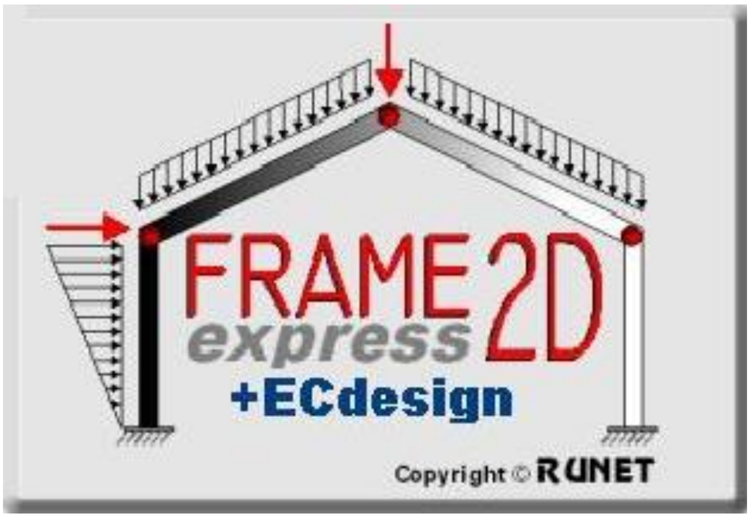


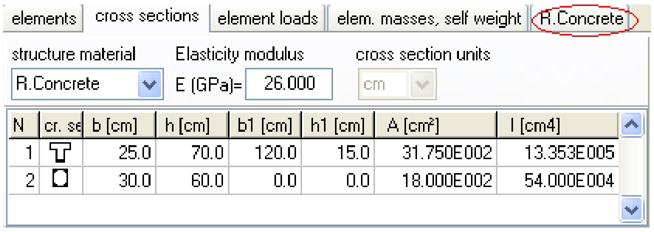
New version for Designing members of Reinforced Concrete, Steel or Timber according to Eurocode 2, Eurocode 3 and Eurocode 5

New-Updated version of Frame2Dexpress:

- Design for reinforced concrete, steel and timber according to Eurocodes.
- Frame prototypes for easy generation of structural models.
- Steel profiles. All the standard steel profiles for the design of steel frame structures.

1. Designing elements from reinforced concrete according to Eurocode 2

After you select material *Concrete* an additional page Concrete shows in the material session.
In this page you input all necessary data for the reinforced concrete design of the frame elements.

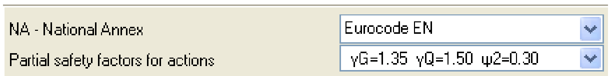


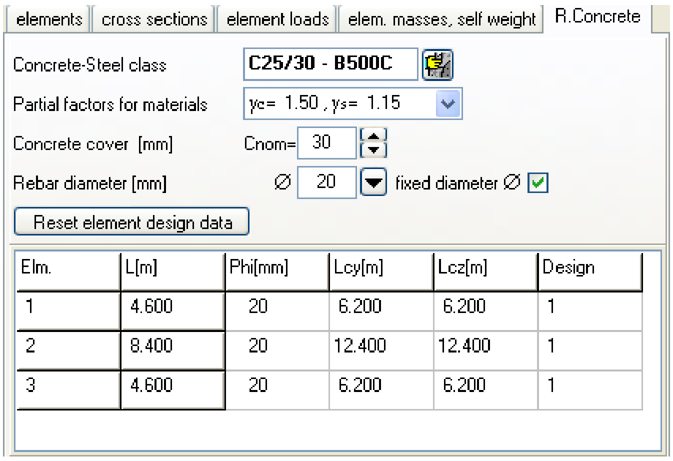
1.1 Design parameters for reinforced concrete

In page Concrete you define parameters for the reinforced design.

1. To select concrete and reinforcing steel class, click.

The concrete and reinforcing steel classes are adjusted according to the selected National Annex.
You can change strength properties for the concrete and reinforcing steel from *Design/Materials/Concrete or Design/Materials/Reinforcing steel.*





2. Partial factors for materials.
This is defined according to National annex, usual values:
γc = 1.50, γs = 1.15.

3. Concrete Cnom, cover in mm.

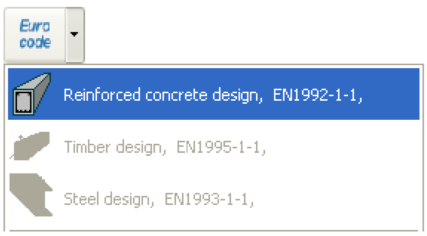
4. Preferable rebar diameter.
If you check *fixed* the selected diameter will be used. If not, some optimum diameter will be used, around the preferable.

5. For every element you can defile:
Lcy: Buckling length for in-plane flexural buckling (meters), usually the length of the member.
Lcz: Buckling length for out-of-plane buckling (meters), usually the length of the member.

6. Design: = 1 The reinforced concrete design of this element is performed.
= 0 This element is skipped in the design.

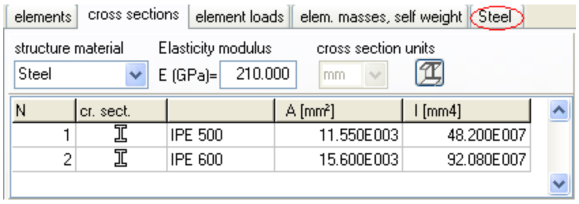
1.2 Reinforced concrete design according to Eurocode 2

Click Reinforced Concrete Design. All the marked with Design = 1 elements will be verified according to Eurocode 2, §6, for axial force, shear and bending moment in ultimate limit state. The design for reinforcement is performed for mid span, left end and right end of each element. The vertical elements in compression (columns) are verified for second order effects according to Eurocode 2, §5.8.3.

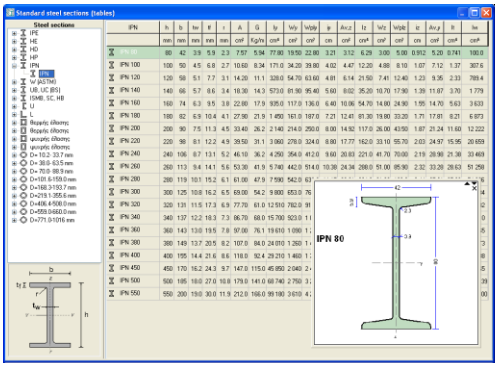


2. Designing steel members according to Eurocode 3

After you select material *Steel* an additional page Steel shows in the material session.
In this page you input all necessary data for the steel design of the frame elements.



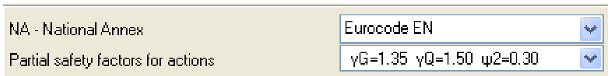
If you click on a line of the table on cross section or click the table with cross sections shows up and you select a standard profile for this element.
In order to proceed with the steel design you have to select standard profiles for all the elements.



2.1 Design parameters for steel

In page Steel you define parameters for the steel design.

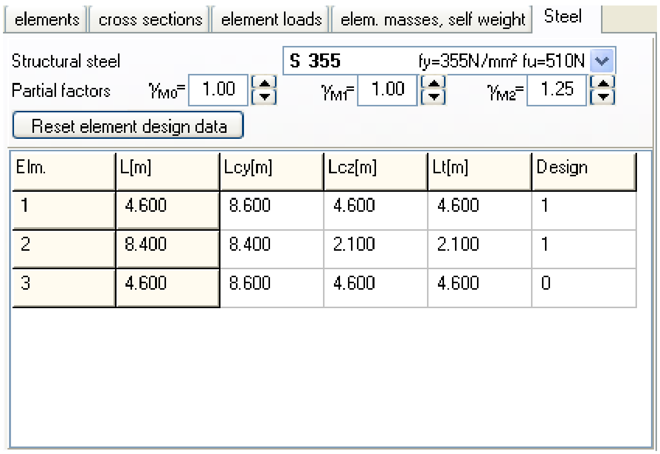
1. Select Steel grade.
The steel grades are adjusted according to the selected National Annex.



You can change properties for structural steel from *Design/Materials/Structural steel.*

2. Partial factors for materials.
This is defined according to National annex, usual values:
$\gamma M0=1.00$, $\gamma M1=1.00$, $\gamma M2=1.25$

3. For every element you can defile:
Lcy: Buckling length for in-plane flexural buckling (meters), usually the length of the member.
Lcz: Buckling length for out-of-plane flexural buckling (meters), usually the distance of lateral supports as purlins for rafters.
Lt: Buckling length for lateral torsional buckling (meters), usually distance of lateral supports as purlins.



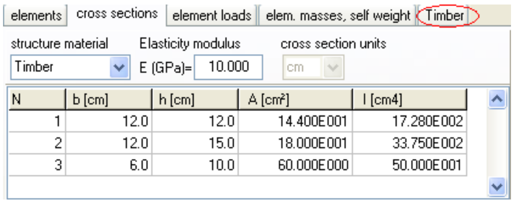
4. Design: = 1 The steel design of this element is performed.
= 0 This element is skipped in the design.

2.2 Steel design according to Eurocode 3

Click Steel design. All the marked with Design = 1 elements will be verified according to Eurocode 3, §6.2, for axial force, shear and bending moment in ultimate limit state, according to §6.3 for flexural and lateral torsional buckling.
The buckling critical lengths are the ones defined in the steel design page. The strength checks are performed for mid span, left end and right end of each element.

3. Designing timber members according to Eurocode 5

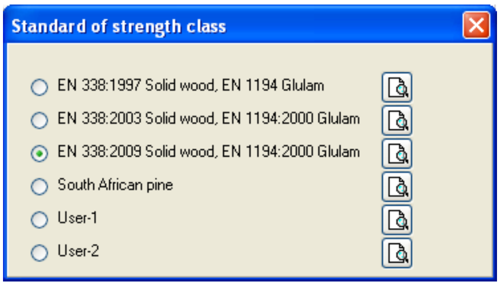
After you select material *Timber* an additional page Timber shows in the material session where you define the additional parameters for timber design.



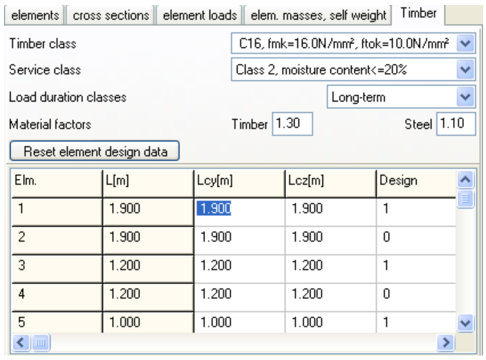
3.1 Design parameters for timber

In page Timber you define parameters for the timber design.

1. Select Timber class. The material properties are according to the selected EN in *Design/Materials/Timber.*



The EN standards are EN338:1997, EN338:2003, or EN 338:2009 or one user defined. Last EN standard is EN 338:2009. You must notice that using an older EN338:1997, EN338:2003 standard with lower defined shear strength the shear strength checks are performed with kcr = 1. Selecting EN 338:2009 (which has increased shear strengths) the shear strength checks are performed with kcr 0.67 as is defined in addition A1:2008 of Eurocode 5 (Eq. 6.13a).



2. Select service class.
Select load duration class. Usually self weight is permanent, snow load and live load is long term, wind load is short term.

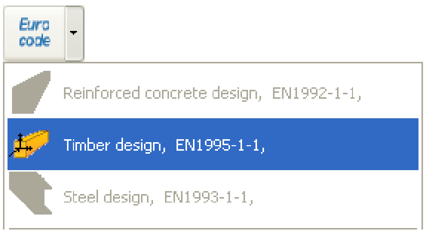
3. Partial factors for materials. This is defined according to National annex, usual values:
$\gamma M=1.30$, for timber and $\gamma M=1.10$ for steel connectors.

4. For every element you can defile:
Lcy: Buckling length for in-plane flexural buckling (meters), usually the length of the member.
Lcz: Buckling length for out-of-plane flexural buckling (meters), usually the distance of lateral supports as purlins for rafters.

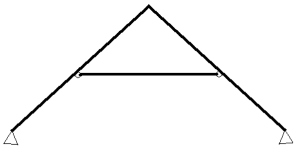
5. Design: = 1 The timber design of this element is performed.
= 0 This element is skipped in the design.

3.2 Timber design according to Eurocode 5

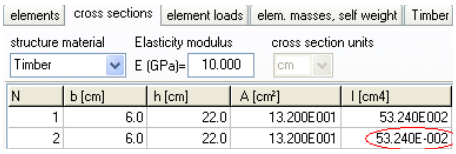
Click Timber Design. All the marked width Design = 1 elements will be verified according to Eurocode 5, §6, for axial force, shear and bending moment in ultimate limit state and according to §6.3 for stability. The buckling critical lengths are defined in the timber design page.
The checks are performed for mid span, left end and right end of each element.



3.3 Design notes



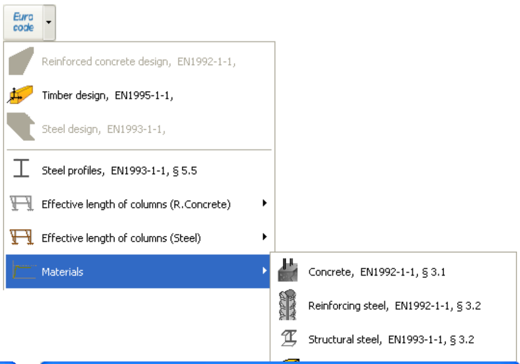
The connections in frame are fixed connections. In case you have to define some timber elements which they are pin-connected to other elements then for these elements define a separate material-section group and after you define the b and h of the cross section, change the moment of inertia to a small number.
Example in the structure showing at the right, the horizontal member carries only axial force.
The moment of inertia of this member has been changed by dividing the original by 10^4.

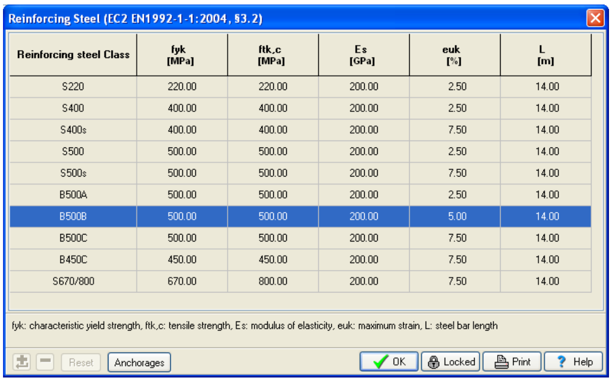


N	b [cm]	h [cm]	A [cm²]	I [cm4]
1	6.0	22.0	13.200E001	53.240E002
2	6.0	22.0	13.200E001	53.240E-002

4. Materials for Reinforcing Concrete, Structural Steel and Timber

The materials for concrete, reinforcing steel, structural steel and timber are adjusted according to the selected National Annex.
You can change material properties from Design/Materials.
In order to change values for materials first you have to unlock the tables with properties.
Click Closed.
For timber you can select one of the EN prototypes. EN338:1997, EN338:2003 or EN 338:2009 or one defined by the user.
The user defined prototype allows you do change the material properties.



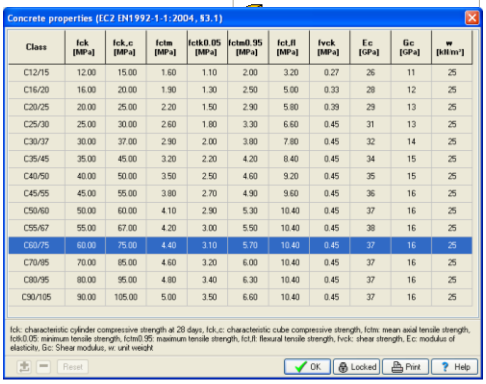


Reinforcing Steel (EC2 EN1992-1-1:2004, §3.2)

Reinforcing steel Class	fyk [MPa]	ftk,c [MPa]	Es [GPa]	euk [%]	L [m]
S220	220.00	220.00	200.00	2.50	14.00
S400	400.00	400.00	200.00	2.50	14.00
S400s	400.00	400.00	200.00	7.50	14.00
S500	500.00	500.00	200.00	2.50	14.00
S500s	500.00	500.00	200.00	7.50	14.00
B500A	500.00	500.00	200.00	2.50	14.00
B500B	500.00	500.00	200.00	5.00	14.00
B500C	500.00	500.00	200.00	7.50	14.00
B450C	450.00	450.00	200.00	7.50	14.00
S670/800	670.00	800.00	200.00	7.50	14.00

fyk: characteristic yield strength, ftk,c: tensile strength, Es: modulus of elasticity, euk: maximum strain, L: steel bar length

Reset Anchorages OK Locked Print Help

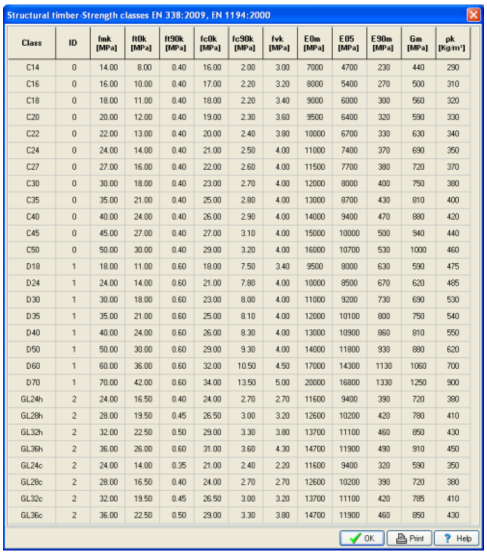


Concrete properties (EC2 EN1992-1-1:2004, §3.1)

Class	fck [MPa]	fck,c [MPa]	fctm [MPa]	fctk0.05 [MPa]	fctm0.95 [MPa]	fct,fl [MPa]	fvck [MPa]	Ec [GPa]	Gc [GPa]	w [kN/m³]
C12/15	12.00	15.00	1.60	1.10	2.00	3.20	0.27	26	11	25
C16/20	16.00	20.00	1.90	1.30	2.50	5.00	0.33	28	12	25
C20/25	20.00	25.00	2.20	1.50	2.90	5.80	0.39	29	13	25
C25/30	25.00	30.00	2.60	1.80	3.30	6.60	0.45	31	13	25
C30/37	30.00	37.00	2.90	2.00	3.80	7.80	0.45	32	14	25
C35/45	35.00	45.00	3.20	2.20	4.20	8.40	0.45	34	15	25
C40/50	40.00	50.00	3.50	2.50	4.60	9.20	0.45	35	15	25
C45/55	45.00	55.00	3.80	2.70	4.90	9.60	0.45	36	16	25
C50/60	50.00	60.00	4.10	2.90	5.30	10.40	0.45	37	16	25
C55/67	55.00	67.00	4.20	3.00	5.50	10.40	0.45	38	16	25
C60/75	60.00	75.00	4.40	3.10	5.70	10.40	0.45	37	16	25
C70/85	70.00	85.00	4.60	3.20	6.00	10.40	0.45	37	16	25
C80/95	80.00	95.00	4.80	3.40	6.30	10.40	0.45	37	16	25
C90/105	90.00	105.00	5.00	3.50	6.60	10.40	0.45	37	16	25

fck: characteristic cylinder compressive strength at 28 days, fck,c: characteristic cube compressive strength, fctm: mean axial tensile strength, fctk0.05: minimum tensile strength, fctm0.95: maximum tensile strength, fct,fl: flexural tensile strength, fvck: shear strength, Ec: modulus of elasticity, Gc: Shear modulus, w: unit weight

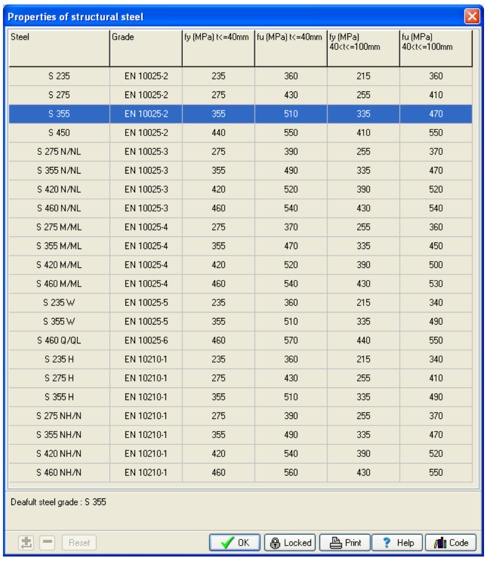
Reset OK Locked Print Help



Structural timber-Strength classes EN 338:2009, EN 1194:2000

Class	ID	fmk [MPa]	ft0k [MPa]	ft90k [MPa]	fc0k [MPa]	fc90k [MPa]	fvk [MPa]	E0m [MPa]	E05 [MPa]	E90m [MPa]	Gm [MPa]	pk [Kg/m³]
C14	0	14.00	8.00	0.40	16.00	2.00	3.00	7000	4700	230	440	290
C16	0	16.00	10.00	0.40	17.00	2.20	3.20	8000	5400	270	500	310
C18	0	18.00	11.00	0.40	18.00	2.20	3.40	9000	6000	300	560	320
C20	0	20.00	12.00	0.40	19.00	2.30	3.60	9500	6400	320	590	330
C22	0	22.00	13.00	0.40	20.00	2.40	3.80	10000	6700	330	630	340
C24	0	24.00	14.00	0.40	21.00	2.50	4.00	11000	7400	370	690	350
C27	0	27.00	16.00	0.40	22.00	2.60	4.00	11500	7700	380	720	370
C30	0	30.00	18.00	0.40	23.00	2.70	4.00	12000	8000	400	750	380
C35	0	35.00	21.00	0.40	25.00	2.80	4.00	13000	8700	430	810	400
C40	0	40.00	24.00	0.40	26.00	2.90	4.00	14000	9400	470	880	420
C45	0	45.00	27.00	0.40	27.00	3.10	4.00	15000	10000	500	940	440
C50	0	50.00	30.00	0.40	29.00	3.20	4.00	16000	10700	530	1000	460
D18	1	18.00	11.00	0.60	18.00	7.50	3.40	9500	8000	630	590	475
D24	1	24.00	14.00	0.60	21.00	7.80	4.00	10000	8500	670	620	485
D30	1	30.00	18.00	0.60	23.00	8.00	4.00	11000	9200	730	690	530
D35	1	35.00	21.00	0.60	25.00	8.10	4.00	12000	10100	800	750	540
D40	1	40.00	24.00	0.60	26.00	8.30	4.00	13000	10900	860	810	550
D50	1	50.00	30.00	0.60	29.00	9.30	4.00	14000	11800	930	880	620
D60	1	60.00	36.00	0.60	32.00	10.50	4.50	17000	14300	1130	1060	700
D70	1	70.00	42.00	0.60	34.00	13.50	5.00	20000	16800	1330	1250	900
GL24h	2	24.00	16.50	0.40	24.00	2.70	2.70	11600	9400	390	720	380
GL28h	2	28.00	19.50	0.45	26.50	3.00	3.20	12600	10200	420	780	410
GL32h	2	32.00	22.50	0.50	29.00	3.30	3.80	13700	11100	460	850	430
GL36h	2	36.00	26.00	0.60	31.00	3.60	4.30	14700	11900	490	910	450
GL24c	2	24.00	14.00	0.35	21.00	2.40	2.20	11600	9400	320	590	350
GL28c	2	28.00	16.50	0.40	24.00	2.70	2.70	12600	10200	390	720	380
GL32c	2	32.00	19.50	0.45	26.50	3.00	3.20	13700	11100	420	785	410
GL36c	2	36.00	22.50	0.50	29.00	3.30	3.80	14700	11900	460	850	430

OK Print Help



Properties of structural steel

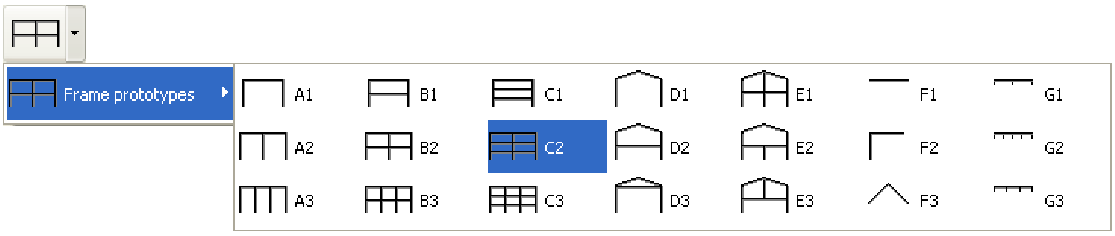
Steel	Grade	fy (MPa) t<=40mm	fu (MPa) t<=40mm	fy (MPa) 40<t<=100mm	fu (MPa) 40<t<=100mm
S 235	EN 10025-2	235	360	215	360
S 275	EN 10025-2	275	430	255	410
S 355	EN 10025-2	355	510	335	470
S 450	EN 10025-2	440	550	410	550
S 275 N/NL	EN 10025-3	275	390	255	370
S 355 N/NL	EN 10025-3	355	490	335	470
S 420 N/NL	EN 10025-3	420	520	390	520
S 460 N/NL	EN 10025-3	460	540	430	540
S 275 M/ML	EN 10025-4	275	370	255	360
S 355 M/ML	EN 10025-4	355	470	335	450
S 420 M/ML	EN 10025-4	420	520	390	500
S 460 M/ML	EN 10025-4	460	540	430	530
S 235 W	EN 10025-5	235	360	215	340
S 355 W	EN 10025-5	355	510	335	490
S 460 Q/QL	EN 10025-6	460	570	440	550
S 235 H	EN 10210-1	235	360	215	340
S 275 H	EN 10210-1	275	430	255	410
S 355 H	EN 10210-1	355	510	335	490
S 275 NH/N	EN 10210-1	275	390	255	370
S 355 NH/N	EN 10210-1	355	490	335	470
S 420 NH/N	EN 10210-1	420	540	390	520
S 460 NH/N	EN 10210-1	460	560	430	550

Deafult steel grade : S 355

Reset OK Locked Print Help Code

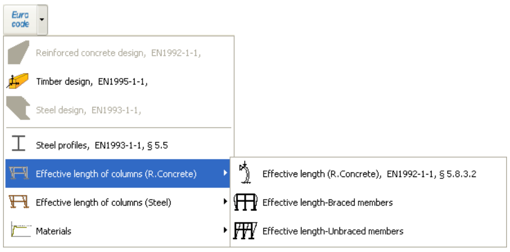
5. Frame prototypes

Selecting a Frame prototypes the program defines the nodal coordinates, support conditions and element properties and connectivity.

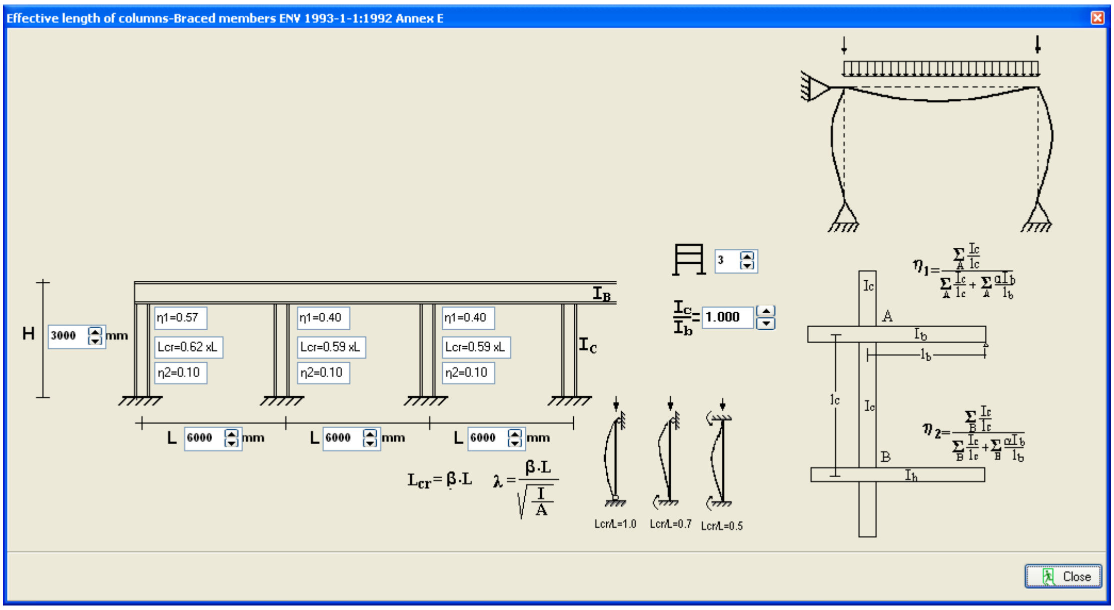


6. Effective lengths for columns

A difficult problem for frames is to define the buckling length for the columns. To help for this special tools have been included in the program (Design/Effective length for columns) for braced and unbraced frames. The curves and tools are based on Eurocode 2 §5.8.3.2 for concrete and Eurocode 3 for steel.



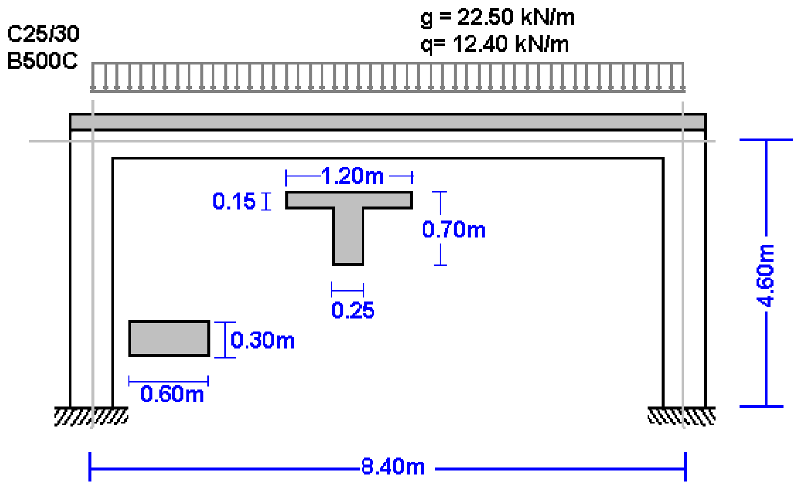
In the appearing windows for computing the effective lengths of columns in braced or unbraced frames, you input the basic frame dimensions and section properties.
For steel frames you input the ratio of flexural stiffness's, column stiffness/beam stiffness.
The critical buckling lengths of the columns are displayed as ratios of the column lengths eg. Lcr = 0.62 x L, 0.59 x L



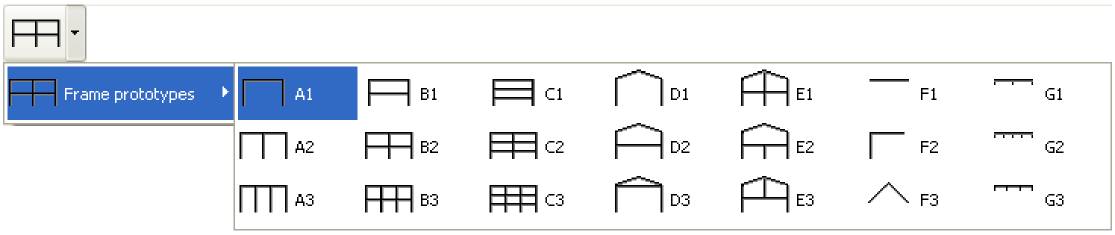
7. Examples

7.1 Example 1

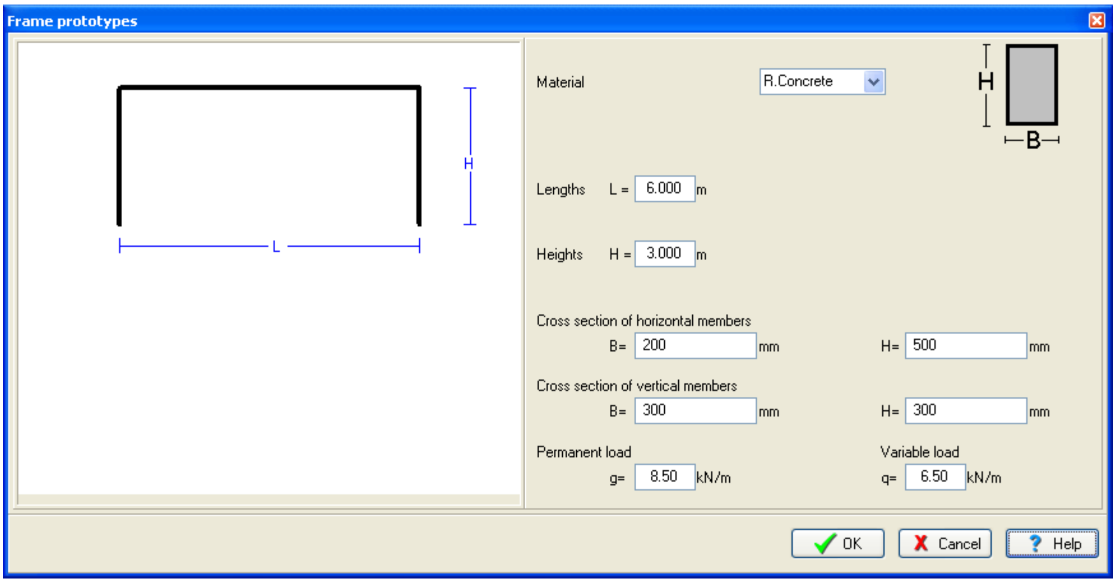
Frame of reinforced concrete 8.40 m x 4.60 m C25/30 B500C



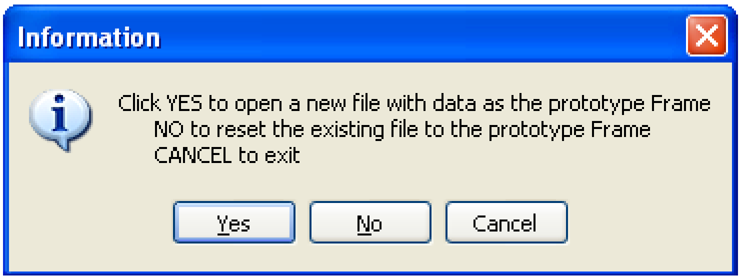
Select a frame from File/Frame prototypes:



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.

NA - National Annex	Eurocode EN
Partial safety factors for actions	γG=1.35 γQ=1.50 ψ2=0.30

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	(uniform)	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

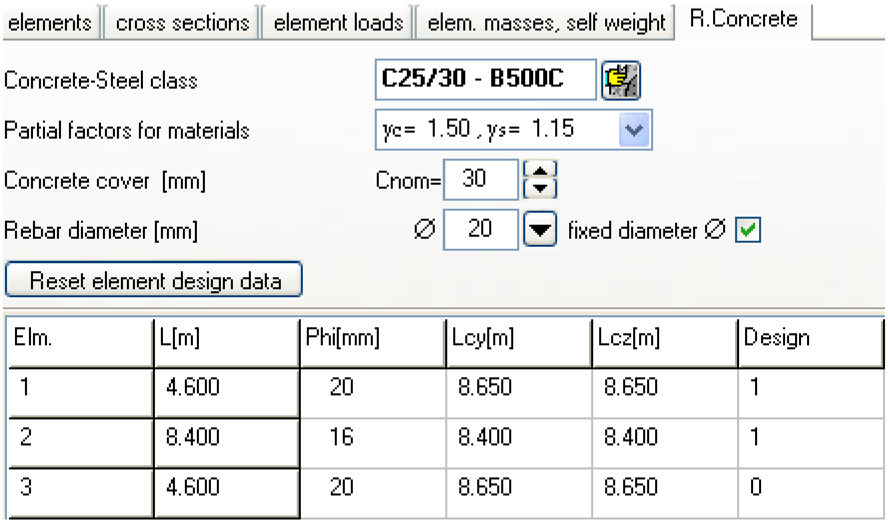
Weight density kN/m3: 25.00 | include self weight in loads and masses: ☑

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

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

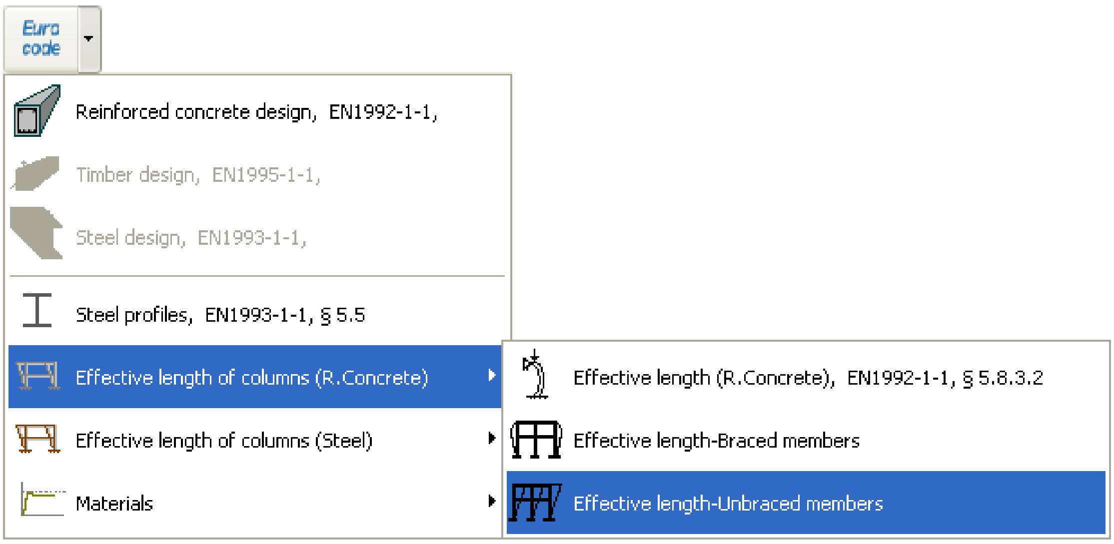
- Concrete. You specify the basic data for the design of reinforced concrete according to Eurocode 2. Select Concrete and Steel class. Partial factors for materials, according to the

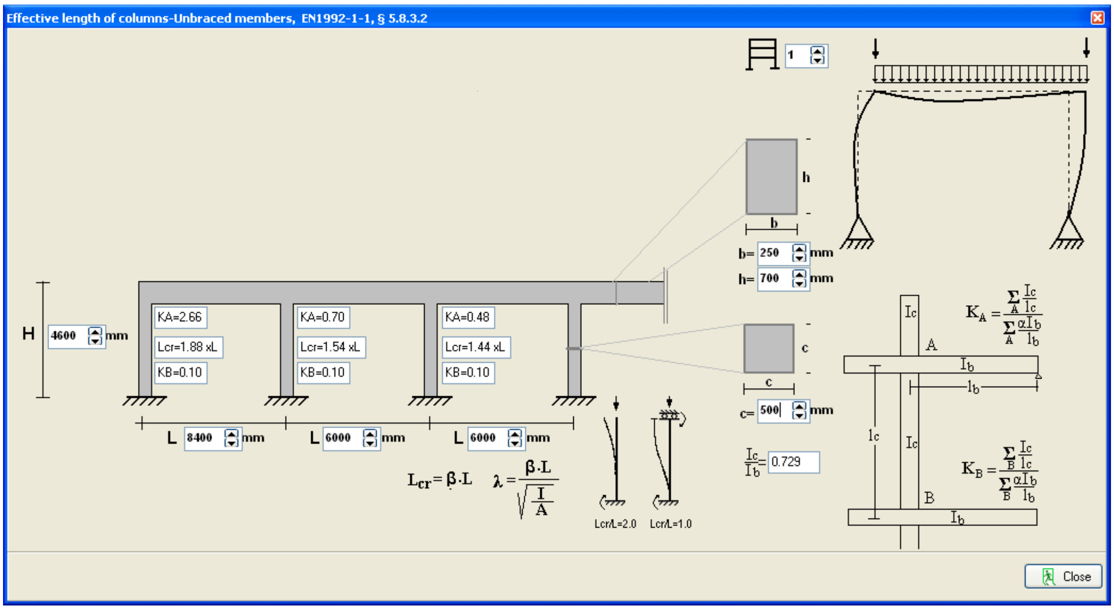
National Annex, for ULS (ultimate limit state) γc=1.50, γs=1.15 and SLS (serviceability limit state) γc=1.00, γs=1.00. Cnom is the concrete cover according to Eurocode 2 §3.4.1.
The rebar diameter is used as the optimum desired by the program. If you check next to the rebar diameter then is only this 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.



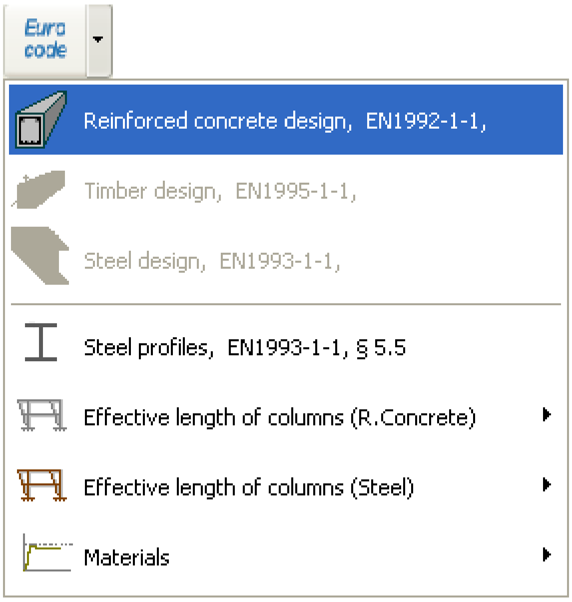
Elm.	L[m]	Phi[mm]	Lcy[m]	Lcz[m]	Design
1	4.600	20	8.650	8.650	1
2	8.400	16	8.400	8.400	1
3	4.600	20	8.650	8.650	0

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

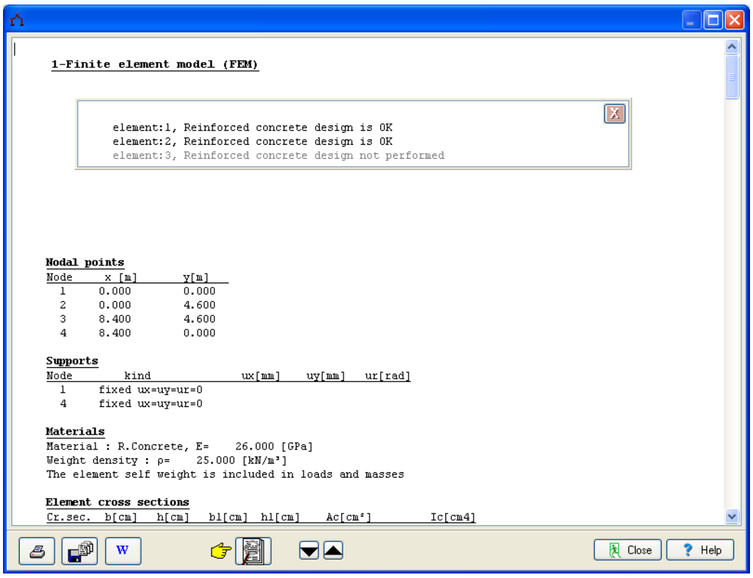




After you give all the data the reinforced concrete design is performed according to Eurocode 2.

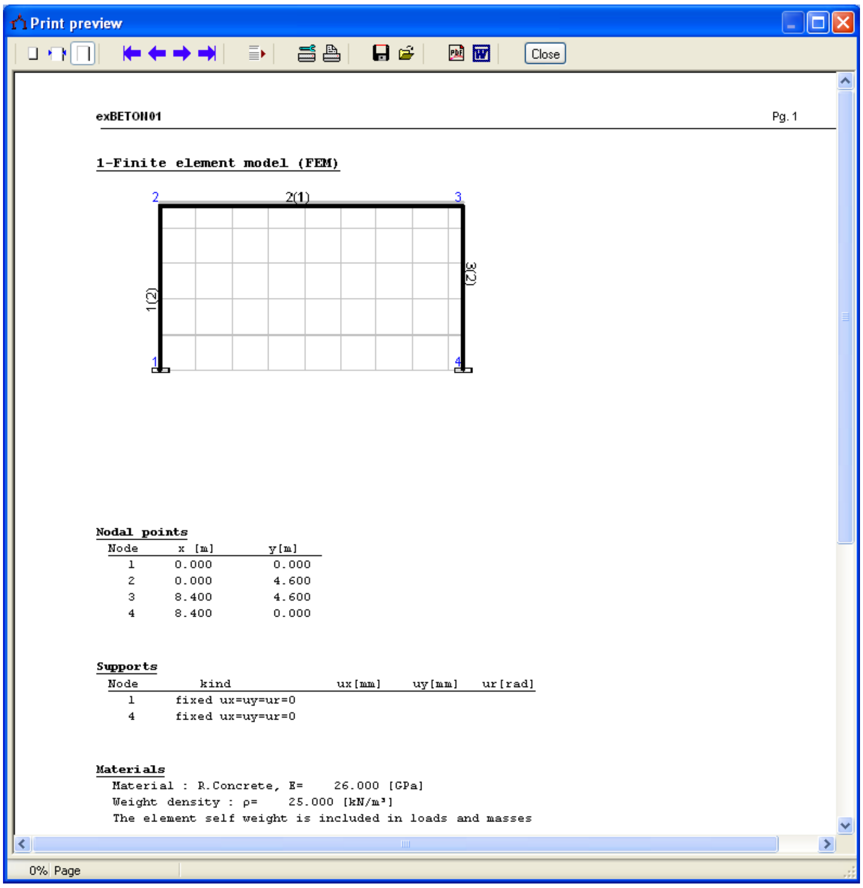


Check if every element is verified in the design.



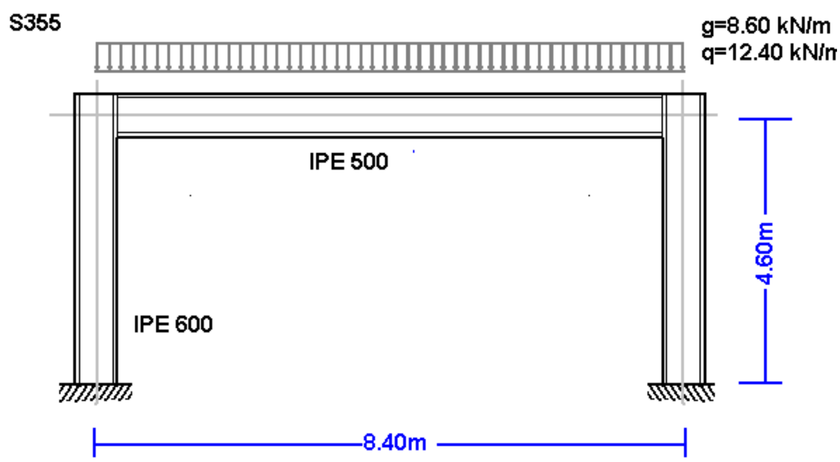
Click for complete formatted report.

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 .

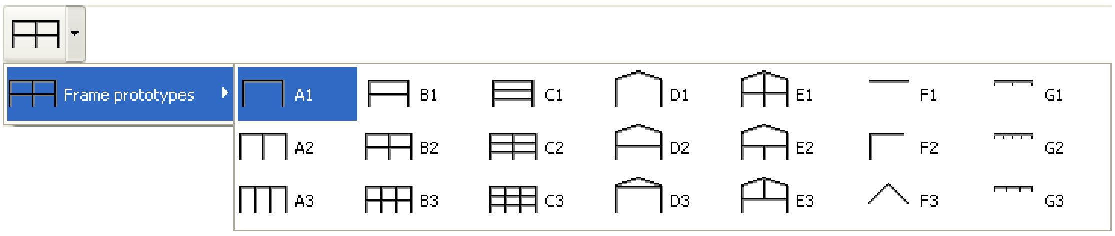


7.2 Example 2

Steel frame 8.40 x 4.60 S355

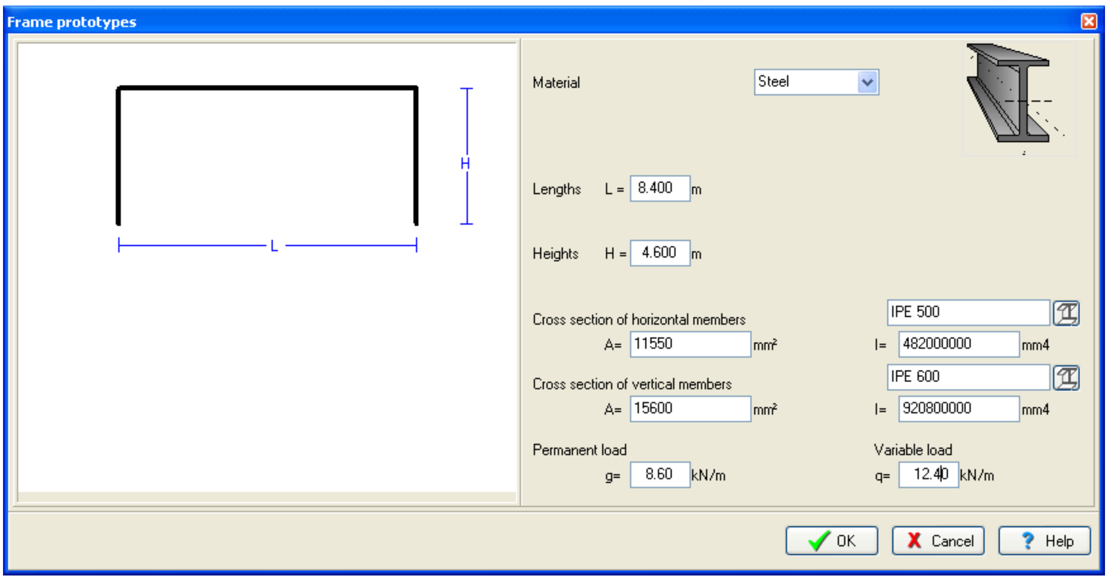


Select a frame from File/Frame prototypes:

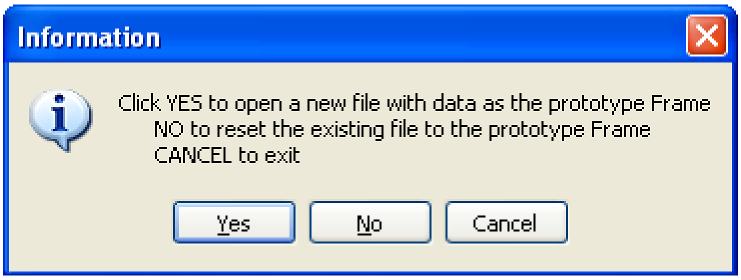


Select material Steel and give the basic dimensions, cross sections and loads.
You can always change and adjust these values afterwards.

For element cross sections click .

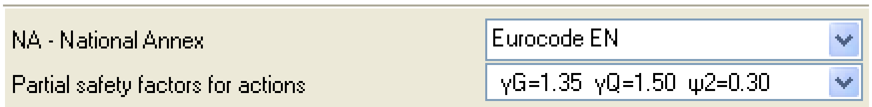


Click Yes and give the file name.



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.

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

structure material: Steel — Elasticity modulus: E (GPa)= 210.00 — cross section units: mm

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.

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

load combination 1.35 xG + 1.50 xQ

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).

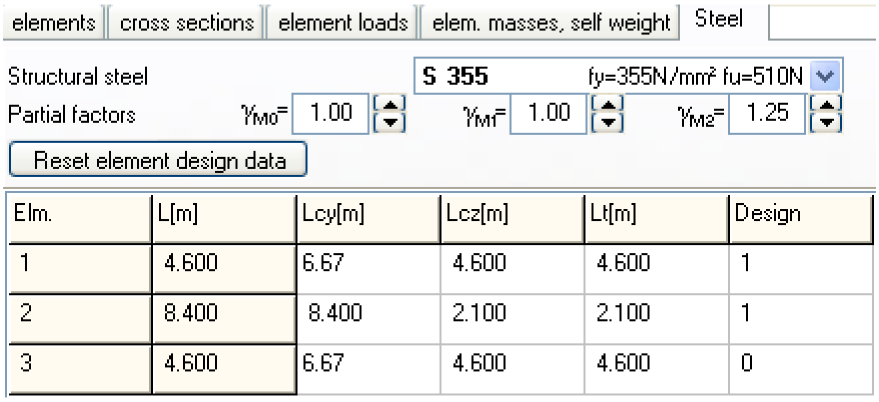
elements | cross sections | element loads | elem. masses, self weight | Steel

Weight density kN/m3: 78.50 — include self weight in loads and masses: ☑

Mass combination: 1.00 xMg + 0.30 xMq

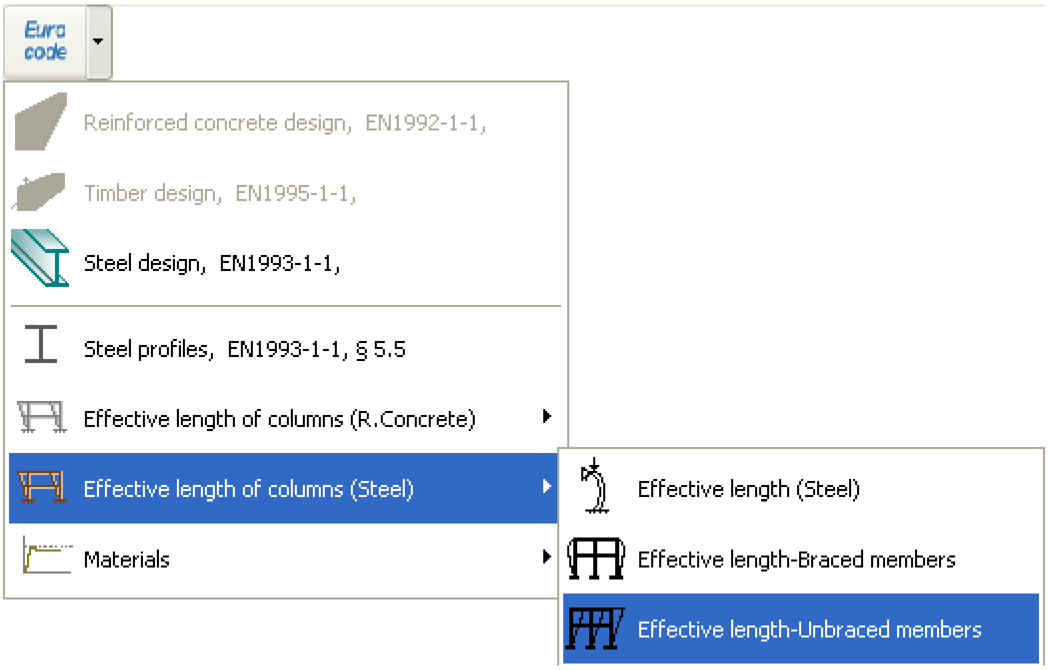
element	Gg [kN/m]	Gq [kN/m]

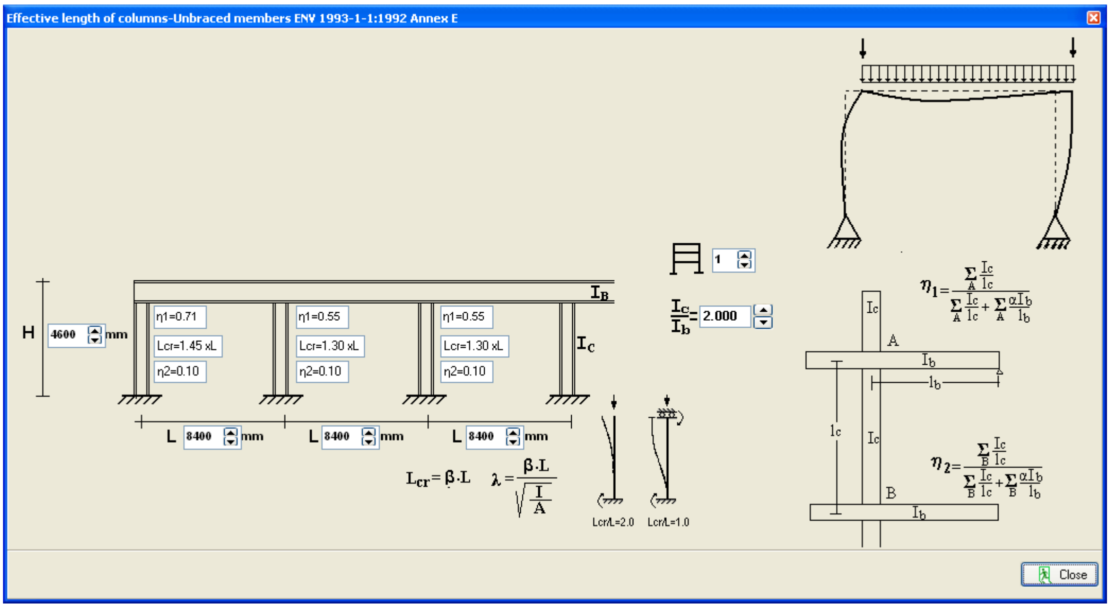
- Steel. You specify the basic data for the design of steel according to Eurocode 3. Select Steel grade. Partial factors for materials, according to National annex, for ULS (ultimate limit state) γM0 = 1.00, γM1 = 1.00, γM2 = 1.25. You have to define the buckling lengths. Lcy buckling length for in plane 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,



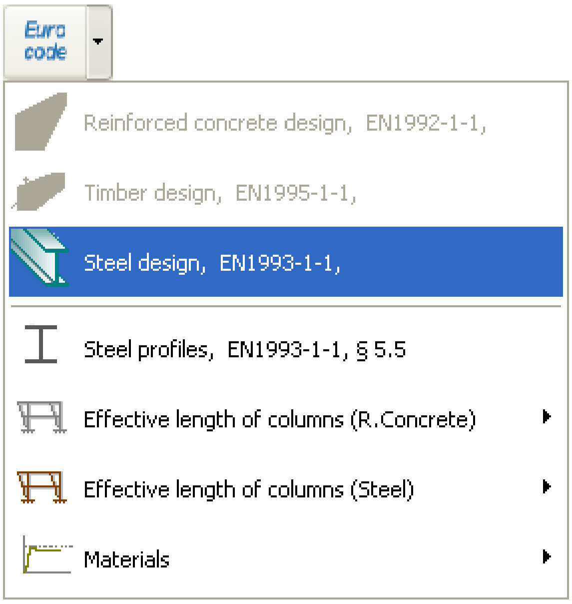
Elm.	L[m]	Lcy[m]	Lcz[m]	Lt[m]	Design
1	4.600	6.67	4.600	4.600	1
2	8.400	8.400	2.100	2.100	1
3	4.600	6.67	4.600	4.600	0

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

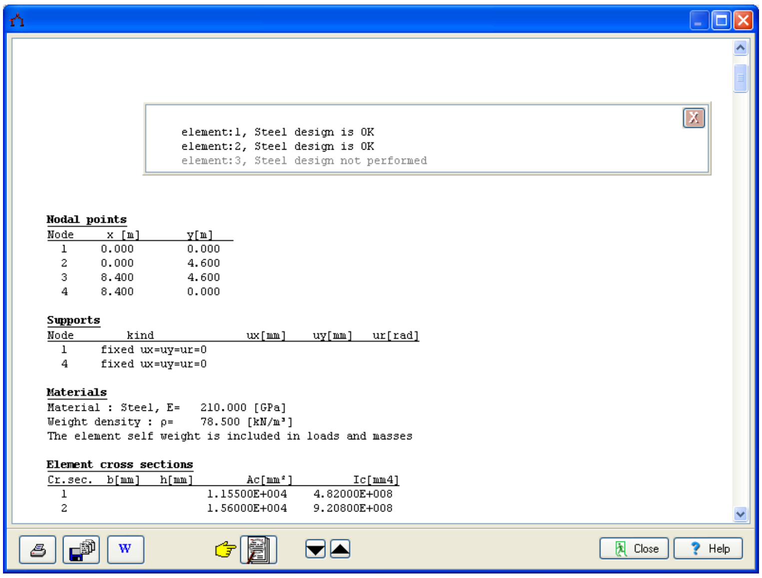




After you give all the data the steel design is performed according to Eurocode 3.

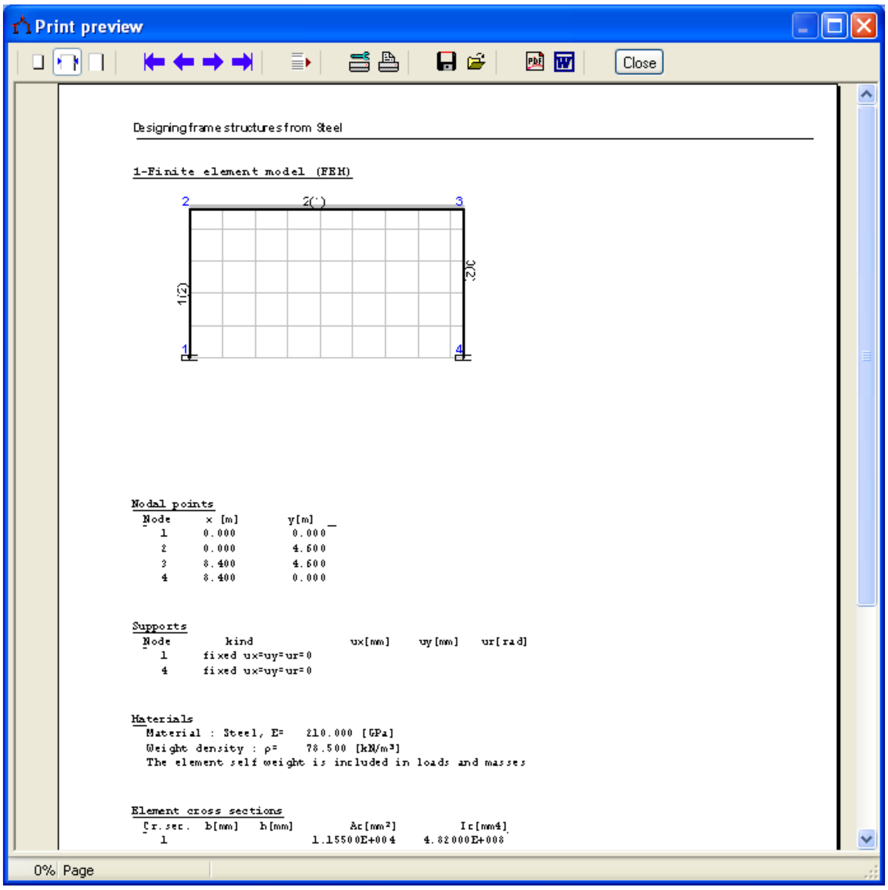


Check if every element is verified in the design,



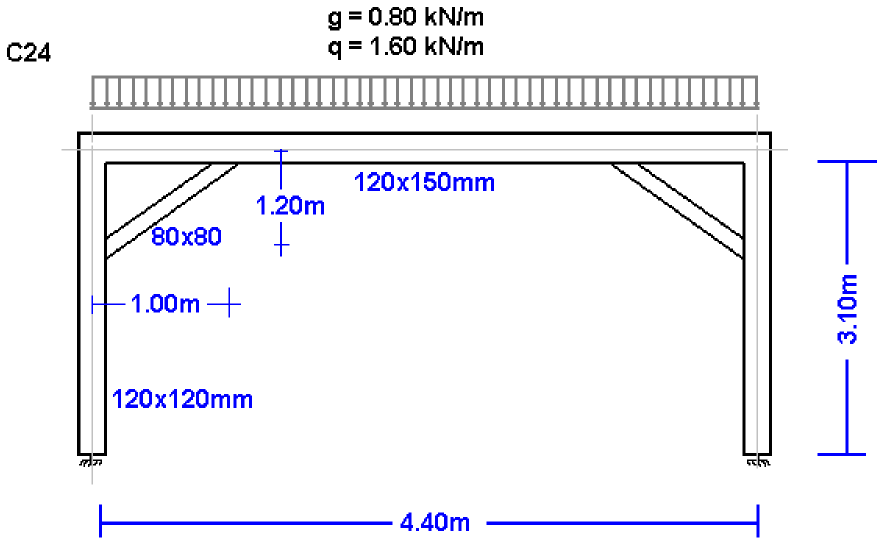
Click for complete formatted report.

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 .

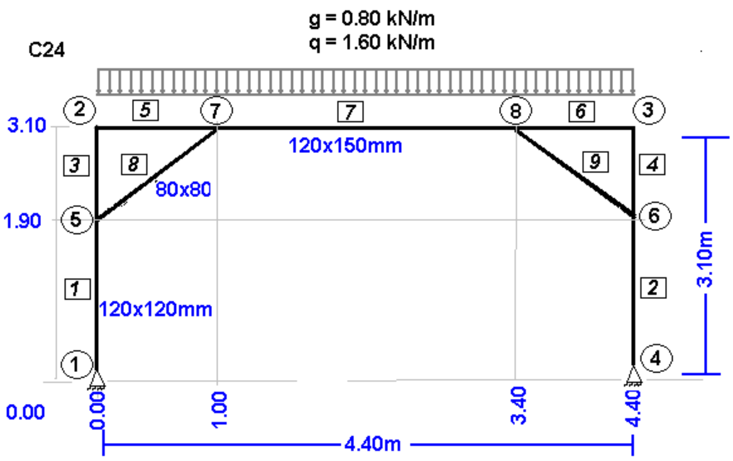


7.3 Example 3

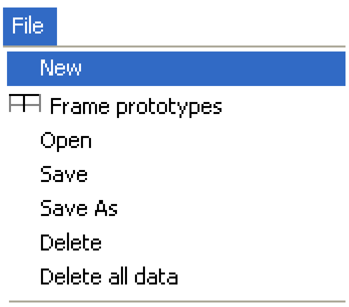
Timber structure 6.40 x 4.60 C24



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.

NA - National Annex	Eurocode EN
Partial safety factors for actions	γG=1.35 γQ=1.50 ψ2=0.30

- 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).

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. 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.

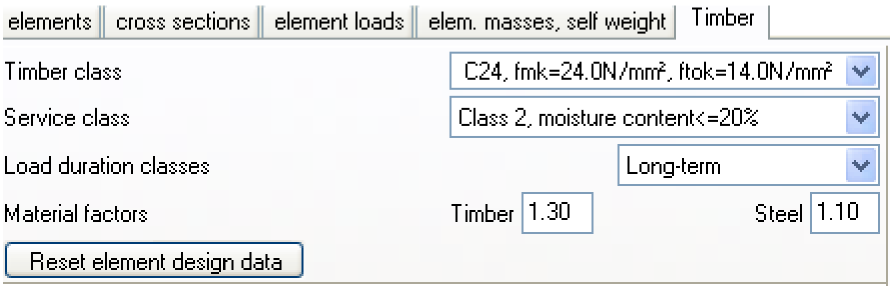
elements | cross sections | element loads | elem. masses, self weight | Timber

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	

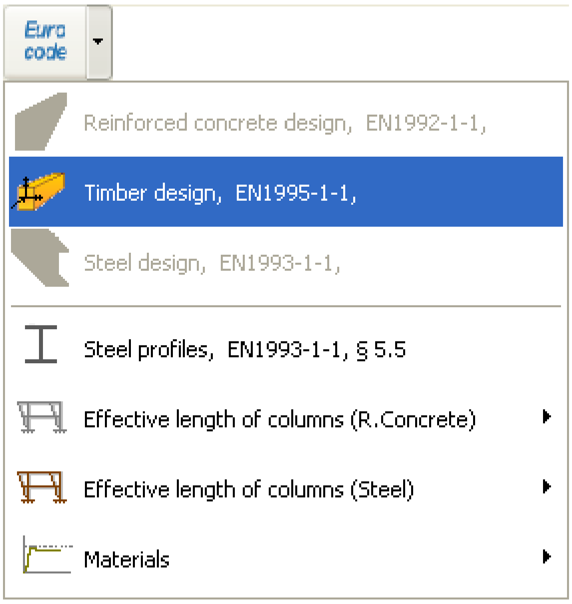
- Element masses self weight. The weight density of the material. If you check to include the self weight in the load and masses, the program adds in dead loads the self weight of each element (unit weight) x (cross section area). The weight density is set automatically by the material (timber 9 kN/m^3).

- 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).

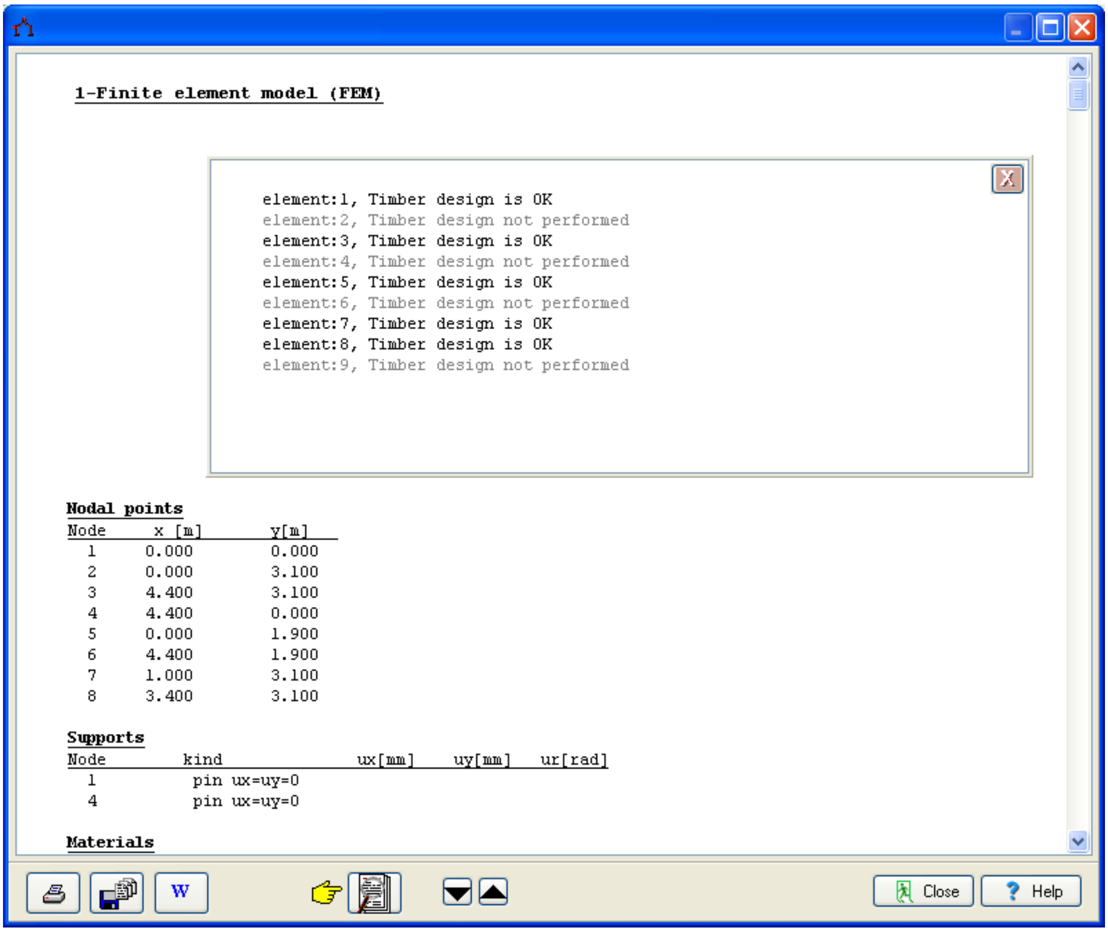


Elm.	L[m]	Lcy[m]	Lcz[m]	Design
1	1.900	1.900	1.900	1
2	1.900	1.900	1.900	0
3	1.200	1.200	1.200	1
4	1.200	1.200	1.200	0
5	1.000	1.000	1.000	1
6	1.000	1.000	1.000	0
7	2.400	2.400	1.200	1
8	1.562	1.562	1.562	1
9	1.562	1.562	1.562	0

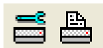
After you give all the data the timber design is performed according to Eurocode 5.



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 .