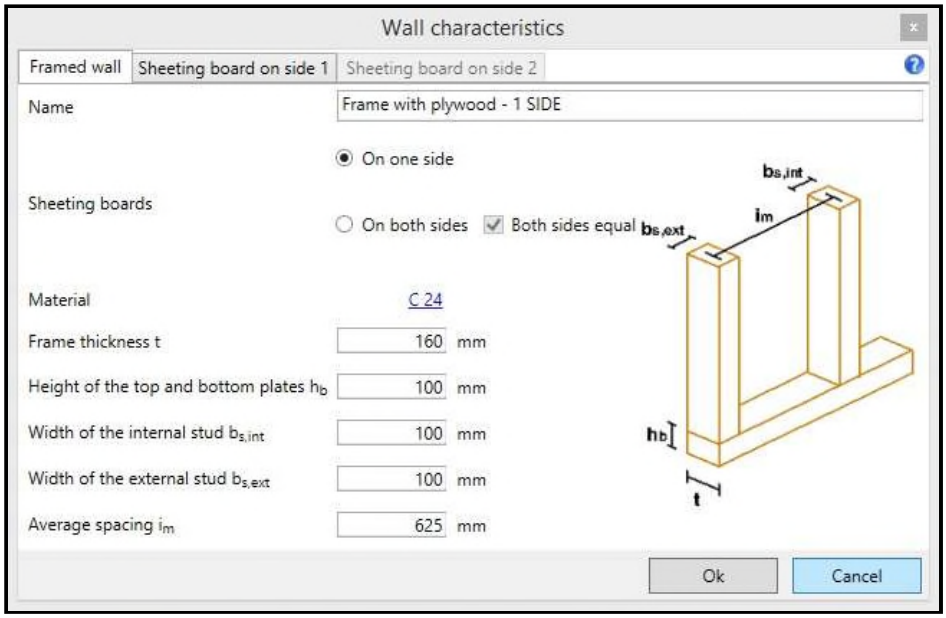
The *Elements* menu allows the user to fully define the geometrical characteristics of the structural elements and the walls connectors (type and number of anchors systems).

4.2.1 Wall type

The command *Wall Type* allows the user to define a wall type and to modify or copy an existing one. The user can select the predefined walls: frame walls or CLT walls.



4.2.1.1 Frame wall



The *Create Frame* command allows the user to define a new wall type, specifying the geometrical and mechanical properties of the frame and of the sheeting boards. In the first window the user can:

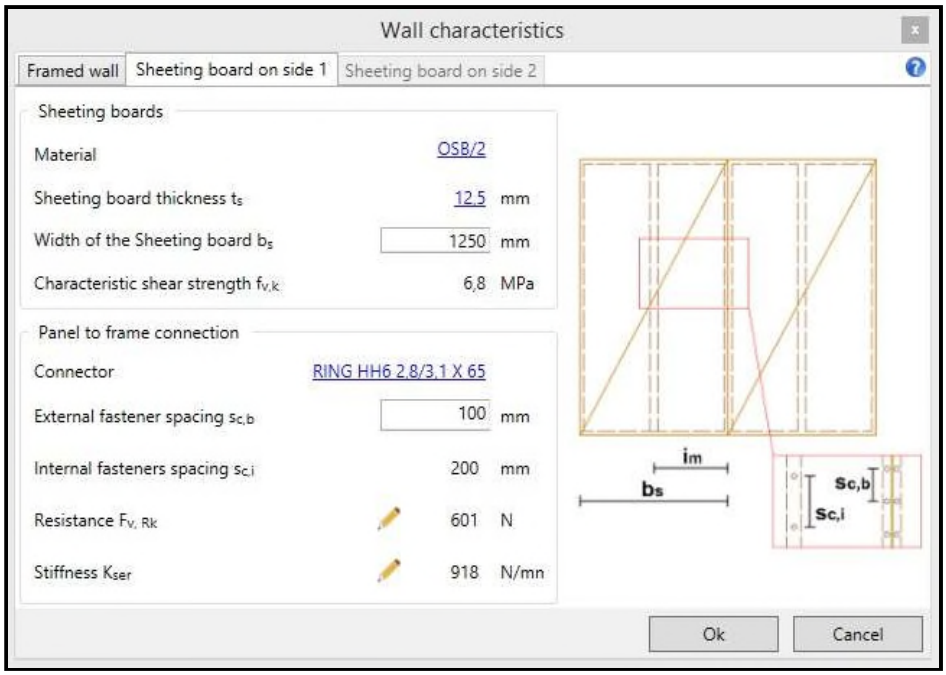
- assign a name to the wall type;
- define the sides number with sheeting boards;
- define the frame material selecting the database data;
- define the geometrical proprieties of the frame elements;

The geometrical proprieties of the frame elements are:

- Frame thickness t;
- Height of the top and bottom frame hb ;
- Width of the internal stud $b_{s,int}$;

- Width of the external stud $b_{s,ext}$;
- Average spacing i_m;

The user can define the geometrical and mechanical characteristics of the sheeting board selecting the windows Sheeting board on side1/side2:



In these windows the user can assign:

- the external sheeting board material selecting OSB, plywood, particleboard or gypsum-fibreboard from the database;
- the connectors type, consulting the database;
- the geometrical characteristics of the boards;
- the external fasteners spacing;

The geometrical characteristics of the boards are:

- Sheeting boards thickness t_s;
- Sheeting boards width b_s;

The internal fasteners spacing $s_{c,i}$ is deduced from the external one $s_{c,b}$ according to UNI EN 1995-1-1: 2005, point 10.8.2: $s_{c,i} = \min(2s_{c,b}$; 300 mm).

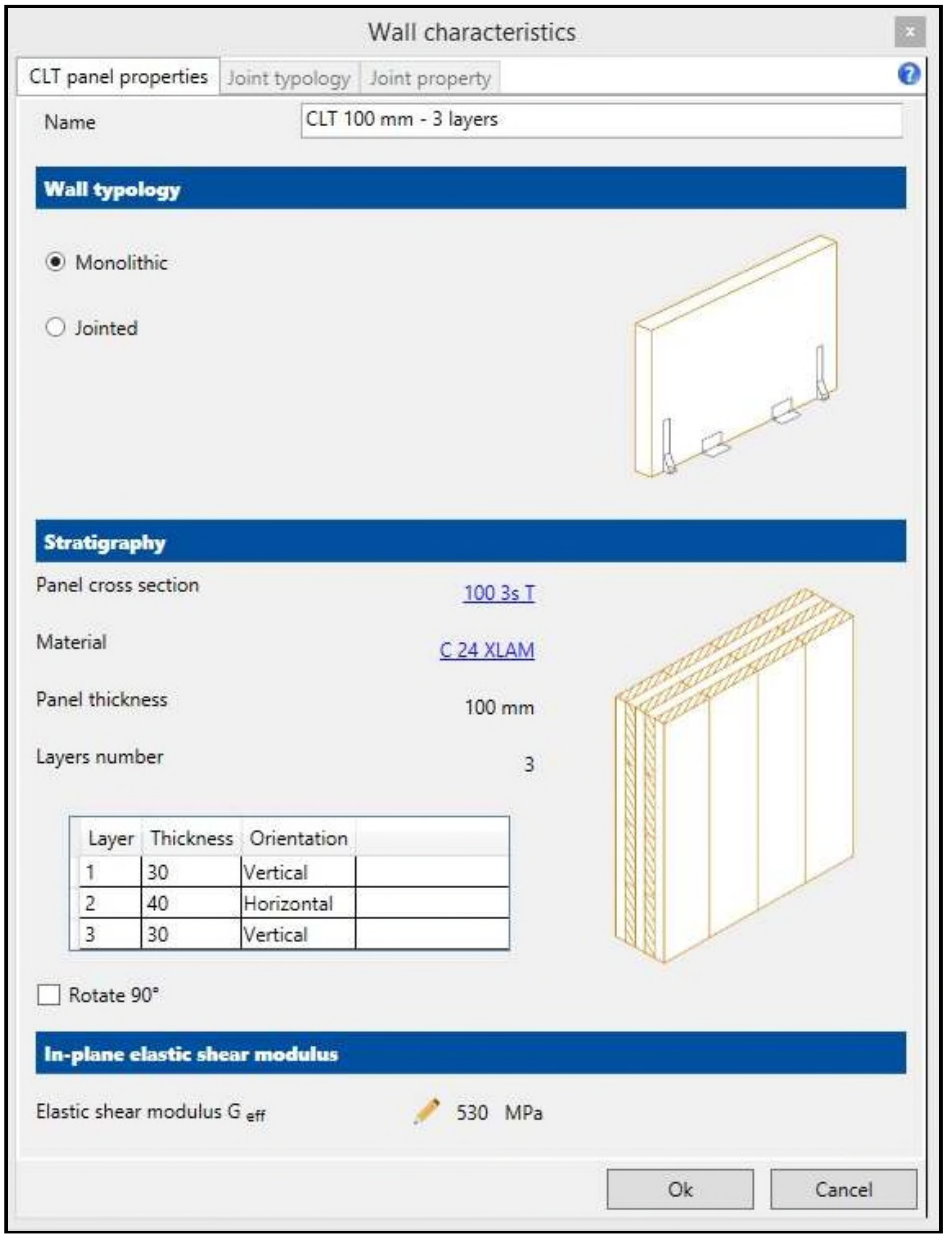
The total strength of the board-frame connection $F_{v,Rk}$, shown in the window, is deduced according to UNI EN 1995-1-1: 2005, point 8.2.2.

The fasteners stiffness K_{ser} , shown in the window, is calculated according to UNI EN 1995-1-1: 2005, point 7.1.

Note:

Left-clicking the mouse on this icon, the user can modify the strength and the stiffness of the wall.

4.2.1.2 CLT



The command *Create CLT wall* allows the user to create a new wall type, defining the geometrical and mechanical characteristics of the CLT panels. In the first window the user has to:

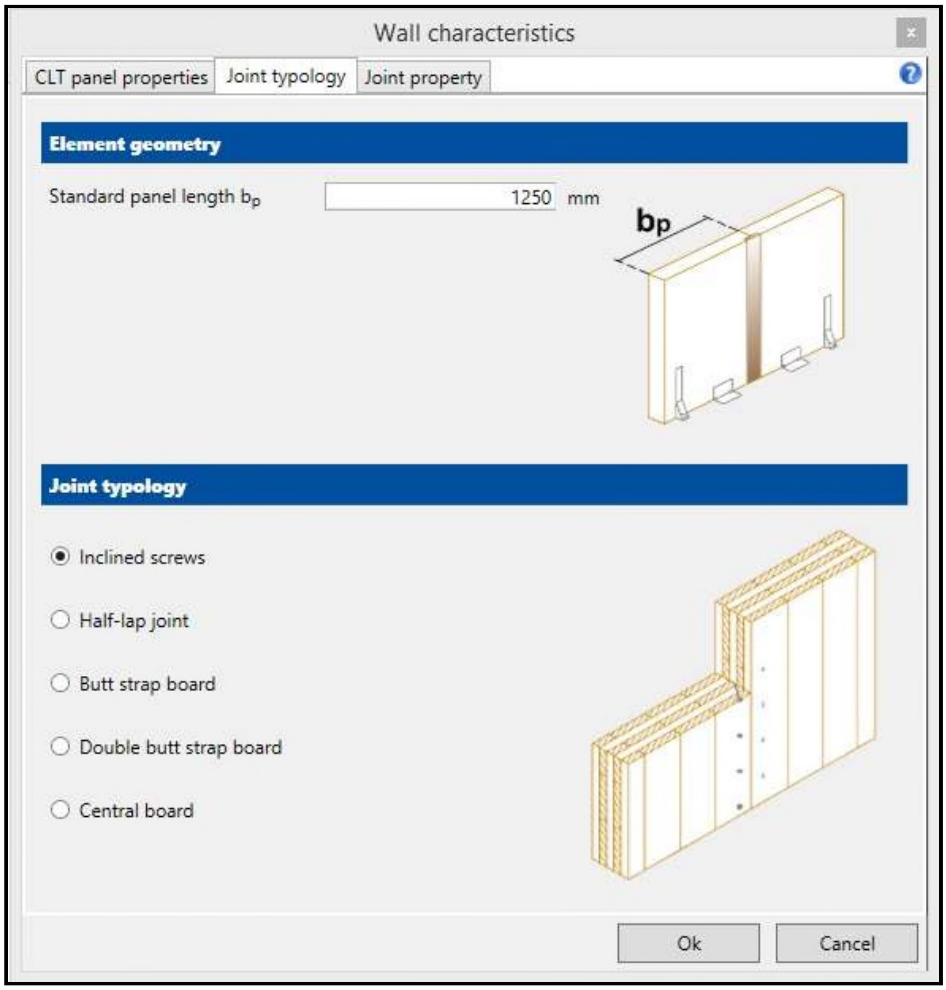
- select, by the checkbox, the monolithic or jointed wall;
- assign the name to the new wall;
- select the panel cross section and the panel material from the Database.

The user can rotate by 90 degrees the orientation of the panel layers.

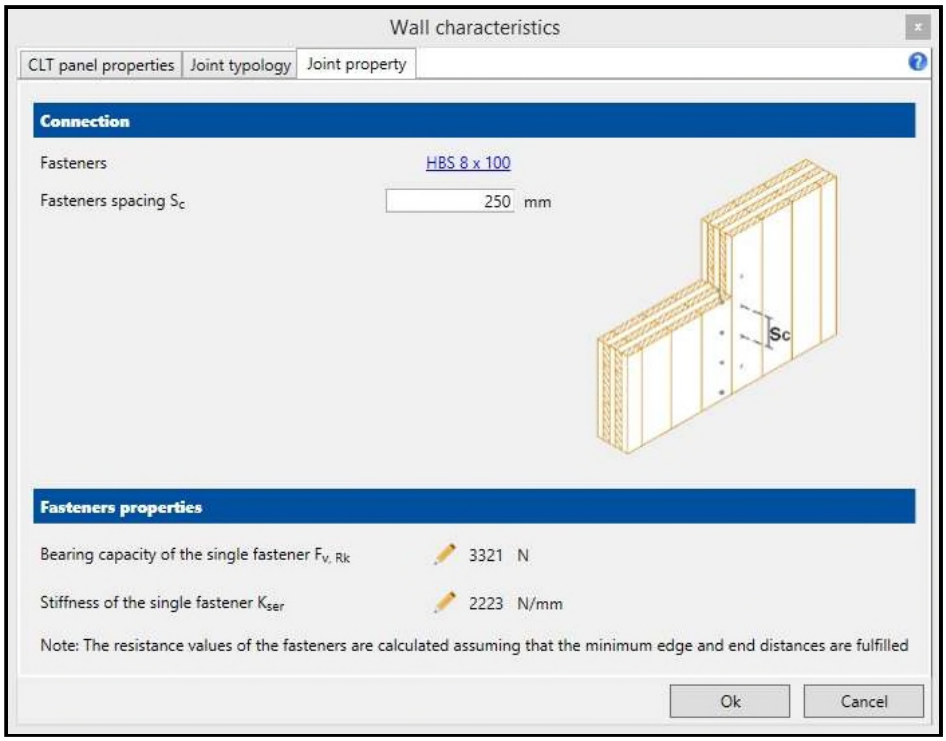
According to the selection, the software calculates the shear modulus G_{eff} , with reference to the model proposed in the publication *Verification of CLT-plates under loads in plane - Bogensperger T. Moosbrugger T. e Silly*". The user can modify the value on the basis experimental results or more accurate models.

If the user selects the jointed wall, the definition of the joints type and proprieties will be done in the following windows:

Joint typology



Joint property



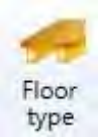
The user can define:

- The joint typology consulting the Database;
- The fasteners spacing S_c;

The bearing capacity of the single fastener $F_{v,Rk}$, provided in the dialog box, is calculated according to the UNI EN 1995-1-1: 2005, point 8.2.2.

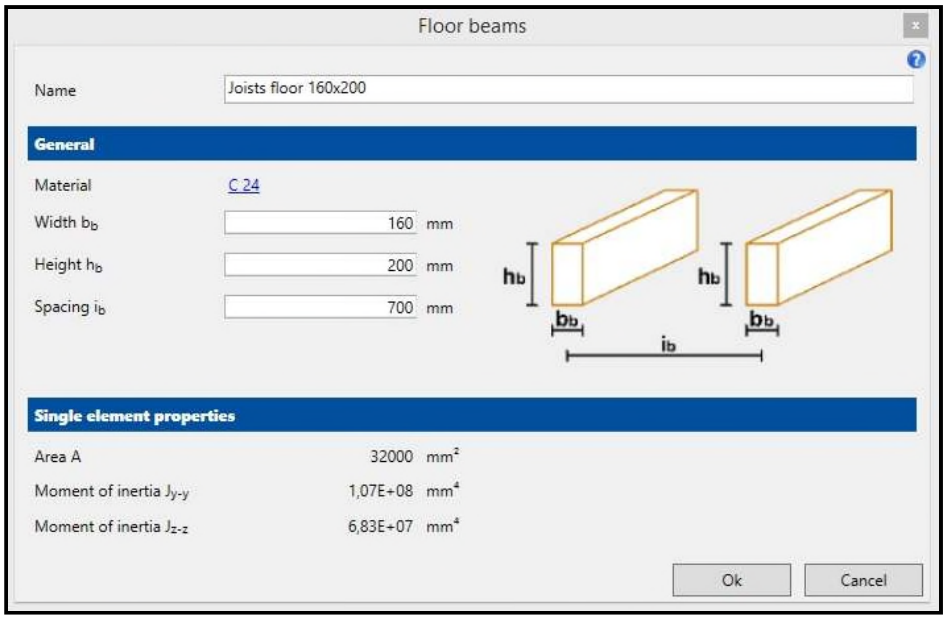
The stiffness of the single fastener K_{ser}, provided in the dialog box, is calculated according to the UNI EN 1995-1-1: 2005, point 7.1.

4.2.2 Floor type



The command *Floor type* provides the definition of the floor type and to modify or copy the existing ones. The user can even select a predefined floor type: joist floor, solid wood floor and CLT floor.

4.2.2.1 Joists floor



The *Create Joists Floor* command is used to create a new floor type and to define the beams characteristics. In the dialog box the user can:

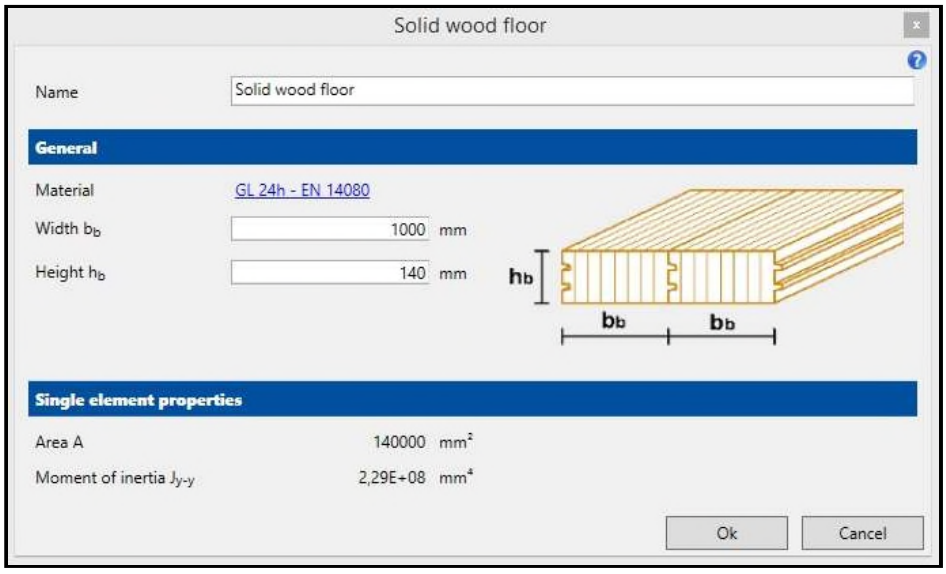
- assign a name to the new floor;
- define the beam material from the internal Database;
- define the beams geometrical proprieties;

The geometrical proprieties of the beams are:

- Beam width b_b;
- Beam height h_b;
- Spacing of the beams i_b;

The assumed value of the inertial proprieties are shown in the dialog box.

4.2.2.2 Solid wood floor



The command *Create a Solid wood floor* allows the user to define a new Solid wood floor type, and to define the geometrical proprieties of the single element.The user has to define, in the dialog box:

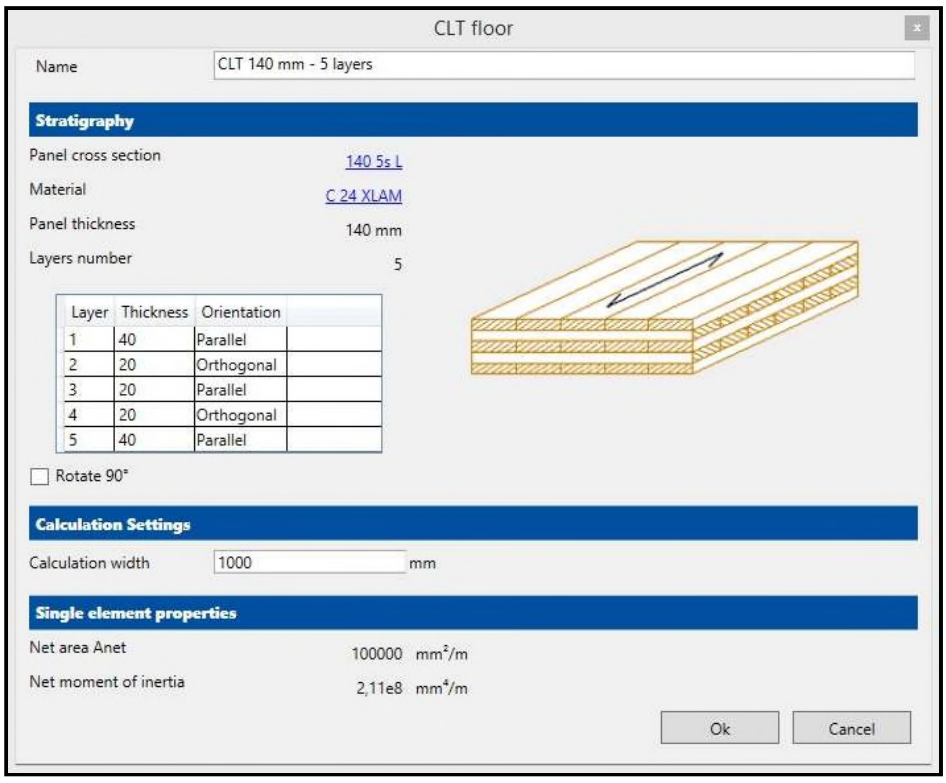
- assign a name to the new floor;
- define the panels material from the internal Database;
- define the geometrical proprieties of the glulam panels;

The user has to define the following geometrical proprieties:

- Panel width b_b;
- Section height of the panel h_b;

The area value and inertial proprieties are provided in the dialog box.

4.2.2.3 CLT floor



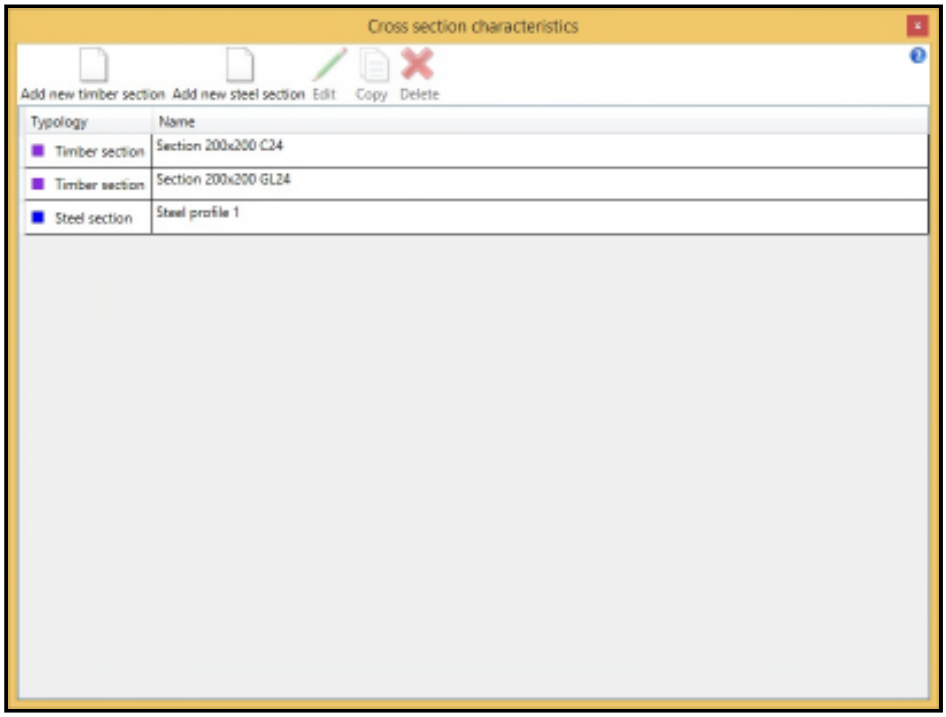
The command *Create a CLT floor* allows the user to define a new CLT floor type, and to define the geometrical proprieties of CLT panels.The user has to define, in the dialog box:

- assign a name to the new floor;
- define the panels stratigraphy from the internal Database;
- define the material proprieties;
- define the orientation of the external layers: parallel or orthogonal to the calculation direction;
- define the calculation width: the width used to verify the panels that is the spacing used to solve the static model;

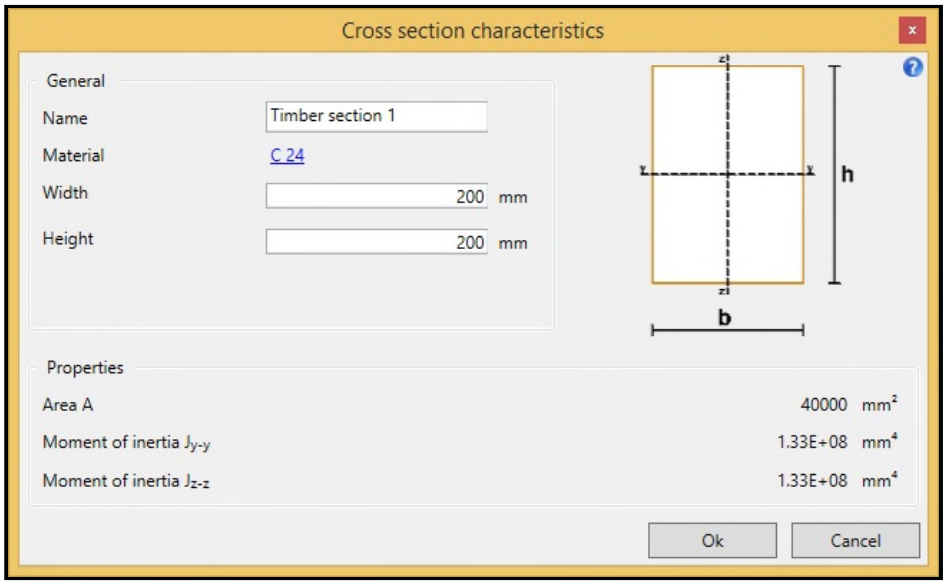
The dialog box provides the inertial proprieties: the net area and the net moment of inertia.

4.2.3 Timber cross section

The command *Cross section* allows the user to define a beam cross section and to modify or copy an existing one. The user can select a predefined cross section.



If the user clicks the *Add new timber section* command, he will define a new section:

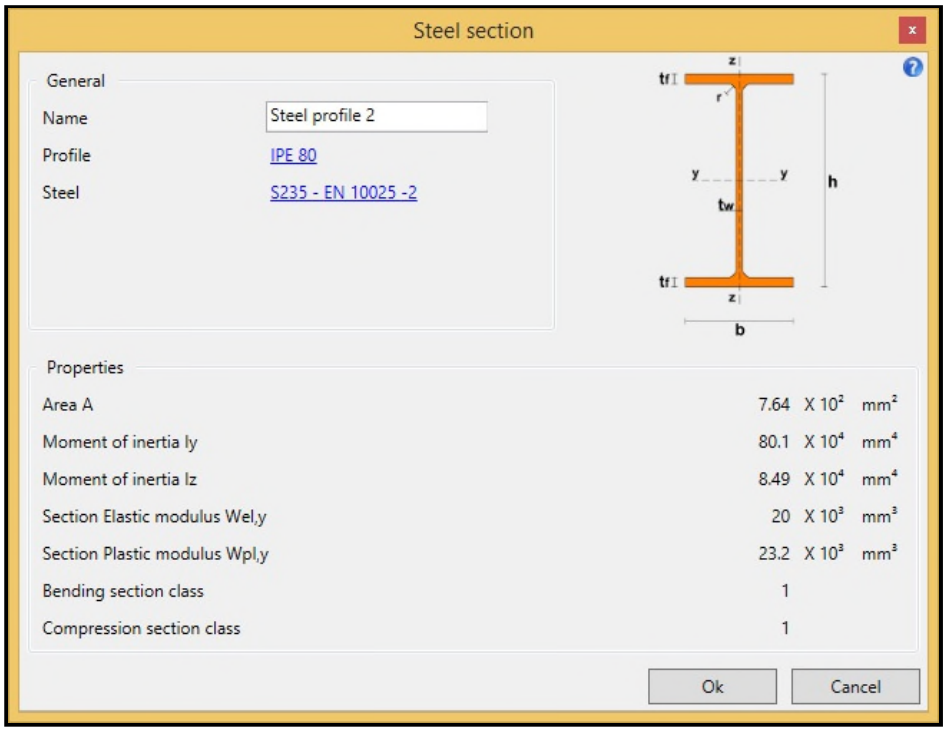


In the dialog box the user has to:

- assign the name to the new section;
- assign the section material, consulting the internal Database;
- assign the geometrical proprieties of the section

The dialog box provides the inertial proprieties: area and inertial moments about the main axes.

If the user clicks the *Add new steel section* command, he will define a new section:



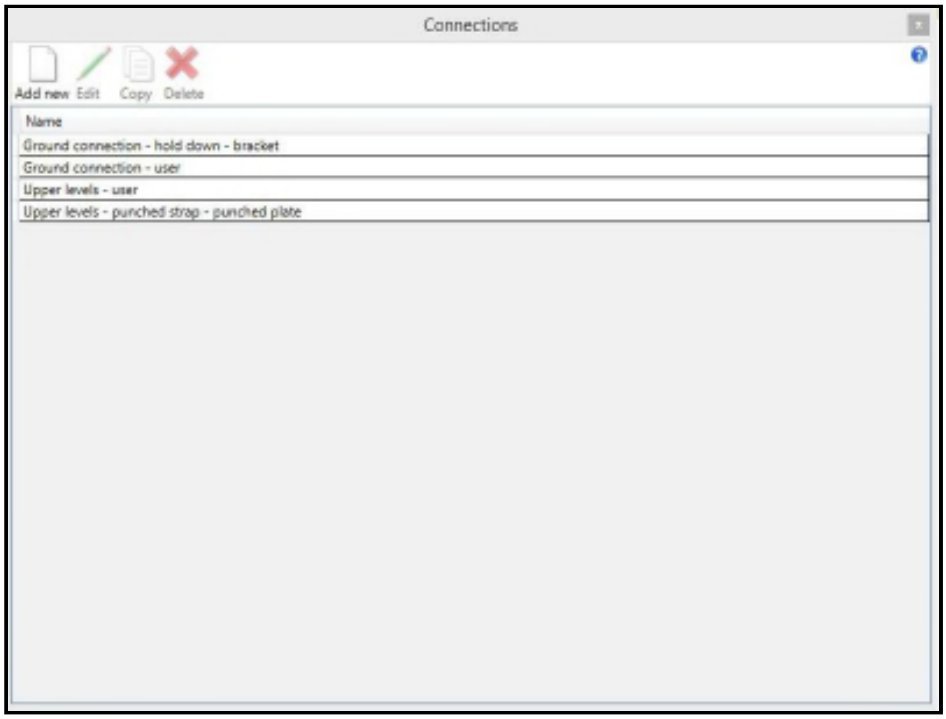
In the dialog box the user has to:

- assign the name to the new section;
- assign the steel profile, consulting the internal Database;
- assign the steel type

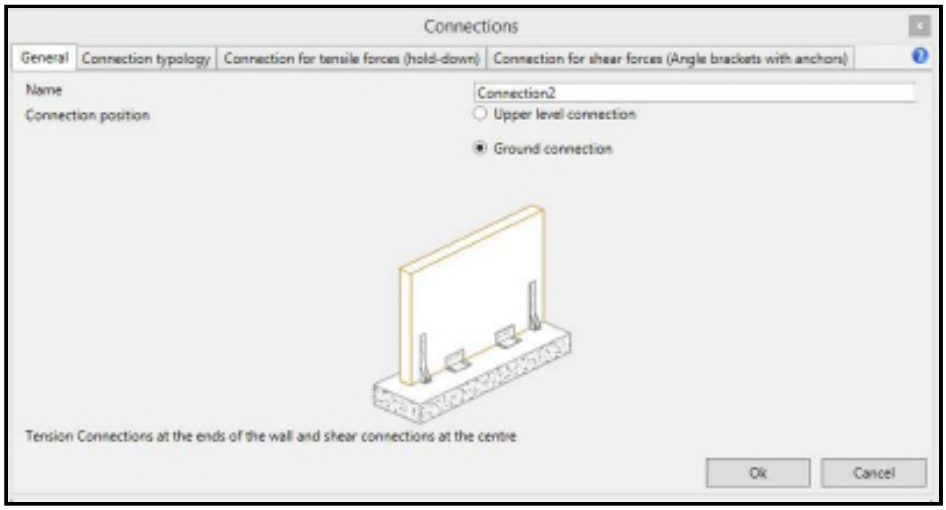
The dialog box provides the inertial proprieties: area, inertial moments, elastic and plastic modulus about the main axes.

4.2.4 Connections

The command is used to define a new connection and to modify or to copy an existing one. The user can also choose a predefined connection.



Selecting the *Add new* command the user can "create" a new connection:



In the first dialog box the user can:

- assign the name;
- assign, by the checkbox, the connection position (ground connection, upper level connection);

According to the connection position, the next dialog box allows the user to define the connection typology (for tension and shear forces).

Note: calculation of the bearing capacity of the nailing - values from certificate

For connections in whose certificate is provided the characteristic value of the nailing resistance (hold down, double hold down, timber to concrete tensile plate, timber to concrete angle bracket, timber to timber angle bracket, timber to concrete shear plate) the following applies.

R_k indicates the characteristic value of the bearing capacity of the nailing assumiong that the minimum edge and end distances are fulfilled and with reference to a characteristic density of the timber equal to 350 kg/m^3. For a density of the used material lower than 350 kg/m^3 in the calculation phase the resistance of the nailing will be corrected using the following eqaution

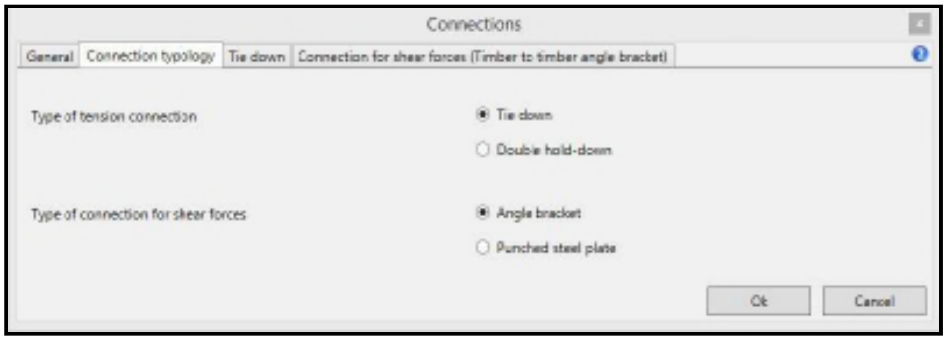
$R_{k,dens} = R_k \cdot (\rho_k / 350)^2$. In the calculation the resistance of the nailing will be evaluated using the actual density of the wood used.

Note: calculation of the resistance of the nailing - calculated values (Johansen theory)

For connections in which the value of the bearing capacity of the nailing is calculated using the theory of Johansen (timber to timber tensile plates and timber to timber shear plates) the following applies.

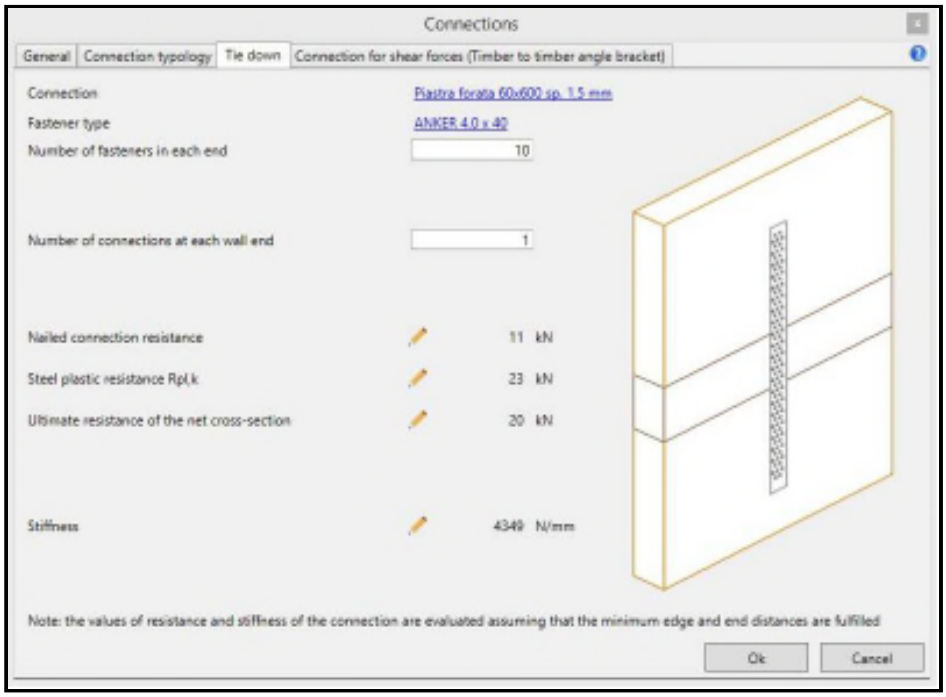
R_k indicates the characteristic value of the bearing capacity of the nailing assuming that the minimum edge and end distances are fulfilled and with reference to a characteristic density of the timber equal to 350 kg/m^3. The characteristic load-carrying capacity of a steel-to-timber connection depends on the thickness of the steel plates: for steel plates of thickness more than or equal to t_{thick} the calculation is done in the hypothesis of thick plate, whereas in the case of plates with thicknesses less than t_{thick} the calculation is done by linear interpolation between the limiting thin (plates with a thickness equal to t_{min}) and thick plate values.

4.2.4.1 Upper level connection



The next dialog boxes allow the user to fully define the connectors used for the chosen connection.

Tension connection ***Tie down***:



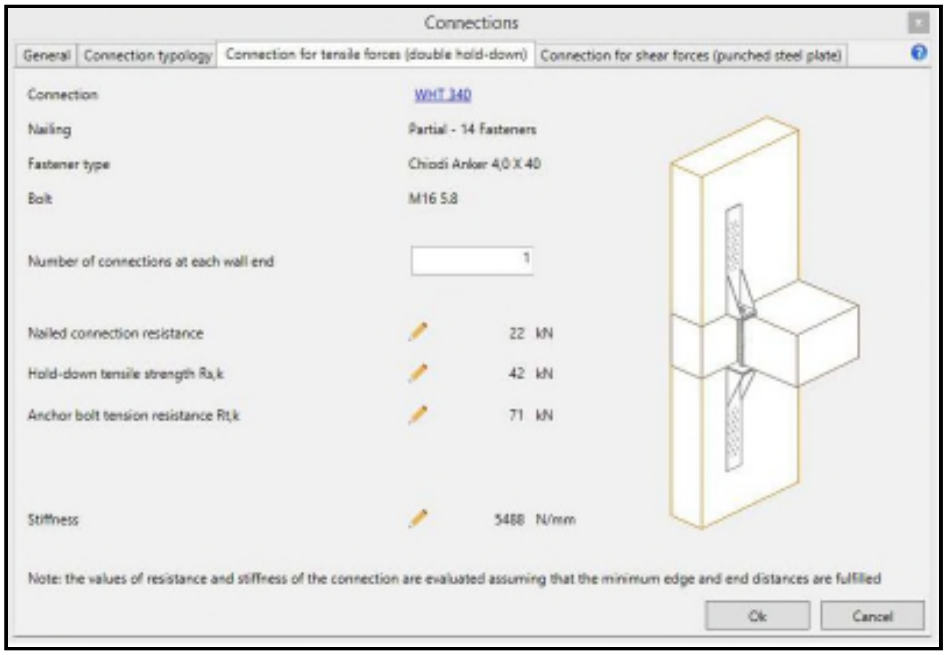
The user can define:

- the connector type;
- the fasteners type;
- the number of fasteners in each end
- the number of connections at each wall end

The dialog box provides:

- the nailed connection resistance $R_{c,k}$;
- the steel plastic resistance $R_{c,pl;}$
- the ultimate resistance of the net cross-section $R_{u,k;}$
- the stiffness value.

Tension connection ***Double Hold-down***:



The user can define in the dialog box:

- the connector type;
- the eccentricity coefficient;
- the number of connections at each wall end;

The dialog provides:

- the fasteners type;
- the bolt type;
- the nailed connection resistance $R_{c,k}$;
- the Hold-down tensile strength $R_{s,k}$;
- the anchor bolt tension resistance $R_{t,k}$;
- the stiffness.

Note: eccentricity coefficient

The tension force acting on the bolt is calculated taking into account the additional moment due to the non-alignment between the external force acting on the vertical flange of the hold down and the bolt itself using a coefficient indicated as k_t.

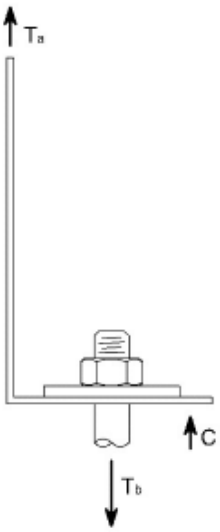
$T_b = T_a \cdot k_t$

where

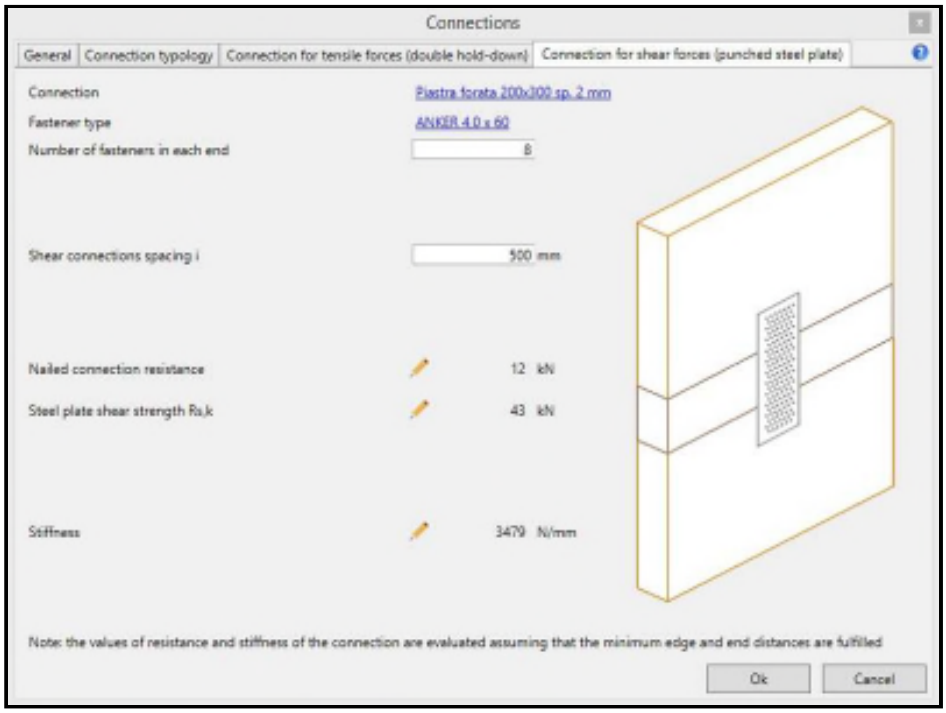
- T_b: is the tension force acting on the bolt increased by the effect of the eccentricity between the flange and bolt

- T_a: is the tension force acting on the hold down

- k_t: is the eccentricity coefficient



Connection for shear forces *Punched steel plate*:



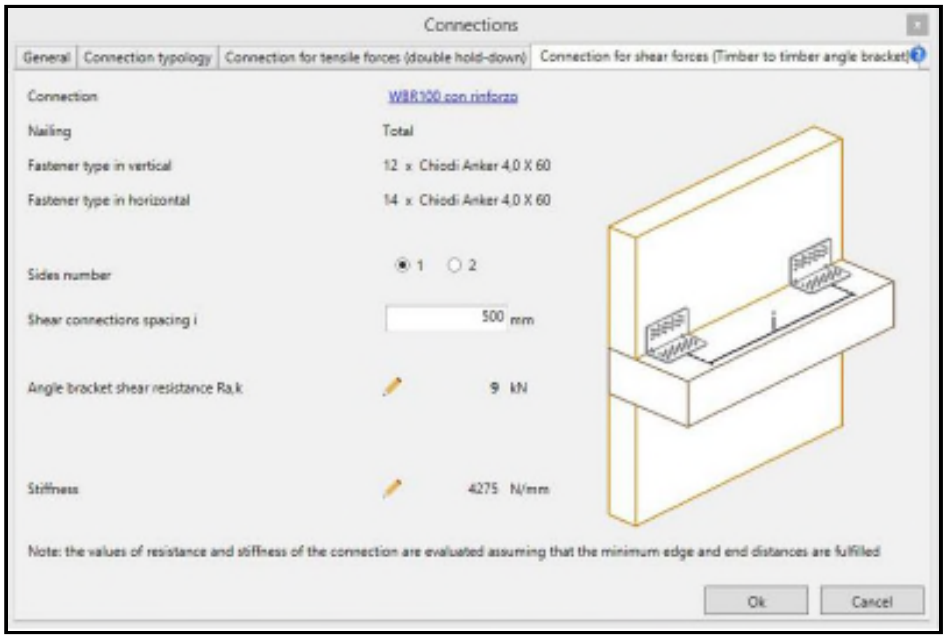
In the dialog box the user can define:

- the connector type;
- the fasteners type;
- the number of fasteners in each end;
- the shear connections spacing;

The dialog box provides:

- the nailed connection resistance $R_{c,k}$;
- the steel plate shear strength $R_{c,pl;}$
- the stiffness.

Connection for shear forces *Angle bracket*:



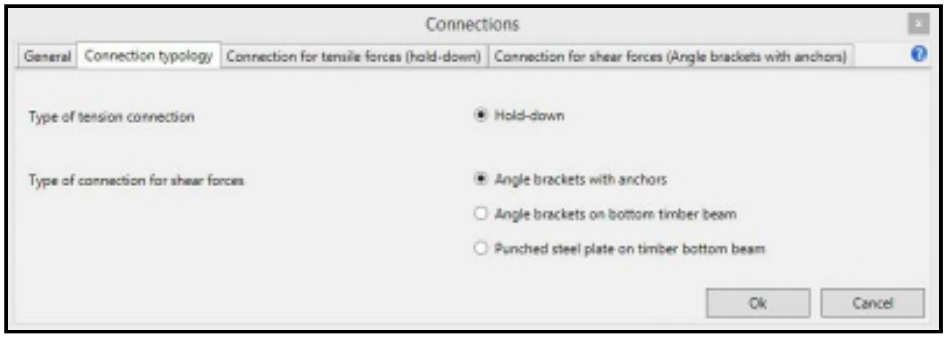
In the dialog box the user can define:

- the connector type;
- the sides number (1/2) on which to apply the connection;
- shear connections spacing *i*;

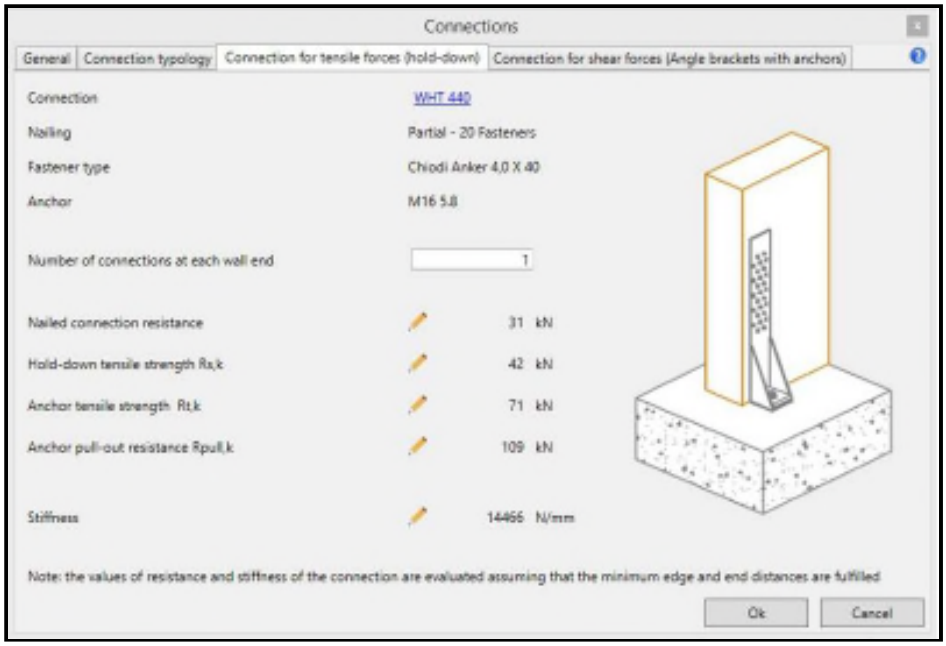
The dialog box provides:

- the fastener type;
- the faster type in vertical;
- the fastener type in horizontal;
- the angle bracket shear resistance $R_{a,k}$;
- the stiffness.

4.2.4.2 Ground connection



Connection for tensile forces ***Hold-Down*:**



In the dialog box the user can define:

- the connector type;
- the eccentricity coefficient;
- the number of connections at each wall end;

The box provides:

- the nailing type;
- the fastener type;
- the anchor type;
- the nailed connection resistance $R_{c,k}$;
- the hold-down tensile strength $R_{s,k}$;
- the anchor tensile strength $R_{t,k}$;
- the stiffness.

Note: eccentricity coefficient

The tension force acting on the anchor is calculated taking into account the additional moment due to the non-alignment between the external force acting on the vertical flange of the hold down and the anchor itself using a coefficient indicated as k_t.

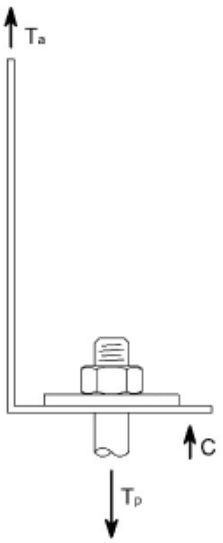
$T_p = T_a \cdot k_t$

where

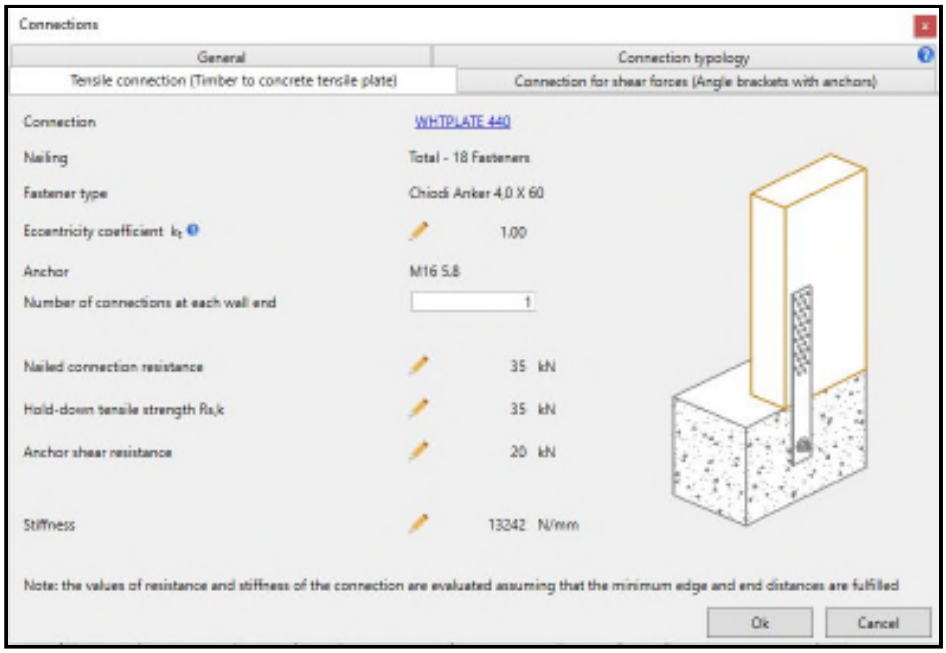
- T_p: is the tension force acting on the anchor increased by the effect of the eccentricity between the flange and the anchor

- T_a: is the tension force acting on the hold down

- k_t: is the eccentricity coefficient



Connection for tensile forces ***Timber to concrete tensile plate***:



In the dialog box the user can define:

- the connector type;
- the eccentricity coefficient k_t;
- the number of connections at each wall end;

The box provides:

- the nailing type;
- the fastener type;
- the anchor type;
- the nailed connection resistance $R_{c,k}$;
- the plate tensile strength $R_{s,k}$;
- the anchor shear strength $R_{v,k}$;
- the stiffness.

Note: eccentricity coefficient

The shear force acting on the anchor is calculated taking into account the redistribution of the total force acting on the plate between all the anchors present. This redistribution is assessed through a partition coefficient, denoted by k_t.

$V_p = T_a \cdot k_t$

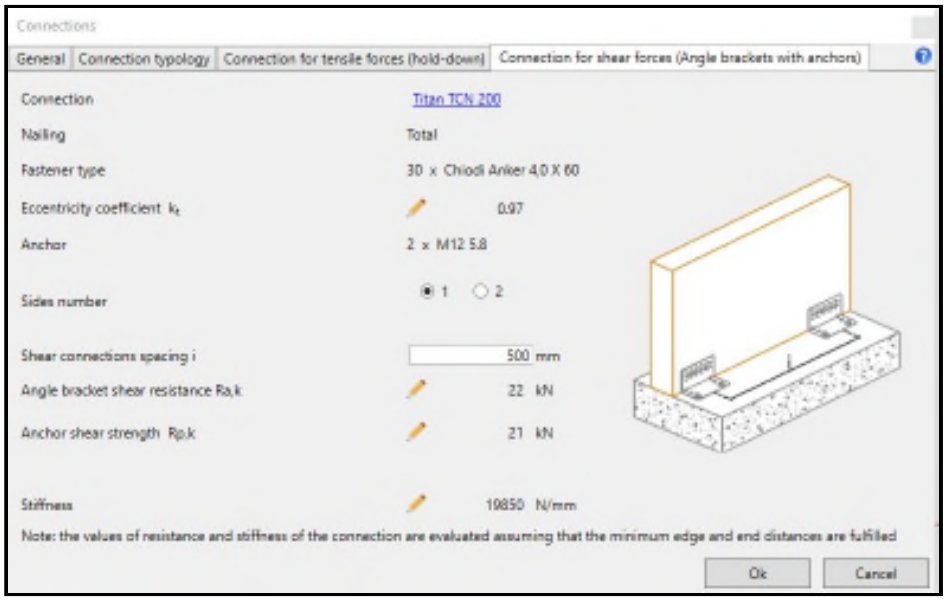
where

- V_p: is the shaer force acting on the anchor

- T_a: is the tension force acting on the metal plate

- k_t: is the eccentricity coefficient

Connection for shear forces ***Angle bracket with anchors***:



In the dialog box the user can define:

- the connector type;
- the eccentricity coefficient;
- the sides number (1/2) on which to apply the connection;
- the shear connections spacing *i*;

The dialog box provides

- the nailing type;
- the fastener type;
- the anchor type;
- the angle bracket shear resistance $R_{a,k}$;
- the anchor shear strength $R_{p,k}$;
- the stiffness.

Note: eccentricity coefficient

The shear force acting on the most loaded anchor is calculated taking into account the additional moment due to the non-alignment between the external forces acting on the vertical flange of the angle bracket and the anchor itself using a coefficient, indicated as k_t.

$V_p = V_a \cdot k_t$

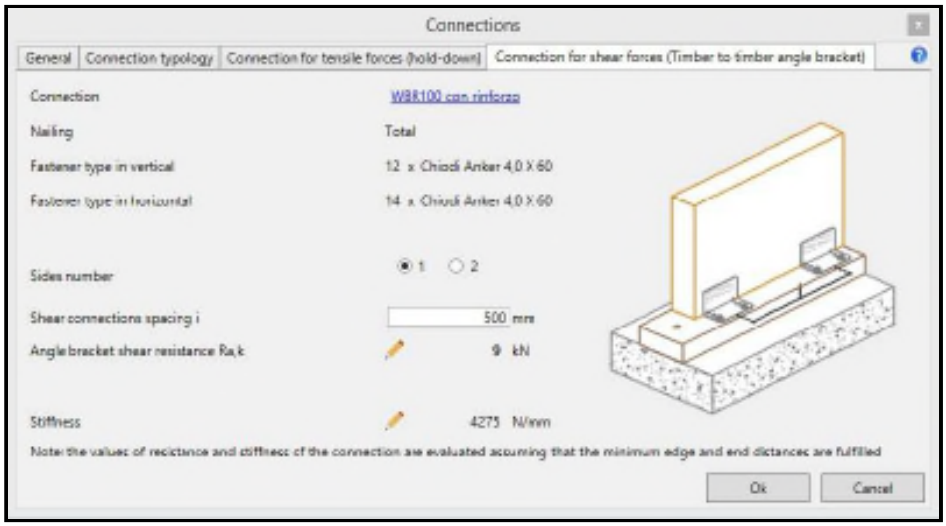
where

- V_p: is the shear force acting on the anchor increased by the effect of the eccentricity between the flange and the anchor

- V_a: is the shear force acting on the angle bracket

- k_t: is the eccentricity coefficient

Connection for shear forces ***Angle bracket on bottom timber beam***:



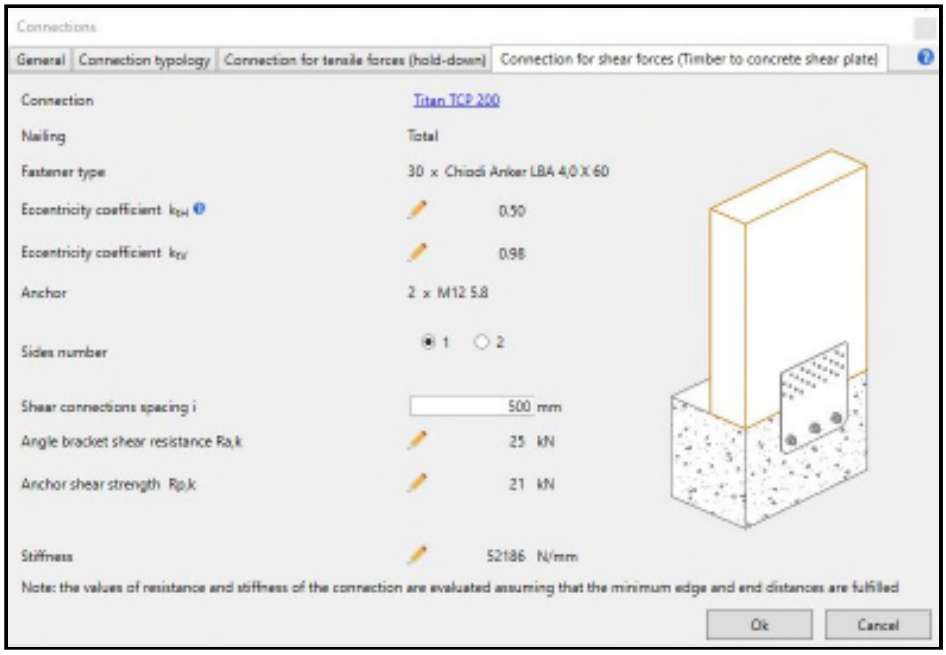
In the dialog box the user can define:

- the connector type;
- the sides number (1/2) on which to apply the connection;
- he shear connections spacing *i*;

The dialog box provides:

- the nailing type;
- the fastener type in vertical;
- the fastener type in horizontal;
- the angle bracket shear resistance $R_{a,k}$;
- the stiffness;

Connection for shear forces***Timber to concrete shear plate***:



In the dialog box the user can define:

- the connector type;
- the eccentricity coefficient k_{tH};

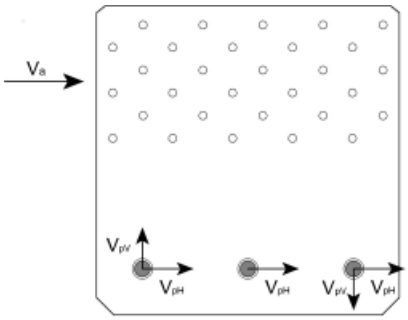
- the eccentricity coefficient k_{tV};
- the anchor type;
- the sides number (1/2) on which to apply the connection;
- the shear connections spacing *i*;

The dialog box provides:

- the angle bracket shear resistance $R_{a,k}$;
- the anchor shear strength $R_{p,k}$;
- the stiffness;

Note: eccentricity coefficient

The shear force acting on the most loaded anchor can be considered composed by two components: a component parallel to the shear force acting on the plate and a component orthogonal to it. The latter one is due to the additional moment associated with the lever arm between the nailing (point of application of external forces) and the row of anchors.



The two components of the shear force acting on the anchor can be evaluated by means of two factorsk_{tH} and k_{tV}

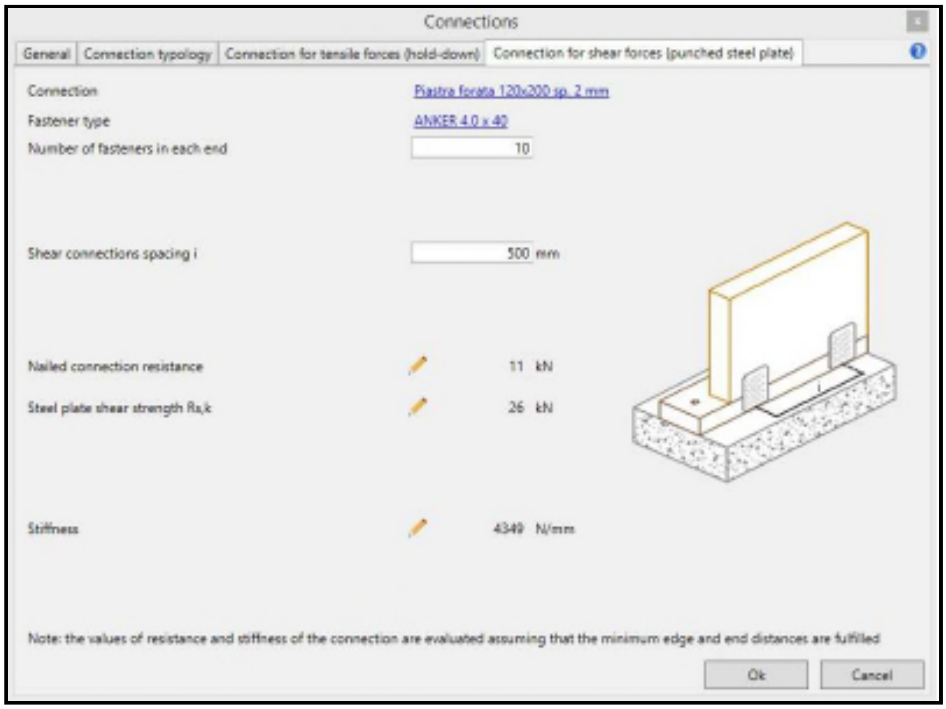
$V_{p,H} = V_a \cdot k_{tH}$

$V_{p,V} = V_a \cdot k_{tv}$

The total shear force acting on the anchor can be calculated with the following expression

$V_p = \left[(V_{p,H})^2 + (V_{p,V})^2 \right]^{1/2}$

Connection for shear forces ***Punched steel plate on timber bottom beam***:



In the dialog box the user can define:

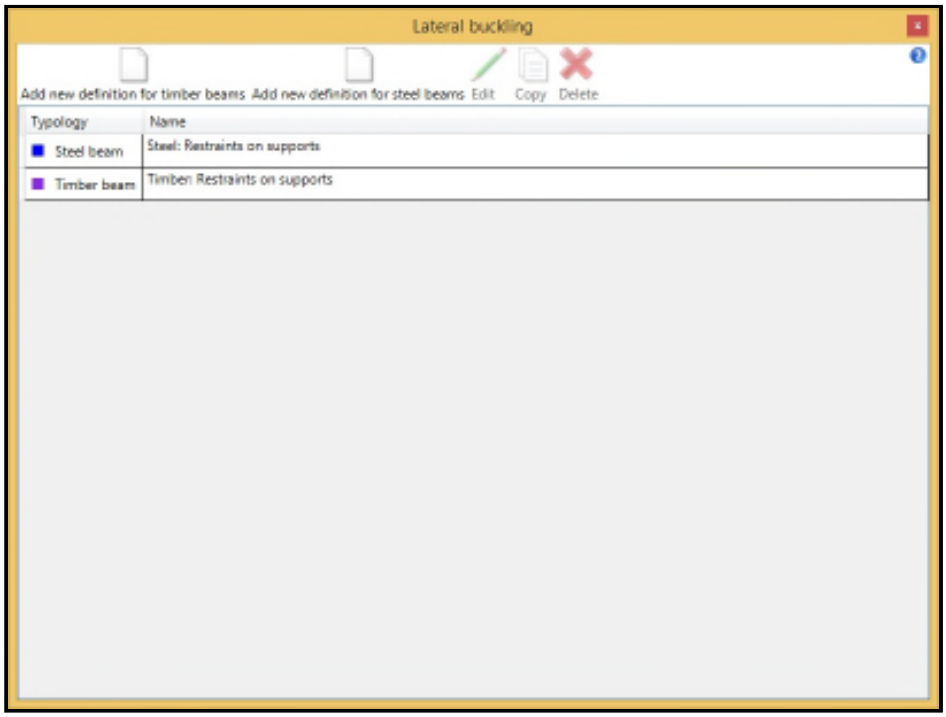
- the connectors type;
- the fasteners type;
- the number of fasteners in each end;
- the shear connections spacing;

The dialog box provides:

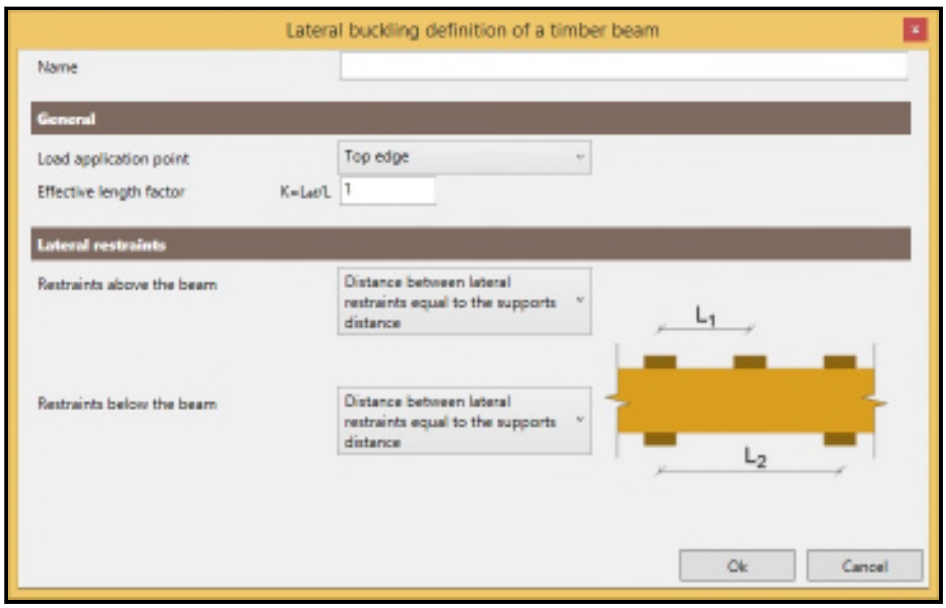
- the nailed connection resistance $R_{c,k}$;
- the steel plate shear strength $R_{s,k;}$
- the stiffness;

4.2.5 Lateral buckling

The command *Lateral buckling* allows the user to define the restraints on supports for a group of beams and to modify or copy an existing one. The user can select a predefined group of restraints on support.



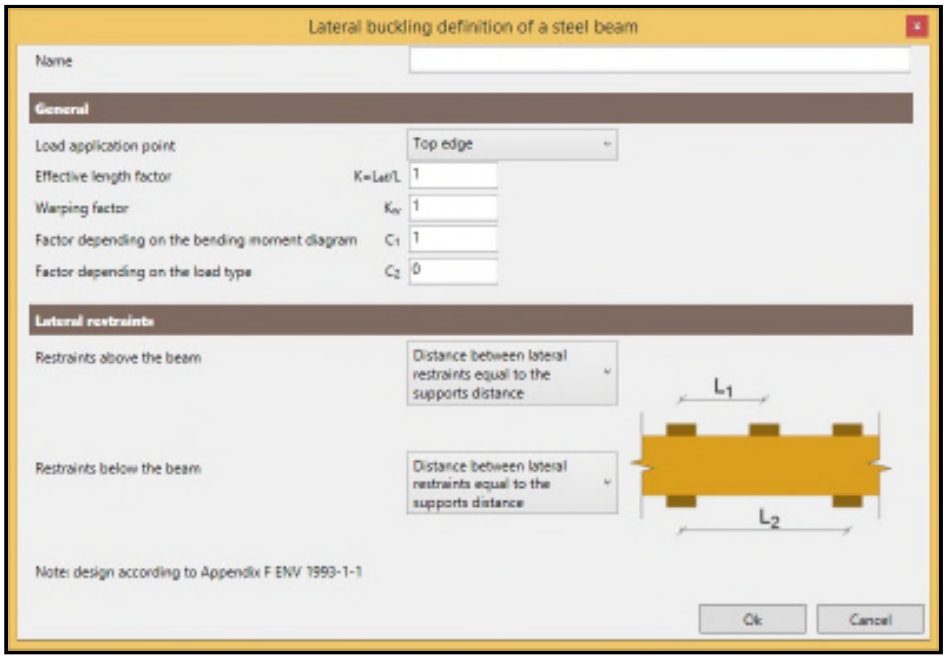
If the user clicks the *Add new definition for timber beams* command, he will define a new group:



In the dialog box the user has to:

- assign the name to the new group;
- assign the load application point;
- assign the effective lenght factor;
- assign the lateral restraints;

If the user clicks the *Add new definition for* steel *beams* command, he will define a new group:



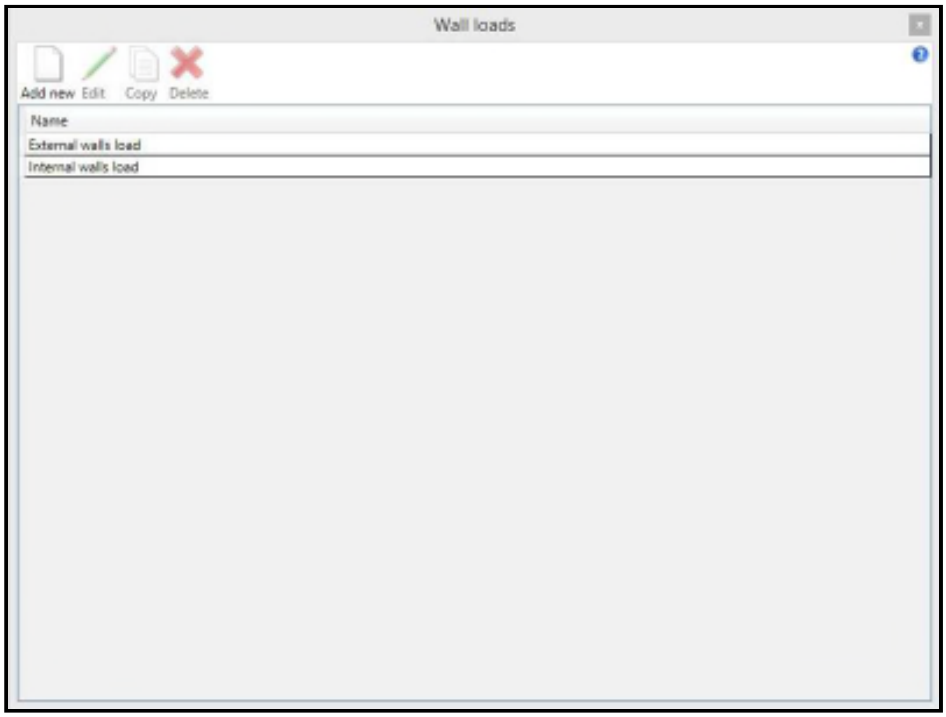
In the dialog box the user has to:

- assign the name to the new group;
- assign the load application point;
- assign the effective lenght factor, the warping factor and factors depending on the bending moment diagram and the load type;
- assign the lateral restraints;

4.2.6 Loads

4.2.6.1 Wall loads

The *Wall loads* command is used to assign loads to the wall and to edit or copy the existing ones (the software distinguishes the loads between internal and external walls):

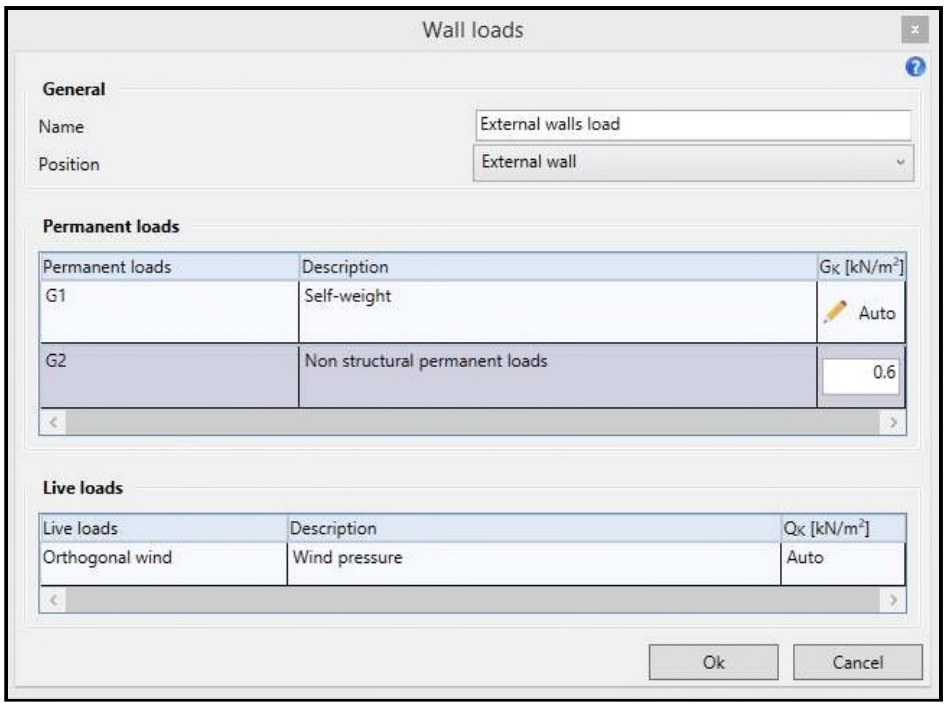


The user can define new wall loads selecting the command *Add new*. In the dialog box the user has to:

- assign the name to the load;
- define the wall position (internal or external);

If the user defines an *External wall* he will:

- modify the self-weight load G_1, calculated automatically by the software, clicking the special icon;
- assign a non structural permanent loads G_2 ;

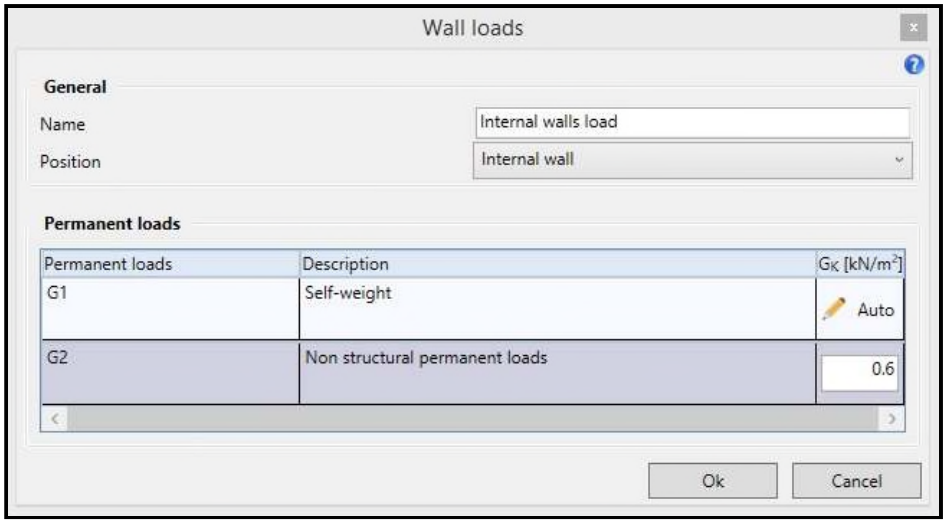


The variable load *Wind pressure* can not be modified, because it is automatically calculated according to the model geolocation;

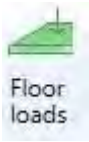
If the user defines an *Internal wall*, he will:

- modify the self-weight load G_1, calculated automatically by the software, clicking the special icon;

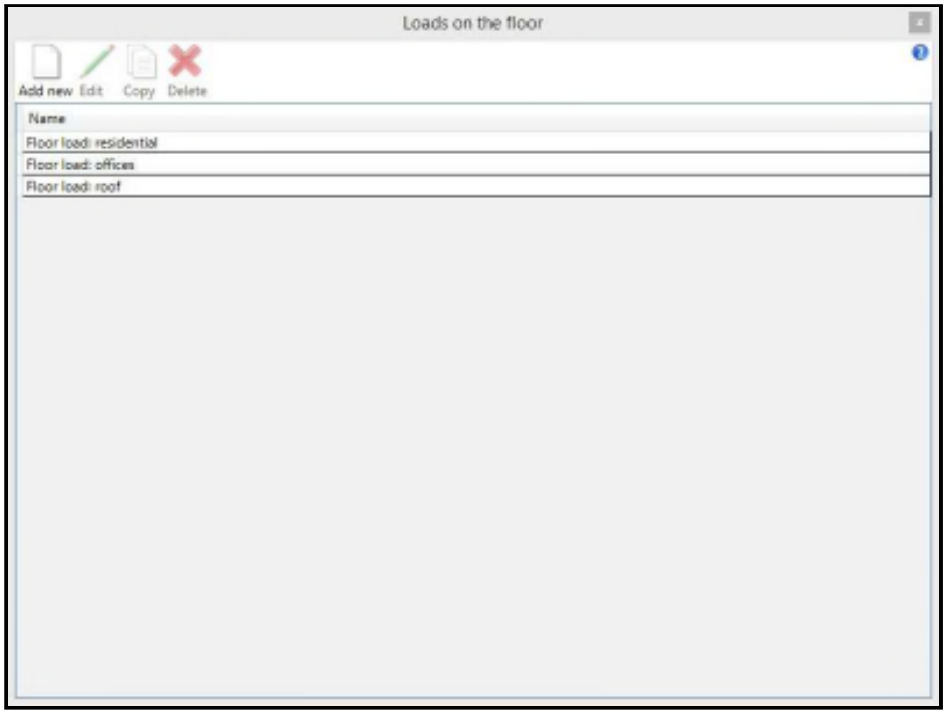
- assign a non structural permanent loads G_2 ;



4.2.6.2 Floor loads;



The command *Floor loads* is used to assign loads to the floors and to edit or copy the existing ones. In the dialog box the user can select the predefined loads depending on the different uses: residential, offices and roof:



If the user select the *Add new* command, he will assign the floor loads. In the dialog box the user has to:

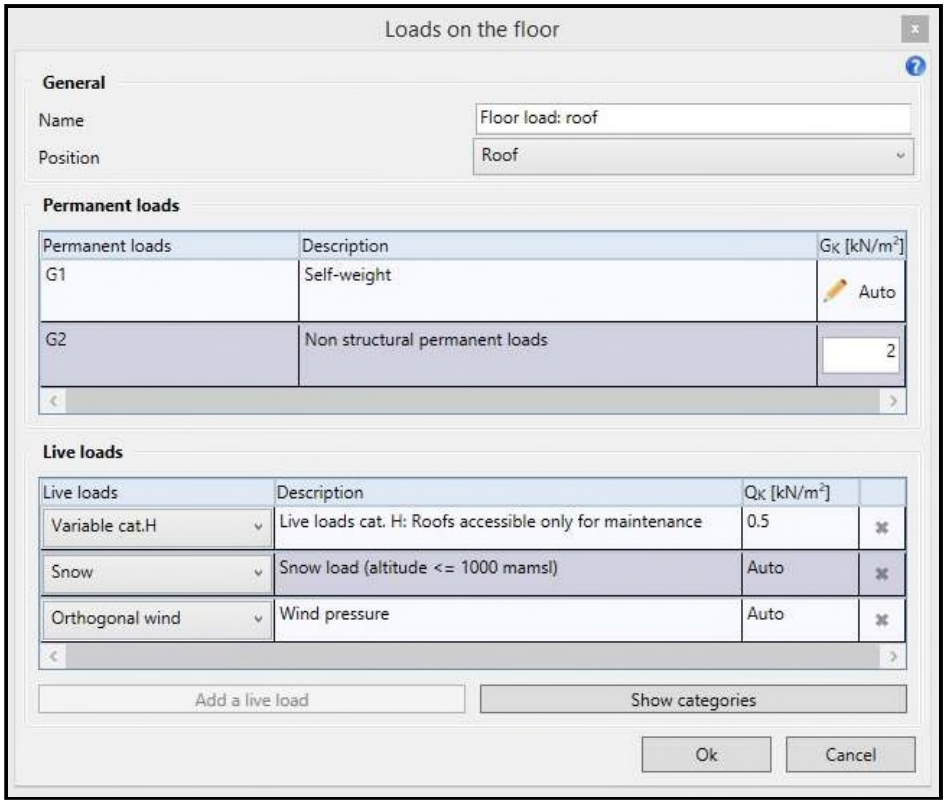
- assign the name to the floor load;
- define the floor position by the drop-down menu (roof or floor);

4.2.6.2.1 Roof

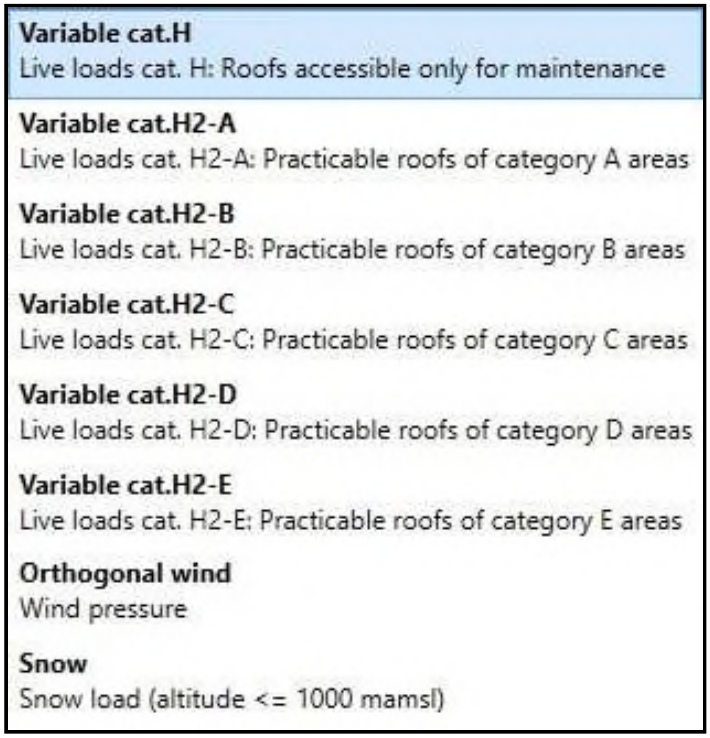
If the user chooses the *Roof* he will:

- modify the self-weight load G_1, calculated automatically by the software, clicking the special icon;

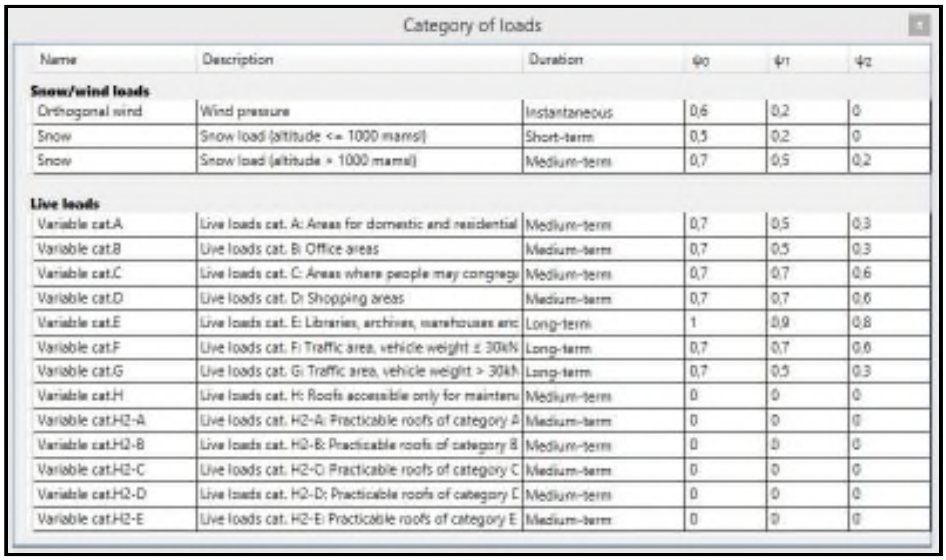
- assign a non structural permanent loads G_2 ;



The user will edit the variable loads by the drop-down menu *Variable,* selecting the load category:

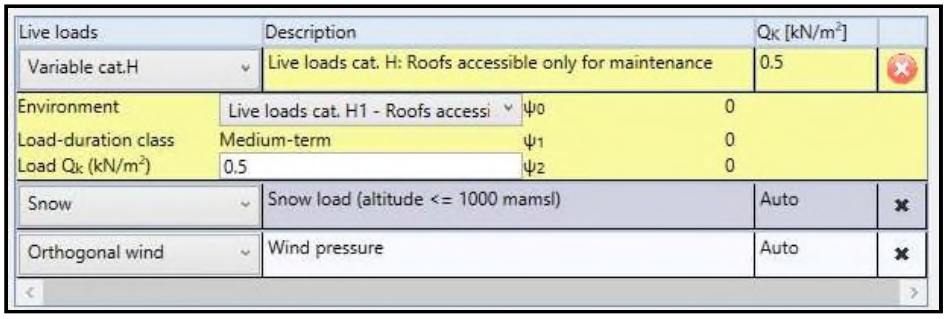


The *Show category* command provides the access to the table which defines the variable loads category, according to the point 3.1.4 of the Italian Technical Code (NTC 2008):



Name	Description	Duration	ψ_0	ψ_1	ψ_2
Snow/wind loads					
Orthogonal wind	Wind pressure	Instantaneous	0,6	0,2	0
Snow	Snow load (altitude <= 1000 mamsl)	Short-term	0,5	0,2	0
Snow	Snow load (altitude > 1000 mamsl)	Medium-term	0,7	0,5	0,2
Live loads					
Variable cat.A	Live loads cat. A: Areas for domestic and residential	Medium-term	0,7	0,5	0,3
Variable cat.B	Live loads cat. B: Office areas	Medium-term	0,7	0,5	0,3
Variable cat.C	Live loads cat. C: Areas where people may congregi	Medium-term	0,7	0,7	0,6
Variable cat.D	Live loads cat. D: Shopping areas	Medium-term	0,7	0,7	0,6
Variable cat.E	Live loads cat. E: Libraries, archives, warehouses anc	Long-term	1	0,9	0,8
Variable cat.F	Live loads cat. F: Traffic area, vehicle weight ≤ 30kN	Long-term	0,7	0,7	0,6
Variable cat.G	Live loads cat. G: Traffic area, vehicle weight > 30kN	Long-term	0,7	0,5	0,3
Variable cat.H	Live loads cat. H: Roofs accessible only for maintenu	Medium-term	0	0	0
Variable cat.H2-A	Live loads cat. H2-A: Practicable roofs of category A	Medium-term	0	0	0
Variable cat.H2-B	Live loads cat. H2-B: Practicable roofs of category B	Medium-term	0	0	0
Variable cat.H2-C	Live loads cat. H2-C: Practicable roofs of category C	Medium-term	0	0	0
Variable cat.H2-D	Live loads cat. H2-D: Practicable roofs of category D	Medium-term	0	0	0
Variable cat.H2-E	Live loads cat. H2-E: Practicable roofs of category E	Medium-term	0	0	0

The user can delete selected loads by clicking the special icon on the right side of the *Vertical loads* window:



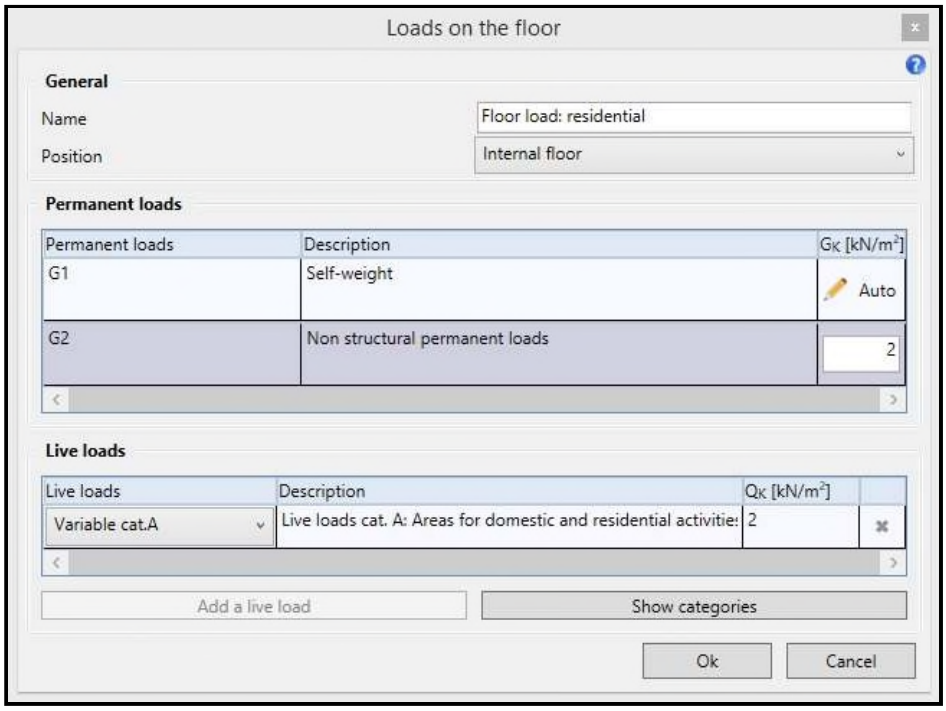
The user can add a new variable loads selecting the *Add a live load* command.

The wind load and the snow load can not be modified by the user because they are automatically calculated according to the model geolocation.

4.2.6.2.2 Internal floor

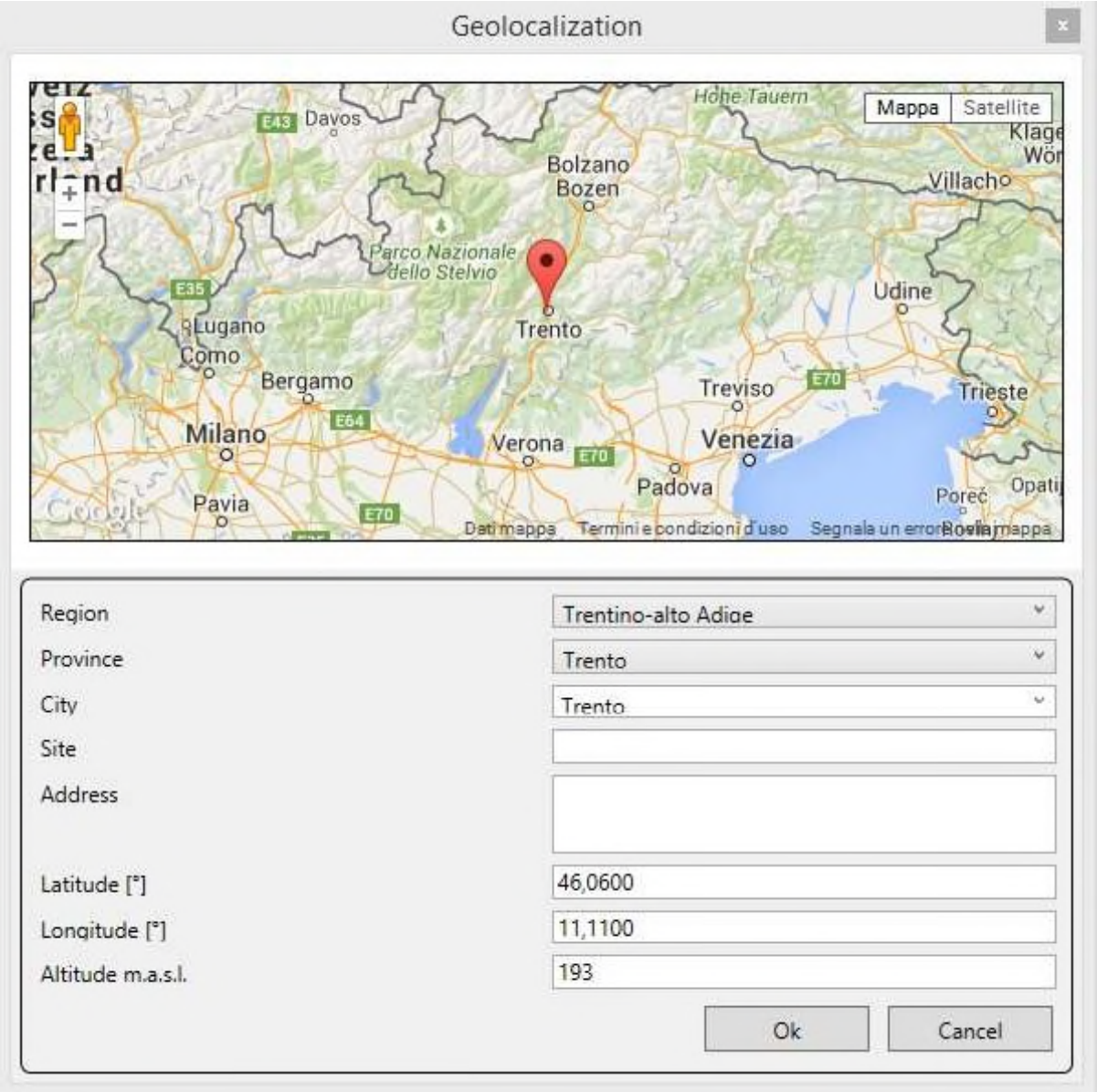
If the user defines the *Internal floor* he will:

- modify the self-weight load G_1, calculated automatically by the software, clicking the special icon;
- assign a non structural permanent loads G_2 ;
- assign the variable load Q_k selecting the corresponding icon;


Loads on the floor
General
Name
Floor load: residential
Position
Internal floor
Permanent loads
Permanent loads
Description
Gk [kN/m²]
G1
Self-weight
Auto
G2
Non structural permanent loads
2
Live loads
Live loads
Description
Qk [kN/m²]
Variable cat.A
Live loads cat. A: Areas for domestic and residential activities
2
Add a live load
Show categories
Ok
Cancel

5 Project Properties Tools

5.1 Geolocation

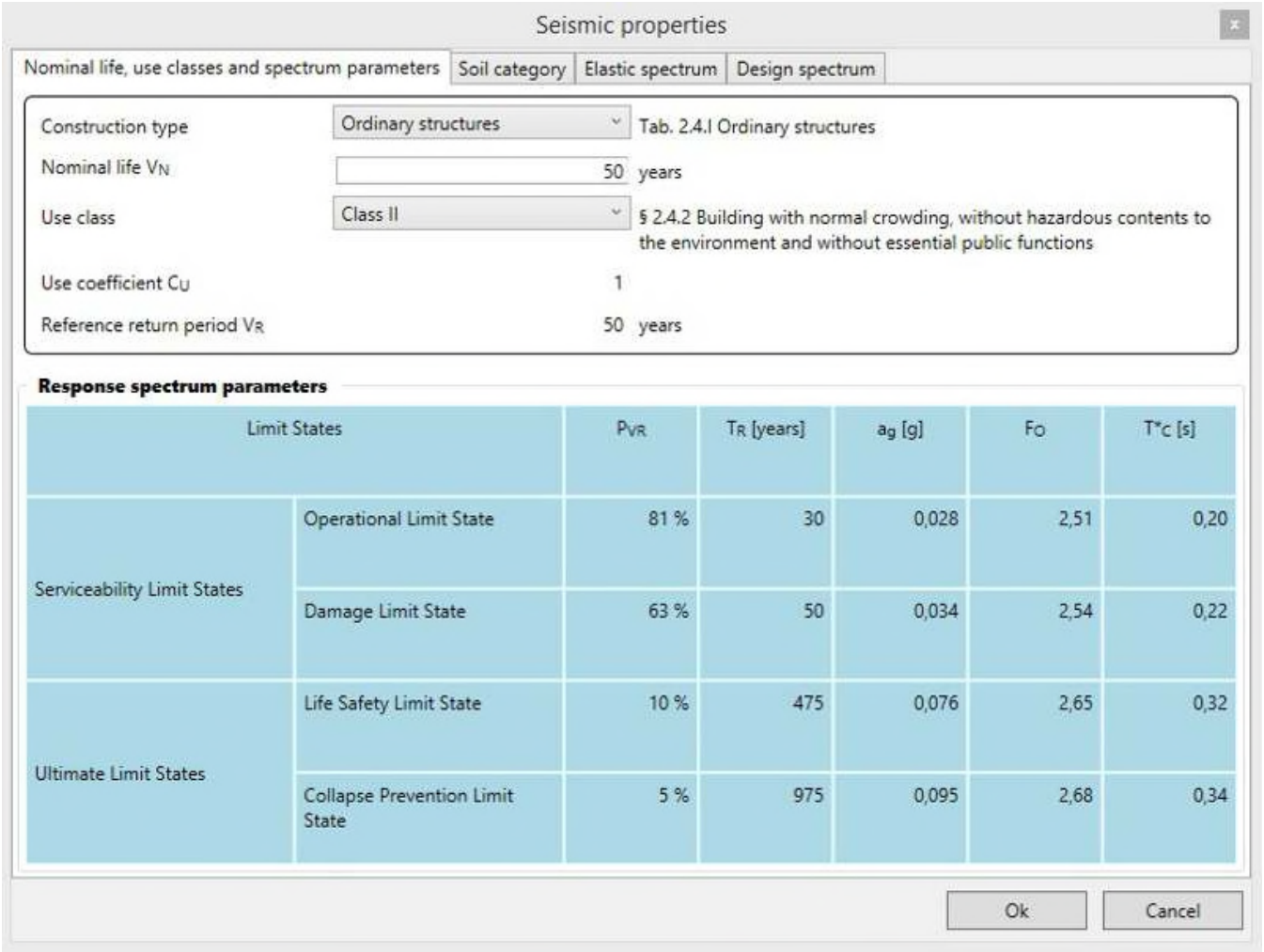


The command *Geolocation* allows the user to define the site of the building. The site can be located in the following ways:

- placing the cursor on map;
- selecting, in the corresponding drop-down menu, the region, the province and the city of the structure site;

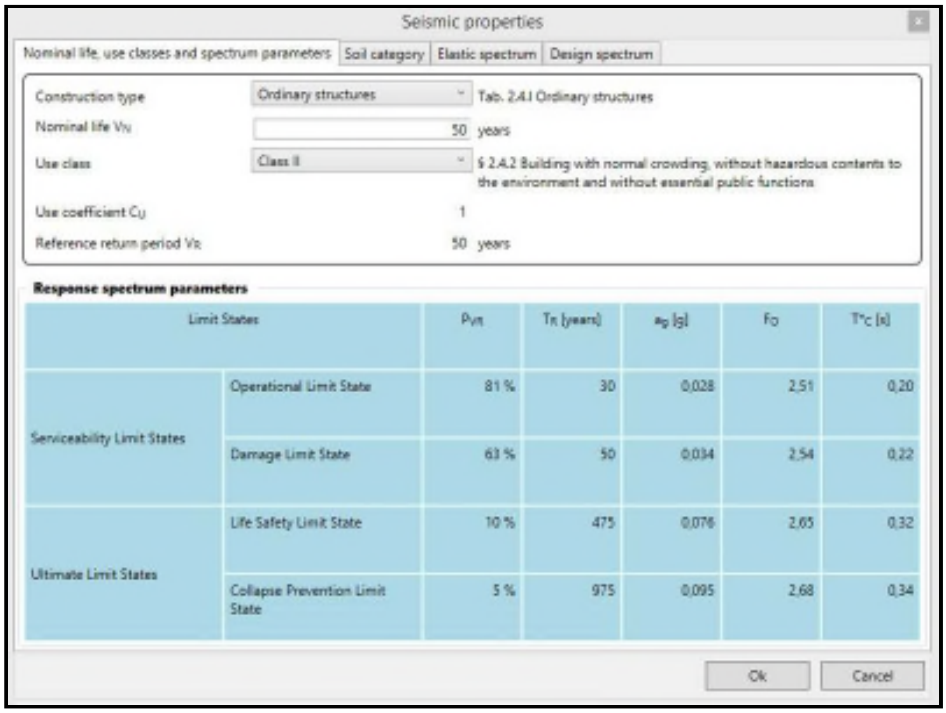
- typing directly the coordinates (latitude e longitude) and the altitude in meters above the sea level.

5.2 Seismic properties



The command *seismic proprieties* allows the user to define the seismic parameters according to the Italian Technical Code: ***Nuove Norme Tecniche per le costruzioni*** (**chapter 3**, **section 3.2**)

5.2.1 Nominal life, use class and spectrum parameters



The user has to define, in the dialog box:

- the construction type (by the drop-down menu);
- the nominal life V_N;
- the use class in the corresponding drop-down menu;

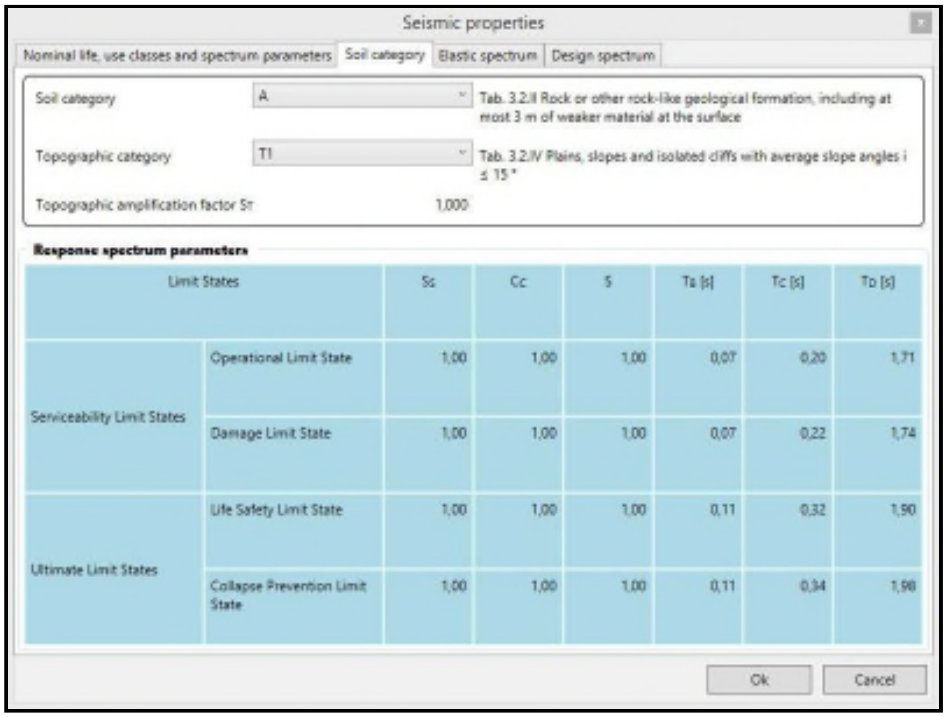
The dialog box provides:

- the use coefficients values C_U, corresponding to the selected use class;
- the reference return period value V_R given by the equation $V_R = V_N \times C_U$;

The window shows the parameters, corresponding to the Serviceability Limit States and Ultimate Limit States, used to define the elastic response spectrum:

- P_R: exceedance probability in the reference return period;
- T_R: reference return period corresponding to the exceedance probability;
- a_g : design ground acceleration;
- F_0: horizontal spectral acceleration amplification factor;
- T^*_C : period when the spectrum constant-velocity starts ;

5.2.2 Soil category



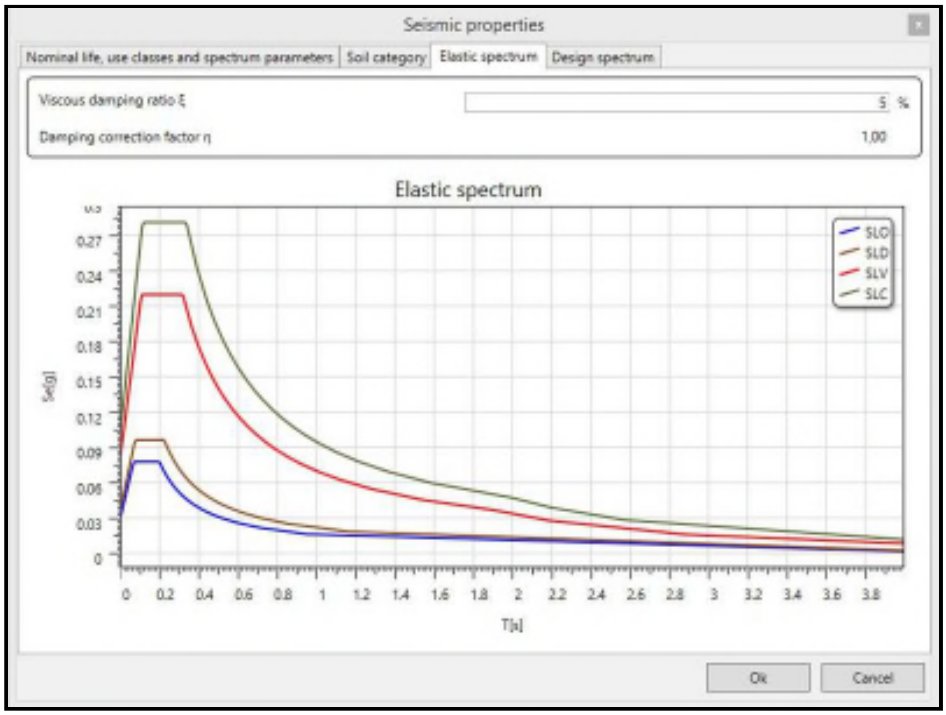
In the dialog box the user will:

- the soil category in the drop-down menu;
- the topographic category in the corresponding drop-down menu;
- the ratio between the height h of the site and the height H of the hill;

The dialog box provides the topographic amplification factor S_T and, corresponding to the Serviceability Limit States and the Ultimate Limit State, the value of the following parameters:

- S_S: is stratigraphic amplification factor;
- C_C: is a coefficient depending on the category of subsoil;
- S: is the Soil Factor depending on the soil category and on the topographic category. The parameter is obtained by the formula $S = S_S \cdot S_T$;
- T_B: is the period when the plateau at constant acceleration of the spectrum starts, defined by the expression $T_B = T_C/3$;
- T_C: is the period when this plateau ends defined by the expression $T_C = C_C \cdot T^*_C$;
- T_D is the value defining the beginning of the constant displacement response range of the spectrum defined by the expression $T_D = 4{,}0 \cdot a_g/g + 1{,}6$;

5.2.3 Elastic spectrum



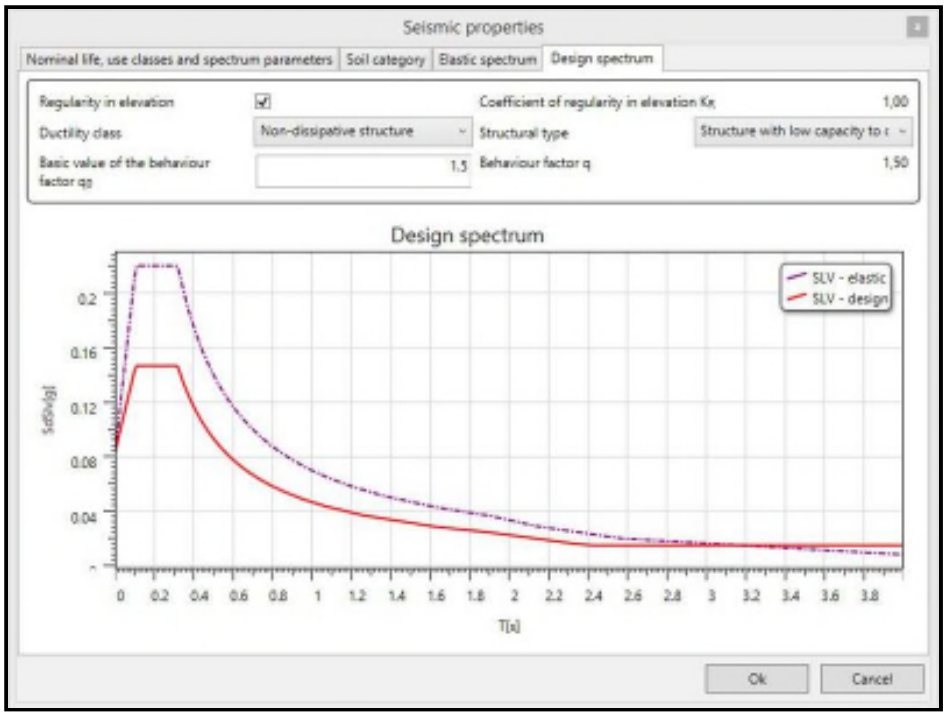
The elastic spectra, corresponding to the Serviceability Limit States and Ultimate Limit States, are displayed in the window. They are fully defined by the user parameters according to the expressions of the **Italian Technical Code** (Norme Tecniche per le costruzioni) at the point **3.2.3.2.1**.

The user must define:

- the value of the *viscous damping ratio* ξ (expressed as a percentage), depending on the materials, structural typology and soil category.

The value of the *viscous damping ratio* depends on the damping correction factor η, which modifies the response spectrum in case of ξ different from 5%.

5.2.3 Design spectrum



The design spectrum is displayed in the dialog box (the red line). This spectrum is obtained applying the *behaviour factor* **q** ; it considers the dissipative capacity of the structure. The Italian Technical Standards (**NTC 2008**), point **3.2.3.5** , provide the following prescription:

If the Ultimate Limit State checks are not carried out through the use of appropriate accelerograms and dynamic analysis by time step integration, in order to design or check the structures, the dissipative capabilities can be brought into account by reducing the elastic forces; the reduction takes ***simply*** *into account the inelastic energy dissipation capacity of the structure, its overstrength and the increase of its own period as a result of plasticisation. In this case, the design spectrum* $S_d(T)$ *to be used both for the horizontal components and for the vertical component, is the corresponding elastic spectrum with the ordinates reduced by replacing in the formulas* ***3.2.3.2.1*** η *with 1/q.*

The user, in the dialog box, must:

- define the structural regularity in elevation;
- define the ductility class;
- modify if necessary, the basic value of the behaviour factor q_0 , provided by the Italian Technical Standars **NTC 2008** , point **7.7.3**;

The dialog box provides:

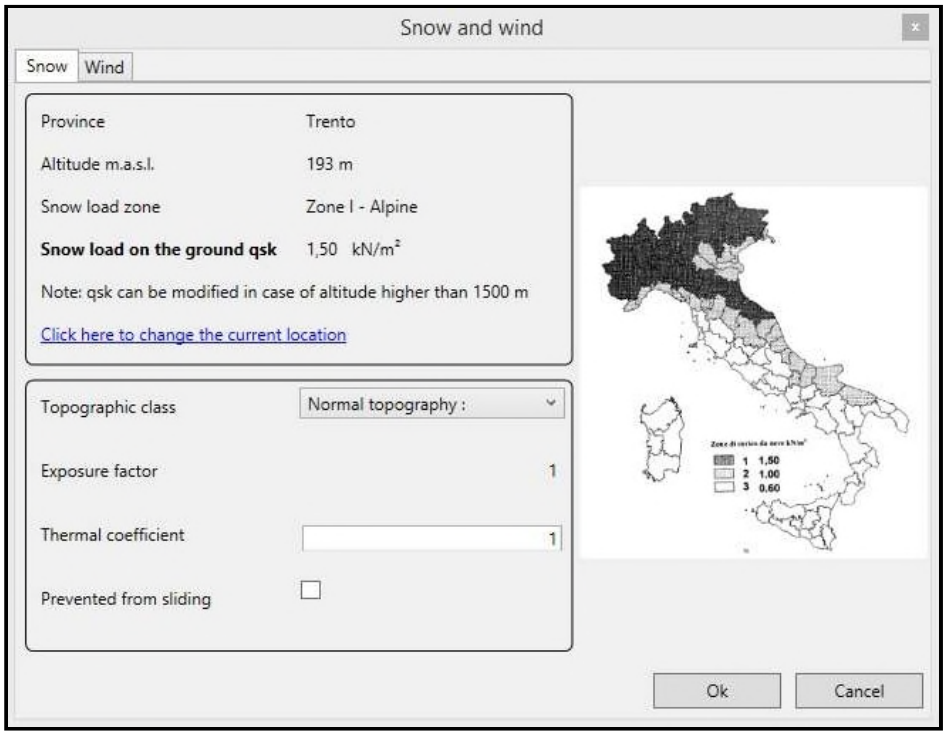
- the coefficient of regularity in elevation K_R, which is equal to 1 in case of regularity in elevation and 0,8 otherwise;
- the behaviour factor calculated by the formula $q = q_0 \cdot K_R$;

5.3 Snow/Wind

The command *Snow/Wind* allows the user to complete the loads definitions according to the structural site geolocation.

The wind load and the snow one are obtained according to the Italian Technical Standards **NTC 2008** at the point **3.3** (*Wind loads*) and **3.4** (*Snow loads*), respectively.

5.3.1 Snow



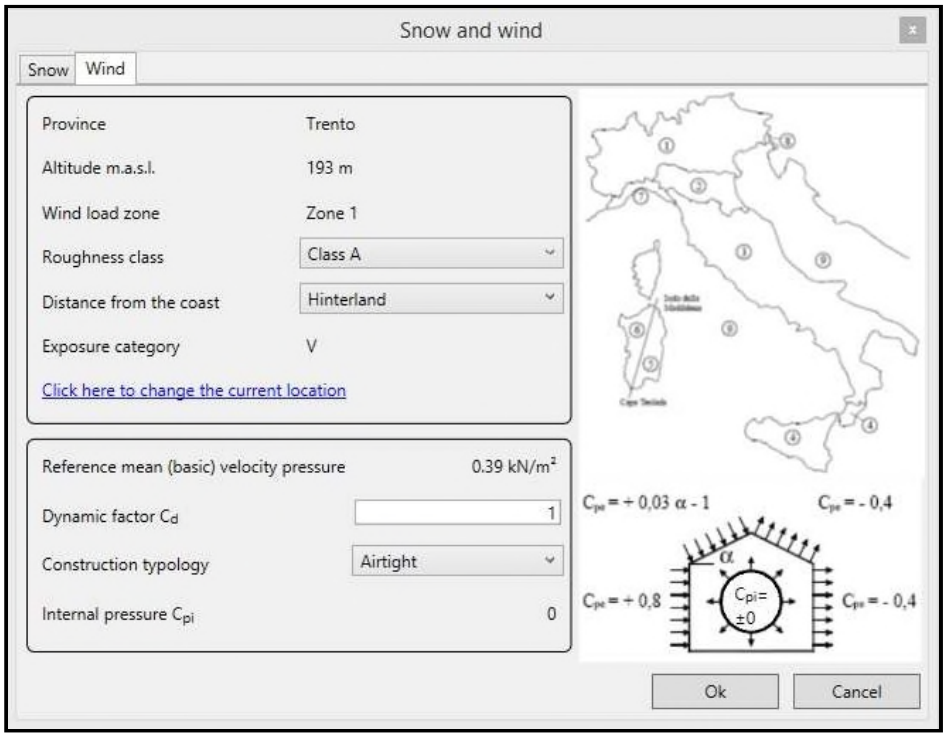
In the dialog box the user has to:

- select, in the drop-down menu, the site topographic class according to the location of the structure;
- edit the exposure factor C_E provided by the software;
- choose, in the checkbox, whether the snow is prevented from sliding;
- edit, if necessary, the site geolocation;

The dialog box provides:

- the province where the structure is located;
- the site altitude in meters above sea level;
- the *Snow load zone* ;
- the snow load on the ground q_{sk};
- the assumed thermal coefficient C_t;

5.3.2 Wind



In the dialog box the user has to:

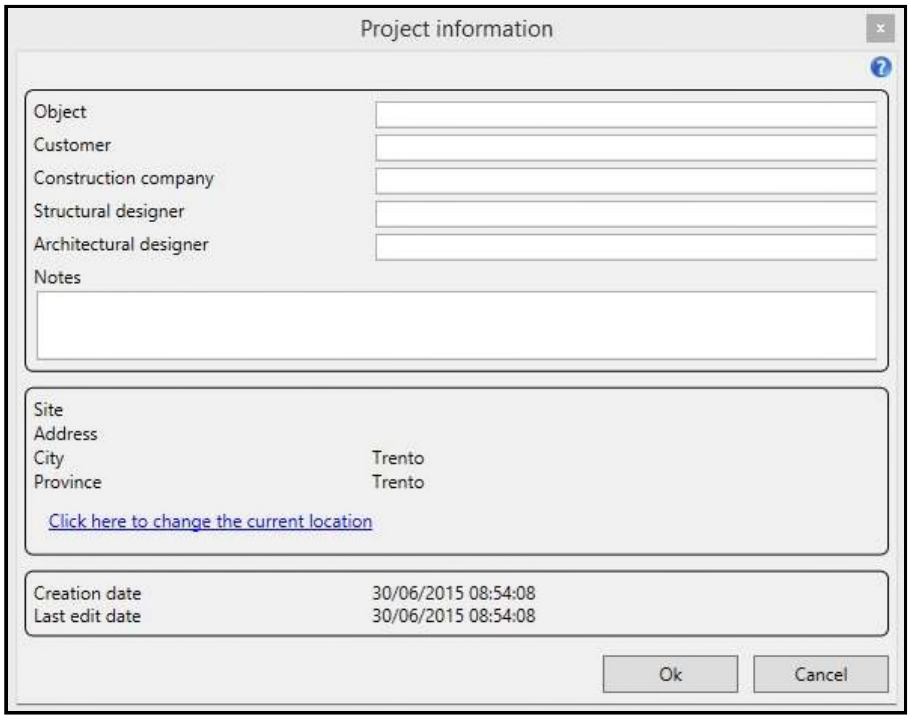
- select, in the drop-down menu, the site topographic class according to the structure location;
- select, in the drop-down menu, the distance from the coast used to define the exposure category;
- modify the dynamic C_d assumed by the software;
- select, in the drop-down menu, the construction typology depending on the external openings; this datum is used to calculate the internal pressure C_{pi};

The dialog box provides:

- the province where the structure is located;
- the site altitude in meter above sea level;
- the *wind load zone*;
- the exposure category of the site;
- the reference mean (basic) velocity pressure q_0;
- the internal pressure C_{pi}.

5.4 Project informations

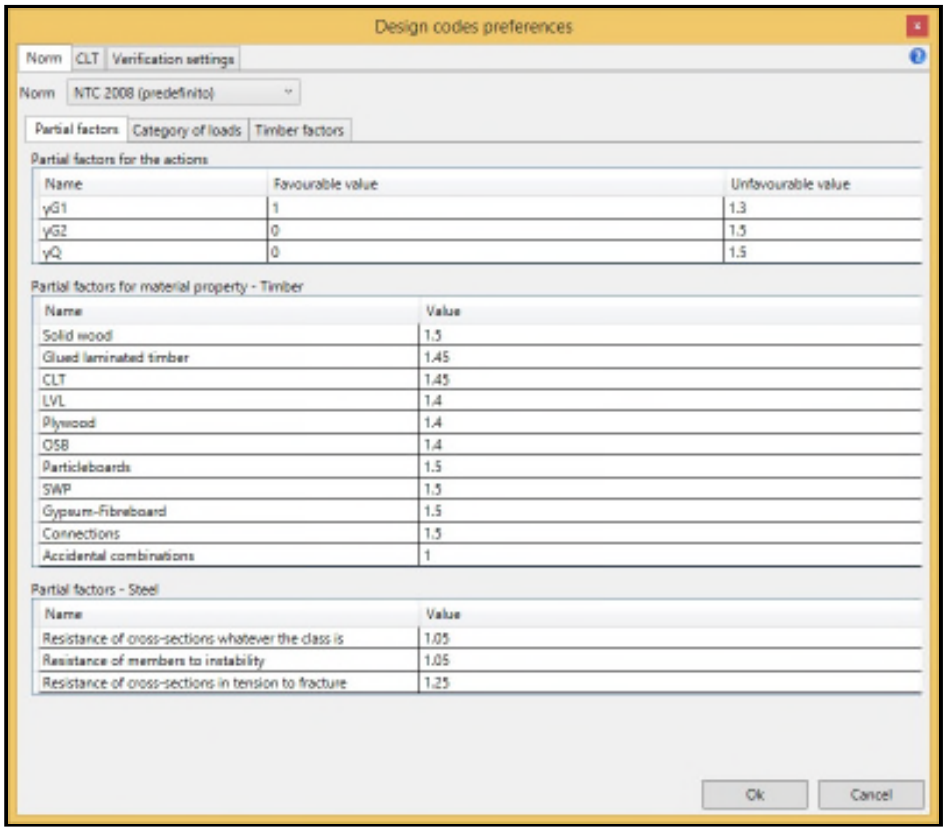
The command *Project informations* allows the user to type, in the dialog box, the informations about the project object, the customer, the construction company, the structural designer and the architectural designer. In the dialog box the user can modify, if necessary, the site geolocation.

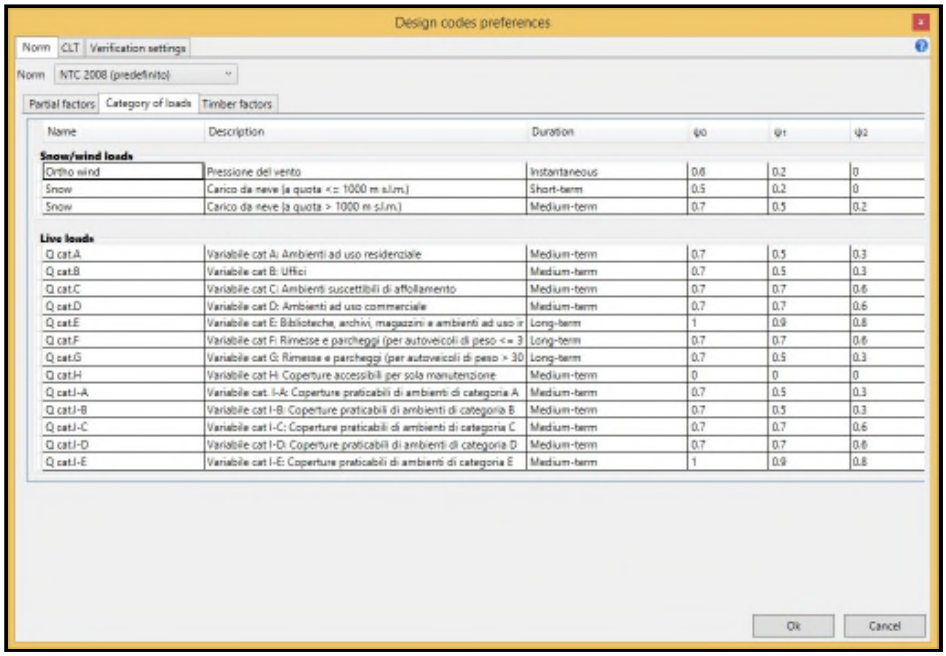


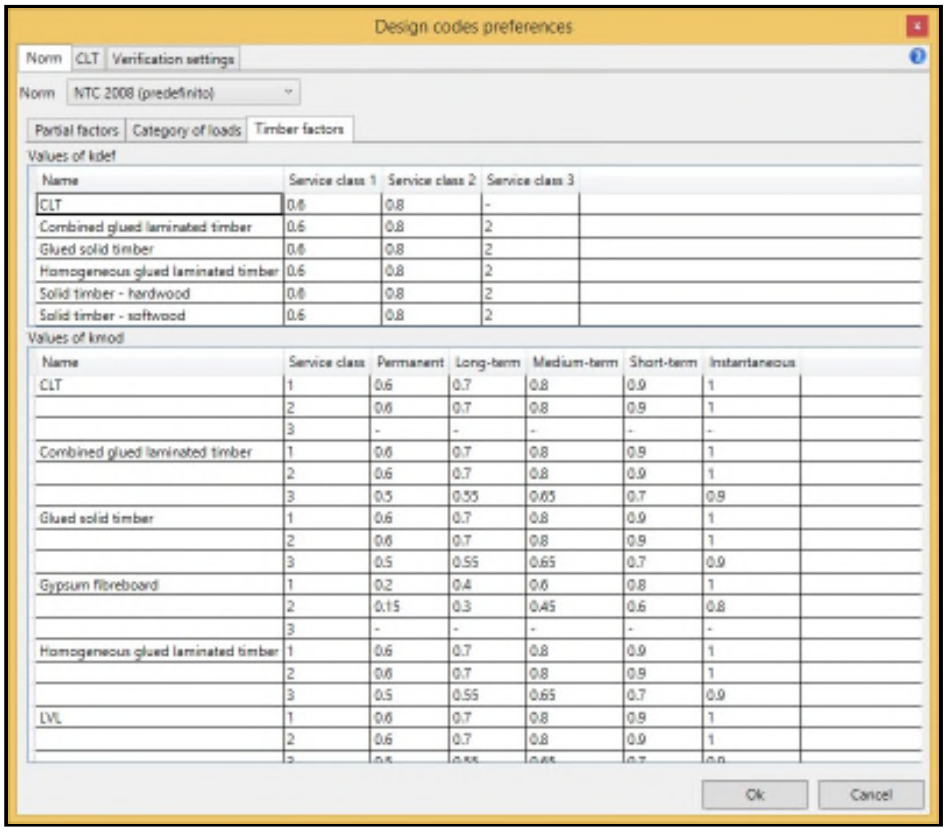
5.5 Design codes preferences

5.5.1 Norm

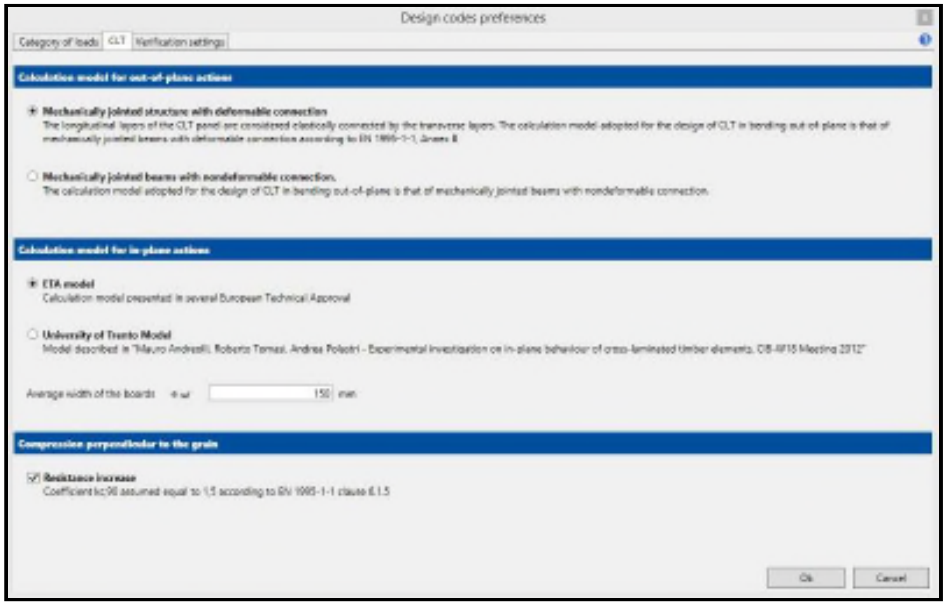
The command provides the access to the table reporting the partial factors for the actions, the partial factors for timber and steel, the loads combination factors which depend on the load duration and the timber factors kded and kmod.


Design codes preferences
Norm
CLT
Verification settings
Norm
NTC 2008 (predefinito)
Partial factors
Category of loads
Timber factors
Partial factors for the actions
Name
Favourable value
Unfavourable value
yG1
1
1.3
yG2
0
1.5
yQ
0
1.5
Partial factors for material property - Timber
Name
Value
Solid wood
1.5
Glued laminated timber
1.45
CLT
1.45
LVL
1.4
Plywood
1.4
OSB
1.4
Particleboards
1.5
SWP
1.5
Gypsum-Fibreboard
1.5
Connections
1.5
Accidental combinations
1
Partial factors - Steel
Name
Value
Resistance of cross-sections whatever the class is
1.05
Resistance of members to instability
1.05
Resistance of cross-sections in tension to fracture
1.25
Ok
Cancel


Design codes preferences
Norm
CLT
Verification settings
Norm
NTC 2008 (predefinito)
Partial factors
Category of loads
Timber factors
Name
Description
Duration
φ0
φ1
φ2
Snow/wind loads
Ortho wind
Pressione del vento
Instantaneous
0.6
0.2
0
Snow
Carico da neve (a quota <= 1000 m s.l.m.)
Short-term
0.5
0.2
0
Snow
Carico da neve (a quota > 1000 m s.l.m.)
Medium-term
0.7
0.5
0.2
Live loads
Q cat.A
Variabile cat A: Ambienti ad uso residenziale
Medium-term
0.7
0.5
0.3
Q cat.B
Variabile cat B: Uffici
Medium-term
0.7
0.5
0.3
Q cat.C
Variabile cat C: Ambienti suscettibili di affollamento
Medium-term
0.7
0.7
0.6
Q cat.D
Variabile cat D: Ambienti ad uso commerciale
Medium-term
0.7
0.7
0.6
Q cat.E
Variabile cat E: Biblioteche, archivi, magazzini e ambienti ad uso
Long-term
1
0.9
0.8
Q cat.F
Variabile cat F: Rimesse e parcheggi (per autoveicoli di peso <= 3
Long-term
0.7
0.7
0.6
Q cat.G
Variabile cat G: Rimesse e parcheggi (per autoveicoli di peso > 30
Long-term
0.7
0.5
0.3
Q cat.H
Variabile cat H: Coperture accessibili per sola manutenzione
Medium-term
0
0
0
Q cat.I-A
Variabile cat. I-A: Coperture praticabili di ambienti di categoria A
Medium-term
0.7
0.5
0.3
Q cat.I-B
Variabile cat I-B: Coperture praticabili di ambienti di categoria B
Medium-term
0.7
0.5
0.3
Q cat.I-C
Variabile cat I-C: Coperture praticabili di ambienti di categoria C
Medium-term
0.7
0.7
0.6
Q cat.I-D
Variabile cat I-D: Coperture praticabili di ambienti di categoria D
Medium-term
0.7
0.7
0.6
Q cat.I-E
Variabile cat I-E: Coperture praticabili di ambienti di categoria E
Medium-term
1
0.9
0.8
Ok
Cancel



5.5.2 CLT



The tab *CLT* allows the user to choose the calculation model used to verify the CLT elements. With regard to the out of plane loads, the user can choose two methods:

- model based on the assumption of ***mechanically jointed structure with deformable connection***;
- model based on the assumption of ***mechanically jointed beam with non-deformable connection***;

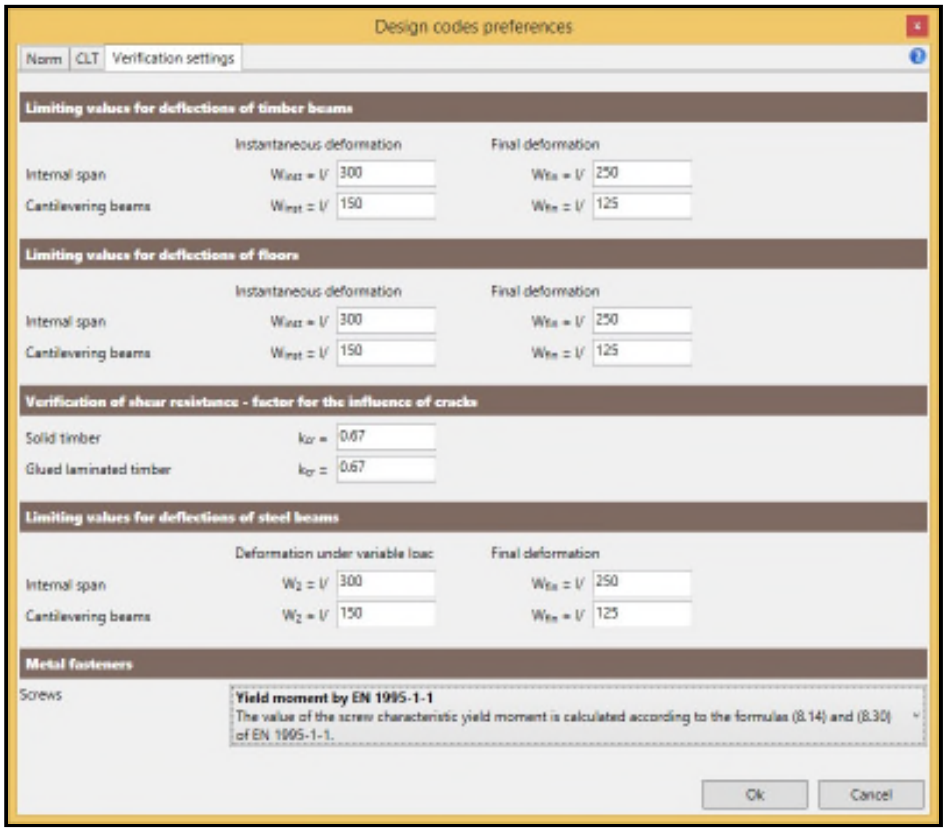
With regard to the in-plane actions, the user can choose between:

- ***ETA model***;
- ***Model developed by the University of Trento***;

The user can even:

- modify the average width of the boards a_{ref}. The parameter is used to calculate the in-plane strength and the assumed shear modulus G_{eff} calculated according to the model proposed in: *Verification fo CLT-plates under loads in plane, Bogensperger T., Moosbrugger T., Silly G."*
- increase the compression strength perpendicular to the grain assuming k_c equal to 1,5.

5.5.3 Verification settings



The *Verification settings* tab provides the access to the dialog box which allows to set the limit deformations of the beams and of the floors. The user can modify the factor for the cracks influence in the shear verification. With regard to the yield moment of the screws, the user can choose between:

- ETA model;
- EN 1995-1-1.

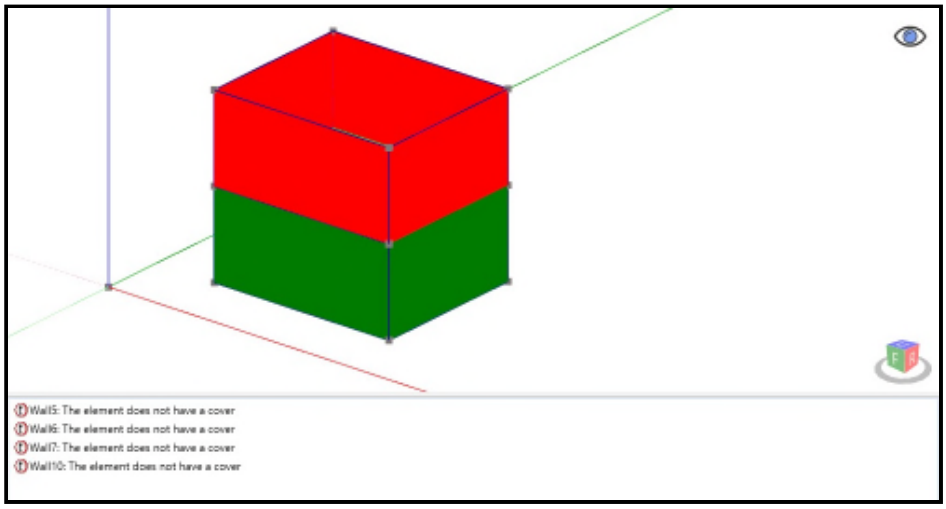
6 Validate and Calculate Tools

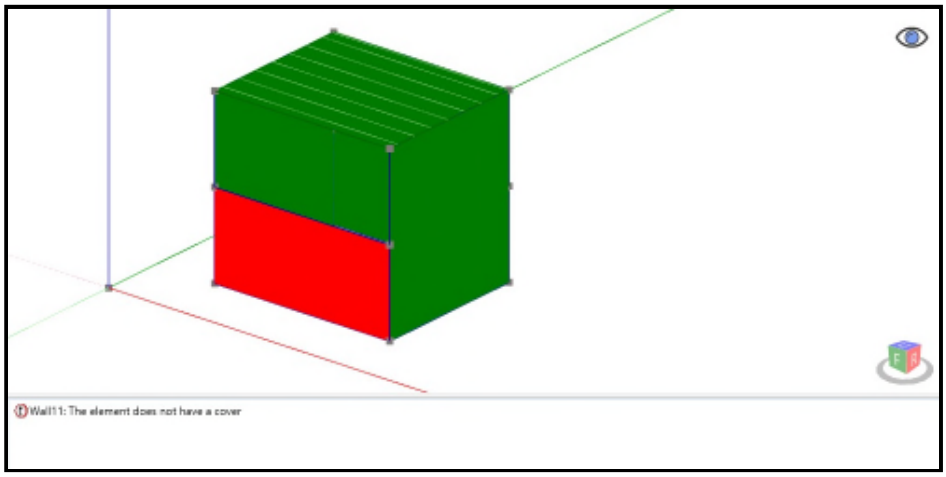
The commands *Validate and calculate* allows to validate the structural model.

6.1 Validate

Clicking the *Validate and calculate* command the software validates the model, finds the bad constrained elements or the structural elements badly defined (referring to the model hypothesis). These structural elements are displayed in red. The following paragraphs describe the most commons case where the model is badly defined.

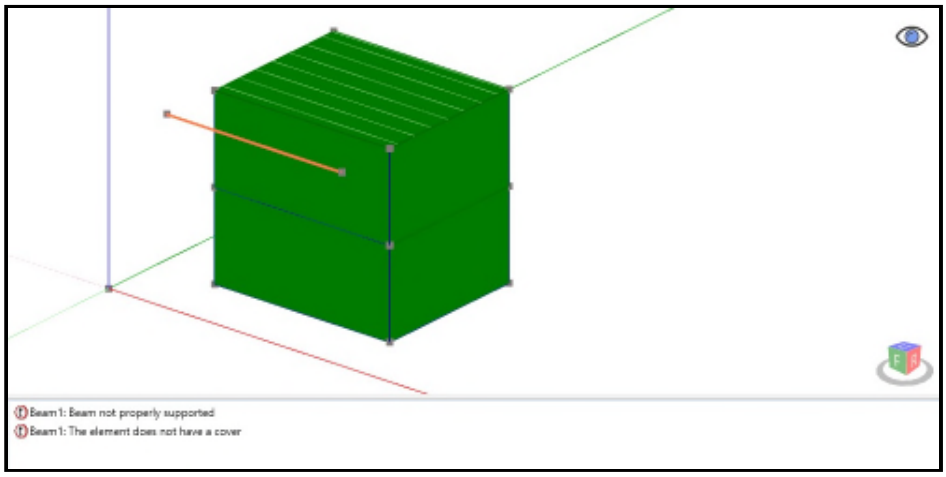
6.1.1 Element without a cover

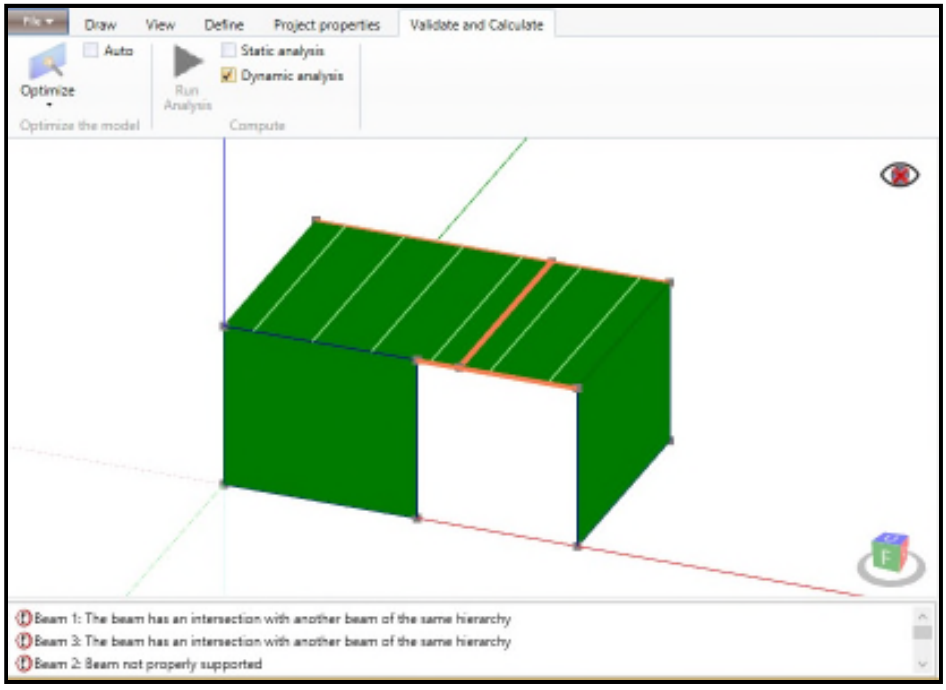




A wall element has to ever support a floor element except the non primary wall. The pillar element, in the same way, must support a beam element.

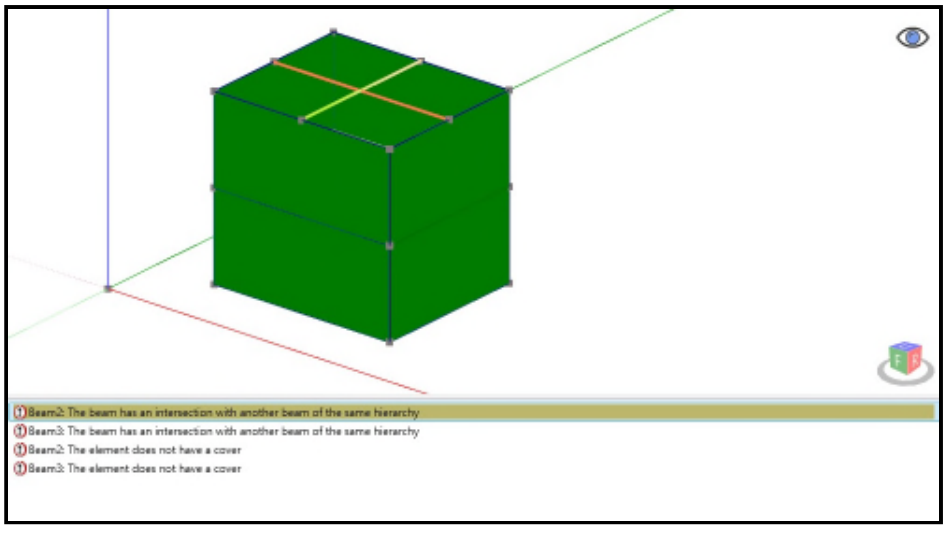
6.1.2 Beam not properly supported





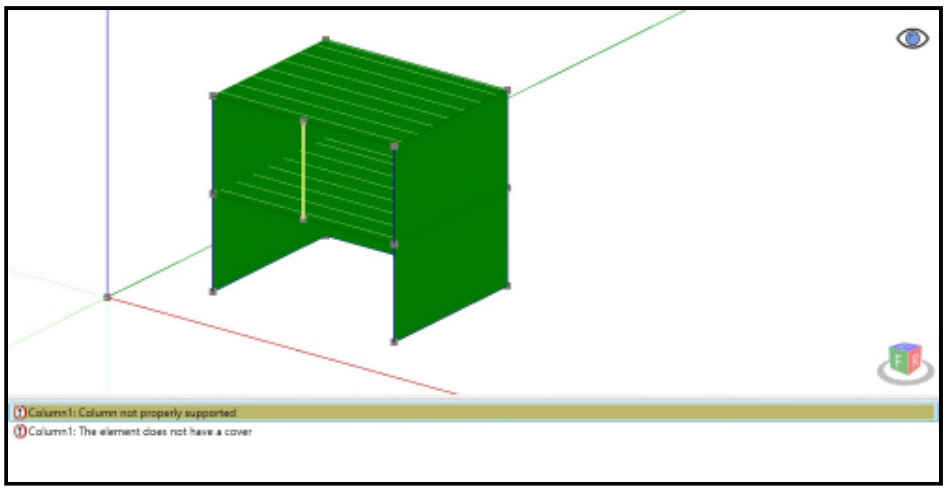
The beam element has to ever be supported by a pillar or a wall. If a beam supports another beam, this last one has to be defined as a secondary element (or 3th and so on).

6.1.3 The beam intersects another beam of the same hierarchy



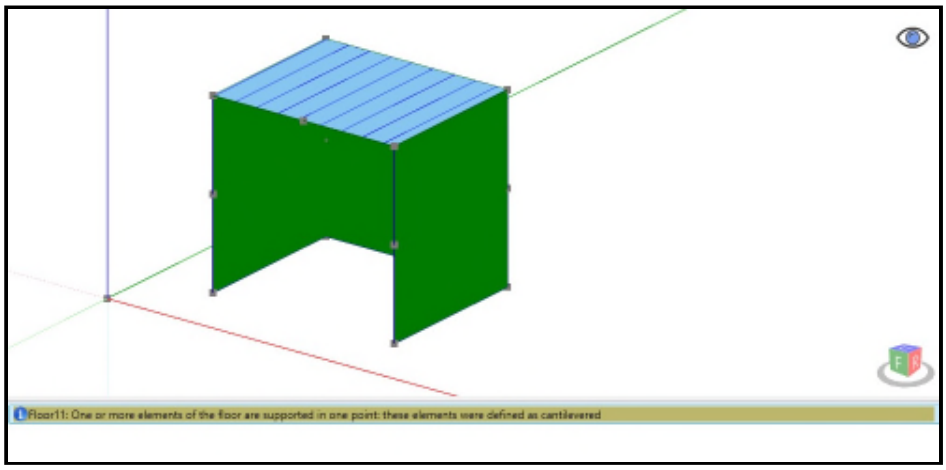
Two elements, belonging to the same hierarchy, must not intersect each other. To overcome the model problem, the user has to assign one of them to a lower hierarchy (2th or 3th).

6.1.4 Pillar not properly supported



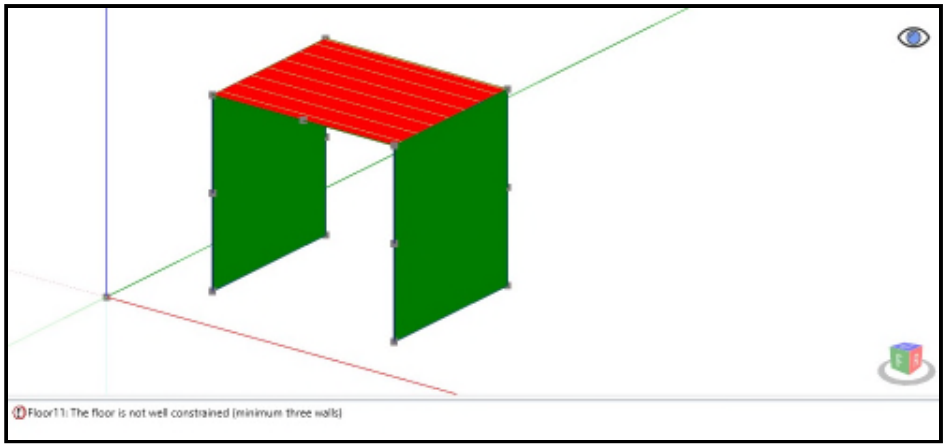
The pillar element must be supported by another pillar, by a wall or by a beam element.

6.1.5 Floor not well supported



The floor must be supported by the walls or by beam element (orthogonal to the floor direction). If the floor is supported in one point, it will be considered as cantilevered.

6.1.6 Floor not well constrained

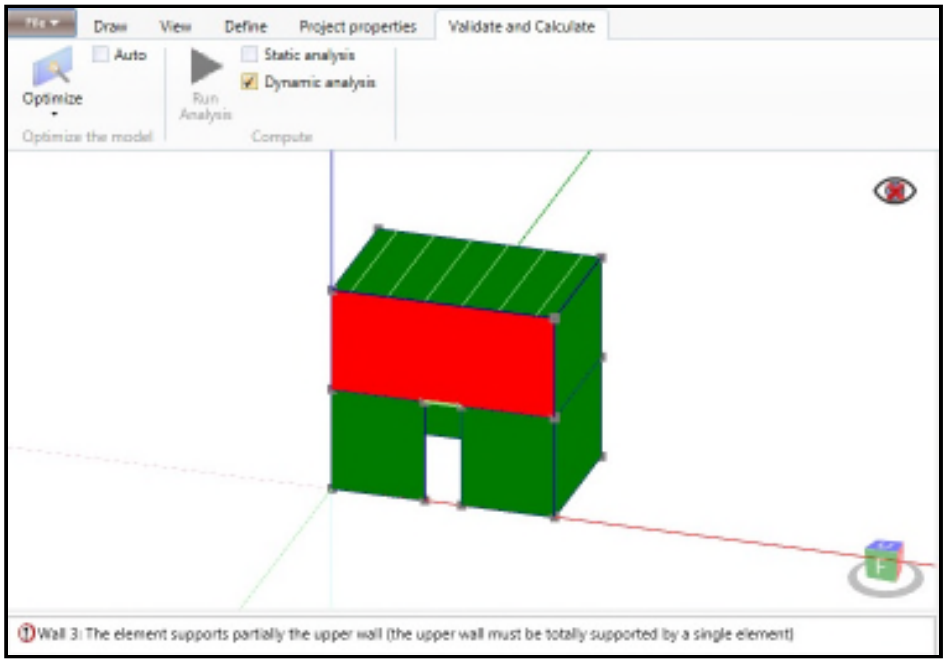


The floor must be supported by at least three walls. The wall elements must be defined as primary, that is enable to overcome the seismic action.

If the floor seems to be well constrained but the software reports the error, the user must verify:

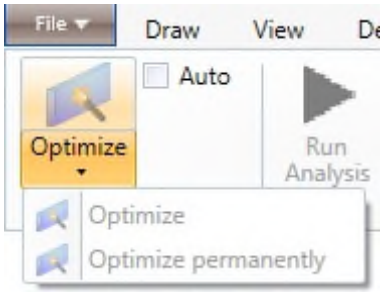
- if all the walls are defined as primary
- if the software automatically assigns to a wall a different hierarchy
- if the walls (at least three) are well positioned

6.1.7 The element partially supports the upper primary wall



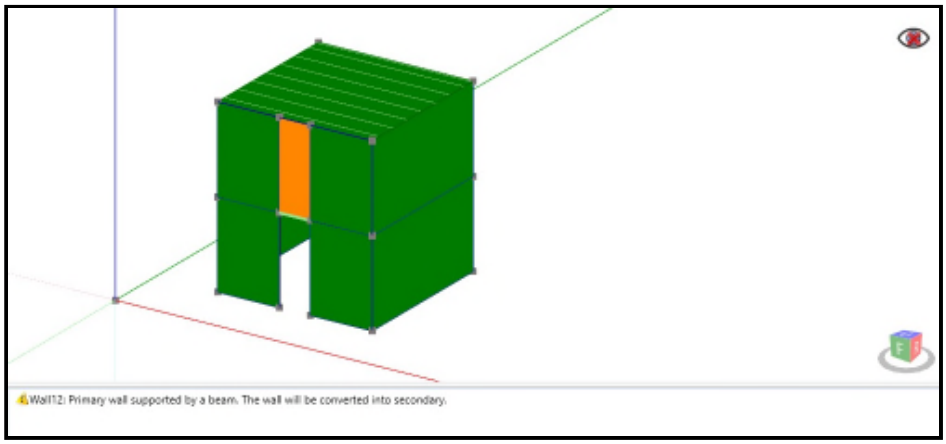
The primary wall element must be totally supported by one single wall.

6.2 Optimize

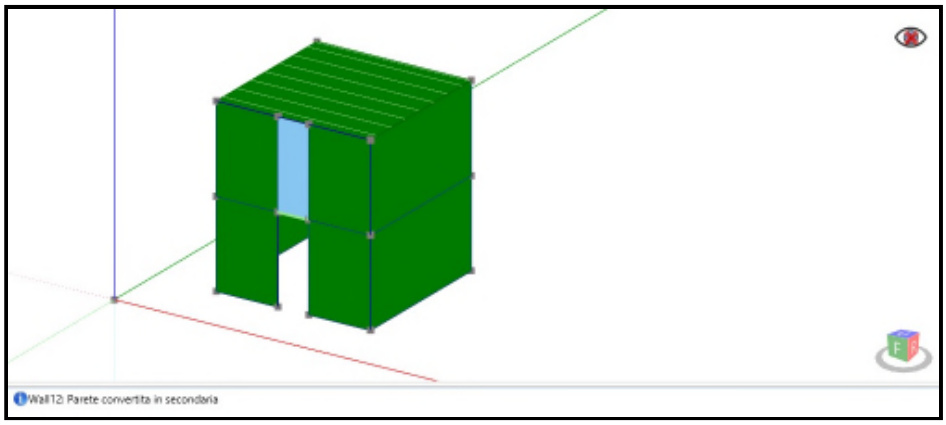


The software *Validate* command identifies the structural element which can not be assumed to be primary elements (element which are resistant to the horizontal loads). The command *Optimize* allows the user to modify this elements hierarchy (displayed in orange). The command Optimize permanently allows the user to permanently change the model after the optimization process

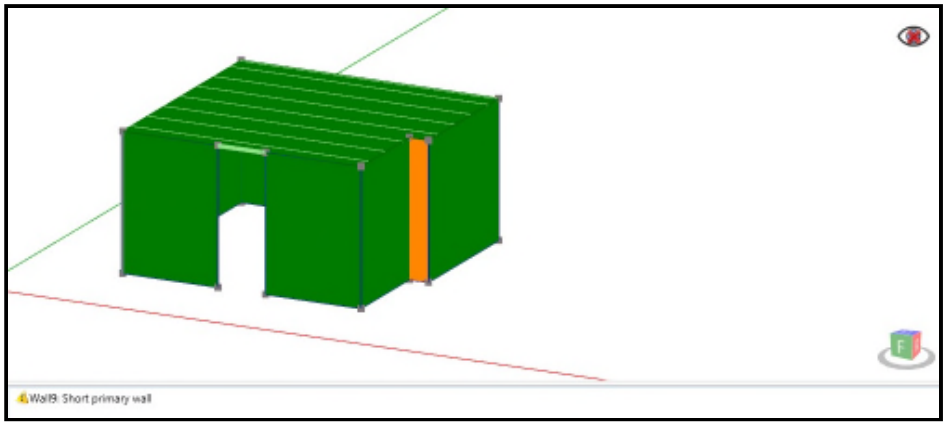
6.2.1 Wall not properly supported



In this case the element, supported by a beam, can not overcome the horizontal loads and the software will classify it as a secondary element.



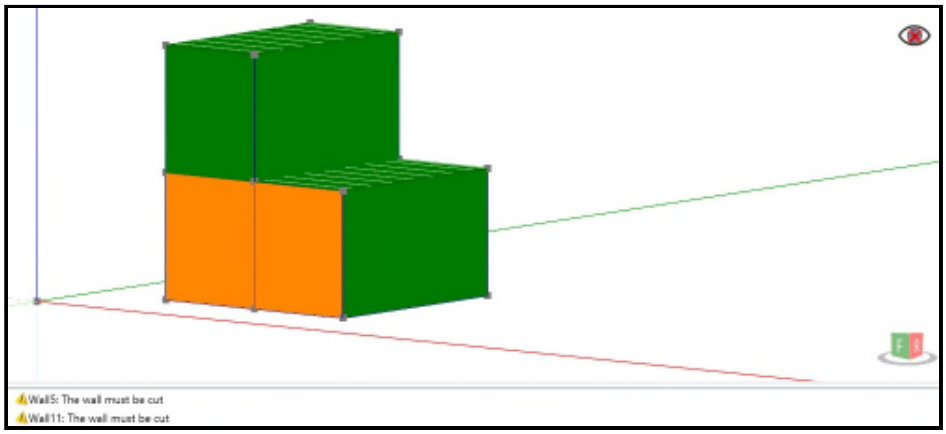
6.2.2 The primary wall is too short

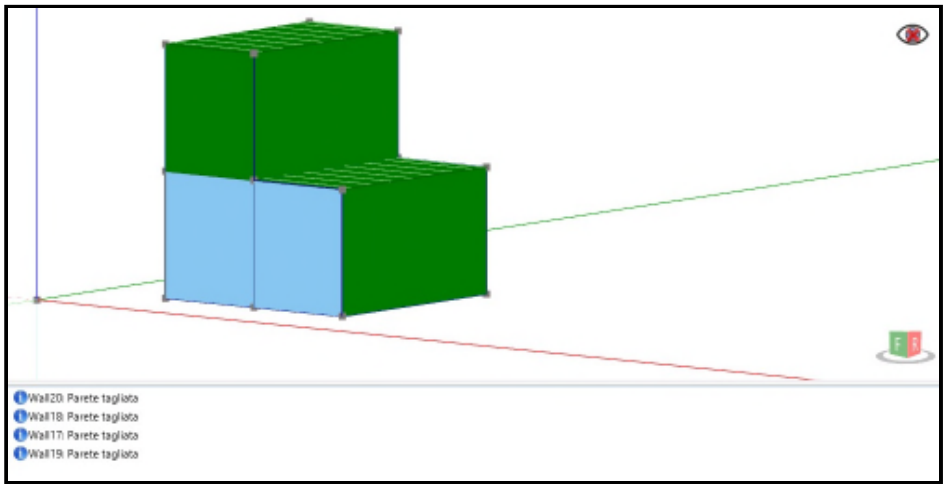


The walls, that are shorter than h/4, will classified as secondary walls.

6.2.3 The wall must be cut

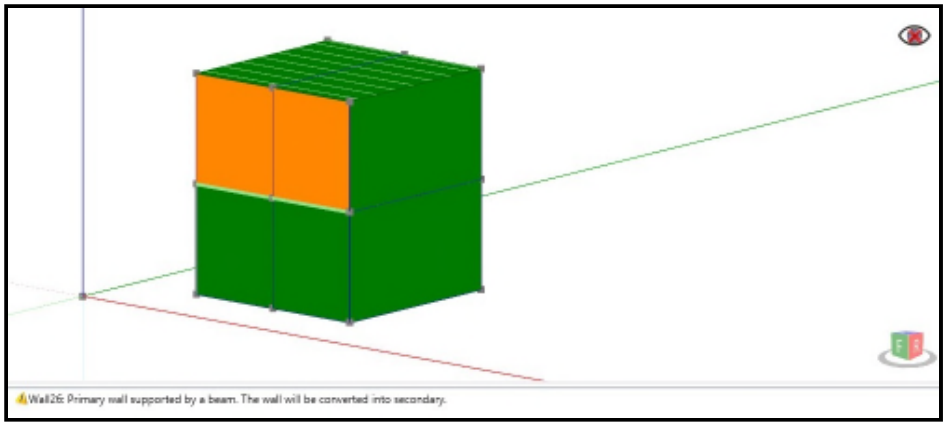
When a primary wall is more than 0.6 m longer than the supported one, the element must be cut where the supported elements "ends".



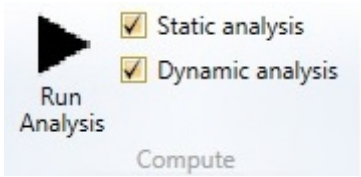


6.2.4 The primary wall is supported by a beam

If a wall is supported by a beam the software will automatically define it as a secondary element. The user could also delete the beam: in this case the beam has not any loads because they are directly transferred by the upper wall to the underlying one.



6.3 Run analysis



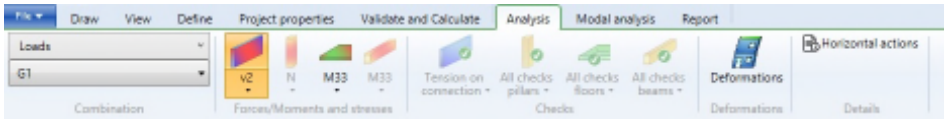
The command *Run Analysis* runs the structural analysis of the model. The model can be correctly analysed when all the elements are green.

The user can run **different analysis**:

- Analysis of vertical loads and horizontal load of the wind;
- Seismic Static Analysis*;
- Seismic Dynamic Analysis*;

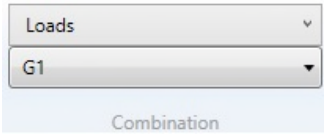
* the software ever run the analysis of vertical load

7. Tools



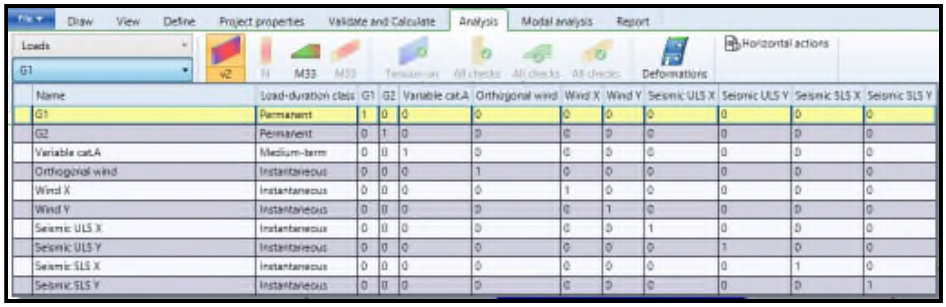
The *Analysis* menu allows the user to look at the element forces and deformation, to control the data and the elements checks.

7.1 Combination



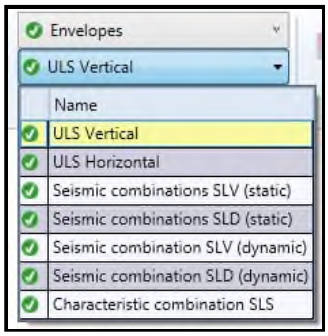
The menus *Combinations* allows the user to select a load or a combination to display the respective forces/deformations of the elements and the checks data. The red symbol near the combination (in the drop-down menu) means the non successful verification.

7.1.1 Loads



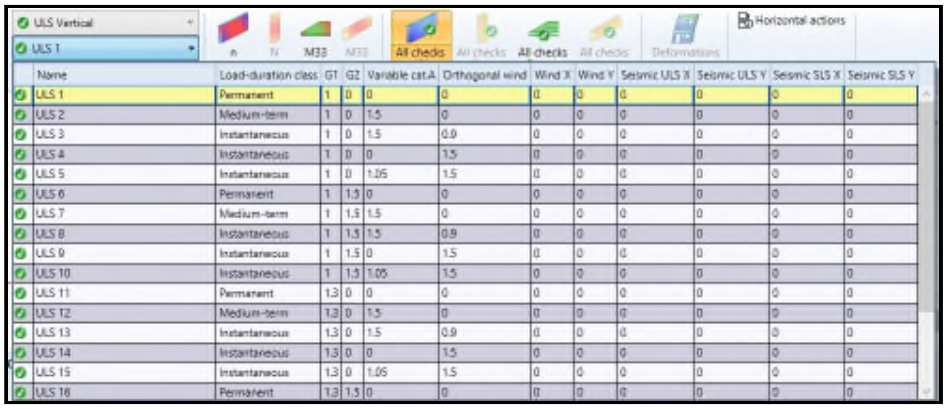
If the user selects *Loads*, the first item in the drop-down menu, he could select the single loads category of which data he wants to display (the forces/deformations levels and the checks data).

7.1.2 Envelopes



If the user selects Envelopes, the second item in the drop-down menu, he could select the combination category of which data he wants to display (the maximum forces/deformations value of all the envelopes corresponding to the selected categories).

7.1.3 ULS Vertical



Name	Load-duration class	G1	G2	Variable cat.A	Orthogonal wind	Wind X	Wind Y	Seismic ULS X	Seismic ULS Y	Seismic SLS X	Seismic SLS Y
ULS 1	Permanent	1	0	0	0	0	0	0	0	0	0
ULS 2	Medium-term	1	0	1.5	0	0	0	0	0	0	0
ULS 3	Instantaneous	1	0	1.5	0.9	0	0	0	0	0	0
ULS 4	Instantaneous	1	0	0	1.5	0	0	0	0	0	0
ULS 5	Instantaneous	1	0	1.05	1.5	0	0	0	0	0	0
ULS 6	Permanent	1	1.5	0	0	0	0	0	0	0	0
ULS 7	Medium-term	1	1.5	1.5	0	0	0	0	0	0	0
ULS 8	Instantaneous	1	1.5	1.5	0.9	0	0	0	0	0	0
ULS 9	Instantaneous	1	1.5	0	1.5	0	0	0	0	0	0
ULS 10	Instantaneous	1	1.5	1.05	1.5	0	0	0	0	0	0
ULS 11	Permanent	1.3	0	0	0	0	0	0	0	0	0
ULS 12	Medium-term	1.3	0	1.5	0	0	0	0	0	0	0
ULS 13	Instantaneous	1.3	0	1.5	0.9	0	0	0	0	0	0
ULS 14	Instantaneous	1.3	0	0	1.5	0	0	0	0	0	0
ULS 15	Instantaneous	1.3	0	1.05	1.5	0	0	0	0	0	0
ULS 16	Permanent	1.3	1.5	0	0	0	0	0	0	0	0

If the user selects *ULS Vertical*, the second menu in the first drop-down menu, he could select the combination category of which data he wants to display. The combinations are obtained according to the **Italian Technical Code** (NTC 2008), point **2.5.3**. In the case of timber structures, the definition of the loads combinations needs to be pointed out. If the load combination considers actions corresponding to different duration classes, the software will assign to the strength the respective k_{mod} factor. That is why the software automatically considers all the combinations: find out the worst one is not elementary.

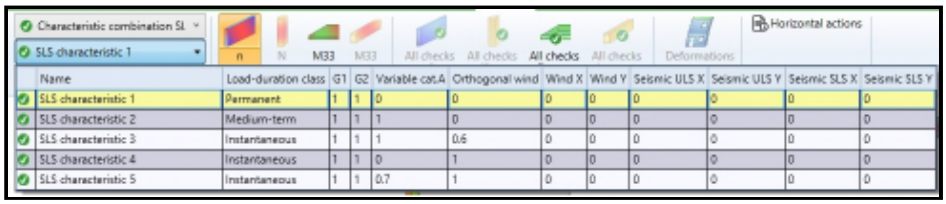
7.1.4 Seismic Combinations

Name	Load-duration class	G1	G2	Variable cat.A	Orthogonal wind	Wind X	Wind Y	Seismic ULS X	Seismic ULS Y	Seismic SLS X	Seismic SLS Y
Seismic ULS 1 ex+ ey+	Instantaneous	1	1	0.3	0	0	0	1	0.3	0	0
Seismic ULS 1 ex+ ey-	Instantaneous	1	1	0.3	0	0	0	1	0.3	0	0
Seismic ULS 1 ex- ey+	Instantaneous	1	1	0.3	0	0	0	1	0.3	0	0
Seismic ULS 1 ex- ey-	Instantaneous	1	1	0.3	0	0	0	1	0.3	0	0
Seismic ULS 2 ex+ ey+	Instantaneous	1	1	0.3	0	0	0	1	-0.3	0	0
Seismic ULS 2 ex+ ey-	Instantaneous	1	1	0.3	0	0	0	1	-0.3	0	0
Seismic ULS 2 ex- ey+	Instantaneous	1	1	0.3	0	0	0	1	-0.3	0	0
Seismic ULS 2 ex- ey-	Instantaneous	1	1	0.3	0	0	0	1	-0.3	0	0
Seismic ULS 3 ex+ ey+	Instantaneous	1	1	0.3	0	0	0	-1	0.3	0	0
Seismic ULS 3 ex+ ey-	Instantaneous	1	1	0.3	0	0	0	-1	0.3	0	0
Seismic ULS 3 ex- ey+	Instantaneous	1	1	0.3	0	0	0	-1	0.3	0	0
Seismic ULS 3 ex- ey-	Instantaneous	1	1	0.3	0	0	0	-1	0.3	0	0
Seismic ULS 4 ex+ ey+	Instantaneous	1	1	0.3	0	0	0	-1	-0.3	0	0
Seismic ULS 4 ex+ ey-	Instantaneous	1	1	0.3	0	0	0	-1	-0.3	0	0
Seismic ULS 4 ex- ey+	Instantaneous	1	1	0.3	0	0	0	-1	-0.3	0	0
Seismic ULS 4 ex- ey-	Instantaneous	1	1	0.3	0	0	0	-1	-0.3	0	0

If the user selects the Seismic Combinations (first drop-down menu) he could select one of the 32 combinations to display the results data. The combinations are obtained by changing the centre of mass, to consider the accidental eccentricity, and by changing the values and the directions of the seismic actions (the 100% of the value in one direction corresponds to the 30 % of the value in the other direction.

The user can also decide to display the Seismic Combinations corresponding to the Static Analysis or to the Dynamic Analysis.

7.1.5 Characteristic combinations SLS



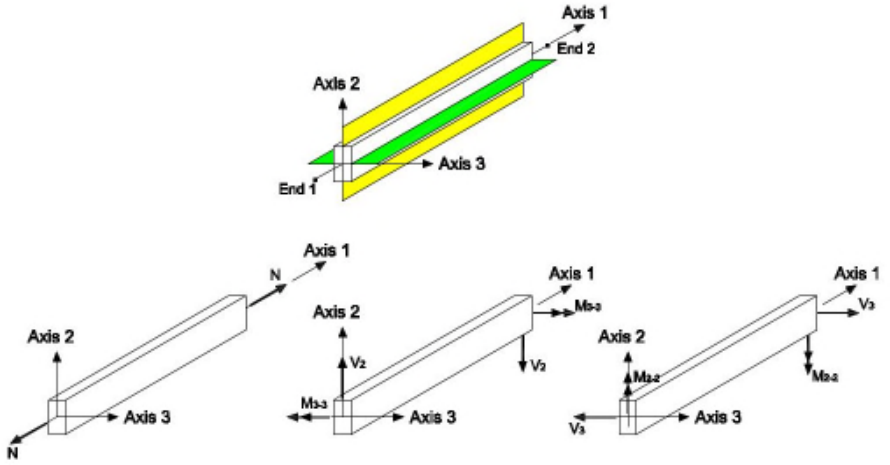
Name	Load-duration class	G1	G2	Variable cat.A	Orthogonal wind	Wind X	Wind Y	Seismic ULS X	Seismic ULS Y	Seismic SLS X	Seismic SLS Y
SLS characteristic 1	Permanent	1	1	0	0	0	0	0	0	0	0
SLS characteristic 2	Medium-term	1	1	1	0	0	0	0	0	0	0
SLS characteristic 3	Instantaneous	1	1	1	0.6	0	0	0	0	0	0
SLS characteristic 4	Instantaneous	1	1	0	1	0	0	0	0	0	0
SLS characteristic 5	Instantaneous	1	1	0.7	1	0	0	0	0	0	0

If the user selects *Characteristic combinations SLS*, second drop-down menu,he could select the combination of which data he wants to display (forces, deformations and elements checks).

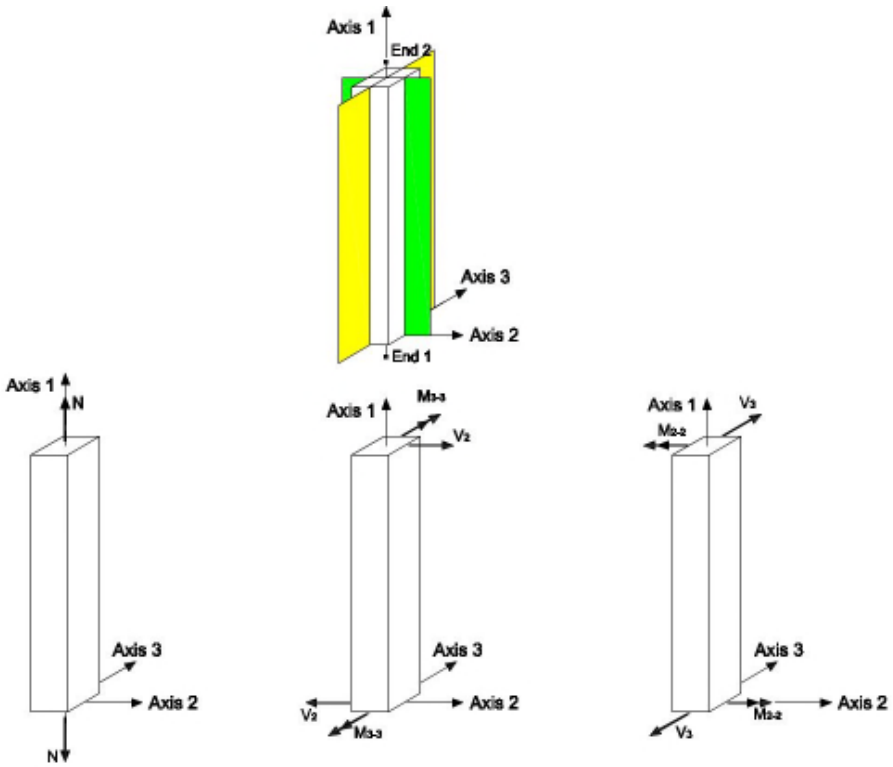
7.2 Forces/Moments and stresses

The menu allows the user to display (with different colour shades) the forces/stresses level of the selected load combinations (for the all type of structural elements). The colours bar, corresponding to the different levels of forces/stresses from the minimum value to the maximum one, is displayed on the bottom side of the screen. The table provides in detail the value of the forces, of the moments and of the stresses of the elements belonging to the same category.

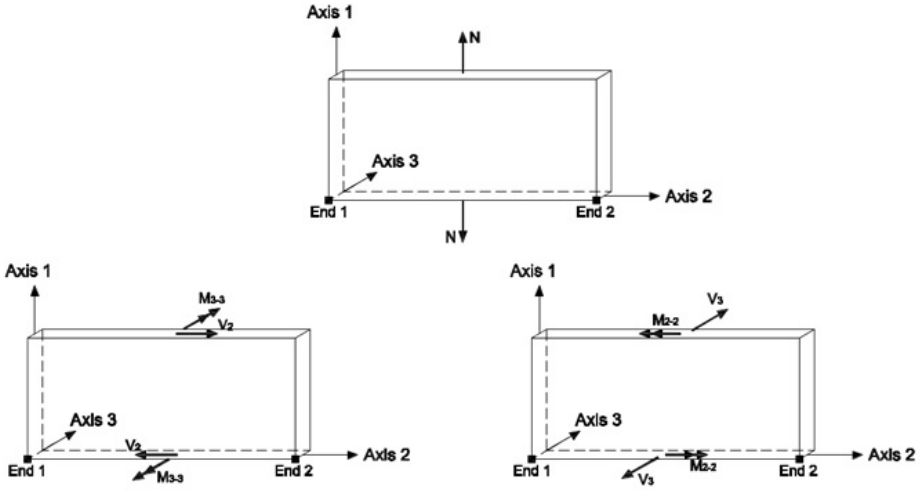
Each element (column, beam, wall, floor) of the structural model has its own coordinate system used to define the properties and the response of the element. The systems are schematically shown in the following figures.



The coordinate system used to define the Pillar (column) element is displayed in the following figure:



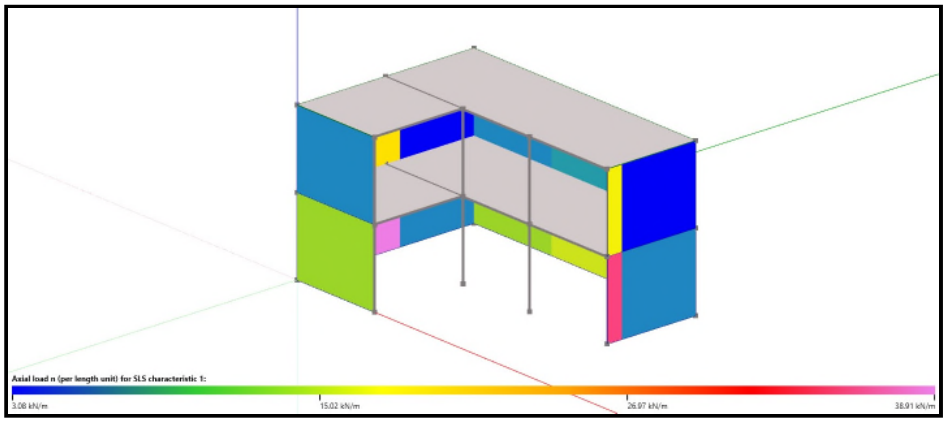
The coordinate system used to define the Wall element is displayed in the following figure:



7.2.1 Wall

The user can select, in the drop-down menu, the structural response he wants to display:

- n: axial stress per unit length;
- m_{22}: bending moment about the local axis 2 (out of plane moment) per unit length;
- v_3: shear along local axis 2 per unit length;
- m_{33}: bending moment about the local axis 2 (in plane moment) per unit length ;
- v_2: shear along local axis 2 per unit length;
- V_a: shear force in the connections;
- T_a: tension force in the connections;



The user can display the element response corresponding the selected combination in the *Combination menu*. (The combinations ULS Vertical do not produce any shear and any out of plane moment in the walls).

The table, below the model display window, provides the maximum of the forces/stresses and moments:

- N: total axial force (integration of *n* along the wall length);

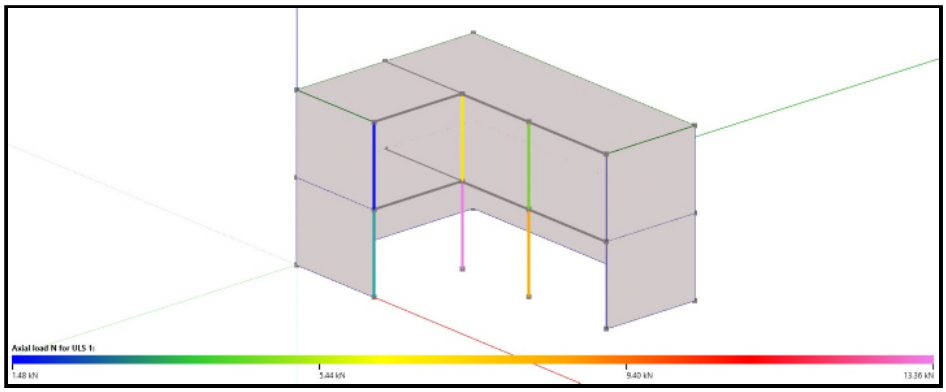
- M_{22}: bending moment about local axis 2 that is the integral of m22 along the wall length;
- V_3: total shear calculated by the integration of v3 along the wall length;
- V_a: shear force of connection;
- T_a: tension force on connection;
- d_r : interstorey drift;

Combination Horizontal ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 1.5, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Instantaneous)

Name	N [kN]	M3-3 [kNm]	V2 [kN]	Va [kN]	Ta [kN]	dr [mm]
Wall1	26.85	17.81	4.73	0.34	0.00	0.12
Wall2	40.33	82.53	22.05	1.10	0.00	0.32
Wall3	13.50	17.81	4.73	0.68	0.00	0.36
Wall4	13.33	31.64	8.53	1.22	3.38	0.66
Wall5	13.42	4.57	1.63	0.12	0.00	0.10
Wall6	20.16	20.80	7.43	0.37	0.00	0.26
Wall7	6.75	4.57	1.63	0.23	0.00	0.33
Wall8	6.66	7.74	2.77	0.40	0.00	0.56

7.2.2 Column

The user can select, in the drop-down menu, the Axial force.



The table, below the model display window, provides the axial force values of all the columns:

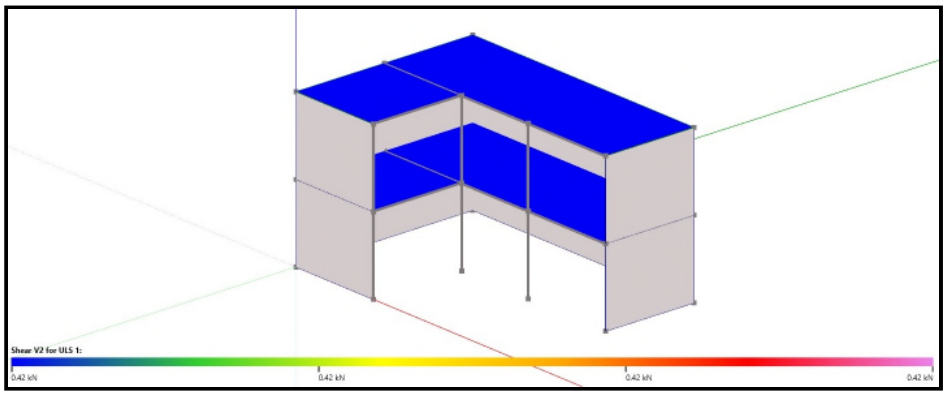
Combination ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 0, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Permanent)

Name	N [kN]
Column1	2.96
Column2	13.36
Column3	8.47
Column4	1.48
Column5	6.68
Column6	4.24

7.2.3 Floor

The user, in the drop-down menu, can select:

- M_{33}: bending moment about the local axes 3 (bending moment in plane 1-2);
- V_{22}: shear force along local axes 2(Shear 2);
- W_{inst}: maximum of the instantaneous deformation;
- W_{fin}: maximum of the final deformation;



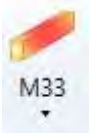
The user can select, in the drop-down menu, the forces/moments or the deformation corresponding to the selected *Combination*.

The table, below the model display window, provides the maximum value of the forces and deformations.

Combination G1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 0, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 |Duration Permanent|

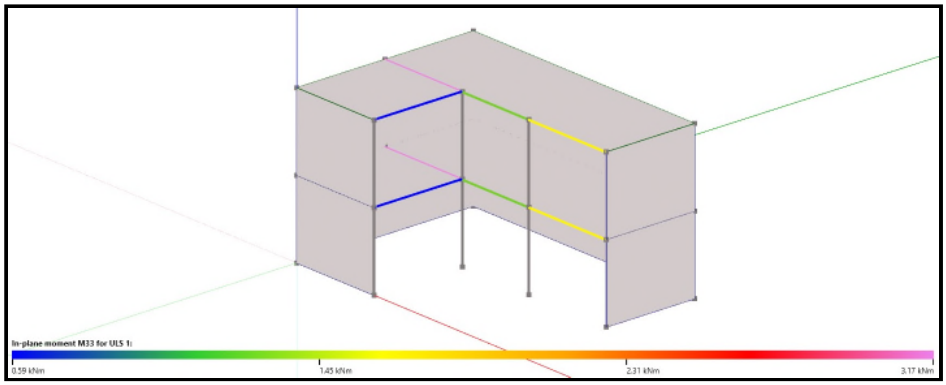
Name	M33 [kNm]	V2 [kN]	Instantaneous deformation [mm]
Floor1	0.29	0.42	0.32
Floor2	0.29	0.42	0.32

7.2.4 Beam



The user, in the drop-down menu, can select the following responses he wants to display:

- M_{33}: bending moment about the local axes 3 (bending moment in plane 1-2);
- V_{22}: shear force along local axes 2(Shear 2);
- W_{inst}: maximum of the instantaneous deformation;
- W_{fin}: maximum of the final deformation;



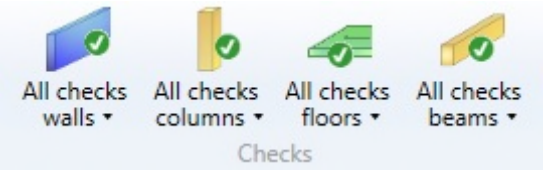
The user can select, in the drop-down menu, the forces/moments or the deformation corresponding to the selected *Combination*.

The table below the model display window provides the maximum value of the forces and deformations.

Combination ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 0, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Permanent)

Name	M33 [kNm]	V2 [kN]
Beam1	1.62	1.84
Beam2	1.19	1.57
Beam3	0.59	0.67
Beam4	1.62	1.84
Beam5	1.19	1.57
Beam6	0.59	0.67
Beam7	3.17	3.60
Beam8	3.17	3.60

7.3 Checks



The table below the model display window provides the elements checks in percentage of the maximum value of the strength. For instance: Bending equal to 68% involves the test passed; vice versa, a Bending moment equal to 103% involves the bending failure of the element.

7.3.1 Wall

The user can select one of the following checks:

- Stability;
- Compression of the wall supports;
- Shear;
- Shear on connections;
- Tension force on the connections;
- Displacement;

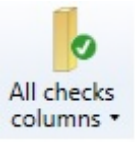
The user can select, in the drop-down menu, the checks he wants to display. The software considers the significant checks depending on the selected combination.

The table below the model display window provides the elements checks in percentage.

Combination Horizontal ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 1.5, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Instantaneous)

Name	N [kN]	M3-3 [kNm]	Stability		Support comp		Shear		Connection for shear force		Connection for tensile force	
Wall1	26.85	17.81	✓	2%	✓	3%	✓	1%	✓	2%	✓	0%
Wall2	40.33	82.53	✓	1%	✓	1%	✓	2%	✓	7%	✓	0%
Wall3	13.50	17.81	✓	2%	✓	2%	✓	1%	✓	5%	✓	0%
Wall4	13.33	31.64	✓	1%	✓	1%	✓	2%	✓	8%	✓	16%
Wall5	13.42	4.57	✓	1%	✓	2%	✓	0%	✓	2%	✓	0%
Wall6	20.16	20.80	✓	1%	✓	1%	✓	1%	✓	6%	✓	0%
Wall7	6.75	4.57	✓	1%	✓	1%	✓	0%	✓	3%	✓	0%
Wall8	6.66	7.74	✓	0%	✓	1%	✓	1%	✓	6%	✓	0%

7.3.2 Column

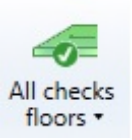


The user can choose to display the stability check.

Combination ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 0, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Permanent)

Name	N [kN]	Stability	
Column1	2.96	✓	1%
Column2	13.36	✓	4%
Column3	8.47	✓	3%
Column4	1.48	✓	0%
Column5	6.68	✓	2%
Column6	4.24	✓	1%

7.3.3 Floor



The user can select one of the following checks:

- Shear;
- Bending;
- W_{inst}: maximum of the instantaneous deformation;
- W_{fin}: maximum of the final deformation;

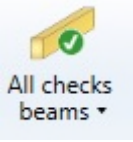
The user can select, in the drop-down menu, the checks he wants to display. The software considers the significant checks depending on the selected combination.

The table below the model display window provides the elements checks in percentage.

Combination ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 0, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Permanent)

Name	M33 [kNm]	V2 [kN]	Instantaneous deformation [mm]	Bending		Shear	
Floor1	0.29	0.42	0.00	✓	3%	✓	2%
Floor2	0.29	0.42	0.00	✓	3%	✓	2%

7.3.4 Beam



The user can select one of the following checks:

- Shear;
- Bending;
- W_{inst}: maximum of the instantaneous deformation;
- W_{fin}: maximum of the final deformation;

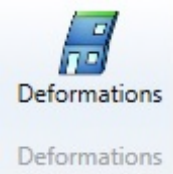
The user can select, in the drop-down menu, the checks he wants to display. The software considers the significant checks depending on the selected combination.

The table below the model display window provides the elements checks in percentage.

Combination ULS 1: G1: 1, G2: 0, Variable cat.A: 0, Orthogonal wind: 0, Wind X: 0, Wind Y: 0, Seismic ULS X: 0, Seismic ULS Y: 0, Seismic SLS X: 0, Seismic SLS Y: 0 (Duration Permanent)

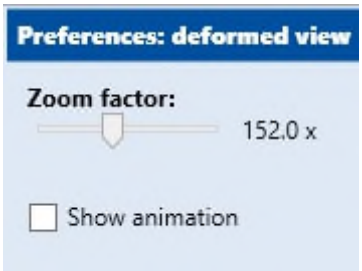
Name	M33 [kNm]	V2 [kN]
Beam1	1.62	1.84
Beam2	1.19	1.57
Beam3	0.59	0.67
Beam4	1.62	1.84
Beam5	1.19	1.57
Beam6	0.59	0.67
Beam7	3.17	3.60
Beam8	3.17	3.60

7.4 Deformations



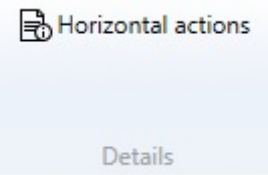
The command *Deformations* allows the user to display the model deformations due to horizontal loads. This command is available only in the case of wind and seismic loads.

Note: the user can display the deformations due to the vertical loads, selecting the element (floor or beam) he wants to examine and the desired combination and then clicking the *Details* button.



It is also possible to increase and to reduce the zoom factor with the slider shown in the previous figure. To deeply understand the structural deformation, the user can also run the model animation.

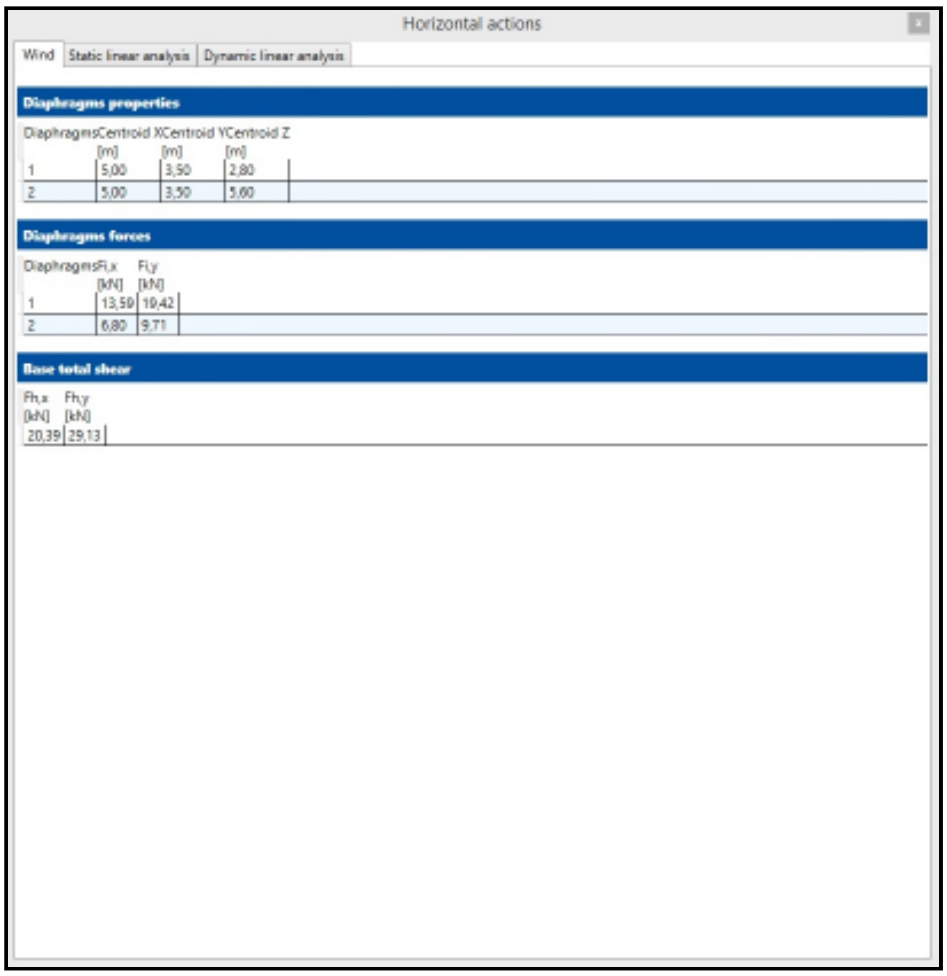
7.5 Details



7.5.1 Details

The command *Details* provides the access to the summary tables which report the horizontal loads (wind and seismic loads) and the most important parameters used to calculate them.

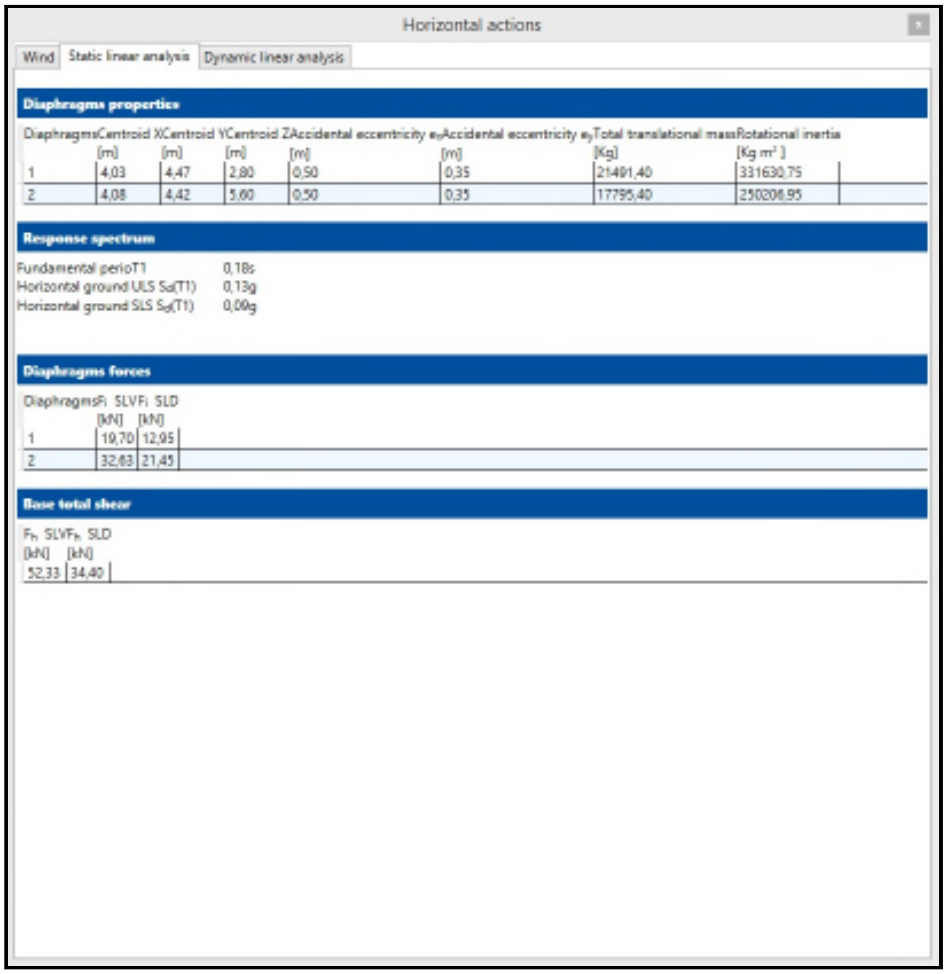
7.5.1.1 Wind



The tables provides, for each structure diaphragms:

- Centroid X: the centroid X coordinate of the entire surface of the walls perpendicular to the Y direction;
- Centroid Y: the centroid Y coordinate of the entire surface of the walls perpendicular to the X direction;
- Centroid Z: height above foundation of the diaphragm;
- $F_{i,X}$: the force acting along X direction on the i-th diaphragm;
- $F_{i,y}$: the force acting along Y direction on the i-th diaphragm;
- $F_{h,x}$: the total force, at the base, acting along X direction;
- $F_{h,x}$: the total force, at the base, acting along X direction;

7.5.1.2 Static linear analysis



The table diaphragm properties provide the following data;

- Centroid X: the centroid X coordinate of the entire surface of the walls perpendicular to the Y direction;
- Centroid Y: the centroid Y coordinate of the entire surface of the walls perpendicular to the X direction;
- Centroid Z: height above foundation of the diaphragm;
- e_x, e_y: the accidental eccentricity in X direction and in Y direction;
- total translational mass: the sum of all the loads and masses acting on each diaphragm;
- the rotational inertia;

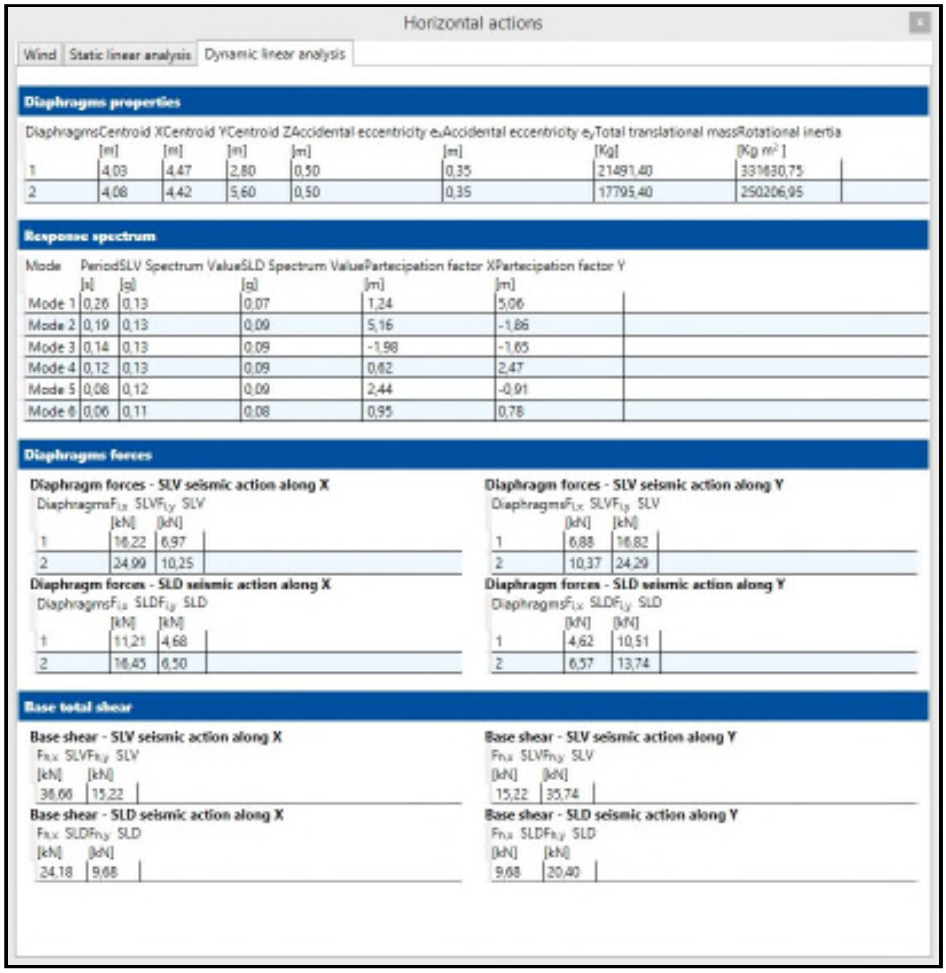
The table *Response spectrum* provides the following data:

- the fundamental period T_1 given by the equation $T_1 = C \cdot H^{3/4}$ where C = 0,5;
- the SLV response spectrum value corresponding to the fundamental period T_1;
- the SLD response spectrum value corresponding to the fundamental period T_1;

The other two tables provide:

- $F_{i,x}$: the force acting along X direction on the i-th diaphragm (in case of SLV and SLD);
- $F_{i,y}$: the force acting along Y direction on the i-th diaphragm (in case of SLV and SLD);
- $F_{h,x}$: the total force, at the base, acting along X direction (in case of SLV and SLD);
- $F_{h,x}$: the total force, at the base, acting along X direction (in case of SLV and SLD);

7.5.1.2 Dynamic linear analysis



The table *Diaphragm properties* provides the following data;

- Centroid X: the centroid X coordinate of the entire surface of the walls perpendicular to the Y direction;
- Centroid Y: the centroid Y coordinate of the entire surface of the walls perpendicular to the X direction;
- Centroid Z: height above foundation of the diaphragm;
- e_x, e_y: the accidental eccentricity in X direction and in Y direction;
- total translational mass: the sum of all the loads and masses acting on each diaphragm;
- the rotational inertia;

The table *Response spectrum* provides the following data:

- the mode number and the corresponding period T_i;
- the SLV and the SLD response spectrum value corresponding to the each period T_i;
- the partecipation factor along X and Y;

The *Diaphgrams forces* table provides:

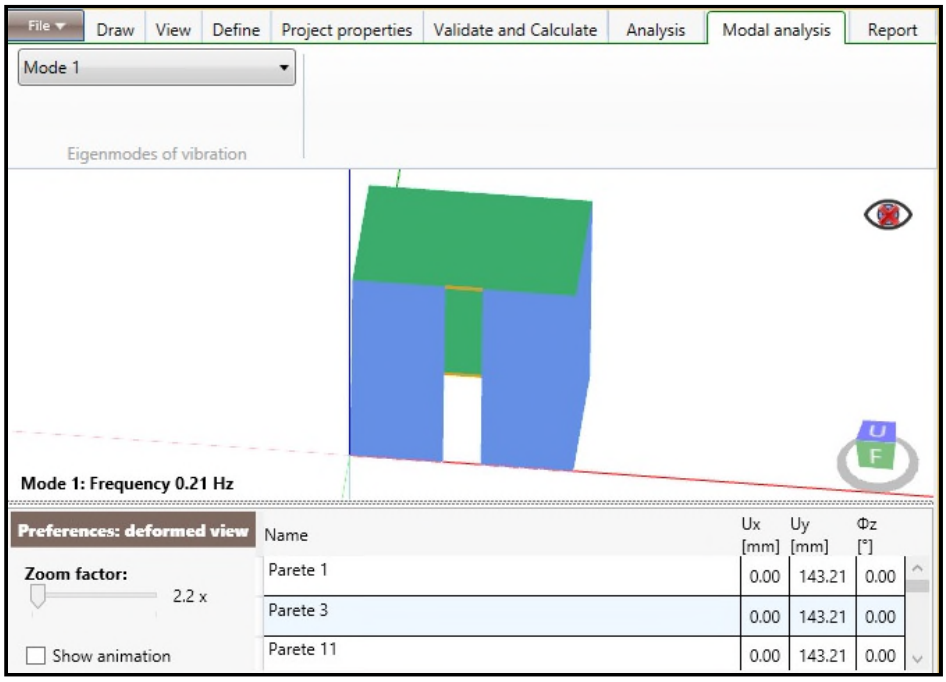
- $F_{i,X}$: the force acting along X direction on the i-th diaphragm (in case of SLV and SLD and for a seismic action along X and along Y);

- $F_{i,y}$: the force acting along Y direction on the i-th diaphragm (in case of SLV and SLD and for a seismic action along X and along Y);

The *Base total shear* table provides:

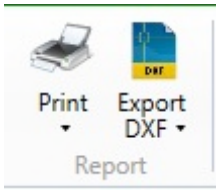
- $F_{h,x}$: the total force, at the base, acting along X direction in case of SLV and SLD and for a seismic action along X and along Y);
- $F_{h,x}$: the total force, at the base, acting along X direction in case of SLV and SLD and for a seismic action along X and along Y);

7.6 Modal Analysis

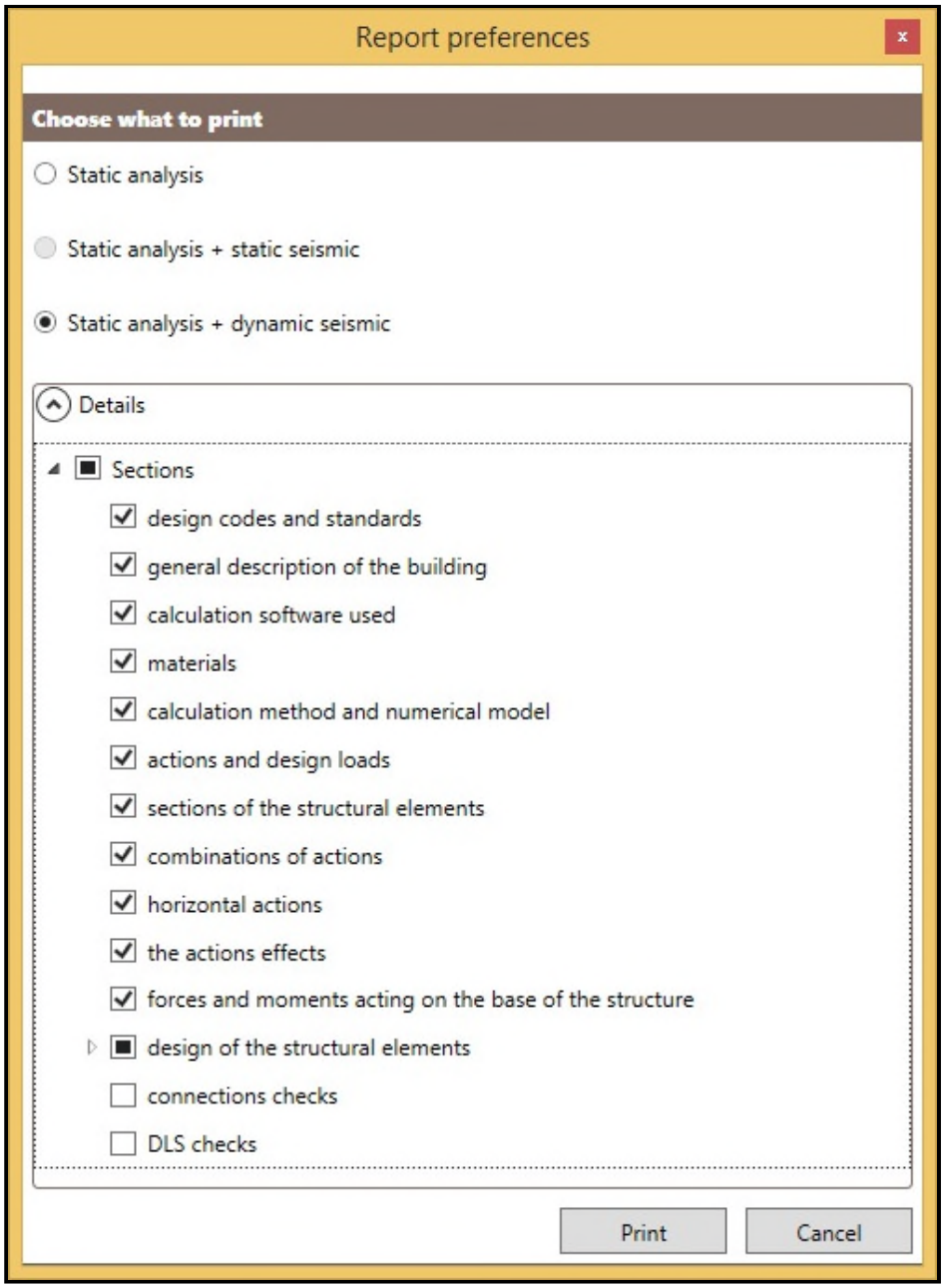


The command *Modal Analysis* allows the user to select a vibration mode to display the respective frequency and the modal deformations.

7.7 Report



7.7.1 Print



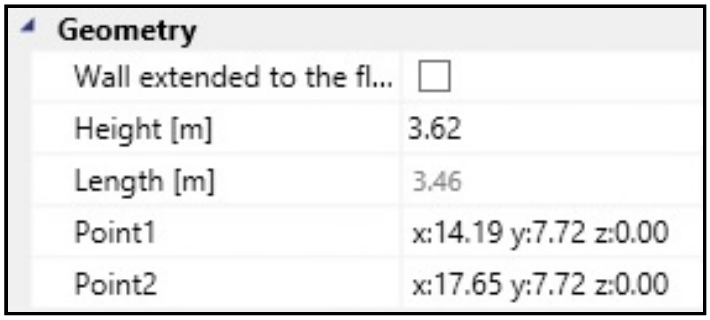
If the user selects *Print*, he could select the report preferences.

8. Elements properties

If the user selects a model element, both on the stage of model design and on the analysis one, a window, on the right side of the screen will appear. The window provides the definition data of the element: geometry, mechanical properties and loads. On the stage of ***model definition***, the user can modify the element properties in the drop-down menu. On the ***analysis*** stage, the box provides the loads/stresses, the elements displacements the design data.

8.1 Wall

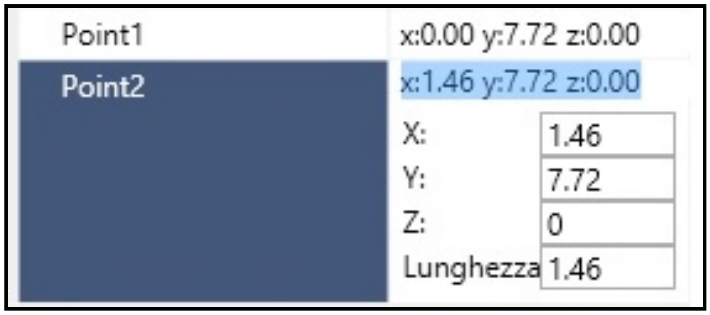
8.1.1 Geometry



The user can:

- extend the selected wall to the overhanging floor or roof from the checkbox;
- modify the element height;
- modify the element length;

To modify the wall length, the user must select the coordinates of one of the end point (Point 1, Point 2). In this way the user can assign new values to the point coordinate or to directly modify the elements length (in the dialog box). If the user modify the length, the software will *lengthen* or *shortens* from the considered point.



8.1.2 Info

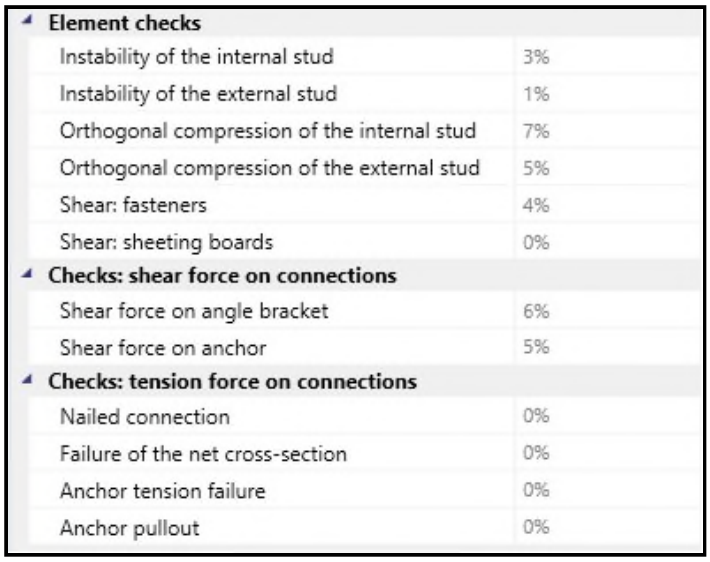
In the ***Info*** box, the user can:

- Modify the wall typology;
- Modify the loads;

- Modify the connection;

The ***Info*** box provides the following informations:

8.1.2.1 Frame Wall



Element checks	
Instability of the internal stud	3%
Instability of the external stud	1%
Orthogonal compression of the internal stud	7%
Orthogonal compression of the external stud	5%
Shear: fasteners	4%
Shear: sheeting boards	0%
Checks: shear force on connections	
Shear force on angle bracket	6%
Shear force on anchor	5%
Checks: tension force on connections	
Nailed connection	0%
Failure of the net cross-section	0%
Anchor tension failure	0%
Anchor pullout	0%

Wall typology

- Typology: the typology of the wall;
- Material: the structural elements material (frames and studs);
- External stud: section area of the external stud;
- Internal stud: section area of the internal stud;
- Sheeting panel side 1: the panel material and type;
- Sheeting panel side 2: the panel material and type;

Loads

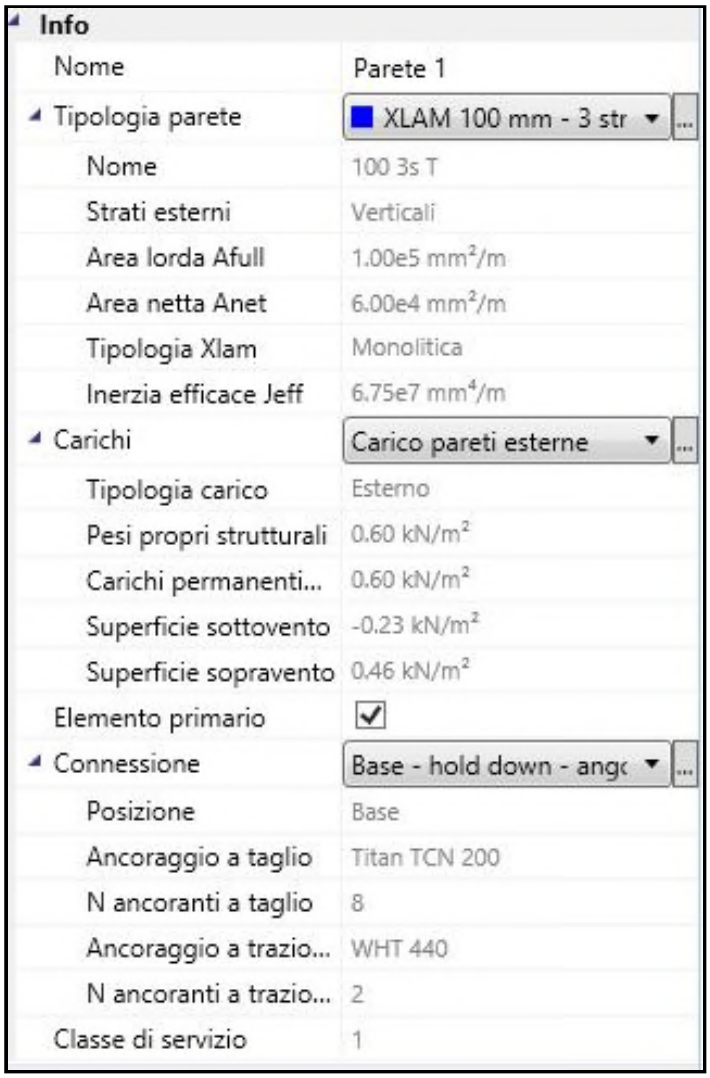
- Load typology: load applied on the internal walls or on the external ones;
- Self-weight load: the self-weight load, per unit area, of the walls;
- Non structural permanent load: load, per unit area, of the surface finishing;
- Leeward surface*: wind pressure on the non directly exposed wall;
- Windward surface*: wind pressure on the non directly exposed wall;
- Primary element: it states the element hierarchy for the horizontal actions;

Connection**

- Position: connection position (ground connection or upper level connection);
- Shear connection: shear connection typology;
- N of shear connection: number of shear connections;
- Tension connection: Tension connection typology;
- N of hold-down/tie-down: total number of tension connections;
- Service class: service class depending on the wood humidity.

* only for external walls; **only in the case of primary wall.

8.1.2.2 CLT wall;



Wall typology

- Typology: wall typology;
- Name: wall definition;
- External layers: orientation of external layers;
- Gross area A_{full}: gross area of the cross section;
- Net area A_{net}: net area of the vertical layers sections;
- CLT typology: wall typology which could be monolithic or jointed;
- Effective moment of inertia J_{eff}: the inertia is obtained from the vertical layer total inertia assumed to be connected by the transversal panels;

Load

- Load typology: load applied on the internal walls or on the external ones;
- Self-weight load: the self-weight load, per unit area, of the walls;
- Non structural permanent load: load, per unit area, of the surface finishing;
- Leeward surface*: wind pressure on the non directly exposed wall;

- Windward surface*: wind pressure on the non directly exposed wall;
- Primary element: it states the element hierarchy for the horizontal actions;

Connection**

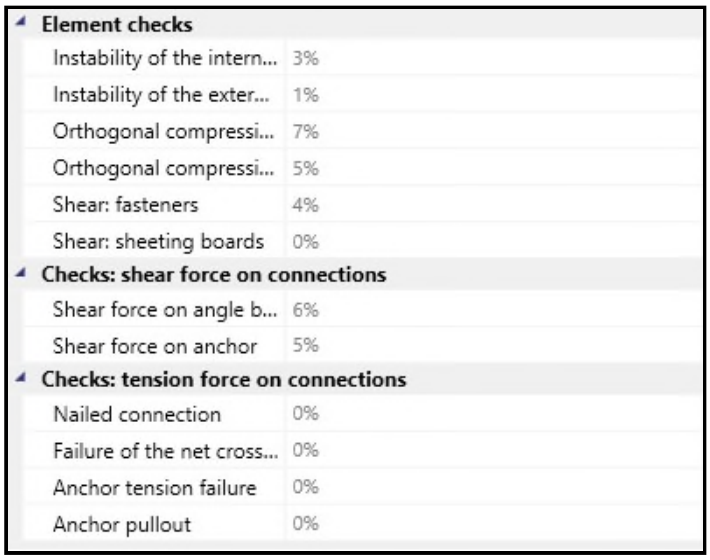
- Position: connection position (ground connection or upper level connection);
- Shear connection: shear connection typology;
- N of shear connection: number of shear connection;
- Tension connection: Tension connection typology;
- N of hold-down/tie-down: total number of tension connection;
- Service class: service class depending on the wood humidity.

* only for external walls; **only in the case of primary wall.

8.1.3 Element checks

The box ***Element checks*** provides the following data:

8.1.3.1 Frame wall



Element checks

- Instability of the internal stud: check of the instability of the internal stud;
- Instability of the external stud: check of the instability of the external stud;
- Orthogonal compression of the internal stud: check of the orthogonal compression of the internal stud;
- Orthogonal compression of the external stud: check of the orthogonal compression of the internal stud;
- Shear fasteners: check of the connection between the sheeting panels and the frame;
- Shear panels: check of the sheeting panels shear;

Check of the shear force connections:

- Shear force on angle bracket: check of the shear force on angle bracket;
- Shear force on the anchor: check of the shear force on the anchor;

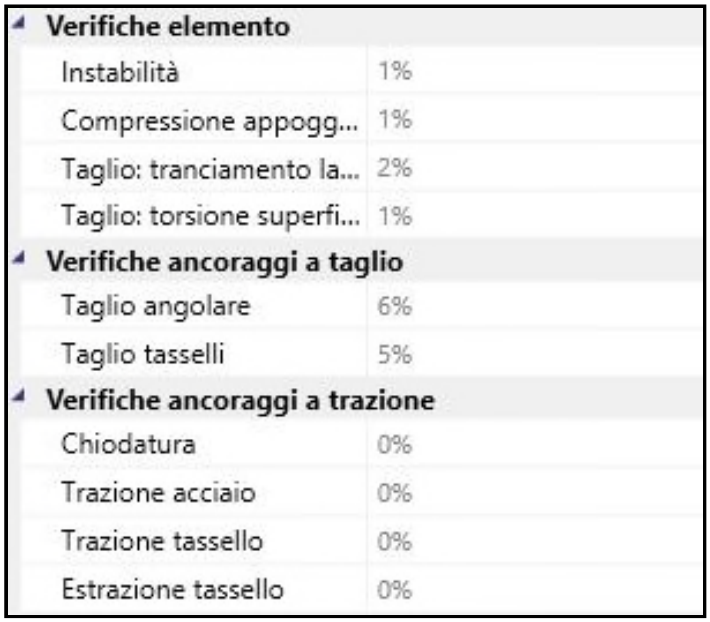
- Nailed connection*: check of the preforated plate nailing (tension connection);
- Failure of the net cross-section*: check of the failure of the preforated plate;

Checks of the tension connection:

- Nailing*: check of the nailed connection resistance;
- Failure of the net cross-section: check of the hold-down tensile strength (steel ultimate tensile strength);
- Anchor tension failure: check of the anchor tension failure;
- Anchor pullout: check of the pull-out resistance;
- Yielding of the gross-section*: check of the steel plastic resistance (net cross-section of the plate);
- Tension failure of the net cross-section:* check of the ultimate tension resistance of the plate net cross-section;

* only in the case of upper level walls;

8.1.3.2 CLT wall



Element checks

- Instability: check of the element instability;
- Support compression: check of the support compression;
- Shear perpendicular to grain: check of shear bearing in the boards ;
- Torsional failure: check the shear failure due to torsion-like mechanism in the gluing interfaces;

Check of the shear force connections:

- Shear force on angle bracket: check of the shear force on angle bracket;
- Shear force on the anchor: check of the shear force on the anchor;
- Nailed connection*: check of the preforated plate nailing (tension connection);
- Failure of the net cross-section*: check of the failure of the preforated plate;

Checks of the tension connection:

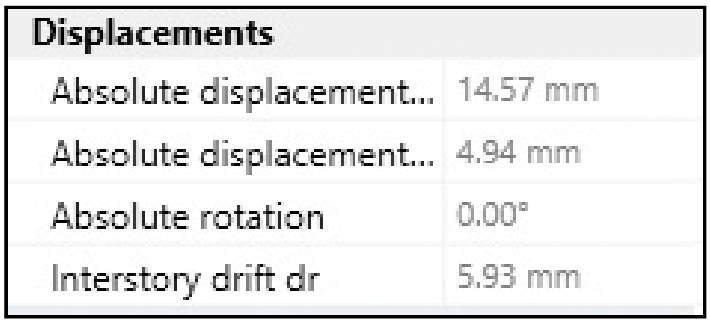
- Nailing*: check of the nailed connection resistance;
- Failure of the net cross-section: check of the hold-down tensile strength (steel ultimate tensile strength);
- Anchor tension failure: check of the anchor tension failure;
- Anchor pullout: check of the pull-out resistance;
- Yielding of the gross-section*: check of the steel plastic resistance (net cross-section of the plate);
- Tension failure of the net cross-section:* check of the ultimate tension resistance of the plate net cross-section;

* only in the case of upper level walls;

8.1.3 Forces

The forces meaning is better explaind in the part 7: **Menu Tools*Analysis.***

8.1.4 Displacement



The data provided in the box *Displacement* do not need to be explained.

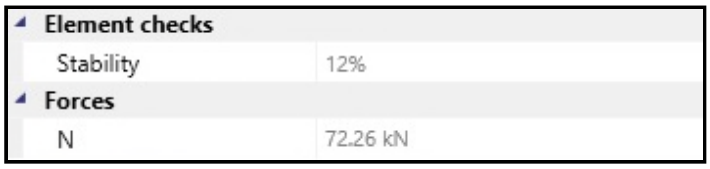
8.2 Column

8.2.1 Geometry/Info

In the box **Geometry** the user can modify the position and the height of the selected Column. In the dialog box **Info** the user can modify the cross-section area. The following informations are reported:

- Name: name of the element;
- Service class: the service class depending on the timber humidity;

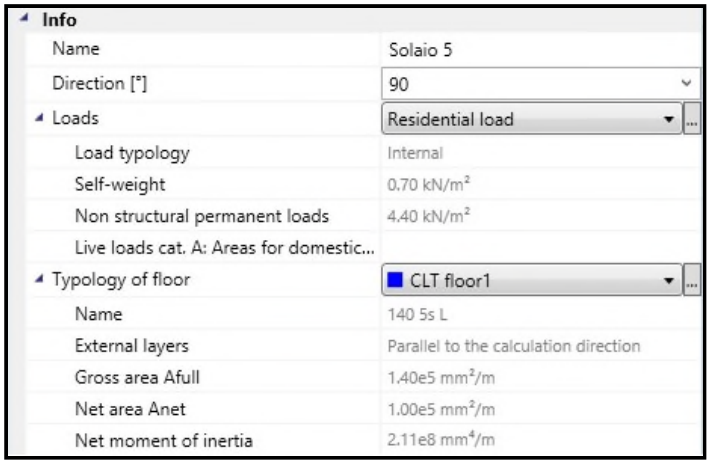
8.2.2 Checks/forces



The dialog box **Element checks and Forces** provides the checks of column instability and the value of the axial force.

8.3 Floor (Roof)

8.3.1 Info



In the dialog box **Info,** , by the drop-down menu, the user can modify:

- the floor direction (of the beams or of the external layers in the case of CLT floors);
- the applied loads;
- the typology fo the floor;

The box provides also the following informations:

Info

- Name: name of the element;

Loads

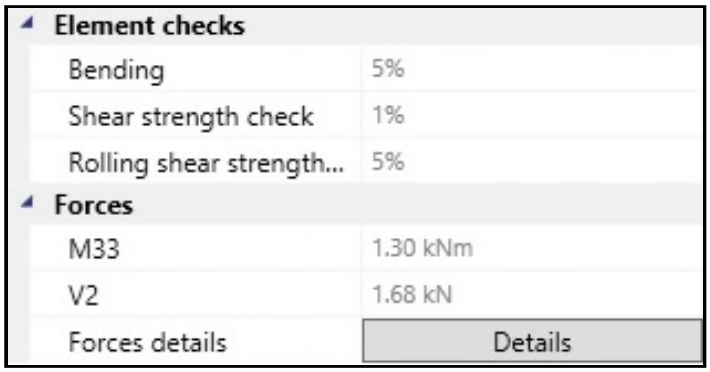
- Load typology (internal or external load);
- Self weight of the floor;
- Non structural permanent load of the floosìr;
- Live loads category;

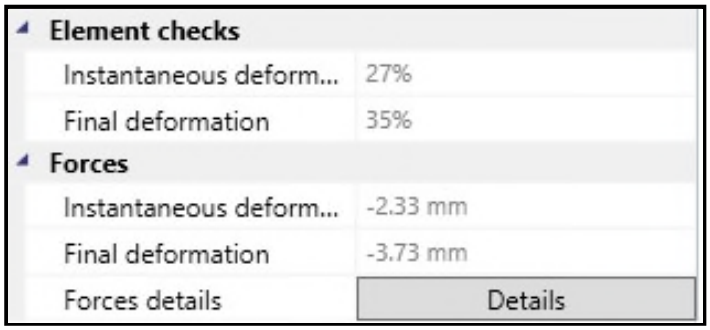
Typology of floor

- External layers*: orientation of the panel external layers (parallel or orthogonal to the calculation direction);
- Area A: area of the cross-section;
- Gross area A_{full} : gross area of the cross-section*;
- Net area A_{net}: cross sectional area of the wall portion considered in the verification *;
- Moment of Inertia J_z**;
- Moment of Inertia J_y:
- Net moment of Inertia*

* only in the case of CLT floor; ** only in the case of solid wood floor;

8.3.2 Checks/Forces

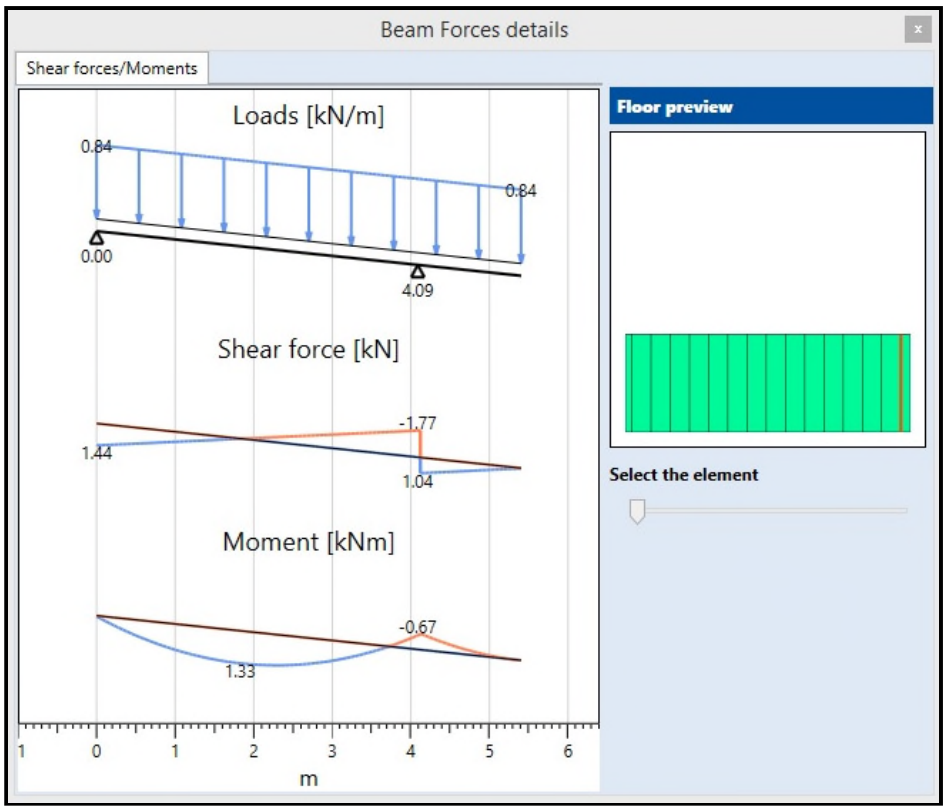




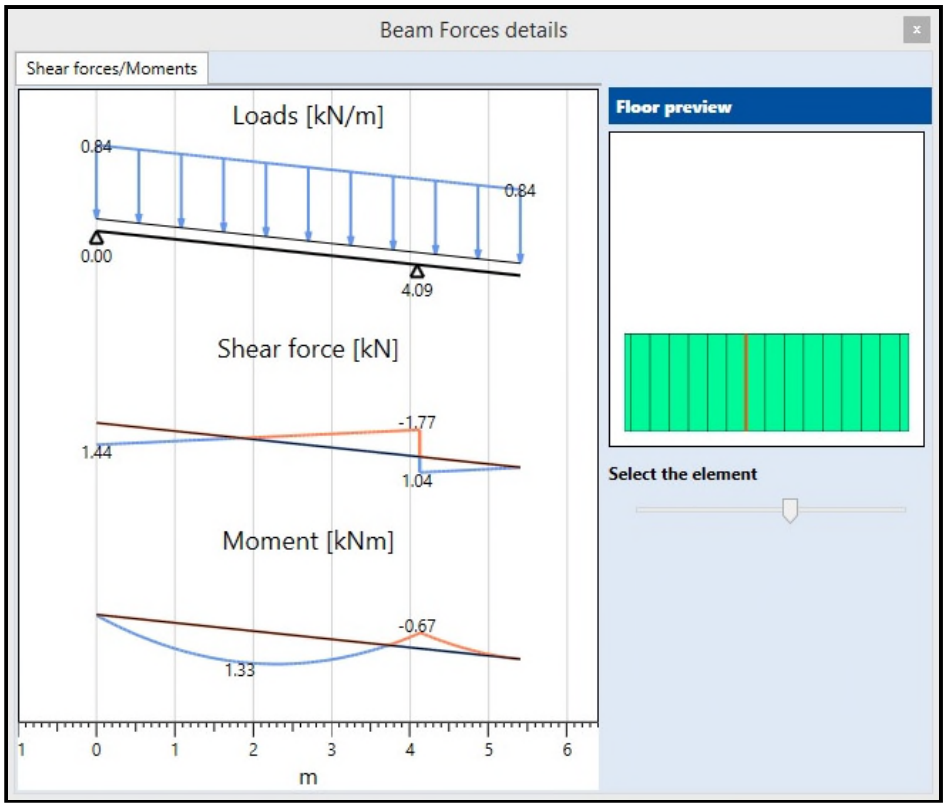
The meaning of the data in the box **Checks and forces** do not need to be explained. The only term that changes, depending on the floor type, is the shear one: in the case of CLT floor the software check both shear strength and rolling shear strength.

8.3.3 Details (Solid Wood Floor/Joists Floor)

If the user selects the command *Details* it will appear a window where the static schemes and the enveloped diagrams of forces (or deformations) are displayed.



The user, moving the cursor in the floor preview, can select the element whose diagram is desired to be displayed:

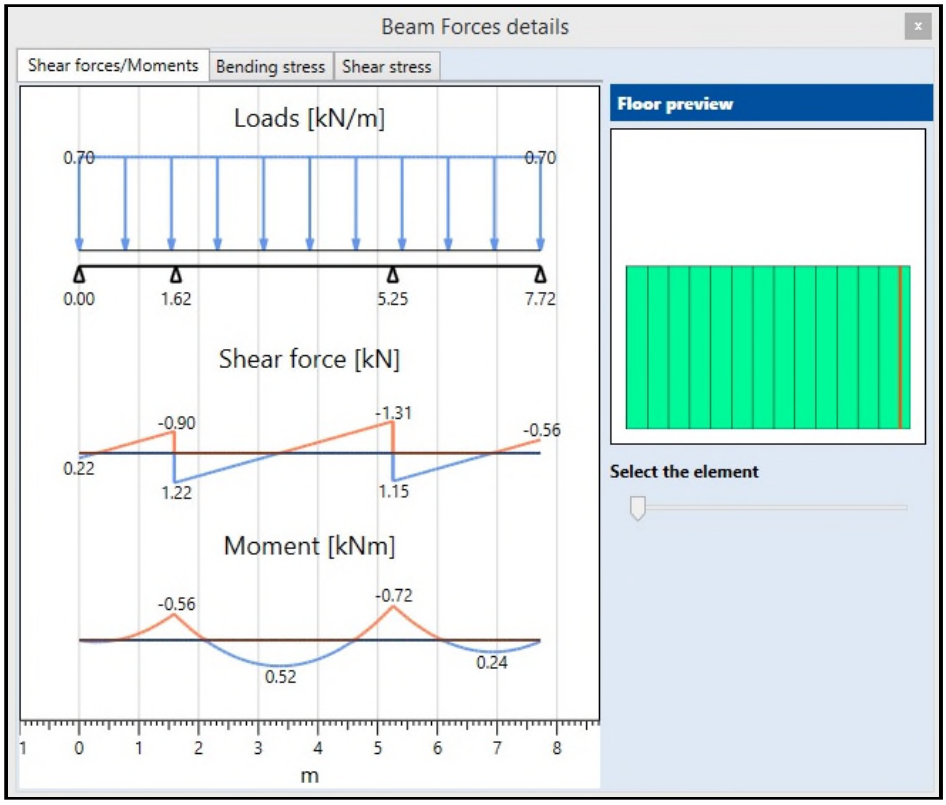


NB: If no values are displayed, these are lower than 0.01 unit;

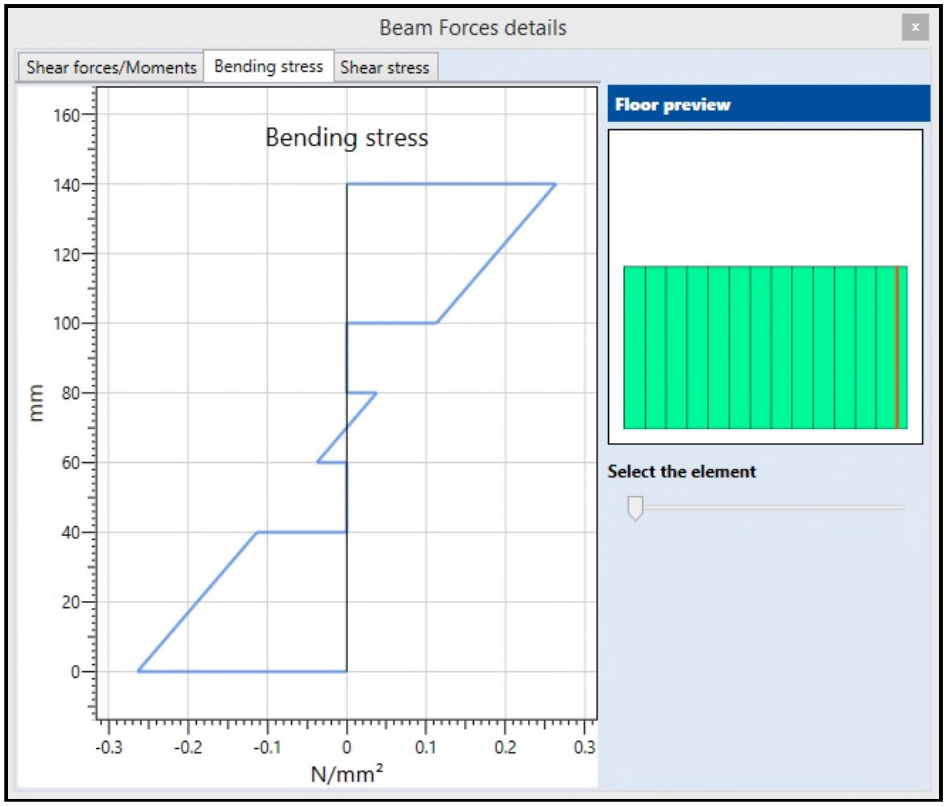
8.3.3 Details (CLT Floor)

If the user selects the command *Details* it will appear a window where the following data displayed.

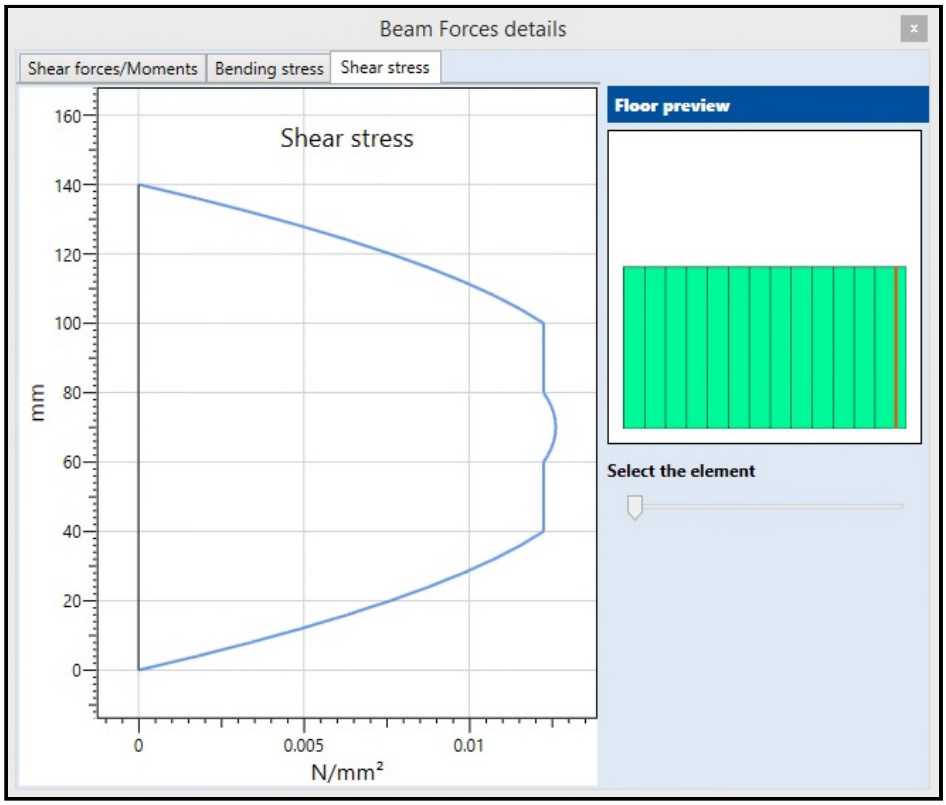
Enveloped diagrams of forces (or deformations)



The bending stresses



The shear stresses



NB: If no values are displayed, these are lower than 0.01 unit;

8.4 Beam

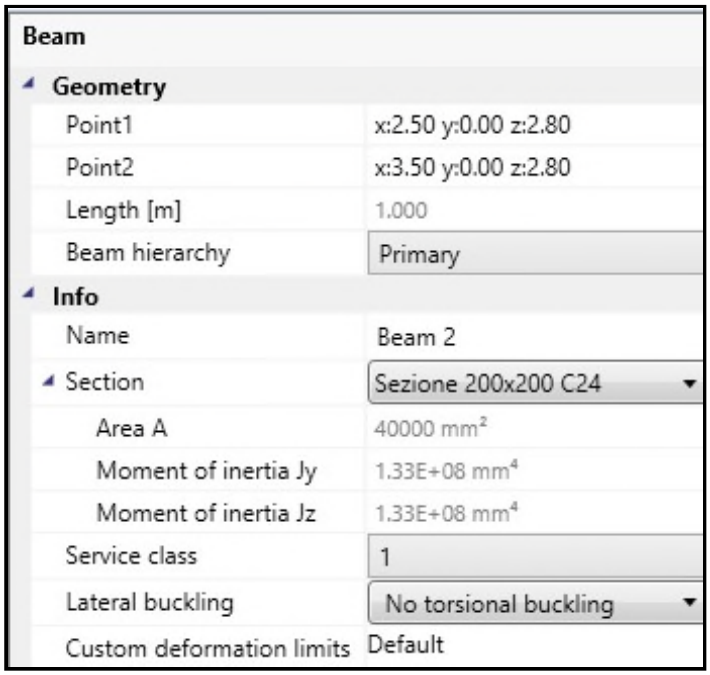
8.4.1 Geometry/Info

In the box **Info,** , by the drop-down menu, the user can modify:

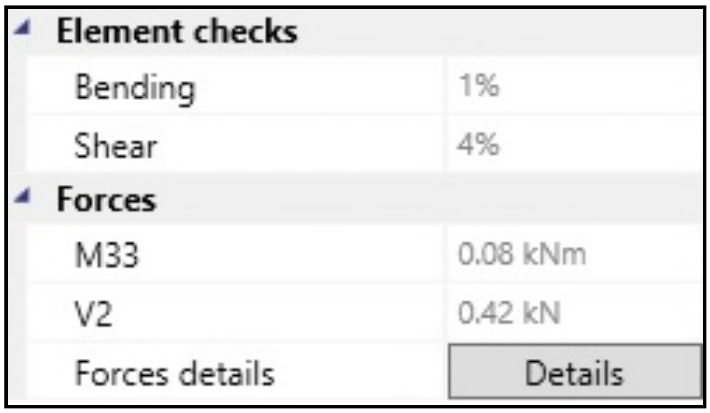
- the length (in the same way as done for the wall elements);
- Cross-section typology;
- Beam hierarchy (Primary element, 2nd, 3th);

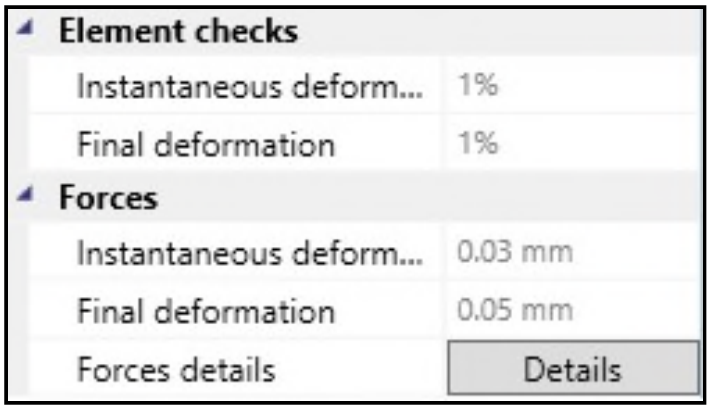
The box also provides the following informations:

- Name: the element name;
- Service class, depending on the timber humidity;
- Lateral buckling;
- Custom deformation limits;



8.4.2 Checks/Forces

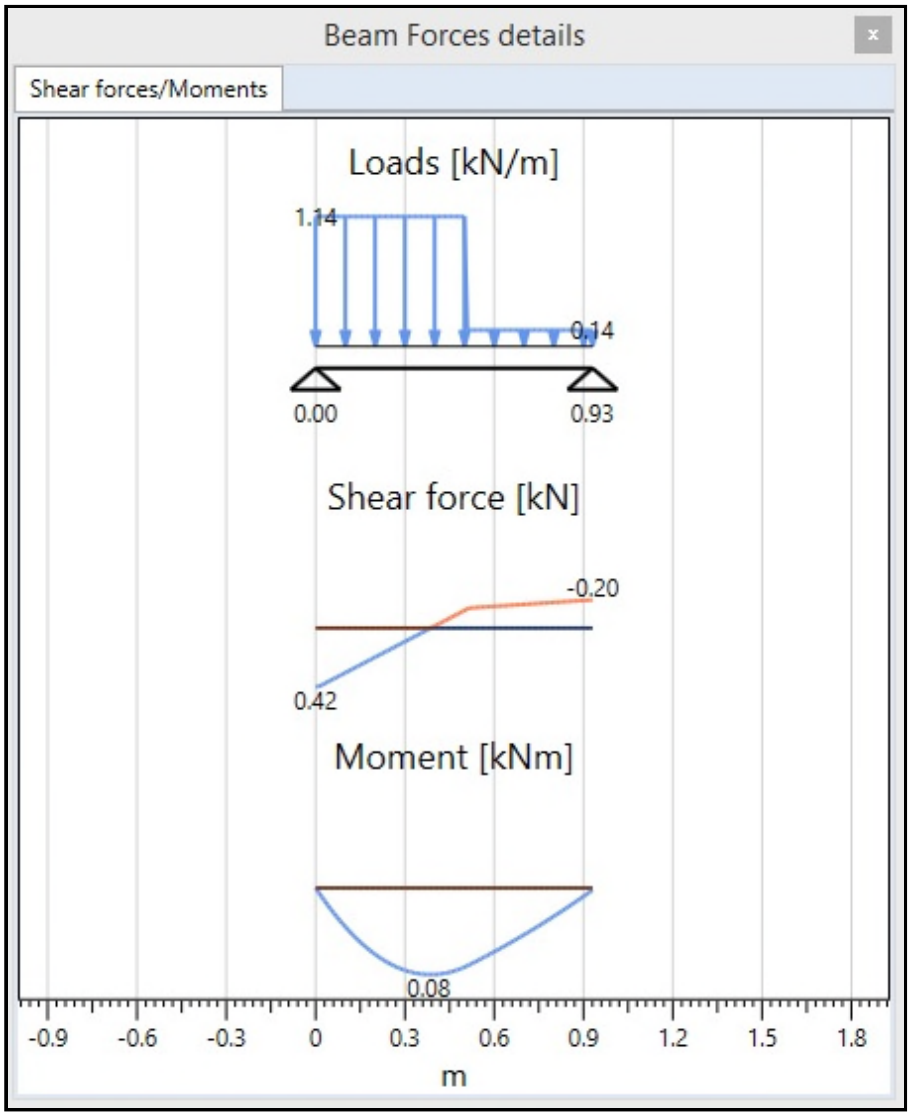




The meaning of the data do not need to be explained.

8.4.3 Details

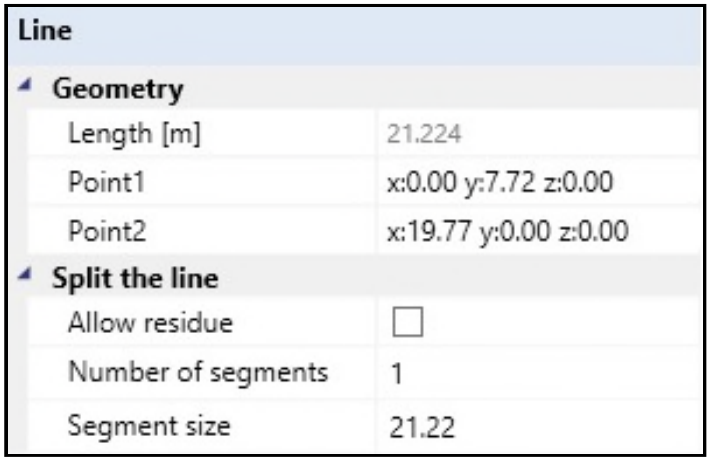
If the user selects the command *Details* it will appear a window where the static schemes and the enveloped diagrams of forces (or deformations) are displayed.



NB: If no values are displayed, these are lower than 0.01 unit;

8.5 Line and Grid

8.5.1 Line

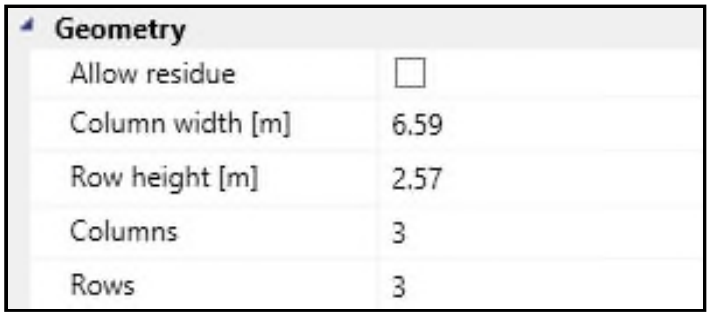


The user can:

- edit the length and the position changing the ends coordinates (see: **8.1.1 *Wall Geometry***).
- choose the number of segments that divide the line;

The option ***allow residue***, allows the user to specify the length of the dividing line segments. If the segment size does not allow to perfectly divide the line, the last segment size will be equal to the residue.

8.5.2 Grid



The user can

- define the number of columns and rows of the grid;
- define the column width and the row height;

In the same way as the line definition (see: **8.5.1**), the user can select the option *Allow residue.*