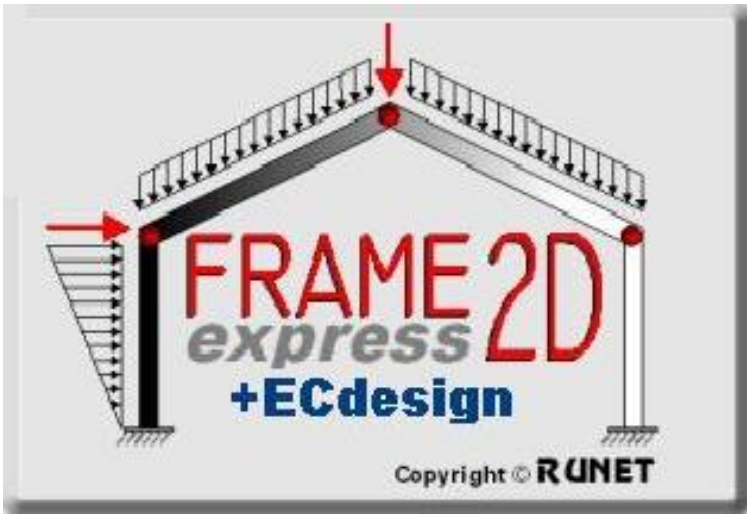
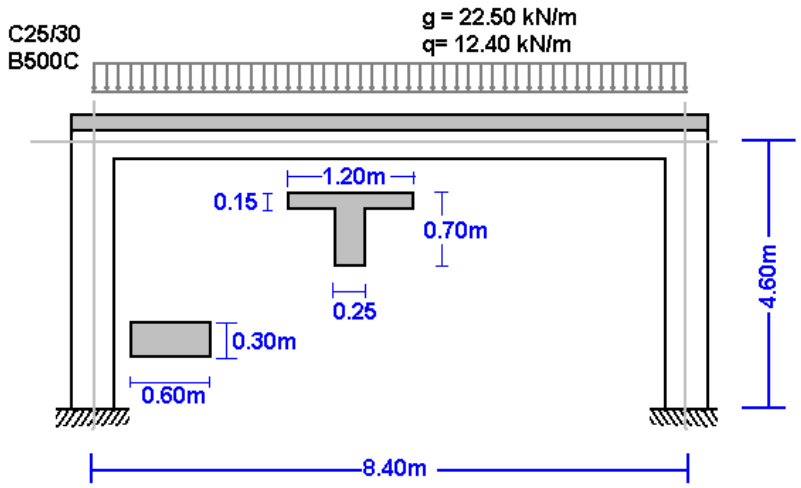


Examples of New version for Designing members of Reinforced Concrete, Steel or Timber according to Eurocode 2, Eurocode 3 and Eurocode 5

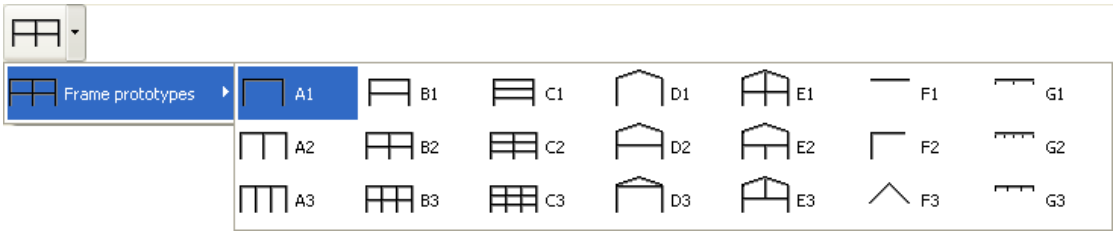
1. Examples

1.1 Example 1

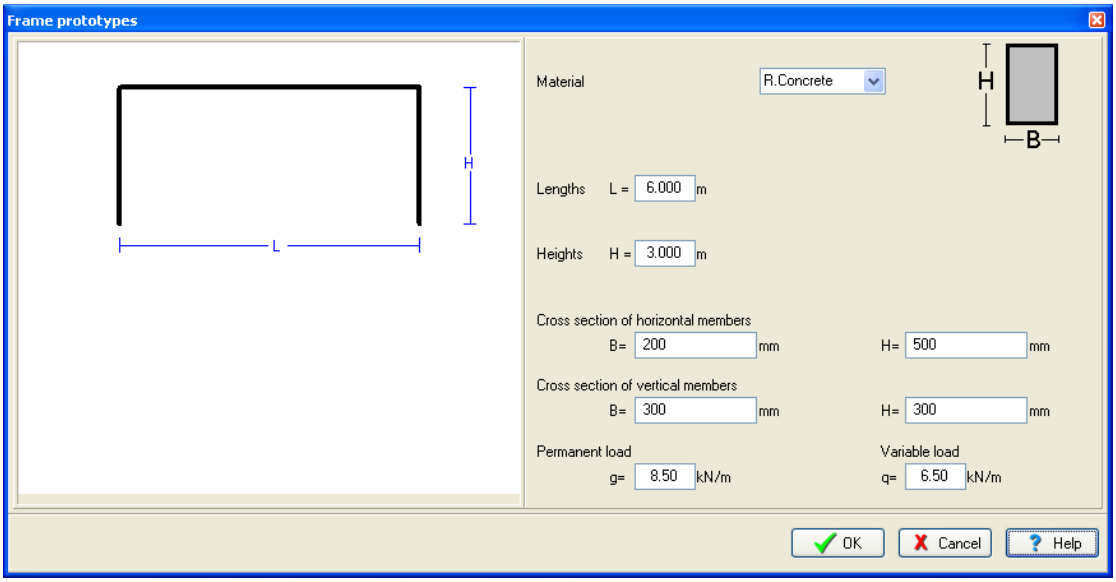
Frame of reinforced concrete 8.40 m x 4.60 m C25/30 B500C



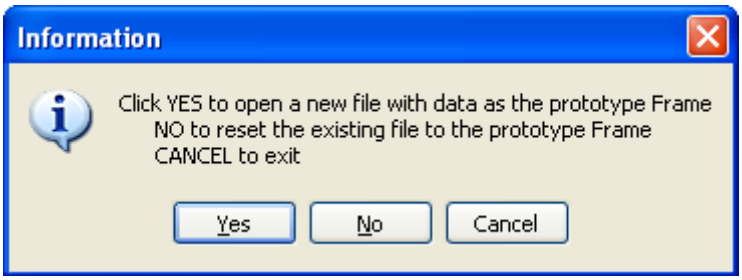
Select a frame from File/Frame prototypes:



Select material R. Concrete and give the basic dimensions, sections and loads.
You can always change and adjust these values afterwards.

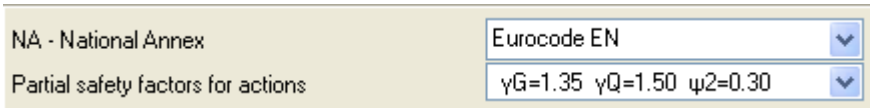


Click Yes and give the file name.



Then check and adjust the rest of the values for the structural model.

Select National Annex of your region and partial safety factors.
Usual values for partial safety factors ULS (ultimate limit state) γG=1.35, γQ=1.50 and SLS (serviceability limit state) γG=1.00, γQ=1.00.



Check the drawing of the structure.

- Nodes. Coordinate system at lower left point.
  Axis x from left to right, axis y from down up.
  The numbering of the nodal points is displayed on the drawing of the structure.

nodes | supports | nodal loads | nodal masses

node	x [m]	y [m]
1	0.000	0.000
2	0.000	4.600
3	8.400	4.600
4	8.400	0.000

- Supports. Nodes 1 and 4 fixed.

nodes | supports | nodal loads | nodal masses

node	support	ux[mm]	uy[mm]	ur[rad]
1	///////	0.00000	0.00000	0.00000
4	///////	0.00000	0.00000	0.00000

- Nodal loads are zero, (in this example there are no loads on the nodal points).

nodes | supports | nodal loads | nodal masses

load combination 1.35 xFg+ 1.50 xFq

node	Fgx[kN]	Fqx[kN]	Fgy[kN]	Fqy[kN]	Mg[kNm]	Mq[kNm]

- Nodal masses are necessary only in dynamic analysis.

- Elements. The element numbering is displayed on the drawing of the structure.
  Nodes A and B are the left and right nodes of each element. Cross section is the number in parenthesis next to each element and represents the number of the section group which properties defined in the page *cross sections*.

elements | cross sections | element loads

element	node A	node B	cr. sect.
1	1	2	2
2	2	3	1
3	3	4	2

- Cross sections. The material is R. Concrete. The modulus of elasticity is automatically adjusted (26 GPa R. concrete, 210 GPa Structural steel and 10 GPa timber).
  Select units for cross section dimensions (eg. cm). For every cross-section group, (1 for horizontal beams, 2 columns) select cross section, T or rectangular cross section.
  The cross section sizes are: b (width), h (height), b1 (effective flange for T section) and h1 (slab thickness for T section). The values for A and I (area and moment of inertia of the cross section) are automatically set from b, h and b1, h1 values.

elements | cross sections | element loads | elem. masses, self weight | R.Concrete

structure material: R.Concrete | Elasticity modulus: E (GPa)= 26.00 | cross section units: cm

N	cr. se	b [cm]	h [cm]	b1 [cm]	h1 [cm]	A [cm²]	I [cm4]
1	T	25.0	70.0	120.0	15.0	31.750E002	13.353E005
2	□	30.0	60.0	0.0	0.0	18.000E002	54.000E004

- Element loads. For every element loaded with distributed load supply one or more loads. Number of loaded element (eg. 2), kind (uniform triangular etc.), load value (dead load g kN/m or live load q kN/m). Careful with dead load, give the additional to the element self weight. The program computes the self weight of the elements if it is checked in the next page (mass self weight) of data. The load direction is (2) downwards for gravity loads and snow load, (1) for wind and pressure and (3) for horizontal loads as seismic load.

elements | cross sections | element loads | elem. masses, self weight | R.Concrete

load combination 1.35 **xG** + 1.50 **xQ**

element	kind	dead g [kN/m]	live q [kN/m]	direction
2	▥	22.500	12.400	↓

- Element masses, self weight. The weight density of the material. If you check to include the self weight in the load and masses, the program adds in dead loads the self weight of each element (unit weight) x (cross section area). The weight density is set automatically by the program (R. concrete 25 KN/m^3, steel 78.50 kN/m^3, timber 9kN/m^3).

elements | cross sections | element loads | elem. masses, self weight | R.Concrete

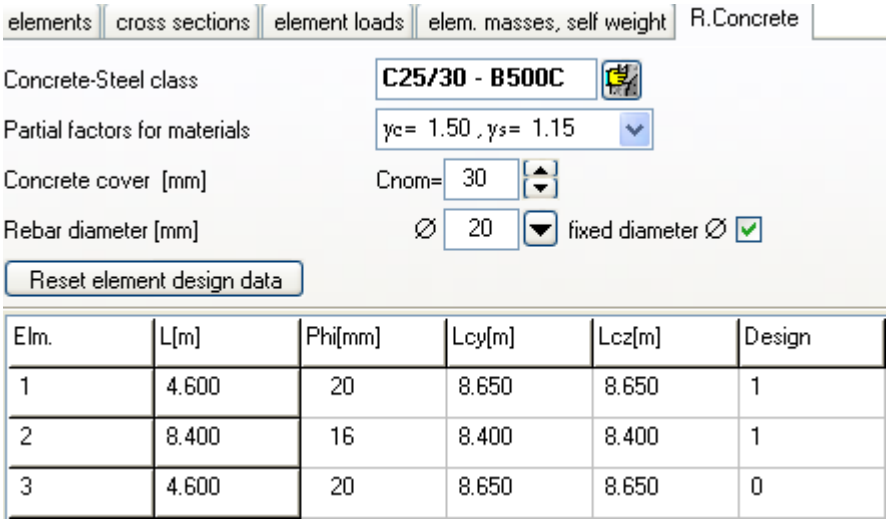
Weight density kN/m3: 25.00 | include self weight in loads and masses ☑

Mass combination: 1.00 **xMg** + 0.30 **xMq**

element	Gg [kN/m]	Gq [kN/m]

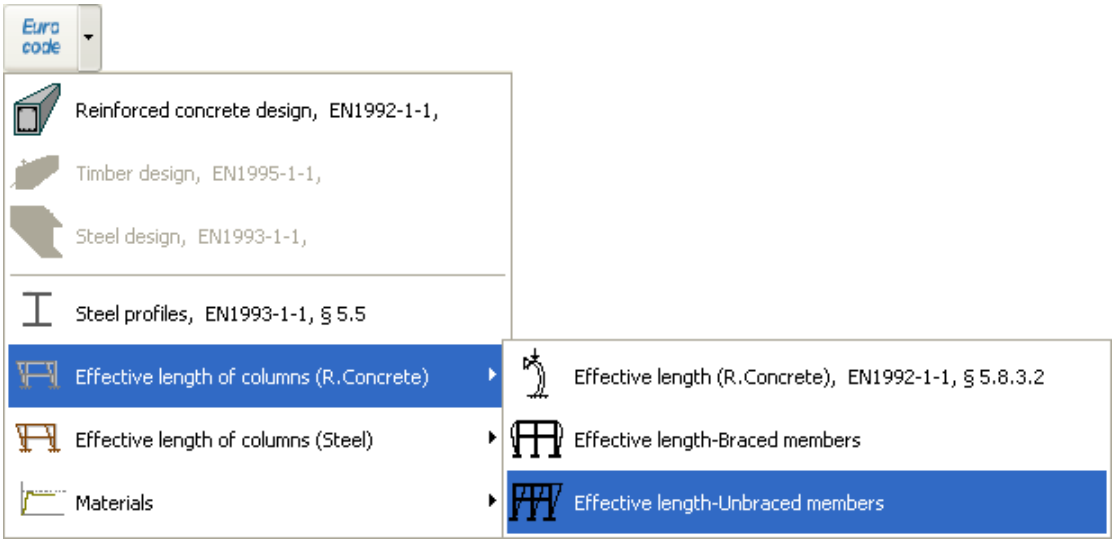
- Concrete. You specify the basic data for the design of reinforced concrete according to Eurocode 2. Select Concrete and Steel class. Partial factors for materials, according to the

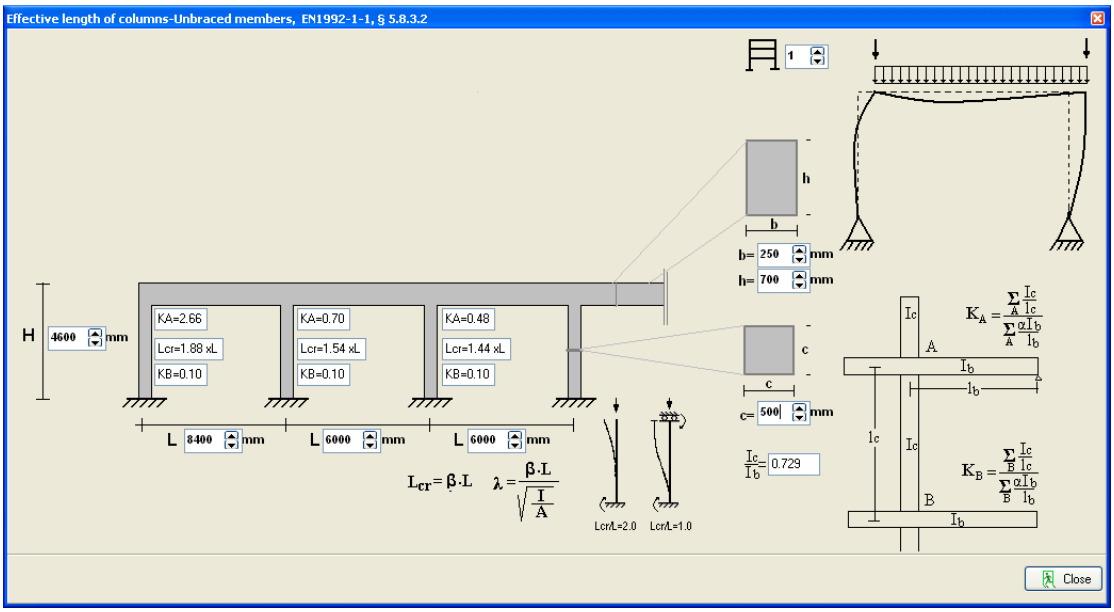
National Annex, for ULS (ultimate limit state) γc=1.50, γs=1.15 and SLS (serviceability limit state) γc=1.00, γs=1.00. Cnom is the concrete cover according to Eurocode 2 §3.4.1.
The rebar diameter is used as the optimum desired by the program. If you check next to the rebar diameter then is only this diameter selected by the program. For every element you may specify in the column Phi [mm] the desired rebar diameter eg. 20 mm for columns and 16 mm for beams.
The buckling lengths Lcy and Lcz for in and out of plane buckling are used for stability checks using second order effects for the columns, according to Eurocode 2 §5.8.3.
In the column Design, mark with 1 the elements which you want to be included in the design of reinforced concrete. In this example the elements 1 and 2 are marked with (1) and element 3 with (0), as there is no need because of symmetry to include element 3 (right column) in the reinforced concrete design.



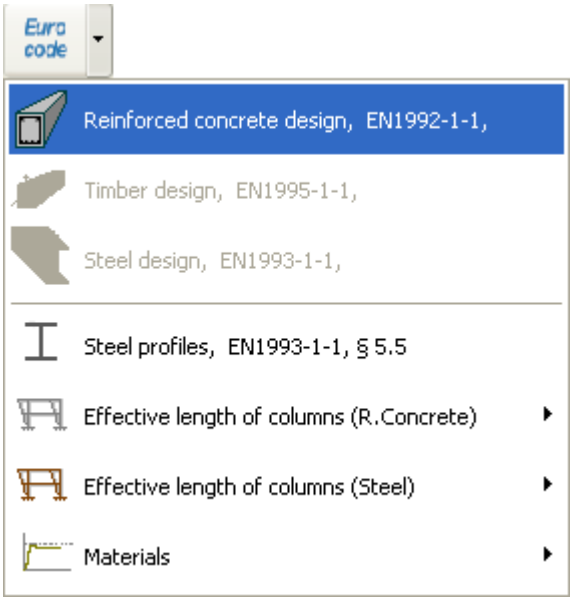
Elm.	L[m]	Phi[mm]	Lcy[m]	Lcz[m]	Design
1	4.600	20	8.650	8.650	1
2	8.400	16	8.400	8.400	1
3	4.600	20	8.650	8.650	0

In order to define the buckling lengths of the columns for unbraced frame according to Eurocode 2 5.8.3.2, use the extra tools of the program Design/Effective length-Unraced members.
For this example we obtain Lcr = 1.88 x L = 1.88 x 4.60 = 8.65 m

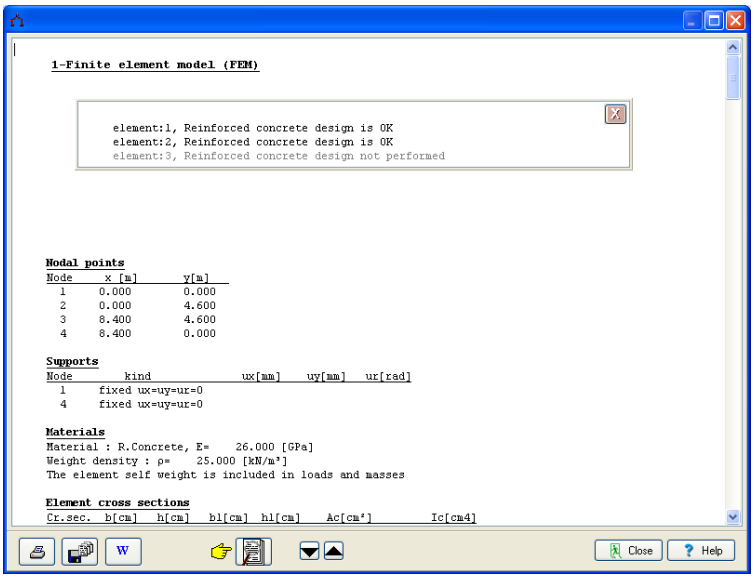




After you give all the data the reinforced concrete design is performed according to Eurocode 2.

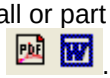


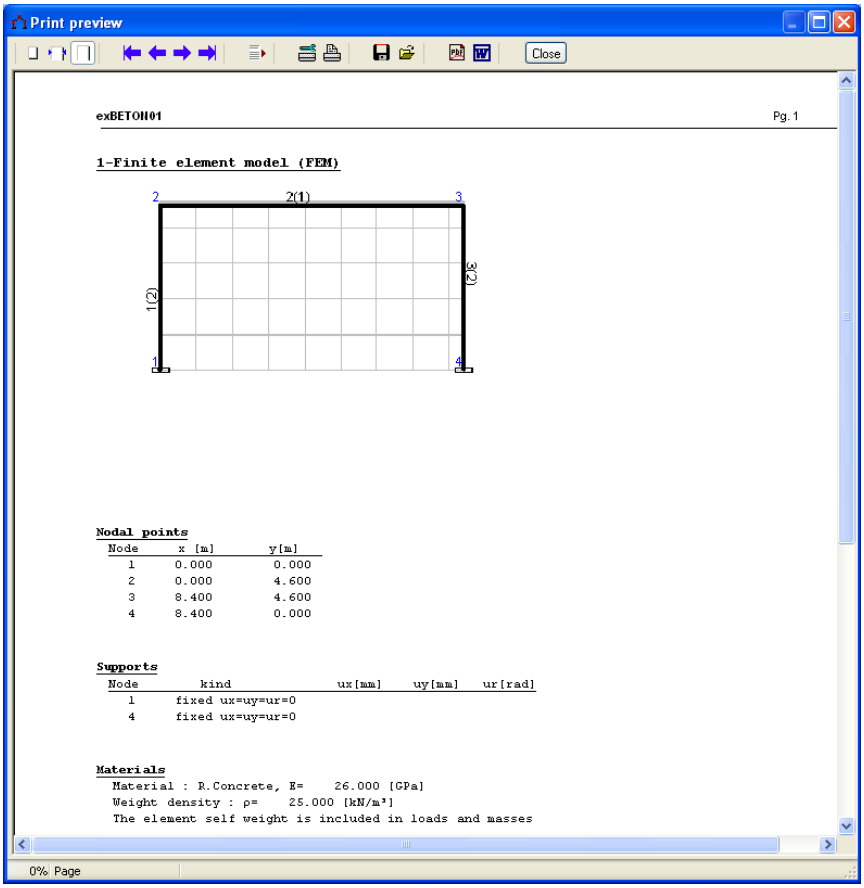
Check if every element is verified in the design.



Click for complete formatted report.

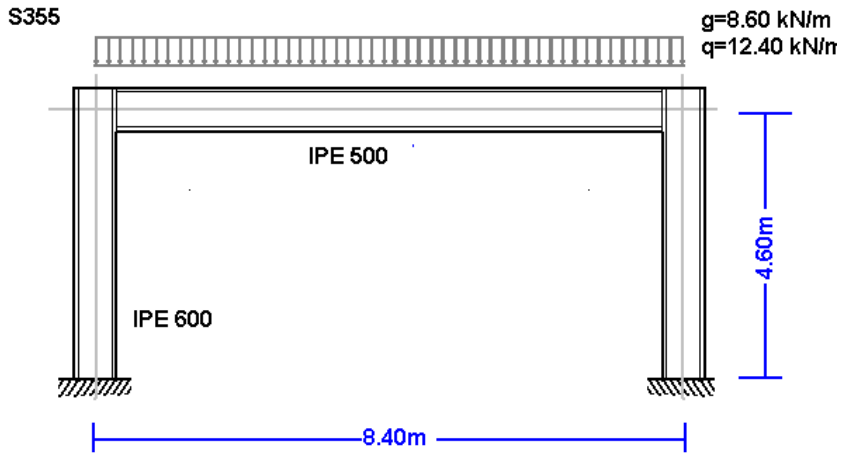
From the report preview you can print all or part (from page to page) of the report

or you can export to PDF or Word files  .

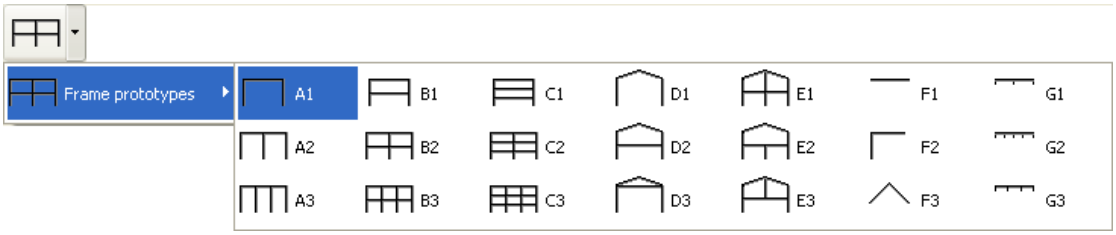


1.2 Example 2

Steel frame 8.40 x 4.60 S355

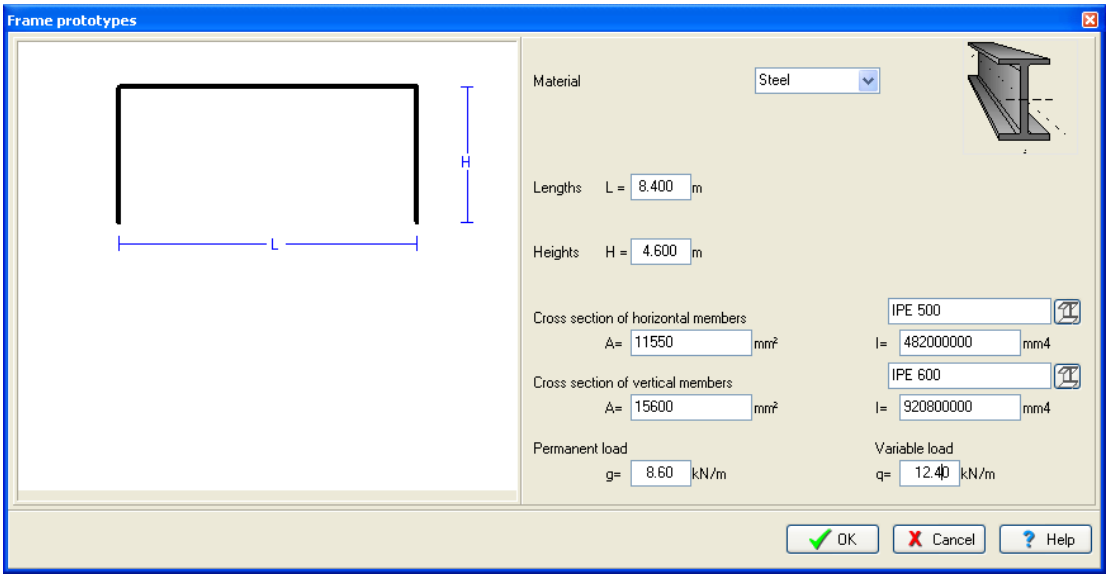


Select a frame from File/Frame prototypes:

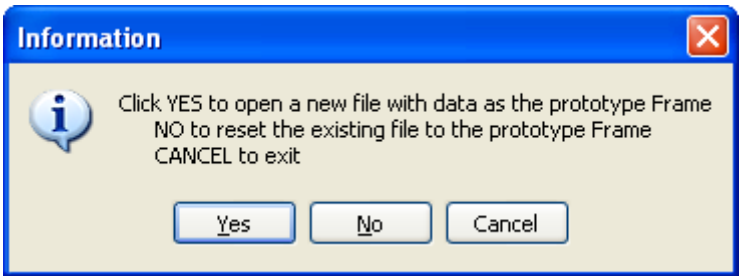


Select material Steel and give the basic dimensions, cross sections and loads.
You can always change and adjust these values afterwards.

For element cross sections click .



Click Yes and give the file name.



Then check and adjust the rest of the values for the structural model.

Select National Annex of your region and partial safety factors.
Usual values for partial safety factors ULS (ultimate limit state) γG=1.35, γQ=1.50 and SLS (serviceability limit state) γG=1.00, γQ=1.00.

Check the drawing of the structure.

- Nodes. Coordinate system at lower left point.
  Axis x from left to right, axis y from down up.
  The numbering of the nodal points is displayed on the drawing of the structure.

nodes | supports | nodal loads | nodal masses

node	x [m]	y [m]
1	0.000	0.000
2	0.000	4.600
3	8.400	4.600
4	8.400	0.000

- Supports. Nodes 1 and 4 fixed.

nodes | supports | nodal loads | nodal masses

node	support	ux[mm]	uy[mm]	ur[rad]
1	///////	0.00000	0.00000	0.00000
4	///////	0.00000	0.00000	0.00000

- Nodal loads. Vertical loads on nodal points 2 and 3, permanent 95 kN and variable 125 kN.
  Sign of loads (-) negative, loads downwards.

nodes | supports | nodal loads | nodal masses

load combination 1.35 xFg+ 1.50 xFq

node	Fgx[kN]	Fqx[kN]	Fgy[kN]	Fqy[kN]	Mg[kNm]	Mq[kNm]
2	0.000	0.000	-95.000	-125.000	0.000	0.000
3	0.000	0.000	-95.000	-125.000	0.000	0.000

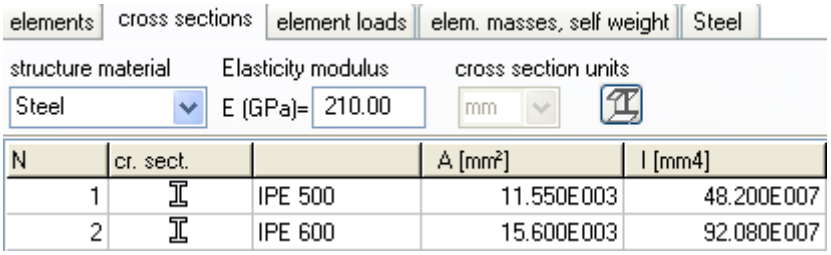
- Nodal masses are necessary only in dynamic analysis.

- Elements. The element numbering is displayed on the drawing of the structure.
  Nodes A and B are the left and right nodes of each element. Cross section is the number in parenthesis next to each element and represents the number of the section group which properties defined in the page cross sections.

elements | cross sections | element loads

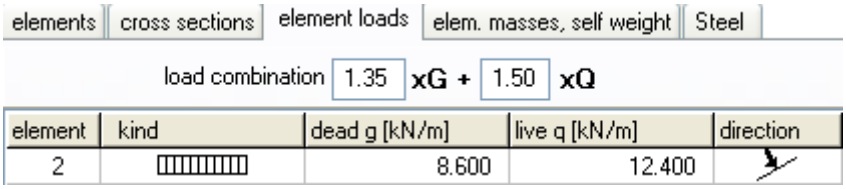
element	node A	node B	cr. sect.
1	1	2	2
2	2	3	1
3	3	4	2

- Cross sections. Material Steel. The modulus of elasticity is automatically adjusted (210 GPa for steel). Select units for cross section dimensions eg. mm). For every cross section group (1 for horizontal beams, 2 columns) select cross section. In the column with the name of the cross section click [...] and the library with all the steel cross sections is display to select cross section type and size.



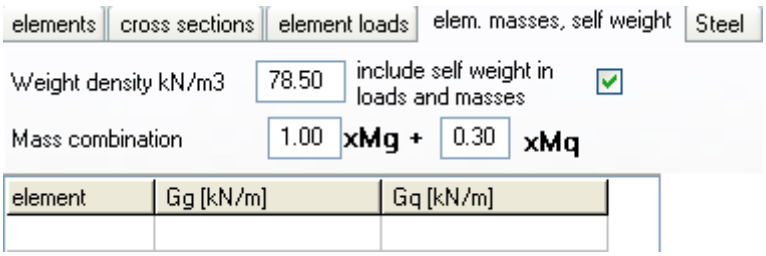
N	cr. sect.		A [mm²]	I [mm4]
1		IPE 500	11.550E003	48.200E007
2		IPE 600	15.600E003	92.080E007

- Element loads. For every element loaded with distributed load supply one or more loads. Number of loaded element (eg. 2), kind (uniform triangular etc.) load value (dead load g kN/m or live load q kN/m). Careful with dead load, give the additional to the element self weight. The program computes the self weight of the elements if it is checked in the next page (mass self weight) of data. The load direction is (2) downwards for gravity loads and snow load, (1) for wind and pressure and (3) for horizontal loads as seismic load.



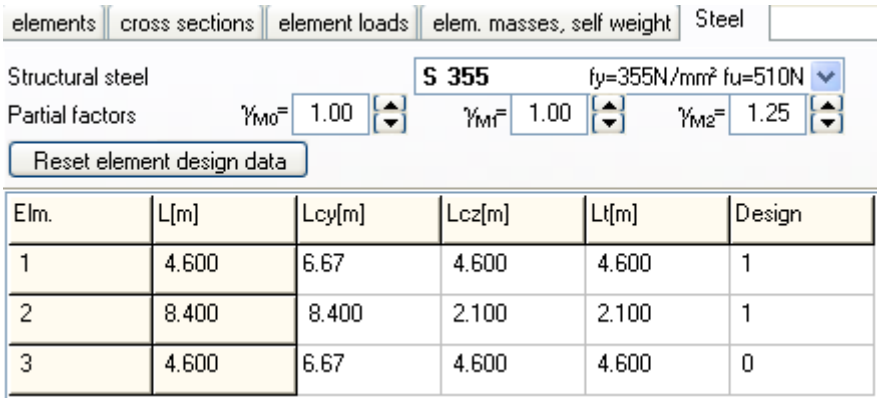
element	kind	dead g [kN/m]	live q [kN/m]	direction
2		8.600	12.400	

- Element masses, self weight. The weight density of the material. If you check to include the self weight in the load and masses, the program adds in dead loads the self weight of each element (unit weight) x (cross-section area). The weight density is set automatically by the program (steel 78.50 kN/m^3).



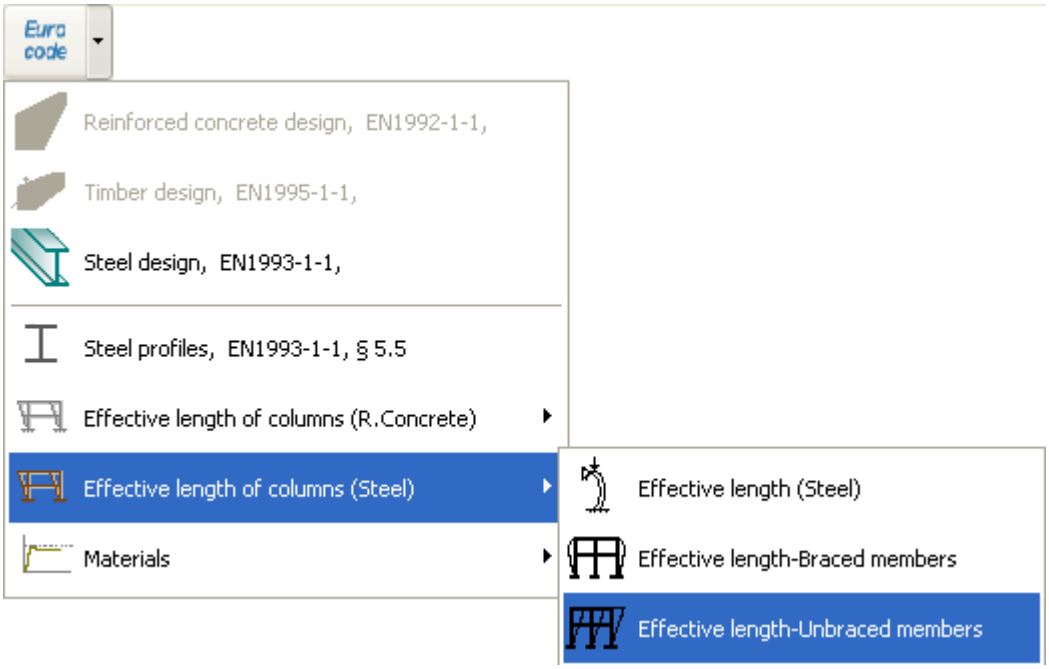
element	Gg [kN/m]	Gq [kN/m]

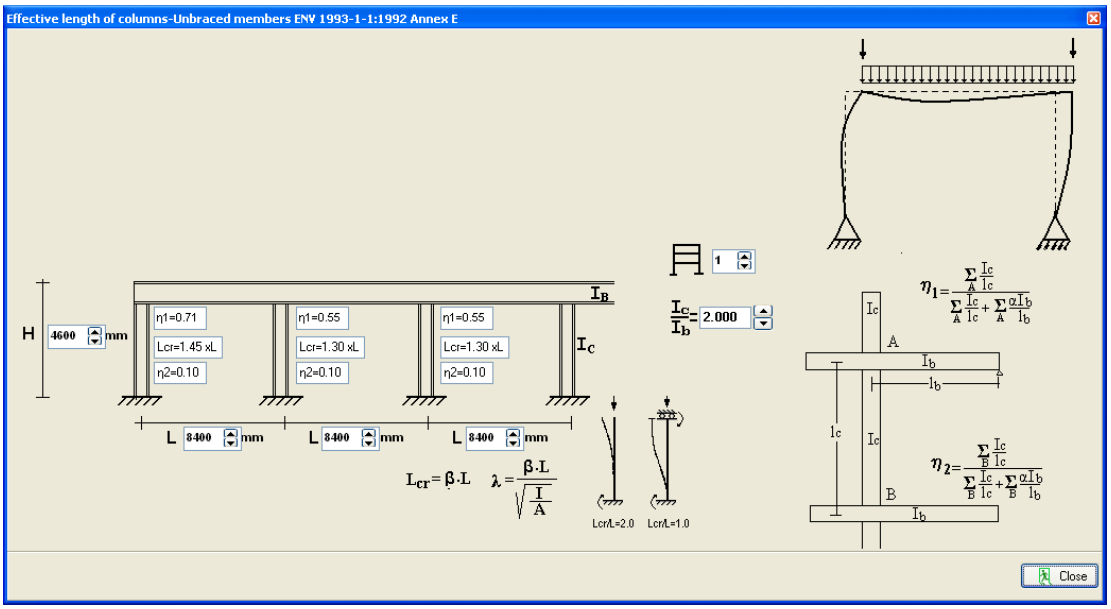
- Steel. You specify the basic data for the design of steel according to Eurocode 3. Select Steel grade. Partial factors for materials, according to National annex, for ULS (ultimate limit state) γM0 = 1.00, γM1 = 1.00, γM2 = 1.25. You have to define the buckling lengths. Lcy buckling length for in plane buckling. For braced frames this is less or equal to the member length, for unbraced members is grater. Lcz buckling length for out of plane buckling and it is defined from the lateral supports. For horizontal beams it is usually the distance of the lateral beams or the purlins. In the column Design mark with 1 the elements which you want to be included in the design of steel according to Eurocode 3. In this example the elements 1 and 2 are marked with (1) and element 3 with (0), as there is no need because of symmetry to include element 3 (right column) in the steel design,



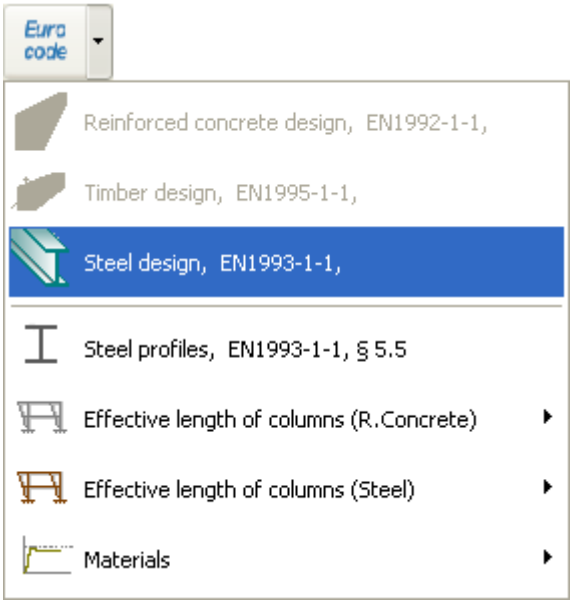
Elm.	L[m]	Lcy[m]	Lcz[m]	Lt[m]	Design
1	4.600	6.67	4.600	4.600	1
2	8.400	8.400	2.100	2.100	1
3	4.600	6.67	4.600	4.600	0

In order to define the buckling lengths of the columns for unbraced and braced frame according to Eurocode 3, use the extra tools of the program Design/Effective length of columns/Unraced members.
For this example we obtain Lcr = 1.45 x L = 1.45 x 4.60 = 6.67 m

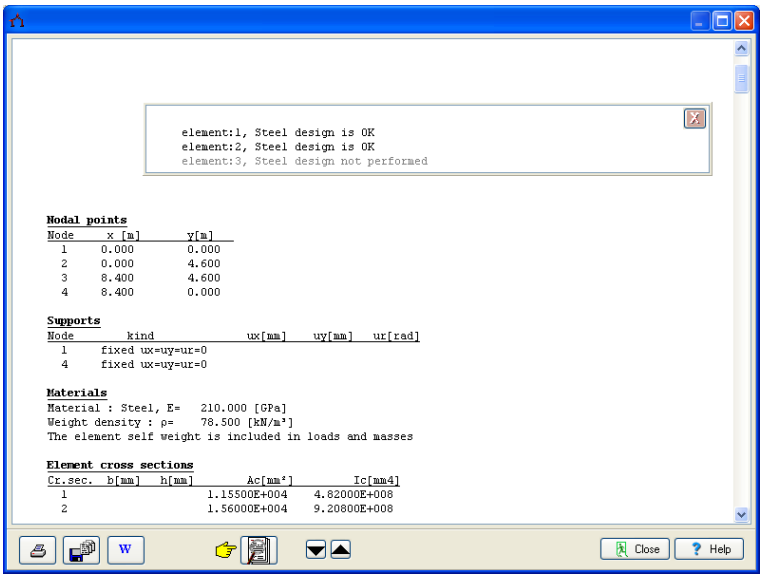




After you give all the data the steel design is performed according to Eurocode 3.

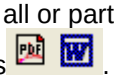


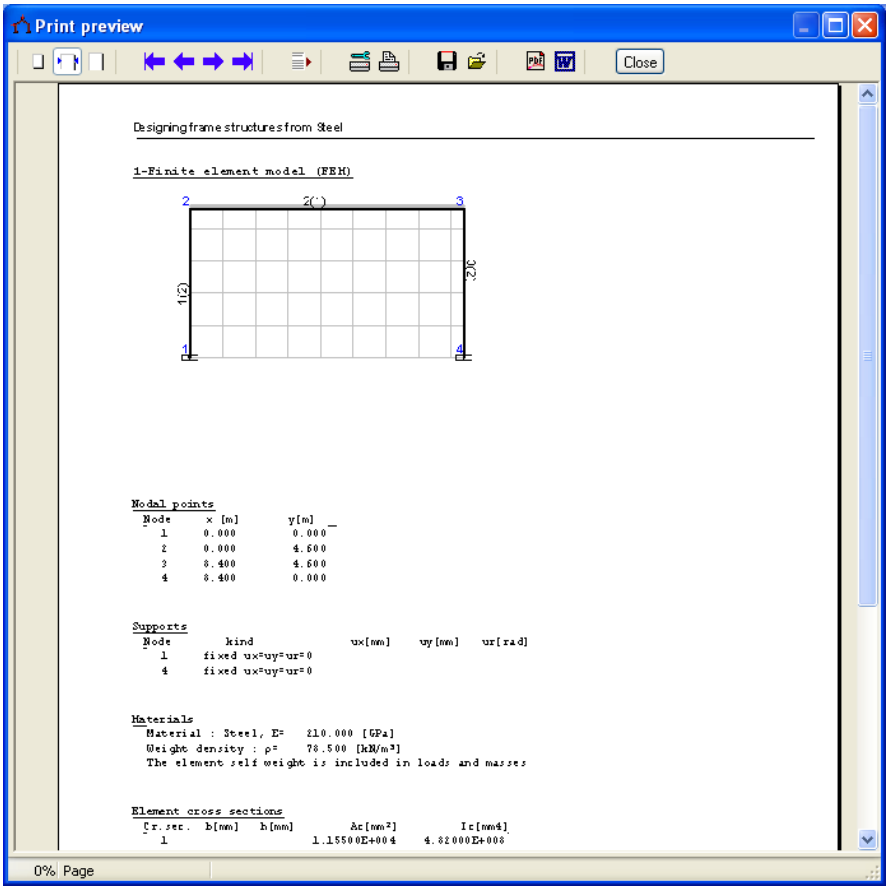
Check if every element is verified in the design,



Click for complete formatted report.

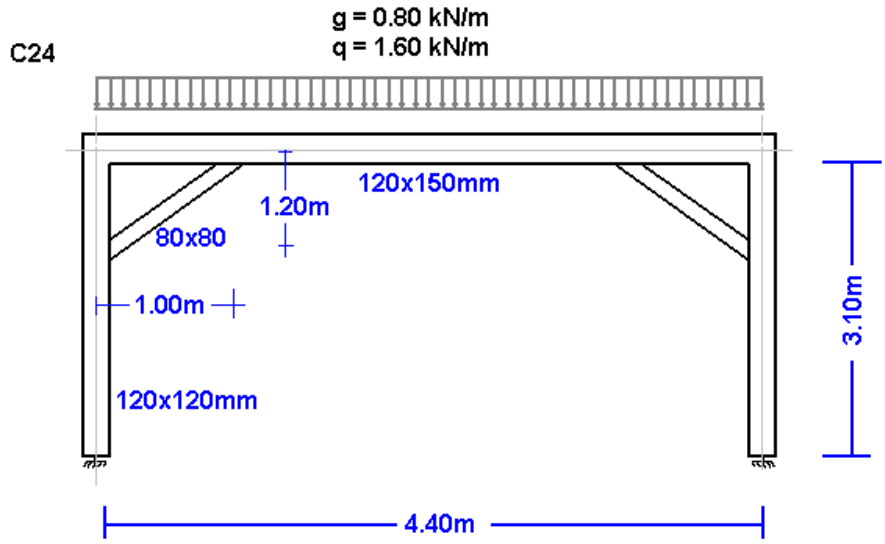
From the report preview you can print all or part (from page to page) of the report

or you can export to PDF or Word files  .

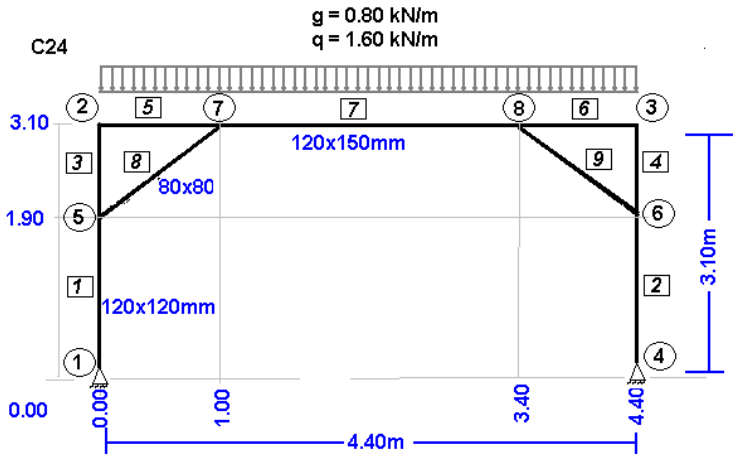


1.3 Example 3

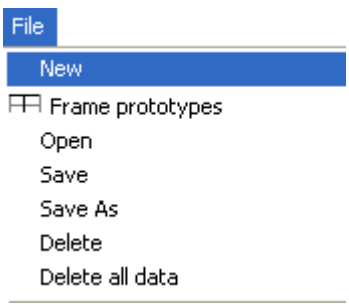
Timber structure 6.40 x 4.60 C24



Structural model



Create a new file:



Supply all the data of the Timber structure.

Select National Annex of your region and partial safety factors.
Usual values for partial safety factors ULS (ultimate limit state) γG=1.35, γQ=1.50 and SLS (serviceability limit state) γG=1.00, γQ=1.00.

- Nodes. Coordinate system at lower left point.
  Axis x from left to right, axis y from down up.
  The numbering of the nodal points is displayed on the drawing of the structure.
  Use for adding or altering lines in the table.

nodes | supports | nodal loads | nodal masses

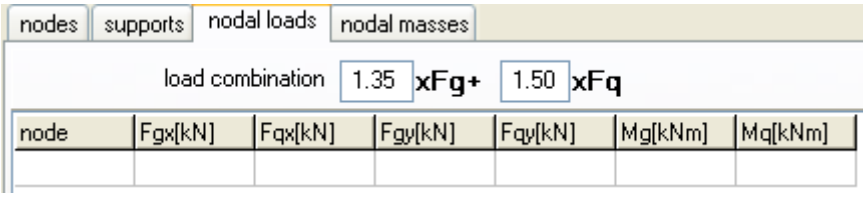
node	x [m]	y [m]
1	0.000	0.000
2	0.000	3.100
3	4.400	3.100
4	4.400	0.000
5	0.000	1.900
6	4.400	1.900
7	1.000	3.100
8	3.400	3.100

- Supports. Nodes 1 and 4 are pin supports. Click support to select support kind.

nodes | supports | nodal loads | nodal masses

node	support	ux[mm]	uy[mm]	ur[rad]
1	△	0.00000	0.00000	0.00000
4	△	0.00000	0.00000	0.00000

- Nodal loads are zero (in this example there are no loads on the nodal points).



- Nodal masses are necessary only in dynamic analysis.

- Elements. For every element according to the numbering of the line drawing of the model supply the element number, the number of the left and right. Number of cross section 1 for vertical elements, 2 for horizontal and 3 for diagonal elements.

elements | cross sections | element loads

element	node A	node B	cr. sect.
1	1	5	1
2	6	4	1
3	5	2	1
4	3	6	1
5	2	7	2
6	8	3	2
7	7	8	2
8	5	7	3
9	8	6	3

- Cross sections. Select material timber. The modulus of elasticity is automatically adjusted (10 Gpa timber). Select units for cross section dimensions (eg cm).
  For every cross section group (1 vertical elements 12x12, 2 horizontal 12x15, 3 diagonal 8x8) supply the cross section sizes b width, h height. The values for A and I (area and moment of inertia of the cross-section) are automatically set from b h. The diagonal elements usually are pin connected with the vertical and horizontal elements. In order to approximate such a model with the program (pin connections for nodes 5 and 7 for element 8), after you give the cross section dimensions b = 8 and h = 8 for group section 3, change the moment of inertia value I to a much smaller value.
  In the example instead of 341.33 has been changed to 34.13 (10 times smaller).
  With this change the diagonal elements become flexible and do not take bending moments (see bending moment diagram).

elements | cross sections | element loads | elem. masses, self weight | Timber

structure material: Timber — Elasticity modulus: E (GPa)= 10.000 — cross section units: cm

N	b [cm]	h [cm]	A [cm²]	I [cm4]
1	12.0	12.0	14.400E001	17.280E002
2	12.0	15.0	18.000E001	33.750E002
3	8.0	8.0	64.000E000	34.133E001

- Element loads. For every element loaded with distributed load supply one or more loads. Number of loaded element (eg. 2), kind (uniform triangular etc.), load value (dead load g kN/m or live load q kN/m. Careful with dead load, give the additional to the element self weight. The program computes the self weight of the elements if it is checked in the next page (mass self weight) of data. The load direction is (2) downwards for gravity loads and snow load, (1) for wind and pressure and (3) for horizontal loads as seismic load.

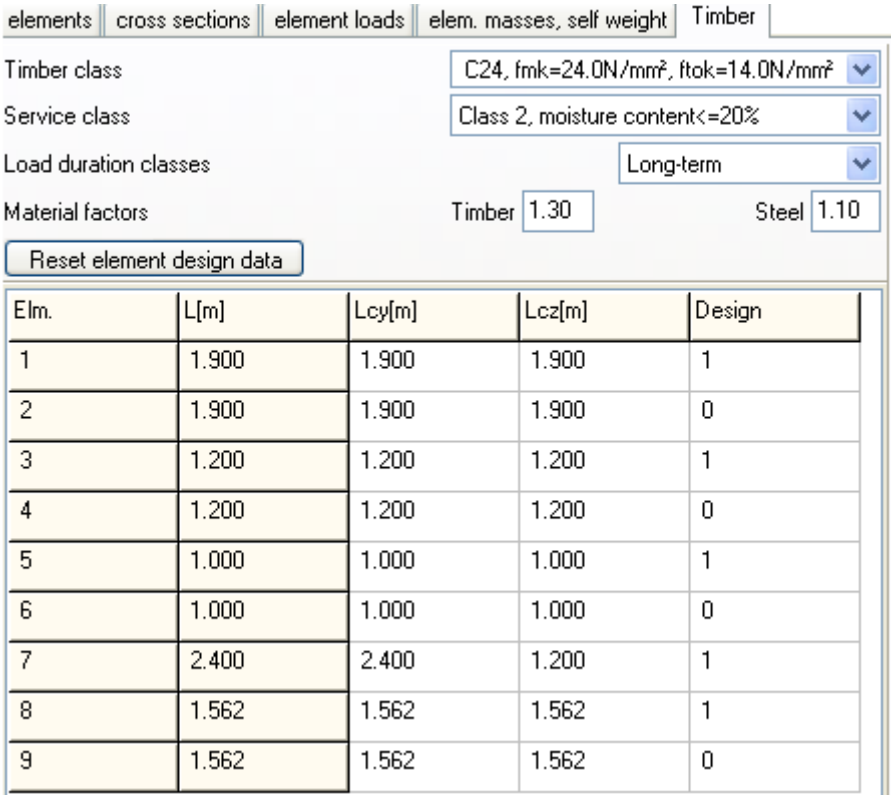
elements | cross sections | element loads | elem. masses, self weight | Timber

load combination 1.35 xG + 1.50 xQ

element	kind	dead g [kN/m]	live q [kN/m]	direction
5		0.800	1.600	
6		0.800	1.600	
7		0.800	1.600	

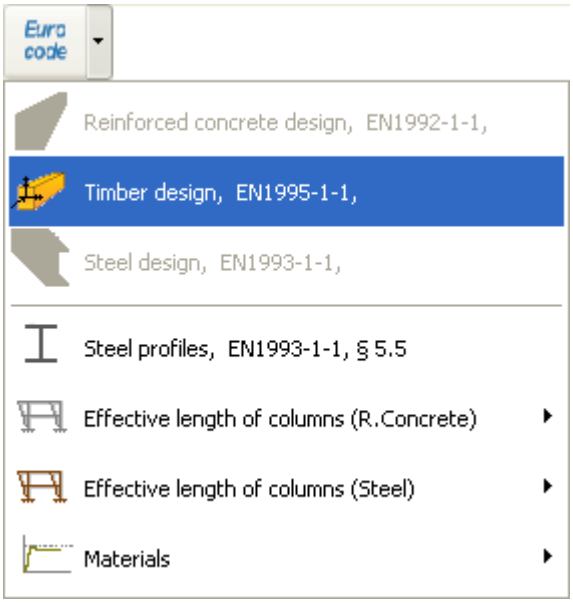
- Element masses self weight. The weight density of the material. If you check to include the self weight in the load and masses, the program adds in dead loads the self weight of each element (unit weight) x (cross section area). The weight density is set automatically by the material (timber 9 kN/m^3).

- Timber. You specify the basic data for the design of the timber members according to Eurocode 5. Select timber class (C24), service class and load duration class. Material factors according to national Annex. For ULS (ultimate limit state) γM=1.30 and for SLS (serviceability limit state) γM=1.00. You have to specify the buckling lengths Lcy and Lcz for in plane and out of plane buckling. For the horizontal elements Lcz is the distance between transverse beams or purlins (1.20 m).

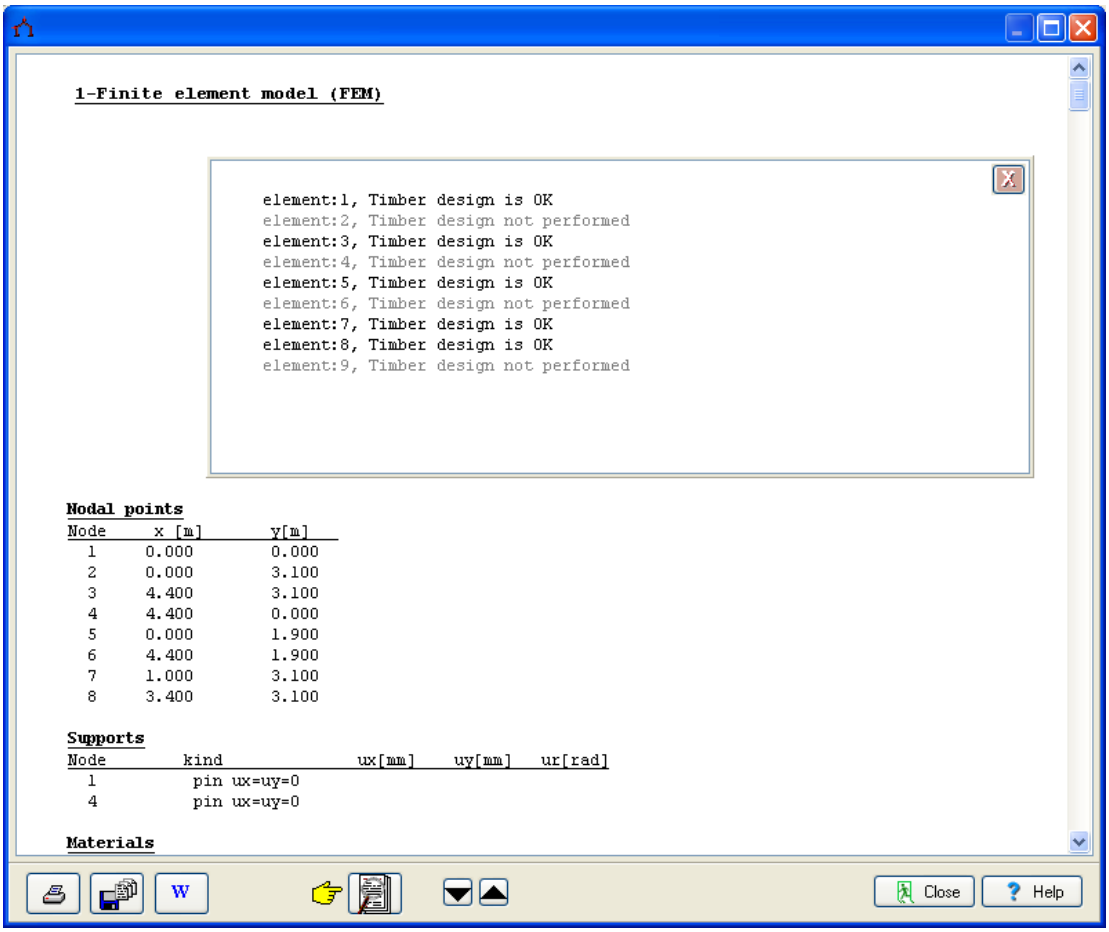


Elm.	L[m]	Lcy[m]	Lcz[m]	Design
1	1.900	1.900	1.900	1
2	1.900	1.900	1.900	0
3	1.200	1.200	1.200	1
4	1.200	1.200	1.200	0
5	1.000	1.000	1.000	1
6	1.000	1.000	1.000	0
7	2.400	2.400	1.200	1
8	1.562	1.562	1.562	1
9	1.562	1.562	1.562	0

After you give all the data the timber design is performed according to Eurocode 5.



Check if every element is verified in the design.



Click for complete formatted report.

From the report preview you can print all or part (from page to page) of the report,

or you can export to PDF or word file .